

DesignBuilder Simulation Documentation

for DesignBuilder v2025.1



DesignBuilder © No part of the manual is to be copied or reproduced in any form without the express agreement of DesignBuilder Software Ltd

Contents

Welcome To DesignBuilder V2025.1	42
Learn About DesignBuilder	44
Recent Files (Opening Screen)	44
User Interface	45
Menu	47
File Menu	47
Export DesignBuilder XML (dsbXML)	49
Import DesignBuilder XML (dsbXML)	49
Export HVAC Loads Report	49
HVAC Loads Options Dialog	49
Viewing The HVAC Loads Report	51
Export To MagiCAD	51
Export CSV Report	52
Edit Menu	53
Go Menu	54
View Menu	54
Tools Menu	54
Help Menu	55
Keyboard Shortcuts	55
Edit Screen	57
DesignBuilder Tips	58
Core Concepts	61
Model Data Hierarchy & Data Inheritance	61
Templates And Components	63
Blocks Introduction	63
Navigating Building Models	65
Using The Navigator	66
Title / Breadcrumbs	67
Customising Model Options	68
DesignBuilder Model Files	68
Guidance For Working With DesignBuilder .dsb Files	68
Licensing & Activation	68
More Detailed Information On Licensing	82
Single User Licences	82
Setting Up A Licence Server For Site Network Licenses	82
Summary Of Process	82
Detailed Steps	83
Troubleshooting	86
Program Options	89
Program Options - User Interface	89
Program Options - Dialogs	94
New Object Dialogs	94
Reports	94
EN ISO 673 Glazing U-value Calculations	94
Detailed HVAC	95
Hideable Messages	95
Program Options - Files	96
Backup	96
Saved Models	96
DesignBuilder XML Files	96
Default Folders	97
Program Options - EnergyPlus	97
EnergyPlus	97
Output File	100
IDF Options	101
IDF Object Naming	102
Distance Between Site Location And Weather File	102
Using Custom Versions Of EnergyPlus	103
Program Options - Limits	103

Program Options - International	104
Units	104
Currency.....	105
Filters	105
Building Models.....	107
Create New Site.....	107
Create New Site - Location	107
Create New Site - Template	107
Building Geometry.....	107
View Controls.....	108
Interaction Of View And Edit Commands.....	108
View Rotation	109
Orbit	110
Zoom Dynamic	111
Zoom Window	111
Fit View	112
Pan View	112
Normal View	112
Walk-Through.....	113
When Using The Mouse	113
When Using Keys.....	113
Look Around	113
When Using The Mouse	114
Surrounding Block/Zone Display	114
Ghost Blocks.....	116
Ghost Selected Blocks	116
Show All Blocks.....	117
Perimeter Drawing Options	117
Shape	117
Polygon	118
Rectangle	118
Circle	119
Line Type	119
Straight Line	120
Arc.....	120
Snaps, Drawing Guides, Protractor And Grids	121
Direction Snaps	122
Axis Snap	122
Parallel Snap	124
Normal (Perpendicular) Snap	125
Offset Snap.....	127
Point Snaps	131
End-point Snap	131
Mid-point Snap.....	132
Edge Snap	132
DXF Snap.....	134
Snap To Lower Perimeters	135
Increment Snap.....	135
Drawing Guides	137
Protractor Tool.....	139
Drawing Grid	141
Construction Lines	142
Drawing Construction Lines.....	143
Removing Construction Lines	147
General Operations	147
Select	148
Move.....	150
Clone (Copy)	152
Mirror	153
Mirror Block.....	153
Mirror Openings	154
Mirror Partitions And Courtyards.....	155

Rotate	156
Stretch	157
Delete	159
Measure	159
Importing Geometric Data	160
Import 2-D Drawing	160
Import Drawing File Wizard	163
DXF File Types	164
Editing DXF Data	167
Bitmap-based File Types	168
Editing Bitmap Data	169
Partitions	170
Import 3-D Model	170
Import 3-D Model	172
Import Options	172
How Does DesignBuilder IDF Import Work?	176
How Does DesignBuilder GbXML Import Work?	176
The Revit DesignBuilder Workflow	177
IDF Import	178
Merge Model Data From File	179
Import Building From External File	180
Exporting Geometric Data	181
Exporting 3-D DXF	181
Exporting GbXML	182
Working With Buildings (Site Level)	182
Add New Building	183
Including Multiple Buildings On The Site	183
Add New Building - Building Tab	183
Model Type	183
Add New Building - Default Data Tab	184
Working At Building Level - Blocks And Solar Collectors	185
Adding Blocks	186
Block Creation Options	191
Block And Zone Dimensions	192
Block Types	194
Building Block	194
Outline Block	196
Using Outline Blocks To Create A Dormer Window	196
Component Block	200
Standard Component Block	200
Ground Component Block	202
Adiabatic Component Blocks	203
Assembly Block	205
Block Forms	208
Extruded Block Form	208
Sloped Wall Block Form	209
Pitched Roof Block Form	210
Pitched Roof Drawing Options	211
Automatic Pitched Roof Geometry	212
Dome Block Form	216
General Block Form	217
Height	217
Auto-Complete Block	217
Follow Path	217
Edit Blocks	220
Drag Face	221
Divide Block Using Cutting Tool	222
Draw Plane Method	222
Select Plane Method	228
Boolean Geometry Operations	233
Edit Block Options	245
Convert Blocks	245

Transfer Blocks.....	246
Add Surface.....	246
Adding Solar Collectors.....	247
Panel Active Direction.....	248
Edit Openings At Building Level.....	249
Working At Block Level	252
Block Level Options	253
Zone Block By Drawing Partitions.....	255
Partition Block Into Zones	257
Deleting Partitions	260
Complex Block Geometry	260
Hanging And Outline Partitions	262
Virtual Partitions	263
Creating Voids (Courtyard Spaces) Within Blocks	264
Soft And Hard Voids.....	265
Open Manifold Blocks - Flip Block Surface Orientation.....	265
CFD Circulation Fans	267
Add CFD Circulation Fan	267
Working Within Zones (Zone Level)	268
Placing Light Sensors.....	269
Working With Surfaces (Surface Level)	270
Draw Window	274
Using Key Ins To Create Openings	275
Draw Sub-Surface.....	277
Draw Door.....	278
Draw Hole	278
Draw Vent	279
Add CFD Boundary.....	279
Edit Surfaces.....	284
Undo/Redo.....	284
Rebuild.....	285
Rezone Block	285
Model Data.....	285
Site Data	286
Location	286
Site Location	287
Site Details	288
Site Details - General.....	288
Site Details - Site Height Variation.....	290
Site Details - Ground	291
Ground Modelling.....	291
Surface.....	293
Ground Monthly Temperatures	295
Ground Deep Temperatures	295
Ground Shallow Temperatures	295
FCFactor Ground Temperatures.....	296
Ground Modelling - Standard Method	296
How It Works.....	296
Ground Temperatures.....	297
Example Application - Basement Or Cellar Under Inclined Ground Slope	297
Kiva Summary.....	299
Two-Dimensional Approximation	301
Numerical Calculations	302
Boundary Conditions.....	305
Multiple Kiva Instances	305
Core Zone Slabs	309
Initialisation And Warm-Up	309
HVAC Autosizing.....	310
Kiva Limitations	310
Validation	311
References.....	311
Horizon	311

Site Details - Mains Water Temperature	311
Site Details - Precipitation + Irrigation	312
Precipitation	312
Site Green Roof Irrigation	313
Site Details - Outdoor Air CO2 And Contaminants	314
Time And Daylight Saving	314
Simulation Weather Data	315
Winter Design Weather Data	316
Sizing Period	316
Summer Design Weather Data	317
Sky	318
Weather Data Modifiers	319
Wind Data	321
Sizing Period	321
Design Temperature Period	322
Yearly Design Temperatures	323
Monthly Design Temperatures	324
Legislative Region And Insulation Standards	324
Include Fuel Emissions	324
Energy Codes/insulation Standards	325
Building Model Data	325
Activity Model Data	326
Zone Type	327
Zone Multiplier	329
Adiabatic Component Blocks	330
Include Zone	330
Floor Areas And Volumes	331
Building Level	331
Zone Level	332
Occupancy	332
Timing	333
Metabolic	334
Clothing	335
Comfort Radiant Temperature Calculation	336
Air Velocity	336
Contaminant Generation And Removal	336
Constant	337
Cutoff Model	338
Decay Source	339
Holidays	340
Compact Schedules	340
7/12 Schedules	341
Typical Workday	341
Environmental Control	341
Heating Setpoint Temperatures	341
Cooling Setpoint Temperatures	341
Humidity Setpoint Temperatures	342
Comfort PMV Setpoints	342
Ventilation Setpoint Temperatures	343
Minimum Fresh Air	345
CO2/Contaminant Setpoints	346
Lighting	346
Domestic Hot Water (DHW)	347
Computer Gains	347
Office Equipment Gains	348
Miscellaneous Gains	349
Calculation	350
Catering Gains	350
Calculation	352
Process Gains	352
Calculation	354
Construction Model Data	354

Templates	354
Constructions	355
Constructions Data.....	356
Semi-Exposed.....	358
Floors	359
Navigator Use - Surface Level	359
Zone Statistics.....	360
Sub-Surface Constructions	360
Internal Thermal Mass.....	361
Adjacency	363
Geometry Areas And Volumes	364
Fixed Surface Thicknesses	367
Considerations For CFD	368
Considerations For Parametric Analysis And Optimisation	368
Void Depths.....	368
Special Conditions	369
Geometry Convention Examples	372
Curtain Glazing - Example	372
Surface Convection	373
Component Block	376
Solar Collector Model Data	380
Photovoltaic Options	381
PV Generator - Simple.....	383
PV Solar Collector Performance - Equivalent One Diode	384
Current	386
Voltage	386
Nominal Operating Cell Temperature	386
Linear Thermal Bridges At Junctions.....	387
Psi Values Including Metal Cladding.....	388
Psi Values NOT Including Metal Cladding	389
Advanced	389
Baffle.....	390
Ground Modelling	392
Kiva	392
Ground Domain.....	392
Ground Domain Slab Tutorial.....	393
Data Entry	393
Simulations.....	397
Simulation Using Multiple Years	399
Airtightness	399
Airtightness (Scheduled Natural Ventilation)	400
DeltaT And Wind Speed Coefficients.....	402
Airtightness (Calculated Natural Ventilation)	403
Calculated Natural Ventilation.....	403
Scheduled Natural Ventilation + Heating And Cooling Design Calculations	405
Construction Cost	406
Cost Of Internal Finishes	406
Openings Model Data (Windows, Doors, Vents, Holes, Sub-surfaces)	406
External Glazing.....	407
Dimensions (Walls).....	408
Frame And Dividers.....	411
Reveal	412
Frame.....	415
Dividers	416
Glazing Assemblies.....	418
Shading.....	418
Window Shading.....	419
Slat Angle Control	422
Operation	423
Limitations	423
Local Shading	423
Airflow Windows.....	423

Window Operation And Free Aperture	425
Operation	425
Free Aperture	427
Scheduled Natural Ventilation.....	430
Internal Glazing.....	430
Roof Glazing.....	431
Dimensions (Roofs).....	431
Frame And Dividers	432
Reveal	433
Frame	436
Dividers	437
Roof Shading	439
Window Shading.....	439
Slat Angle Control	443
Operation	443
Limitations	443
Window Operation And Free Aperture	443
Operation	443
Free Aperture	445
Scheduled Natural Ventilation.....	448
Doors	449
External Doors	449
Internal Doors.....	449
Door Placement	450
Door Operation	450
Calculated Natural Ventilation.....	451
Scheduled Natural Ventilation.....	451
Vents And Holes	451
External Vents	452
Roof Vents	452
Internal Vents	453
Holes	453
Vent Placement	454
Vent Operation	454
Calculated Natural Ventilation.....	455
Scheduled Natural Ventilation.....	455
Wind Pressure Coefficients	456
Lighting Model Data	456
General Lighting	457
Task And Display Lighting	460
Lighting Control.....	461
Lighting Areas	461
Procedure.....	462
Glare	465
Algorithms	465
Output	465
Setting The Position Of Daylight Sensors.....	465
Exterior Lighting	466
Lighting Cost.....	467
HVAC Model Data	467
Simple HVAC.....	468
Auxiliary Energy	468
NCM Setting.....	469
Separate Fans And Pumps Setting.....	469
Mechanical Ventilation	469
1-Room Ventilation.....	470
2-Ideal Loads	471
Outside Air	472
Operation	473
Heating.....	474
Heating Plant.....	474
Local Heating Units	475

1-Convective	475
2-Radiant/Convective Units	476
Sizing Zone Equipment	477
Operation	478
Cooling	478
Supply Air Conditions	479
Cooling Plant	480
Operation	481
Humidity Control	481
Humidification	481
Dehumidification	481
Humidity Control Linkage With Heating And Cooling Schedules	482
Detailed HVAC	482
Natural Ventilation	482
Scheduled Natural Ventilation Data	483
Natural Ventilation	483
Operation	484
Outdoor Temperature Limits	484
Delta T Limits	485
Delta T And Wind Speed Coefficients	486
Summary Of Scheduled Natural Ventilation Operation Rules	487
Internal Windows, Doors And Vents	487
Calculated Natural Ventilation Data	488
Mixed Mode Zone Equipment	492
Wind And Rain	493
Temperature Control	493
Enthalpy Control	494
Dewpoint Control	494
Advanced	495
Mixed Mode Cooling	496
Mixed Mode Tutorial	499
Earth Tubes	504
Operation	505
Control	505
Fan Electric Consumption Configuration	506
Pipe Specifications	506
Soil Heat Transfer Calculations	507
Flow Term Coefficients	507
Outputs	508
Domestic Hot Water	509
Air Temperature Distribution	510
Air Temperature Distribution - Dynamic Gradient	511
Upper Conditions	512
Lower Conditions	512
Operation	513
Heights	513
How It Works	513
Limitations	513
Air Temperature Distribution - Three Node Displacement Ventilation	514
Advanced	515
Air Temperature Distribution - Under Floor Interior	515
Custom Diffuser Coefficients	516
Height Settings	517
Advanced	517
Air Temperature Distribution - Under Floor Exterior	517
Custom Diffuser Coefficients	518
Height Settings	519
Advanced	519
HVAC Cost	520
Environmental Impact Factors	520
On Site Electricity Generation	521
How To Model Photovoltaic (PV) Systems	523

How To Model A Standalone PV Solar System - Summary	523
How To Model A Building Integrated PV (BIPV) Solar System - Summary.....	524
Electric Load Centre.....	526
General	526
Cost.....	531
Generator List	532
How To Model Wind Turbine Systems	532
How To Model A Wind Turbine System - Summary	532
Wind Turbines.....	533
Vertical Axis	536
Power Coefficients	537
Electrical Storage	537
Simple Storage.....	538
Heat Gain To Zone.....	538
Simple Options	538
Battery	539
Heat Gain To Zone.....	540
Battery Storage	540
Battery Life	542
Inverters.....	542
Inverter - Simple.....	542
General	542
Simple	543
Heat Gains To Zone.....	543
Inverter - Function Of Power	543
General	543
Function Of Power	544
Heat Gains To Zone.....	545
Inverters - Lookup Table	545
General	545
Lookup Table	545
Efficiency At Nominal Voltage.....	546
Heat Gains To Zone.....	546
Economics.....	546
Tariff Analysis.....	547
Construction Costs.....	547
Life Cycle Cost	547
Tariff Analysis	548
Background	548
Working With Tariff Analysis	548
Tariff Analysis - Tariff Tab	549
Tariff	549
Schedules	550
Other Settings	553
Tariff Analysis - Charge Tab.....	554
Charge Selection.....	555
Charge : Simple	555
Charge : Block.....	556
Tariff Analysis - Qualify Tab	558
Tariff Analysis - Ratchet Tab	560
Ratchet.....	562
Tariff Analysis - User Variables Tab	564
Variables Used.....	565
Variable N Settings	565
January Through December Values	565
Tariff Examples	565
List Of Examples	565
Create Base Model - Optional Preliminary Steps For Examples A-J	566
Example A – Flat Energy Charge	567
Example B – Block Energy Charge.....	570
Example C – Block Energy And Demand Charges	573
Example D - Seasonal Energy Charges With Minimum Charge	577

Example E – KWh/kW Energy Charges.....	582
Example F – Seasonal Time Of Use Energy	586
Example G – Blocks Within Blocks	592
Example H – Real Time Pricing	597
Example I – Selling And Net-Metering	601
Example J - Dual Ratchet Demand Tariff	612
Further Examples	620
Construction Costs	620
Example	620
Construction Costs - Line Item	622
Line Item N Inputs	623
Construction Costs - Adjustments	623
Construction Costs - Reference.....	624
Life Cycle Cost.....	626
Output	626
Life Cycle Cost - Parameters Tab	627
Discount Settings	627
Dates For Study	628
Other Settings	630
Life Cycle Cost - Recurring Costs Tab	631
Recurring Cost N Settings	631
Life Cycle Cost - Non-Recurring Costs Tab.....	632
Non-Recurring Cost N Settings.....	632
Life Cycle Cost - Use Price Escalation Tab	633
Use Price Escalation N Settings	634
Escalation Data	635
Life Cycle Cost - Use Adjustment Tab.....	635
Adjustment N Settings	635
Adjustment Data.....	636
Miscellaneous	636
Outputs - General	636
Output Options - Heating Design.....	637
Output Options - Cooling Design.....	637
Simulation Output Options	638
General	639
Daylighting Outputs	640
CFD.....	640
CFD Boundary Defaults	641
CFD Boundary Conditions Editor	642
Edit Boundary Conditions	643
Import EnergyPlus Boundary Conditions	643
Preparing Simulations To Generate CFD Boundary Conditions	644
Infiltration In CFD.....	645
Assembly Library.....	646
Add Assembly To Library	646
Export/Import.....	646
Updating The Assembly Library	647
Detailed Equipment	647
Edit Detailed Equipment	648
Cost Data Overview	649
1. Super-Structure And Sub-Structure Costs.....	649
2. Building Cost Summary Data	650
Building Cost Summary Data	650
A Substructure	651
B Shell	651
C Interiors.....	652
D Services.....	652
E Equipment And Furnishings	652
F Special Construction And Demolition	652
G Sitework And Utilities	652
Z General Requirements And OH And P	652
Facilitating Works.....	653

Substructure	653
Superstructure.....	653
Internal Finishes.....	654
Fittings, Furnishings And Equipment	654
Services	654
Complete Buildings And Building Units	654
Work To Existing Buildings	654
External Work.....	655
Main Contractors Preliminaries.....	655
Main Contractor's Overhead And Profits	655
Project Design Team Fee	655
Risk Allowance Estimate.....	655
Inflation Allowance	655
Model Data Management	656
Model Data Grid View (MDGV).....	656
Management Dialog	656
Model Data Grid View - Viewing And Editing Model Data	658
Layout Selection.....	658
Launching The View/Edit Grid Dialog	658
Grid Limitations	669
Model Data Grid View - Importing Model Data.....	669
Import Limitations	671
Model Data Grid View - Layout Management.....	671
Model Data Grid View - Layout Editing.....	674
Layout Details	675
Attribute Visibility Rules	676
Attributes Commonly Used In Visibility Rules	680
Model Data Grid View Examples	681
General Steps	681
Load Data From Template	683
Filter Selection	687
General	687
Zones	687
Surfaces	688
Openings.....	689
Save Data To Template	689
Example	690
Reload Templates	691
Clear Data To Default	692
Merging Zones	693
Selective Zone Merging	694
Merging Zones - Discussion	696
Zone Level Data For Internal Surfaces	698
Load Whole Building Infiltration	699
Correct Data	699
Naming Model Objects.....	700
EnergyPlus IDF Names	702
Detailed HVAC.....	703
Overview	703
Accessing Detailed HVAC	704
Detailed HVAC Contents	705
Detailed HVAC Terminology	705
HVAC Loops	705
Air Loops	709
Air Loop - Generic Single Duct	710
Air Loop - Generic Dual Duct	710
Air Loop - Unitary Heat Cool	711
Air Loop - Unitary Heat Pump Air To Air.....	712
Air Loop - Unitary Heat Pump Water To Air	712
Air Loop - Unitary Generic.....	713
Air Loop - Dedicated Outdoor Air System	714
Plant And Condenser Loops	715

Hot Water Plant Loop	715
Chilled Water Plant Loop	716
Primary/Secondary Chilled Water Plant Loop	717
Domestic Hot Water Plant Loop	719
Solar And Auxiliary Heating Loops	720
Condenser Loop	724
VRF Loops	726
Loop Data	727
Air Loop Data	727
General	727
Sizing	727
Heating	729
Cooling	730
DOAS Loop Data	731
General	731
Heating	731
Cooling	731
Plant And Condenser Loop Data	731
General	732
Flow Type	732
Temperatures	733
Flow Rates	733
Sizing	734
Operation	734
Plant Equipment Operation	734
Scheme	735
Range	736
Range Equipment	736
Solar Loop Data	736
General	736
Flow Type	737
Temperatures	737
Flow Rates	737
Temperature Protection	738
Differential Thermostat	738
Sizing	739
Operation	739
Plant Equipment Operation	739
Scheme	739
Range	740
HVAC Zones And Zone Groups	740
HVAC Component Hierarchy And Navigation	745
HVAC System Control - Setpoint Managers	745
Setpoint Manager Used With Draw-Through Fans	746
Working At HVAC System Level	746
Load HVAC Template Dialog	747
Select The HVAC System Template	747
Select Building Zones	749
Air System Data	750
Hot Water System Data	751
Chilled Water System Data	752
Loading The Template	753
Save HVAC Template	754
Add Zone Group Tool	756
Add Loop Tool	757
Auto Connect Tool	757
Working At Sub-Loop Level	759
Add Boiler Tool	759
Add Chiller Tool	759
Add Cooling Tower Tool	760
Add Water Heater Tool	760
Add Pump Tool	760

Add Heating Coil Tool	761
Add Cooling Coil Tool	761
Add Humidifier Tool	762
Add Splitter/Add Mixer Tool	762
Add Supply/Return Plenum Tool	763
Add Water Heater Heat Pump	765
Add Setpoint Manager	766
Connect Components Tool	767
Working At HVAC Zone Group Level	768
Working At HVAC Zone Level	768
Add Zone Air Distribution Unit Tool	768
Add Zone Forced Air Unit Tool	769
Add Zone Radiant Surface Tool	770
Add Zone Heated Floor Tool	770
Add Zone Chilled Ceiling Tool	771
Add Zone Radiative-Convective Unit Tool	772
Add Zone High Temperature Radiant Heating	772
Add Zone Water Heater Heat Pump	773
Add Zone Extract Tool	774
Add Zone Exhaust Fan Tool	775
Working At HVAC Component Level	776
Autosizing HVAC Components	776
Autosizing In EnergyPlus	777
HVAC Sizing Tips	779
Differences Between Autosizing Simulations And Cooling Design Calculations	779
Humidity Control	780
VAV With Reheat	780
CAV With Reheat	786
Unitary Heat Cool And Unitary Heat Pump Humidity Control	786
HVAC Components	787
Air Loop Components	787
Air Handling Units	787
Generic Air Handling Unit (AHU)	788
General	788
Operation	789
Night Cycle	789
Extract Fan	790
Mixed Mode	790
Outdoor Air System (Common To All AHUs)	790
DOAS	791
Recirculation	793
Economiser	794
Outdoor Air Schedules	797
Demand Controlled Ventilation	799
Heat Recovery (Common To All AHUs)	800
Pre-treatment Coils	807
Unitary Generic AHU	808
General	809
Operation	810
Control	810
Supply Air Flow	811
Extract Fan	814
DX Coil	814
Compressor	814
Heat Pump	814
On-Cycle And Off-Cycle Power Use	815
Supplemental Heater	815
Mixed Mode Zone Equipment	815
Outdoor Air System	816
Unitary Heat Cool AHU	816
General	816
Operation	816

Supply Air Temperature	817
Supply Air Flow Rates	817
Supply Fan	817
Extract Fan	818
Thermostat Location	818
Dehumidification	818
Night Cycle	818
Mixed Mode	818
Outdoor Air System	819
Unitary Air To Air Heat Pump AHU	819
General	819
Operation	820
Supply Air Flow Rates	820
Supply Fan	820
Extract Fan	821
Thermostat Location	821
Dehumidification	821
Supplemental Heater	821
Night Cycle	822
Mixed Mode	822
Outdoor Air System	822
Unitary Water To Air Heat Pump AHU	822
General	823
Operation	823
Supply Air Flow Rates	823
Supply Fan	823
Thermostat Location	824
On Cycle Power Use	824
Dehumidification	824
Cooling Coil	824
Compressor	825
Supplemental Heater	825
Coil Water Flow Rate	825
Night Cycle	825
Mixed Mode	826
Advanced	826
Outdoor Air System	826
DOAS AHU	826
General	827
Operation	827
Connecting To Standard AHUs	827
Plenum Data	827
Fans	828
Fan - Constant Volume	828
General	829
Flow Rates	829
Motor	830
Operation	830
Algorithm	830
Fan - Constant Volume Outputs	830
Fan - Variable Volume	831
General	831
Flow Rates	832
Motor	832
Fan Coefficients	833
Operation	833
Algorithm	833
Fan - Variable Volume Outputs	834
Fan - On/Off	834
Operation	835
Algorithm	836
Fan - On/Off Outputs	836

Fan - System Model	837
General	837
Electric Power	837
Flow Rates	838
Motor	838
Speed Control	839
Operation	840
Fan - Zone Exhaust	840
Fan Data	842
Operation	844
Humidifier - Steam Electric	844
General	844
Operation	845
Humidifier - Steam Electric Outputs	845
Evaporative Cooler Data	846
Evaporative Cooler - Direct CelDekPad	847
General	848
Settings	849
Operation	849
Evaporative Cooler - Direct Research Special	849
General	849
Settings	850
Recirculating Water	851
Temperature Limits	851
Blowdown	852
Operation	852
Evaporative Cooler - Indirect CelDekPad	852
General	853
Settings	854
Secondary Fan	854
Operation	855
Evaporative Cooler - Indirect Wet Coil	855
General	856
Settings	856
Secondary Fan	857
Blowdown	857
Operation	857
Evaporative Cooler - Indirect Research Special	857
General	858
Settings	858
Recirculating Water	860
Secondary Air	860
Temperature Limits	861
Blowdown	862
Operation	862
Plant Loop Heat Pumps	862
Configuring GSHPs In DesignBuilder	864
Heat Pump - Cooling	866
General	866
General	866
Rated Flow Rate And Capacity	867
Performance Curves	867
Companion Component	867
General	868
Flow Rates And Capacity	868
Performance Curves	869
Condenser	869
Heat Pump - Heating	869
General	869
General	870
Rated Flow Rate And Capacity	870
Performance Curves	870

Companion Component	871
General	871
Flow Rates And Capacity	871
Performance Curves	872
Condenser	872
Generating Heat Pump Coefficients From Manufacturers Data	872
Processing Catalogue Data	873
Reference Condition Explanation And EquationFit Functionality Verification	874
Ground Source Heat Pump (GSHP) Example	875
Air To Water Heat Pump / ASHP Data	884
General	885
Compressor Settings	886
Flow Rate Settings	887
Inlet Air Settings	887
Fan Settings	888
Parasitic Load Settings	888
Air To Water Heat Pump Coil	888
General	889
Evaporator Settings	889
Condenser Settings	890
Crankcase Heater Settings	891
Performance Settings	891
Air To Water Heat Pump Example	893
District Heating	896
General	896
Performance Curves	896
General Tab	897
Linear	897
Quad Linear	898
Quint Linear	901
Quadratic	903
Cubic	904
Exponent	905
Bi-quadratic	906
Bi-cubic	908
Quartic	910
Rectangular Hyperbola 2	911
Double Exponential Decay	912
Curve Plot Tab	913
Performance Curve Coefficient Tools	914
How DesignBuilder Plots Performance Curves	914
Example: Chiller EIR Modifier For Temperature Curve (Biquadratic)	917
Important Checks When Using Any Performance Curve Coefficients Tool	922
Additional Information	923
EnergyPlus Curve Fit Tool	923
Editing Zone Group Data	924
General Tab	924
Heating And Cooling Calculation Sequence Tab	925
Zone Defaults	926
Editing HVAC Zone Data	926
Edit HVAC Zone Data - General Tab	927
General	927
Thermostat Schedules	927
Comfort PMV Setpoint Schedules	927
Humidistat Control	928
CO2 And Contaminant Control	928
Zone Air Distribution Effectiveness	929
Sizing	930
Cooling Sizing	930
Heating Sizing	932
Outdoor Air Sizing	933
Dedicated Outdoor Air System Sizing	934

Load Outdoor Air Model Data	935
Edit HVAC Zone Data - Sequence	936
Load Distribution Scheme	936
Heating Calculation Sequence	936
Cooling Calculation Sequence	936
Sequential Heating Fraction Schedules	937
Sequential Cooling Fraction Schedules	937
Edit HVAC Zone Data - Target	937
Define HVAC Zone Affected By Changes	937
Setpoint Manager	938
1-Scheduled Setpoint Manager	939
2-Warmest Setpoint Manager	939
3-Coldest Setpoint Manager	940
4-Multi-Zone Minimum Humidity Average Setpoint Manager	940
5-Multi-Zone Maximum Humidity Average Setpoint Manager	940
6-Multi-Zone Humidity Minimum Setpoint Manager	941
7-Multi-Zone Humidity Maximum Setpoint Manager	941
8-Single Zone Humidity Minimum Setpoint Manager	942
9-Single Zone Humidity Maximum Setpoint Manager	942
10-Outdoor Air Reset Setpoint Manager	943
11-Single Zone Heating Setpoint Manager	944
12-Single Zone Cooling Setpoint Manager	945
13-Multi Zone Cooling Average Setpoint Manager	945
14-Multi Zone Heating Average Setpoint Manager	946
15-Follow Ground Temperature	947
16-Follow Outdoor Air Temperature	947
17-Dual Setpoint	948
DHW	948
Water Outlet Group	949
Drain Water Heat Exchanger	949
Water Outlet	950
General	951
Flow Rate	951
Heat Gain To Zone	951
Zones Served	951
Water Heater	952
Standard Ratings	953
Water Heater Outputs	953
Water Heater - Mixed	957
General	957
External Heating Plant Connection	958
Temperature Settings	958
Ambient Temperature Settings	958
Internal Heating Element	959
Internal Heating Element Settings	959
On-Cycle Settings	960
Off-Cycle Settings	961
Use-Side Settings	961
Source-Side Settings	961
Sizing	962
Water Heater - Stratified	964
General	964
External Heating Plant Connection	965
Temperature Settings	965
Ambient Heat Transfer Settings	966
Internal Heating Element	967
Internal Heating Element Settings	967
Auxiliary Heating Element	967
Auxiliary Heating Element Settings	968
On-Cycle Settings	968
Off-Cycle Settings	969
Inlet Mode	969

Use-Side Settings	970
Source-Side Settings	970
Stratification Settings	971
Sizing	971
Solar Hot Water	973
Solar Heating System Control	973
Freeze Prevention	975
Additional Controls	976
Solar Collector Data	976
General	976
Solar Collector Surface	976
Control	976
Performance	977
Tempering Valve	978
Plant And Condenser Loop Equipment Data	978
Boilers	979
General	979
Efficiency	980
Water Outlet	982
Part Load Ratios	983
Boiler Outputs	983
Chillers	985
Chiller - Constant CoP	985
General	985
Condenser	987
Flow Rates	987
Basin Heater	987
Generic Chiller Outputs	988
Chiller EIR	991
General	991
Condenser	993
Temperatures	993
Flow Rates	994
Performance Curves	994
Part Load Settings	996
Heat Recovery	997
Generic Chiller Outputs	997
Chiller Electric EIR Outputs	1000
Chiller:Electric:EIR Technical Description	1001
Overview	1001
Model Description	1001
Chiller - Reformulated EIR	1003
General	1004
Temperatures	1005
Flow Rates	1006
Performance Curves	1006
Part Load Settings	1008
Heat Recovery	1009
Chiller - Absorption	1010
General	1010
Condenser	1012
Temperatures	1012
Flow Rates	1012
Part Load Settings	1012
Generator Heat Input Part Load Ratio Curve	1013
Pump Electric Use Part Load Ratio Curve	1013
Generator Hot Water Supply	1013
Chiller - Absorption Indirect	1014
General	1014
Condenser	1015
Temperatures	1016
Flow Rates	1016

Part Load Settings.....	1016
Part-Load Settings Curves	1017
Correction Curves	1017
Generator Hot Water Supply.....	1018
District Cooling	1018
General	1018
Cooling Towers.....	1019
Cooling Tower - Single Speed And Two Speed.....	1021
General	1022
Airflow	1023
Performance Input Method	1023
Water Flow	1024
U-Factor Times Area Value	1024
Capacity	1024
Temperature.....	1024
Free Convection Regime	1025
Basin Heater Settings	1026
Blowdown.....	1026
Capacity Control.....	1027
Multi-Cell Tower Settings	1027
Cooling Tower - Variable Speed Merkel	1027
General	1028
Airflow	1029
Performance Input Method	1029
Water Flow	1029
U-Factor Times Area Value	1029
Capacity	1030
Fan Power.....	1030
Temperature.....	1031
Free Convection Regime	1031
Basin Heater Settings	1032
Blowdown.....	1033
Multi-Cell Tower Settings	1033
Cooling Tower - Variable Speed.....	1034
General	1034
Airflow	1035
Model Type	1035
Water Flow	1036
Fan Power.....	1036
Temperature.....	1036
Free Convection Regime	1037
Basin Heater Settings	1037
Blowdown.....	1037
Multi-Cell Tower Settings	1038
Cooling Tower - Model Coefficients	1038
Cool Tools	1039
York Calc.....	1040
Fluid Cooler.....	1042
General	1043
Fan Settings	1044
High Speed Fan Settings	1044
Low Speed Fan Settings	1045
Pumps.....	1046
Pump - Constant Speed	1046
General	1046
Pump Settings.....	1046
Pump - Constant Speed Outputs	1047
Pump - Variable Speed.....	1048
General	1048
Pump Settings.....	1048
Part Load Performance	1049
Pump Coefficients	1049

Pump - Variable Speed Outputs	1050
Ground Heat Exchanger	1050
Ground Heat Exchanger - Vertical	1050
General	1052
Ground Heat Exchanger Template	1052
Ground Heat Exchanger Type	1052
Ground Properties	1052
Flow Rate	1052
Borehole And Pipe Geometry	1053
Grout And Pipe Properties	1053
Other	1053
G Function Data Pairs	1054
Sizing Vertical Ground Heat Exchangers	1055
Import Vertical Ground Heat Exchanger Data From IDF	1057
Ground Heat Exchanger - Pond	1066
General	1067
Ground Heat Exchanger Template	1067
Ground Heat Exchanger Type	1067
Ground Heat Exchanger - Pond	1067
Ground Heat Exchanger - Surface	1068
General	1069
Ground Heat Exchanger Template	1069
Ground Heat Exchanger Type	1070
Ground Heat Exchanger - Surface	1070
VRF Outdoor Unit	1070
VRF Outdoor Unit - General Tab	1072
General	1072
Control	1073
Piping	1073
Compressor	1075
Defrost	1075
Condenser	1076
VRF Outdoor Unit - Cooling Tab	1078
General	1078
Cooling Capacity Ratio Modifier Function Of Temperature Curves	1079
Cooling Energy Input Ratio (EIR) Modifier Function Of Temperature Curves	1080
Other Curves	1081
VRF Outdoor Unit - Heating Tab	1081
General	1081
Heating Capacity Ratio Modifier Function Of Temperature Curves	1082
Heating Energy Input Ratio (EIR) Modifier Function Of Temperature Curves	1083
Heating Energy Input Ratio Modifier Function Of Part-Load Ratio Curves	1084
Other Curves	1084
VRF Outdoor Unit - Heat Recovery Tab	1084
General	1084
Cooling	1085
Heating	1086
Fluid To Fluid Heat Exchanger	1087
General	1092
Operation	1092
Heat Exchanger Settings	1092
Temperature Limits	1094
Sizing Factor	1095
Chilled Water Storage	1095
Chilled Water Storage - Mixed	1101
General	1101
Temperature Settings	1102
Ambient Heat Transfer Settings	1102
Use Side Settings	1102
Source Side Settings	1103
Chilled Water Storage - Stratified	1103
General	1103

Temperature Settings.....	1104
Ambient Heat Transfer Settings.....	1105
Use Side Settings.....	1105
Source Side Settings.....	1106
Inlet Mode	1106
Stratification Settings	1106
Ice Thermal Storage	1107
General	1107
Operation	1108
Curve Timestep.....	1108
Charging Settings.....	1109
Discharging Settings	1109
Parallel Or Series Configuration With Chiller	1109
Controls And Chilled Water Plant Loop Operation Scheme Settings	1110
Zone Unit Data	1112
Air Distribution Units.....	1112
Target Tab.....	1113
Constant Volume No Reheat ADU	1113
General	1114
Outdoor Air	1114
Operation	1114
Constant Air Volume With Reheat ADU	1114
General	1115
Heating Coil.....	1115
Operation	1115
Advanced	1115
Variable Air Volume With Reheat ADU	1116
General	1116
Air Flow	1118
Heating Coil.....	1118
Outdoor Air	1119
Operation	1119
Advanced	1120
Variable Air Volume With Reheat And Variable Speed Fan ADU	1120
General	1120
Air Flow	1120
Heating Coil.....	1121
Operation	1121
Advanced	1121
Variable Air Volume With No Reheat ADU	1121
General	1122
Outdoor Air	1123
Operation	1123
Series Powered Induction Unit With Reheat ADU	1123
General	1123
Air Flow	1124
Heating Coil.....	1124
Operation	1124
Advanced	1124
Parallel Powered Induction Unit With Reheat ADU	1124
General	1125
Supply Air Flow Rates.....	1125
Heating Coil.....	1126
Operation	1126
Advanced	1126
Four Pipe Induction Unit ADU	1126
General	1127
Heating Coil.....	1127
Cooling Coil.....	1127
Operation	1128
Advanced	1128
Dual Duct Constant Air Volume ADU	1128

General	1128
Operation	1129
Dual Duct Variable Air Volume ADU	1129
General	1129
Operation	1129
Cooled Beam	1130
General	1132
Performance Parameters	1133
Operation	1134
Variable Air Volume Outdoor Air ADU	1134
General	1135
Operation	1135
Radiant Surface Data	1135
Heated Floors	1136
Chilled Ceilings	1139
Target Tab	1142
Radiant Surface - Constant Flow	1143
General	1143
Tube Settings	1143
Flow Settings	1144
Pump Settings	1144
Operation	1144
Heating	1145
Control	1145
Cooling	1146
Control	1147
Condensation Control	1148
Radiant Surface - Variable Flow	1148
Tube Settings	1149
Heating	1150
Control	1150
Cooling	1151
Control	1152
Condensation Control	1153
Operation	1154
Checking Sizing Results For Design	1154
Radiant Surface - Electric	1154
General	1154
Operation	1155
Heating	1155
Heated Floor	1156
Target Tab	1157
Troubleshooting Heated Floors	1158
Coverings Suitable For Heated Floors	1159
Heated Floor - Constant Flow	1160
General	1160
Tube Settings	1161
Flow Settings	1162
Pump Settings	1162
Control	1162
Operation	1163
Heated Floor - Variable Flow	1163
Tube Settings	1164
Flow Settings	1165
Control	1165
Operation	1166
Chilled Ceiling	1166
Target Tab	1169
Troubleshooting Chilled Ceilings	1169
Chilled Ceiling - Constant Flow	1170
General	1170
Tube Settings	1171

Flow Settings.....	1172
Pump Settings.....	1172
Control.....	1172
Condensation Control	1173
Operation	1174
Chilled Ceiling - Variable Flow	1174
General	1174
Tube Settings.....	1175
Flow Settings.....	1176
Control.....	1176
Condensation Control	1177
Operation	1177
Radiative-Convective Heater Data	1178
Target Tab.....	1178
Hot Water Convector.....	1178
General	1178
Operation	1179
Electric Convector	1179
Operation	1180
Hot Water Radiator	1181
General	1181
Operation	1182
Advanced	1182
Electric Radiator	1183
General	1183
Operation	1184
High Temperature Radiant Heater	1184
General	1184
Operation	1185
Heating.....	1185
Zone Forced Air Units	1187
Target Tab.....	1187
Fan Coil Unit - Four Pipe.....	1187
Outdoor Air Supply	1189
Cooling Coil.....	1189
Heating Coil.....	1190
Advanced	1190
Fan Coil Unit - Four Pipe Outputs	1190
Packaged Terminal Heat Pump	1191
Fan	1192
Supply Air Flow Rates.....	1193
Outdoor Air Supply	1193
Compressor.....	1194
Supplemental Heater	1194
Operation	1195
Advanced	1195
Packaged Terminal Heat Pump Outputs	1195
Packaged Terminal Air Conditioner.....	1198
Fan	1199
Supply Air Flow Rates.....	1200
Outdoor Air	1200
Operation	1201
Packaged Terminal Air Conditioner Outputs	1201
Zone Water To Air Heat Pump.....	1203
Application.....	1203
General	1206
Operation	1206
Fan	1206
Supply Air Flow Rate.....	1207
Outdoor Air Supply	1207
On Cycle Power Use.....	1208
Cooling Coil.....	1208

Compressor.....	1208
Supplemental Heater	1208
Coil Water Flow Mode.....	1209
VRF Indoor Unit	1209
General	1210
Operation	1210
Supply Air Flow Rates.....	1210
Outdoor Air Supply.....	1210
Parasitic Electric Energy	1211
Supply Fan	1211
Heating Capacity.....	1211
Heating And Cooling Coil Data	1212
Heating Coils.....	1212
Heating Coil - Water.....	1212
General	1213
Water Temperatures	1214
Air Temperatures	1214
Operation	1214
Controller.....	1214
Heating Coil - Water Outputs	1215
Heating Coil - Electric.....	1216
General	1216
Operation	1216
Heating Coil - Electric Outputs	1217
Heating Coil - Fuel	1217
Part Load Performance	1218
Operation	1218
Heating Coil - DX.....	1219
General	1219
Compressor.....	1220
Defrost.....	1221
Performance Curves	1221
Crankcase Heater	1223
Operation	1223
Heating Coil - DX Outputs.....	1223
Heating Coil - VRF - DX	1224
General	1225
Operation	1225
Airflow	1225
Capacity	1225
Heating Coil - Water To Air Heat Pump	1226
General	1226
Equation Fit Coil Type Settings	1227
General	1227
Temperatures.....	1227
Performance Curves	1228
Parameter Estimation Coil Type Settings	1228
Cooling Coils.....	1228
Cooling Coil - Water	1228
General	1229
Operation	1230
Controller.....	1230
Cooling Coil - Water Outputs	1231
Single Speed DX Cooling Coil.....	1232
General	1233
Operation	1234
Performance Curves	1234
Latent Capacity Degradation	1235
Condenser.....	1236
Evaporative Condenser	1236
Crankcase Heater	1237
Basin Heater	1238

Run Settings	1238
DOAS Settings	1239
DX Cooling Coil Outputs	1239
Cooling Coil - Two Stage With Humidity Control DX	1240
General	1240
Operation	1241
Normal Mode Stage 1 Coil Performance	1241
Normal Mode Stage 1+2 Coil Performance	1241
Dehumidification Mode 1 Stage 1 Coil Performance	1241
Dehumidification Mode 1 Stage 1+2 Coil Performance	1242
Crankcase Heater	1242
Basin Heater	1242
Run Settings	1242
DX Cooling Coil Outputs	1243
Cooling Coil - VRF - DX	1243
General	1243
Operation	1243
Airflow	1243
Capacity	1244
Cooling Coil - Water To Air Heat Pump	1244
General	1245
Temperatures	1245
Performance Curves	1246
Others	1246
Parameter Estimation Coil Type Settings	1246
Generators	1247
Generator - Internal Combustion Engine	1247
General	1248
Operation	1248
Power Output	1248
Fuel	1249
Part Load Ratios	1249
Exhaust	1250
Exhaust U-Factor Times Area Curve	1250
Heat Recovery	1250
Generator - Micro Turbine	1251
General	1251
Operation	1252
Power Output	1252
Power Consumption	1253
Fuel	1253
Electrical Efficiency	1254
Combustion Air	1254
Exhaust	1255
Heat Recovery	1255
Generators - Modelling Guide	1257
Model Options	1262
Data - Model Options (Building)	1263
Construction And Glazing Model Option	1264
Gains Data Model Option	1264
Timing Model Option	1267
HVAC Model Options	1267
Natural Ventilation And Infiltration Model Options	1272
Imported Surfaces	1274
Advanced - Model Options (Building)	1274
Simplification	1274
Adjacency Settings	1276
Natural Ventilation Options	1276
Calculated Natural Ventilation	1277
Scheduled Natural Ventilation	1277
Lighting	1277
Filters	1278

Component Blocks - Model Options	1278
Heating Design Calculation Options - Model Options (Building)	1279
Winter Design Day	1279
Operation For 7/12 And Typical Workday Schedules	1279
Calculation Options - Heating Design	1280
Calculation Options	1280
System Sizing	1281
Output Options - Heating Design	1281
Advanced Calculation Options - Heating Design	1282
General Solution	1282
Other	1283
Cooling Design Calculation Options - Model Options (Building)	1283
Calculation Options - Cooling Design	1283
Calculation Description	1283
Calculation Options	1284
Summer Design Day	1285
Cooling System Sizing	1286
ASHRAE Cooling Sizing	1287
Supply Air Conditions.....	1287
Unitary DX Cooling Sizing	1287
Solar Options - Cooling Design	1288
Output Options - Cooling Design	1288
Advanced Calculation Options - Cooling Design	1289
General Solution	1289
Convection	1290
Shading	1290
Other	1290
Simulation Calculation Options - Model Options (Building)	1291
Simulation Calculation Options - Simulation Options	1291
Calculation Description	1291
Simulation Period	1291
Output Intervals For Reporting.....	1293
Simulation Calculation Options - Calculation Options	1293
Simulation Calculation Options - Solar	1296
Shadow Calculations.....	1300
Simulation Calculation Options - Advanced	1301
General Solution	1301
Finite Difference Settings	1302
Airflow Network	1303
Convection	1303
Warmup.....	1303
Shading.....	1304
Include IDF Data	1305
Other	1306
Simulation Calculation Options - Output	1306
Output Data.....	1306
Graphable Outputs - Energy	1307
Graphable Outputs - Comfort And Environmental	1308
Detailed Daylight Outputs	1311
Summary Outputs	1312
Miscellaneous Outputs.....	1324
Time Setpoints Not Met Tolerances.....	1324
Display - Model Options (Site)	1325
Display Model Options - Show Data For Calculations	1325
Display Model Options - Edit Screen	1326
Display Model Options - Visualisation	1342
Drawing Tools - Model Options (Site)	1343
Drawing Tools - Draw Block	1344
Drawing Tools - Direction Snaps	1344
Drawing Tools - Point Snaps	1344
Drawing Tools - Drawing Guides	1344
Drawing Tools - Object Rotation	1344

Drawing Tools - Protractor Tool	1344
Drawing Tools - Miscellaneous Edit	1345
Block - Model Options (Block)	1345
Block Geometry	1346
Transition Angles	1347
Project Details - Model Options (Building)	1347
Project Details	1348
Owner Details	1348
Analyst Details	1348
Notes	1348
Carbon - Model Options (Building)	1348
LCA	1348
Glazing Carbon Model	1348
Renewables	1349
One Click LCA	1349
Interzone Airflow	1349
Add An Airflow Path	1350
Editing Existing Airflow Paths	1351
Outputs	1351
Components And Templates	1351
Components	1353
Construction Components	1354
Construction Layer Data	1354
General	1355
Definition	1355
Calculation Settings	1355
Layers	1356
Photovoltaic Panel	1358
Bridging	1359
C-Factor And F-Factor Constructions	1362
C-Factor (Underground Wall)	1363
F-Factor (Ground Floor)	1363
Surface Properties	1363
Construction Image	1364
Construction Calculated Data	1364
Construction Cost Data	1366
Internal Source	1366
Condensation Analysis	1368
Options	1368
Example Output	1370
Boundary Data	1372
Materials	1372
Material Thermophysical Properties	1373
Material Thickness	1373
Detailed Properties	1373
Simple Resistance	1374
Vapour Resistance	1374
Surface Properties	1375
Radiance Daylighting	1376
Green Roof	1377
Green Roof Data	1379
Embodied Carbon	1387
Embodied Carbon Report	1388
Phase Change Materials	1388
Solution Algorithm	1389
Phase Change Material - Basic	1390
Phase Change Properties	1390
Temperature-Enthalpy Curves	1390
Phase Change Material - Hysteresis	1392
Phase Change Properties	1392
Phase Change Hysteresis	1392
Liquid State	1392

Melting Curve	1393
Solid State	1393
Freezing Curve	1393
Moisture Transfer	1393
Material Cost Data	1394
Glazing Components	1394
Glazing Layer Data	1395
General	1395
Definition Method	1395
Layers	1396
Photovoltaic Panel	1396
Radiance Daylighting	1397
Glazing Calculated Data	1400
Simple Glazing Definition	1401
Glazing Cost Data	1402
Pane Components	1402
Panes - General	1403
Panes - Thermal Data	1403
Panes - Solar Data	1404
Panes - Visible Data	1404
Panes - Long-wave (Infra-red) Properties	1404
Spectral Data	1405
Glass Optical Properties Conversion	1406
Load Glass Data From Berkeley Lab WINDOW	1406
Importing New IGDB Data	1408
Pane Group Components	1408
Schedules	1409
7/12 Schedules - Defined Using Profiles	1410
Compact Schedules	1412
Field-Set (Through, For, Interpolate, Until, Value)	1413
Day Schedule	1416
Defining Setpoint Temperature And RH Schedules	1417
Schedules Timing Model Option	1417
Typical Workday Timing Model Option	1417
Schedules And Design And Sizing Simulations	1418
Heating And Cooling Design Calculation	1418
HVAC Autosizing Simulations	1418
Simulation Hourly Weather Data	1418
Hourly Weather Data - General	1420
Hourly Weather Data - Statistics	1420
Add New Hourly Weather Data	1420
Add Hourly Weather Data To The Library	1421
Add Hourly Weather Data To The Model	1423
Weather Data Analysis	1424
Input Data	1425
Options	1426
Modify	1427
Edit/Review	1428
Converting From Other Formats	1428
Hourly Weather Data Report Files	1429
Audit / Log File (.audit)	1429
Statistical Report File (.stat)	1430
Design Day Calculations Output (.ddy)	1434
Sources Of EnergyPlus Hourly Weather Data	1436
Other Sources	1437
Profiles	1437
Textures	1438
Window Gas	1439
Custom Gases Only	1440
Metabolic Rates	1441
Local Shading Components	1441
Local Shading - General	1442

Local Shading - Louvres	1443
Local Shading - Sidefins	1443
Local Shading - Overhangs	1444
Window Blinds	1445
Window Blinds - General	1445
Window Blinds - Slat Data	1446
Slat Properties.....	1446
Slat Beam Solar Properties.....	1447
Slat Diffuse Solar Properties.....	1448
Slat Beam Visible Properties	1448
Slat Diffuse Visible Properties.....	1448
Slat Infra Red Properties.....	1449
Openings.....	1449
Window Blinds - Shade Data	1449
Shade Properties	1450
Openings.....	1451
Window Blinds - Electrochromic Switching	1453
SageGlass Electrochromic	1453
Vent Components	1454
Vents - General	1454
General	1454
Visual	1455
Conduction	1455
Vents - Airflow.....	1455
Airflow	1455
Holiday Schedule.....	1455
Holiday Schedule - General	1456
Holiday Schedule - List	1456
Ground Domain.....	1456
Ground Domain - General Tab	1456
Slab	1459
Basement	1460
Ground Domain - Soil.....	1460
Soil Thermal Properties	1460
Soil Moisture Content.....	1460
Ground Domain - Ground Undisturbed Temperatures	1460
Finite Difference	1461
Kusuda-Achebach.....	1461
Xing	1462
Ground Domain - Insulation	1462
Horizontal Insulation.....	1463
Vertical Insulation	1463
Kiva Foundation.....	1463
Technical Description.....	1463
General	1465
Initial Indoor Temperature	1466
Interior Horizontal Insulation	1467
Interior Vertical Insulation	1467
Exterior Horizontal Insulation	1468
Exterior Vertical Insulation	1469
Footing Wall	1469
Custom Blocks	1471
Kiva Foundation Example Cases	1473
Raft Slab	1473
Foundation Footings	1474
Slab-on-Grade Foundation With Additional Perimeter Insulation	1475
Kiva Foundation Settings	1475
General	1475
Soil Thermal Properties	1476
Surface Properties	1476
Extent	1476
Options.....	1477

Moisture Transfer Data	1477
General	1478
EMPD	1478
Moisture Equilibrium Constants	1480
HAMT Settings	1480
Sorption Isotherm	1481
Suction	1481
Redistribution	1481
Diffusion	1482
Thermal Conductivity	1482
HAMT Limitations	1483
Fuel Emission Factors	1483
General	1483
Electricity, Natural Gas, Fuel Oil 1 Etc Tabs	1483
Emission Factors	1485
Fuel Emissions Library Data	1488
Outputs	1488
Horizon Component	1489
Horizon	1489
Currency	1490
General	1490
Templates	1491
Activity Templates	1493
Activity Templates - General	1493
Activity Templates - All Gains	1493
Activity Templates - Occupancy	1494
Activity Templates - Other Gains	1494
Activity Templates - DHW	1494
Activity Templates - Environmental/Comfort	1494
Construction Templates	1495
Construction Templates - General	1495
Construction Templates - Simple	1495
Construction Templates - Walls	1495
Construction Templates - Roofs/Floors/Slabs/Ceilings	1496
Construction Templates - Doors	1496
Construction Templates - Airtightness	1496
Construction Templates - Thermal Mass	1496
Glazing Templates	1497
Glazing Templates - General	1497
Glazing Templates - Glazing	1497
Glazing Templates - Shading	1497
Facade Templates (Types)	1497
Facade Templates - General	1498
Facade Templates - Openings	1498
Lighting Templates	1498
Lighting Templates - General	1499
Lighting Templates - Output	1499
Lighting Templates - Control	1499
HVAC Templates	1500
HVAC Templates - General	1501
HVAC Templates - Ventilation	1501
Natural Ventilation	1501
Mechanical Ventilation	1502
HVAC Templates - Heating And Cooling	1503
Heating	1503
Heating Supply Air Conditions	1504
Cooling	1504
Cooling Supply Air Conditions	1504
HVAC Templates - Humidity Control	1505
HVAC Templates - Air Temperature Distribution	1505
HVAC Templates - Cost	1506
Location Templates	1506

Location Templates - Location	1507
Location Templates - Winter Design Weather	1507
Location Templates - Summer Design Weather	1507
Location Templates - Simulation Weather	1507
Crack Templates	1508
Crack Templates - General	1508
Crack Templates - Openings	1508
Crack Templates - Walls	1509
Crack Templates - Floors/Ceilings	1509
Crack Templates - Roofs	1509
Legislative Region Templates	1509
Legislative Region Templates - General	1510
Legislative Region Templates - Standard Insulation	1510
Regions, Energy Codes, Construction Templates And Constructions - How They Interact	1510
Legislative Region Templates - Emissions	1511
CO2 Emissions	1511
Energy Code Templates	1512
Energy Code Templates - General	1512
Energy Code Templates - Envelope (component)	1512
Energy Code Templates - Envelope (building)	1513
Energy Code Templates - Energy Performance	1513
Sector Templates	1513
Times Zone Templates	1513
Wind Pressure Coefficient Templates	1514
Import Components/Templates	1514
Export Components/Templates	1515
Export Limitations	1516
Library Management	1517
Opening Screen - Component Libraries	1517
Template Library Management	1519
Backup Component/Template Libraries	1520
Restore Component/Template Libraries	1520
Outputs And Calculations	1523
Visualisation	1525
Visualisation - Model Data	1525
Constructions And Glazing	1525
Linear Thermal Bridging	1529
Visualisation - Rendered View	1530
Visualisation Controls	1530
Sunpath	1531
Show Shadows, Time, Day, Month	1533
Show North Arrow	1535
Show Ground Plane	1535
Field Of View	1536
Antialiasing	1538
Dynamics	1539
Visualisation Rendered View Limitations	1540
Section Cut And Clipping Plane	1540
Show Surface Edge Lines	1541
Show Surrounding Blocks/Zones/Surfaces	1545
Creating Movies	1548
Viewing AVI Files	1549
Heating Design Calculation	1549
Heating Design Calculation Options Dialog	1550
Calculation Options - Heating Design	1550
Calculation Options	1551
System Sizing	1551
Output Options - Heating Design	1552
Advanced Calculation Options - Heating Design	1553
General Solution	1553
Other	1553
Heat Loss Breakdown Results	1553

Reviewing Heating Design Output.....	1554
Data Options	1554
Y-Axis Options	1555
Appearance.....	1556
EnergyPlus IDF Input Data	1556
Heating Design Summary Table.....	1556
Summary Display Options	1556
Summary Results	1557
Cooling Design Calculation	1557
Cooling Design Calculation Options Dialog	1558
Calculation Options - Cooling Design	1559
Calculation Description	1559
Calculation Options	1559
Summer Design Day	1560
Cooling System Sizing	1561
ASHRAE Cooling Sizing	1562
Supply Air Conditions.....	1562
Unitary DX Cooling Sizing	1562
Solar Options - Cooling Design	1563
Output Options - Cooling Design.....	1564
Advanced Calculation Options - Cooling Design.....	1565
General Solution	1565
Convection	1565
Shading	1565
Other	1566
Cooling Design Detailed Results	1566
Reviewing Cooling Design Output.....	1568
Data Options	1568
Y-Axis Options	1569
Appearance.....	1570
EnergyPlus IDF Input Data	1570
Summary Cooling Design Table	1570
Summary Display Options	1570
Summary Results	1571
Simulation Using Hourly Weather Data.....	1574
Simulation Calculation Options Dialog	1574
Simulation Calculation Options - Simulation Options	1575
Calculation Description	1575
Simulation Period	1575
Output Intervals For Reporting.....	1576
Options - Simulation.....	1576
Simulation Calculation Options - Calculation Options	1577
Simulation Calculation Options - Solar.....	1580
Shadow Calculations.....	1583
Simulation Calculation Options - Advanced.....	1585
General Solution	1585
Finite Difference Settings	1585
Airflow Network	1586
Convection	1587
Warmup.....	1587
Shading	1588
Include IDF Data	1588
Other	1589
Simulation Calculation Options - Output	1589
Output Data.....	1589
Graphable Outputs - Energy	1590
Graphable Outputs - Comfort And Environmental	1591
Detailed Daylight Outputs	1595
Summary Outputs	1596
Miscellaneous Outputs.....	1608
Time Setpoints Not Met Tolerances.....	1608
Simulation Manager.....	1609

Version Of EnergyPlus.....	1611
Simulation Manager Dialog	1612
Simulation Servers Dialog.....	1615
Installing The Simulation Manager On A Server	1616
Install The Simulation Manager Server.....	1617
Connect The Simulation Manager On The Client Machine Running DesignBuilder	1617
JESS Online	1618
How To Access JESS	1619
Simulation Display Options	1619
Data Options	1620
Y-Axis Options	1621
Appearance.....	1622
EnergyPlus IDF Input Data	1622
Simulation Detailed Results	1622
Building Level Only	1626
Surface And Opening Levels	1627
Hourly And Sub-Hourly Results Times	1628
Comfort Analysis	1628
Temperature Distribution	1629
Simplified ASHRAE 55-2004 Graph Related Outputs	1631
CIBSE TM52 Overheating Assessments	1632
CIBSE TM59 - Modelling And Analysis Guide	1634
CIBSE TM59:2017 - Modelling And Analysis Guide	1636
Overview	1636
TM59 Modelling Process	1637
Running The TM59 Analysis.....	1645
TM59:2017 Outputs	1648
CIBSE TM59:2026 - Modelling And Analysis Guide	1650
Overview	1650
TM59:2026 Modelling Process	1651
Assessment Criteria (A To D)	1661
Running The TM59 Analysis.....	1663
TM59:2026 Output Report	1665
Simulation Summary Output	1667
Parametric Analysis	1668
Parametric Analysis Contents.....	1670
Parametric Analysis - Parametric Settings Dialog	1670
Analysis Type Tab.....	1670
Outputs Tab	1670
Variables Tab	1671
Parametric Analysis - Analysis Type	1671
Analysis Type.....	1671
Parametric Analysis - Outputs Dialog	1672
Parametric Analysis - Design Variables Dialog.....	1672
Design Variable.....	1673
List Variable Option Ordering.....	1674
Target.....	1674
Key.....	1674
Parametric Analysis Calculation Options	1675
Parametric Analysis Calculation Options - General Tab	1675
Calculation Description	1675
Parametric Options	1675
Advanced	1676
Parametric Analysis Graph	1677
Parametric Analysis Calculation Options - Simulation Manager Tab	1677
Parametric Analysis Results	1677
Parametric Analysis Display Options	1678
Export Results.....	1679
Parametric Analysis Limitations	1679
Parametric Analysis Tutorials	1679
Parametric Analysis - Basic Tutorial	1679
Parametric Analysis Tutorial - Future Climate Study	1684

Parametric Analysis Tutorial - Custom Script KPI - TM59 Comfort	1690
Step 1 - Create A Base Model	1690
Step 2 – Run A Simulation To Check The Base Model	1693
Step 3 – Adding Parametric Design Variable	1693
Step 4 – Set Up Parametric Outputs.....	1694
Step 5 – Write Python Script For The Custom Script KPI.....	1695
Step 6 - Re-Run A Standard Simulation	1697
Step 7 - Run The Parametric Simulation	1697
Parametric Analysis Tutorial - Custom Script Variable - Advanced Parametric Facade.....	1698
Optimisation	1703
Optimisation - Parametric Settings Dialog	1705
Analysis Type Tab.....	1705
Objectives Tab	1705
Additional Outputs Tab	1706
Constraints Tab.....	1706
Variables Tab	1707
Optimisation - Analysis Type	1708
Optimisation - Objectives Dialog	1709
Optimisation - Additional Outputs Dialog	1709
Optimisation - Constraints Dialog	1709
Optimisation - Design Variables Dialog	1710
List Variable Option Ordering.....	1711
Target.....	1711
Key	1712
List Of Variables	1712
Variables List.....	1712
Summary Outputs (KPIs)	1717
List Of KPIs	1718
Optimisation Calculation Options Dialog	1722
Optimisation Calculation Options Dialog - General Tab	1723
Calculation Description	1723
Optimisation Options	1723
Advanced Options	1725
Discussion On Selecting Settings	1729
Multi-Objective Pareto Scatter Graph	1729
Single Objective Graph	1729
Optimisation Calculation Options - Simulation Manager Tab	1729
Optimisation Results.....	1730
Optimisation Limitations	1731
Optimisation Tips And Troubleshooting.....	1731
Tips.....	1731
Troubleshooting	1732
Optimisation Technical Background	1733
How It Works.....	1733
Constraints	1734
Pareto Archived Elitism	1735
Bibliography	1735
Optimisation Tutorials.....	1736
Optimisation Basic Tutorial.....	1736
Optimisation Tutorial - Custom EMS Variable	1739
Step 1 - Create A Base Model	1739
Step 2 - Set Up The Optimum Start Control Script	1740
Step 3 - Run Standard Simulation	1741
Step 4 - Make Optimisation Objective Settings	1741
Step 5 - Make Optimisation Variable Settings	1742
Step 6 - Re-Run A Standard Simulation	1742
Step 7 - Run Optimisation	1743
Step 7 - Analyse Results.....	1743
Optimisation Tutorial - Custom Script Variable	1745
Step 1 - Create A Base Model	1745
Step 2 - Make Optimisation Objective Settings	1746
Step 3 - Make Optimisation Variable Settings	1746

Step 4 - Write Script For Custom Script Variables	1747
Step 5 - Run A Standard Simulation	1750
Step 6 - Run Optimisation	1750
Step 7 - Analyse Results	1751
Optimisation Tutorial - Custom Script KPI Example 1 - Db_eplusout_reader Python Libraries	1752
Step 1 - Create A Base Model	1753
Step 2 - Run A Standard Simulation	1754
Step 3 - Make Optimisation Objective Settings	1754
Step 4 - Make Optimisation Variable Settings	1756
Step 5 - Write Custom Script	1757
Step 6 - Re-Run A Standard Simulation	1758
Step 7 - Run The Optimisation	1758
Optimisation Tutorial - Custom Script KPI Example 2 - Reading The EUI KPI Using EpNet (C#) And Eppy (Python)	1760
Step 1 - Create A Base Model	1761
Step 2 - Run A Standard Simulation	1762
Step 3 - Make Optimisation Objective Settings	1762
Step 4 - Make Optimisation Variable Settings	1763
Step 5 - Write Custom KPI Script	1765
Step 6 - Re-Run A Standard Simulation	1766
Step 7 - Run The Optimisation	1766
Optimisation Tutorial - Custom Script KPI Example 3 - Using EMS + Python Scripts To Optimise Controls For Comfort And Heating	1770
Step 1 - Create A Base Model	1770
Step 2 - Run A Simulation To Check The Base Model	1772
Step 3 - Make Optimisation Objective Settings	1772
Step 4 - Make Optimisation Variable Settings	1774
Step 5 - Write Custom EMS Script To Calculate Fanger % Discomfort Area	1774
Step 6 - Write Python Script For The Custom Script KPI	1776
Step 6 - Re-Run A Standard Simulation	1777
Step 7 - Run The Optimisation	1777
Optimisation Tutorial - Custom Script KPI Example 4 - Adding Python Script For TM52 Comfort To Example 3	1778
Step 1 - Switch On TM52 Outputs	1778
Step 2 - Add Constraint Setting	1779
Step 3 - Write Python Script For The Custom Script KPI	1779
Step 6 - Re-Run A Standard Simulation	1781
Step 7 - Run The Optimisation	1782
Uncertainty And Sensitivity Analysis	1783
Overview	1783
Core Methodology For Uncertainty And Sensitivity Analysis	1783
DesignBuilder Implementation	1784
Uncertainty And Sensitivity Analysis Contents	1786
Uncertainty And Sensitivity Analysis - Parametric Settings Dialog	1786
Analysis Type Tab	1786
Outputs Tab	1786
Design Variables Tab	1787
Uncertainty And Sensitivity Analysis - Analysis Type	1788
Analysis Type	1788
Uncertainty And Sensitivity Analysis - Outputs Dialog	1790
Outputs	1790
Uncertainty And Sensitivity Analysis - Design Variables Dialog	1791
Design Variables	1791
Distribution Parameters	1796
Transform Distribution	1807
Variable Values	1846
Target	1847
Graph	1848
Uncertainty And Sensitivity Analysis - Calculation Options Dialog	1848
Uncertainty And Sensitivity Analysis - Calculation Options General Tab	1848
Calculation Description	1849

Uncertainty/Sensitivity Analysis Options	1849
Uncertainty/Sensitivity Analysis Options	1850
Advanced Options	1852
Discussion On Selecting Settings	1853
Uncertainty And Sensitivity Analysis - Calculation Options Simulation Manager Tab	1853
Uncertainty And Sensitivity Results	1853
Display Options: Uncertainty/Sensitivity Analysis	1853
Uncertainty Analysis Outputs	1854
Histogram Display Options	1855
Uncertainty Analysis Report	1855
Sensitivity Analysis Outputs - Standardised Regression Coefficient	1857
Graph Controls	1857
Sensitivity Analysis Report	1857
Checking Regression Results	1866
Uncertainty And Sensitivity Analysis - Tips And Tricks	1867
Speeding Up Uncertainty And Sensitivity Analysis	1867
Running Sensitivity Analysis As A Precursor To Optimisation	1867
Ensure Inheritance Path	1868
Switching On Shading When Shading System Type Is A Variable	1868
Uncertainty And Sensitivity Analysis Tutorials	1868
Uncertainty And Sensitivity Analysis - Basic Tutorial	1868
Step 1 - Create A Base Model	1869
Step 2 - Run A Standard Simulation	1869
Step 3 - Define Problem On Uncertainty And Sensitivity Analysis Settings Dialog	1870
Step 4 - Open Calculation Options Dialog	1873
Step 5 - Start Uncertainty And Sensitivity Analysis	1874
Step 6 - Analyse Results	1874
Uncertainty And Sensitivity Analysis - Advanced Tutorial	1878
Step 1 - Create A Base Model	1879
Step 2 - Run A Standard Simulation	1880
Step 3 - Define Problem On Uncertainty And Sensitivity Analysis Settings Dialog	1880
Step 4 - Open Calculation Options Dialog	1894
Step 5 - Start Uncertainty And Sensitivity Analysis	1894
Step 6 - Analyse Results	1895
ASHRAE 90.1 Reporting	1898
Data Visualisation	1898
Legend	1904
Airflow Visualisation Controls	1905
Using The Data Visualisation Tools	1906
Sub-Dividing Surfaces	1907
Computational Fluid Dynamics (CFD)	1909
Creating Model Geometry For CFD	1913
Finite Volume Grid And Geometric Modelling Considerations	1913
Building Levels And CFD Analyses	1920
CFD Boundary Conditions	1920
Boundary Conditions For Internal CFD Analyses	1921
Default Wall And Window Temperatures	1921
Zone And Component Block Surface Boundaries	1922
Component Blocks & Assemblies As CFD Boundaries	1922
Advanced	1924
Non-Orthogonal Boundary Settings	1924
New External CFD Analysis	1926
Wind	1927
Site Domain Factors	1928
New Internal CFD Analysis	1929
Editing The CFD Grid	1930
CFD Grid Information	1932
Setting Up CFD Cell Monitor Points	1933
CFD Calculation Options	1935
Buttons	1936
Turbulence Model	1936
Discretisation Scheme	1936

Iterations	1937
Isothermal	1937
Surface Heat Transfer	1937
Surface Heat Transfer Coefficients	1937
Initial Conditions	1937
Cell Monitor	1938
Residual Display	1938
Dependent Variable Control Settings	1938
Residuals And Cell Monitor Graphs	1939
Convergence Difficulties And Troubleshooting	1939
Displaying Results	1940
CFD Comfort Calculations	1943
Comfort Calculation Options	1943
Comfort Results	1944
E Age Of Air And Air Change Effectiveness	1945
CFD Analysis Manager	1946
Importing And Exporting Results	1949
Example - CFD External Analysis	1949
Creating The Model	1949
Creating A New CFD External Analysis	1950
Setting Up Monitor Cells	1951
Running The CFD Calculations	1952
Displaying CFD Results	1952
Display 3-D Contours	1954
Example - CFD Internal Analysis Basics	1955
Creating The Model	1955
Default Wall And Window Boundary Temperatures	1956
Adding Surface Supply Diffuser And Extract Grille	1956
Creating A New CFD Internal Analysis	1957
Running The CFD Calculations	1958
Displaying CFD Results	1958
Example - CFD Internal Analysis Including Assemblies	1959
Creating The Radiator Assembly	1959
Positioning Radiators Within The Block	1961
Creating A New CFD Internal Analysis	1962
Editing The CFD Grid	1963
Setting Up Monitor Cells	1965
Running The CFD Calculations	1966
CFD Display Options	1966
Comfort Calculations	1967
Change CFD Analysis	1967
Appendix - CFD Calculations And Convergence	1968
CFD Analysis Update	1969
Daylighting Calculations	1969
Daylighting Model Preparation	1970
Daylighting Calculation Options	1974
Calculation Description	1974
General Options	1975
Simulation Options	1977
Advanced Options	1978
Daylighting Calculation Options - General	1983
General Simulation Options	1983
Annual Options	1983
Point-in-time Options	1987
Daylighting Calculation Options - LEED	1990
LEED Simulation Options	1991
Option 1 (Annual) Options	1991
Option 2 (Point-in-time) Options	1992
Daylighting Calculation Options - BREEAM	1995
BREEAM Simulation Options	1995
Option 4b (Annual) Options	1995
Option 4a (Point-in-time) Options	1996

Daylighting Calculation Options - LM-83-12	1999
LM-83-12 Simulation Options	2000
Daylighting Calculation Options - Right To Light	2000
Daylighting Calculation Options - Green Star	2001
Green Star Simulation Options	2001
Daylighting Outputs And Display Options	2002
View	2002
Show Object Options	2002
Visible Working Planes	2003
Zone Labels	2003
Result Values	2003
Colour	2004
Daylighting Display Options - General	2006
Margin	2007
Metric Pass Thresholds (Annual)	2009
Thresholds (Point In Time)	2009
Scale	2010
Point In Time Illuminance Results	2011
Annual Results	2013
Daylighting Display Options - LEED	2015
Margin	2017
Metric Pass Thresholds (Annual)	2017
Metric Pass Thresholds (Point In Time)	2017
Scale	2018
LEED Version-Dependent Settings	2019
LEED V4.1 Option 1 (Annual) Results	2019
LEED v4.1 Option 2 (Point In Time) Results	2021
LEED v4 Option 1 (Annual) Results	2024
LEED v4 Option 2 (Point In Time) Results	2027
LEED v3 NC 2009 IEQ 8.1 Results	2029
LEED v2 NC 2. EQ 8.1 Results	2032
Daylighting Display Options - BREEAM	2034
Margin	2035
Metric Pass Thresholds (Annual)	2035
Metric Pass Thresholds (Point In Time)	2035
Scale	2036
BREEAM Option 4a Results	2037
BREEAM Option 4b Results	2039
Daylighting Display Options LM-83-12	2041
Daylighting Display Options - Right To Light	2041
Scale	2041
Right To Light Results	2043
Daylighting Display Options - Green Star	2047
Results	2047
Daylighting Metrics	2048
Point In Time Illuminance Daylighting	2048
Annual Climate-Based Daylighting	2049
Right To Light	2052
Guidance For Calculating Daylighting Credits For LEED, BREEAM, Green Star Etc	2055
BREEAM Daylighting Guidance	2056
BREEAM Daylighting Calculations For Mixed-Use Buildings	2056
Radiance Reference	2057
Radiance Implementation	2057
Cost And Carbon	2057
Construction Cost	2057
LCA And Embodied Carbon	2058
One Click LCA Integration	2058
User Guide	2059
One Click LCA Example	2062
Model Setup	2062
Export To One Click LCA	2065
DesignBuilder Embodied Carbon Calculations	2066

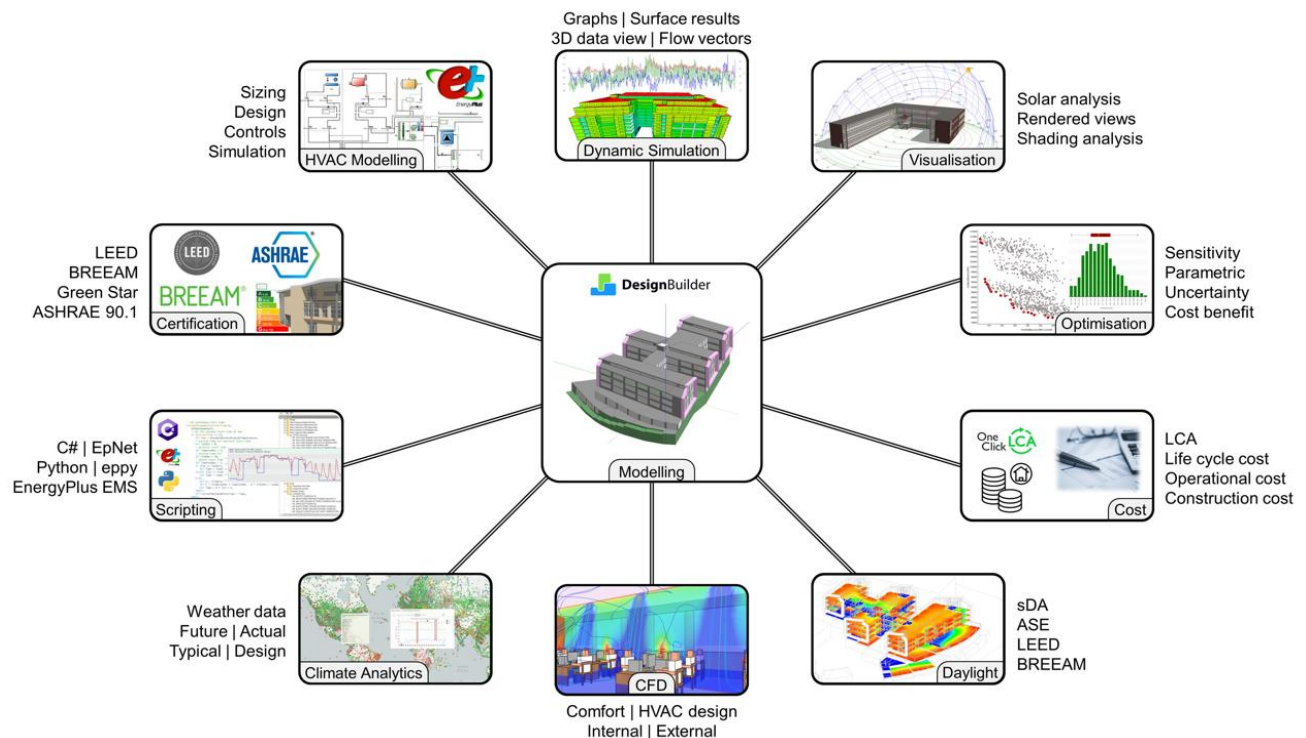
Embodied Carbon Results	2066
Reporting	2069
Export Data	2069
Report Topics	2070
Compile Reports	2070
Insights	2071
Insights Preparation	2071
Registering And Signing In	2071
Exporting Parametric Results To Insights	2073
Insights Upload Options Dialog	2075
Insights Data Management	2076
List Of Projects	2076
List Of Analyses	2077
Insights Data Analysis	2079
Chart Types	2079
Insights Bubble Charts	2079
Insights Parallel Coordinates Plots	2083
Insights Line Charts	2087
Insights Tutorials	2090
Insights Tutorial - Basic Parametric Analysis	2090
Insights Tutorial - Basic Optimisation	2095
HVAC Loads Report	2100
UK Certification	2100
Extensibility	2100
Scripting	2101
Plugins	2102
Energy Management System (EMS)	2102
EMS Edit Script Dialog	2104
General	2105
Script	2105
Utilities	2106
Add Output	2106
Add Program	2108
Add Sensor	2109
Loops	2110
Data	2111
Outputs	2112
Add Actuator	2112
Loops	2113
Data	2113
Add Internal Variable	2113
Loops	2114
Data	2114
Add EMS Variable	2115
Add DB Variable	2116
DesignBuilder Erl	2116
Loops	2117
Optional Arguments	2119
Boolean Expressions - Include/exclude Blocks Of Code	2121
Parametric Variables	2121
DB Erl Changes To Standard EnergyPlus Erl	2121
CS-Script	2121
EpNet Library	2122
Python Scripting	2122
Python Scripting Libraries	2123
Db_eplusout_reader	2123
Eppy	2123
Adding Custom Libraries	2123
Python Scripting Examples	2124
Python Scripting Example 1 - Db_eplusout_reader Basics	2124
Step 1 - Create A Base Model	2124
Step 2 - Write Custom Script	2125

Step 3 - Run Simulation And Check Results	2126
Support - Getting Help	2127
Modelling Guides And Tutorials	2127
Modelling Guides	2127
Help Tutorials	2127
Video Tutorials	2128
Introductory Tutorials	2129
Basic Geometry	2129
Basic Model Data	2129
Calculations	2129
Detailed HVAC	2130
Natural Ventilation	2130
Daylighting	2130
CFD	2130
Other Video Learning Resources	2130
Frequently Asked Questions	2130
Modelling Advice	2133
Working With Large Models And Speeding Up Simulations	2133
Creating And Editing Large Models	2133
Modelling Strategy To Reduce EnergyPlus Simulations Times	2133
Modelling Strategy For Successful Radiance Simulations In Large Models	2135
ASHRAE 90.1 Appendix G PRM Reference	2135
Convert An Existing Simulation Model To An ASHRAE 90.1 Model	2138
Unmet Hours	2138
Natural Ventilation Modelling	2140
Scheduled Natural Ventilation And Infiltration	2140
Calculated Natural Ventilation And Infiltration	2141
DCV Modelling	2143
Background	2143
CO2 Control / DCV Modelling Example	2144
Adjacent Buildings	2157
Electrochromic Glazing	2158
Trombe Walls	2159
Trombe Wall Example	2160
Double Skin Façades	2162
Double Skin Façade Example	2163
Autosizing Heating And Cooling Equipment	2164
Simple HVAC Autosizing	2164
Detailed HVAC Autosizing	2164
Daylight Saving And Local Time	2165
Solar Radiation Distribution	2165
Solar Distribution	2167
Warehouse With Office	2171
Start Up And Pre-conditioning	2177
Warmup And Starting Simulations On A Weekend	2177
EnergyPlus Background Information	2179
EnergyPlus Documentation	2179
Exporting IDF Files	2179
Reading ESO Results Files Back Into DesignBuilder	2180
EnergyPlus Version Compatibility	2181
EnergyPlus IDF Window Dimensions	2181
Calculation Of DesignBuilder Output From EnergyPlus Report Variables	2181
EnergyPlus Errors And Warnings	2190
Errors	2190
Warnings	2193
Maximum Iteration Errors	2195
EnergyPlus Daylight Calculations	2197
EnergyPlus Thermal Comfort	2198
Fanger Comfort Model	2199
Pierce Two-Node Model	2200
KSU Two-Node Model	2201
EnergyPlus Date Definition	2202

When DesignBuilder Output Is Different From Unprocessed EnergyPlus Output ...	2203
Parametric Simulations	2203
EnergyPlus Daylight Map Output.....	2204
How To Generate And View Extra EnergyPlus Outputs	2205
Requesting The Extra Outputs.....	2205
Adding Extra Output Reports Using "Include IDF Data"	2206
Viewing Extra Output Reports.....	2207
Tutorial - Viewing Direct And Diffuse Solar Radiation Incident On A PV Panel	2207
DesignBuilder Files, Location And Extensions.....	2211
DesignBuilder XML (dsbXML)	2213
Hardware Requirements	2214
Results Viewer	2214
Results Viewer Data Display	2217
Detailed Results - Timestep, Hourly, Daily, Monthly, Run Period Tabs	2218
Summary And Summary Charts Results	2221
Hourly Heat Map	2224
CIBSE TM52 Results	2226
CIBSE TM59 Results	2226
Working With Multiple Data Sets	2227
Zooming	2228
Create Composite Plot.....	2228
Crosshair And Tooltips.....	2229
Data Comparison	2230
Styling	2231
Results Viewer Menu And Toolbar Commands.....	2231
Top Menu Bar	2231
Toolbar	2232
Context Menu.....	2235
Results Viewer Options Dialog	2235
General Tab	2235
Units Tab	2236
Graph Tab	2237
Grid Tab	2240
Dataset Comparison Tab	2240
Heat Map Tab	2240
Results Viewer Supported Dataset File Formats	2241
.eso EnergyPlus Simulation Results File	2241
.sql EnergyPlus Simulation Results File	2242
.htm/.html EnergyPlus HTML Summary Results File.....	2242
.epw EnergyPlus Format Hourly Weather File.....	2242
.csv Spreadsheet File.....	2242
.drb Results Viewer Binary File.....	2244
Diagnostics	2245
Finding The Diagnostics Files.....	2245
Calculation Of Air Density	2245
Pitched Roof Geometry - Technical Description.....	2246
Calculation Of Simple HVAC Energy Consumption From EnergyPlus Loads	2248
Component Blocks Used For Local Shading	2249
Data Recovery.....	2250
Standards Used In DesignBuilder.....	2251
Thermal Comfort Calculator	2253
Calculating Fan Pressure Rise	2256
Importing Custom Templates And Components.....	2257
Lock/Unlock Model	2258
Command Line Processing	2258
Available Processes.....	2259
Example Usage.....	2260
Batch Processing	2261
Limitations	2262

Welcome To DesignBuilder V2025.1

DesignBuilder is a user-friendly modelling environment where you can work (and play) with virtual building models. It provides a range of environmental performance data such as: energy consumption, carbon emissions, comfort conditions, daylight illuminance, maximum summertime temperatures and HVAC component sizes.



Some typical uses of DesignBuilder are:

- Calculating the impact of various design options on building energy consumption.
- Evaluating façade options for overheating and visual appearance.
- Thermal simulation of naturally ventilated buildings.
- Reporting savings in electric lighting due to use of natural daylight.
- Prediction of daylight performance through Radiance simulations.
- Visualisation of site layouts and solar shading.
- Calculating heating and cooling equipment sizes.
- Detailed simulation and design of HVAC and natural ventilation systems including the impact of supply air distribution on temperature and velocity distribution within a room using CFD
- ASHRAE 90.1 and LEED energy models.
- Economic analysis based on construction costs, utility costs and life cycle costs (LCC)
- UK, Ireland and France Building regulations and certification reports
- Design optimisation with multiple objectives, constraints and design variables.
- Life-cycle analysis (LCA)
- Communication aid at design meetings.
- An educational tool for teaching building simulation to architectural and engineering students.



DesignBuilder uses the [EnergyPlus](#) dynamic simulation engine to generate performance data.

Learn About DesignBuilder

If you're new to DesignBuilder you are probably keen to start using the program right away, but there's a few things you should do before you dive in:

- Watch the [Getting Started Tutorials](#).
- Read [Important tips about software use](#).

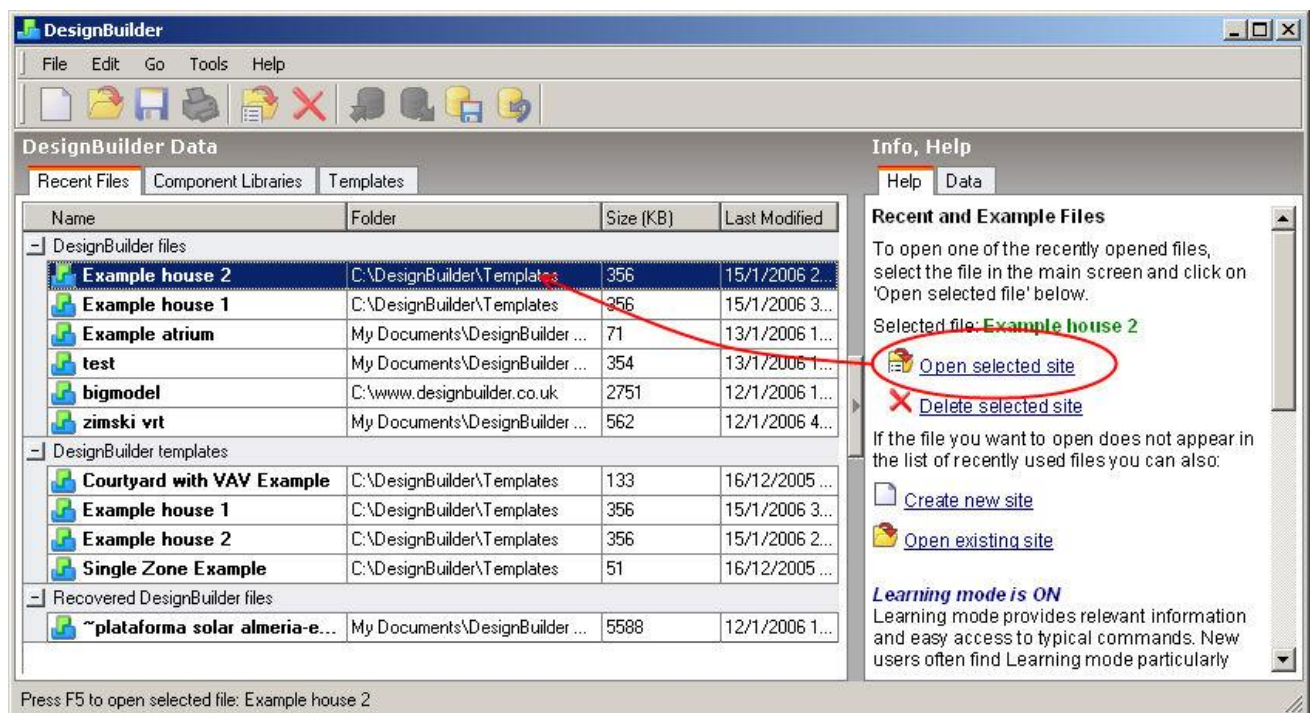
A few minutes spent with these resources will be time well spent!

There is a range of other freely available learning resources on the [Webinars](#) and [Other Learning Materials](#) sections of the DesignBuilder website.

Recent Files (Opening Screen)



The Recent Files Opening Screen has a list of the files used in your previous DesignBuilder modelling sessions. To access one of these files again you can double-click on it in the grid, or single click in the grid to select it and then 'Open selected site' from the Info Panel on the Toolbar.



You can also:

- [Create a new file \(site\)](#) from this screen.
- Open existing dsb file from the **DesignBuilder Data** folder. This is located beneath the main **Documents** by default.

User Interface

The DesignBuilder user interface has been designed from first principles for ease of use. Each screen has a specific purpose which is described in the Info panel on the right of the screen along with context sensitive commands and other relevant information.

The Info panel is only displayed in [Learning mode](#). If you would prefer to have simultaneous access to the Model Data and the Edit View you can switch Learning mode off from the [Program Options](#) dialog from the Tools menu.

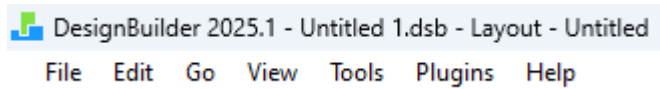
The user interface is described for each group of screens:

- [Opening Screen - Recent Files](#)
- [Opening Screen - Component Libraries](#)
- [Opening Screen - Template Libraries](#)
- [Edit Screen](#)
- [Visualisation Screen](#)

See also the [DesignBuilder User Interface – Modelling: Opening screen, loading a model, screen layout, navigation](#) tutorial

Menu

The DesignBuilder menu is located at the top of the screen and provides a full reference of all the currently available commands.



The following sub-menus are available from the main menu:

- [File](#) - access file operations
- [Edit](#) - options for editing the model
- [Go](#) - options for navigating and obtaining data
- [View](#) - options for adjusting the view of the model
- [Tools](#) - miscellaneous tools
- [Plugins](#) - if you have installed one or more plugin, their menu options will be listed below the top-level Plugins menu item
- [Help](#) - documentation and other support resources on use of the program.

See also: "**Keyboard Shortcuts**" on page **55**

File Menu

The File menu can be accessed from the horizontal [top bar menu](#) at the top of the screen.

Tip: see the "DesignBuilder Model Files" on page 68 help page for advice on file storage, backups etc.

The following options are available when a file is open:

General

- **New file** (Ctrl+N) - create a new DesignBuilder dsb file.
- **Open file** (Ctrl+O) - Open an existing DesignBuilder dsb file.
- **Close** (Ctrl+F4) - close the currently open DesignBuilder dsb file and returned to the opening screen.
- **Print** - Print current view
- **Load results file** - load a [previously generated ESO file](#).

Save

- **Save** (Ctrl+S) - Save currently open DesignBuilder dsb file. If no file is currently loaded then this command saves the libraries.
- **Save without output results** - save the current DesignBuilder dsb file without output results reducing the file size. Use this option if you plan to send the dsb file by e-mail or forums etc.
- **Save as** (F12) - save the DesignBuilder dsb file using a different name/location.
- **Save as template** - save the DesignBuilder dsb file as a template so it can be used as a starting point for subsequent new projects.

Import

- [Import IDF model](#) - import a model saved in the EnergyPlus IDF file format.
- [Import model from XML](#) - imports a model previously exported in the DesignBuilder XML model file format.
- **Import component blocks from XML** - imported component blocks and their attribute settings from a file previously exported using the **Export component blocks as XML** menu command.

- [Merge model data from file](#) - import the attribute settings, components and templates, HVAC system, assemblies and component blocks from a previously created DesignBuilder model into a newly modified one.
- [Import 2-D drawing file](#) - import a 2-D DXF floorplan into the model for tracing over.
- [Import 3-D model](#) - import a gbXML model.
- [Import building from external file](#) - import building(s) into the current model from an external file.
- [Import assembly library](#) - imports an [assembly library](#) from an external asl file previously created using the [Export assembly library](#) command.
- [Import library data](#) - import library data from an existing DDF file.

Export

- [Export EnergyPlus Heating design IDF file](#) - export an IDF file using options for heating design calculations.
- [Export EnergyPlus Cooling design IDF file](#) - export an IDF file using options for cooling design calculations.
- [Export EnergyPlus Simulation design IDF file](#) - export an IDF file using options for simulation using hourly weather data.

Note: the IDF export options are only available for fully licensed users. If you are using an evaluation version and would like to test the IDF export tools as part of your evaluation of the software please contact [DesignBuilder sales](#).

- [Export model as XML](#) - exports the model in DesignBuilder XML (dsbXML) model file format. Exported models include all geometry, attribute and custom components and templates, but not results.
- [Export selected component blocks as XML](#) - exports any selected building level component blocks in XML format. The exported file can later be reimported back in either manually through the [Import component blocks from XML](#) menu command or through a [script](#) as part of a parametric analysis or optimisation study. You must be at building level and have component blocks selected to access this command.
- [Export HVAC Loads Report](#) - exports the model in an industry-standard PDF format.
- [Export model image](#) - export the currently displayed image of the model.
- [Export 3-D DXF model](#) - export the model in 3-D DXF format.
- [Export gbXML model](#) - exports the current building geometry in gbXML format including layered constructions, materials, glazing and shading devices for use in another compatible program.
- - exports an IFC file with heating and cooling design results for loading into MagiCAD.
- [Export to MasterClima 11300](#) - exports the current building data in a format compatible with MasterClima 11300. Contact [SGM](#) our Italian reseller for more details.
- [Export CSV report file](#) - export a summary of the model in spreadsheet format.
- [Export assembly library](#) - exports the current [assembly library](#) to an asl file for importing into another model.
- [Export CFD results slices](#) - from the CFD screen, when results are being displayed in slices, this command allows you to export the result values to a csv file for further analysis and checking.
- [Export library data](#) - export library data in DDF format.

Folder

- [EnergyPlus folder](#) - opens a Windows Explorer window in the folder used by DesignBuilder for storing temporary EnergyPlus files.
- [Weather data folder](#) - opens a Windows Explorer window in the folder where the hourly weather files are stored.
- [Library data folder](#) - opens a Windows Explorer window in the folder where library data files are stored.
- [Template projects folder](#) - opens a Windows Explorer window in the folder where dsb file templates are stored. File templates are available for selection on the second tab of the Add new project dialog.

Export DesignBuilder XML (dsbXML)

The **File > Export > Export model as XML** menu command exports the model in DesignBuilder XML (dsbXML) model file format. The exported model includes all geometry, attribute and custom components and templates, but not calculation results. It can be subsequently imported into any other version of DesignBuilder that supports dsbXML.

DesignBuilder XML files have extension .xml.

The [Include library data in exported DesignBuilder XML](#) program option setting can be used to control whether library data is included in the exported dsbXML file.

See also:

- "DesignBuilder XML (dsbXML)" on page 2213
- [Import DesignBuilder XML \(dsbXML\)](#)
- [Include library data in exported DesignBuilder XML](#)

Import DesignBuilder XML (dsbXML)

The **File > Import > Import model from XML** menu command imports a DesignBuilder XML file previously exported using the **File > Export > Export model as XML** menu command.

After loading the file, DesignBuilder automatically saves the imported model in DSB format to the default model folder (usually Documents\DesignBuilder Data). It uses the name of the dsbXML file as a base and appends a number to ensure uniqueness. For example, when importing the file model.xml, the imported mode will be automatically saved as model1.dsb.

Note: When importing dsbXML files into DesignBuilder, any library component and template data stored in the file is replaced with the latest library data. Imported user-defined data is always retained.

See also:

- "DesignBuilder XML (dsbXML)" on page 2213
- [Export DesignBuilder XML \(dsbXML\)](#)
- [Include library data in exported DesignBuilder XML](#)

Export HVAC Loads Report

Select the [File > Export > Export HVAC Loads Report](#) menu option to export combined heating and cooling loads summary data for each zone in an industry-standard PDF report format.

HVAC Loads Options Dialog

First you must make some settings on the **HVAC Loads Options** dialog to define the data to be displayed:

HVAC Loads Options

Project name
Help Example Project

Design alternative
Alternative system No.1

☒ Heating ☒ Cooling

Totals method
2-System

Cooling peak method
2-Coincident

Cooling coincident definition method
1-Building

Assessor
John Smith

Company
John Smith Ltd

Help Cancel OK

Note: Before selecting the [Export HVAC Loads Report](#) menu option, you should make sure that you have heating and cooling design results already calculated. If this is not the case, you will receive an error message when pressing the OK button on the options dialog.

Project name

This defaults to the building name.

Design alternative

Enter an appropriate name to describe this design alternative.

Heating

If Heating design results are to be included in the HVAC Loads Report, tick this checkbox.

Cooling

If Cooling design results are to be included in the HVAC Loads Report, tick this checkbox.

Totals method

You can sum cooling design total loads in 2 different ways

- **1-Building** - loads are summed for the whole building.
- **2-System** - loads are summed for each [System](#) type allowing you to calculate system (AHU)/block loads for sizing AHUs.

Cooling peak method

Summary cooling design results can be displayed either for the time and month of the overall maximum building cooling load or for the individual zone.

- **1-Non-coincident** - Summary results are for the time and month of maximum cooling load for each zone.
- **2-Coincident** - All summary results are for the time and month of the maximum building total cooling load.

Cooling coincident definition method

When the **Peak method** is set to **2-Coincident** you can specify whether the coincident time for cooling design results is calculated for the building as a whole, i.e. for all zones, or whether it is calculated separately for each system. Select from:

- **1-Building**, where the time of peak cooling for zones is calculated by summing hourly totals for all zones in the building, or,
- **2-System**, where the time of peak cooling for zones is defined considering all zones in each system.

Assessor

Enter your assessor name for the record.

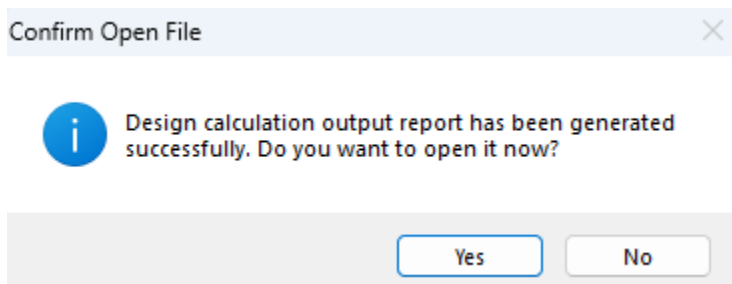
Company

Enter the name of your company.

Viewing The HVAC Loads Report

After pressing the OK button on the HVAC Loads Options dialog you will be asked to provide a filename to save the PDF report.

After saving the report you are asked if you would like to open the report.



If you click "Yes", DesignBuilder will open the PDF report in whatever software is associated with the PDF file extension on your machine.

The "HVAC Loads Report" on page 2100 help topic includes details on the exported report.

Export To MagiCAD

You can export a file in IFC format with heating and cooling loads results for [loading into MagiCAD](#).

Update Space Parameters

Totals

Changed elements: 44

Unchanged elements: 0

Errors: 0

Changed elements

Errors

All elements

Space Number	Space Name	☒ Return air flow [l/s]		☒ Heating dry bulb [°C]		☒ Cooling dry bulb [°C]		☒ Total sensible heat gain [W]		☒ Supply air dry bulb [°C]		☒ Supply air flow [l/s]		Note
		Original	New	Original	New	Original	New	Original	New	Original	New	Original	New	
100	Meeting Room	0,00	57,28	-273	20	-273	25	0	1541	-273	17	0,00	58,91	
103	Office	0,00	17,23	-273	20	-273	25	0	79	-273	17	0,00	17,70	
104	Office	0,00	17,23	-273	20	-273	25	0	79	-273	17	0,00	17,70	
105	Office	0,00	17,23	-273	20	-273	25	0	79	-273	17	0,00	17,70	
102	Meeting Room	0,00	56,80	-273	20	-273	25	0	1890	-273	17	0,00	58,41	
101	Storage	0,00	28,40	-273	20	-273	25	0	827	-273	17	0,00	29,21	
106	Office	0,00	16,94	-273	20	-273	25	0	79	-273	17	0,00	17,40	
107	Office	0,00	16,94	-273	20	-273	25	0	79	-273	17	0,00	17,40	
108	Office	0,00	16,94	-273	20	-273	25	0	79	-273	17	0,00	17,40	
110	Lobby	0,00	110,90	-273	20	-273	25	0	837	-273	17	0,00	114,00	
109	Office	0,00	16,94	-273	20	-273	25	0	79	-273	17	0,00	17,40	
111	Office	0,00	17,21	-273	20	-273	25	0	759	-273	17	0,00	17,70	
112	Office	0,00	16,92	-273	20	-273	25	0	759	-273	17	0,00	17,40	
113	Office	0,00	16,92	-273	20	-273	25	0	759	-273	17	0,00	17,40	

Check all parameters

Uncheck all parameters

Accept changes

Close

About MagiCAD

Fully integrated within Autodesk's Revit and AutoCAD platforms, MagiCAD offers a set of powerful modelling functions for each MEP discipline enabling integrated system calculations. The integration with MagiCAD allows you to feed your DesignBuilder load calculation results back into Revit and from there undertake ductwork and pipework design.

The DesignBuilder > MagiCAD integration works as follows:

1. Export the architectural model directly from Revit by using the native Revit gbXML export.
2. Load the gbXML into DesignBuilder and make any further changes to the model as required.
3. Perform the Heating and Cooling design calculations in DesignBuilder.
4. Export the results using DesignBuilder's **File > Export > Export MagiCAD** menu command. This produces a simplified IFC file containing the results.
5. From within MagiCAD click on the [Import from Design Builder](#) tool. Select the exported IFC file to start the import process. Where the space names and numbers from the active Revit project match the equivalent in the IFC file, the results are loaded to the Revit space instances. In case of mismatches, the spaces can be mapped manually.

Export CSV Report

The CSV report includes an inventory of all the materials, constructions and glazing systems used in the model as well as a summary of each zone with walls, floors, roofs etc, glazing, airflow paths, heating and cooling setpoint temperatures.

Note: The areas of surfaces and openings provided in the CSV report are the same as those that would be used in an EnergyPlus simulation. Under some circumstances these can be very slightly different from the areas specified in the model. For example, in the case where a full height window has top and bottom edges touching the top and bottom of the parent surface DesignBuilder must trim the window slightly to fit it on the surface before providing it to EnergyPlus (and to the CSV report). This is because EnergyPlus requires a small margin between the edges of openings and the host surface. In other words, the edges of child surfaces (openings) cannot be coincident with the edges of the host surface. More specifically, DesignBuilder trims openings where necessary to ensure a minimum margin of 1mm between opening and surface edges.

The report also includes the same [cost](#) and [embodied carbon](#) data that can be obtained from the [Cost and Carbon](#) tab.

To generate the CSV report use the **File > Export > Export CSV report file** menu command. This opens the Report Options dialog with the following settings.

Description

The description to be included within the report.

Report type

The options are:

- **1-Basic** - The basic report.
- **2-ISO 13790** - Same as the basic report but with additional heat flow items included with each line of surface heat transfer data to help with an ISO 13790 analysis.

Edit Menu

The Edit menu can be accessed from the horizontal [top bar menu](#) at the top of the screen.

General

- **Undo** <Ctrl+Z> - undo last operation
- **Redo** <Ctrl+R> - redo last operation
- **End stream** <Enter> - end the partition stream creating a floating partition
- **Cancel current operation** <Esc> - cancel the current operation returning to select mode
- **Undo last point** <Esc> - undo the last point drawn when drawing a block base perimeter or when drawing partitions. Note that when drawing arcs, the **Undo last point** command doesn't undo the whole arc, it operates on one arc line segment at a time. To undo the whole arc, press <Esc> as many times as there are lines in the arc.
- **Clear to default** - clear model data to defaults
- **Load data from template** <Ctrl+L> - loads data from template
- **Reload templates** - reloads data from template throughout model where the template is non-default
- **Transfer blocks** - transfers blocks from one building to another
- **Convert selected outlines to blocks** - convert any selected outlines to component or building blocks
- **Drag block face** - start the drag face operation on the currently selected block
- **Divide block** - divide the currently selected block using the cutting tool
- **Measure** <Ctrl+M> - measure length, angles or areas in the model

Draw

- **Draw block** - add a new block (building level only)
- **Draw internal perimeter** - draw a void/courtyard (block level only)
- **Draw partition** - draw partition streams (block level only)
- **Draw window** - draw windows (surface level only)
- **Draw sub-surface** - draw sub-surface (surface level only)
- **Draw door** - draw doors (surface level only)
- **Draw hole** - draw holes (surface level only)
- **Draw vent** - draw vents (surface level only)
- **Draw construction line** - draw one or more construction lines
- **Place assembly** - place an assembly from the library
- **Flip block surface orientation** - applies only to open manifold blocks imported from BIM models

Remove

- **Remove DXF data** - remove all current DXF data
- **Remove all construction lines** - remove all construction lines
- **Remove all outlines** - remove all outline blocks

Select

- **Select** - returned to select mode
- **Select all** - select all objects in current view

Equipment (zone level, Equipment tab when Detailed gains model option selected)

- [Add](#) - at an item of equipment
- [Edit](#) - edit currently selected equipment
- [Delete](#) - delete currently selected equipment

Opening screen component and template edit

- [Add](#) - add a new component/template as per currently selected item
- [Edit](#) - edit currently selected component/template
- [Delete](#) - delete currently selected component/template

Edit currently selected model objects

- [Move](#) <Ctrl+V> - move selected objects
- [Clone](#) <Ctrl+C> - clone (copy) selected objects
- [Rotate](#) <Ctrl+T> - rotate selected objects
- [Stretch](#) <Ctrl+J> - stretch selected objects
- [Delete](#) - delete selected objects

General

- [Building model options](#) <F11> - open Model options dialog
- [Add building](#) <Ctrl+A> - add a new building (site level only)

Go Menu

The Go menu can be accessed from the horizontal [top bar menu](#) at the top of the screen.

View Menu

The View menu can be accessed from the horizontal [top bar menu](#) at the top of the screen.

Tools Menu

The Tools menu can be accessed from the horizontal [top bar menu](#) at the top of the screen.

General

- [Program options](#) - open program options dialog.
- [Model data grid view](#) - view the model data using a grid view, selecting from a range of view template
- "Optimisation - Parametric Settings Dialog" on page 1705 - **define settings for optimisation and parametric analysis**
- [Rebuild](#) - Rebuilds all secondary geometry including block perimeters, zones, default openings, adjacencies (block and zone) and all display lists from the base building elements.
- [Merge zones](#) - control the way zones are combined.
- [Rezone block](#) - refresh the zoning in the current block.
- [Interzone Airflow](#) - define scheduled air flow rates between zones.
- [Import library components to model](#) - **import current component and template libraries to the model replacing existing ones.**
- [Export data](#) - export results/graphics results from the current screen.
- [Compile report](#) - generation of a report based on previously created [Report topics](#).
- [Hourly weather data](#) - open the hourly weather data dialog.

- [Thermal comfort calculator](#) - calculate PMV and PPD from air and radiant temperatures, RH, clothing and air speed.
- [Show log file](#) - show a log file containing information on your computer and all significant actions taken during the current DesignBuilder session.
- [Building Cost Summary](#) - open the dialog to edit building summary costs.

Results screens

- [Update calculated data](#) - refreshes the results first opening the Calculation options dialog allowing you to enter new options.
- [Lock Y-axis min/max values](#) - locks the extent of the Y-axes so that they keep their current maximum and minimum values allowing consistent comparison with the future results display.
- [Optimisation Analysis Settings](#) - access optimisation settings.
- [View EnergyPlus results](#) - If you are not using the Simulation Manager, this will open the most recently simulated .eso and .htm results into the Results Viewer, or if you are using the Simulation Manager to run simulations, this option will load the most recently loaded set of results for display in DesignBuilder into the Results Viewer.

Libraries (opening screen)

- [Backup libraries](#) - backup component and template libraries
- [Restore libraries](#) - restore previous component and template library backups

Help Menu

The Help menu can be accessed from the horizontal [top bar menu](#) at the top of the screen.

- [Index](#) - access program help using the Index.
- [Search](#) - access program help using the Search facility.
- [Contents](#) - access program help using the hierarchical Contents.
- [Licence](#) - enter serial number and licence keys
- [Tutorials](#) - access DesignBuilder web tutorials
- [About](#) - DesignBuilder version number

Keyboard Shortcuts

Some menu commands can be accessed more conveniently through keyboard shortcut key presses. The tables below provide a list of all of the available shortcuts.

Change screens

<F7>	Edit screen
<F8>	Visualise screen
<F9>	Simulation screen
<Ctrl + F9>	Heating design screen
<Shift + F9>	Cooling design screen

File operations

<Ctrl + N>	New file
<Ctrl + O>	Open file
<Ctrl + S>	Save file
<Ctrl + F4>	Close currently open file
<F12>	Save As

Edit

<F11>	Model options
<Ctrl + F>	Flip Block Surface Orientation

<Ctrl + T>	Rotate
<Ctrl + J>	Stretch
<Ctrl + Z>	Undo last change
<Ctrl + R>	Redo last change
<Ctrl + L>	Load data from template
<Shift + Ctrl + T>	Follow path
<Shift + Ctrl + C>	Remove all construction lines
<Ctrl + M>	Measure tool
<Ctrl + K>	Add block
<Ctrl + B>	Draw partition
<Ctrl + D>	Draw construction line
<Ctrl + V>	Move
<Ctrl + C>	Copy (Clone)
	Delete currently selected data
<Shift + Ctrl + A>	Add building

View

<Shift + Ctrl + F>	Fit view
<Shift + Ctrl + Z>	"Zoom Dynamic" on page 111
<Shift + Ctrl + W>	"Zoom Window" on page 111
<Shift + Ctrl + N>	Pan view
<Shift + Ctrl + O>	Orbit view
<Shift + Ctrl + L>	Look around
<Shift + Ctrl + K>	Walk-through
<Shift + Ctrl + D>	Toggle surrounding block/zone display

View rotation

The [view rotation](#) can be edited using the drop list in the top right of the screen or through these shortcut keys.

<Ctrl + A>	Axonometric view
<Ctrl + P>	Plan view
<Ctrl + ↑>	Front view
<Ctrl + ↓>	Back view
<Ctrl + →>	Right view
<Ctrl + ←>	Left view

Navigation

<F2>	Go up
<F4>	Go to previous object
<F5>	Go to next object

Miscellaneous

<F1>	Context sensitive help
<Ctrl + X>	Export data
<Ctrl + W>	Compile report
<Ctrl + U>	Update results
<Ctrl + Shift + L>	Lock axis

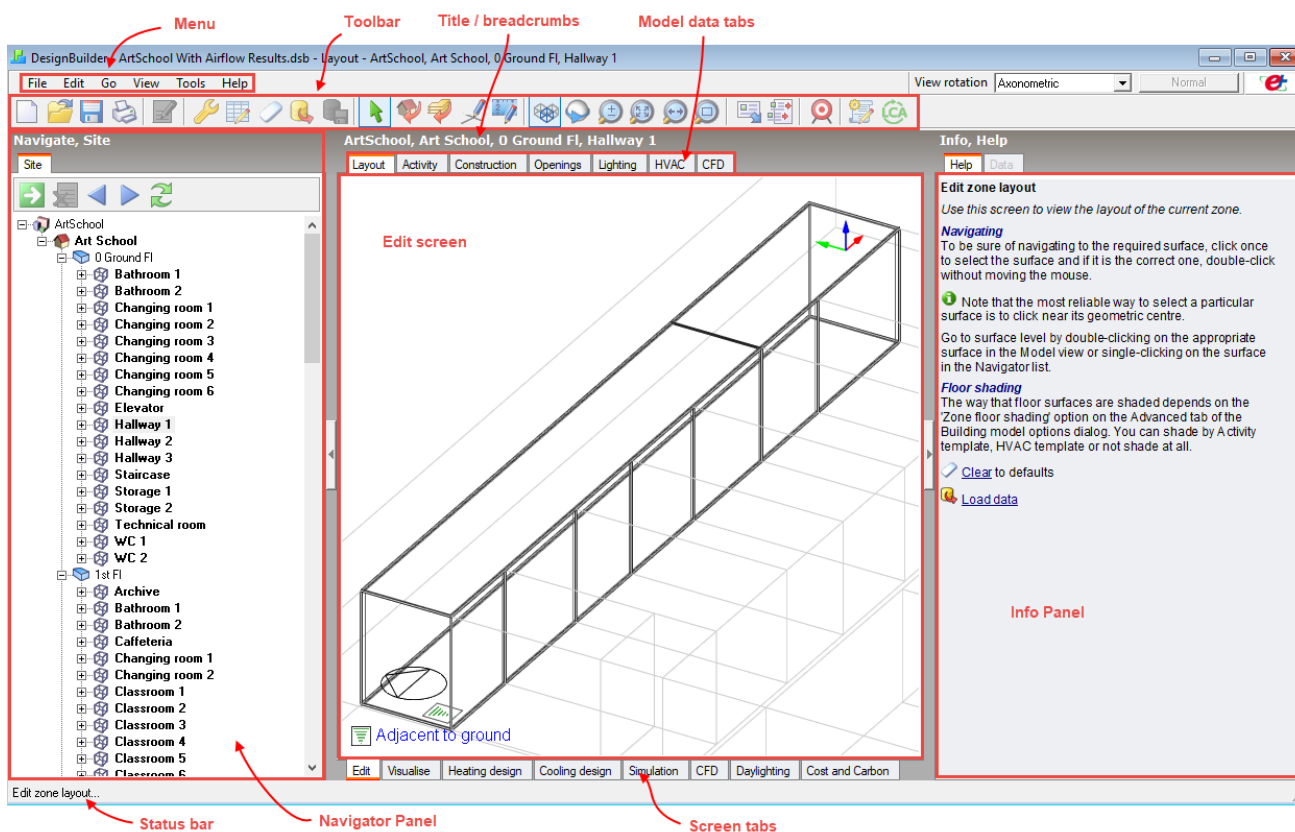
- <Ctrl + Shift + G> Show log file
- <Ctrl + Shift + X> Show file import log
- <Ctrl + Shift + S> Change solar position when on Visualise screen and shadows and the Sunpath diagram are both displayed

Edit Screen

The diagrams below illustrate the various components of the DesignBuilder Edit Screen with [Learning Mode](#) switched on and off.

Learning Mode On

With Learning mode switched on the Edit screen will look similar to the image below.

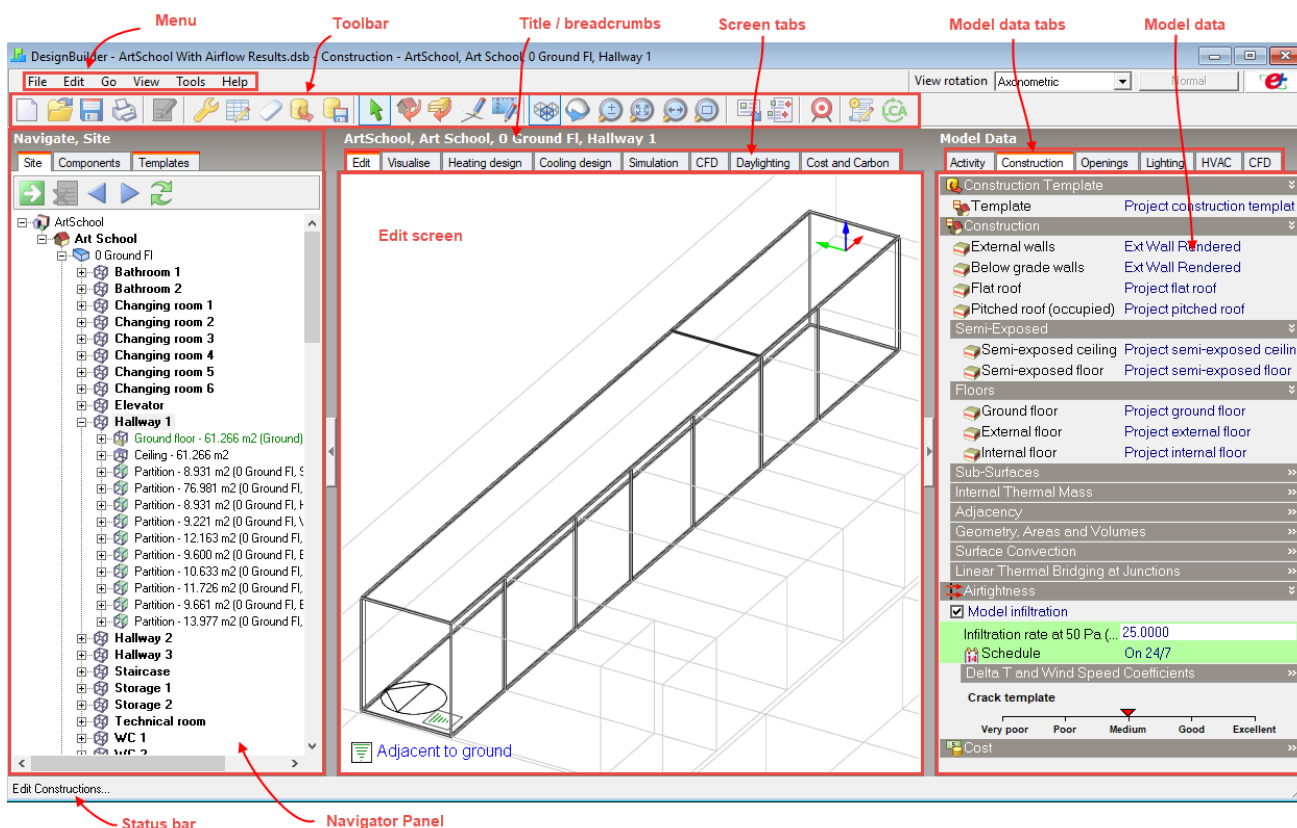


Use the Model data tabs to access Activity, Construction, Openings, Lighting, HVAC etc [model data](#).

See also the [DesignBuilder User Interface – Modelling: Opening screen, loading a model, screen layout, navigation](#) tutorial

Learning Mode Off

The image below shows the same model but with Learning mode Off. Edit Screen and Model data are visible simultaneously and there is no Info Panel..



DesignBuilder Tips

Miscellaneous information you should know before starting 'serious work' with DesignBuilder:

1. Buildings are put together by [drawing blocks](#). Block creation and edit operations take place at the [building level](#).
2. Blocks can be subdivided into more than one zone by drawing partitions. Partitioning takes place at the [block level](#).
3. Default data is [inherited](#) from the level above, so blocks inherit their data from the building level, zones inherit their data from blocks, surfaces from zones and openings from surfaces. It is therefore possible to make global changes to the model by changing data at building level. It is possible to override default by editing the data. So for example, you could change the external wall construction for a whole block.
4. [Data templates](#) are the source of default data in DesignBuilder. They can also be used for [loading data](#) to the model en masse. For example you can load the 'Energy code - heavyweight' construction template at the building level to change the construction of the entire building to thermally heavyweight elements which comply with local energy codes/building regulations.
5. You can clear non-default data (i.e. data that has been explicitly entered) to its default state by using the [Clear to default](#) command.
6. When drawing a block which touches another block, you should use the shift key to lock the base perimeter to the correct plane before drawing. You can release the shift key once you have placed the first point of the perimeter.
7. Use the snap points - DesignBuilder block geometry is quite easy to work with but you must use the positioning tools provided to ensure that blocks connect exactly. The only way to do this is to use snaps - do not position blocks and partitions relative to each other 'by eye'.
8. Large building models can take a long time to simulate. You should familiarise yourself with the [Working with Large Models](#) help topic before creating a large/complex DesignBuilder model.
9. [DesignBuilder dsb files](#) are already compressed so there is no need to 'Zip' them when attaching to e-mails or support forums.
10. Understand DesignBuilder [constructions](#), in particular the way floors and ceilings are defined.
11. Name your zones. Do this by clicking twice (not double-clicking) on the name in the Navigator and typing the name. This is especially useful when [merging](#) zones and when checking adjacencies of individual surfaces in the Navigator.

12. If you are exporting IDF data for use outside DesignBuilder please read the [Exporting EnergyPlus IDF Files](#) topic.

Core Concepts

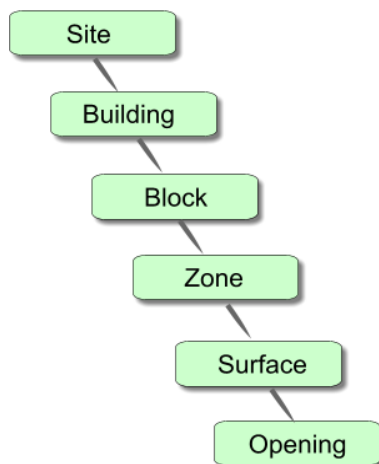
Some core concepts you should be familiar with before attempting to use DesignBuilder on a real project are:

- [Model data hierarchy and data inheritance](#)
- [Templates and components](#)
- [Blocks](#)
- [Navigating building models](#)
- [Customising model detail](#)
- "DesignBuilder Model Files" on page 68

Model Data Hierarchy & Data Inheritance

See also the [Model data hierarchy and inheritance](#) tutorial

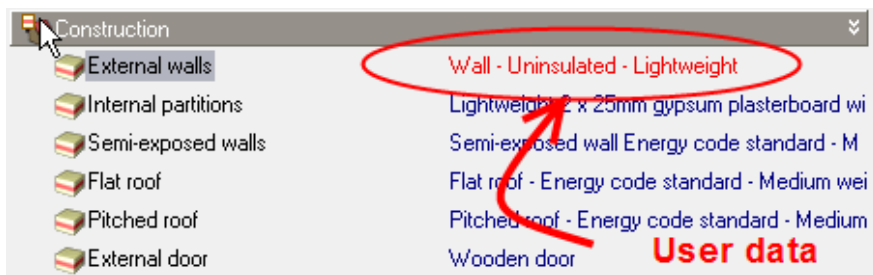
DesignBuilder models are organised in a simple hierarchy:



Default data is inherited from the level above in the hierarchy, so block data is inherited from building level, zone data is inherited from block data and surface data from zone data. This arrangement allows you to make settings at building level which can become active throughout the whole building; or make settings at block level to change data for all zones/surfaces in the block. So for example, if the External wall construction is set to 'Wall 1' at building level, then this will be the default External wall for all blocks in the building. It is possible to change the inherited defaults in any of these blocks by making an explicit selection (say 'Wall 2'). All the zones in the block whose external wall construction was set to 'Wall 2' have that default External wall construction.

The construction and opening defaults are inherited down to surface level where the data is actually used in the calculations. For example it is the External wall construction model data at the surface level that defines the actual construction used for that external wall. The External wall model data set at zone, block and building level has no effect on the model (other than by providing the default data for the level below in the hierarchy).

To assist in identifying where data is inherited from DesignBuilder uses a colour code. Data set at building level or user defined data 'hard set' at any level below will appear in red. All data inherited at levels below will be in blue (see below).



The simplest way of determining where data is inherited from is to start at the lowest level, a surface or opening, and go up through the levels until the data turns red. This then is the level from which the data has been inherited.

The DesignBuilder hierarchy mechanism is a fast way of setting data globally in a building model, but you must take care to *enter the minimum amount of data* to get the most from the inheritance system. For example if all zones in a particular block have the same activity called say 'Office work', you should set the activity at block level and not multiple times at zone level. Equally if all the blocks in the building have the same activity, you should set the data at the building level and allow the blocks to inherit the data from the building. By keeping the amount of user data to a minimum, you will find it quicker to make changes at a later date. Clearly it is much faster to change a few Model data items at building and block level than it is to make multiple changes at zone or surface levels.

When entering construction data, you could set the construction for each surface in the model but in practice this would be an extremely inefficient process. It is faster and less error-prone to set the construction as high up in the hierarchy as possible, possibly at building level (if all external walls in the building have that external wall construction), or at block level if all walls in that block have the construction (and other blocks don't).

Most data at the building level is 'user data' as buildings do not inherit their data from the site level.

Site data holds location and regional settings as well as a list of buildings.

To undo user defined 'hard set' data run the Edit > [Clear to default](#) menu command and select **All data** or **Custom openings** and specify level (block, zone or surface) down to which the data is to be cleared.

Defaults

In short the rules are:

- [Construction](#) and [Openings](#) data is used at the *surface* level, i.e. all construction and opening data at building, block and zone level is default data and only data at surface level is actually used to define the model.
- [Activity](#), [Lighting](#), [HVAC](#), [Equipment](#) data is used at the *zone* level, i.e. all Activity, Lighting, HVAC, Equipment data at building and block level is default data.

A common question is "*Why do I see Pitched roof constructions at building/block/zone level on the Constructions tab when my building/block/zone doesn't have a pitched roof?*"

The answer is that the Pitched roof construction setting at building, block and zone levels should be thought of as the default setting for any pitched roofs that may (or may not) exist in the model further down in the hierarchy. If there are no pitched roofs in the model then the data is simply not used.

Custom openings

When [custom](#) (user-drawn) openings have been created at surface level these override any openings that would otherwise have been generated through the [facade](#) layout Model data for the surface.

Component blocks

[Component blocks](#) inherit their data from building level.

Partition data inheritance

There are 2 types of partition surface:

- [Partitions](#) separating zones within a single block. These partitions within a block inherit their data from the block level. Why? They can't inherit from zone level because there are 2 zones adjacent to the partition so they inherit from the next level up in the hierarchy - blocks.
- [Inter-block partitions](#) which separate zones in 2 different blocks have 2 parent blocks and so it is not possible for the data to inherit from block level for these surfaces so data inherits directly from building level.

See also: "Zone Level Data for Internal Surfaces" on page 698

Templates And Components

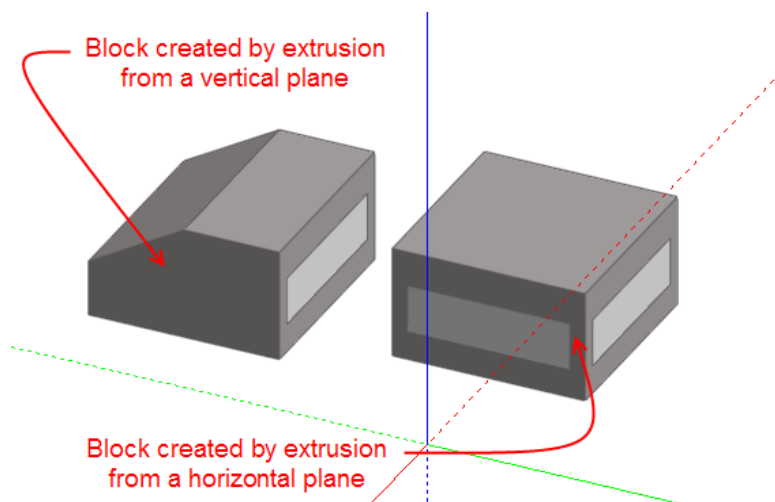
- [Templates](#) are databases of typical generic data.
- [Components](#) are databases of individual data items (e.g. a construction type, material, window pane) .

DesignBuilder allows you to build up your own [libraries of templates and components](#). Note that each user on a computer maintains their own database of templates and components.

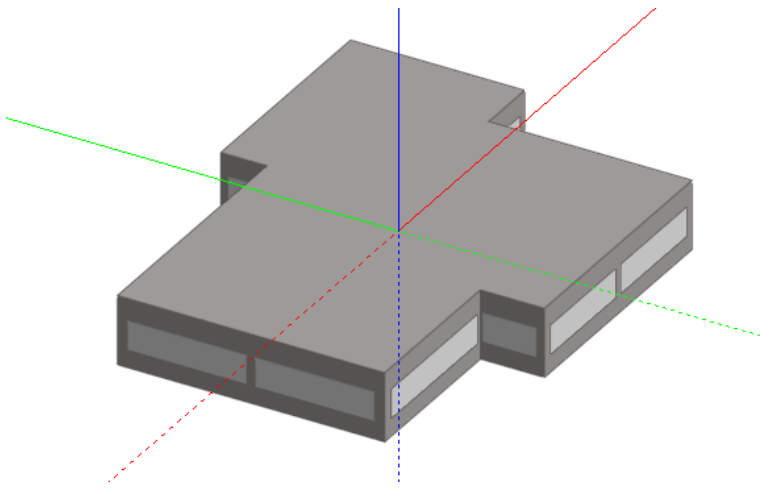
Template and component libraries are loaded into your model at the time it is created and new custom data from other computers can be [imported](#) at any time.

Blocks Introduction

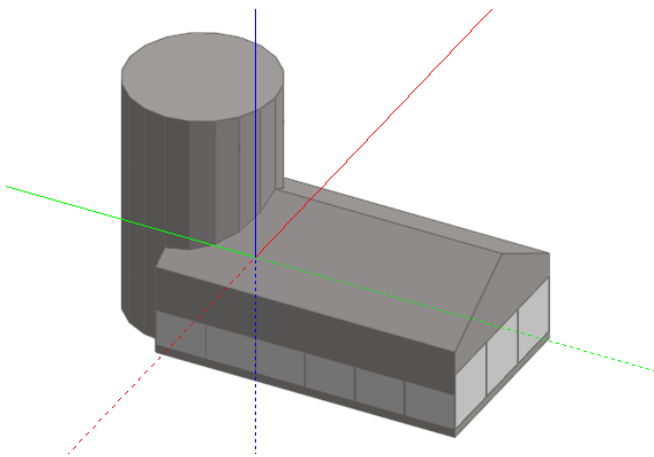
[Blocks](#) are the basic geometric shapes that you use to assemble a DesignBuilder model. The idea of blocks is to provide fairly simple objects that you can easily move around in 3-D space in a similar fashion to that of building a physical model using bricks from a construction set. [Blocks are created](#) by drawing 2-D perimeters on horizontal or vertical planes and then extruding these perimeters to form 3-D shapes:



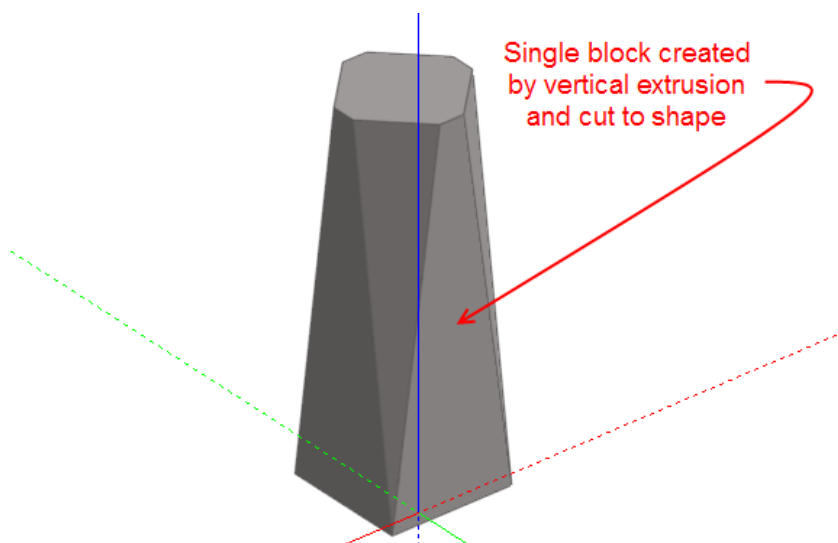
Blocks are versatile objects and once created can easily be modified by moving their surfaces, by [stretching](#) them and even by [cutting](#) them using arbitrary planes. Typically, you might use a block to represent an entire building storey:



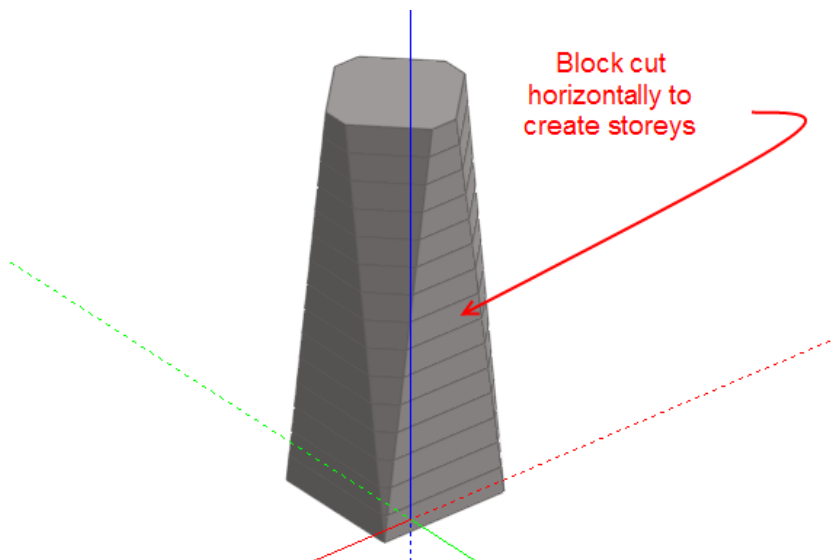
Or a particular building design might lend itself to an obvious division into blocks:



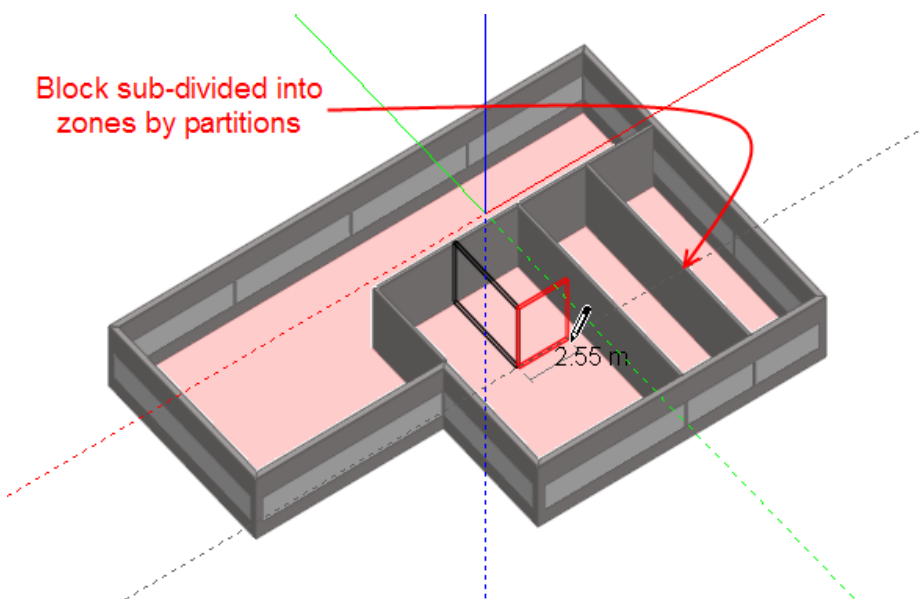
You can model an entire building as a single block:



And then cut the block into storeys:



Once the overall geometry of a model has been established with blocks, you can go 'into' a block and sub-divide it into a number of rooms or zones using partitions:



If you sub-divide a block internally using partitions, you can still modify the block by stretching, cutting, rotating, etc. and where possible the internal geometry will be kept.

Also blocks can have boolean geometry operations applied such as [Union](#), where 2 blocks are combined into a single new block, [Difference](#) (subtract), in which one block is subtracted from the other to form a single new block and [Intersection](#), where a new block is formed where the 2 original blocks intersect.

Navigating Building Models

[Quick overview](#) of the navigation tools in action

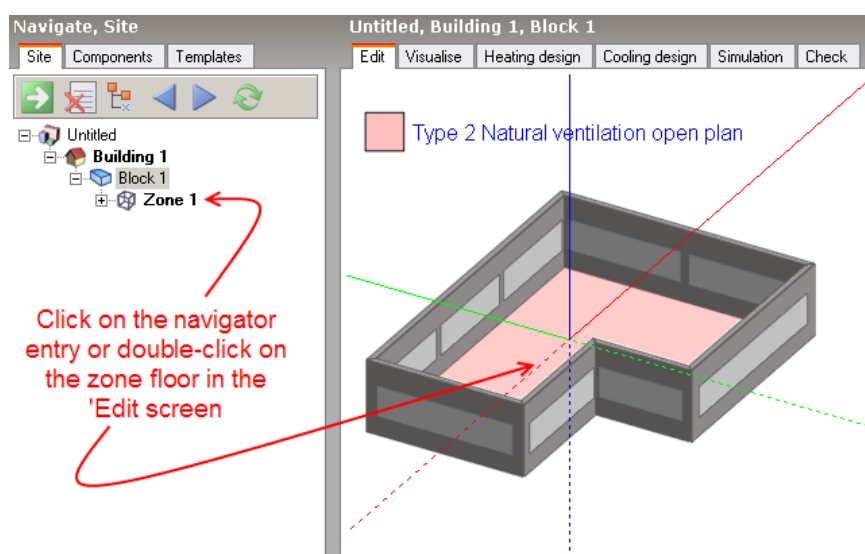
There are various ways of navigating (moving around) DesignBuilder models:

- Single clicking the appropriate entry in the Navigator panel is the most versatile way because every object in the building has an entry in the Navigation panel. Note that if the [Double click to navigate site](#)

[hierarchy](#) program option is set then you must double-click in the navigator to go to the associated object.

- Double-clicking on the appropriate object in the Edit Screen takes you to it.
- Single-clicking on the appropriate object in the Edit Screen then right mouse clicking on the object and selecting the 'Go to...' menu option at the top.
- Use the [Title / breadcrumbs](#) at the top of the screen to navigate up one or more levels.
- Go to the level up in the hierarchy by pressing <F2> or by right mouse clicking in the Edit Screen and selecting the 'Go up...' menu option.
- The Next <F5> and Back <F4> Navigation buttons allow you to move sequentially through the model (avoiding surfaces). This approach can be slow but can be useful when it is necessary to methodically work through the model for any reason.

The Figure below shows how to either click on the Navigator entry or double-click on the Edit Screen to go to 'Zone 1' from 'Block 1'.



Using The Navigator

The Navigator panel in the top left of the screen includes a full list of the model hierarchy, starting at Site Level and going down to Building, Block, Zone, Surface, Opening Levels. Also included are Component and Assembly Blocks and the HVAC hierarchy when using Detailed HVAC.

The following objects can be renamed by clicking on the currently selected the Navigator Treeview node:

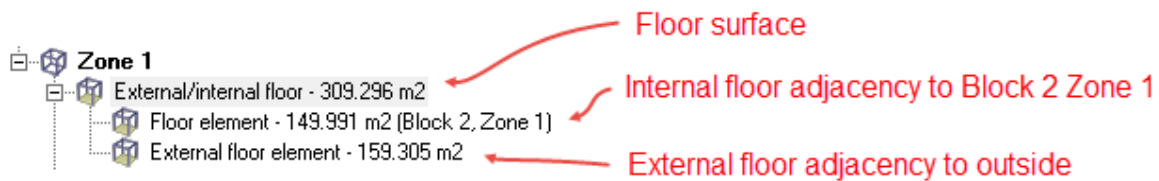
- Site
- Building
- Block
- Zone
- Component block
- Assembly instance
- Solar panel
- High-level HVAC system components (e.g. Loops, Setpoint Managers, Zone groups)

The following objects cannot be renamed through the Navigator TreeView:

- Surface
- Opening (Window, Door, Vent, Subsurface, Hole, CFD boundary condition)
- HVAC system sub-components which only inherit their names from the level above.

Note: The areas quoted in Surface Navigator nodes are gross areas, i.e. they include the areas of any openings they may contain.

Note: The level below Surface in the Navigator includes any openings that are placed on the surface as well as the breakdown of 1 or more adjacencies to other zones or the outside. For example, a floor surface may include two adjacencies: an external floor and an internal floor, as shown in the screenshot below.

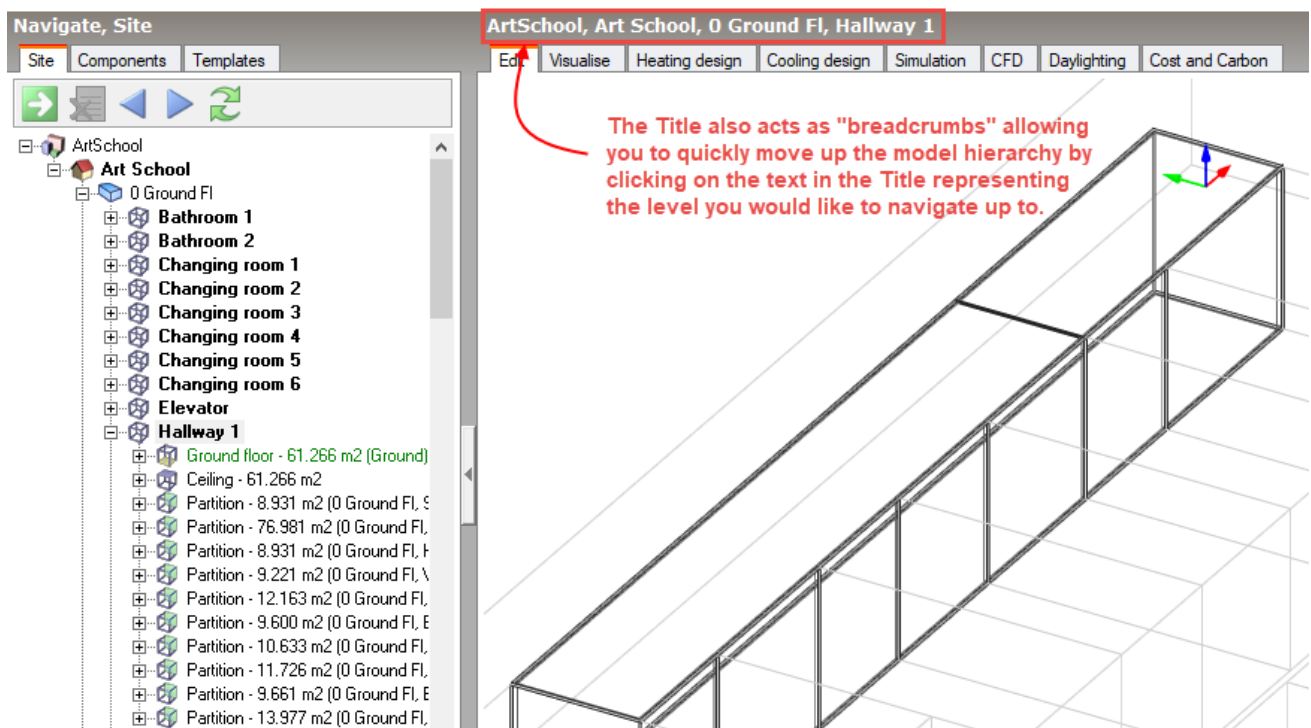


See also [HVAC Component Hierarchy and Navigation](#).

Title / Breadcrumbs

The Title at the top of the screen tells you which object you are currently navigated to. For example in the screenshot below the title tells us that we are navigated to "Hallway 1" in block "0 Ground Fl" in building "Art School" on the site "Art School".

The title also acts as "breadcrumbs" allowing you to quickly move up the model hierarchy by clicking on the text in the Title representing the level you would like to navigate to. As you move the mouse over the Title text, the various elements are underlined like a link on a webpage to indicate what would happen if you click with the mouse.



The Title / breadcrumbs are available on all screens, including when editing [Detailed HVAC](#) systems.

Note: Alternative ways to navigate up the hierarchy by one level are by pressing the <F2> key or by right mouse clicking in the Edit Screen and selecting the 'Go up...' menu option.

Customising Model Options

DesignBuilder models can be customised to allow you to enter the level of detail which is appropriate to your building design or modelling project. In early-stage design work you will generally want to use generic selections for most Model Data and so would choose the corresponding 'low data requirement' model options.

As the building design firms up, more information becomes available and you can change the model options to use more specific data tailored to the project and less generic data.

The 'standard' Model Option set is loaded at the point the building is created and you should generally keep these options unless you have specific reason to change.

Model Options are set in the [Model Options dialog](#) from the Edit menu.

DesignBuilder Model Files

DesignBuilder stores all of its main model data in a single file with .dsb extension making it easy for you to send project models along with associated results to colleagues and clients. A .dsb file includes:

- Building geometry
- Model data settings
- HVAC data
- Model components and templates
- Any user-defined weather files
- Any user-defined textures
- All results for Heating design, Cooling design, Simulation, Daylighting, Cost/Carbon, Optimisation and CFD (including all project variations)

See also:

- "Program options - Files" on page 96
- "Data Recovery" on page 2250

Guidance For Working With DesignBuilder .dsb Files

DesignBuilder .dsb files should be saved to and loaded from the local hard disk. You should specifically not work with to the following locations:

1. Dropbox or similar folders which can lock files for synching for storage on cloud servers. **If you try to work on Dropbox or OneDrive folders your data will eventually become corrupted as DesignBuilder tries and fails to save to a locked file.**
2. Network drives. Less risky than Dropbox folders but a risk nevertheless.
3. Removable data sticks.

If you need to store your data using any of the above locations then you should copy your .dsb data stored on the local disk to Dropbox, network or data sticks outside DesignBuilder once you have closed the software down.

Licensing & Activation

[Help menu > Licence](#)

DesignBuilder consists of a modelling Graphical User Interface (GUI) connected to a number of modules each providing a particular functionality. Modules include Simulation, Visualisation, HVAC, CFD etc. You will need a license for each of the modules you intend to use. If you require more than one module then the most

convenient way to obtain DesignBuilder licences is usually to purchase a [DesignBuilder package](#). Each package consists of 2 or more module licences, each module giving access to a particular aspect of DesignBuilder functionality.

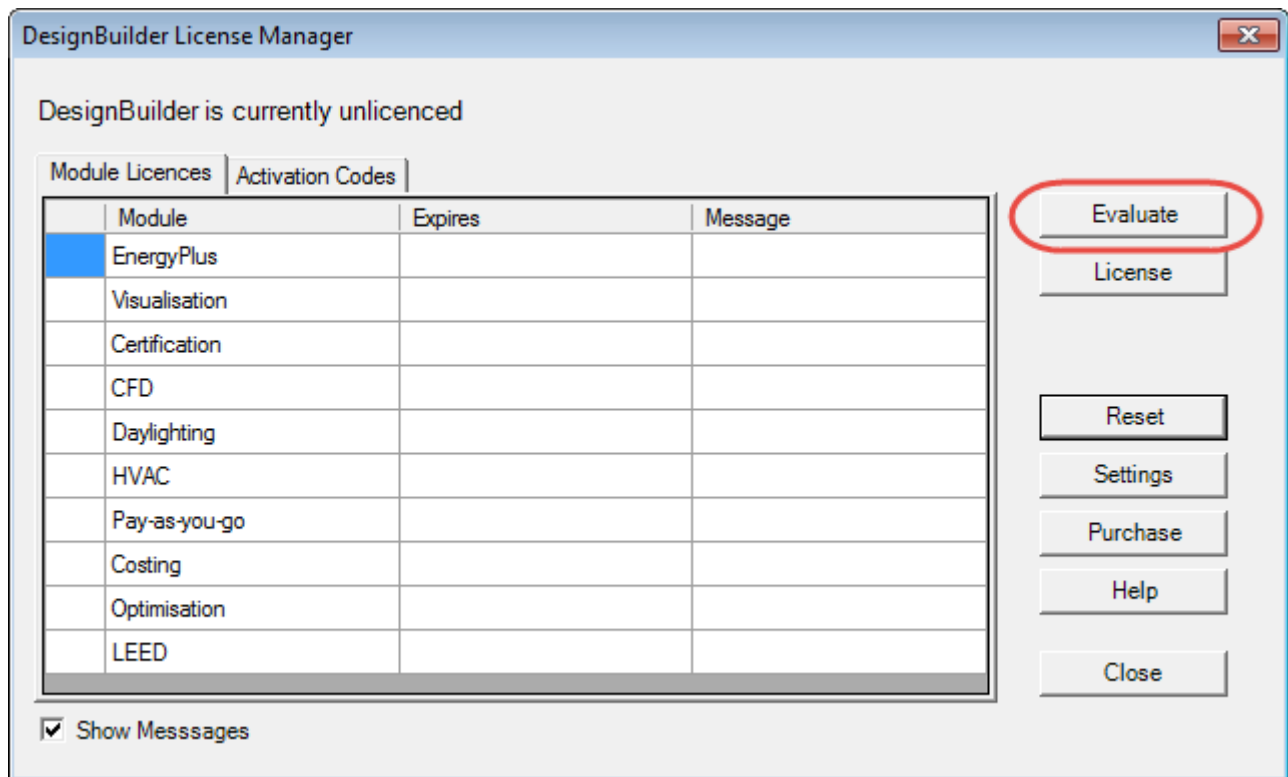
There are various types of licence and ways of licensing all accessed from the **Licence Manager** dialog (**Help > Licence** menu option). The types of licence are:

- **Single user** licence which allows you to install DesignBuilder onto a single computer. If you need to use the software on a second computer you can deactivate the licence on the first one and activate on the second. You can move the licence from one computer to another freely in this way.
- **Site network** licence where a site license server is installed on one computer on the network and provides module licenses on demand to DesignBuilder users on the network. With a site network license you purchase individual DesignBuilder modules in quantities sufficient to support your concurrent use requirements.

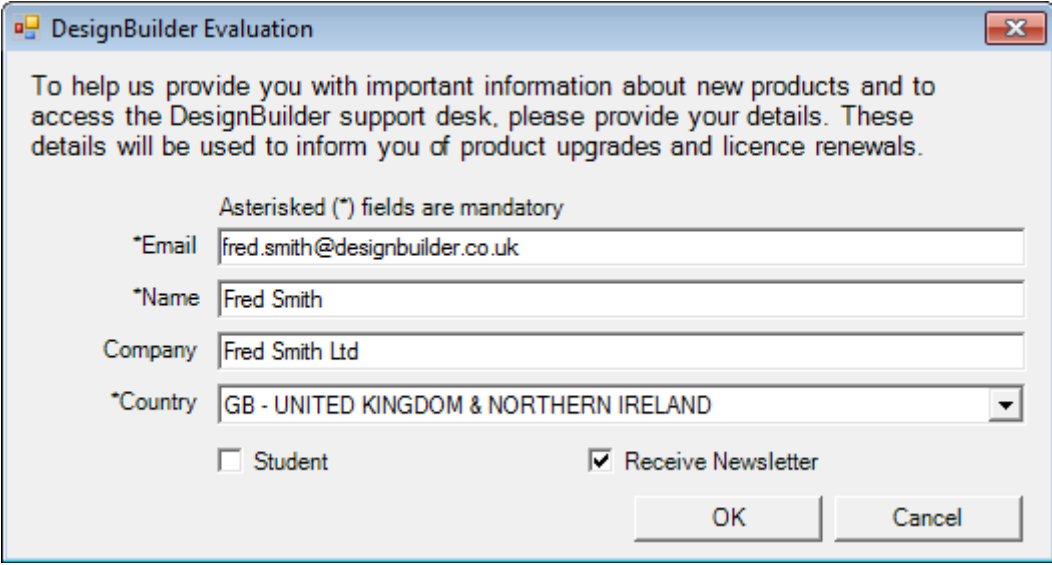
For details on how to licence the software please read on.

Evaluation

When you first install DesignBuilder you can request an evaluation license to be sent by email. To do this open the Licence Manager dialog and click on the **Evaluate** button (this button will only be enabled if no other licences are loaded on the machine).



This takes you to a new dialog where you can enter your user details including an email address.



DesignBuilder Evaluation

To help us provide you with important information about new products and to access the DesignBuilder support desk, please provide your details. These details will be used to inform you of product upgrades and licence renewals.

Asterisked (*) fields are mandatory

*Email

*Name

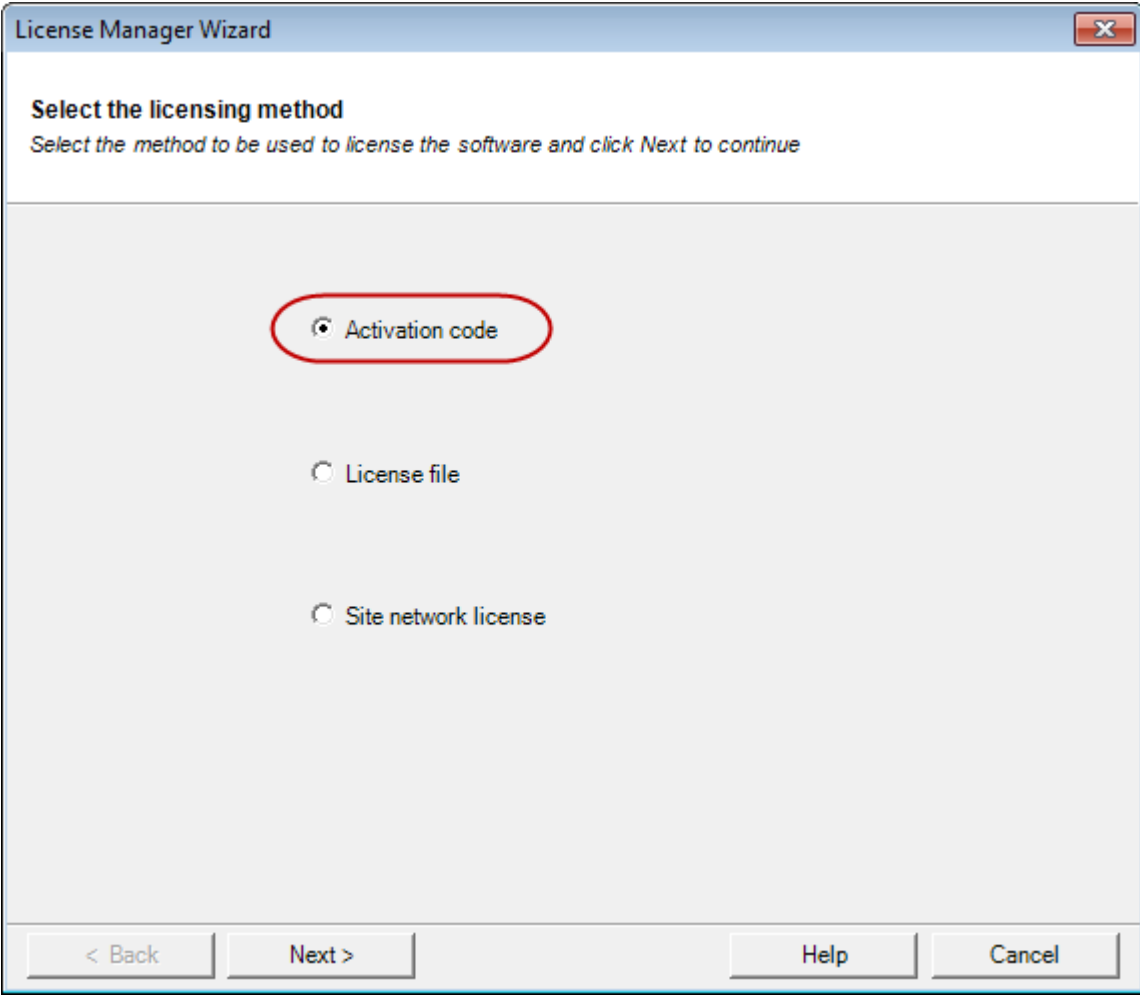
Company

*Country

☐ Student ☒ Receive Newsletter

OK Cancel

You will then be sent an email with an activation code. Copy the activation code from the email and return to the Licence Manager dialog, pressing the **Licence** button to open the **Licence Manager Wizard**.



License Manager Wizard

Select the licensing method

Select the method to be used to license the software and click Next to continue

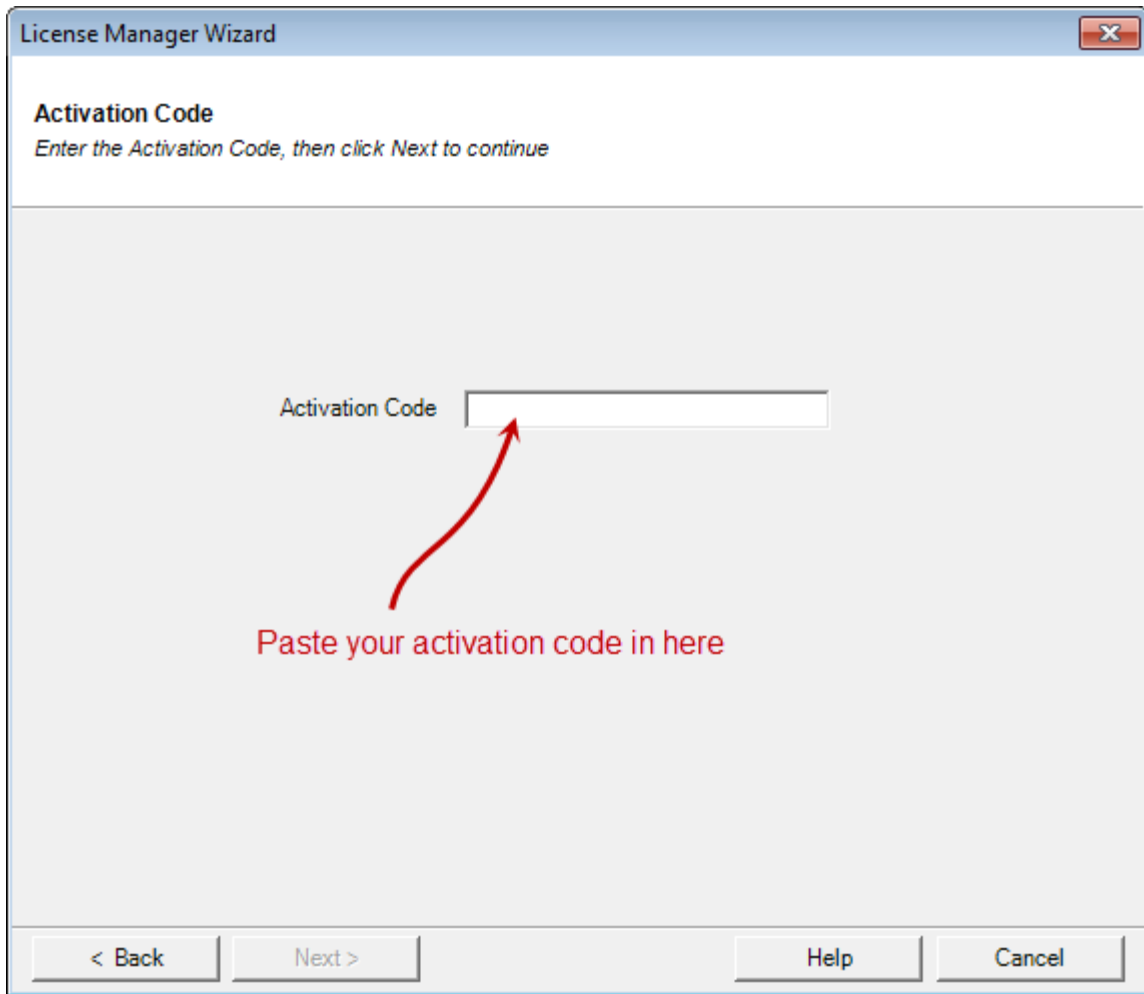
☒ Activation code

☐ License file

☐ Site network license

< Back Next > Help Cancel

Here select the **Activation code** option, confirm your user details and press Next. On the next page paste your activation code into the text box.



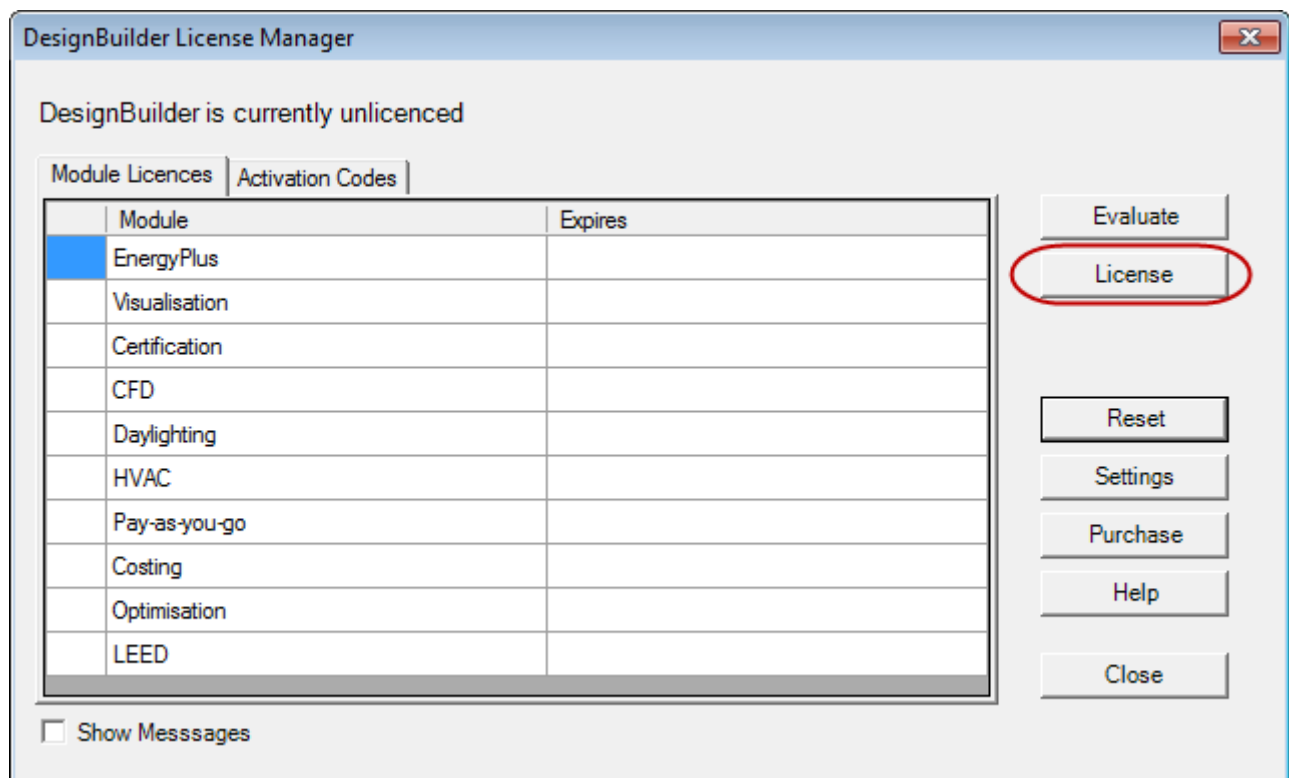
The image shows a screenshot of the 'License Manager Wizard' window. The title bar at the top says 'License Manager Wizard' with a close button on the right. Below the title bar, the text 'Activation Code' is displayed in bold, followed by the instruction 'Enter the Activation Code, then click Next to continue'. In the center of the window, there is a text input field. To the left of the input field is the label 'Activation Code'. A red arrow points from the text 'Paste your activation code in here' (written in red) to the input field. At the bottom of the window, there are four buttons: '< Back', 'Next >', 'Help', and 'Cancel'.

Evaluation licences are valid for 30 days from the date of request and allow you to try out a fully functional version of the software to check that it meets your needs and is compatible with your computer. You can read about restrictions with evaluation licences on the [main website](#).

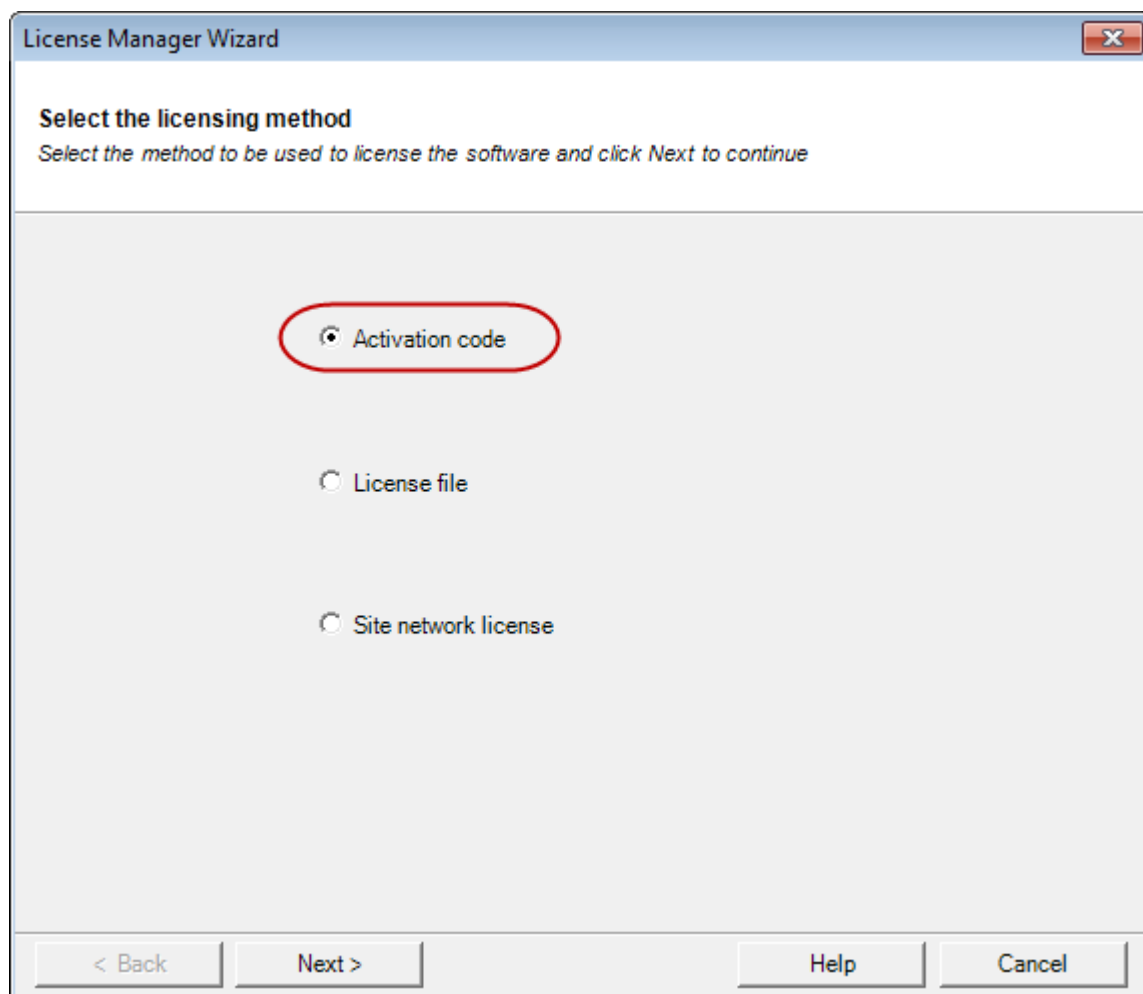
Activation

When you purchase single user licences to use DesignBuilder, you are sent an e-mail containing an 8-character code. You should copy (Ctrl-C) the activation code from the e-mail and follow these steps in DesignBuilder:

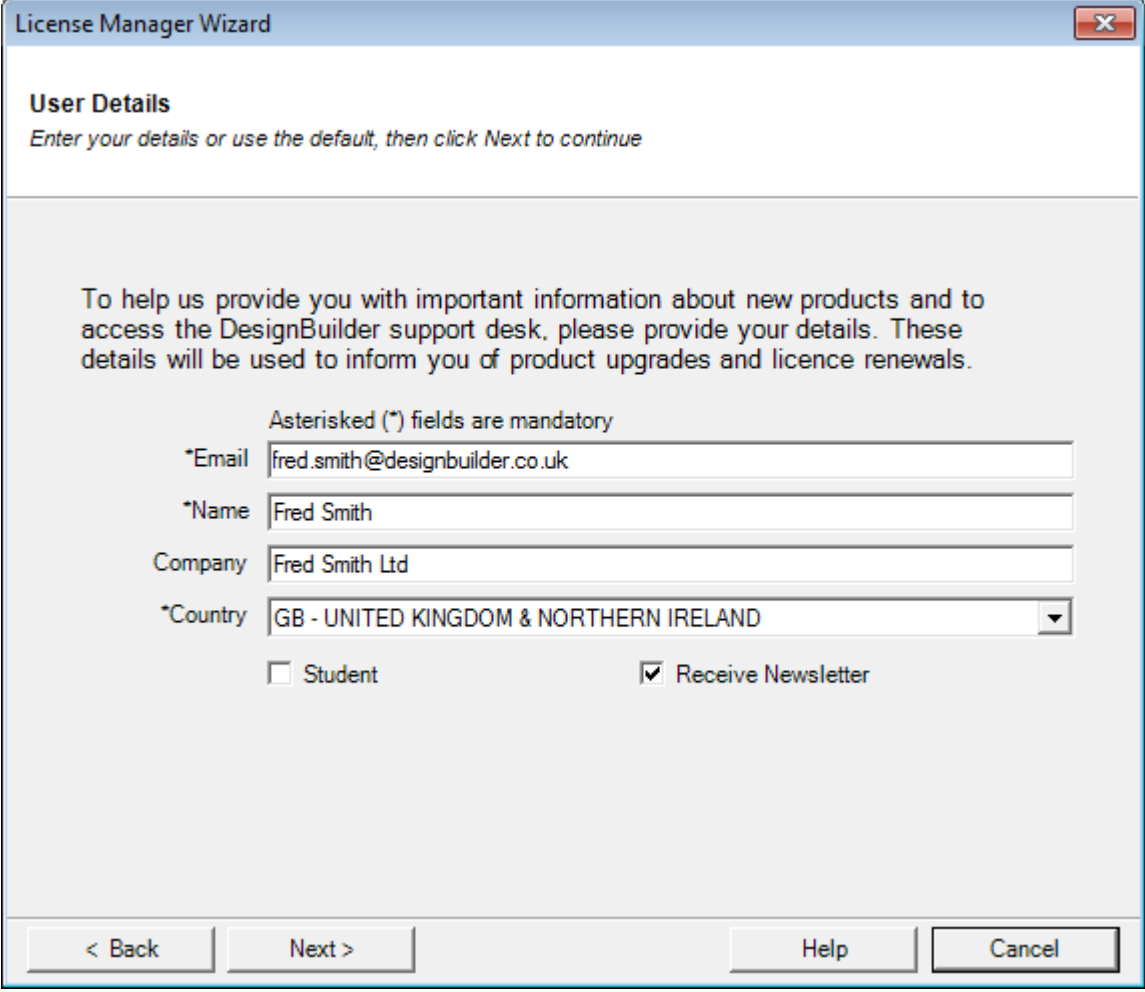
Open the **Licence Manager** dialog (**Help > Licence**). Click on the **Licence** button to open the **Licence Manager Wizard**.



Here select the **Activation code** option and press Next.



Enter your user details including a valid email address and press Next.



The image shows a 'License Manager Wizard' window with a title bar containing a close button. The main heading is 'User Details' with a subtitle 'Enter your details or use the default, then click Next to continue'. A paragraph explains that details are needed for product information, support desk access, and license renewals. A note states 'Asterisked (*) fields are mandatory'. The form includes fields for Email (fred.smith@designbuilder.co.uk), Name (Fred Smith), Company (Fred Smith Ltd), and Country (GB - UNITED KINGDOM & NORTHERN IRELAND). There are checkboxes for 'Student' (unchecked) and 'Receive Newsletter' (checked). Navigation buttons at the bottom are '< Back', 'Next >', 'Help', and 'Cancel'.

License Manager Wizard

User Details
Enter your details or use the default, then click Next to continue

To help us provide you with important information about new products and to access the DesignBuilder support desk, please provide your details. These details will be used to inform you of product upgrades and licence renewals.

Asterisked (*) fields are mandatory

*Email

*Name

Company

*Country

☐ Student ☒ Receive Newsletter

< Back Next > Help Cancel

On the next page paste your activation code into the text box and press Next.

License Manager Wizard

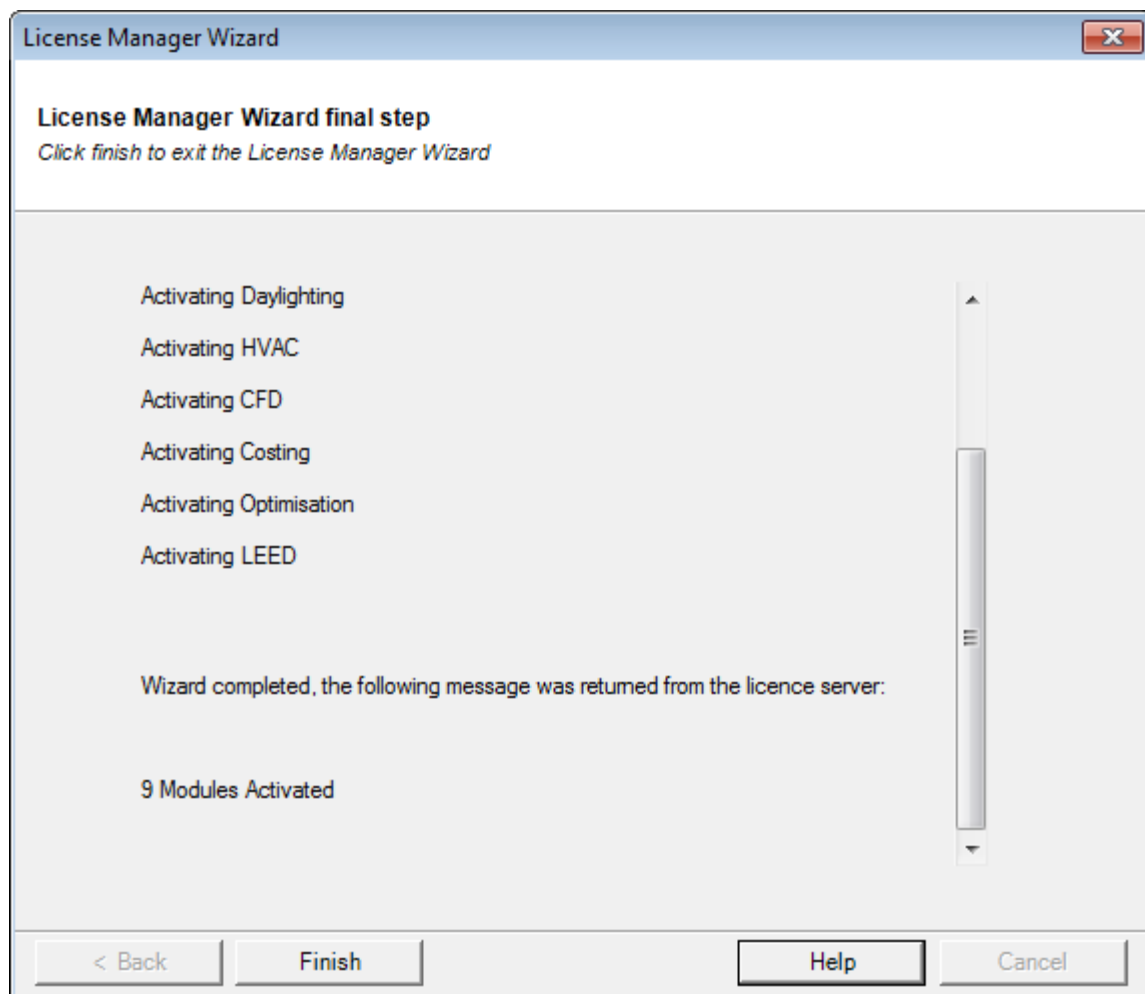
Activation Code
Enter the Activation Code, then click Next to continue

Activation Code

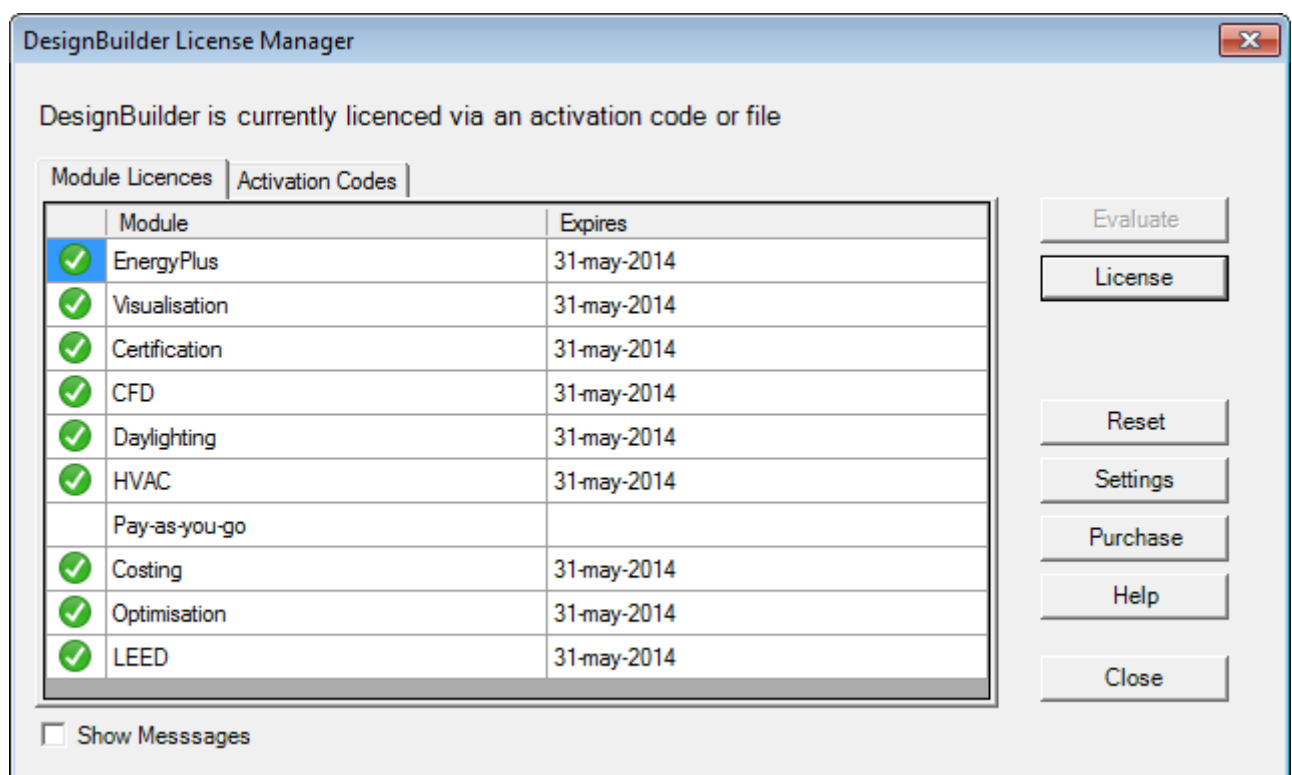
Paste your activation code in here

< Back Next > Help Cancel

You should see a message to say that the various modules have been activated.



When you press Finish, if the activation was successful, you will be shown the Modules Licence tab displaying the licence status of each module.



Note 1: If you purchased a [Single User Licence](#) then you will be able to activate the code on a single computer.

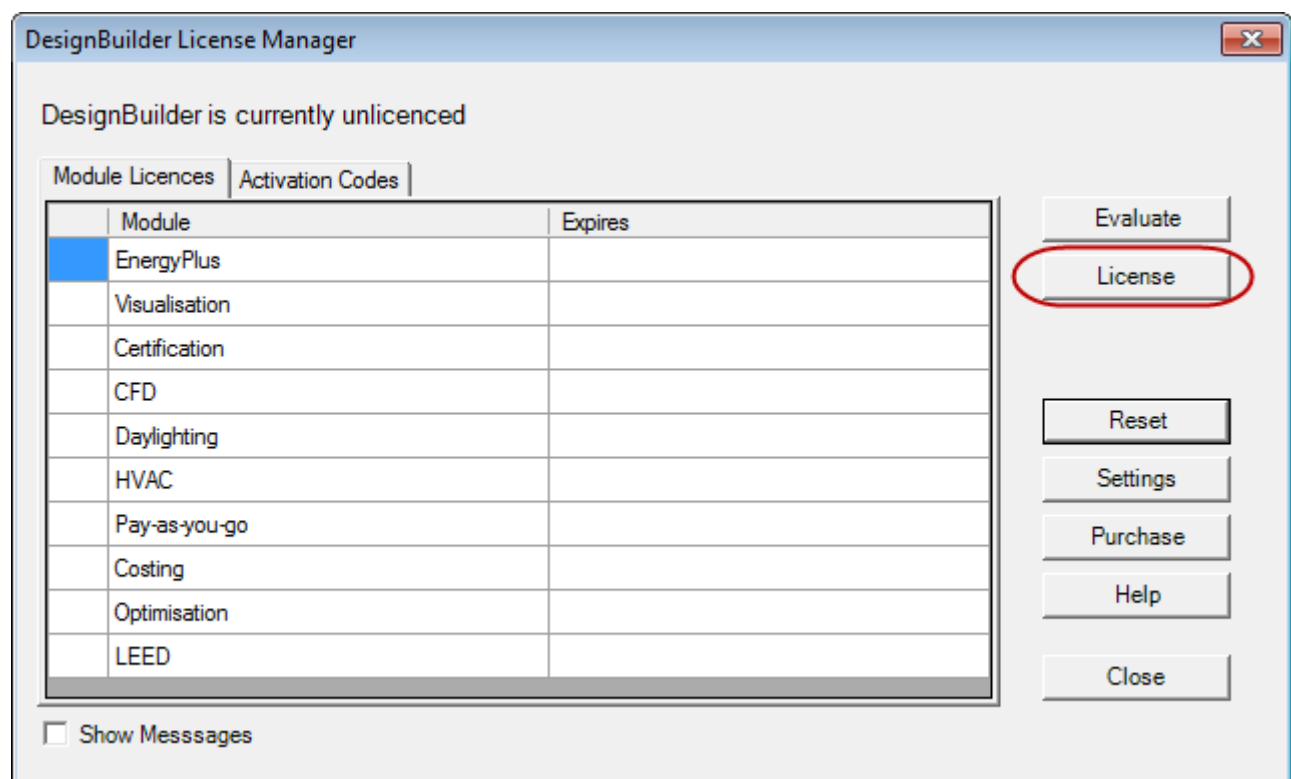
Note 2: You must be connected to the Internet to activate and deactivate DesignBuilder licenses. If your network has a strict firewall policy then you should ensure that DesignBuilder.exe is allowed to access the www.designbuilder.co.uk website.

Licence file

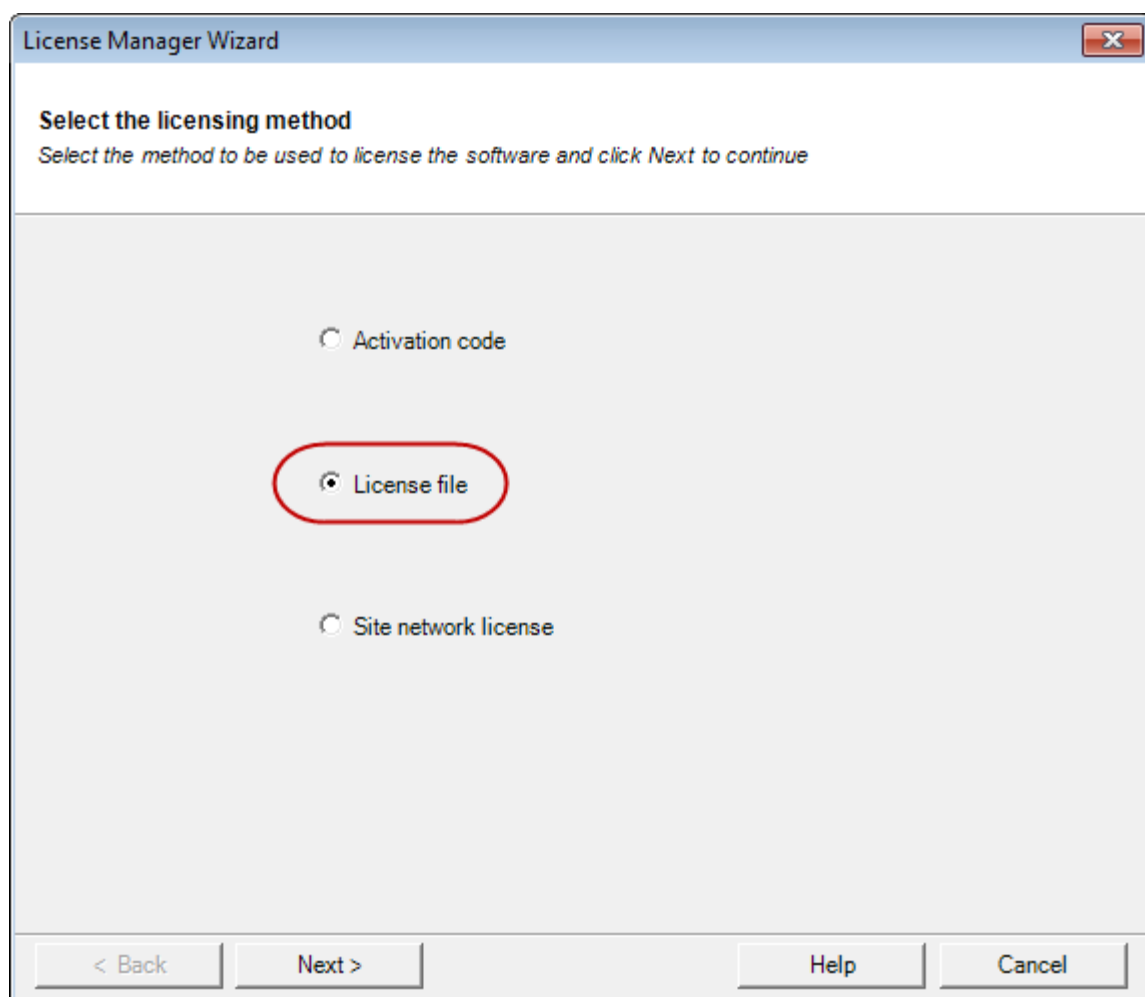
You may have been supplied with a licence file with extension .lic containing your licence or extended evaluation. Typical reasons for using a .lic file are when on a training course where you may not have access to the Internet for activation or for customers who have purchased licences but either do not have Internet access or have strict firewall policies preventing the standard web-based activation from working.

To activate DesignBuilder modules using a .lic file follow the instructions below. Before starting make sure the lic file you received has been saved to a place where you can access it, e.g. desktop, a pen drive, documents folder etc. The location does not have to be permanent as DesignBuilder will store a copy of the lic file in a safe place on the local machine.

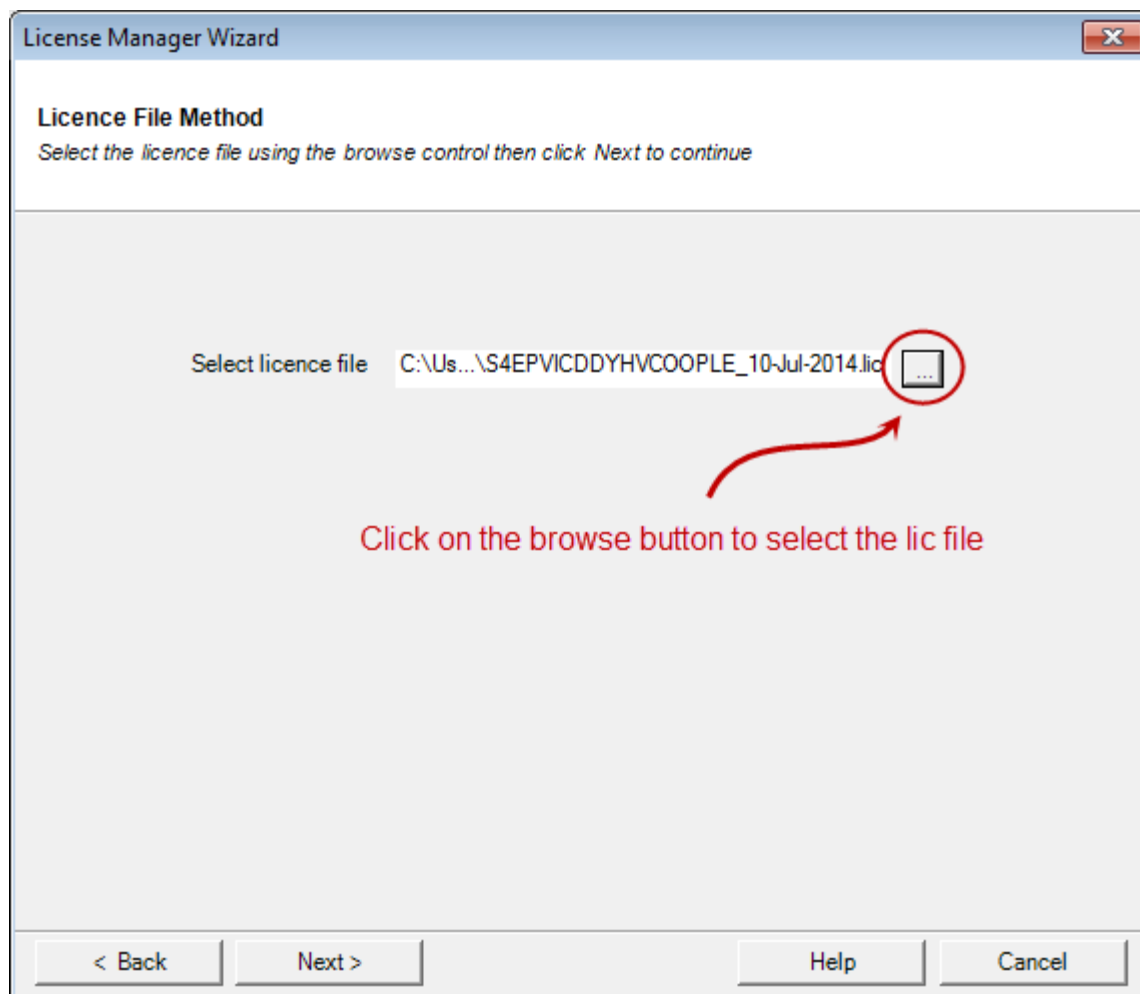
Open the **Licence Manager** dialog (**Help > Licence**). Click on the **Licence** button to open the **Licence Manager Wizard**.



Select the **Licence file** option and press Next.



Click on the browse button, select the Licence file and press Next.



Press the Finish button to complete the action.

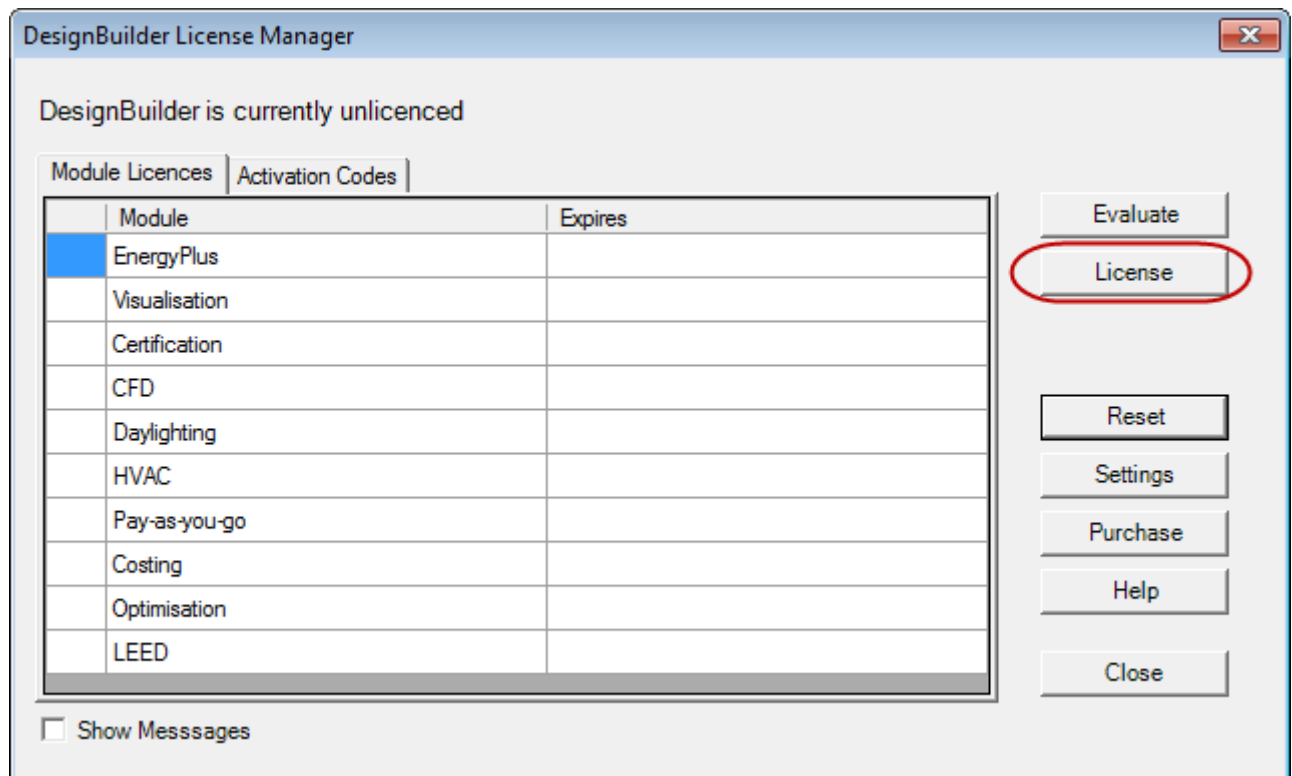
Site Network Licensing (client side)

With site network licenses each client computer on the network can "check out" licenses from the license server application which manages the allocation of licenses. The system is designed to provide a maximum number of concurrent users according to the number of seats purchased. For example if you purchased a site licence with 10 seats for the EnergyPlus module then the software can be installed on any number of computers but only a maximum of 10 people will be able to access DesignBuilder EnergyPlus simulations at any one time. If an eleventh person on the network tries to check out an EnergyPlus license (by starting DesignBuilder with EnergyPlus selected in the License dialog) then they will receive a message to say that the seats are all in use and to wait until one becomes available.

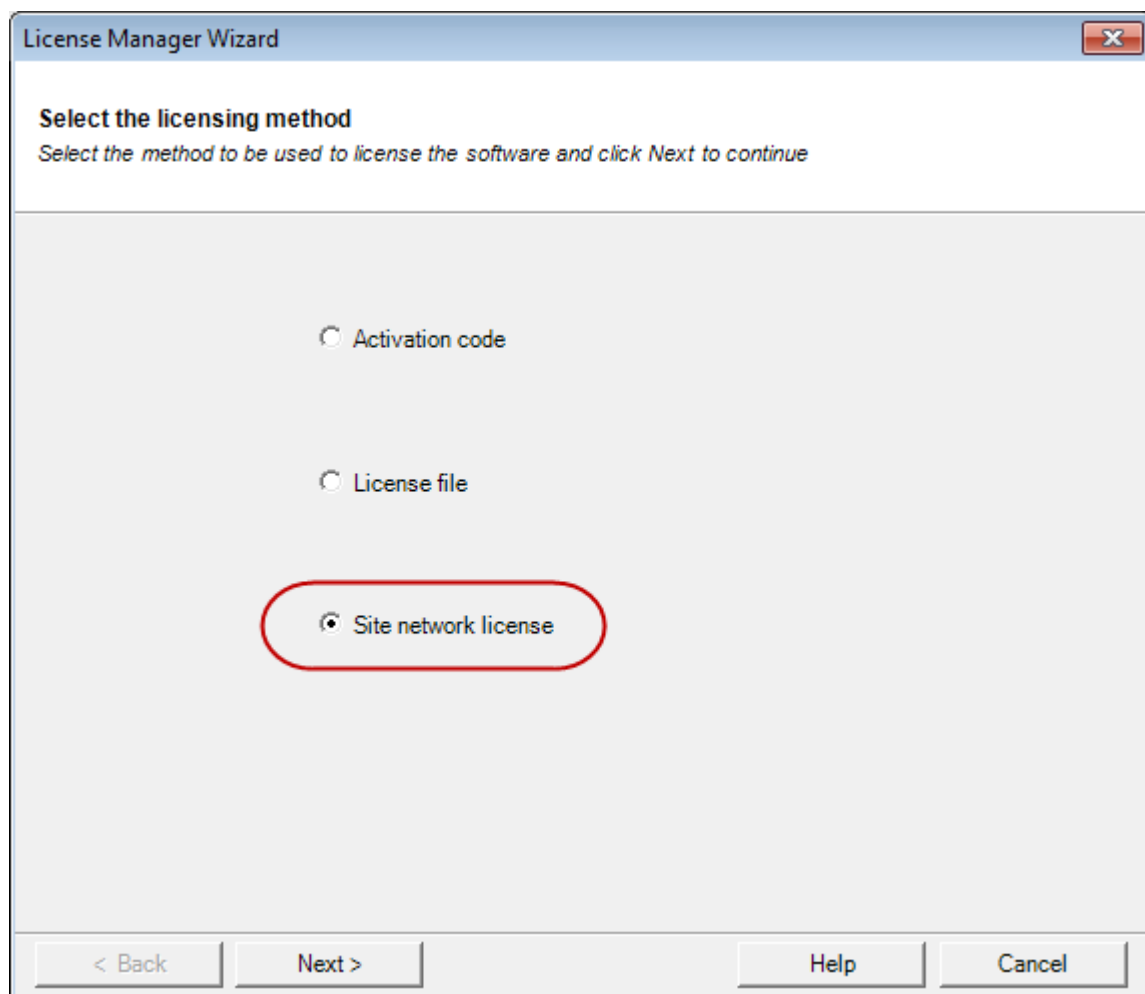
DesignBuilder site licensing requires a simple server application to run on the network to provide licenses. Each client computer has a setting to tell the software to run in site license mode and where to find the Licence Server on the network. The site licensing system is quite straightforward but does require a network administrator with adequate IT skills to set it up.

To set up site network licensing on a *client* computer (i.e. the computer that is to run DesignBuilder) follow the instructions below.

Open the **Licence Manager** dialog (**Help > Licence**). Click on the **Licence** button to open the **Licence Manager Wizard**.

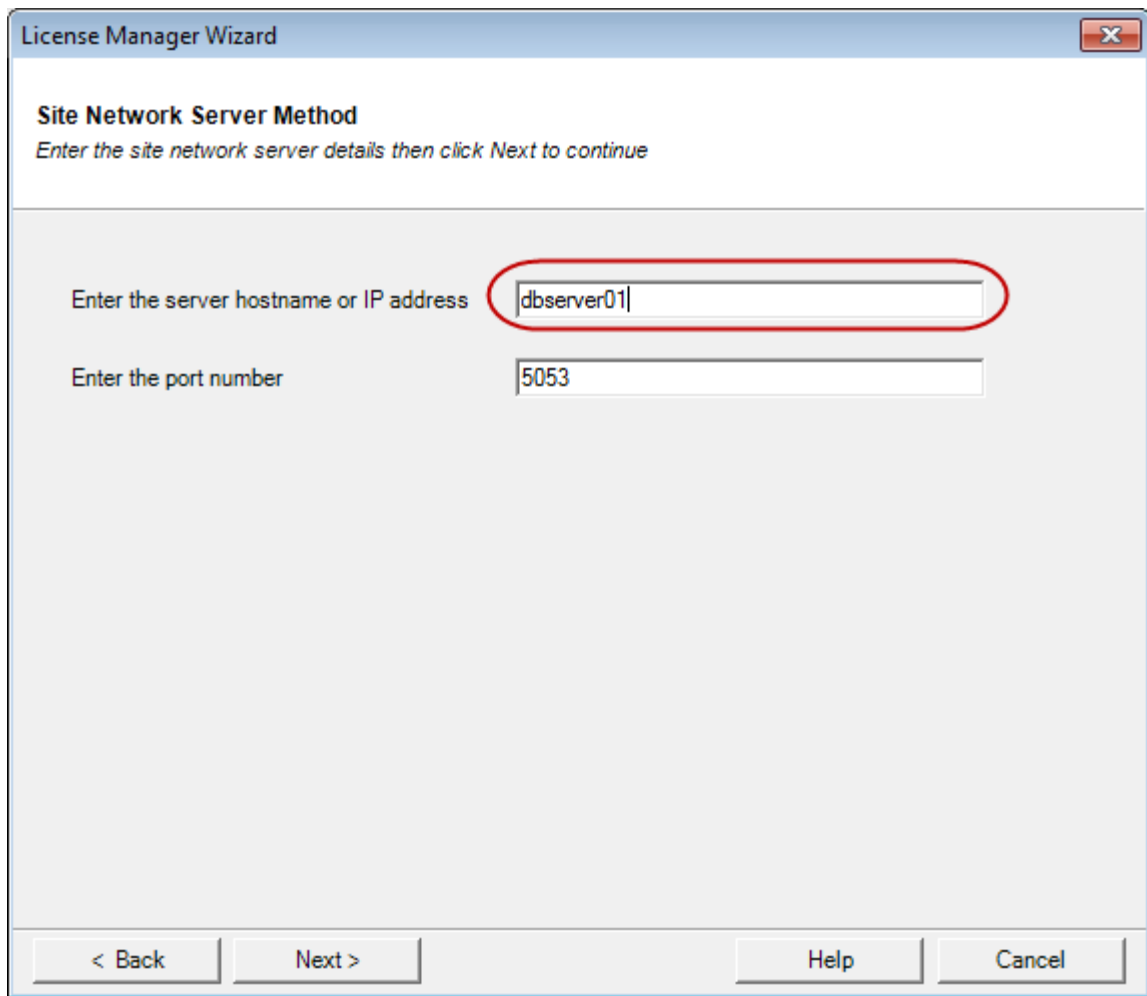


Select the **Site network licence** option and press Next.



On the second page enter:

- a. Either the name of the server or its IP address.



The image shows a Windows-style dialog box titled "License Manager Wizard". It has a close button (X) in the top right corner. The main heading is "Site Network Server Method", followed by the instruction "Enter the site network server details then click Next to continue". There are two input fields: the first is labeled "Enter the server hostname or IP address" and contains the text "dbserver01"; the second is labeled "Enter the port number" and contains the text "5053". The first input field is circled in red. At the bottom, there are four buttons: "< Back", "Next >", "Help", and "Cancel".

You can find the IP address of your server by typing **ipconfig** in the Windows Command box on the server machine.

```

C:\WINDOWS\system32\cmd.exe
C:\>ipconfig

Windows IP Configuration

Ethernet adapter VMware Network Adapter VMnet8:

    Connection-specific DNS Suffix  . : 
    IP Address. . . . . : 192.168.87.1
    Subnet Mask . . . . . : 255.255.255.0
    Default Gateway . . . . . : 

Ethernet adapter VMware Network Adapter VMnet1:

    Connection-specific DNS Suffix  . : 
    IP Address. . . . . : 192.168.254.1
    Subnet Mask . . . . . : 255.255.255.0
    Default Gateway . . . . . : 

Ethernet adapter Local Area Connection:

    Connection-specific DNS Suffix  . : 
    IP Address. . . . . : 192.168.1.16
    Subnet Mask . . . . . : 255.255.255.0
    Default Gateway . . . . . : 192.168.1.254

Ethernet adapter Wireless Network Connection:

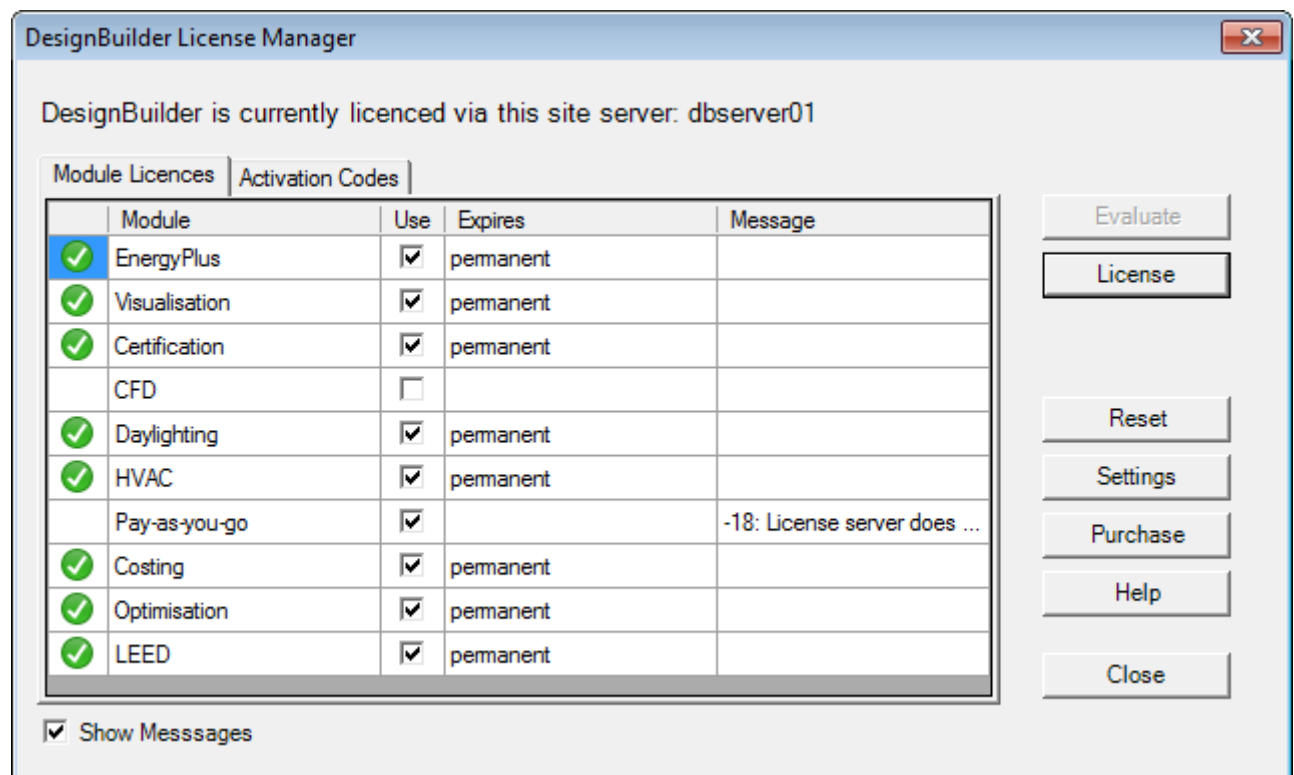
    Connection-specific DNS Suffix  . : 
    IP Address. . . . . : 192.168.1.15
    Subnet Mask . . . . . : 255.255.255.0
    Default Gateway . . . . . : 192.168.1.254

Ethernet adapter Bluetooth Network Connection:
  
```

- b. The port number which will usually be 5053.

When you click the Finish button the connection to the License Server is established based on the port number and IP address you provided on the dialog.

If the site network licence connection was successful you will be shown the Modules Licence tab displaying the licence status of each module.



Once back to the main License Manager dialog you can check out or check back in module licenses through the check boxes in the Use column for each module.

If the server host name or IP address was incorrect, if that server is not running or if the Licence Server service isn't running then the licences will not be accessed and error messages are displayed.

More Detailed Information On Licensing

- [Setting up a Licence Server for Site network licenses](#)

Single User Licences

A single user licence allows you to install DesignBuilder on a single computer. You can move the licence from one computer to the other freely. For example, if you need to use the software on a second computer you can deactivate the licence on the first one and activate on the second.

When you order a DesignBuilder single user license, you will receive an 8-digit activation code in an email. To use this to activate the software follow these steps:

1. Download and install DesignBuilder (if you haven't already done so).
 - a. For **Simulation** work, download from the main [Downloads > Release Software](#) menu option.
 - b. For **UK and Ireland Energy Assessors**, download from the [Downloads > UK Energy Assessors](#) menu option.
2. Run DesignBuilder.
3. In DesignBuilder, open the [License dialog](#) which can be accessed from the top Help menu
4. Press the **License** button
5. On the "Select the licensing method" page, select the **Activation Code** option and press the **Next** button.
6. Fill out your user details including your email address, name etc. and press the **Next** button.
7. On the "Activation Code" page, enter the Activation code. The easiest way to do this is usually to open the email you received with the activation code, copy it so the text is on the clipboard and paste the code into the dialog.
8. Ensure that you have an Internet connection.
9. Then press the **Next** button.
10. This will cause DesignBuilder to automatically download licences for all modules associated with the activation code and activate them. You will then be able to use the software and run functionality associated with each activated license. Press the **Finish** button.

More details can be found in the [License dialog](#) help section.

Setting Up A Licence Server For Site Network Licenses

Site network licenses require a license server application to be installed and run on a server on the network. DesignBuilder runs on the client machines on the same Windows network and obtains licences from the license server on a concurrent basis.

Summary Of Process

In summary, the steps to set up DesignBuilder Site Licenses are:

1. Download the **designb_rlm_v11.zip** file from the [Download > Server Software](#) area of the DesignBuilder website
2. Extract the files in the **designb_rlm_v11.zip** file to the folder on the server to be used to store DesignBuilder site license files (e.g. c:\RLMSERVER)
3. Obtain the License file by clicking on activate.cmd, following instructions on the webpage and saving the license file to the site license folder.
4. Run the RLM server as a service and start the service.

5. On client machines which are to run the DesignBuilder software, open the Licence Manager dialog (**Help > License** menu option) and enter the ip address or hostname of site license server host.

The above steps are detailed below.

Detailed Steps

Obtain your License File

With Site Network Licenses the License file is obtained through the activation code you received when ordering the software.

- a. Before you run the RLM server you will need to obtain your license file and save it into the RLM installation folder. That's the folder containing rlm.exe and other files listed above. We recommend setting up the rlm folder on a local drive on the server itself.
- b. You should have received an activation code from your site license purchase. This is an 8-digit number and is needed to activate the license.
- c. Open the RLM installation folder in Windows Explorer and double click on **activate.cmd**
- d. This loads a web page at www.designbuilder.co.uk with a form for you to enter your activation code.



DesignBuilder
Software **Network License Activation**

Please enter your **activation code** and click **Activate License** to generate the license file.

Activation Code:

Hostid:

IP Address:

Hostname:

This produces a license file activated specifically for your network license server host.
You will need to download the license file and save it for use by the RLM license server.

- e. Other values in the form such as Hostid are automatically filled in for you. After entering your activation code click **Activate License** and if successful you will be presented with a download link for your activated license file.
- f. Download the license file and save in your **RLM installation folder**.

If your server cannot access the Internet

If your server computer has no internet connection then activate.cmd will not be able to connect a browser window to the designbuilder.co.uk web server and you will have to activate manually by following these steps:

1. Open a command window with admin rights on the server.
2. Change directory to the RLM folder (e.g. type "cd \RLM SERVER").
3. Obtain your server hostid by running the command **rlmutil rlmhostid -q 32**
4. Obtain the ip address using the command **rlmutil rlmhostid -q internet**
5. Obtain the hostname with command **rlmutil rlmhostid -q host**
6. Make a note of the **hostid, ip address, hostname** and your DesignBuilder **activation code** and go to a computer with internet connection and run:

https://www.designbuilder.co.uk/license/activate_site.php

7. Enter your details in to the form and download the resulting .lic file.
8. Place a copy of the .lic file in the site license installation folder on your host server.

Running the RLM Server

The server can be run directly by double clicking [rlm.exe](#). This runs a command window and logs activity to the window. This is useful when you first run the license server as you'll be able to see the license server starting up correctly and loading the license file as well as DesignBuilder clients obtaining licenses from the server.

Running RLM as a Service

It is best to set RLM up to run as a Windows service rather than in the command window, otherwise you'll need to remember to manually restart the license server when the computer reboots.

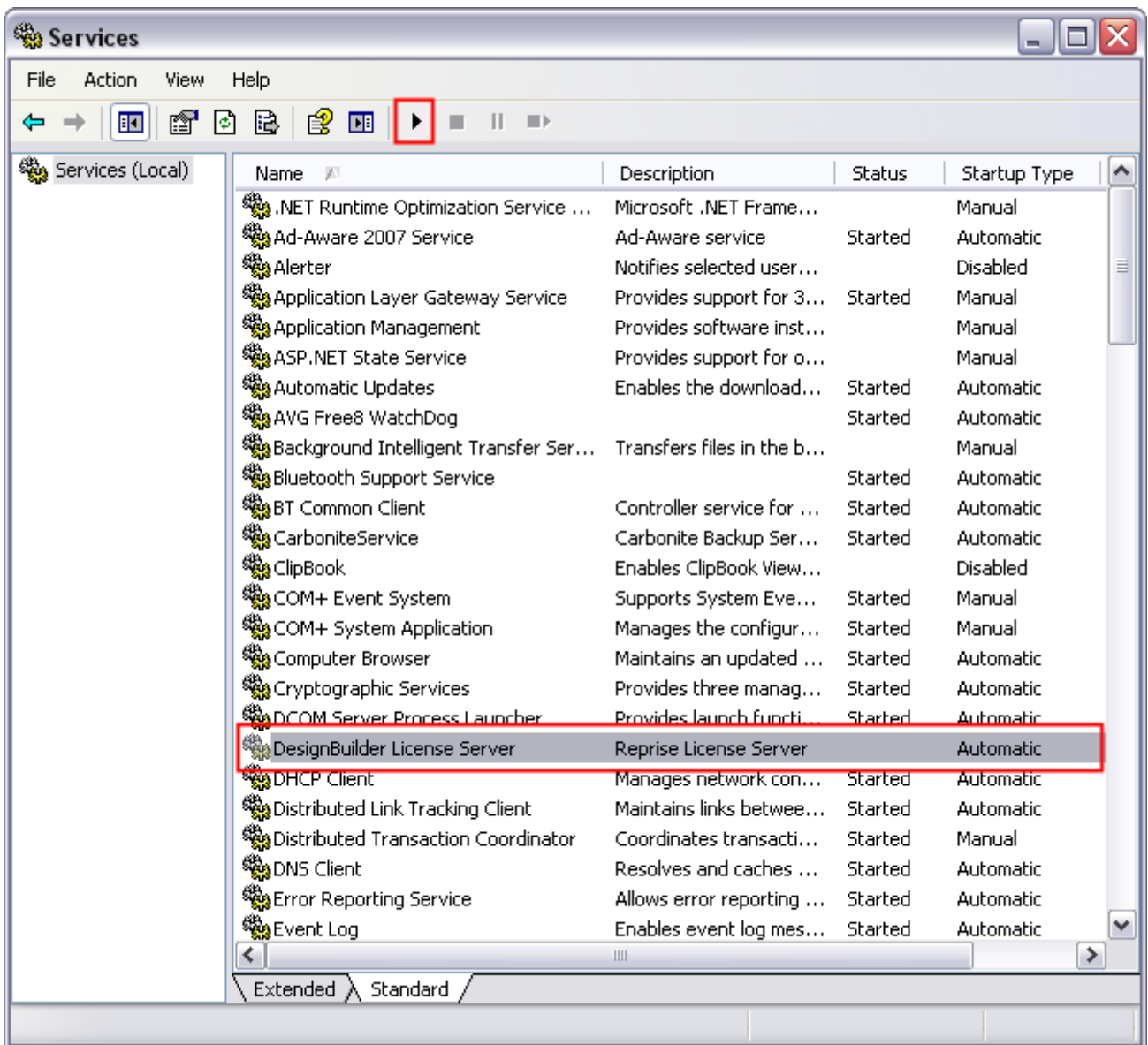
If you are running RLM in a command window you should close that before starting up RLM as a service.

To create the Windows service to run RLM automatically double click on [service_install.cmd](#) or use **Manage Windows Service** in the admin interface.

If the service_install.cmd does not run then open a command window with elevated privileges from Start> right click on Command prompt to run as admin, then cd to RLM folder and type service_install and press Enter key. See below for details.

Note: The service will be added to Windows services but not started.

Use Control Panel > Services to start **DesignBuilder License Server**.



Once started in this way the service will restart automatically if the computer is rebooted. By default the service will write a log file **designb_rlm.log** into the rlm installation folder. If you want this in a different location you can use the admin interface (above) to set up the service then you can specify the service name and the log file location.

You may experience problems with permissions when setting up the service. You can either disable the UAC or try right clicking **service_install.cmd** and choosing "Run as administrator".

Even this doesn't always give you access to the Service Control Manager. You can check this by running a cmd prompt, changing folder to the license server installation folder and running the RLM install service command (below).

If you edit **service_install.cmd** you'll see the command being executed is:

```
rlm -install_service -dlog "%CD%\designb_rlm.log" -service_name "DesignBuilder License Server" -c "%CD%"
```

If you see this error returned "Error: Access to Service Control Manager denied" you can get over this by running cmd as administrator and then run the install command. To do this right click Command Prompt on the start menu and run as administrator. Change folder to the license manager installation folder and run the **rlm.exe -install_service** command given above and it should work.

In case of trouble, add inbound windows firewall rules to server machine, to always allow TCP ports 5053, 5054. If you set fixed ISV port for port-forwarding method of off-site access (See Troubleshooting section)

then also add and fixed ISV port e.g.49670 as an inbound TCP port-allow rule. If necessary, also allow TCP ports 2600 and 2700 inbound.

More information on this can be found in the Reprise License Manager Knowledge base at <https://reprisesoftware.com/knowledge-base>

RLM Server Admin Interface

RLM provides a browser-based administration interface to the license server. To run this, double click on the **admin_interface.cmd** file in the RLM folder using Windows Explorer.

Refer to the RLM End User Manual (double click **online_manual.cmd**) for more details on using the admin interface.

Note: DesignBuilder does not use the Activation page of the admin interface.

Reprise Software, developers of the RLM licensing system, always maintains a current version of the RLM End User's Manual here:

<https://reprisesoftware.com/docs/admin/basics-admin.html>

Adding a lic file for extra seats

If you purchase additional seats for a site network license system, the new licenses are provided in the form of a lic file download. The process of obtaining the additional lic file is the same as described above under [Obtaining your License File](#). The new lic file does not replace the existing lic files, but is used in addition to the existing licenses. The license server reads these multiple lic files, licenses from multiple lic files adding to give the total number of seats of concurrent use that are allowed.

Setting up Client Machines

This steps involved in pointing client machines to the License Server is described on the main [Licensing & Activation](#) page. From DesignBuilder you can check out or check back in module licenses using the License Manager which can be accessed from the Help menu.

Troubleshooting

Unreachable server

Normally when the Client machine can't see server it displays "-17 communications error" and "-111 Connection refused at server". In that case the access to the server is probably being blocked.

The RLM Licence server uses random ports. So, the best solution is not to allow exceptions by port, but instead by program, eg. open all ports used by "rlm.exe".

But if the server is behind a firewall that you have to specify pinholes to get access then follow these instructions:

In order to specify what port the second service uses you will have to edit the lic file.

On the ISV designb line of the lic file (that will be the 2nd line) add the following:

- port=nnnnn
- where nnnnn is the port number. There must be a space before the word "port".

Usage of licence server from off-site (via WAN)

Method 1: VPN with a sufficiently fast connection.

The remote user will need to login through your VPN connection and access the server that is hosting the network license. DesignBuilder site licence is IP based.

Method 2: Port forwarding

After activating the site license activation code, download the .lic license file, open the file in a text editor, with admin rights. Example:

```
HOST localhost 780bc61a  
ISV designb port=49670  
LICENSE designb designbuilder_n.
```

Now, on the second line you should see "ISV designb" but the "port=..." part is probably not present.

Be sure to insert after "designb", a space, followed by "port=", followed directly by a port number – one example is 49670.

Then save the lic file and restart the service.

This instructs the licence server to use a fixed "ISV port" as specified, instead of randomly choosing the port number from a range each time the server is started (default behaviour, good for security but it makes port-forwarding for off-site access unachievable).

In the router used by the licence server, set up port forwarding from WAN TCP port 5053, to LAN TCP port 49670 or whatever you chose as your fixed ISV port. Forward to the LAN static IP of the licence server machine.

All clients, whether on LAN or WAN, must then access the server using port 5053. When on WAN, for host name they must use the server's WAN static IP or DDNS host address as the host name. This means when a staff member moves in/out of the office with their laptop, they have to switch the hostname in their DesignBuilder (client) licence settings (unless you have something clever set up on your network to make this seamless). The VPN approach is probably more convenient in this respect.

Note: It is the customer's responsibility to consider security concerns when opening ports etc.
DesignBuilder Software Limited shall not be held responsible for damages due to a security breach.

The DesignBuilder licence server is based on the Reprise Licence Manager (RLM) software in case you have prior knowledge of this platform.

Program Options

The Program options dialog is accessed from the [Tools menu](#).

The Program options apply to the overall application no matter which file is loaded. There are six tabs:

- [User interface](#)
- [Dialogs](#)
- [Files](#)
- [EnergyPlus](#)
- [Limits](#)
- [International](#)

See also the [Program and Model Options](#) tutorial

Program Options - User Interface

User Interface tab of the [Program options](#) dialog

Learning

You should generally select **Learning** mode while you gain experience with DesignBuilder. In Learning mode, the [Info panel](#) is shown to the right of the main screen, providing additional information and commonly used tools relevant to each screen. Model data is accessed from tabs along the top of the screen.

With Learning mode switched off, the Info window is not shown and Model data is shown on the right of the screen alongside the Model edit screen. This allows you to see the effects of updates to data as they are made. The Program help commonly refers to this setting as "non-Learning mode".

See the [New Project and Learning Mode Tutorial](#)

Help system

Select one of:

- **1-Web-help** - the default option where the latest help is displayed from the DesignBuilder servers using the **Web Help** system. This option has the advantage of providing the most up-to-date help but it requires an Internet connection and is slower than...
- **2-Local help** - a chm format help file called **Manual.chm** is downloaded. Once the help file is downloaded no Internet connection is required for help.

Technical details on the 2-Local help option

When using **2-Local help**, the help file is called **Manual.chm** and is downloaded and stored on the local hard disk at:

C:\Users\<Username>\AppData\Local\DesignBuilder\Help\V6.0

You can download the help file once while you have an Internet connection and manually copy it to other computers as required when on or off-line, remembering to select the **2-Local help** Program option on the other computers.

Use Slider controls

Some significant data items are (by default) controlled using a 'slider' which allows rapid selection of typical values. If however you need to enter precise values (for example when modelling an existing building) or if you need to enter values outside the slider range you may prefer to type this data in a text box. In this case uncheck the 'Use slider controls' data. Common reasons for this are to enter [Computer gain](#) densities for IT suites or dealing rooms.

Note: when using DesignBuilder with [IP units](#), slider controls are always switched off regardless of this setting.

Sort lists

By default DesignBuilder automatically sorts navigator and other data lists alphabetically. This can make it easier to find the data you are looking for but also increases the time taken to generate lists and so slows some program operations requiring lists to be generated. You can request unsorted lists by unchecking this option.

Line smoothing in the display

DesignBuilder can optionally smooth lines in the Edit screen to provide a crisp clear view of axes, North arrow, construction lines etc without 'jaggies'. Note that this option does not affect the surfaces of the model itself.

On some less powerful computer displays, some line types in the model edit screen can be hard to see with this option selected. You should ensure that this option is unchecked if you have this problem on your computer. The default setting is off.

If you have a good quality graphics card you may prefer to leave this option unchecked and switch on hardware antialiasing on the control panel provided by the software that comes with the graphics card. This will improve image quality without the need to change DesignBuilder settings.

Automatically refresh site navigator lists

By default DesignBuilder automatically refreshes the contents of the site navigator list on the left of the screen as the building is modified so it is in synch with the model. However, when working on large models, frequent rebuilding of the navigator list can slow progress especially when drawing partitions or renaming zones. You can uncheck this option to avoid this problem.

Warning: You should be careful when using this option as the Navigator list can quickly become out of date when you are editing a model. You should be especially wary of clicking on objects in the Navigator which no longer exist in the model as this can cause DesignBuilder to crash. For this reason **we can only recommend unchecking this option for advanced and "careful" users!**

You can manually refresh the navigator lists by clicking on the [refresh list](#) icon above the Navigator panel (below).



Double click to navigate site hierarchy

By default you can navigate to an object in the site hierarchy (building, block, zone etc) by single-clicking in the Navigator. You can change this behaviour by checking the [Double click to navigate site hierarchy](#) program option. In this case you must double-click to go to objects in the Navigator and a single click highlights the object in the Edit view (if you are already at the level above). For example if you are at block level and you single click on one of the zones in the block the zone is highlighted in red in the Edit view.

Tip: Enabling this option can be useful for identifying blocks, zones, surfaces by single-clicking in the Navigator. It can also save time when renaming many zones at once from block level as it becomes possible to click on the zone Navigator entries to rename them without navigating down to each zone and then having to navigate back up again to block level to rename the next zone.

Show 'Load data from template' dialog when template selected

When you select a new template by clicking on the browse button in the Model data, by default the data in the template is loaded directly to the current object. However if this option the 'Load data from template' dialog is opened giving you the opportunity to select various options and load other templates. This option is switched off by default.

Show model data departures from template in bold

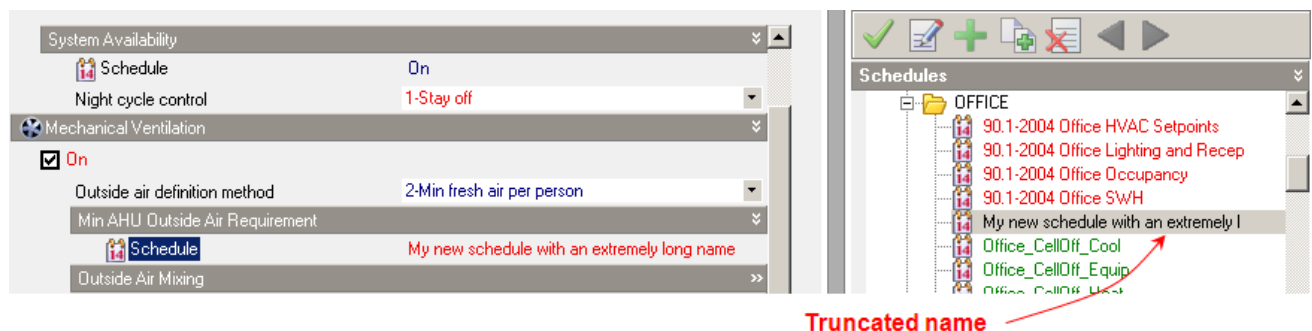
Select this option to show any differences in model data relative to the selected template in bold. This option is switched on by default.

Default surface normal view

When working at [surface level](#) it can be useful to get a normal view of the surface by clicking on the [Normal button](#) in the top right of the screen. By default this will show a normal view of the surface as viewed from **outside**. You can use the **Default surface normal view** program option to instead view from the **inside**.

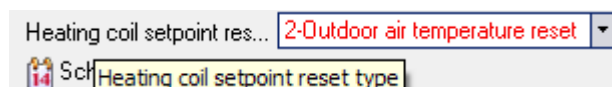
Maximum characters shown in data display lists

You can control the number of characters displayed in data lists on the right hand panel. This provides a workaround to a problem with the list component used in DesignBuilder which always scroll to the right when an item with a long name is selected. This scrolling can be inconvenient so we have implemented the option to control the amount of text displayed. By default this is set to 35 characters but you can use a larger value if you prefer.



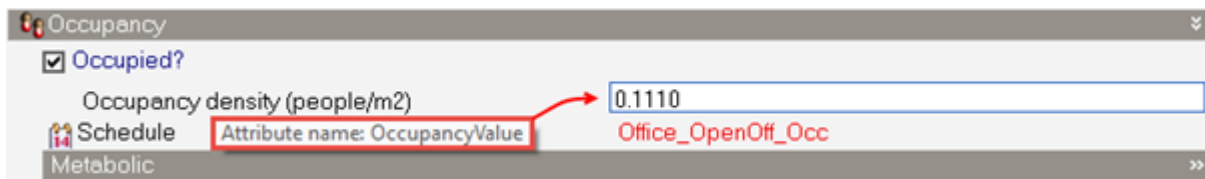
Show tooltips in Model data

With small window sizes or when Learning mode is switched off there is not always enough space to display the caption for each Model data item so DesignBuilder provides the option for tooltips to appear when the mouse hovers over a truncated caption. See screenshot below.



Show attribute names in tooltips

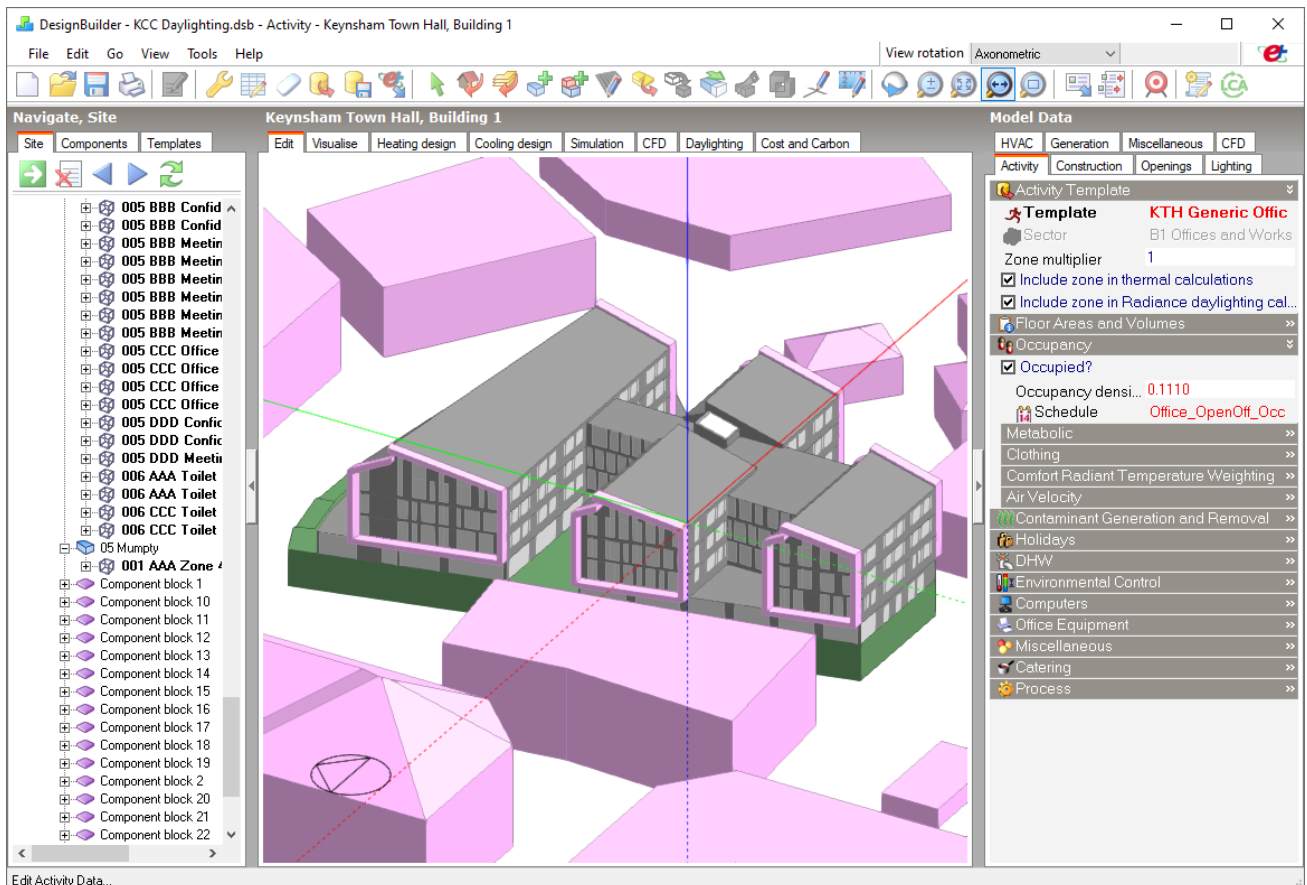
When the **Show tooltips in Model data** option is active you can choose to display the names of model attributes in tooltips that appear over user interface items. These may need to be accessed when writing EMS, CS-Script and Python scripts and plugins.



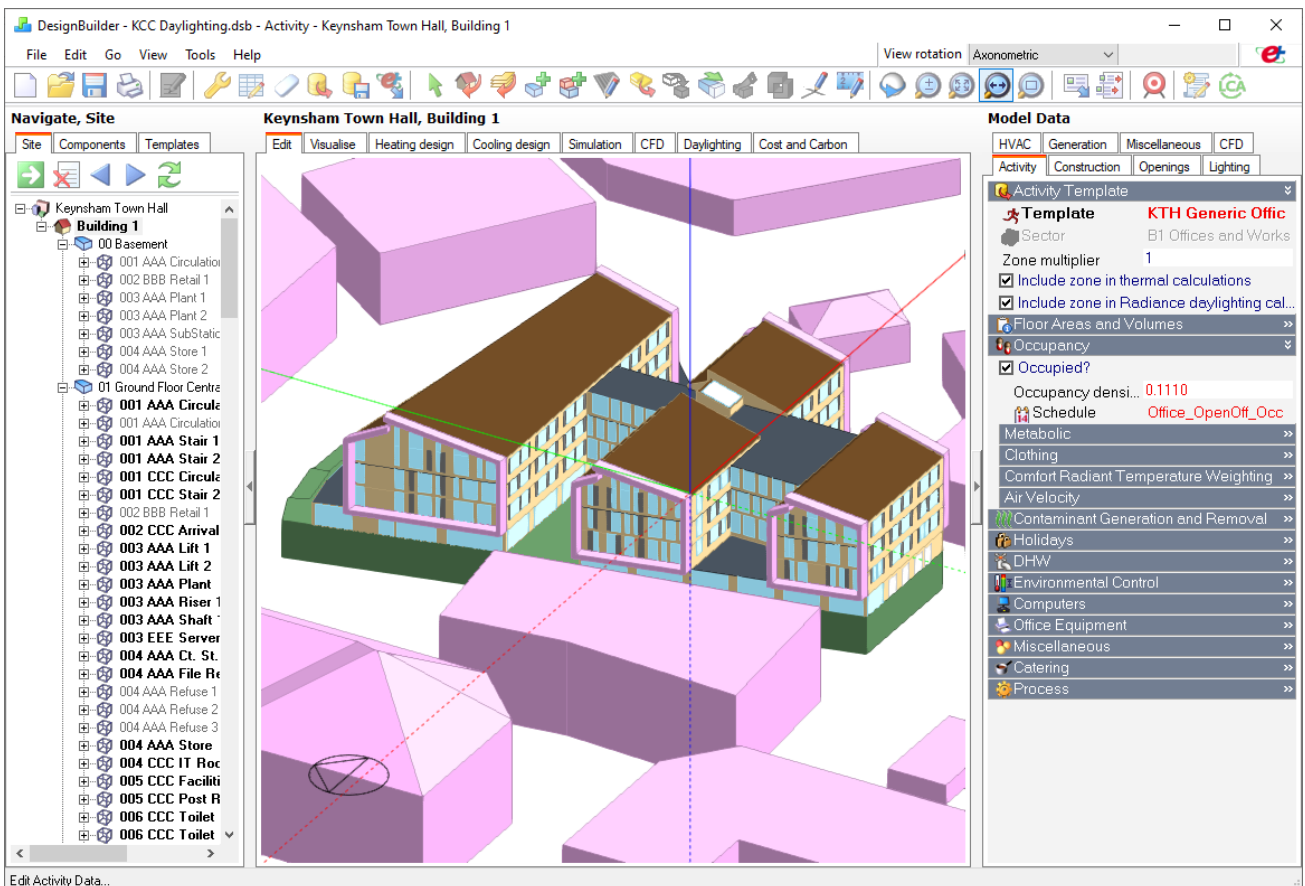
Tip: A full list of all model attributes can be found in the [Extensibility Tools User Guide](#).

Colour scheme

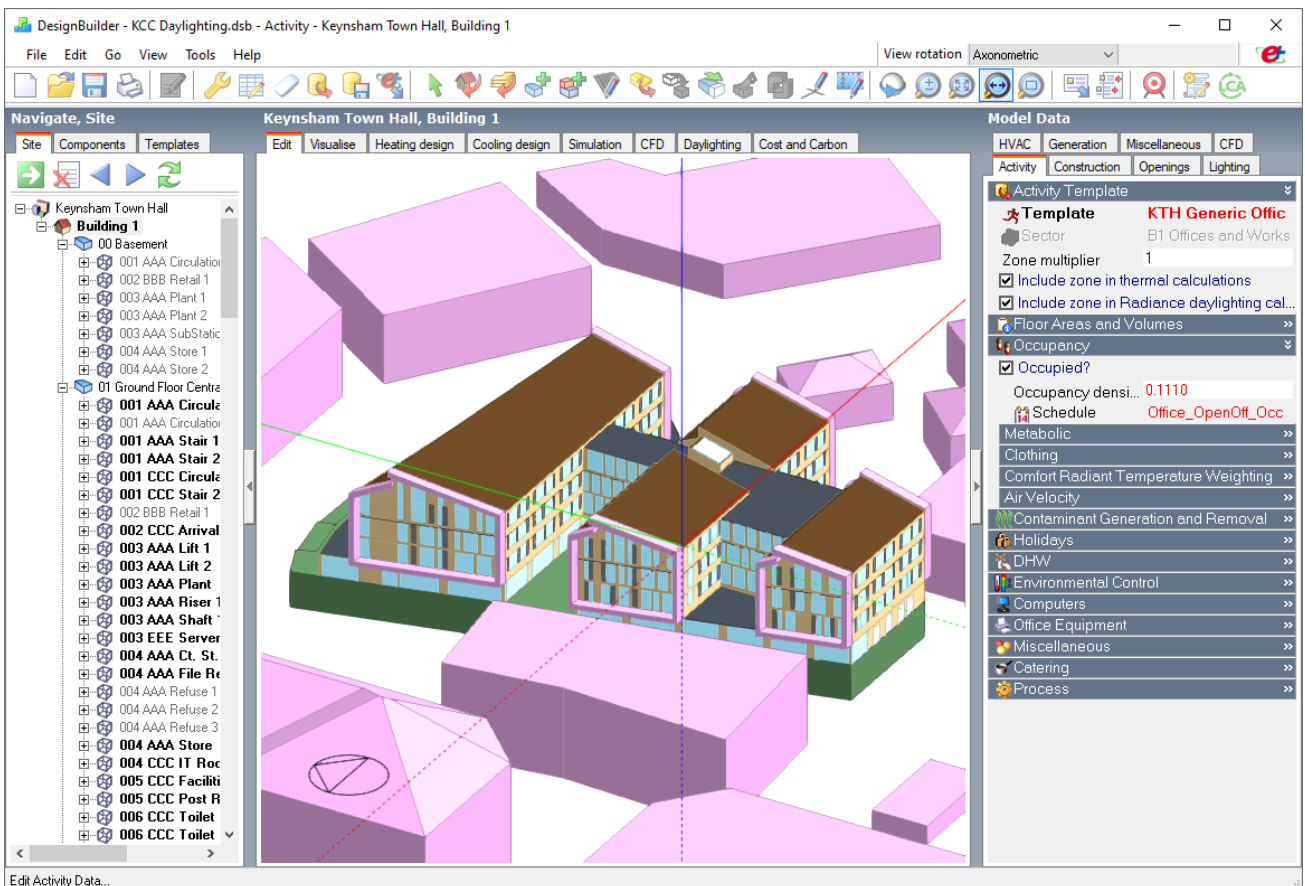
You can control most of the colours used in the DesignBuilder user interface by selecting from a list of pre-defined Colour schemes. The screenshots below show how the 3 Colour schemes provided with DesignBuilder look when loaded.



Classic (Default scheme)



Architectural light



Architectural

Tip: To further customise the appearance, you can create your own Colour scheme and select that. When creating your own Colour scheme, it is best to start by copying one of the existing schemes and editing the copy.

Program Options - Dialogs

Dialogs tab of the [Program options](#) dialog

Auto calculate

Check this option if you prefer DesignBuilder to automatically start calculations (typically opening the calculation options dialog first) when you move to a new calculation tab. For example when you click on the Simulation screen and there aren't any results available to display yet, by default DesignBuilder shows the Update button, giving you opportunity to update the results. However if you activate the **Auto calculate** program option, DesignBuilder will instead open the Simulation calculation options dialog. This was the default behaviour in v5.1 and earlier, but in v5.2 and later it is necessary to click on the Update button to start a calculation.

Calculation dialogs

These options control whether a calculation option dialog appears by default before every calculation. Note that a dialog always appears when you press Update.

New Object Dialogs

New project

When you create a new site (project), a dialog box appears by default allowing you to enter basic data on the new project. If you prefer not to see this dialog in future and just use default data, uncheck this checkbox.

New building

When you add a new building, a dialog box appears by default allowing you to enter basic data on the new building. If you prefer not to see this dialog in future and just use default data, uncheck this checkbox.

Reports

Default report destination

When you export data (typically by clicking on the [Export](#) toolbar icon) there are 3 options for the destination of the report. You can define which option you most frequently use by choosing from one the available default options:

- **1-File** - save the data as a file.
- **2-Clipboard** - copy the data to the clipboard for pasting into another application such as Word or Excel.
- **3-Report topic** - store the data as a [report topic](#) which can later be included as a page in a report by using the [Compile report](#) option.

EN ISO 673 Glazing U-value Calculations

There are some options for the calculation of the U-value according to EN ISO 673.

Fixed mean gas temperature (10°C)

You can choose to fix the mean gas temperature to 10°C which is the requirement for strict adherence to EN ISO 673. Unchecking this option allows the gas temperature for each gas layer to be calculated individually for a more accurate calculation of gas conduction (and hence the glazing U-value) to be made.

Outside air temperature

You can define the outside air temperature to be used as boundary conditions in the calculation of glazing U-value. The outside air temperature should be 0°C for strict adherence to ISO 673.

Inside air temperature

You can define the inside air temperature to be used as boundary conditions in the calculation of glazing U-value. The inside temperature should be 20°C for strict adherence to EN ISO 673.

Detailed HVAC

Default HVAC zones target

By default in Detailed HVAC when you edit an HVAC zone equipment object, such as an HVAC zone, a FCU a chilled ceiling or a sub-component within zone equipment such as a zone heating coil, any changes made on the dialog apply only to that object. That is **1-This HVAC Zone only** is selected by default on the [Target tab](#) of the dialog. However you can use this selection to choose a different default option allowing changes made to be saved to the equivalent component in all HVAC zones in the group by default if that is what is required. Options are:

- **1-This HVAC Zone only** - the data on the dialog will be saved for the current HVAC zone only.
- **2-Multiple (all settings)** - all of the data on the dialog will be saved for all HVAC zones in the group
- **3-Multiple (modified settings only)** - only the changed settings on the dialog (shown in red) will be saved for all HVAC zones in the group.

Show HVAC zone default data on the HVAC zone group dialog

When an HVAC zone is added to an HVAC zone group, default values are taken from the HVAC zone group. This default HVAC zone group data is normally hidden because it can confuse new users who expect changes made to HVAC zone default settings on the HVAC zone group dialog to be inherited by child HVAC zones.

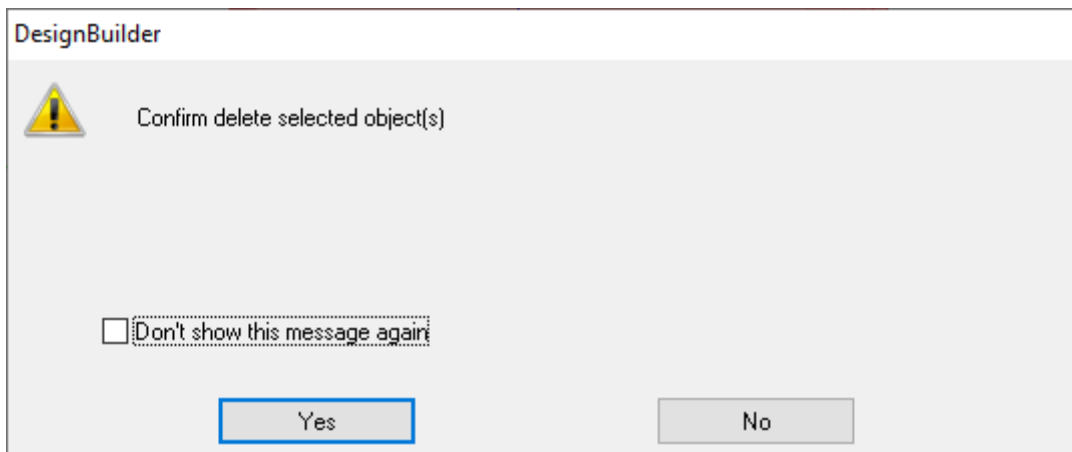
This option allows more experienced users who understand that the HVAC zone group defaults are only used at the point when the HVAC zone is added to the group to control defaults. It is switched off by default.

Tip: The most important application of HVAC zone group default data is when [saving HVAC templates](#) where specific HVAC zone settings are required. Any HVAC zone settings made in the source model are not saved with an HVAC template, but the settings on the HVAC zone group are saved and applied to zones added to the HVAC zone group when the template is subsequently loaded in the destination model.

Hideable Messages

Confirm object deletion

Check this option if you would like DesignBuilder to prompt you to confirm before deleting any objects in the model. You may need to use this option if you have previously checked the checkbox on the delete object dialog (below) and would now like to change back to the default behaviour of confirming before deleting.



Program Options - Files

Files tab of the [Program options](#) dialog

Backup

DesignBuilder will by default make automatic backups of your current working file. Check the **Automatic backup** option box and select a backup interval to enable this option.

DesignBuilder copies the auto backup file to the [Backup](#) folder below [DesignBuilder Data](#) folder.

See also [Data recovery](#).

Note: DesignBuilder keeps files in the [Backup](#) folder for up to 30 days and removes old backups each time the program is run.

Saved Models

Save DesignBuilder library data to models

Warning: Switching this option on causes the entire DesignBuilder library to be stored in every dsb model file that you save. This will cause dsb files to become larger than they need to be. This option should normally be switched off. You should only switch it on if advised to do so by DesignBuilder Support.

DesignBuilder XML Files

Include library data in exported DesignBuilder XML

This program option setting can be used to control whether or not library data is included in exported dsbXML files. Available options are:

- **1-None** - the default option, excludes library data from the XML, resulting in significantly smaller file sizes.
- **2-All** - includes all library data in the XML, resulting in a self-contained data set but a very large file size.

When exporting for subsequent importing to DesignBuilder v2025.1 and later, you should generally use the default **1-None** option to minimise the file size. In this case, only user-defined components and templates are stored in the exported file.

If, on the other hand, the file is intended for loading into a 3rd party tool or into DesignBuilder v7.3 or earlier, you should generally use the **2-All** program option to include all library data in the tables. This ensures that data for all components and templates referenced within the model is included in the XML, making the data set self-contained. The dsbXML file size can be very large when including all library data.

Note: Library data is identified as any component or template in a data table with an Id value of less than 10,000. User-defined custom data, on the other hand, has an Id of 10,000 or more. Custom data is always exported regardless of the program option setting.

See also:

- "DesignBuilder XML (dsbXML)" on page 2213
- [Export DesignBuilder XML \(dsbXML\)](#)
- [Import DesignBuilder XML \(dsbXML\)](#)

Default Folders

DesignBuilder stores the most recently used folders for:

- Data file storage - default folder for .dsb .skh and .ddf exports.
- Picture file export - default folder for pictures and movie exports.
- Report file export
- DXF file import/export

You can edit these default folders on this tab.

Program Options - EnergyPlus

EnergyPlus tab of the [Program options](#) dialog

EnergyPlus

EnergyPlus version

You can select the version of EnergyPlus to use in your simulations. This can be useful to target a specific version you may wish to use or when a new version becomes available which might provide new functionality or address previous problems.

Tip: You should generally use the recommended DLL option as it is tightly integrated with DesignBuilder and provides progress feedback.

DesignBuilder comes with 2 internal versions of EnergyPlus:

- **DLL** - the default option, runs the internal version of EnergyPlus supplied with the program asynchronously giving visual feedback of progress and allowing interaction with the computer during the simulation.
- **Executable** - an executable version of the program running in a command window.

Alternatively you can select an [executable version of EnergyPlus supplied by DOE](#). If you are [exporting IDF data](#) for use outside DesignBuilder you should select the appropriate DOE version to avoid [incompatibilities](#).

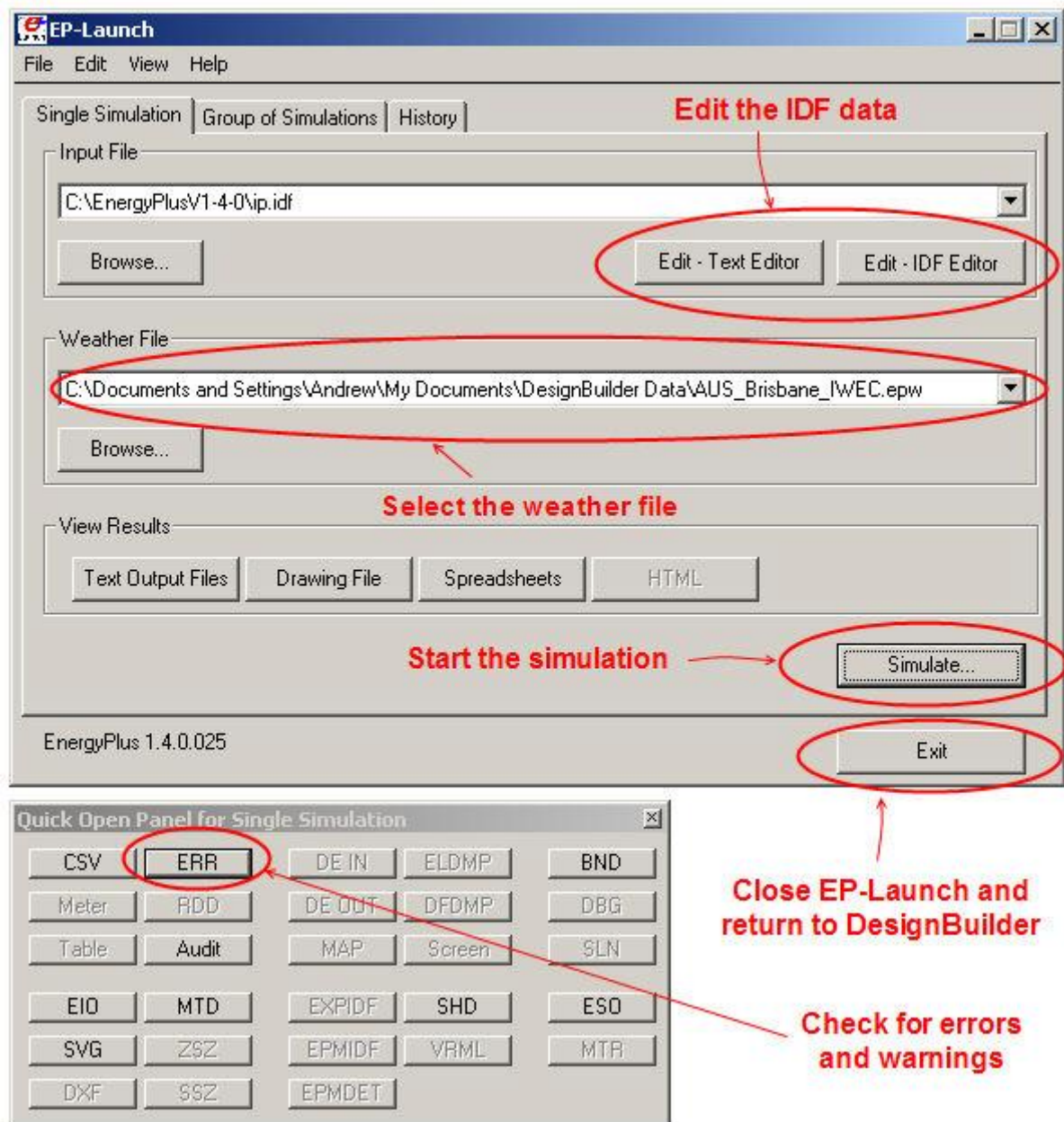
Note: the EnergyPlus version selected here is used for simulations within DesignBuilder but also as the target version when exporting IDF data for use outside DesignBuilder.

Mode

If you select an executable version (internal or DOE) there are four options available to control the way EnergyPlus is executed when simulating within DesignBuilder:

- **1-Executable (minimised)** - runs the executable invisibly - you will just see a minimised window on the Task bar but not the EnergyPlus command window.
- **2-Executable (visible)** - runs the executable showing EnergyPlus screen output in a command window.
- **3-Executable (debug)** - runs the executable showing EnergyPlus screen output in a command window and pauses after simulation - intended for advanced EnergyPlus users who may want to see the stack dump following a crash.
- **4-Executable (EP-Launch)** - runs the DOE-supplied EP-Launch program, waiting until this is closed before loading results. This option is only possible if you have selected a DOE-supplied version of the EnergyPlus executable. This option can be useful for making quick minor changes to the IDF data prior to simulation. Any changes made to the IDF data from within EP-Launch will be included in the simulation but will not be retained for future simulations. **Info on the EP-Launch option.**

The screenshot below shows EP-Launch being used to control EnergyPlus simulations from DesignBuilder.



Note 1: If you use this method you must make sure to set the correct weather file manually.

Note 2: Any changes you make to the IDF data, either in the IDF Editor or using a text editor, will be saved in a file called ip.inp in the EnergyPlus folder. This file will be overwritten the next time a simulation is requested so if you wish to save the any changes you make to IDF data you should use the File > Export option and edit the data and make simulations outside DesignBuilder.

Note 3: If you want to run EnergyPlus without DesignBuilder then use the File > Export > Export EnergyPlus IDf file > Simulation option to generate the idf file and load this using EP-Launch outside DesignBuilder.

EnergyPlus	
EnergyPlus version	Version 1.4.0.025
Mode	4-Executable (EPLaunch)
EnergyPlus export folder	C:\EnergyPlus\1-4-0

EnergyPlus export folder

When using a DOE EnergyPlus executable version you must also select the folder where the selected version is installed.

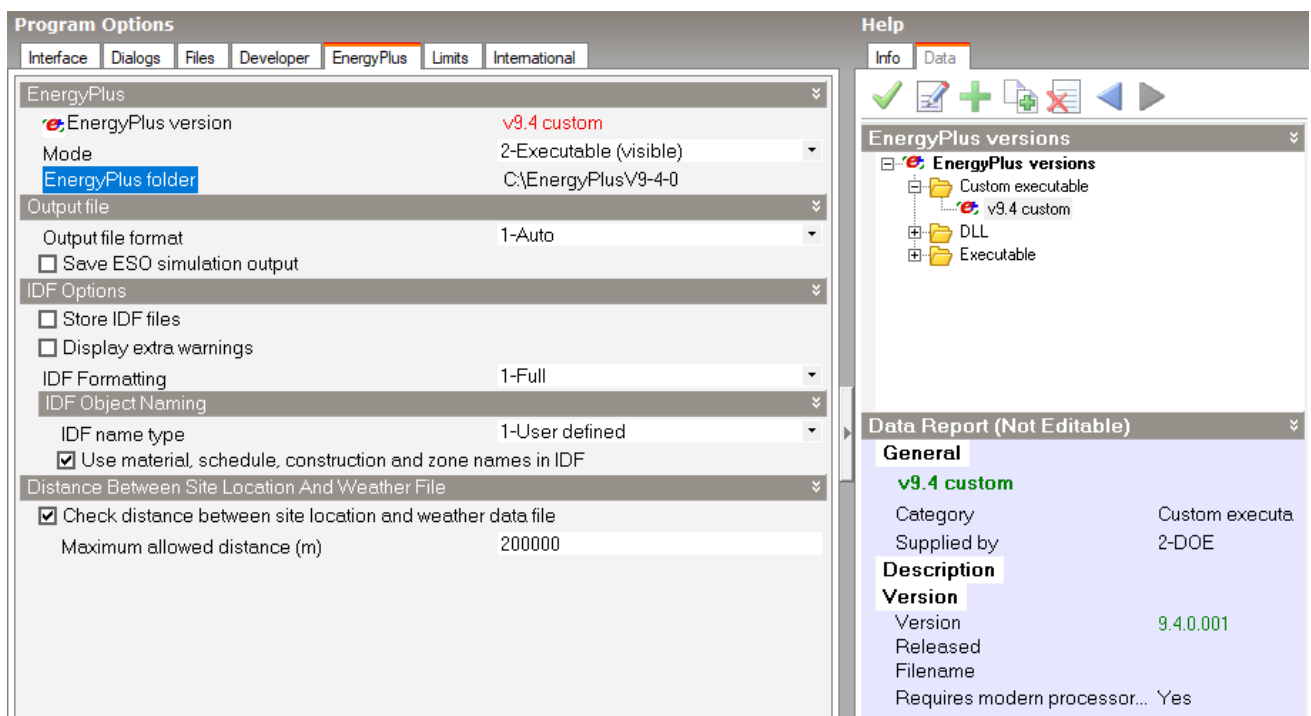
When you select any option other than the default DLL option, the necessary EnergyPlus files are downloaded seamlessly prior to the first simulation following the change.

Note: If you wish to install a different DLL version of EnergyPlus to that which is recommended, you must have administrative rights on the computer.

DesignBuilder are licensed distributors of EnergyPlus. All versions of EnergyPlus supplied by DesignBuilder have been compiled by the DOE.

Running a DOE Version of EnergyPlus

To run a version of EnergyPlus downloaded from the DOE website you should create a new [EnergyPlus version](#) using the **Custom** category and make the appropriate settings. The screenshot below shows the EnergyPlus tab on Program options in this case.



When doing this it is important to consider that the only versions officially supported by DesignBuilder are the ones in the DLL list, and of those, the only one that has been fully tested is the default DLL option. The other options provided should generally work but no guarantees are provided. You may also be able to create and use new custom EnergyPlus versions not listed under the DLL category *if the IDF /ESO file input/output formats are the same as one of the options provided by DesignBuilder*. In this case you should look out for possible problems with simulations not running or results not loading correctly.

Output File

Output file format

This option allows you to define the format of the EnergyPlus Summary "CSV" output file. Select from:

- **1-Auto** - the default option where DesignBuilder selects either the appropriate option based on the machine locale. Normally it will use the tab format for machines using

- **2-Tab** - ensures that EnergyPlus generates a file called eplustbl.tab in which the tab character is used to separate the values, regardless of the machine locale.
- **3-CSV** - ensures that EnergyPlus generates a file called eplustbl.csv in which the comma (,) character is used to separate the values, regardless of the machine locale.

Save ESO simulation output

The ESO file is the primary output from EnergyPlus used by DesignBuilder to access simulation results. ESO output files will be generated by EnergyPlus following Heating design, Cooling design and Simulation calculations. You should normally leave this option checked to ensure that results are generated.

If you choose to **Save ESO simulation output** then you can also select options to **Prompt for filename** and **Auto load ESO** described below.

Note: These three settings only apply when the Simulation Manager is not being used.

The ESO files can be loaded later using the File > [Load results file](#) menu option, provided the model hasn't changed too much. In this case the rules are as follows:

1. The DesignBuilder model must still have same zones and surfaces it had at the time the simulation data was generated
2. If you have modified these files and carried out further simulations in EnergyPlus you should find that the results will read in OK provided you haven't changed zone or surface names in the IDF data.
3. You cannot read in ESO files generated from IDF data which was entered by hand (or by other software) because the zone and surface references will be different.
4. Results must be loaded to the correct screen - you can't for example read Heating design results into the Simulation screen.
5. *Simulation only:* the results to be read in must have been generated using the same [simulation period](#) and number of [timesteps](#) currently set in the [Simulation options](#).

Prompt for filename

Tell DesignBuilder whether you wish to be prompted for a filename after each simulation. If you choose not to be prompted then DesignBuilder will automatically generate output filenames based on the name of the dsb file, the calculation type, the [calculation description](#) and the date. It places the file in the [EnergyPlus folder](#). If you choose to be prompted the file can be stored anywhere on your computer system.

Auto load ESO

Check this option to automatically load the ESO results file after simulations. If you have been warned that there will be a lot of data and the program may crash on reading results you may wish to switch this option off and reload the results later. It is often possible to read in very large ESO files after DesignBuilder has restarted following a simulation and this can be a useful for avoiding a crash due to too much data. You may also wish to switch the option off to avoid waiting for the results to read in when carrying out parametric simulations.

This option only refers to Simulation results not to Heating/Cooling Design results.

IDF Options

Store IDF files

When this option is checked the EnergyPlus IDF input file associated with the simulation is stored with model and can be accessed later even after the file in the EnergyPlus folder has been overwritten. This option is off by default to help conserve memory for other operations.

Display extra warnings

Requesting Display extra warnings causes the statement:

Diagnostics, DisplayExtraWarnings;

to be included in the IDF data. This option is intended for use by experienced EnergyPlus users who understand the significance of the extra warning messages generated.

IDF Formatting

The IDF data can be formatted using the options:

- **1-Full** - all indentation and comments are included in the IDF data and user-defined names are used as object identifiers. You should avoid this option for very large models as the amount of text generated can cause issues with memory.
- **2-Basic** - the default option where indentation and some minimal comments are included in the IDF data. Choose this option if your model is large but you would like to retain user-defined names in the IDF object names.
- **3-Compressed** - the IDF data is fully compressed by removing indentation and comments and most user-defined names are replaced with numeric identifiers. Choose this option if your model is very large and you wish to reduce the amount of IDF text being processed. When the IDF data uses more than about 10MB of disk space DesignBuilder is unable to process the quantity of text involved and if this happens when the **1-Full** or **2-Basic** options are set it will attempt to regenerate the IDF data using the **3-Compressed** option to reduce the memory requirement. If IDF data is being generated twice before your simulations you can speed the process by selecting this option from the start ensuring that IDF data is only generated once.

IDF Object Naming

IDF name type

Normally DesignBuilder uses human readable names in the IDF based on the names used in the model. However you can override this behaviour by changing this option. Select from:

- **1-User-defined** - the default option where human readable user-defined names are used wherever possible in the IDF data.
- **2-Numeric** - where machine-generated numeric IDs are used instead of readable names used in the model.

Note: If you are using an evaluation licence then the software behaves as if the **2-Numeric** setting has been selected, regardless of the selection here.

Use material, construction and zone name names in IDF

You can override the overall way that IDF names are written specifically for materials, constructions and zones by checking/unchecking this checkbox.

Distance Between Site Location And Weather File

Check distance between site location and weather file

The locations defined by the site location (latitude and longitude) should match with that of the Hourly weather file. If you would like DesignBuilder to enforce a minimum distance then check this option.

Maximum allowed distance

If the above **Check distance between site location and weather file** option is checked then enter the maximum distance allowed between the location defined by the [site latitude/longitude](#) and the [latitude/longitude set for the Hourly weather file](#) here (in m or ft). The default setting is 200,000m i.e. 200km.

Using Custom Versions Of EnergyPlus

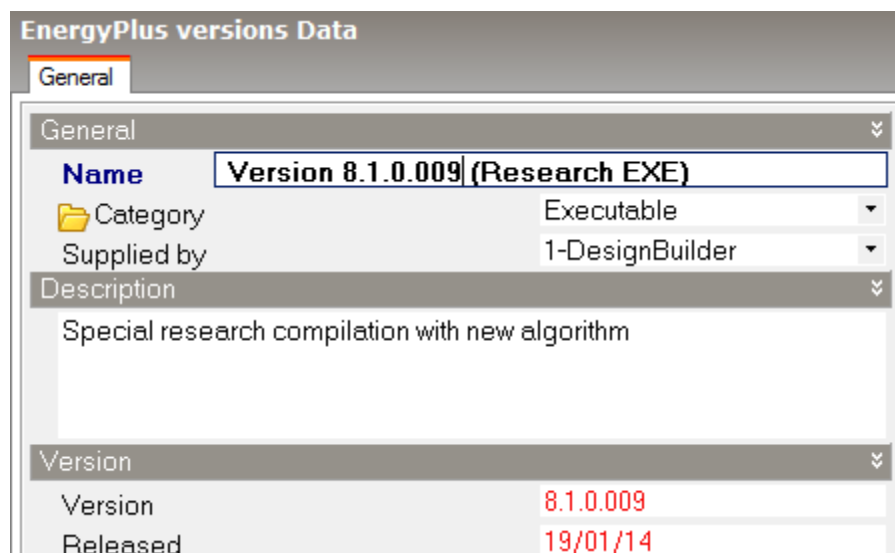
If you have compiled a special version of EnergyPlus you can use it to run simulations from within DesignBuilder by following the instructions below. Note that the custom version must be based on one of the versions supported by DesignBuilder and ideally the default version. The instructions are easiest if the version number of your custom compilation is the exactly the same as that of a version that is supported by DesignBuilder.

When the custom version has the same version number as a supported version

1. Recompile your new version of EnergyPlus into EXE form (not DLL)
2. Select the EXE option from the EnergyPlus program options corresponding to the custom version.
3. This downloads and installs the appropriate standard version of EnergyPlus from the DesignBuilder servers to the DesignBuilder [EnergyPlus folder](#) (if it hasn't already been downloaded).
4. Copy your custom EnergyPlus.exe (and Eplus+.idd if this has changed) to the DesignBuilder [EnergyPlus folder](#) overwriting the standard the files downloaded by DesignBuilder.
5. Test by running simulations from within DesignBuilder to ensure that your custom version is being run.

When the custom version has a new version number

1. Follow instructions 1-3 above selecting the standard version most similar to your custom version. This ensures that all standard files are installed ready to be overwritten with custom files.
2. Create a new version component from the EnergyPlus tab on the Program options dialog and enter the custom version on the dialog. See below.



3. Copy all of the EnergyPlus files which have changed typically EnergyPlus.exe and Energy+.idd to the DesignBuilder [EnergyPlus folder](#).
4. Test by running simulations from within DesignBuilder to ensure that your custom version is being run.

Program Options - Limits

Limits tab of the [Program options](#) dialog

Maximum number of circle segments

DesignBuilder does not work with circular objects directly. Instead, any circular perimeters, openings etc drawn are converted to a stream of line segments to represent the circle. DesignBuilder prevents you from generating circles with more segments than this user-defined setting.

Note: you are advised not to exceed the default setting of 32 to prevent overly complex objects from being created.

Maximum number of arc segments

DesignBuilder does not work with arc objects directly. Instead, any arc segments in perimeters and openings etc are converted to a stream of line segments to represent the arc. DesignBuilder prevents you from generating arcs with more segments than this user-defined setting.

Note: you are advised not to exceed the default setting of 32 to prevent overly complex objects from being created.

Maximum name length

This is the maximum length of name in characters for materials and constructions. If DesignBuilder finds names longer than this it will prevent any calculations requested.

Maximum EnergyPlus ESO file size for importing hourly and sub-hourly results (KB)

Before DesignBuilder starts loading an EnergyPlus result set from an ESO file it checks the size of the file. If the file is larger than the value entered here in KB then you will be prevented from loading hourly and sub-hourly results from the file. In this case you will be asked whether you would like to load the results without hourly and sub-hourly results and if so the load continues but hourly and sub-hourly results will not be available for viewing. If you select "No" then the load is cancelled and no results will be available.

This safeguard has been included to prevent users from attempting to load excessively large datasets that could cause the software memory limits from being exceeded.

Program Options - International

International tab of the [Program options](#) dialog

Language

DesignBuilder currently supports 5 languages:

- **English** - native language of DesignBuilder
- **Italian**
- **French**
- **Spanish** - available only from our Spanish speaking representatives [Aurea Consulting \(Spain\)](#), [Sol Arg \(Mexico\)](#) and [Energy Arg \(Chile\)](#)
- **German**

When you change the language you must restart DesignBuilder for all screens and dialogs to be refreshed.

Units

Units

Two options for units are available:

- **1-Metric (SI)** units and,
- **2-Inch-Pound (IP)** units.

Because the program uses SI units internally, and because the databases which come with the program have mainly been built up using SI units, US users may find some limitations with the user interface when in IP mode. For example the [sliders](#) used on some dialogs are replaced by text boxes.

Currency

Currency

Select the [currency](#) to be used in all cost model data and economic calculations. This setting is used to localise all cost calculation results.

Filters

Region filter

The region filter allows you to see only the data relevant to your particular region. Note that if you have a file loaded then DesignBuilder will use the region associated with the location of the site. The region in this dialog is only used when no file is loaded. Data tables which can be filtered in this way are:

- Constructions
- Glazing
- Materials
- Metabolic rates
- Schedules
- Holidays
- Activity templates
- Construction templates
- Glazing templates
- Lighting templates
- HVAC templates
- DHW templates
- Sectors

Show ASHRAE data

Check this option if you would like to be able to view and access ASHRAE components and templates in tables listed below.

- Constructions
- Glazing
- Materials
- Schedules
- Activity templates
- Construction templates
- Glazing templates
- Lighting templates

Show NCM data

Check this option if you would like to be able to view and access components and templates that are specifically associated with the England and Wales region.

Show Spanish data

Check this option if you would like to be able to view and access components and templates that are specifically associated with the Spain region.

Show other region data

Use this option to display/hide component and template data from other regions.

Show 'General' data

Use this option to display/hide component and template data from the 'General' region. Options are:

1-Never - never show General data

2-Always - always show General data

3-When no region specific data - only show General data when there is no region specific data available.

Show program 'Early' data

Use this option to display/hide special 'Early' and other 'broad-brush' typical system component and template data. Options are:

1-Never - never show 'Early' data

2-Always - always show 'Early' data

3-When no region specific data - only show 'Early' data when there is no region specific data available.

Note: When categories of data are hidden using the program options described above this does not mean they are deleted from the database - the data is still there but it does not appear on lists.

Italian Law

When this option is checked construction layer thickness values are exported to XML in a way that is consistent with Italian Law requirements. This option only affects R-value layers when using the [Export to MasterClima 11300](#) menu option.

Building Models

The process for creating a new building model typically follows a sequence:

1. [Create new site.](#)
2. [Create new building.](#)
3. [Draw and manipulate building blocks to create building geometry.](#)
4. [Partition building into thermal zones.](#)
5. Set [Model Data](#).
6. [Add any 'custom openings'](#) (windows, doors etc) by drawing up the surface level.
7. Size heating and cooling systems.
8. Check design by carrying out summer and winter simulations displaying hourly data.
9. Once model has been checked over short periods, you can use it with confidence for annual simulations.

Create New Site



You can create a new site (dsb file) from the Opening screen. In DesignBuilder, the term 'Site' is synonymous with the file or project because each file can only contain one site. When you request a new site, the **Add New Site** dialog is opened which allows you to select the location and a template:

- [Location](#)
- [Template](#)

See also the [Starting a New Project](#) Tutorial.

Create New Site - Location

Select the Location Template as a source of location and weather data for the site.

The Location defines the geographical location and weather data for all buildings on this site. You will be able to load data from other location templates or override the default data from the Location Model Data tab at Site level.

Create New Site - Template

You can either select 'Blank' to create a clean site, or select a template to base your site on an existing site.

If you have a DesignBuilder file you would like to use as a template for future sites save or copy it to the ['Templates'](#) folder (accessed from the **File | Folders | Template projects** menu command).

Building Geometry

This section describes the various facilities within DesignBuilder to create and edit model geometry.

- "View Controls" on page 108
- "Perimeter Drawing Options" on page 117
- "Snaps, drawing guides, protractor and Grids" on page 121
- "Construction Lines" on page 142
- "General operations" on page 147
- "Importing Geometric Data" on page 160
- "Exporting Geometric Data" on page 181
- "Working with Buildings (Site Level)" on page 182
- "Working at Building Level - Blocks and Solar Collectors" on page 185
- "Working at Block Level" on page 252
- "Working within Zones (Zone Level)" on page 268
- "Working with Surfaces (Surface Level)" on page 270
- "Edit Openings at Building Level" on page 249
- "Undo/Redo" on page 284
- "Rebuild" on page 285
- "Rezone Block" on page 285

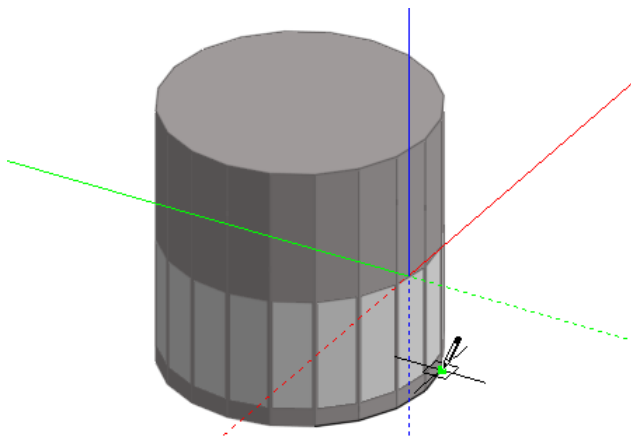
View Controls

A range of view controls are provided to allow you to change the point of view for the model and how the model is to be shown on the screen:

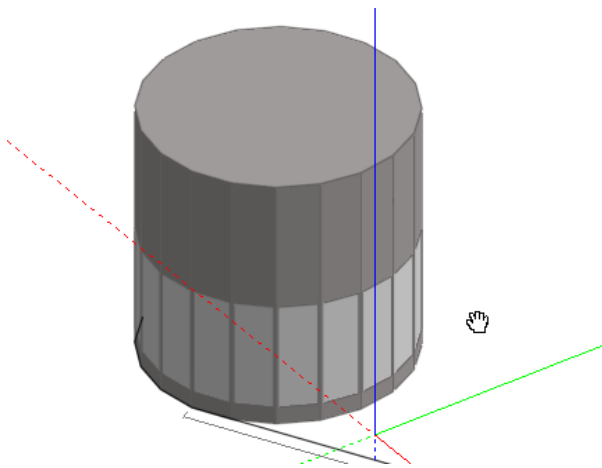
- [View Rotation](#)
- [Orbit](#)
- [Zoom](#)
- "Zoom Dynamic" on page 111
- "Zoom Window" on page 111
- [Fit view](#)
- [Pan view](#)
- "Normal View" on page 112
- "Walk-Through" on page 113
- "Look Around" on page 113
- "Surrounding Block/Zone Display" on page 114
- "Ghost Blocks" on page 116

Interaction Of View And Edit Commands

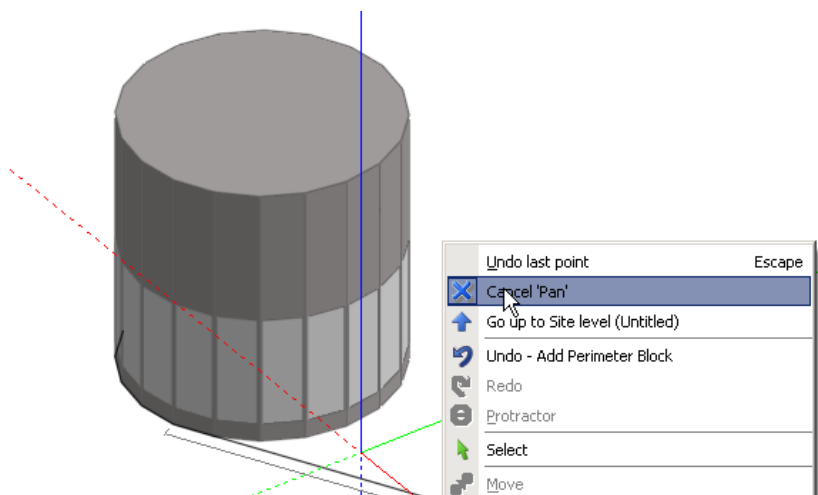
When on the Edit screen, the View tools can be activated at any time during operation of the drawing tools. Control is returned to the active drawing tool after completing the view process. For example to draw a block perimeter adjacent to an existing cylindrical block:



To continue drawing the perimeter around the other side of the cylindrical block, click on the [Orbit](#) tool and orbit the model to reveal the other side and then use the [Pan](#) tool to centre the model:



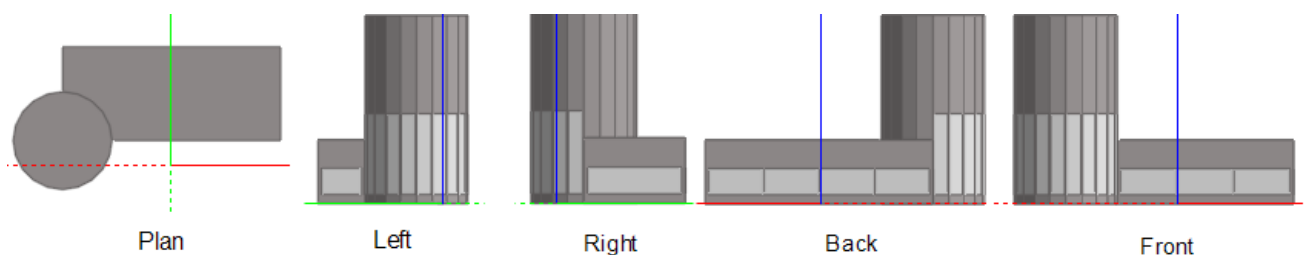
You can then resume the drawing process either by pressing the <Esc> key or selecting 'Cancel Pan' from the right-click menu:



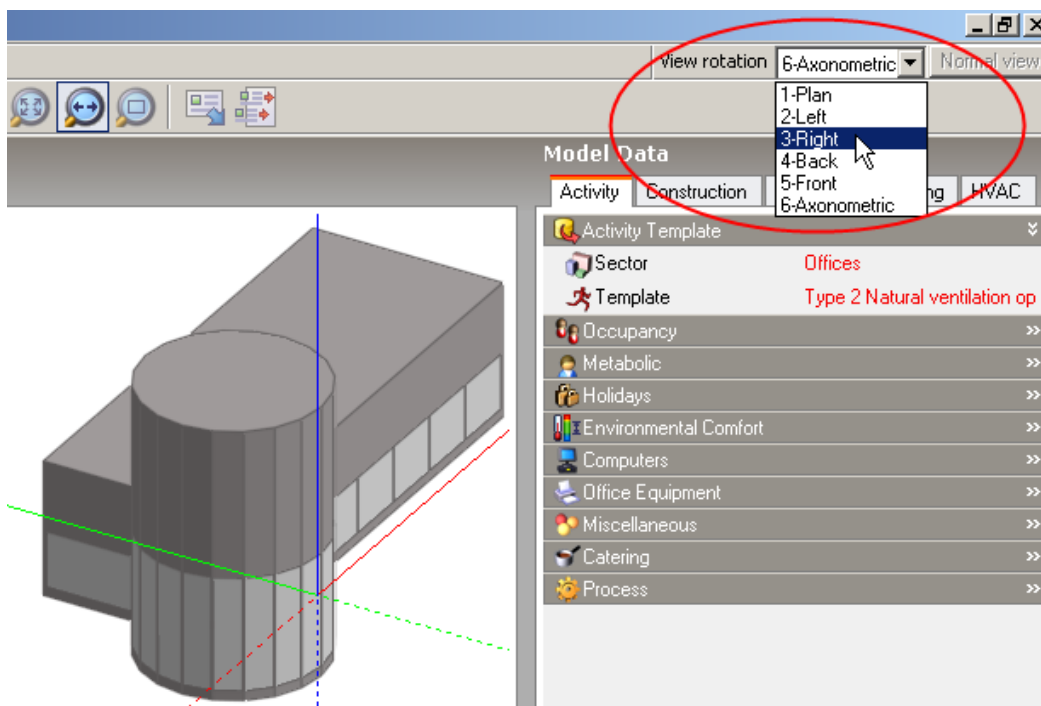
View Rotation

Accessible from the [Edit](#), [Visualisation](#), [Daylighting](#) and [CFD](#) screens at all levels

DesignBuilder provides several pre-defined standard view rotations:



You can select one of these standard views from the view rotation drop list:



The default 'Axonometric' setting is designed to provide a good general 3-D view of the various components at each model level.

Alternative ways to access the view rotation commands are:

- For **Axonometric** view, press the A keyboard key.
- For **Plan** view, press the P keyboard key.
- For **Right** view, press the R keyboard key.
- For **Left** view, press the L keyboard key.
- For **Back** view, press the B keyboard key.
- For **Front** view, press the F keyboard key.

Orbit

Accessible from the Edit, [Visualisation](#), [Daylighting](#) and [CFD](#) screens at all levels



The **Orbit** tool is used to rotate the view around the model. To orbit the model click on the **Orbit** tool and then press the left mouse button and keep the button pressed as you move the mouse cursor, the view will orbit around the model.

Alternative ways to access the **Orbit** command are:

- Hold <Shift> key and move mouse while pressing the right mouse button
- Move mouse while pressing the centre mouse button/wheel (if you have one).
- Hold the <Ctrl> + <Shift> + O keys to start the **Orbit** command then hold left mouse button and move mouse.

Tip: It can be helpful to understand how the orbit command works to ensure smooth operation. When the orbiting command is started, DesignBuilder needs to calculate a centre point around which to rotate the view. To do this a ray is fired into the centre of the screen, which should ideally intersect with a part of the model that you want to see. If this is not the case and the ray to the centre of the screen instead finds a far distant point, then the centre of orbit will be that far distant point. The effect of this when you start to move

the mouse to change views can be rather disconcerting as the model will tend not to orbit as expected. **To avoid this sort of problem it is important to ensure that the object being viewed is in centre of the screen and that any holes between the building blocks are not in the centre.** If you do encounter any strange effects with the Orbit command, an easy solution is to select a new view from the [View Rotation](#) drop list in the top right of the screen (e.g. the **Axonometric** option) or to use the [Fit View](#) command.

Note: when using the **Orbit** command after the [Zoom Window](#) command you may see a very rapid and uncontrolled movement. This is usually because of the way that the previous Zoom Window operation was done. That operation works best if the 2 corners selected to create the zoom window area are both on the part of the model you wish to view. If either of the corners selected are outside the model (e.g. on the white background or in distant parts of the model) then the operation can have these unintended effects. This happens because the centre point of the **Orbit** operation is in the far distance.

Zoom Dynamic

Accessible from the Edit, [Visualisation](#), [Daylighting](#) and [CFD](#) screens at all levels



To zoom in and out from the current model point of view, click on the **Zoom** tool, then press the left mouse button and keep the button pressed as you move the mouse cursor, the view will zoom in as you move the mouse cursor down and out as you move it up.

Tip: To centre the view on the current mouse position and start zooming, press the Z key first.

Alternative ways to access the **Zoom dynamic command** are:

- Hold <Ctrl> key and move mouse while pressing the right mouse button.
- Hold <Ctrl> key and move mouse while pressing the centre mouse button (if you have one).
- Hold the <Ctrl> + <Shift> + Z keys to start the **Zoom dynamic** command then hold left mouse button and move mouse.
- As mentioned above press Z to centre the view on the current mouse position and start zooming by pressing left mouse button and moving mouse.

Zoom Window

Accessible from the Edit, [Visualisation](#), [Daylighting](#) and [CFD](#) screens at all levels



The **Zoom Window** tool is used to zoom in on a selected area of your model. To create a window on an area of the model, click on the **Zoom Window** toolbar icon and move the mouse cursor to a corner of the window you want to create, then press the left mouse button and keep the mouse button pressed as you drag the window, when you release the mouse button, the view will zoom to the extents of the defined window.

Alternative ways to access the **Zoom Window** command are:

- <Ctrl> + <Shift> + W keyboard keys.

Note: the **Zoom Window** operation works best if the 2 corners selected to create the zoom window area are both on the part of the model you wish to view. If either of the corners selected are outside the model (e.g. on the white background or in distant parts of the model) then the operation can have unintended effects when applying a subsequent [Orbit](#) command where the centre point of the Orbit operation is in the far distance.

Fit View

Accessible from the Edit, [Visualisation](#), [Daylighting](#) and [CFD](#) screens at all levels



The **Fit view** tool is used to zoom your view to a distance which will enable the whole model to be made visible and centred in the 'Edit' window.

Alternative ways to access the **Fit View** command are:

- Press the w keyboard key.
- <Ctrl> + <Shift> + F keyboard keys.

Pan View

Accessible from the Edit, [Visualisation](#), [Daylighting](#) and [CFD](#) screens at all levels



The **Pan** tool is used to move the model point of view vertically and horizontally across the plane of the screen. To pan the view, click on the **Pan** tool, then press the left mouse button and keep the button pressed as you move the mouse cursor, the point of view will pan horizontally as you move the mouse cursor left and right and vertically as you move it up and down.

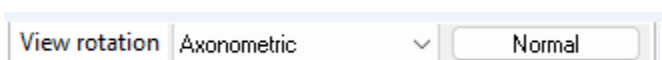
Alternative ways to access the **Pan** command are:

- Hold <Shift> + <Ctrl> keys and move mouse while pressing the right mouse button
- Hold <Shift> key and move mouse while pressing the centre mouse button (if you have one).
- Hold the <Ctrl> + <Shift> + N keys to start the **Pan** command then hold left mouse button and move mouse.

Normal View

Accessible from the Edit screen at Surface level

At the [surface level](#), The **Normal** button is displayed at the rightmost end of the toolbar when at surface level and can be used to rotate the current view to be normal to the surface and to automatically zoom in so that the extents of the surface fill the view.



The [Default surface normal view](#) Program option allows you to control whether the normal view is from the outside or the inside.

Walk-Through

Accessible from the [Visualisation](#) and [CFD](#) screens at Building level



When you are at building level, the Walk-through view command allows you to move through the model using the mouse or keyboard to control the direction of movement and of the camera. It provides a useful way to obtain specific views of the model more quickly than is possible using the normal [Orbit](#), [Zoom](#), [Pan](#) etc tools. For example to look more closely at a particular area of a CFD results slice or to gain a specific interior view when analysing shading on the Visualisation screen.

Tip: One key feature of the Walk-through command is that the z (height) position of the camera is maintained constant. This helps to give the impression of walking through the model. To change floors, or to otherwise change the z-position, it can be convenient to use the [Pan](#) command. You can gain quick access to the Pan command without stopping the Walk-through by holding the <Shift> key and the middle mouse button/wheel down while moving the mouse.

When Using The Mouse

While holding the mouse left button down, move the mouse around the screen in the direction you wish to move. Move the mouse cursor up the screen to move forwards and down the screen to move backwards. Likewise, to turn left move the mouse to the left of the screen, and to turn right move the mouse to the right of the screen. You can combine forward/backward movement and left/right direction changes by moving the mouse diagonally towards the appropriate corner.

Other tools often used together with the Walk-through tool are [Look around](#) or [Orbit](#) (to change the camera direction), and [Pan](#) to move vertically up and down, which is not possible with the Walk-through command.

Tip: You can move more quickly by holding the <Shift> key down while using the above mouse commands.

When Using Keys

The Walk-through tool is most responsive when using the mouse, but it is also possible to access it using the keyboard as follows.

Use the <Up> and <Down> keys to move forwards or backwards in the current direction.

Use the <Left> and <Right> keys to change direction.

Tip: The best way to learn how to use the Walk-through tool is to try it for yourself!

Look Around

Accessible from the [Visualisation](#), [Daylighting](#) and [CFD](#) screens at Building level



When you are at building level, the Look Around view command allows you to change the direction of camera view using the mouse. It allows you to obtain specific views of the model more quickly than is possible using the normal [Orbit](#), [Zoom](#), [Pan](#) etc tools. For example to look more closely at a particular area of a CFD results slice or to gain a specific interior view when analysing shading on the Visualisation screen.

When Using The Mouse

While holding the mouse left button down, move the mouse around the screen in the direction you wish to view. Move the mouse cursor up the screen to look up and down the screen to look down. Likewise, to look left move the mouse to the left of the screen, and to look right move the mouse to the right of the screen. You can combine forward/backward/left/right view changes by moving the mouse diagonally towards the appropriate corner.

Other tools often used together with the Look Around tool are "Walk-Through" on page 113 and [Orbit](#) (to change the camera direction), and [Pan](#) to move vertically up and down, which is not possible with the Walk-through command.

Surrounding Block/Zone Display

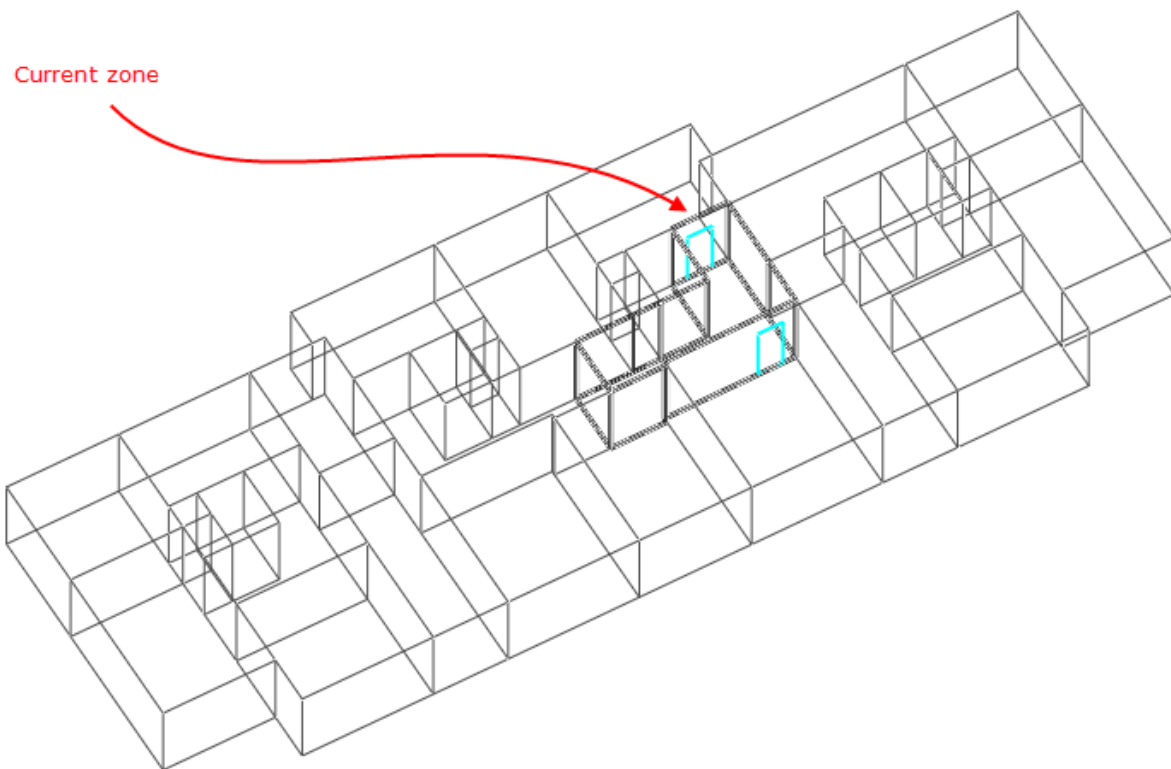
Accessible from the Edit Screen at Block and Zone Levels



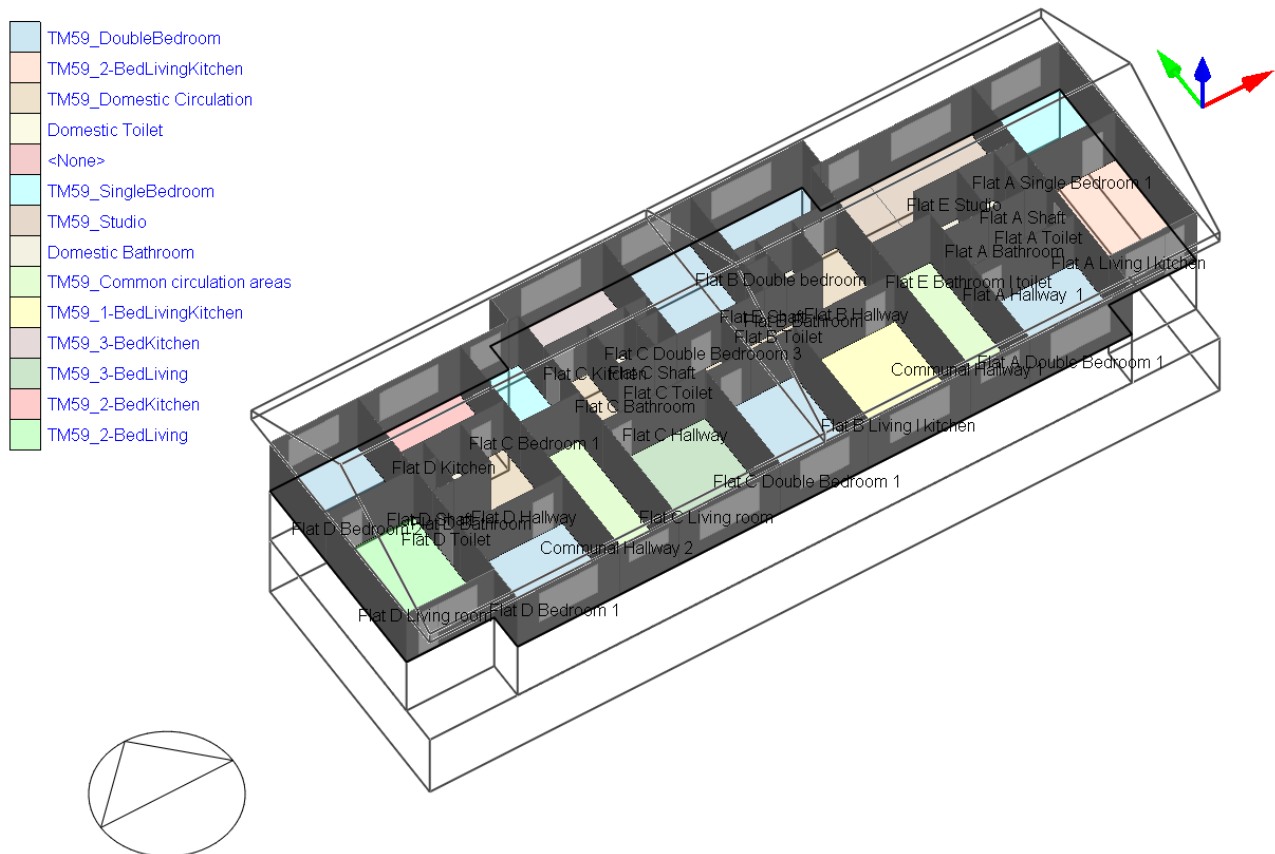
It is sometimes useful to be able to gain a quick visual indication of the location of the current block or zone in the context of the rest of the building or block. The **Surrounding block/zone display** toolbar and menu command provides a convenient way to control:

- a. the visibility of other blocks in the current building (when working at block level) and,
- b. the visibility of other zones in the current block (when working at zone level).

Note: The same functionality can also be accessed from the [Surrounding block display](#) and the [Surrounding block zone display](#) options on the Display tab of the Model options dialog, however, it is usually easier to use the toolbar icon. Note that the Model options dialog provides additional functionality in that it also allows a zone to be seen in the context of all the other zones in the building if required. This less commonly required option is not available from the toolbar which only allows zones to be seen in the context of the rest of the current block.



Example use of the **Surrounding block/zone display** command at zone level



Example use of the **Surrounding block/zone display** command at block level

Ghost Blocks

Accessible from the Edit Screen at Building Level

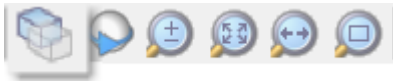
There may be times when it is helpful to be able to ghost one or more blocks in the model so that they become semi-transparent, allowing blocks behind them to be accessed and selected for editing. The **Ghost selected blocks** and **Show all blocks** toolbar icons described here provide the most convenient way to control the ghosting of blocks in the model.

The Ghost blocks mechanism applies to both building blocks and component blocks.

An alternative (slower) method of controlling building block ghosting is to use the [Active](#) checkbox on the Blocks tab of the Model options dialog.

Ghost Selected Blocks

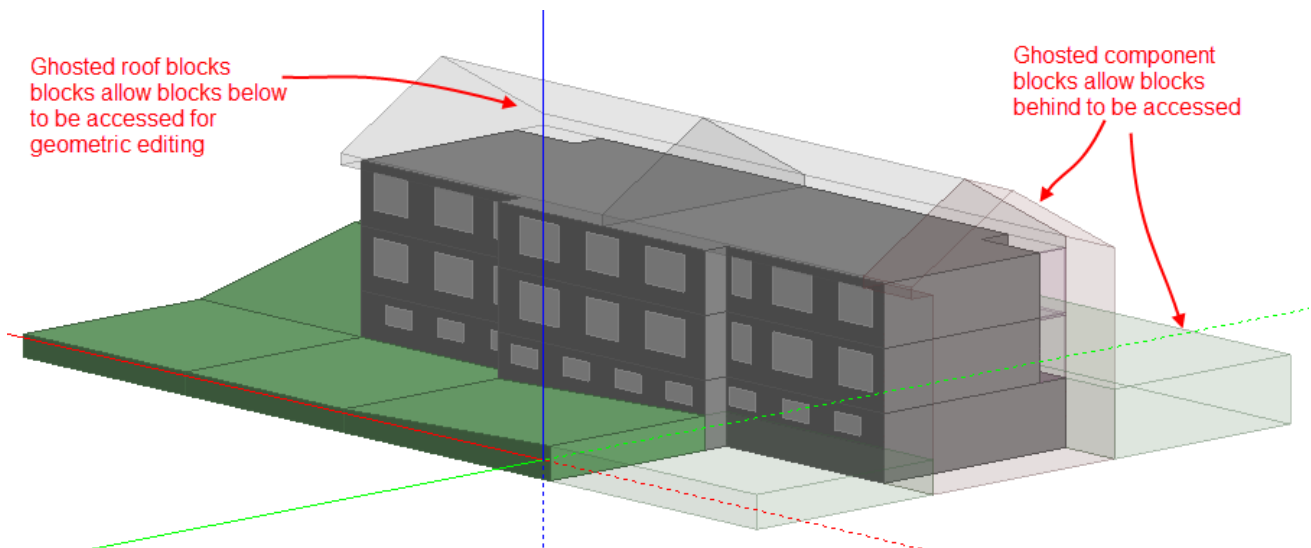
Accessed from Edit Screen, Building Level



This tool is available on the Edit screen at building level when one or more blocks are currently selected.

To ghost one or more blocks:

- Go to building level and select the building and/or component blocks to be ghosted.
- Click on the **Ghost selected blocks** toolbar icon.



Ghost selected blocks example

Note: Once in a ghosted state, blocks cannot be selected. There are 2 ways to "un-ghost" them:

- Navigate to building level and use the **Show all blocks** command (below).
- Navigate to the block to be un-ghosted, open the Model Options dialog and on the Block tab, check/uncheck the [Active](#) checkbox. This allows the ghosting of blocks to be controlled independently and can be useful for un-ghosting just one block at a time.

Show All Blocks

Accessed from Edit Screen, Building Level

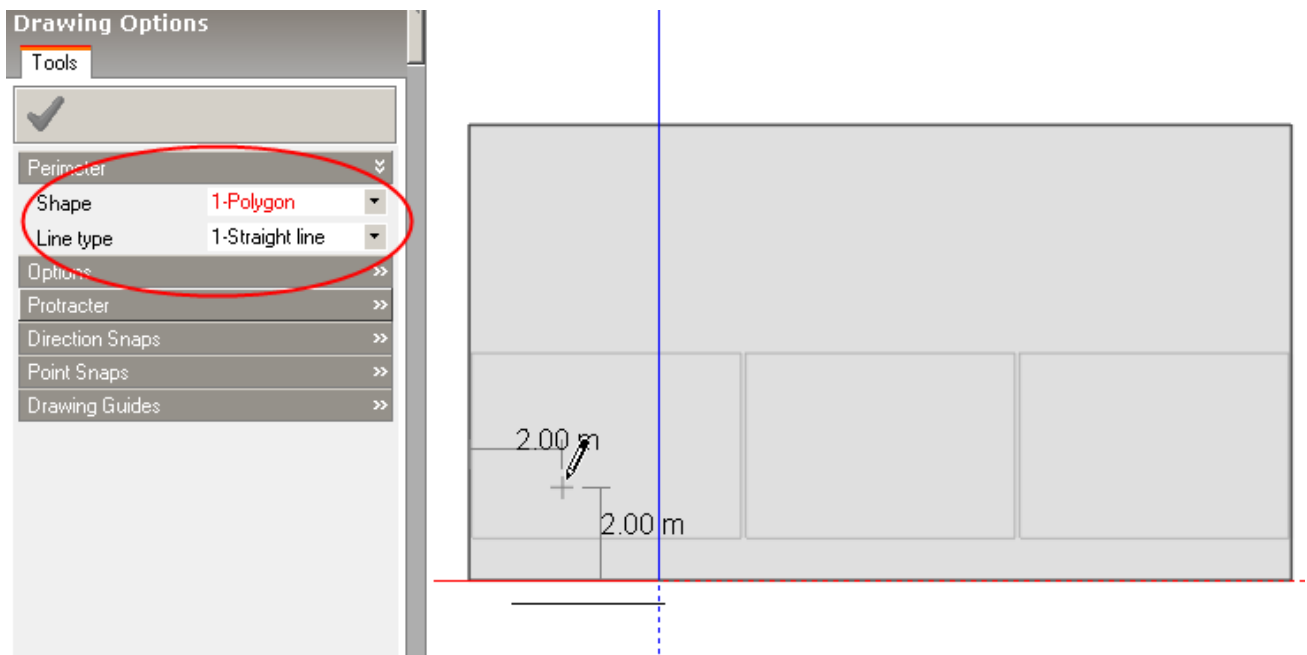


This tool is available on the Edit screen at building level when one or more blocks are currently ghosted. To display or "unghost" all blocks in the building:

- Go to building level.
- Click on the **Show all blocks** toolbar icon

Perimeter Drawing Options

Blocks, courtyards and surface openings are all created using perimeters. When any of these element drawing tools are activated, perimeter options can be accessed from the 'Drawing Options' data panel, for example when the [Draw window](#) tool is active at the surface level:



See also the [Blocks and dimensioning protocols](#), [Adding blocks and using the measure tool](#), [Add surface, increment snaps](#), [Drawing arcs, circles and domes](#) Tutorials

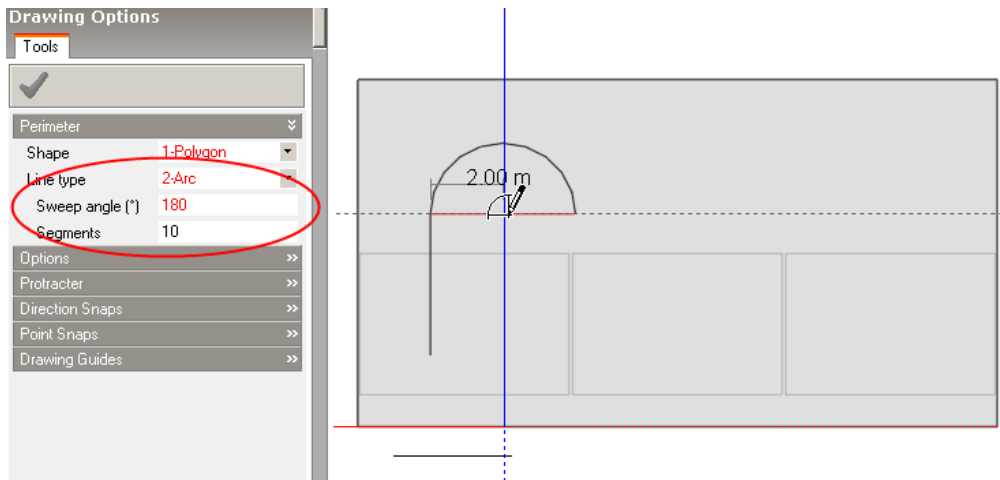
Shape

Three perimeter shapes are provided to enable specific perimeter topologies for blocks and surface openings to be drawn efficiently:

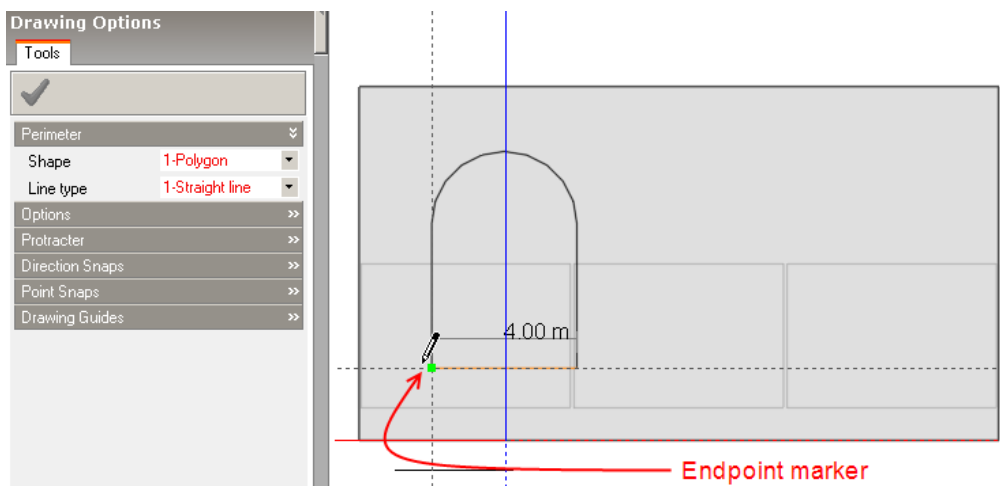
- **1-Polygon**
- **2-Rectangle**
- **3-Circle**

Polygon

The **1-Polygon** is generally the most useful shape and can be used to draw block and opening perimeters of any topology. The **1-Polygon** shape also allows you to change the line type from a straight line to an arc. To draw a perimeter using the **1-Polygon** shape, start drawing the first line by clicking anywhere in the active drawing plane to place the starting point, then move the mouse cursor and click to place the line end-point, you can use the [snaps and the protractor](#) to help with alignment. Continue placing perimeter lines in the same way. If at any time you want to cancel the last line that you placed, you can press the ESC key or select the **Undo last point** option from the right-click menu. If you want to incorporate an arc within the perimeter, you can change the [line type](#) to **2-Arc** and then place the centre-point of the arc:

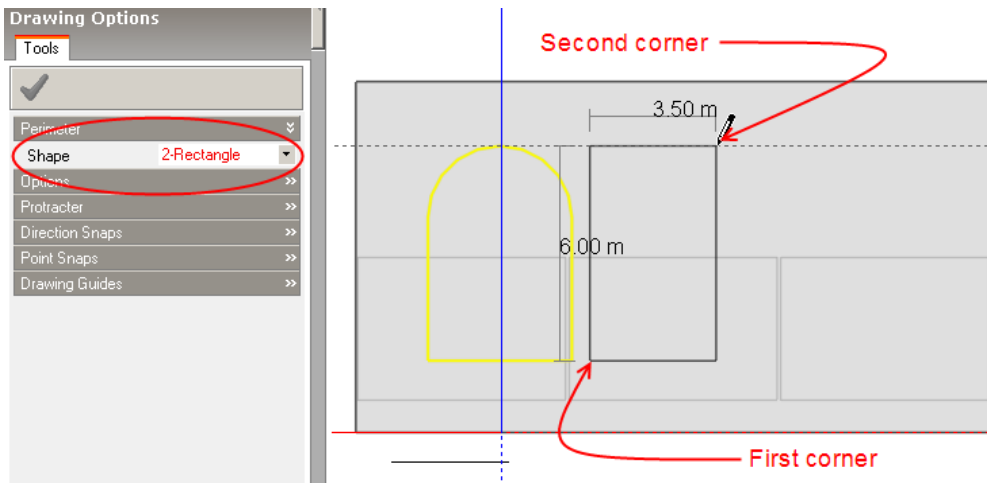


To complete the perimeter, move the mouse cursor to the beginning of the first perimeter line, which activates the [end-point](#) snap and click on it to connect the lines:



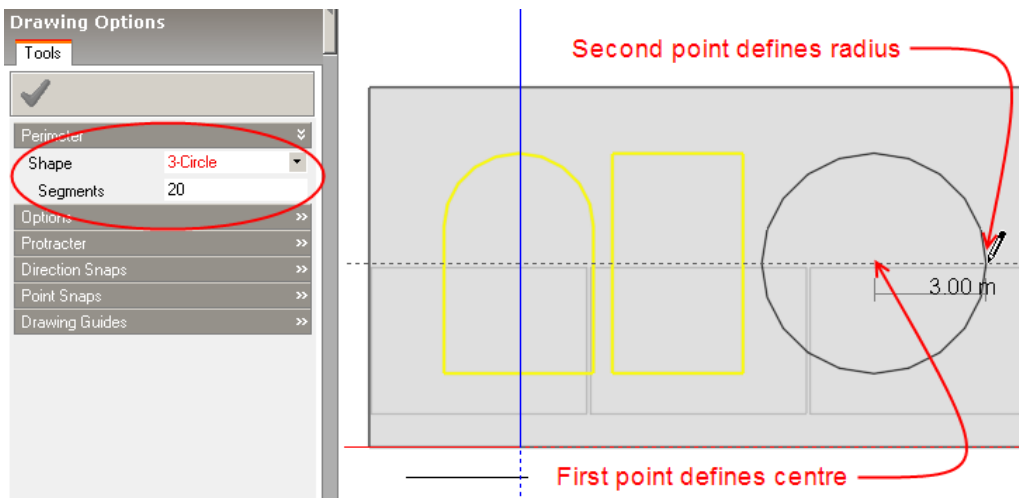
Rectangle

The **2-Rectangle** shape allows you to draw perimeters by placing diagonally opposite corners of a rectangle:



Circle

The **3-Circle** shape is used to draw circular perimeters by placing the circle centre and then setting the radius by placing a second point:



The associated **Segments** option allows you to control the accuracy of the circle. In DesignBuilder geometry, circles are actually composed of polygons and the accuracy of the representation can be controlled by defining the number of segments (or lines) used in generating the polygon. Generally speaking, the greater the number of segments used will result in more memory usage and processing time and so, particularly in the case of blocks, it's advisable to use the default number of segments where possible.

More information on using the Circle shape can be found in the [Drawing arcs, circles and domes](#) tutorial.

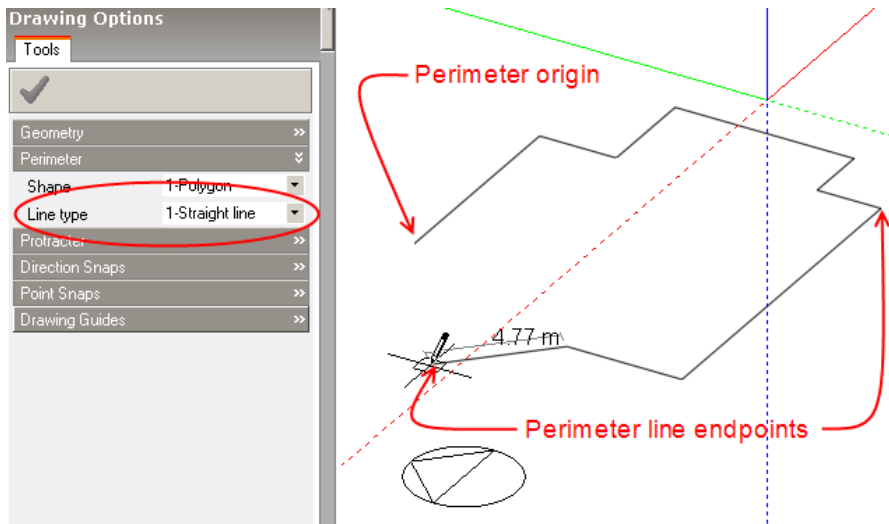
Line Type

When the perimeter [shape](#) is set to **1-Polygon**, you can set the perimeter **Line type** to one of:

- **1-Straight line** or
- **2-Arc**

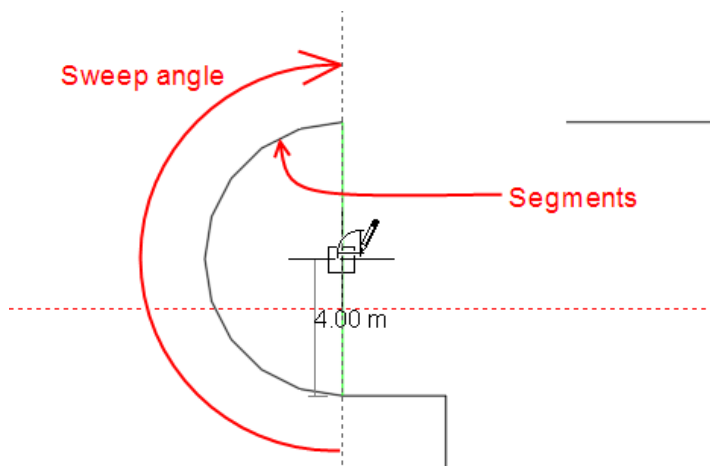
Straight Line

When the line type is set to **1-Straight line**, a perimeter is drawn by placing the origin and then a series of line end-points:



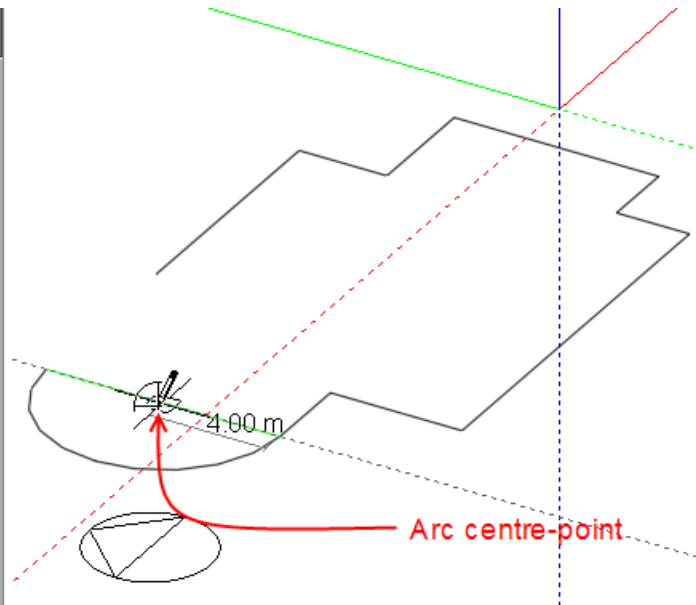
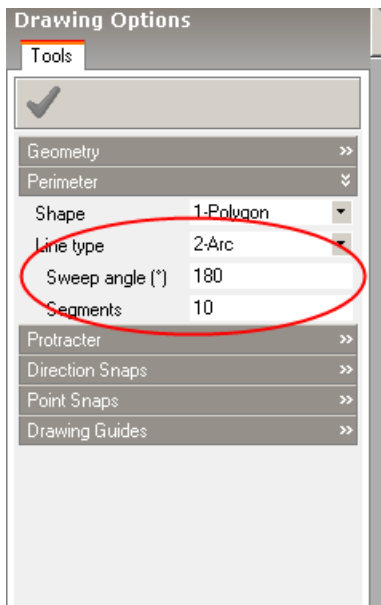
Arc

To incorporate an arc within a perimeter, set the line type to **2-Arc**. An arc is defined using the sweep angle and number of segments:



A negative sweep angle can be used to create an arc in an anti-clockwise direction. In DesignBuilder geometry, arcs are created using a number of segments or straight lines and the accuracy of the representation can be controlled by defining the number of segments. Generally speaking, the greater the number of segments used will result in more memory usage and processing time and so, particularly in the case of blocks, it's advisable to use the default number of segments where possible.

To draw the arc, place the arc centre:



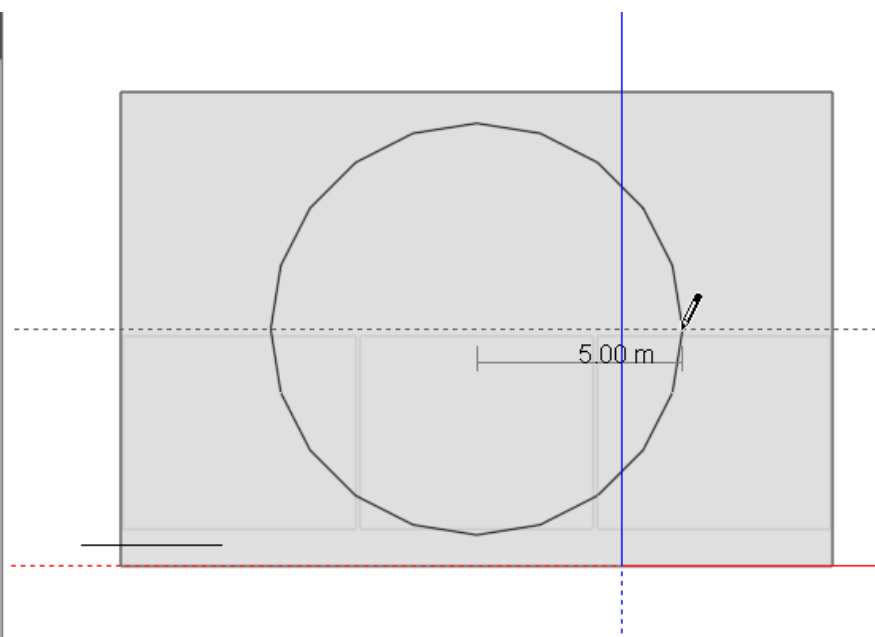
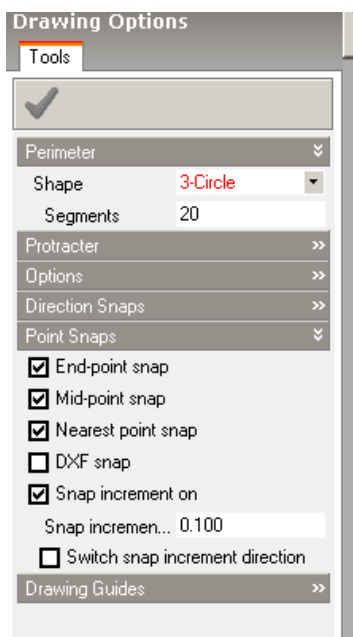
More information on using the Arc line type can be found in the [Drawing arcs, circles and domes](#) tutorial.

Snaps, Drawing Guides, Protractor And Grids

A number of drawing aids are provided to enable block perimeter lines, partitions, etc. to be drawn precisely:

- [Direction Snaps](#)
- [Point Snaps](#)
- [Drawing Guides](#)
- "Protractor tool" on page 139
- "Drawing Grid" on page 141

These tools are made available in the Drawing Options panel in the bottom left of the screen when the drawing tools are activated.:



Direction Snaps

When drawing perimeter lines, block partitions or surface openings direction snaps can be used to draw lines:

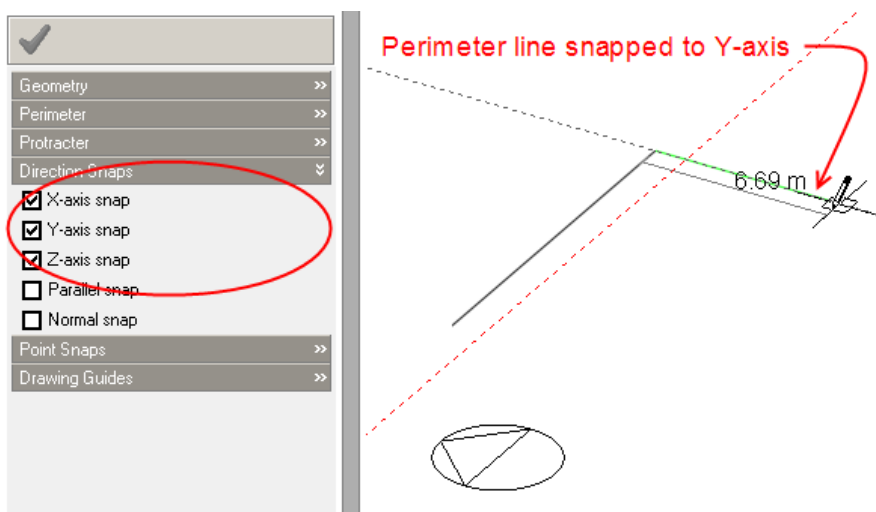
- [Parallel with the active drawing plane axes \(Axis snap\)](#),
- [Parallel with existing elements \(Parallel snap\)](#) or
- [Perpendicular to existing elements \(Normal snap\)](#).

See also the [Adding blocks and using the measure tool](#) Tutorial

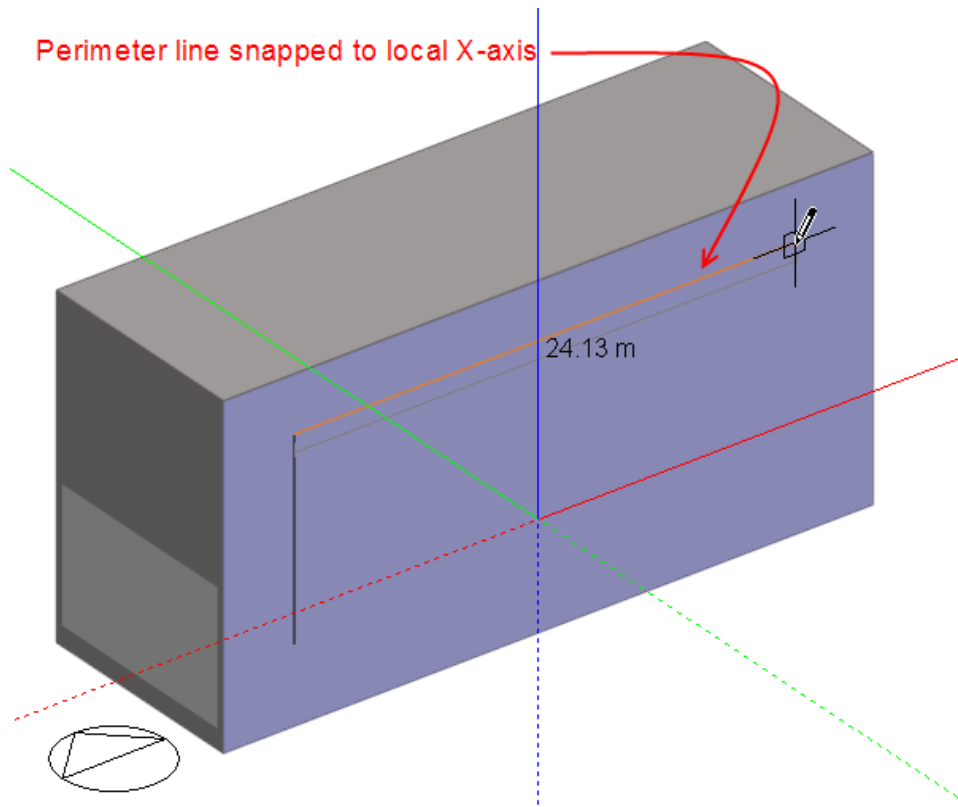
Axis Snap

The axis snap check boxes are displayed under 'Direction snaps' in the 'Drawing Options' data panel when the various drawing commands are active. The axis snap is used to draw lines parallel with the major axes of the active drawing plane.

As you move the cursor around to draw a line, the line will change colour and 'stick' slightly as it becomes parallel with one of the major axes:

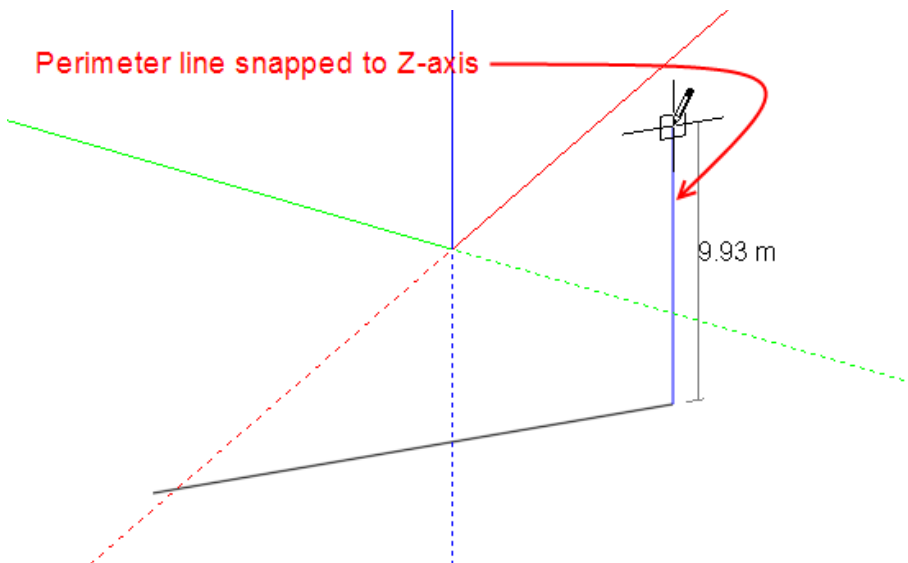


You can use axis snap to draw block perimeters on planes of any orientation. If the plane is non-horizontal, the axis snap applies to the local coordinate system of the plane as if it were rotated to the horizontal:

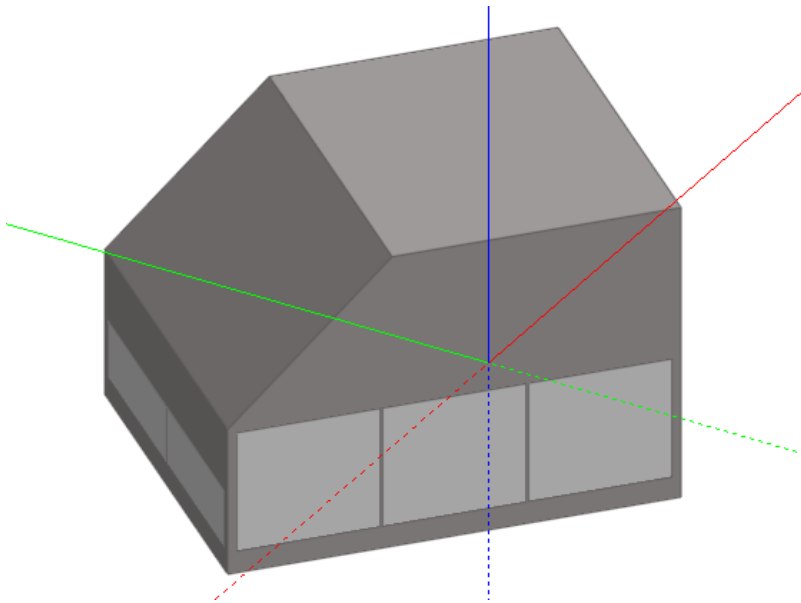


If the axis snap is operating on a horizontal plane, the perimeter line will change colour to red if the line is snapped to the X-axis or green if it's snapped to the Y-axis. However if the active drawing plane is not horizontal, the line will change colour to orange for the X-axis and purple for the Y-axis.

At the building level, when the active drawing plane is horizontal, you can also snap block perimeters and [construction lines](#) to the Z-axis:



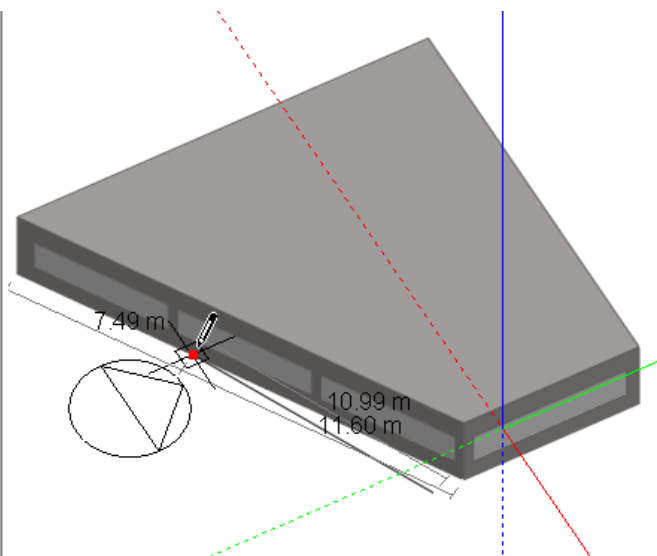
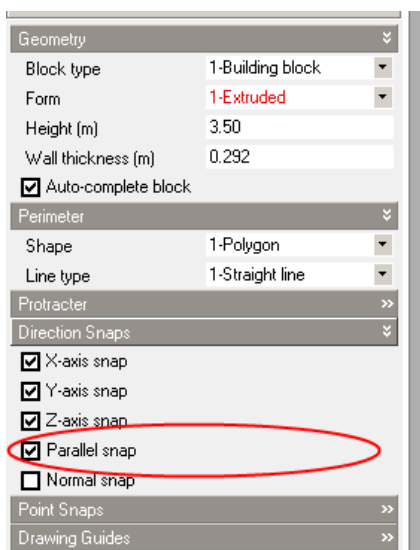
which allows you to draw horizontally extruded blocks, this process is explained under [Adding Blocks](#)



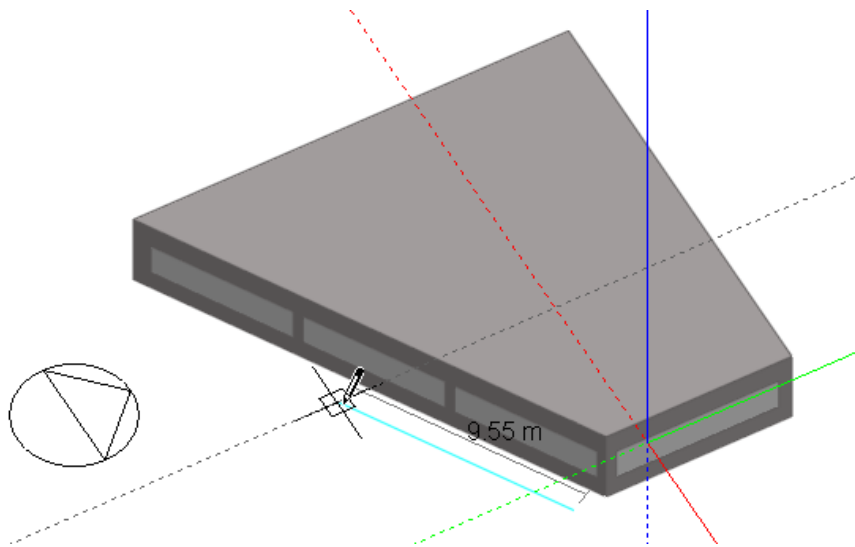
See also the [Blocks and dimensioning protocols](#) tutorial.

Parallel Snap

The parallel snap is used to draw lines parallel with the edges of existing objects. The parallel snap check box is displayed under 'Direction snaps' in the 'Drawing Options' data panel when the various drawing commands are active. To draw a block perimeter line to be parallel with an existing block edge, switch the parallel snap on and move the mouse cursor to touch the edge with which the current perimeter line is to be parallel:



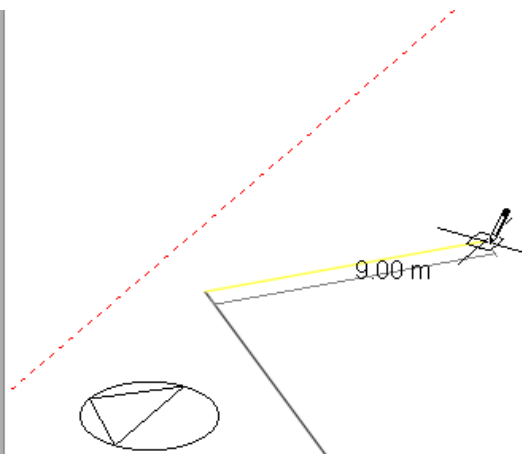
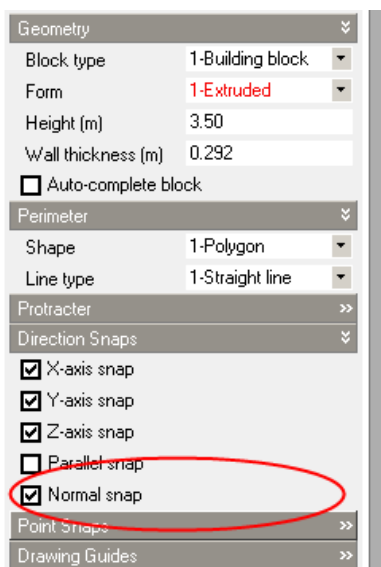
This will enable the active perimeter line to infer a direction from the selected edge and you can lock this inference by pressing the SHIFT key to prevent a parallel inference being picked up from an unwanted edge. So press the SHIFT key and move the mouse cursor to a position approximately parallel with the edge, the cursor will 'stick' when the parallel snap operates and the perimeter line will change colour to cyan:



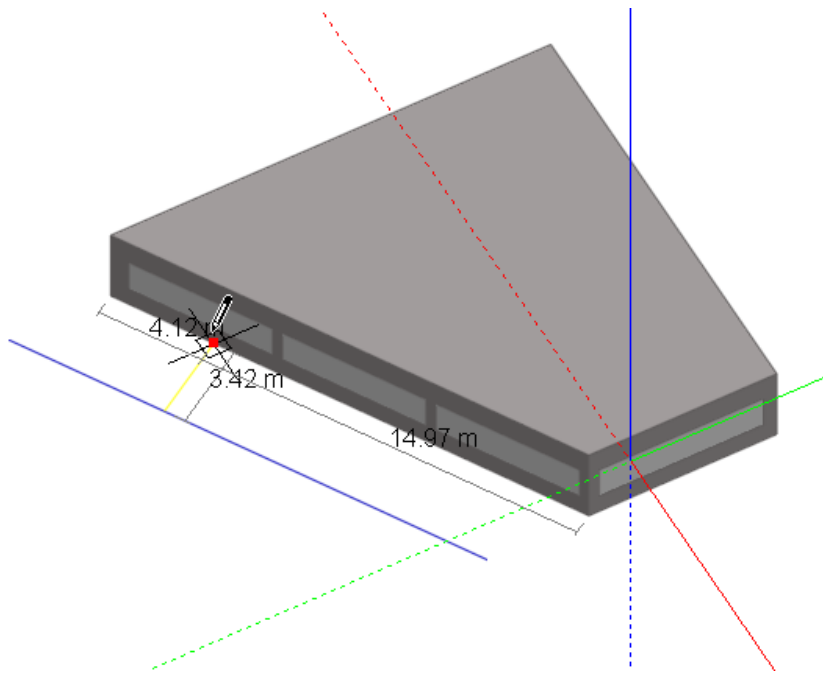
When the parallel snap is in operation, intersections are automatically calculated with [drawing guides](#) to allow the two to be combined, so you can snap to the intersection of the two.

Normal (Perpendicular) Snap

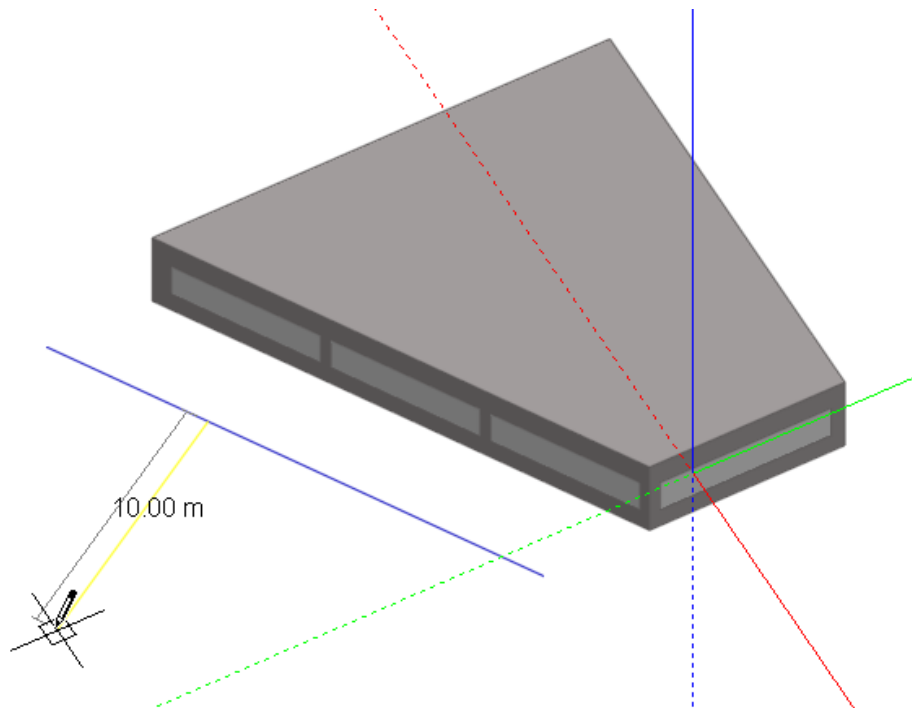
The normal snap is used to draw lines perpendicular to the edges of existing objects. The normal snap check box is displayed under 'Direction snaps' in the 'Drawing Options' data panel when the various drawing commands are active. When the normal snap is switched on and you draw a line string, each line will automatically snap to be perpendicular with its predecessor if it exists:



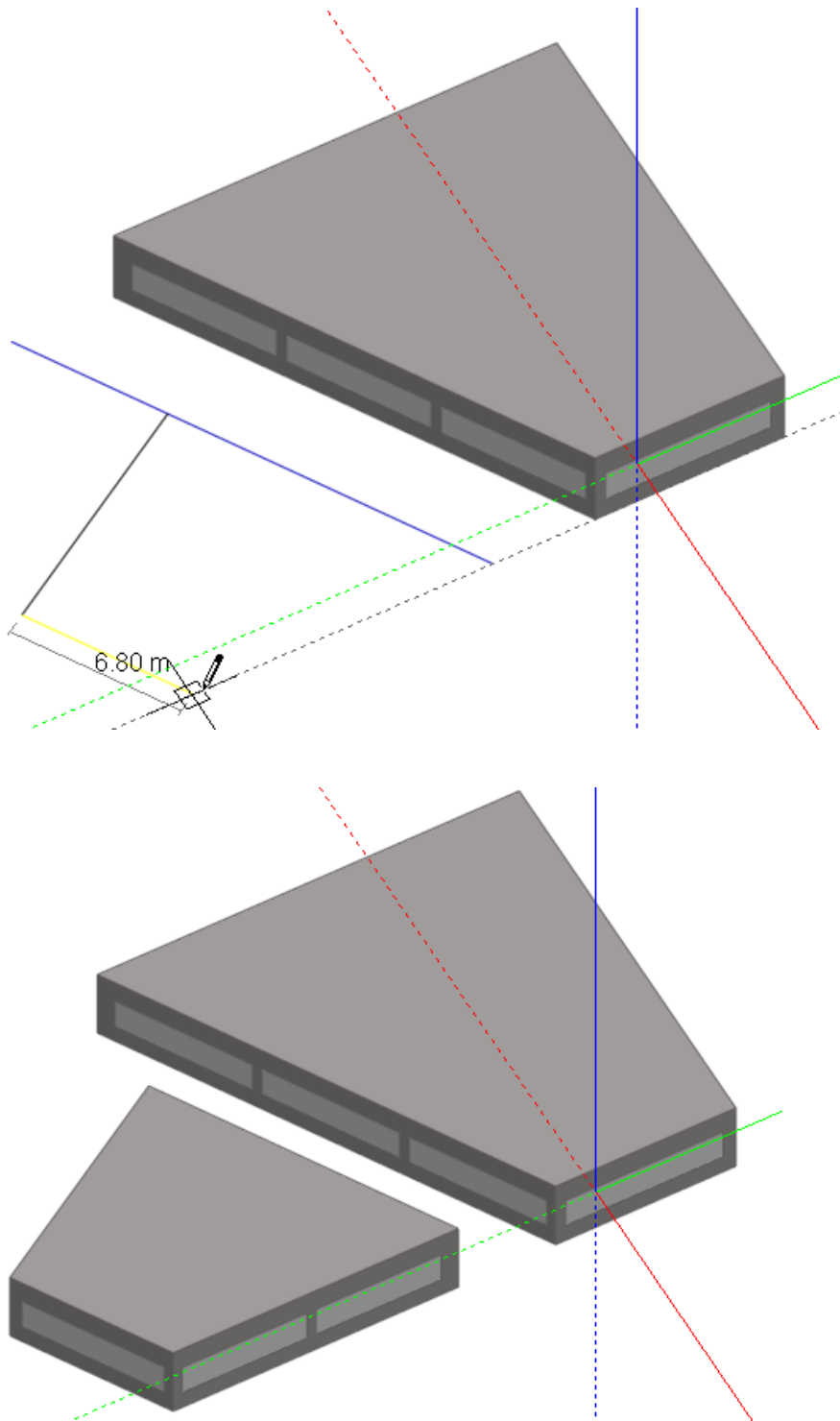
To draw a block perimeter line to be perpendicular with the edge of an existing block, move the mouse cursor to touch the edge and then move it to be approximately perpendicular to it, the cursor will automatically snap to a line which is exactly perpendicular.



You can lock the perpendicular snap by pressing the SHIFT key which will prevent the cursor from picking up an unwanted direction inference. In this case, a [construction line](#) has been placed [parallel](#) with the block edge to help in positioning the new block. The perimeter line locks to a perpendicular from the construction line:



When the normal snap is in operation, intersections are automatically calculated with [drawing guides](#), allowing you to snap to the intersection:

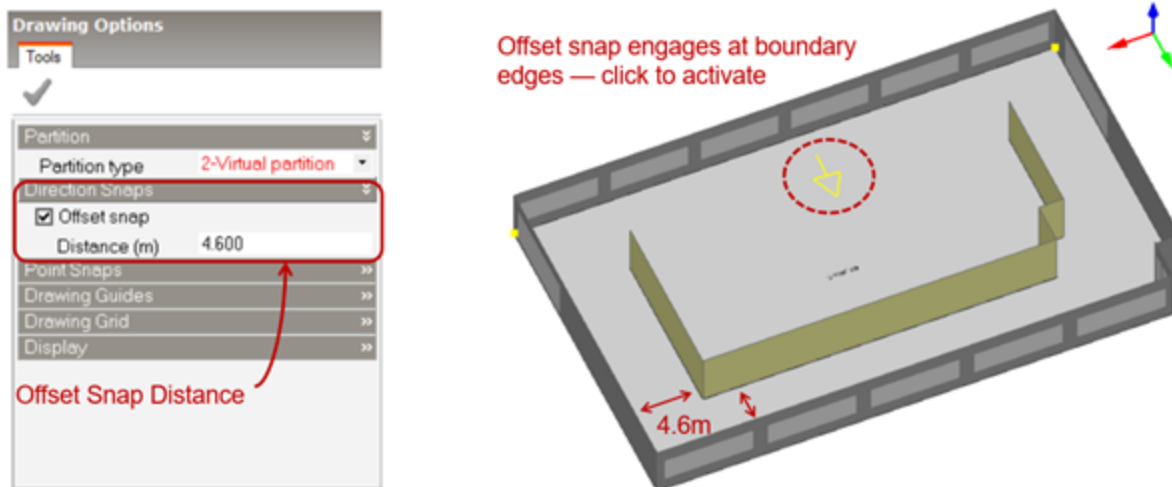


Offset Snap

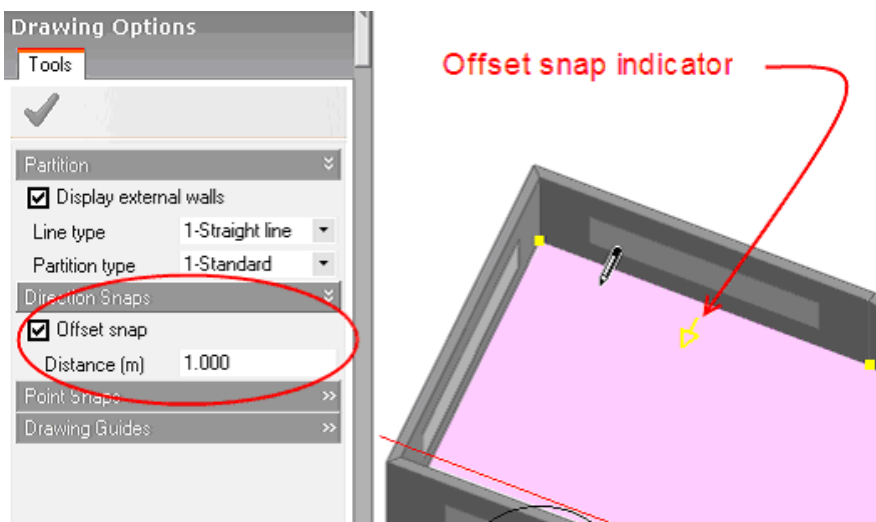
The offset snap is used to draw [partitions](#) and [voids](#) parallel to existing walls and partitions at a fixed offset distance. This help topic describes the use of offset snap for drawing partitions, but the information is equally applicable to drawing voids at a fixed offset from the perimeter.

The offset snap check box is displayed under [Direction snaps](#) in the **Drawing Options** data panel when the [Draw partitions](#) command is active.

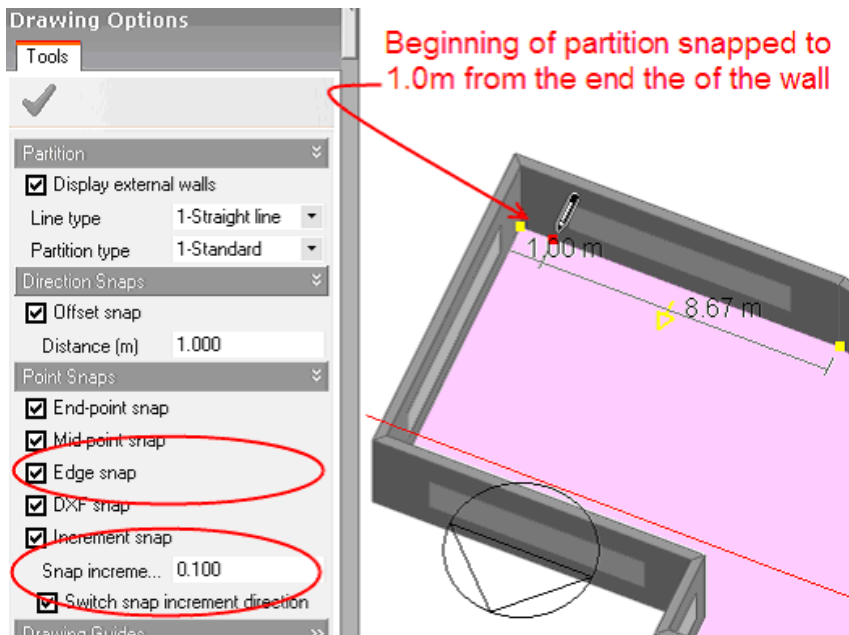
The offset snap can be used whenever a string of partitions is to be located at a fixed offset from a perimeter, but can be specifically applied to dividing large open plan spaces up into 'perimeter' and 'core' zones for thermal simulation.



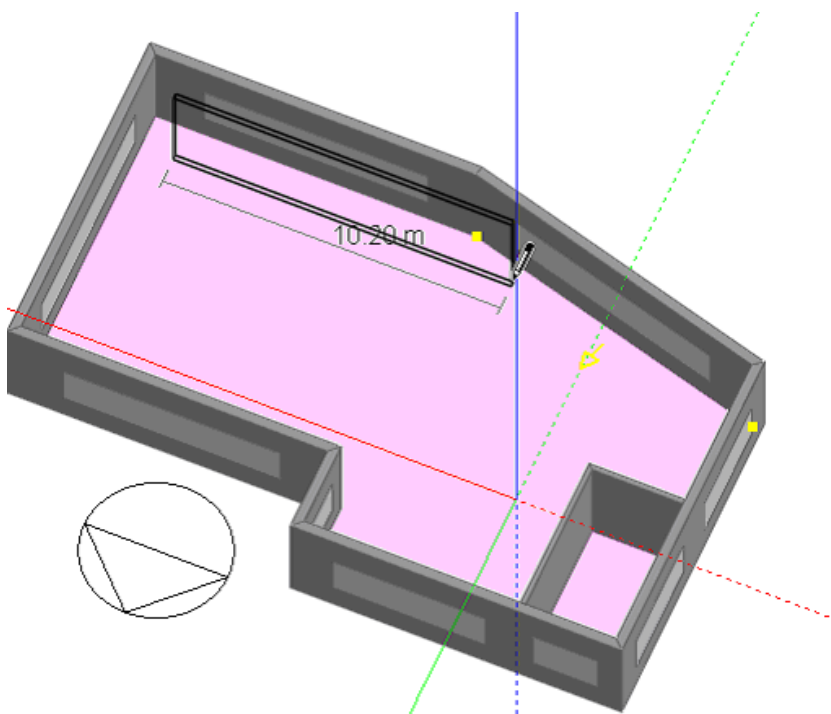
When the offset snap is switched on, an additional **Distance** item will be displayed under the offset snap check box in the Drawing options panel which allows you to define the offset distance. When you first switch on the offset snap, having clicked on the [Draw partition](#) icon, you will see offset snap indicators displayed as you move the cursor within the snap range of an existing wall or partition:



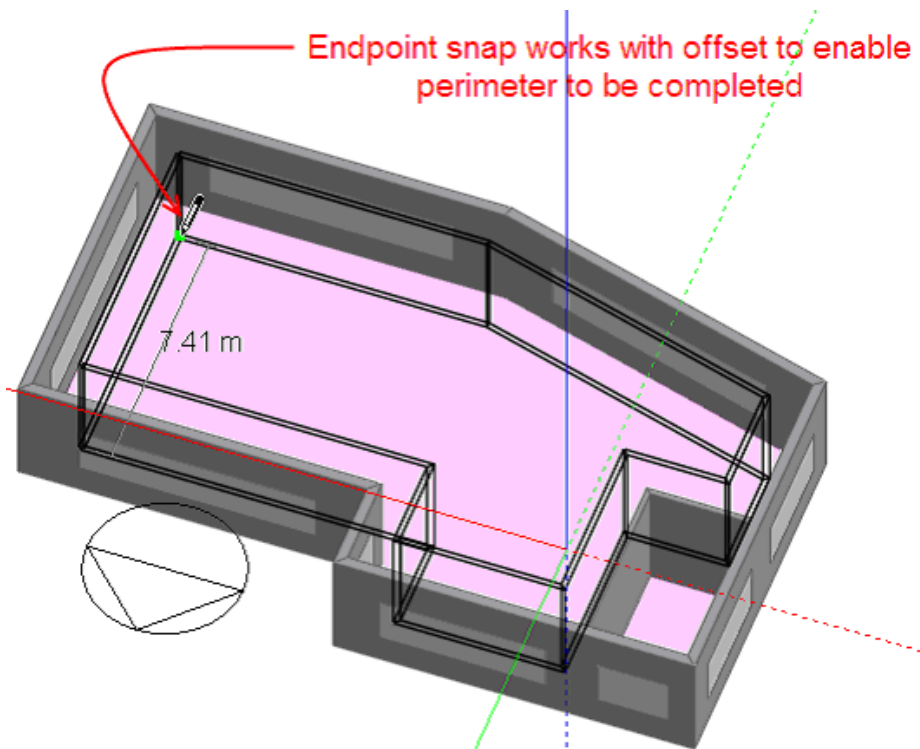
The offset snap works in conjunction with the other snaps and so, for example, you can start an offset partition at a fixed distance from the beginning of an existing wall using the [edge snap](#) in combination with the [increment snap](#):



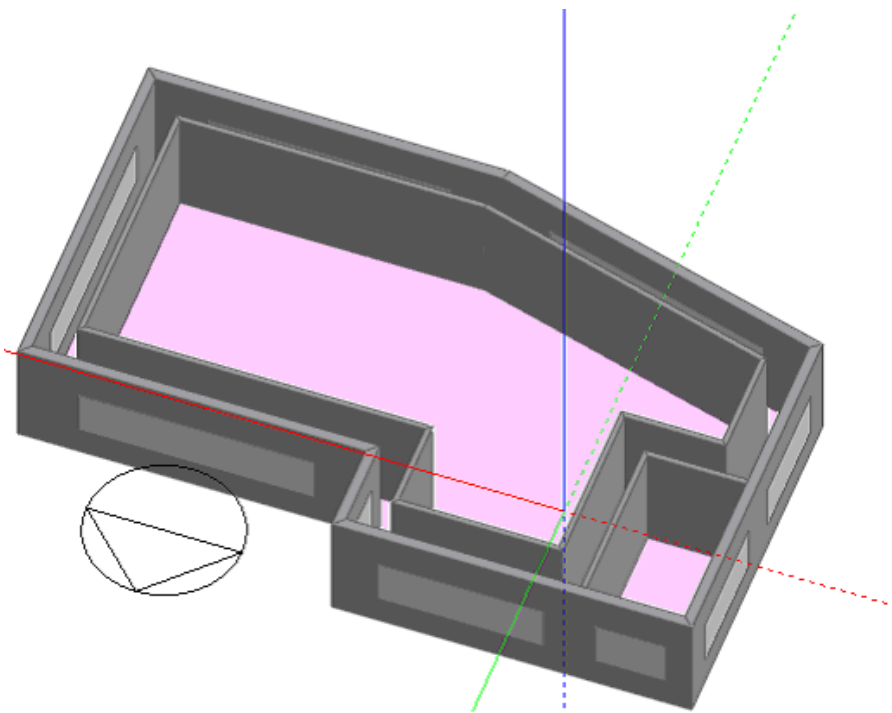
After placing the beginning of the partition, the 'rubber-banded' partition will be automatically snapped to the offset from the selected wall and as you move the cursor to the next connected wall segment, the offset snap indicator will appear again:



As you move the cursor, you will see that the offset snap interacts with other snaps allowing you to position partitions accurately. If you find that the other snaps are interfering with the offset snap, you can switch them off by pressing the SHIFT key.



Completed perimeter:



Note: The offset snap command prevents access to the normal end-point, mid-point and edge snaps. You must switch off offset snap to re-enable these.

Note: If you override the offset snap (by snapping to a drawing guide or edge), the offset snap has to be discontinued until the command (partition or void) is re-started.

Point Snaps

When the various point snaps are switched on, a coloured indicator is displayed when the mouse cursor is close to a key-point on an existing element allowing a perimeter or partition line end-point to be positioned precisely at the selected key-point.

See also the [Blocks and dimensioning protocols](#) and [Adding blocks and using the measure tool](#) tutorials.

The following point snaps are available:

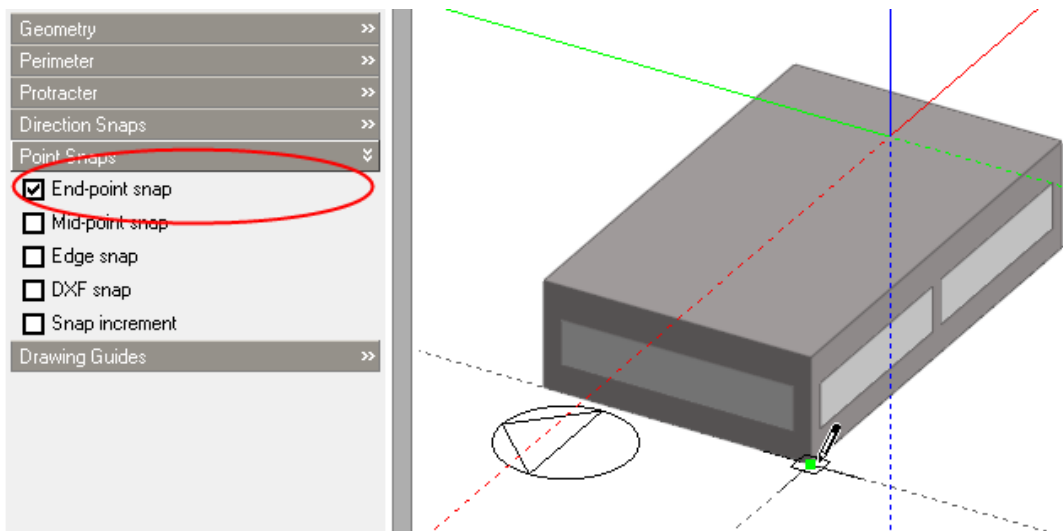
- [End-point snap](#)
- [mid-point snap](#)
- [Edge snap](#)
- [DXF snap](#)
- [Snap To Lower Perimeters](#)
- [Increment snap](#)

Note: In some cases Snaps can interfere with attempts to draw to a particular point and in this case you can temporarily switch off the appropriate snap(s) using the check boxes in the Drawing options panel in the bottom left of the screen.

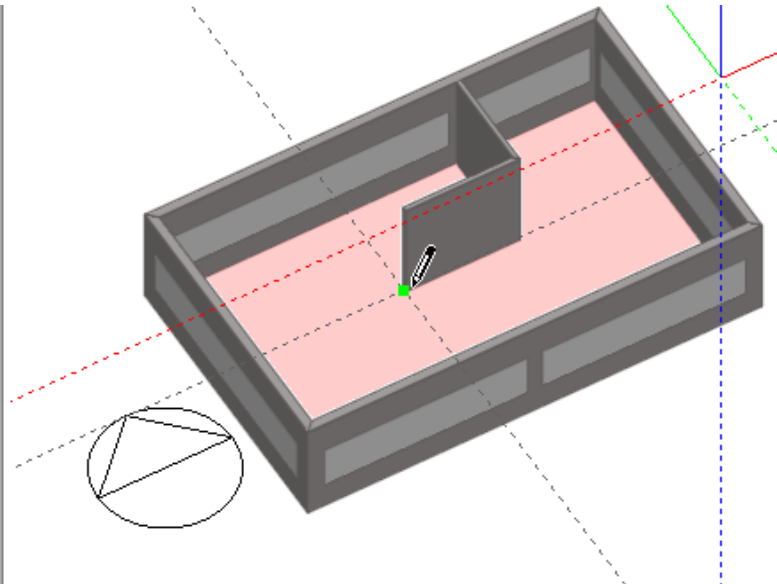
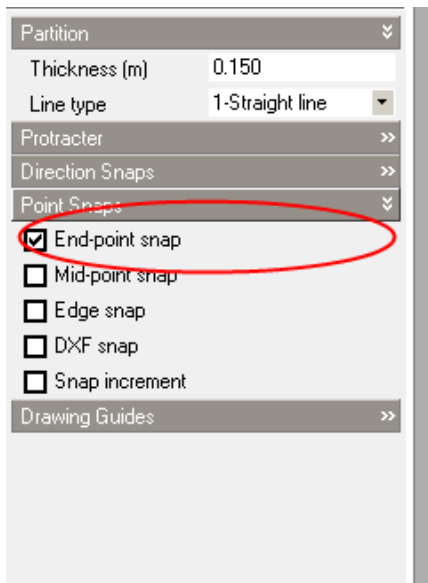
End-point Snap

This snap allows you to snap lines to the end-point of an existing edge. A green indicator is displayed when the mouse cursor is within the snap range of an end-point:

For example, to snap the starting point of a new block perimeter to the end-point of an existing block edge, move the mouse cursor onto the background and press SHIFT to lock the drawing plane to the ground plane. Keep the SHIFT key pressed and move the mouse cursor close to the required edge marker to activate the end-point snap:



Similarly to snap the starting point of a new block partition to the end-point of an existing partition, make sure that the 'End-point' snap is switched on and move the mouse cursor close to the partition end-point to activate the snap:



See also the [Adding blocks and using the measure tool](#) tutorial.

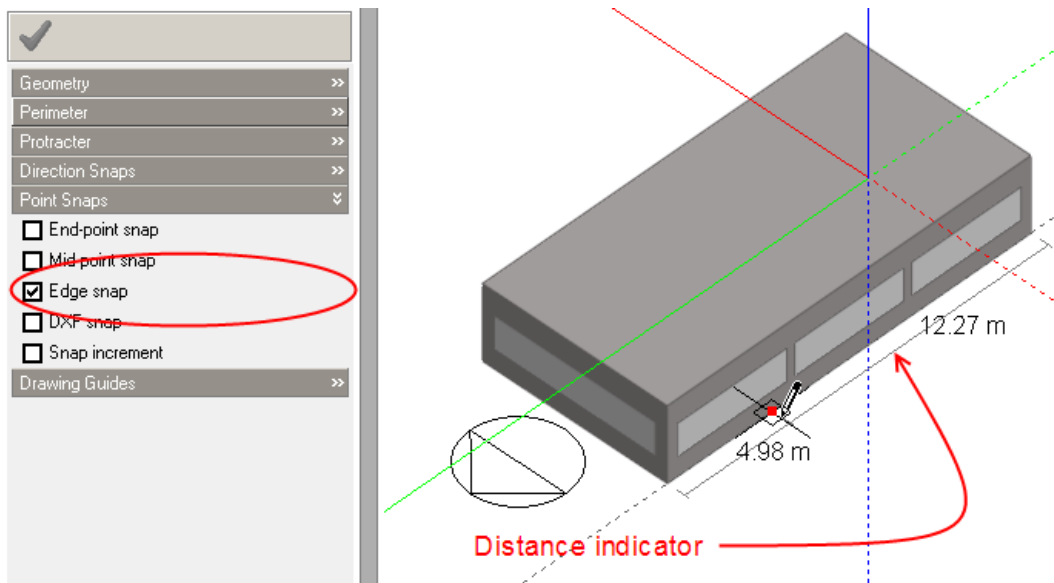
Mid-point Snap

For details of using the 'Mid-point' snap, please refer to the [End-point](#) snap.

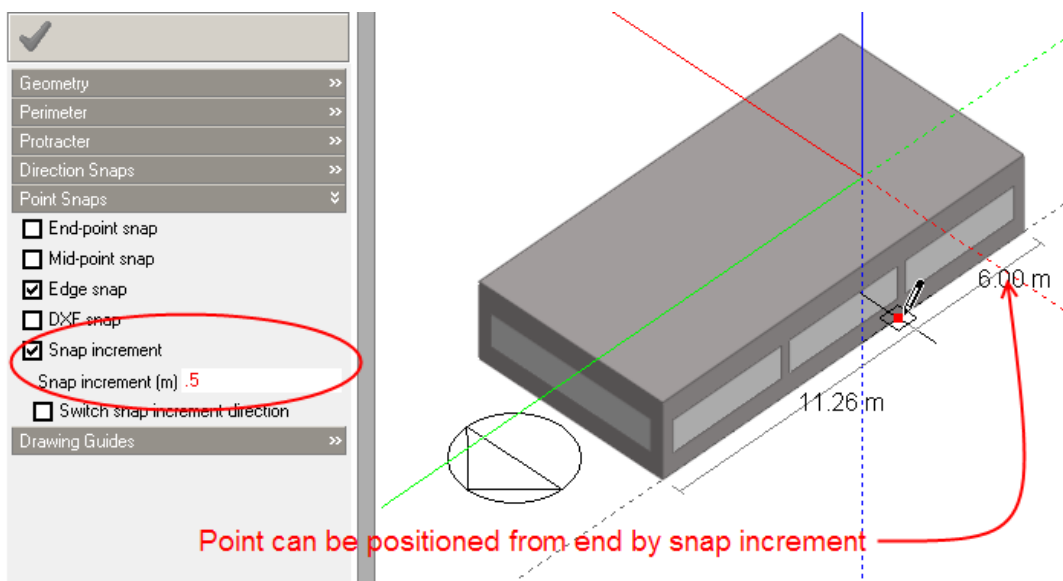
See also the [Adding blocks and using the measure tool](#) tutorial.

Edge Snap

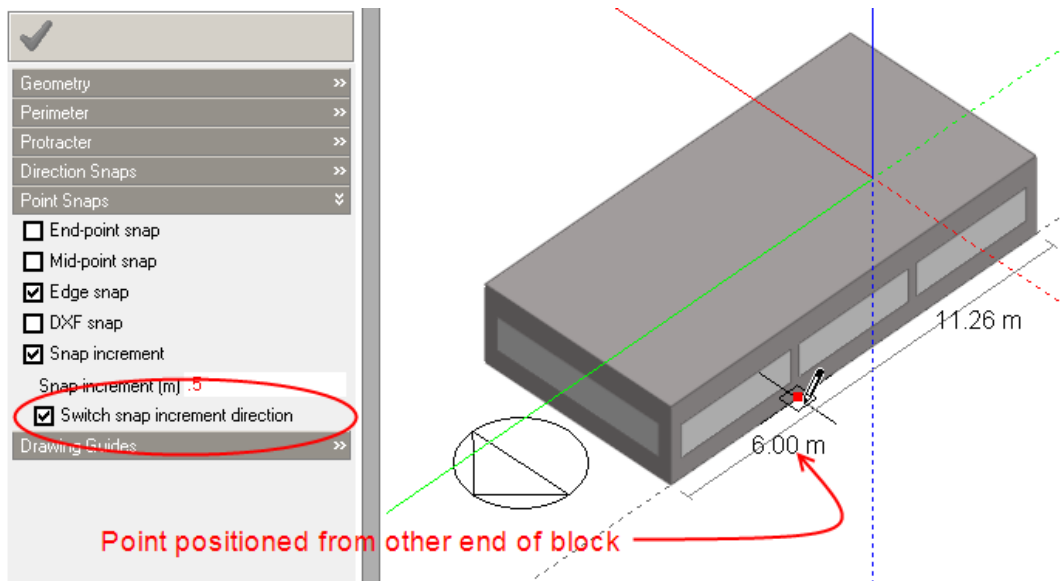
The edge snap allows you to snap to any point on the edge of an existing object. The edge snap can be used in conjunction with the increment snap to enable a point to be located along an edge by moving it in defined increments. A red indicator is displayed when the mouse cursor is within the snap range of an edge and distance indicators are also displayed to help in locating a point a specific distance from either end of the edge:



By switching the increment snap on, the point can be moved along the edge in defined increments. For example, if you wanted to start a block perimeter line at 6.0 metres from the end of the block, after switching on the increment snap, you could set the snap increment to say 0.5m and then move the mouse cursor to the edge at a distance of 6.0 metres from the end-point:



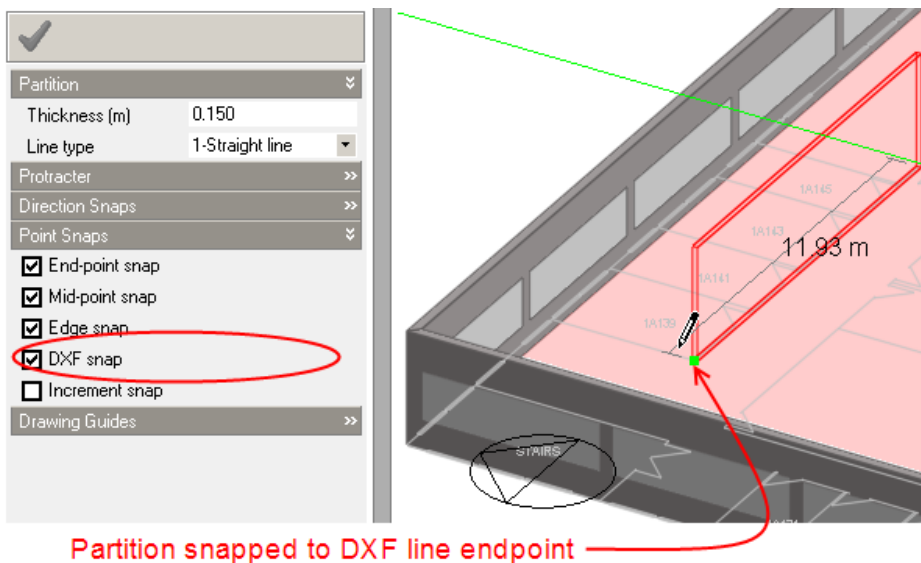
If you wanted to locate the beginning of the perimeter line at a specific distance from the other end of the edge, you can switch the end of the edge from which the increment snap operates by toggling the 'Switch snap increment direction' switch:



See also the [Adding blocks and using the measure tool](#) tutorial.

DXF Snap

The DXF snap only applies when [DXF data](#) has been imported from an external file. The DXF snap enables you to snap to the end-points of DXF lines:



Note: You should generally switch DXF snap off when drawing partitions at block level and simply use the DXF lines as a visual guide. This ensures that partitions correctly snap to perimeters instead of snapping to DXF lines which may be very close to the perimeter but not actually touching.

See also the [Importing drawing files](#) tutorial

Snap To Lower Perimeters

This snap allows you to snap to adjacent lower block perimeters and partitions. This snap is switched on by default but it can be useful sometimes to switch it off to avoid snapping to partitions in the block below.

When this snap is active, lines are also displayed at block level to represent any partitions and block perimeters from the block(s) below. Sometimes these lines can cause confusion, especially if a 2-D drawing has been loaded as well, and in this case it may help to switch the **Snap to lower perimeters** option off.

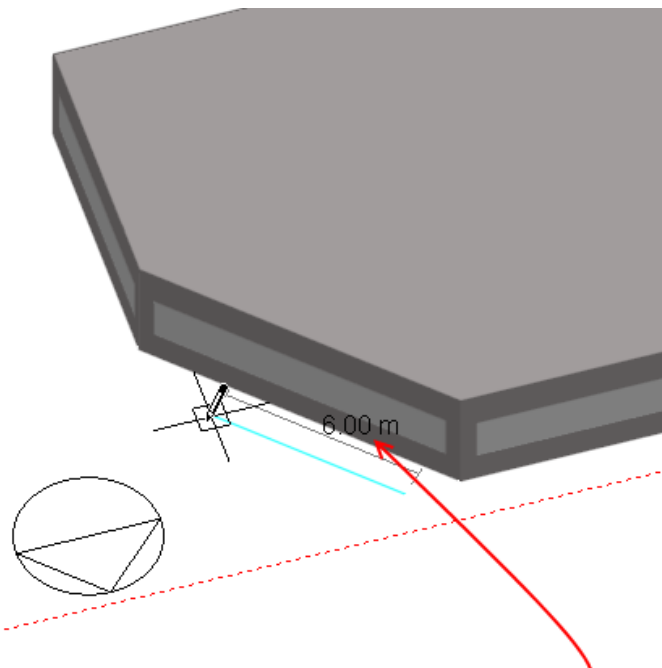
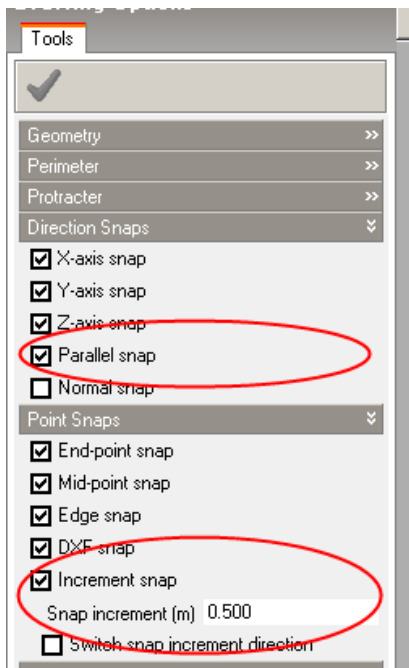
Increment Snap

The increment snap applies to any process that involves moving a point or a surface in a particular direction. Such processes include drawing block partitions or perimeter lines which have an active [direction snap](#) or dragging the surface of a block normal to the original surface position. The increment snap allows you to move a point in a particular direction by defined increments. You can think of the increment snap acting in the same way as a conventional CAD drawing grid and in fact if you were just to use the axis snaps it would have the same effect, however the increment snap is more powerful in that it provides this type of incremental snap along any defined vector not just the axis vectors.

See also the [Add surface, increment snaps](#) tutorial

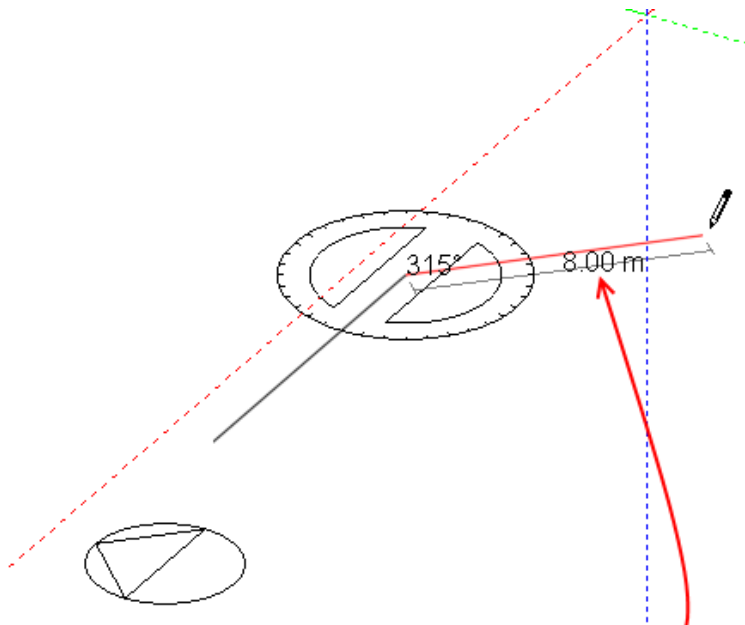
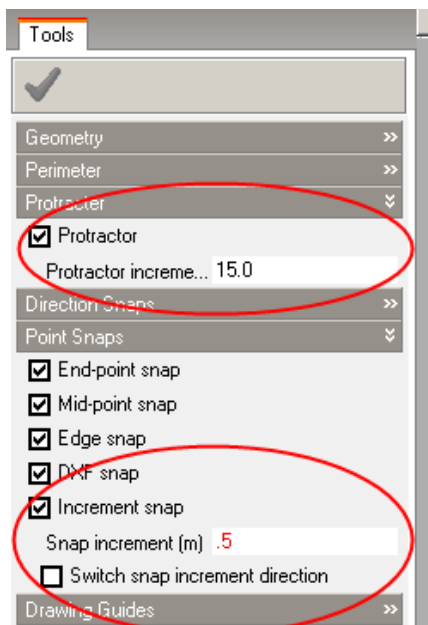
The increment snap may be particularly useful during early stage design when you want to place geometry with reasonable accuracy but don't want to be constantly keying-in distances.

Note: the increment snap will only come into operation when you are drawing a line or moving a surface along a defined vector. For example if you are drawing a block perimeter parallel with an existing block edge, the increment snap will operate when the parallel snap is active and in this case you would be able to move the mouse cursor along the parallel vector in increments of 0.5m:



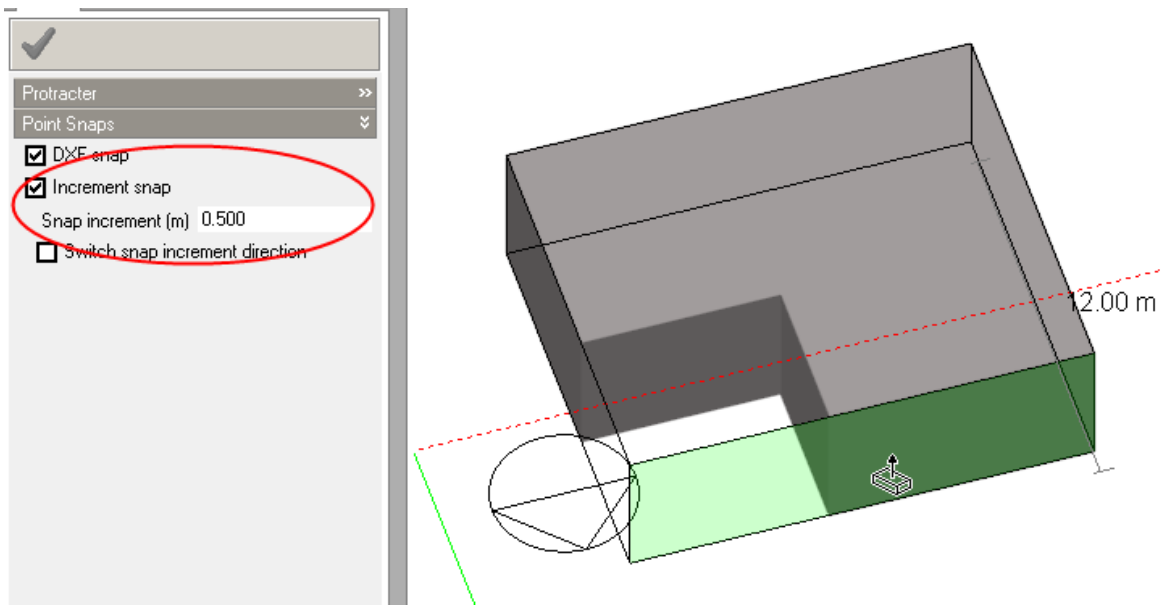
Increment snap operates when any of the direction snaps are active

The increment snap also works in conjunction with the [Protractor](#):



Increment snap operates when line snapped to protractor angle

You can use the increment snap to control the location of block faces when you extrude a block on creation or drag faces during an [edit](#) or [stretch](#) operation:



The increment snap also comes into operation when an edge snap is active, allowing you to snap to the edge by increments from either end, please see [edge snap](#) for details.

Switch snap increment direction

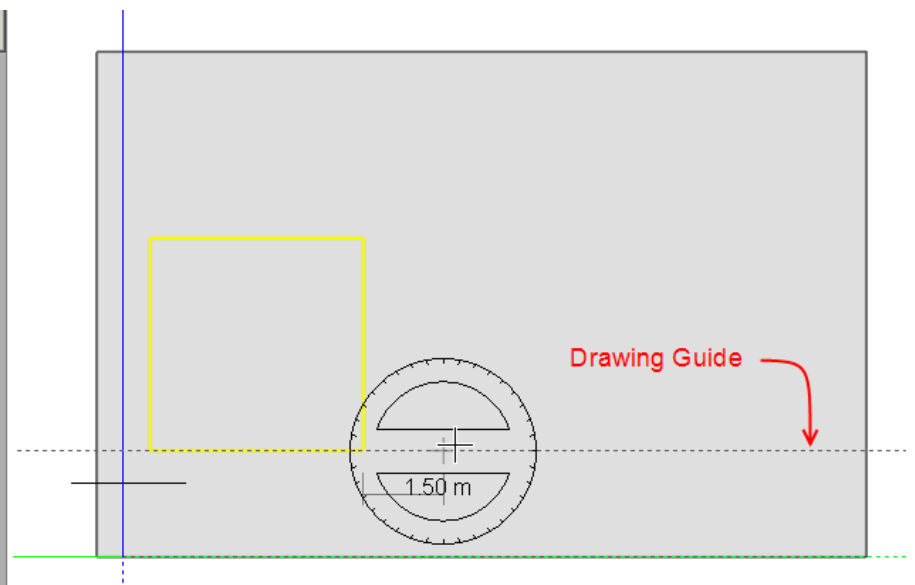
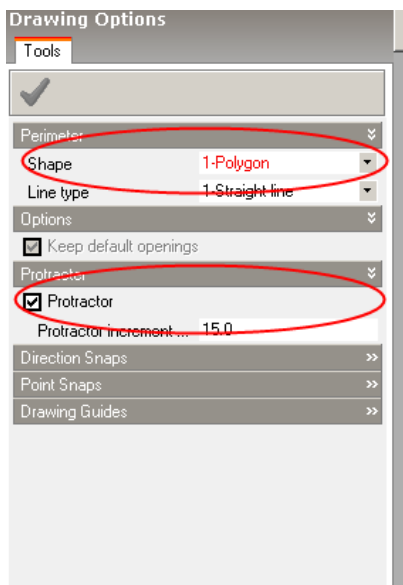
Increment snap provides increments between the 2 end points of the reference line and increments can only be generated from one end point at a time. You can reverse the direction so that increments are generated from the other end of the reference line by checking the **Switch snap increment direction** drawing option. This option is found immediately below the Increment snap option (screenshot above).

Drawing Guides

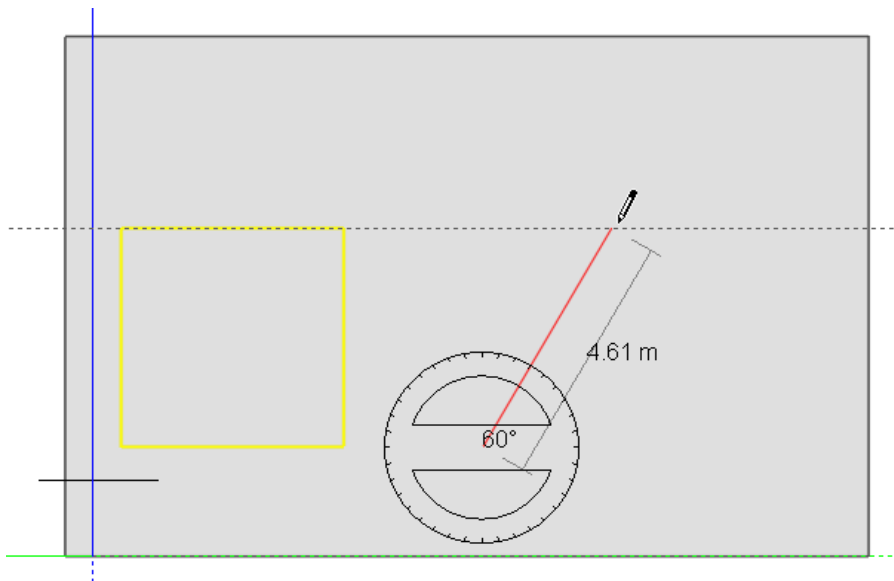
Drawing guides are invisible lines that are automatically generated as soon as a drawing plane becomes active during a drawing operation. Drawing guides are parallel with the active drawing plane X and Y axes and run through all end-points and mid-points of existing object edges, which lie in the same plane as the drawing plane. As you move the cursor in the vicinity of a drawing guide, it will become visible and you can then snap the cursor to any point on the guide. When drawing guides are used in conjunction with [Direction Snaps](#) or the [Protractor](#) they offer a powerful mechanism for locating perimeter lines in relation to existing object edges.

See also the [Adding blocks and using the measure tool](#) tutorial

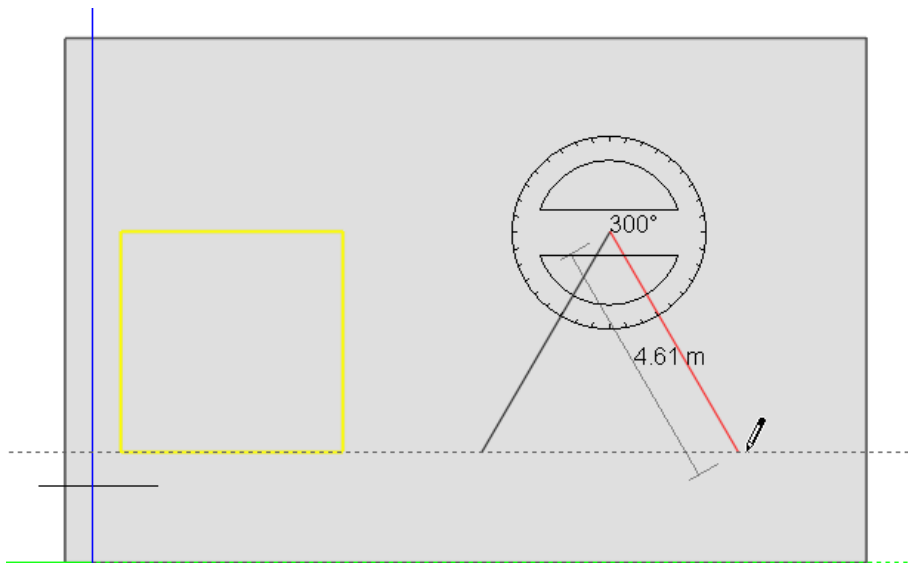
For example to create a triangular window in a surface with the same height as an existing opening, first click on the [Draw window](#) tool and check that the perimeter shape is set to 'Polygon', switch the protractor on and then move the cursor to align approximately with the lower edge of the existing window which will automatically snap the cursor to the required drawing guide:



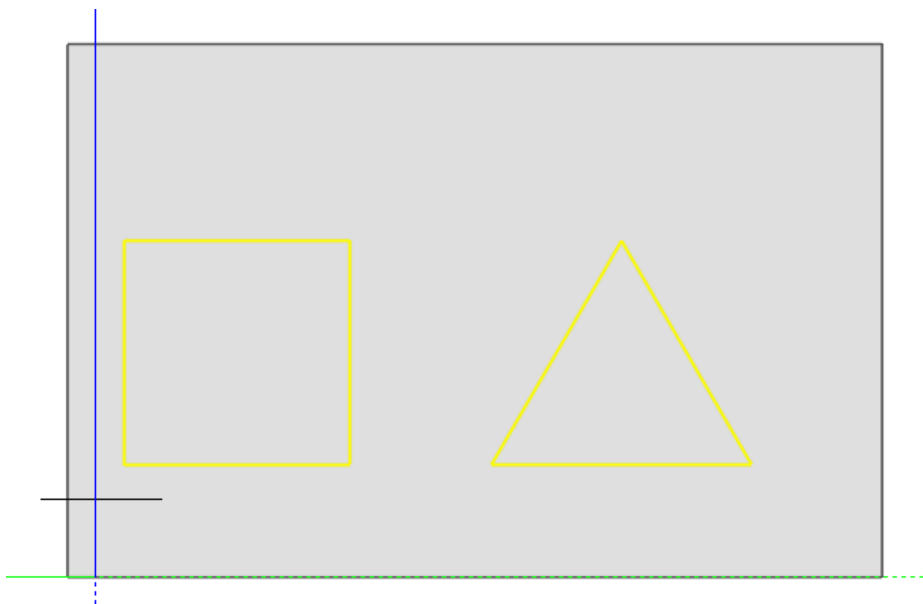
Click to place the window origin and move the cursor to snap to the drawing guide associated with the existing window upper edge, the protractor angle lock will automatically combine with the drawing guide:



Click to place the apex of the window and then place the next point in a similar fashion:



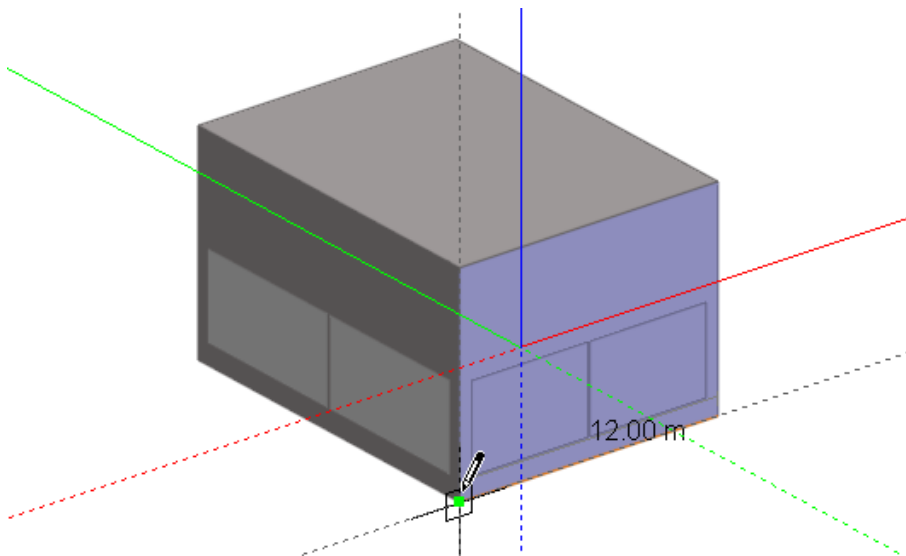
Completing the window perimeter:



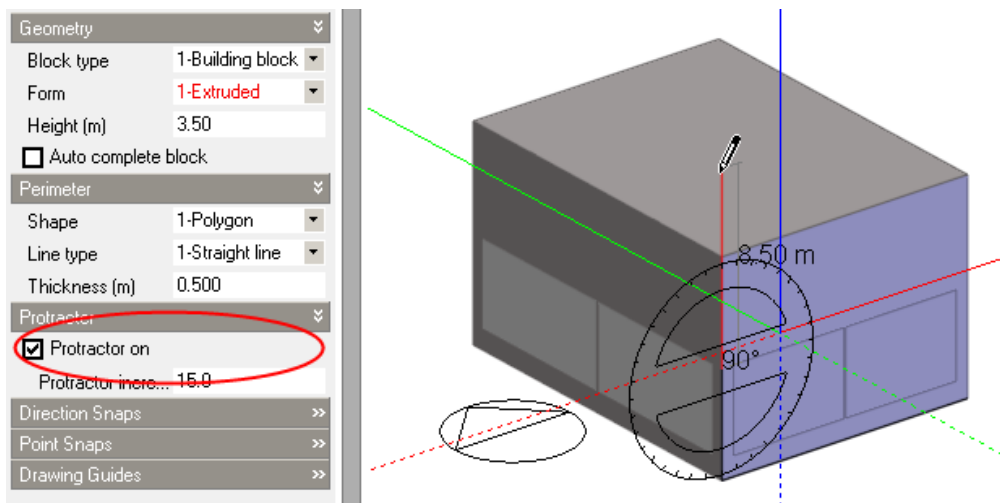
Note: In some cases Drawing guides can interfere with attempts to draw to a particular point and in this case you can temporarily switch off Drawing guides using the check boxes in the Drawing Options panel in the bottom left of the screen.

Protractor Tool

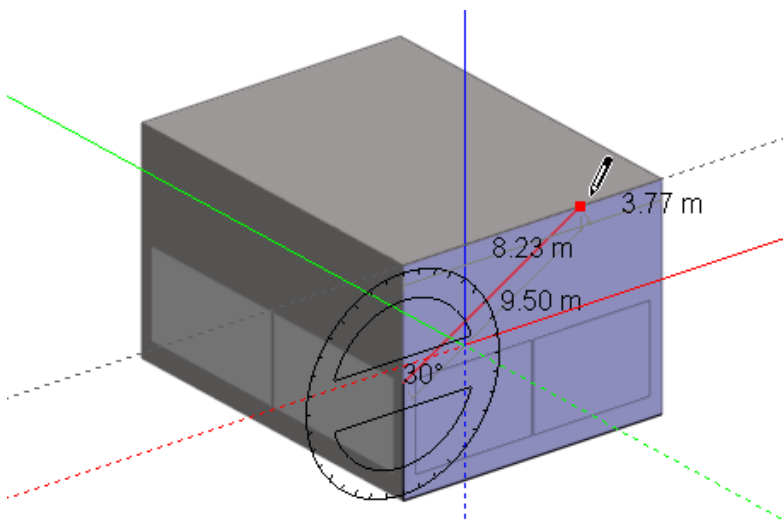
The protractor tool is used to draw perimeter lines and partitions at specified angles in relation to the current drawing plane. The protractor can be activated from the 'Drawing options' data panel, which is displayed when any of the drawing tools are used. For example, to create an offshoot projecting from an existing block with a roof slope of 30°, move the cursor over the existing block surface, press the SHIFT key to lock the drawing plane and draw the first perimeter line, snapping to the base edge using [end-point snap](#):



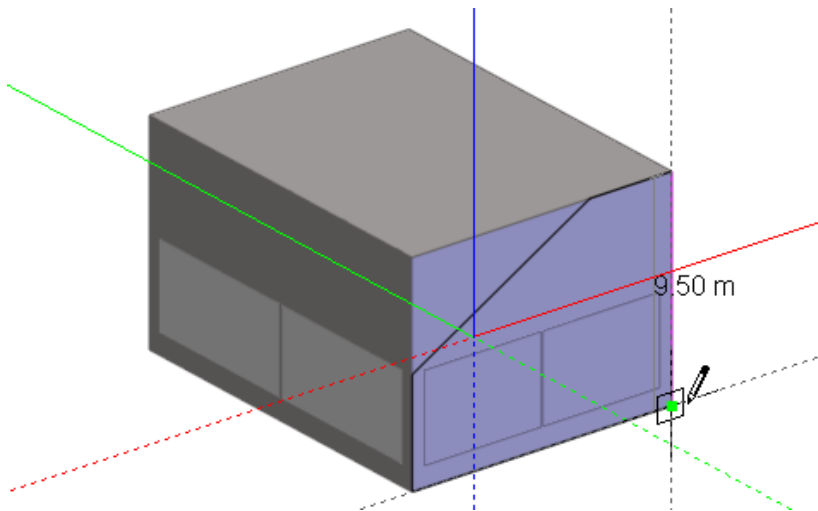
Draw the second perimeter line to connect to the mid-point of the block edge and then activate the protractor:



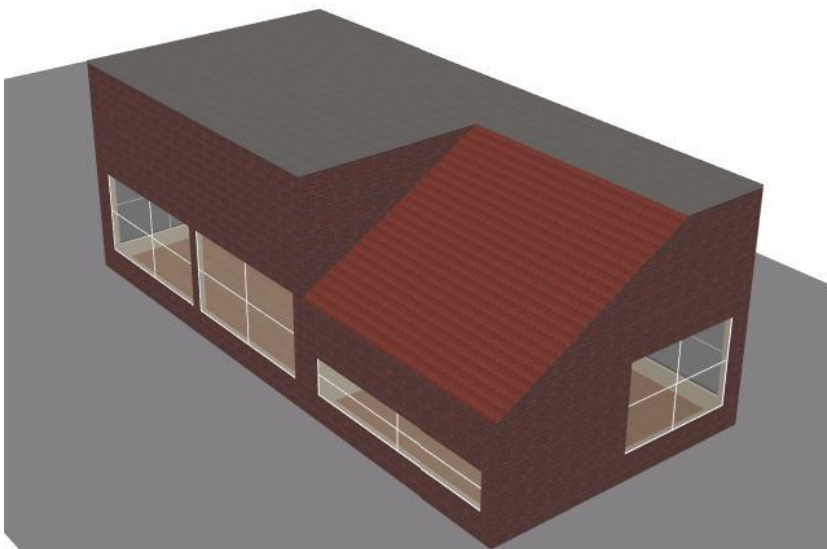
The protractor will automatically snap to the current drawing plane and as you move the mouse cursor, the perimeter line will snap to the protractor in angular increments, which can be set in the 'Drawing options' data panel. The rotation angle is displayed at the centre of the protractor. Move the cursor until the displayed angle is 30° and then move the snapped line towards the upper block edge. When the line end-point nears the edge, it will 'stick' to the edge and the [edge snap](#) marker will be displayed:



Click to accept the point and complete the perimeter by snapping to the block face vertices using the [end-point](#) snap:



Drag the perimeter to complete the offshoot:



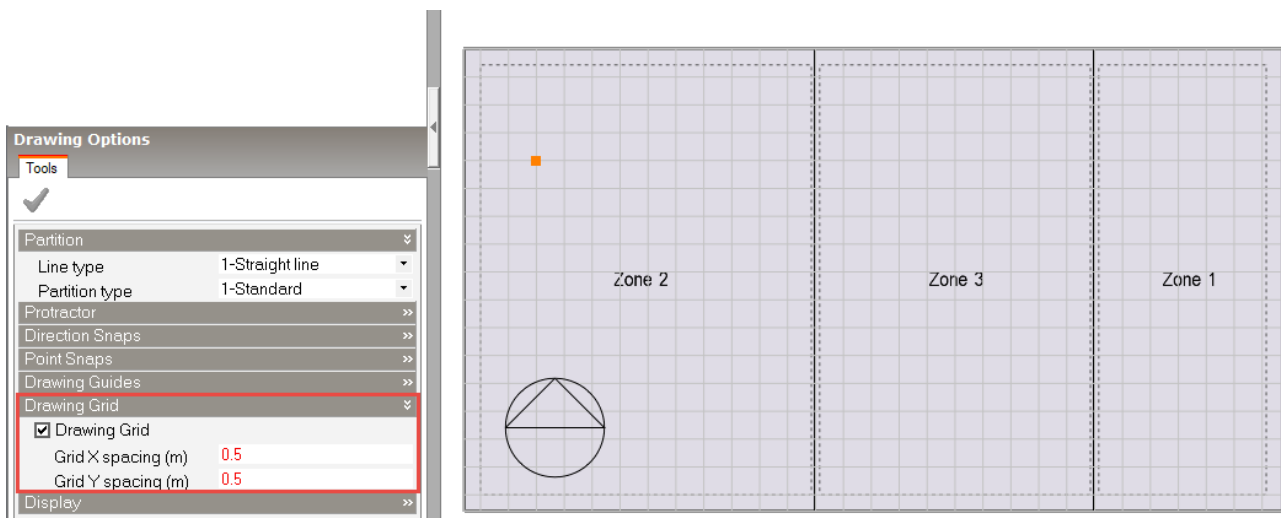
See also the [Cutting blocks, protractor tool](#) Tutorial

Drawing Grid

The **Drawing Grid** Drawing option provides a different, more visual way to obtain snap points based on a grid when drawing new objects at Block and Surface levels:

- **Block** when using any block level drawing tool ([Partitions](#), assembly, [Construction lines](#))
- **Surface** when using any surface level drawing tool ([Window](#), [Subsurface](#), [Hole](#), [Door](#), [Vent](#), [Boundary conditions](#), [Construction lines](#)).

The Drawing Grid can be accessed when at block or surface level from the bottom left **Drawing Options** panel when adding a new object. See the screenshot below which shows the Grid in use at block level.



The Grid can be used in combination with [Increment snap](#) and the other [Point Snap](#) options.

Note: Once the **Drawing Grid** option has been selected on the **Drawing Options** panel, the actual display of the grid only becomes active when a drawing tool has been selected (e.g. [Draw Window](#) at the surface level).

Set drawing grid origin

When the **Drawing grid** display is active, an additional **Set drawing grid origin** tool becomes available on the main Toolbar: 

This tool enables the drawing grid to be dimensioned from any key point in the display. For example, at the surface level, the corner of an existing window could be used to set the grid origin and then additional openings could be added relative to that existing window.

See also: "Increment Snap" on page 135

Construction Lines

Construction lines can be used to help in the creation of precise geometry. You can [draw construction lines](#) using the [snaps and the protractor](#) to lie at defined locations and then use these lines as a framework from which or to which various object geometries can be projected or attached.

At the building level, construction lines can also be snapped to the Z-axis to form vertical planes from which block geometry can be extruded.

Construction line snap points can be picked up at all levels regardless of the level at which they were drawn. For example all of these operations are possible:

- Draw construction lines at building level and pick up snap points for these at surface level for accurate drawing of openings etc.
- Draw construction lines at block level and access them at zone and surface levels for positioning openings, lighting sensors, CFD fans and assemblies.
- Draw construction lines at surface level to indicate the position of a shading device relative to windows and use snap points to draw and position a component block at building level.

Note: Because Construction lines are not part of the model (in the same way that [outline blocks](#) are not part of the model) they do not appear in the Navigator panel.

See also:

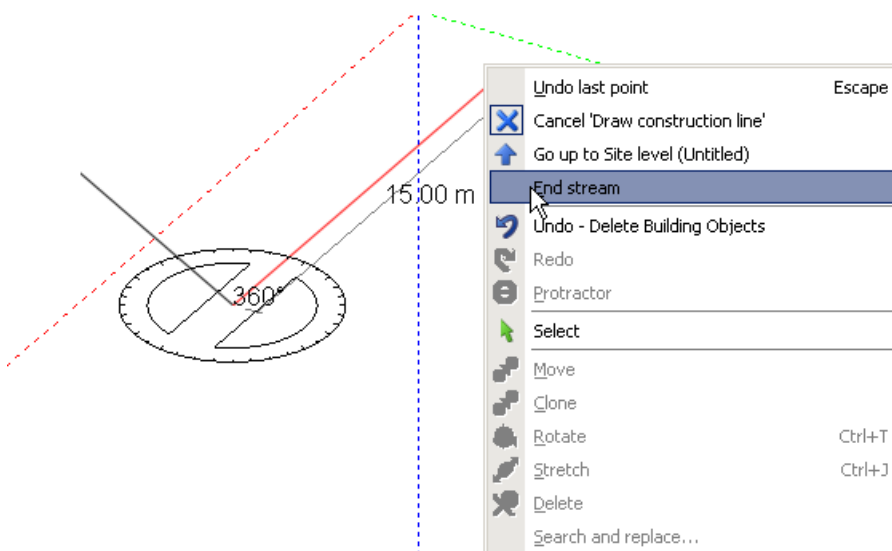
- "Drawing Construction Lines" on page 143
- "Removing Construction Lines" on page 147

More information on how to work with construction lines can be found in the [Shading using standard component blocks](#) and [Adding a roof, construction lines](#) tutorials.

Drawing Construction Lines

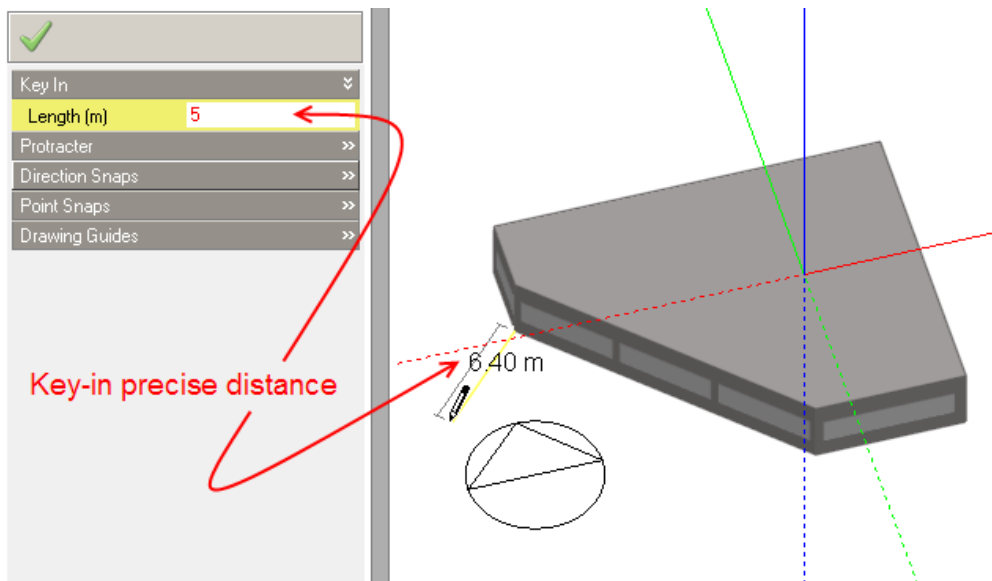


Construction lines are drawn in the form of line streams. To draw a construction line, click on the **Place construction line** tool and start drawing the first line by clicking anywhere in the active drawing plane then move the mouse cursor and click to place the line end-point, you can use [point](#) snaps, [direction](#) snaps, [drawing guides](#) and the [protractor](#) to help with alignment. Continue placing construction lines in this way. If at any time you want to cancel the last line that you placed, you can press the <Esc> key or select **Undo last point** from the right-click menu.

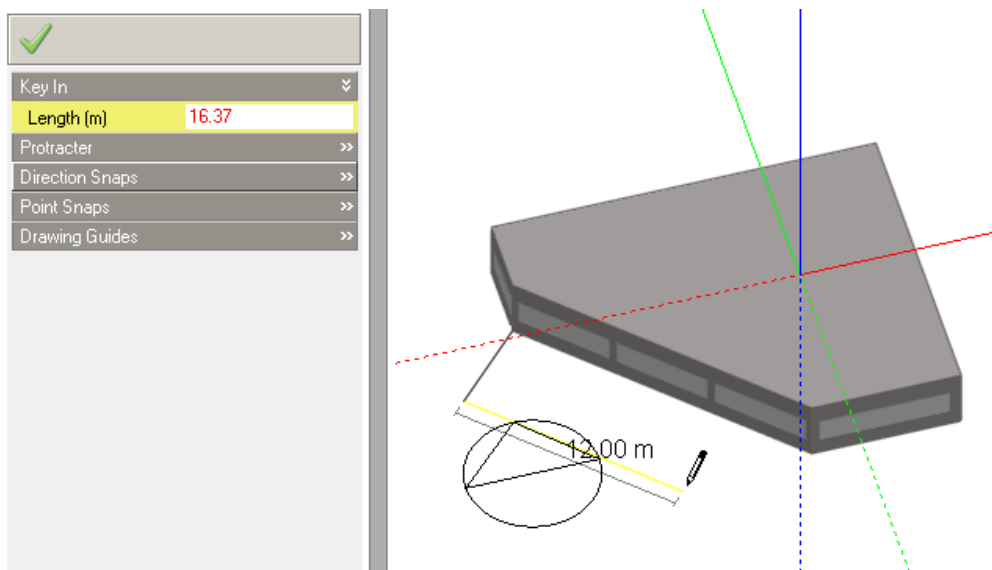


Note: Although each construction line is a 1-D object, when placing a stream of construction lines DesignBuilder considers the lines to lie on a single drawing plane. Once you have defined three points in the line stream, that defines the drawing plane and all subsequent points will be located in that plane. The line stream can be ended at any time by pressing the <Enter> key and then the next segment can be positioned on any other plane.

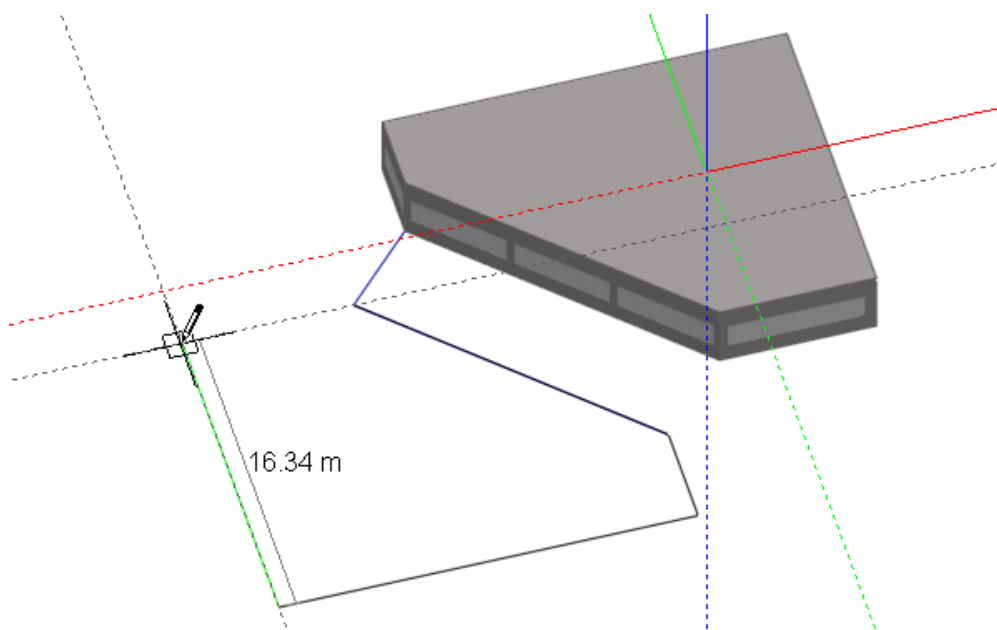
Construction lines have numerous applications in forming temporary attachment points during the creation of precise geometry. As a simple example, if you want to place a block at a precise distance from the angled wall of an existing block, you could switch the [Normal](#) snap on, click on the **Place construction line** tool and draw a perpendicular line a precise distance from the wall. To draw the perpendicular line, place the first point at an end-point of the wall baseline, move the mouse cursor to touch the baseline and press <Shift> to lock the direction inference, then move the mouse cursor to activate the perpendicular lock and key-in the required distance:



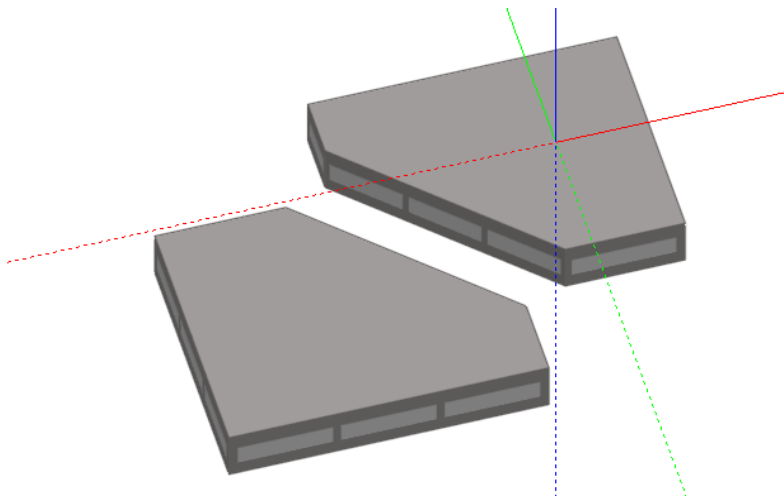
You could then place a second construction line normal to the first line and key-in the required length:



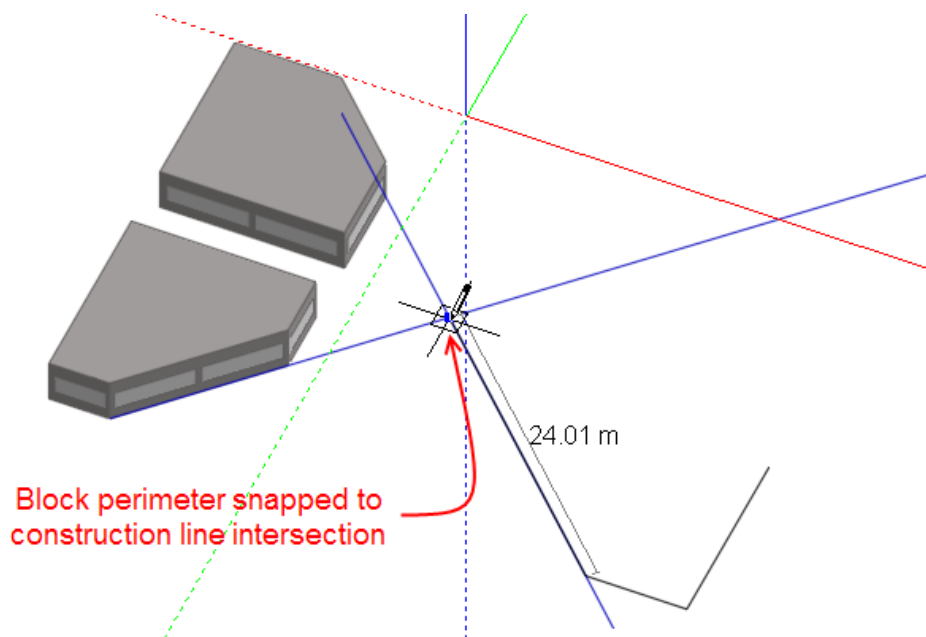
You can then snap to the construction line to draw the new block perimeter:



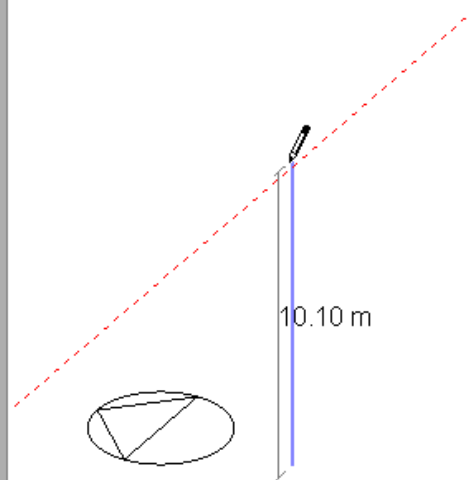
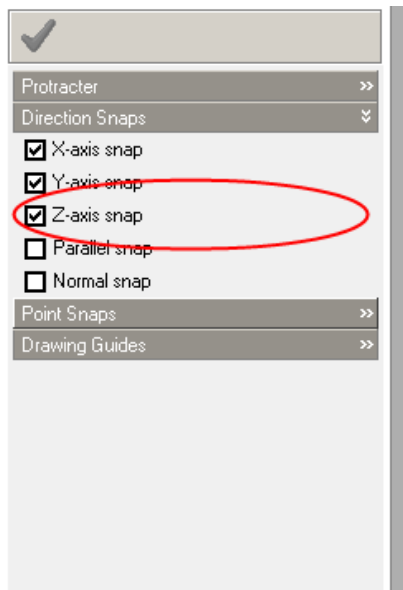
The finished block:



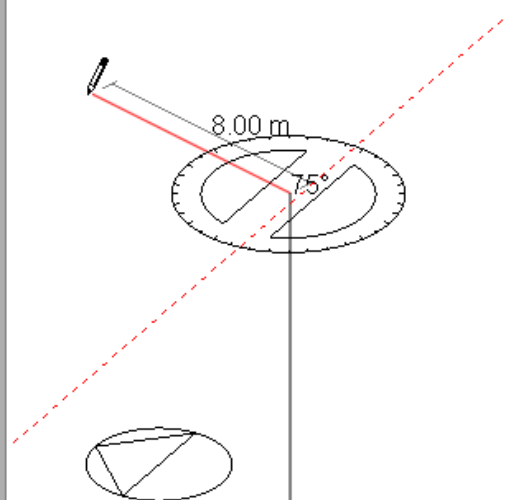
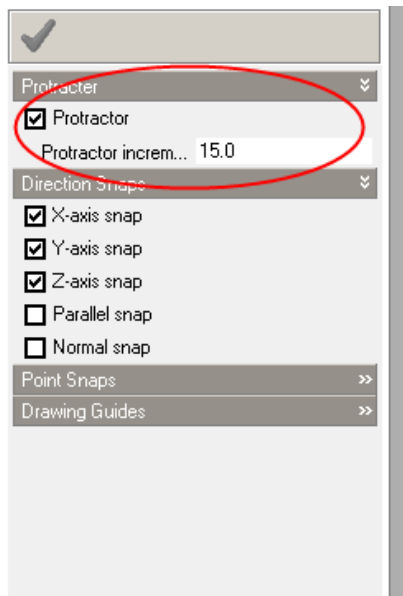
You can also snap to the intersection of construction lines:



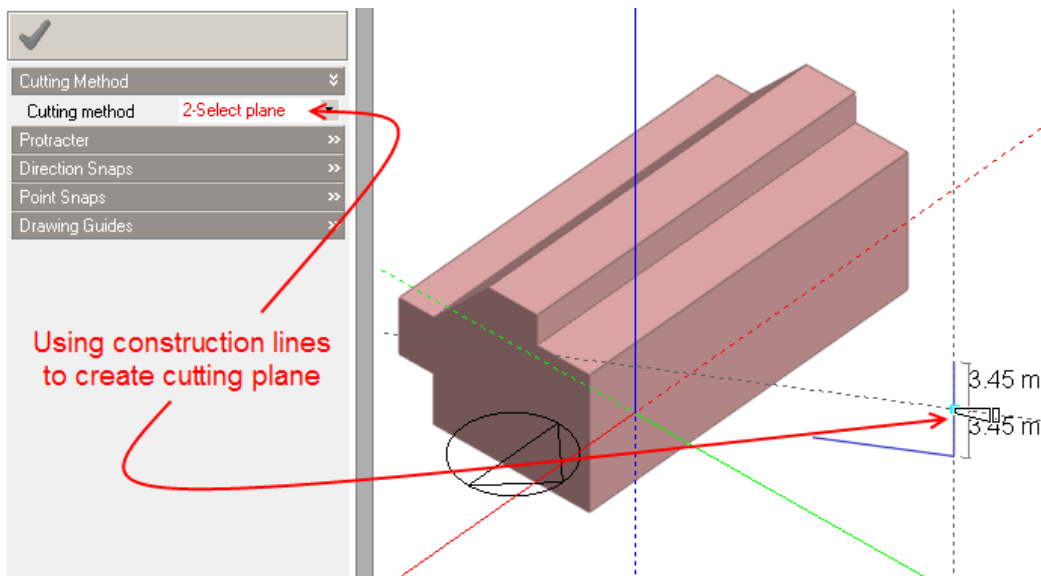
At the building level, construction lines can be snapped to the Z-axis in order to create vertical planes to which block geometry can be attached. To draw construction lines in a vertical plane, switch the Z-axis snap on, click on the 'Place construction line' tool and place the first point on the ground plane or any other horizontal plane. Move the mouse cursor to snap the current line to the Z-axis:



You can then use the protractor to place the next line in order to accurately define a plane:



After placing the second line, the two construction lines form a vertical plane that can be used for extruding blocks or as a plane to be used as a cutting plane.



Note: Construction lines can be copied, rotated, scaled etc like any other object.

Note: When drawing construction lines at **block level** it is possible to get a fuller range of snaps by selecting the **2-Zone inner surfaces** Snap mode.

See also the [Adding a roof, construction lines](#) tutorial

Tip: Construction line streams are used as a part of the process when using the [Follow Path](#) tool.

Removing Construction Lines

You can remove construction lines by [selecting](#) and [deleting](#) them individually, or you can remove all construction lines from the model by selecting the **Remove all construction lines** command from the [Edit](#) menu.

See also the [Adding a roof, construction lines](#) tutorial

General Operations

This section describes general geometric operations:

- [Select](#)
- [Move](#)
- [Clone \(copy\)](#)
- "Mirror" on page 153
- [Rotate](#)
- [Stretch](#)

- [Delete](#)
- "Measure" on page 159

Select



The **Select** tool acts like pressing the <Esc> key to cancel another command and enable graphical selection of model objects.

Selecting a single object

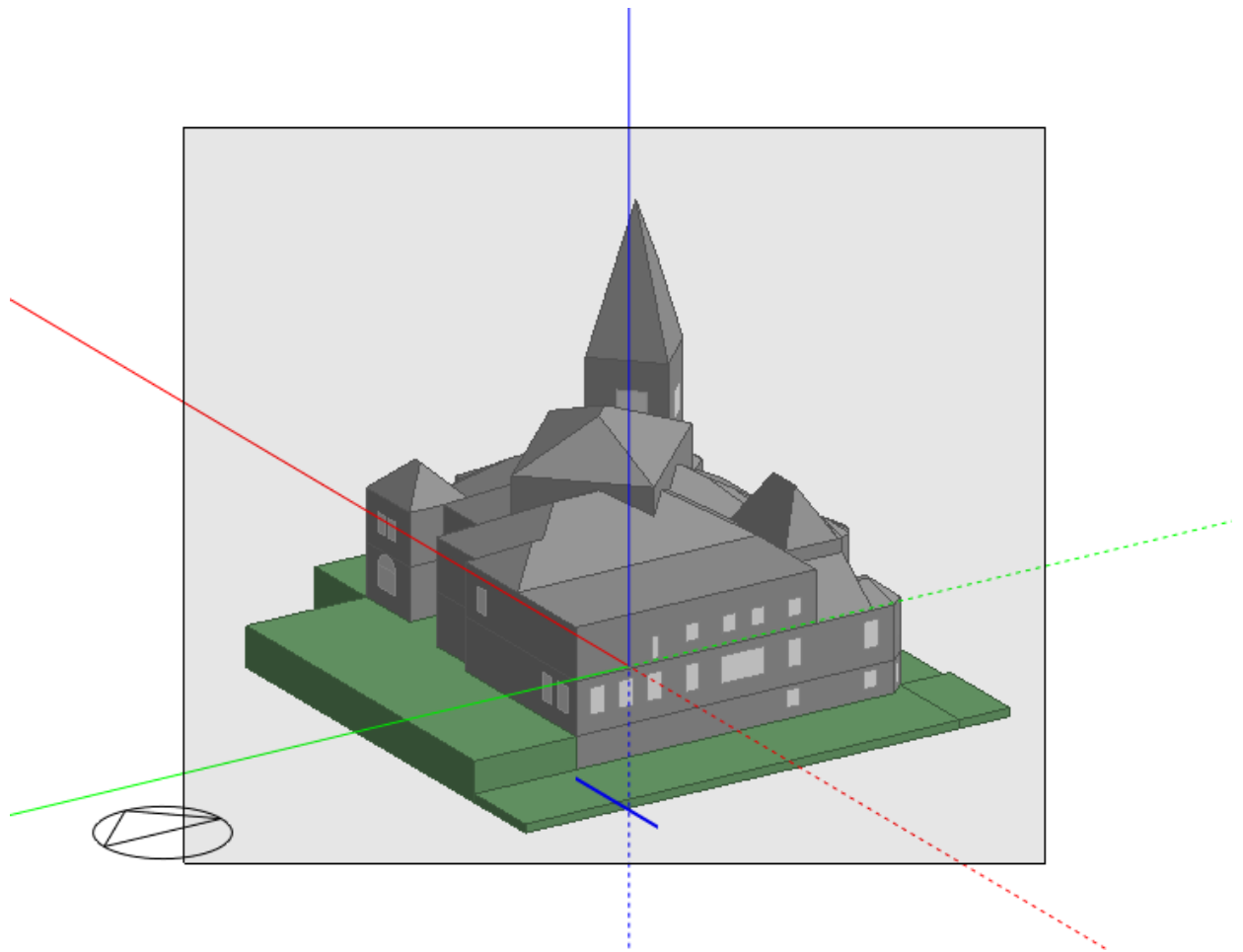
To select an object, click on the **Select** tool, or press <Esc>, or select **Select** from the right-click context menu, then move the mouse cursor to the object to be selected and click with the mouse to select it. The colour of the object temporarily changes to light red to indicate that it is selected.

Selecting multiple objects

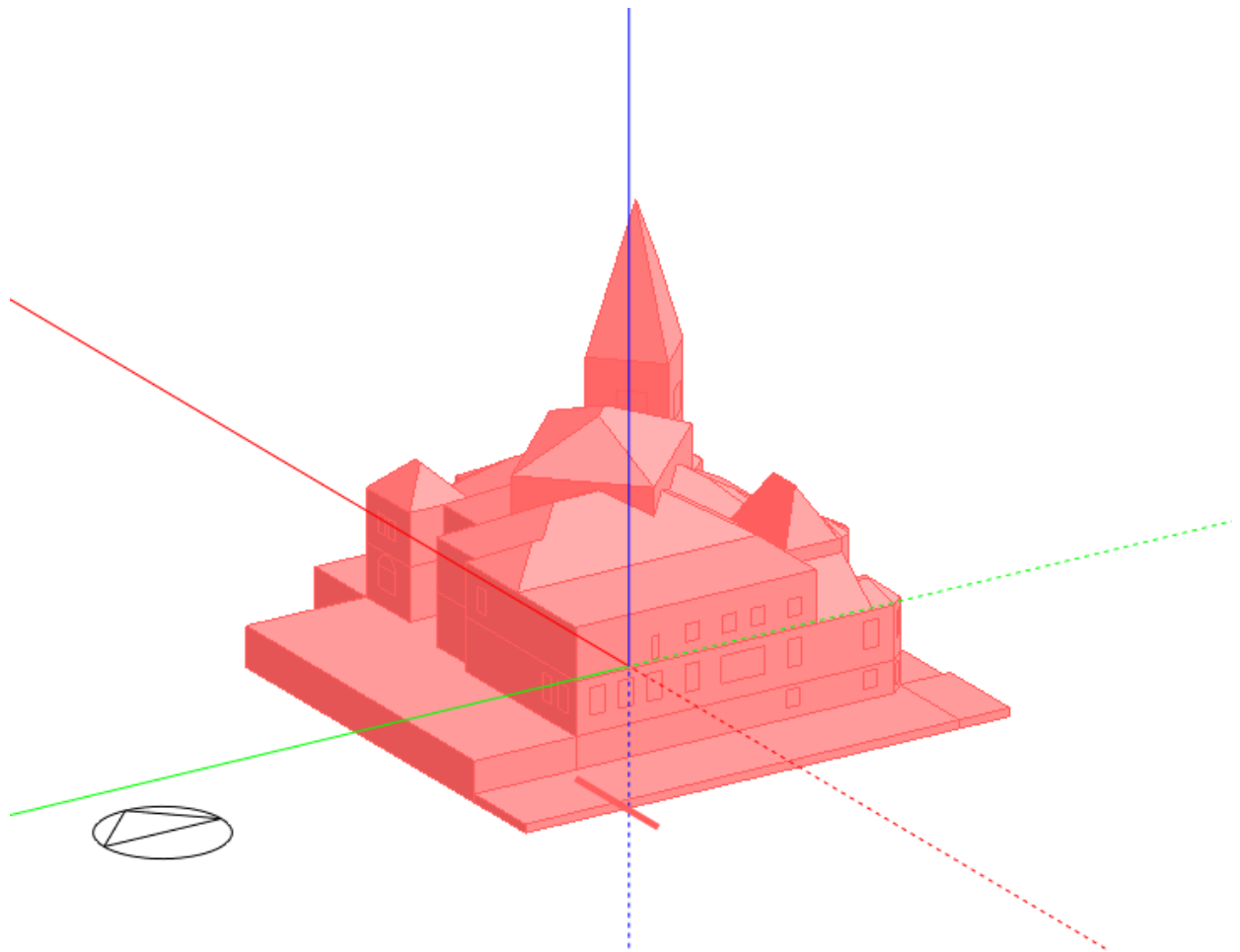
To select multiple objects, use one of the following methods:

- Press and hold the <Ctrl> key while clicking on objects, or,
- Drag a selection rectangle (aka "rubber band") by pressing and holding the left mouse button and dragging a window across the required selection area. When you release the mouse button, all the objects that are entirely within the selection window will be selected. Any objects that are only partially within the selection rectangle will not be selected. This is often the preferred selection method when many objects need to be selected as it doesn't require each item to be clicked on individually.

The screenshots below illustrate the process.



Dragging a selection rectangle around all blocks in a model



On releasing the mouse key, the objects are selected.

Tip: If you need to select most objects within an area, but not all of them, you can use the selection rectangle tool to select all objects and then deselect the objects that do not need to be selected by holding the <Ctrl> key and clicking on them.

Note: When using the selection rectangle please be aware that any openings in the selection rectangle that are hidden by model surfaces will also be selected. It is therefore important to carefully check which openings have been selected when using the select command before carrying out a significant operation on the selection set, such as deletion.

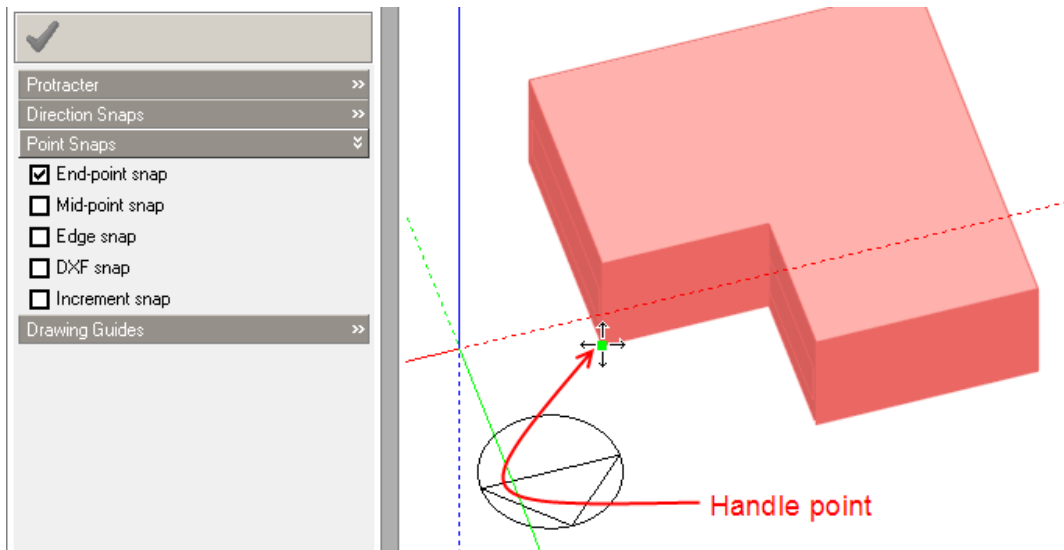
See also the [Selecting, moving, cloning and deleting objects](#) tutorial.

Move

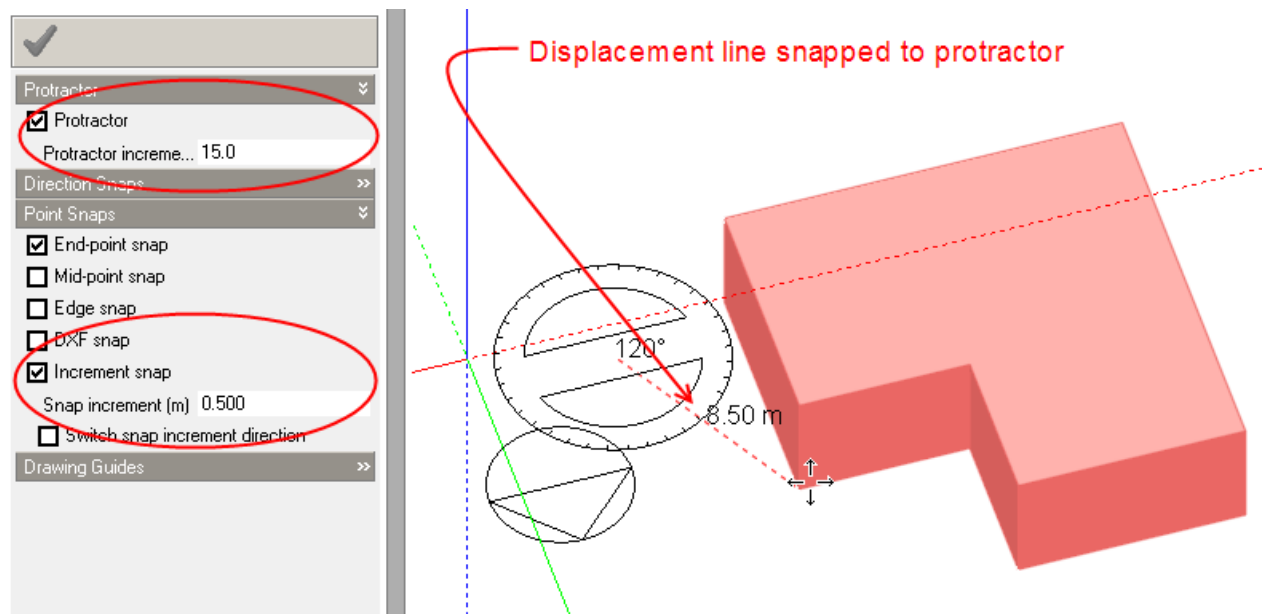


You can use the **Move** command to move blocks, partitions, courtyards and openings. You can also use Move to [set the position of daylight sensors](#). To move an object, first select it and then click on the **Move** tool or select 'Move' from the right-click context menu. If you want, you can move several objects at the same time by

doing a [multiple selection](#). You then need to define a 'handle point' for the move operation, the handle point can be any point on a model surface or ground plane but more usually will be a vertex on the object that you want to move, so make sure that the end-point snap is switched on, move the mouse cursor to the required object vertex and snap to it:

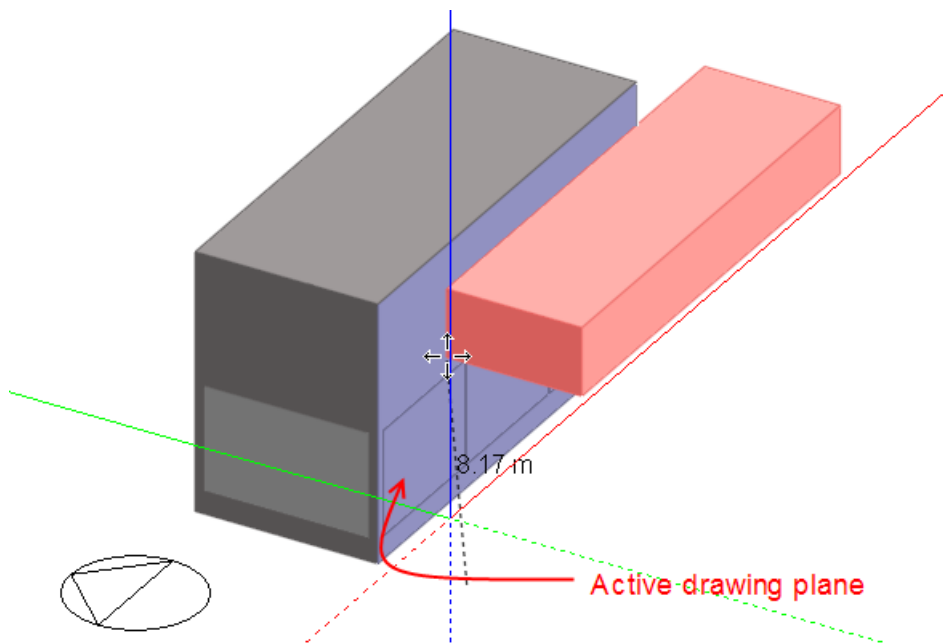


When you release the mouse button, you will be able to move the object to any required location. As you move the object, a displacement line is displayed to show how far the object has been moved, the displacement line can be snapped to the [axes](#) or the [Protractor](#). When a [direction snap](#) is active the [increment snap](#) can also be used:

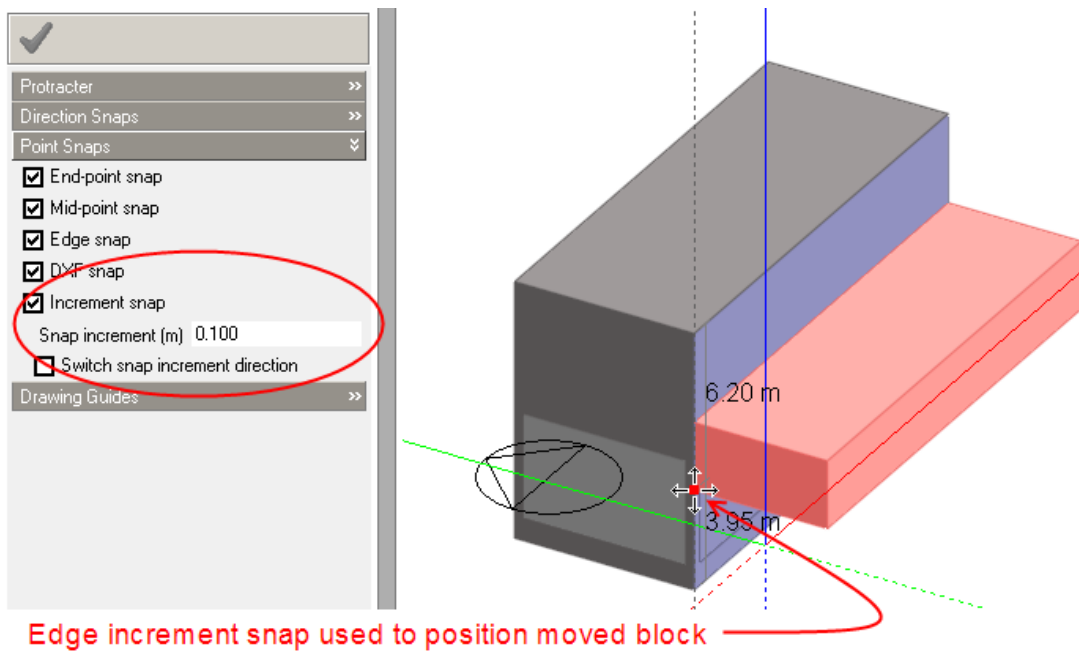


You can then click the mouse button to place the object.

At the building level, blocks can be moved in three dimensions onto any existing block face in the model. As you move the block around the model, each block face will highlight in blue as the block handle point is moved across it, indicating that the face has become the active drawing plane.



You can lock the active drawing plane by pressing the <Shift> key to position the block:



See also the [Selecting, moving, cloning and deleting objects](#) tutorial.

Clone (Copy)



For details of using the **Clone** tool, please refer to the [Move](#) tool. The process is identical except that when cloning, the original block is retained.

See also the [Selecting, moving, cloning and deleting objects](#) tutorial.

Mirror

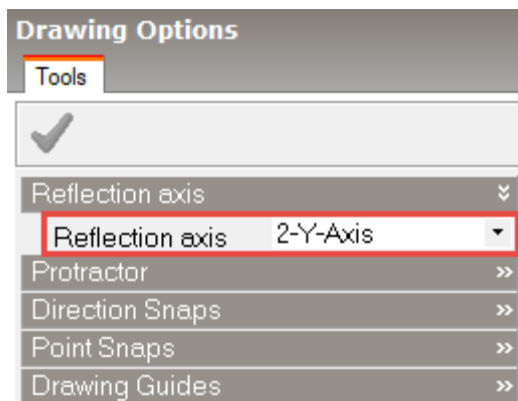


Mirrored copies of blocks, partitions, courtyards and openings can be created through the **Mirror** command. Objects can be mirrored in either the x or y axis. The process involved in using the mirror command is described below.

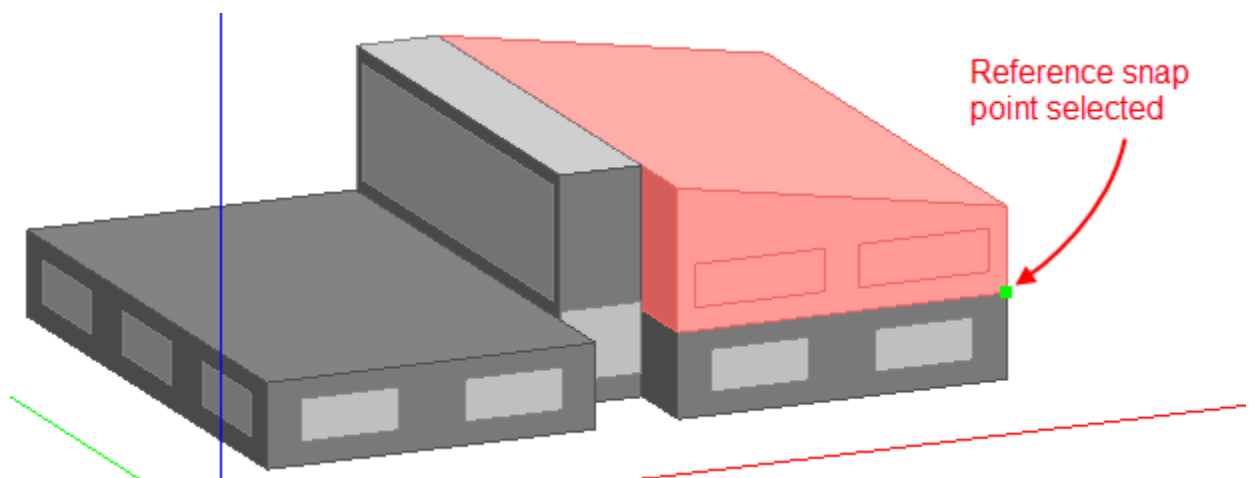
Mirror Block

To use the Mirror command to create a mirrored copy of a building block:

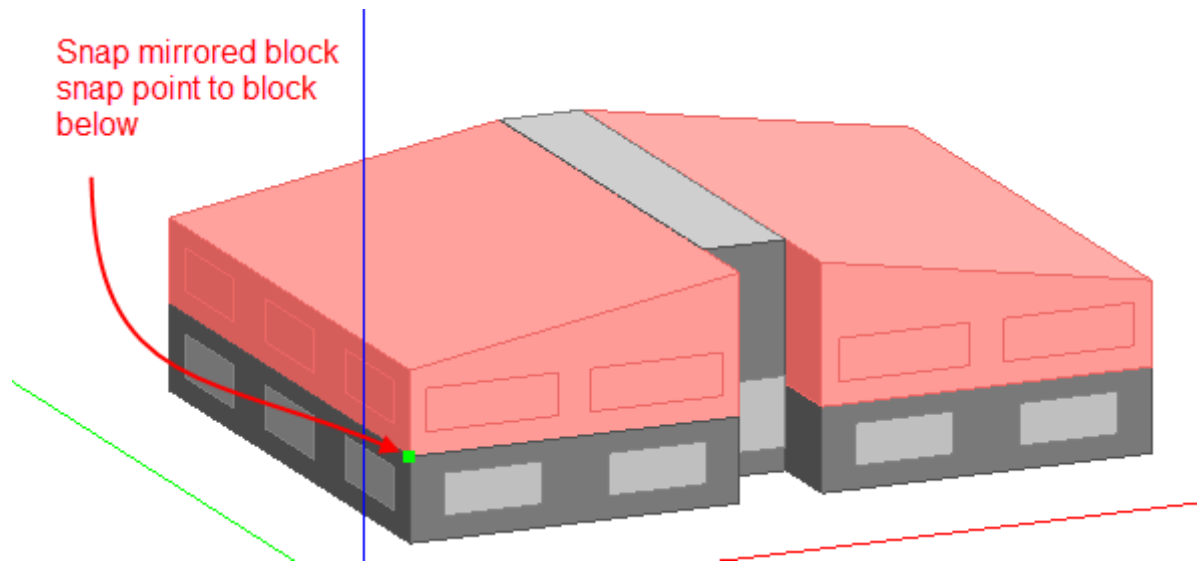
1. Go to building level (or block level if you wish to mirror a block-level component block or assembly).
2. Select the block to be mirrored.
3. Click on the **Mirror** toolbar icon.
4. In the Drawing Options panel select the Reflection axis. Choose from x or y-axis as the mirror plane.



5. Select the reference point on the existing block. You should choose a point whose corresponding mirrored point can be used to accurately snap the new mirrored copy in place.



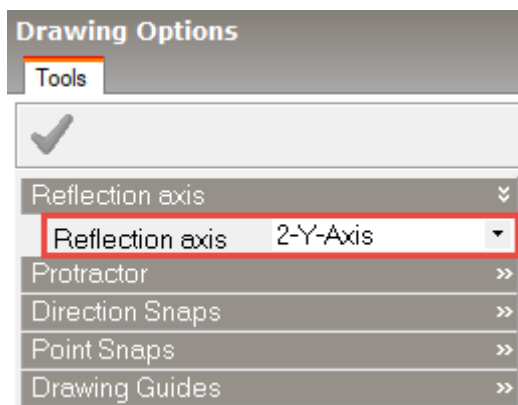
6. Once you have selected the reference point the mirrored copy is created and can be positioned as required, usually by snapping the mirrored reference point to the required place.



Mirror Openings

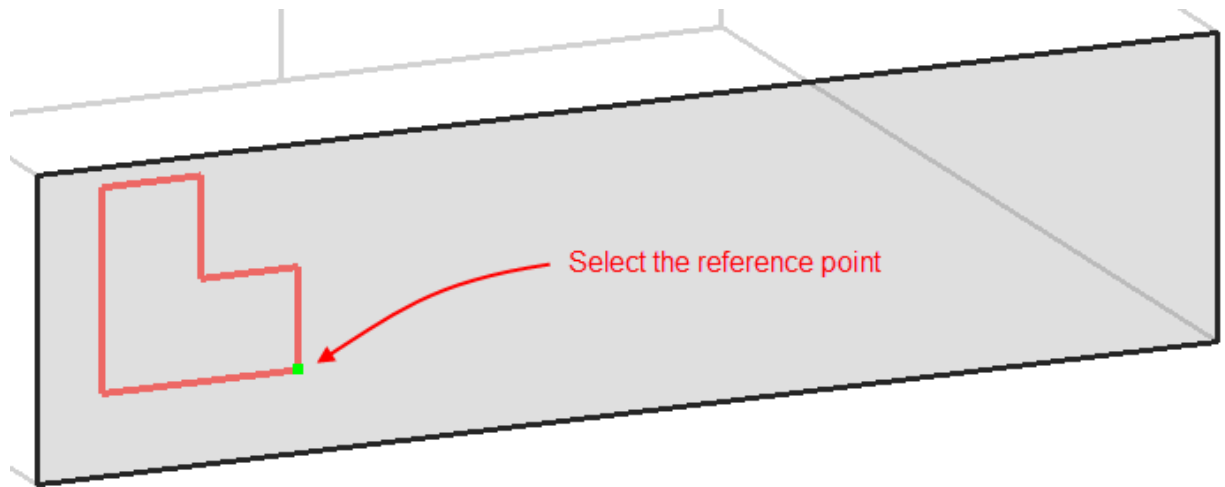
A similar process is used to create mirrored copies of a selected set of openings either at building or surface level. The instructions below are for working at surface level, but the process is the same when working at building level.

1. Go to the surface containing the openings to be copied and mirrored.
2. Select the openings.
3. Click on the **Mirror** toolbar icon.
4. In the Drawing Options panel select the Reflection axis. Choose from x or y-axis as the mirror plane.

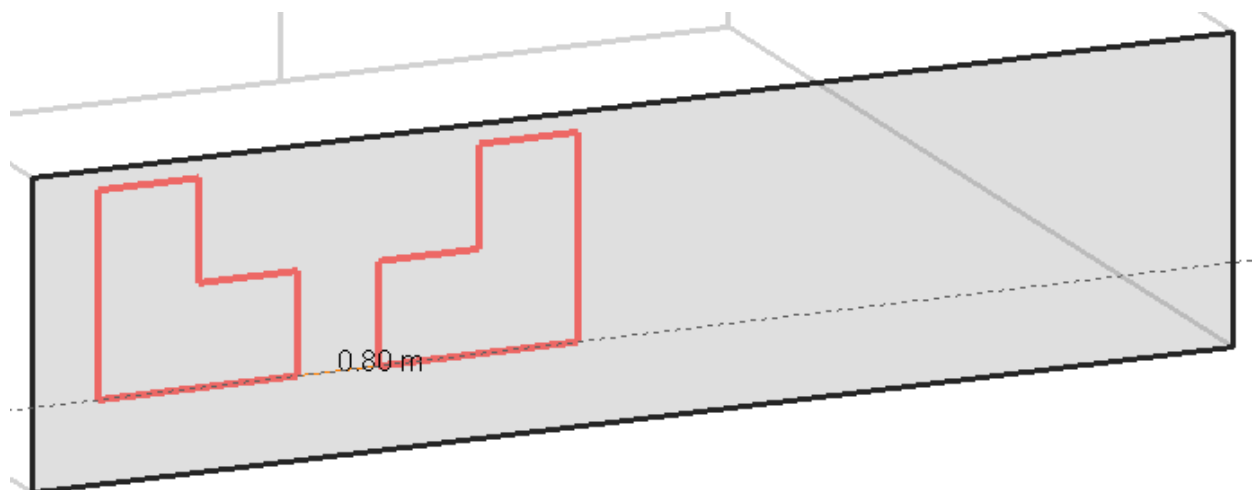


For surfaces, choosing to mirror in the x-axis means to mirror in a line in the horizontal direction and the y-axis means that the mirror line is in the vertical direction.

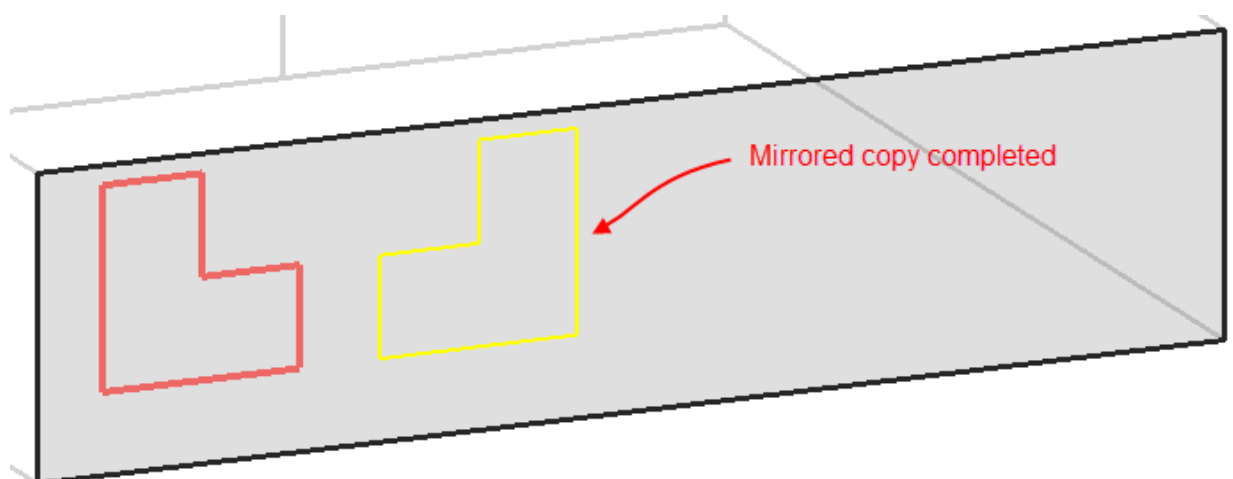
5. Select the reference point on the existing set of selected openings by clicking with the mouse. It is usually convenient to choose an end point snap point whose corresponding mirrored point can be used to accurately snap the new mirrored copy in place.



6. Once you have selected the reference point the mirrored copy is created and can be positioned as required, usually by snapping the mirrored reference point to the required place. You can key in an exact distance to move the reference point while creating the mirrored copy.



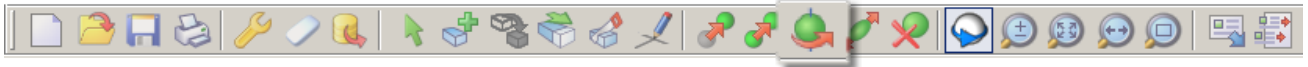
7. Once the mirrored copy is in place, click with the mouse to actually create the new opening(s).



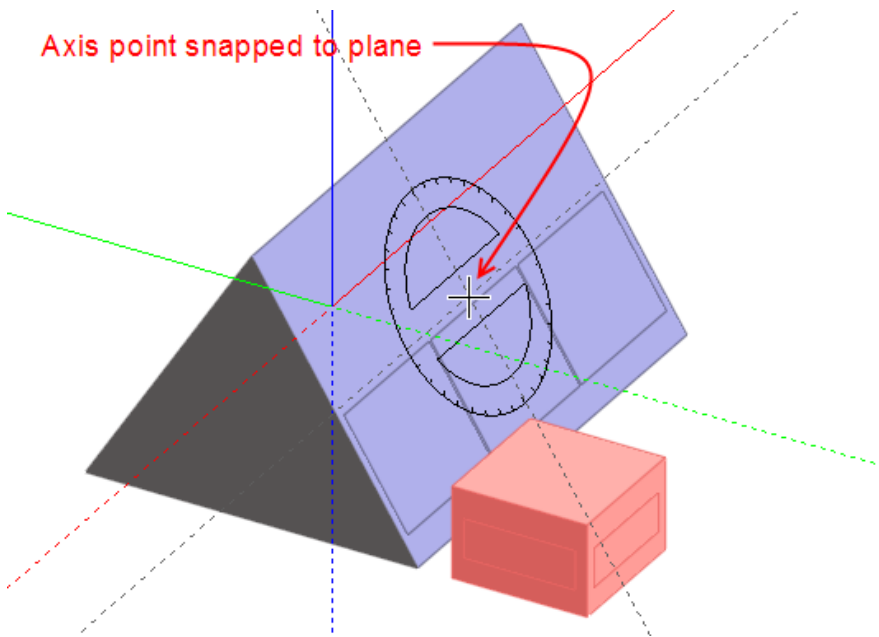
Mirror Partitions And Courtyards

The process of mirroring partitions and courtyards is carried out at block level and is similar to the processes described above for blocks and openings.

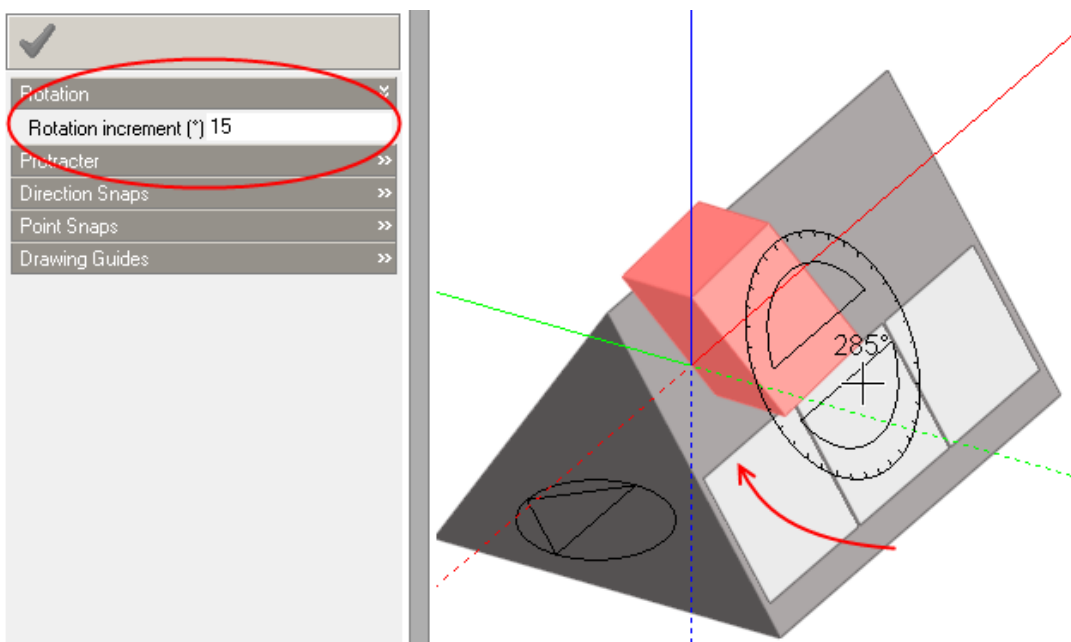
Rotate



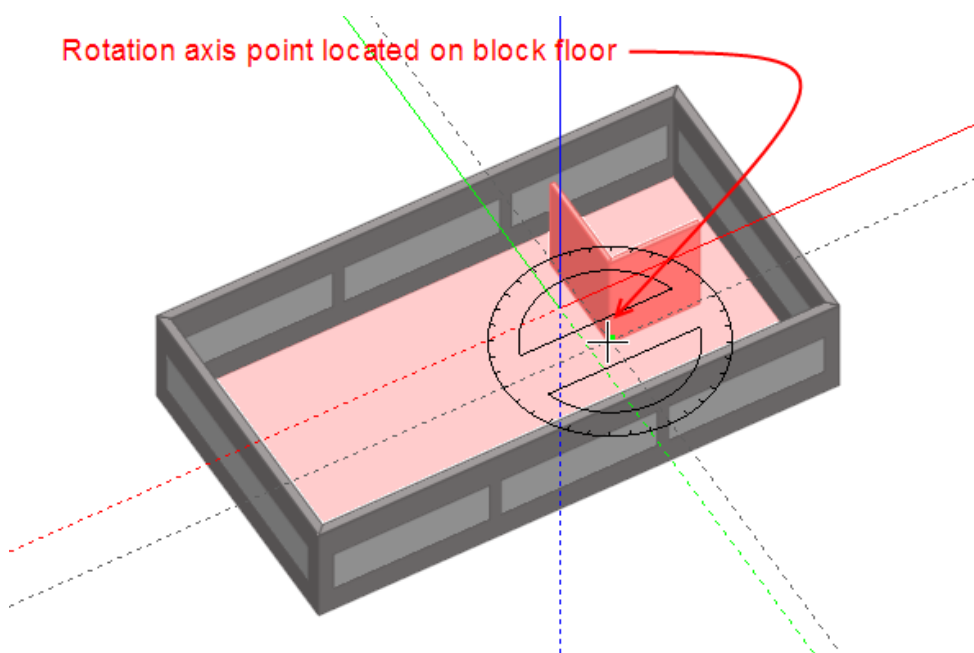
You can rotate blocks, partitions, courtyards and openings. To rotate an object, first select it and then click on the 'Rotate' tool or select 'Rotate' from the right-click menu. If you want, you can rotate several objects at the same time by doing a [multiple selection](#). As you move the cursor back into the Edit screen, the Protractor will be displayed at the cursor position. At the building level, you can rotate a block around an axis point defined on any plane and as you move the cursor from surface to surface in the model, each surface will highlight in blue to show that it's the active drawing plane and the protractor will automatically rotate into the same plane:



You define the rotation axis about which the object will be rotated by clicking the mouse when the mouse cursor is in the required location, you can use the various [snaps](#) and [drawing guides](#) to locate the axis point. After placing the axis point, you can then rotate the object around the axis:



In the case of partitions and courtyards at the block level, the active drawing plane for rotation is always the plane of the block floor and in the case of surface openings, the active drawing plane is always the surface plane:

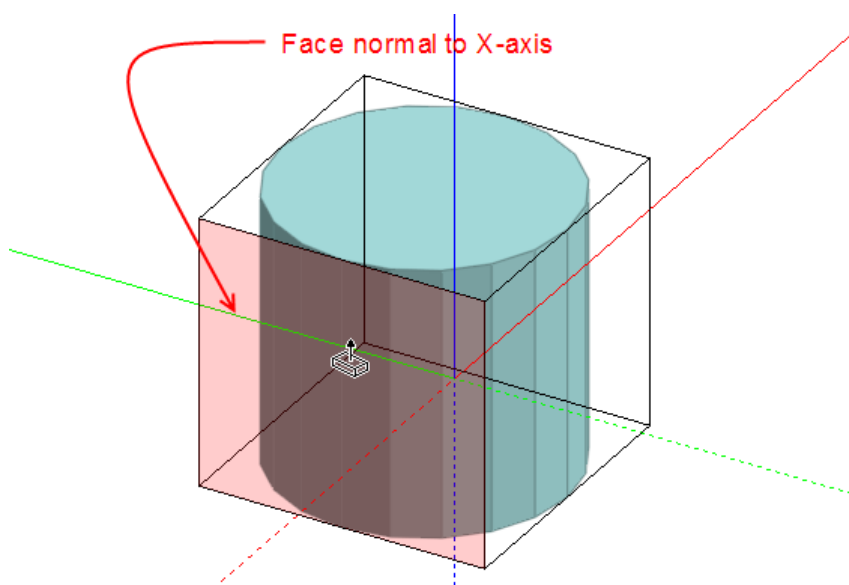


See also the [Stretch, drag face and rotate tools](#) tutorial.

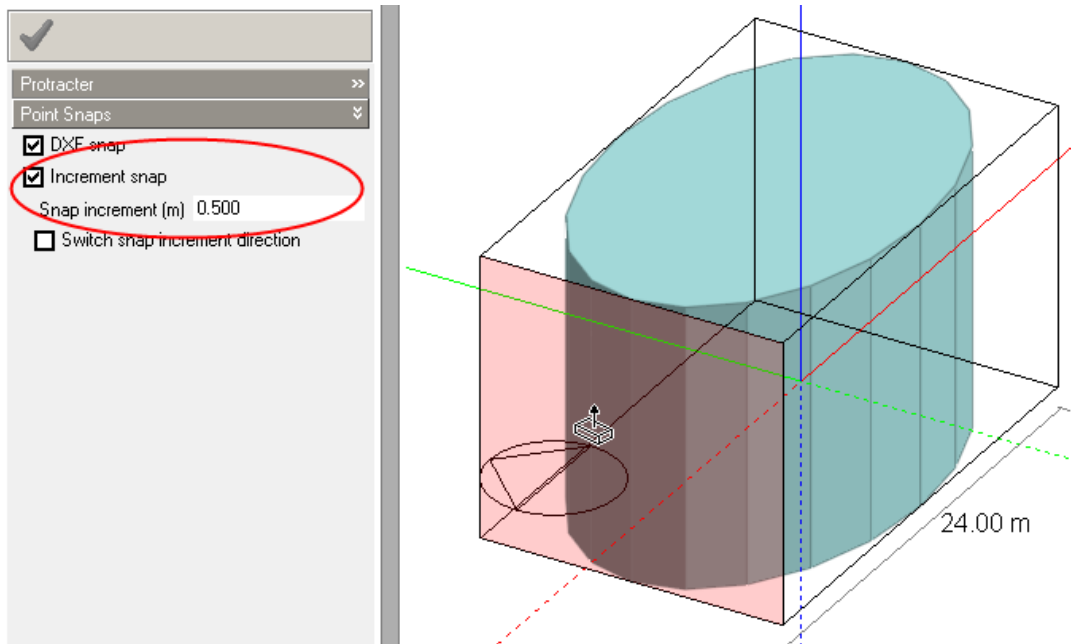
Stretch



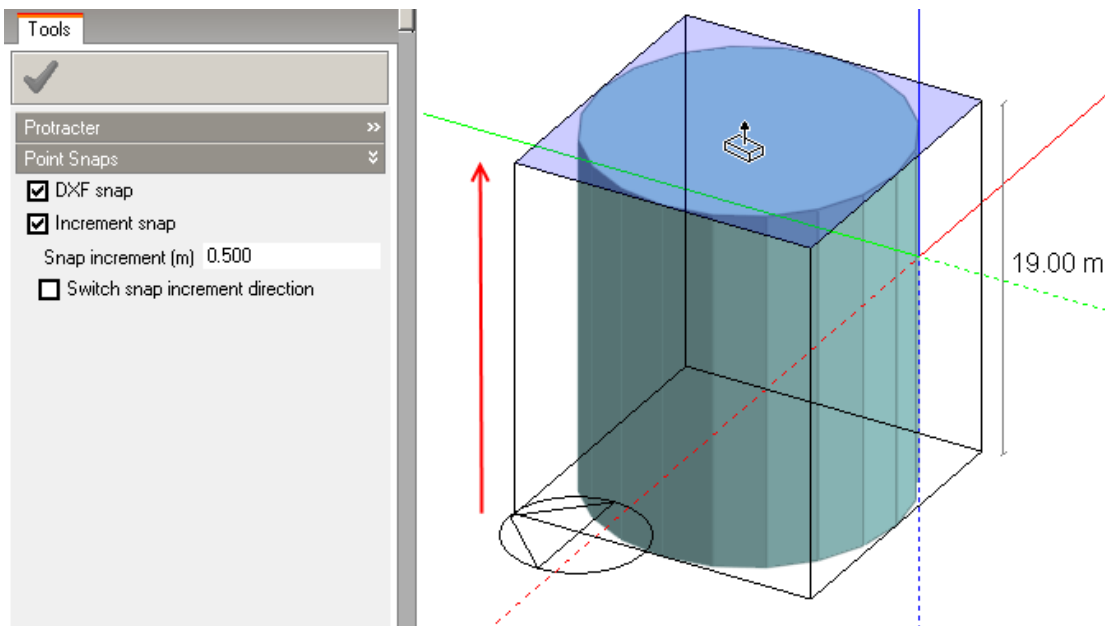
Building, component and outline blocks can be stretched (scaled) when at the building level. To stretch a block, go to building level then select it and click on the **Stretch** toolbar icon, or select 'Stretch' from the right-click menu. When you activate the Stretch tool, a wire frame is displayed around the extent of the block and as you move the cursor across the faces of this frame, each face will highlight in the colour of the axis to which the face is normal:



If you then click on one of these highlighted faces, you can then stretch the block along the axis normal to the face. If the increment snap is switched on, the block can be stretched in increments:



In the case of the Z-axis, the stretch operation will only operate along the positive Z-axis:



Tip: By default the stretch will take place in the direction defined by the face that you select. However to stretch in both directions equally, hold the <Shift> key down while dragging.

Tip: You can also key in the *new dimension* using the keyboard. Note the difference to the [Drag face](#) tool where the *increment* in the extrusion is keyed in.

Note: You can only stretch one block at a time.

See also the [Stretch, drag face and rotate tools](#) tutorial.

Delete



To delete an object, first select the object you want to delete and then click on the **Delete** tool or select 'Delete' from the right-click menu. If you want, you can delete several objects at the same time through [multiple selection](#).

Tip: It is usually quicker to simply press the <Delete> key on the keyboard.

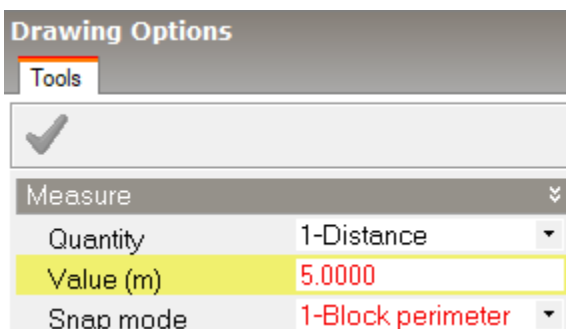
See also the [Selecting, moving, cloning and deleting objects](#) tutorial.

Measure

The measure tool allows you measure distance, angles and areas *on single planes* in the model. To use the tool, navigate to the part of the model you wish to measure, click on the Measure toolbar icon, select the type of measurement to be made in the Display options panel on the left of the screen. The options are:

- **1-Distance** - Click twice on a plane to define the length between the 2 points.
- **2-Angle** - Click three times on a plane to define the area between the 2 lines.
- **3-Distance** - Draw an enclosed polygon by clicking with the mouse on a single plane.
- **4-Perimeter** - Draw an enclosed polygon by clicking with the mouse on a single plane.

The requested data is displayed highlighted in the Drawing Options panel.



Note: When measuring at block level it is possible to get a fuller range of snaps by selecting the **2-Zone inner surfaces** Snap mode.

See also the [Adding blocks and using the measure tool](#) tutorial.

Importing Geometric Data

There are 2 ways to import geometric data into DesignBuilder:

- [Importing floor plans](#) where 2-D floor plan data is imported either from CAD or scanned paper drawings.
- [Import 3-D BIM](#) data using the gbXML data format.

See also:

- "Merge Model Data from File" on page 179 to import the attribute settings, components and templates, HVAC system, assemblies and component blocks from a previously created DesignBuilder model into a newly imported BIM model.

More information on Importing model geometry can be found in the [Importing drawing files](#) tutorial

Import 2-D Drawing

DXF and other bitmap-type drawing data formats can be imported in the form of 2-D floor plan or elevation layout drawings. These can be used to trace block perimeters, partitions and façades as a fast way to enter the model geometry. A typical process is as follows:

1. Create new dsb site file, select location etc.
2. Import floor plan data (described below).
3. Go to block level and draw partitions by tracing over the drawing data.
4. Import other DXF floor plans for other stories if necessary.

The process of importing elevation drawings for drawing opening often takes place after the main block geometry definition and partitioning has been completed as follows:

1. Import elevation DXF data to the ground plane (you cannot import 2-D drawing data directly to any other plane).
2. Move the DXF object to a wall surface by selecting it and using the [Move](#) command.
3. Go to surface level and trace windows, doors and any other openings by snapping to the DXF data.

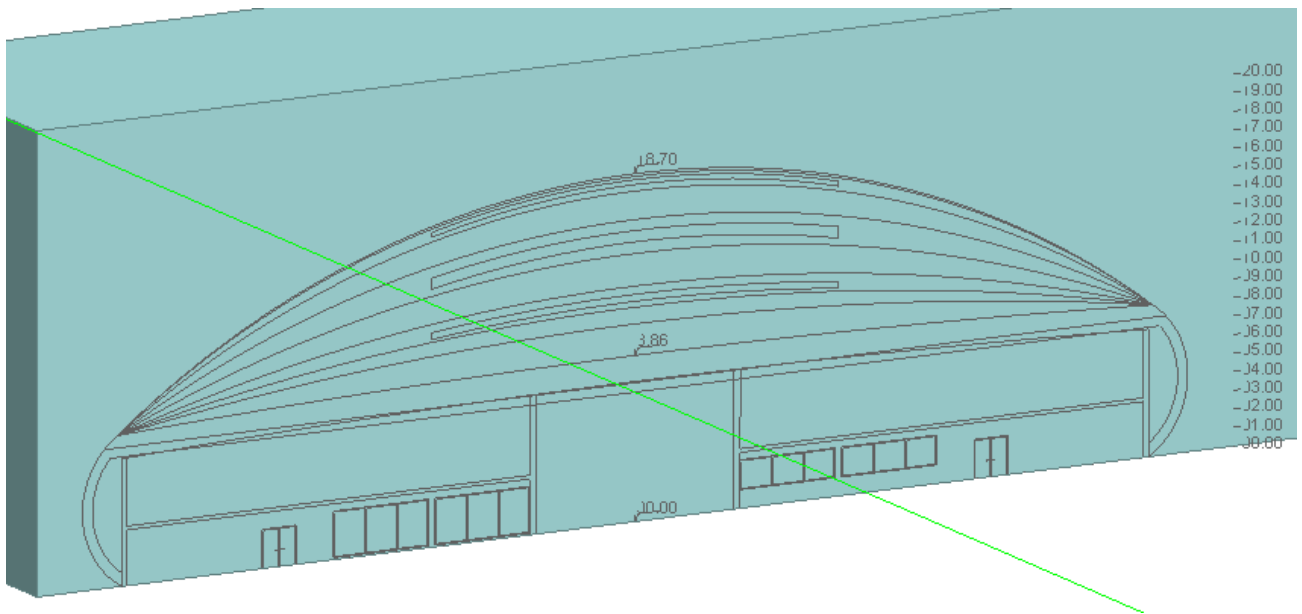
It is also possible to use elevation drawings to create the original block form as described below.

Note: It is necessary to switch on the [Show imported 2-D drawing at zone and surface levels](#) Display model option on when using an imported elevation to show the position of windows, doors etc at surface level.

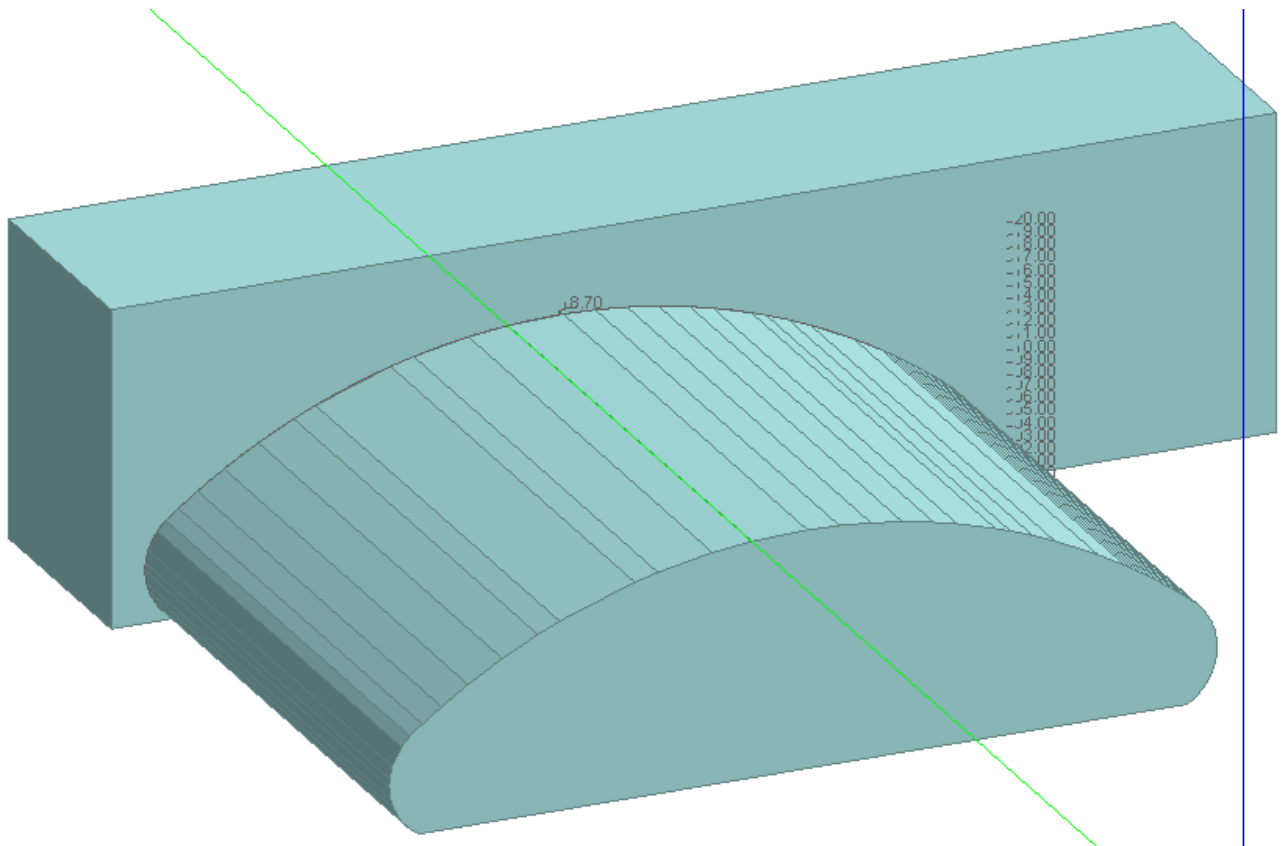
Tip: When a 2-D drawing is loaded, sometimes the drawing data can come between the mouse cursor and objects to be accessed. In this case, pressing the <Shift> key while left mouse clicking allows the drawing to become "transparent" to any clicks, making it easier to access model objects such as blocks and openings, which are behind the drawing data.

Using an Elevation Drawing to Create a Block with Openings

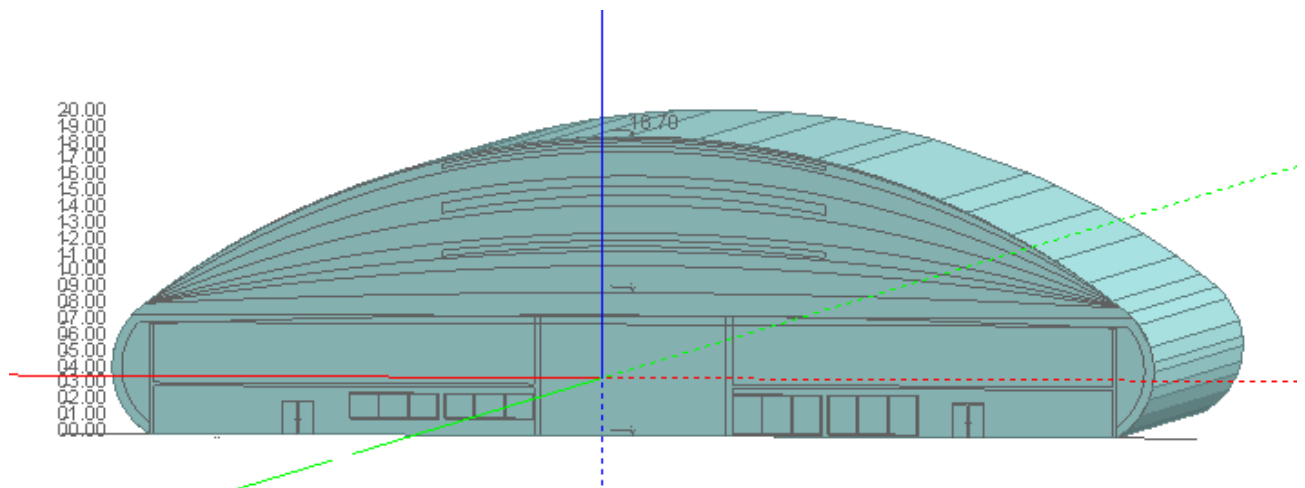
Elevation drawings can also be used for creating horizontally extruded blocks of complex section such as the aircraft hanger elevation drawing below, placed on an outline block ready to create a horizontally extruded block shape.



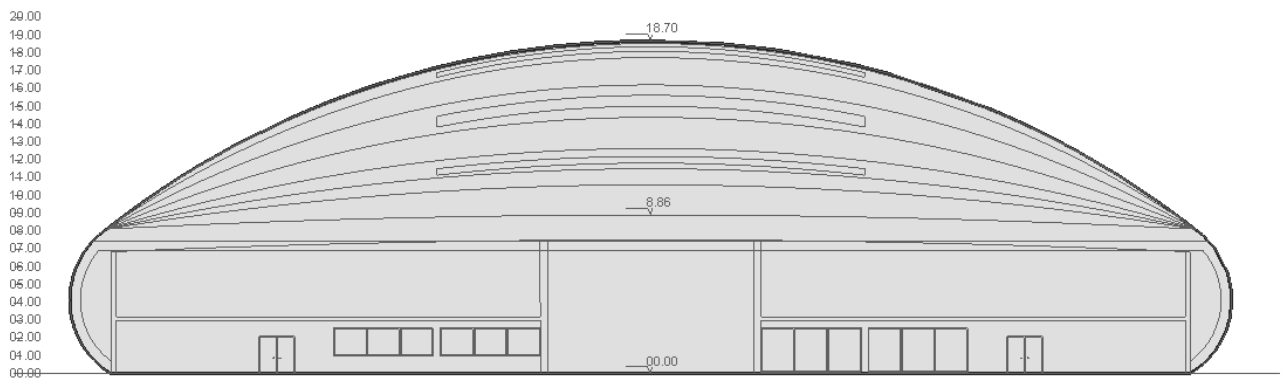
Using the [Add surface](#) tool, trace the outline of the hanger external façade:



The Cutting tool can be used to remove the original rectangular part of the outline block. The image below shows the outline block from the other side with the rectangular part of the outline block removed so we can see the drawing coplanar with the block facade:

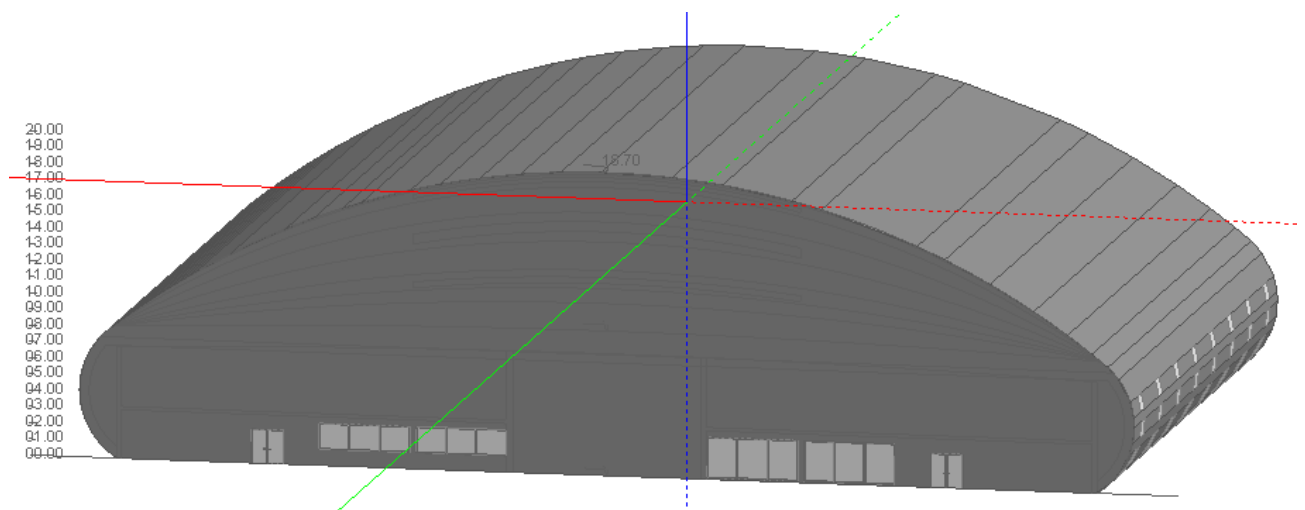


The hanger outline block has now been converted to a building block and once navigated to surface level we are ready to use the drawing to help trace the windows and doors onto the surface:



Note: Make sure that the correct surface is selected when navigating to surface level

Completed hanger block with windows and doors drawn at building level:



More information on Importing floor plans can be found in the [Importing drawing files](#) tutorial

Import Drawing File Wizard

You can import 2-D DXF floor plan files created in AutoCAD or other CAD programs or the other bitmap-based drawing formats listed below. To start the process click on the **Files > Import > Import 2-D drawing file** menu command. This starts a the Import Drawing File Wizard.

With the import wizard open, select the File type to import. You can select from:

- **1-DXF** - 2-D CAD data. This option allows you to snap to the end-points of the DXF line segments, the other bitmap-based formats below do not allow snapping.
- **2-PDF** - PDF drawing file (single page). **Note only PDF files with a single page will import successfully into DesignBuilder.**
- **3-BMP** - Bitmap drawing file.
- **4-JPG** - JPEG drawing file.
- **6-GIF** - GIF drawing file.
- **7-TIFF** - TIFF drawing file.

When you have selected the file type then select the file by clicking into the 'Filename' control. Click on the ellipsis button to open the file browser dialogue box.

Import Drawing File

Imported drawing file

Select the floor-plan file to import

DXF file		<<
File type	1-DXF	▼
Filename	<Select file>	...
Scale and Position		<<
Units	1-Metres	▼
Attach at height (m)	0.000	

Cancel Help Back Next Finish

Once the floor plan has been imported you can add blocks by tracing over the [external perimeter](#) corners of the floor plan.

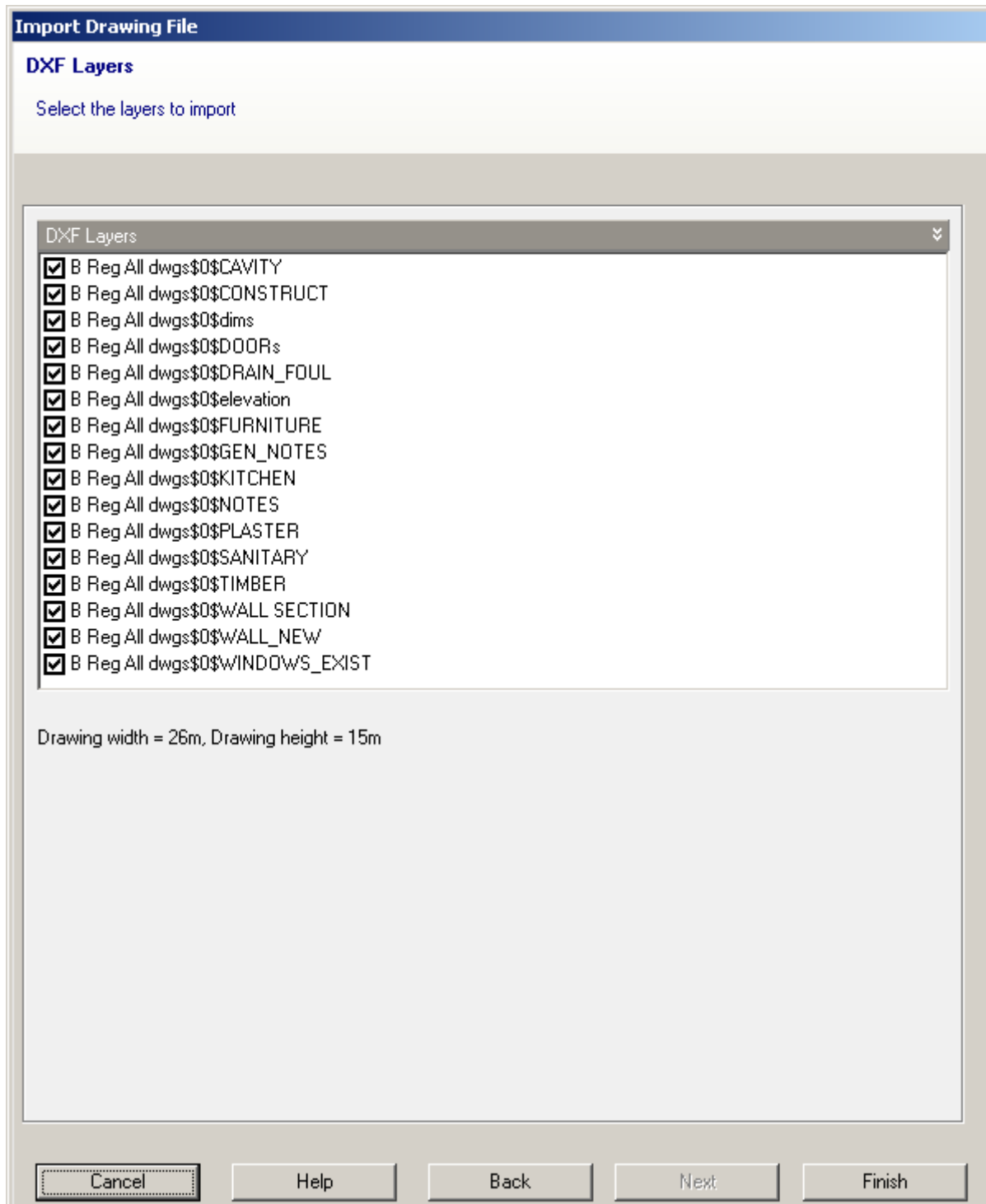
DXF File Types

While on the Import drawing file wizard, if you selected a DXF file you must then select the units used in the DXF file from the **Units** drop list. You should be able to find this out from the creator of the CAD model - you are likely to get 'out of range' errors if you get it wrong. For example the CAD drawing may have been defined in millimetres and if you have selected metres as the units, when DesignBuilder tries to read the floor plan everything is 1000x too big and meters become kilometres! The DesignBuilder modeller has a range of several hundred metres and if any data falls outside this, the 'out of range' error occurs.

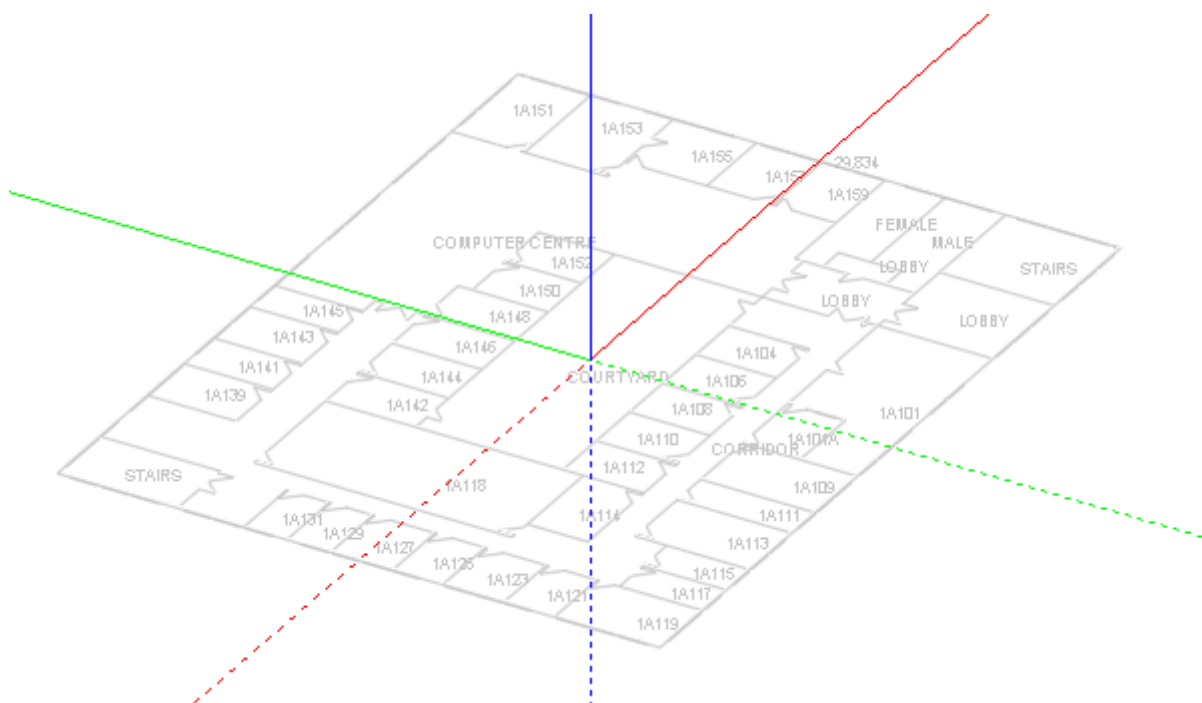
DesignBuilder supports ASCII DXF, and not binary DXF format. You can convert binary files to ASCII using a utility such as [DDS-CAD Viewer](#).

Tip: DesignBuilder is not able to import DXF files that are already open in other applications, so if you have been checking the DXF file that you plan to import in AutoCAD or other CAD tool, you should make sure to close the file before attempting to import it into DesignBuilder.

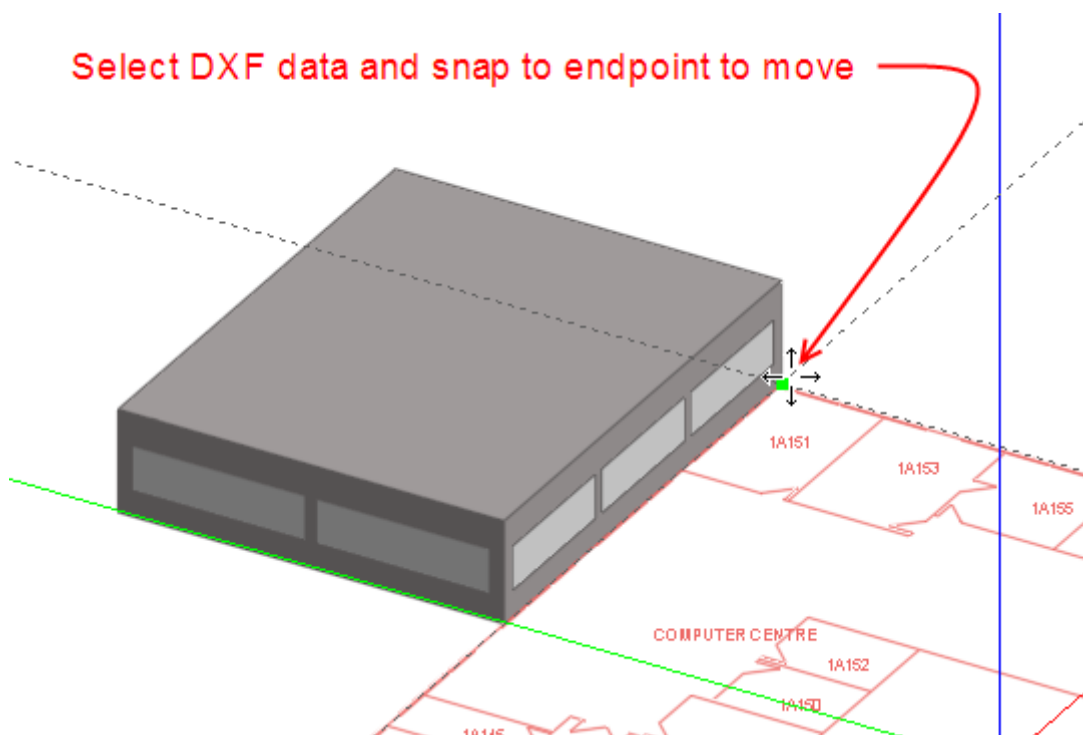
You must also set the attachment height at which the 2-D DXF floor plan will be attached in the DesignBuilder model. The DXF data can be moved around the model once it is imported, so you do not need to know the exact attachment height at this stage - you can import the DXF data at zero height and move it to the correct location using the [Move](#) command. Click on the 'Next' button to set up layer visibility:



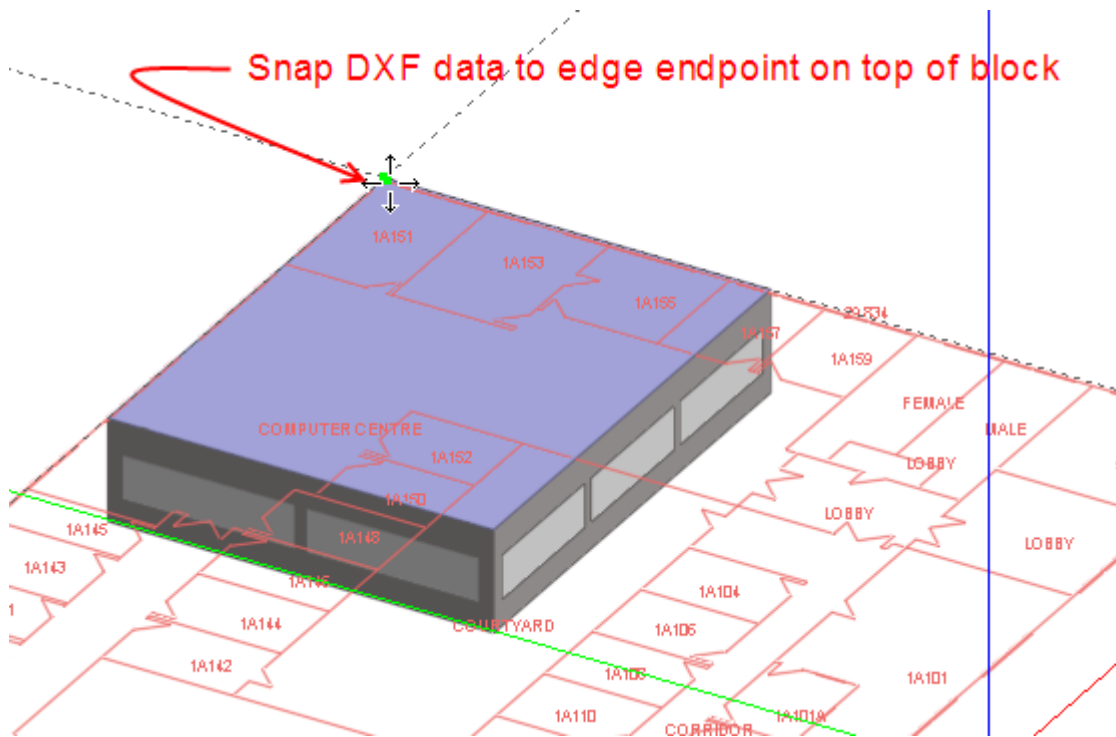
You can use these controls to switch off unnecessary layers in the DXF data. As you check/uncheck the checkboxes you will see the effect of the change in the Edit view under the Wizard. Click on the 'Finish' button to import the data:



At the **building level**, you can [select](#), [move](#) and [delete](#) DXF data. When you select DXF data, you may need to zoom in to make the selection. You can move DXF file in 3-D and snap the data to the edges of blocks or end-points of [construction lines](#):



Move the DXF data on top of the existing block and snap to an edge end-point:



When you have imported the DXF data you can use the Measure tool to check that the dimensions are correct.

Import DXF floor plans for other stories if necessary. Note that only one set of DXF data can be imported at a time. Once the CAD data has been imported into the model, the source DXF file is no longer needed by DesignBuilder and can be moved or deleted.

Editing DXF Data

Once the DXF data has been imported to the model, it must be selected to allow these operations:

- [Move](#) - move the DXF data
- [Delete](#) - delete the DXF data (not the source file itself).

Also the [Set X-axis for imported DXF](#) and [Scale DXF drawing](#) menu commands are available from the [Edit menu](#) when DXF data is loaded.

Set X-axis for imported DXF

When the command is started click twice to define the end points of a line in the direction of the X-axis on the imported DXF data. After the operation the DXF data is rotated to align with the DesignBuilder X-axis. Normally DXF data will already be orientated to the grid and this command is not required.

Scale DXF drawing

This command allows you to scale the DXF data to be larger or smaller. This may be necessary if the scale of the DXF data was unknown at the import stage. When the command is started click twice to define the end points of a reference line on the DXF data of known length (e.g. a dimension arrow or known length on a building wall). Then either move the cursor to define the new length of the reference line and click again or type in the dimension of the reference line. So a typical sequence might be:

1. Start [Edit > Scale DXF drawing](#) command.
2. Single click on one end of a large dimension line.
3. Single click on the other end of the dimension line.
4. Type in the dimension in m or ft (depending on the international [units](#) setting) followed by <Enter>.

Bitmap-based File Types

When the file type is one of the bitmap based formats (PDF, BMP, JPG, PNG, GIF, TIFF collectively referred to here as 'bitmap') the next page of the Wizard provides a preview of the drawing and some options.

Invert bitmap

Check this option if you wish the colours in the bitmap to be inverted. For example if the picture is white on a black background then it will generally be best to invert these for use in DesignBuilder.

Auto scale bitmap after loading drawing

If you wish the 'scale bitmap' command to be automatically started when the Import Drawing File Wizard is closed then check this option.

Quality slider

By reducing the number of pixels in the bitmap it is possible to significantly reduce the size of the imported drawing data and hence speed subsequent drawing 'scale' and 'rotate' operations. The default Quality setting of 0.5 gives a good trade-off between image size and readability but if the drawing file is particularly large you may decide to reduce the quality to reduce the memory overhead or, if the drawing is difficult to read you could increase Quality.

Opacity slider

You can control the opacity (transparency) of the drawing data from 0 to 1. A value of 1 means that the imported drawing will be completely opaque and 0 means it will be completely transparent. The default is 0.5.

Import Drawing File

Preview

C:\Users\darrenf1\Downloads\14775-311C1 - Block B First Floor (A1).pdf



Bitmap-based files may be used for:

1. Existing buildings having paper floor plans that have been scanned into electronic format or,
2. Early stage design sketch ideas are available only in bitmap form.

Note: bitmap drawings do not allow end-point snap like DXF but can be an equally effective option for rapid geometry entry in DesignBuilder.

Editing Bitmap Data

Once the floor plan data has been imported to the model, it must be selected to allow these operations:

- [Move](#) - move the bitmap data.
- [Delete](#) - delete the bitmap data.

Also the **Set X-axis for imported bitmap** and **Scale bitmap drawing** commands are available from the [Edit menu](#) when bitmap data is loaded.

Set X-axis for imported bitmap

Sometimes scanned bitmap files are not exactly aligned to the X-axis and you can use this command to re-orientate the drawing. When the command is started click twice to define the end points of a line in the direction of the X-axis on the imported bitmap data. After the operation the bitmap data is rotated to align with the DesignBuilder X-axis.

Scale bitmap drawing

This command allows you to scale the bitmap data to the correct size using a known dimension in the bitmap image.

When the command is started click twice to define the end points of a reference line on the bitmap data of known length (e.g. a dimension arrow or known length on a building wall). Then either move the cursor to define the new length of the reference line and click again or type in the dimension of the reference line. So a typical sequence might be:

1. Start **Edit > Scale bitmap drawing** command.
2. Single click on one end of a large dimension line.
3. Single click on the other end of the dimension line.
4. Type in the dimension in m or ft (depending on the international [units](#) setting) followed by <Enter>.

Important Note: This process will always be necessary for bitmap drawing formats as the scale is not defined at the import stage.

Partitions

You can also use the imported floor plan data to draw internal partitions. To do this go to block level and draw partitions by tracing over using the floor plan partition data. You should take care to connect partitions to the external perimeters. A common mistake is to snap partitions to DXF data lying very close to the external perimeter but not close enough to create a connection with the external perimeter.

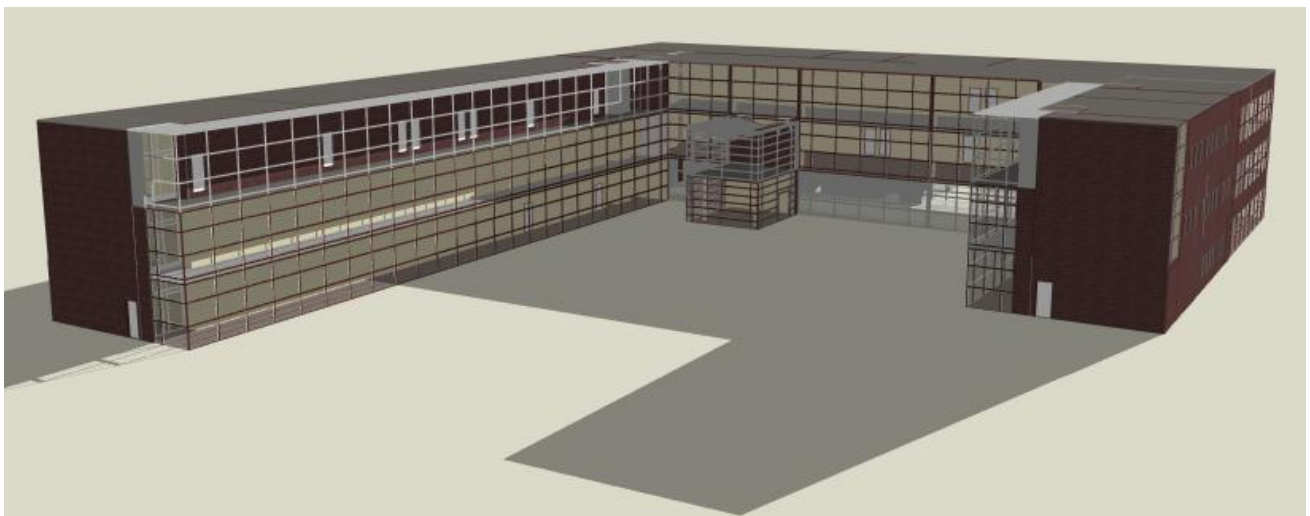
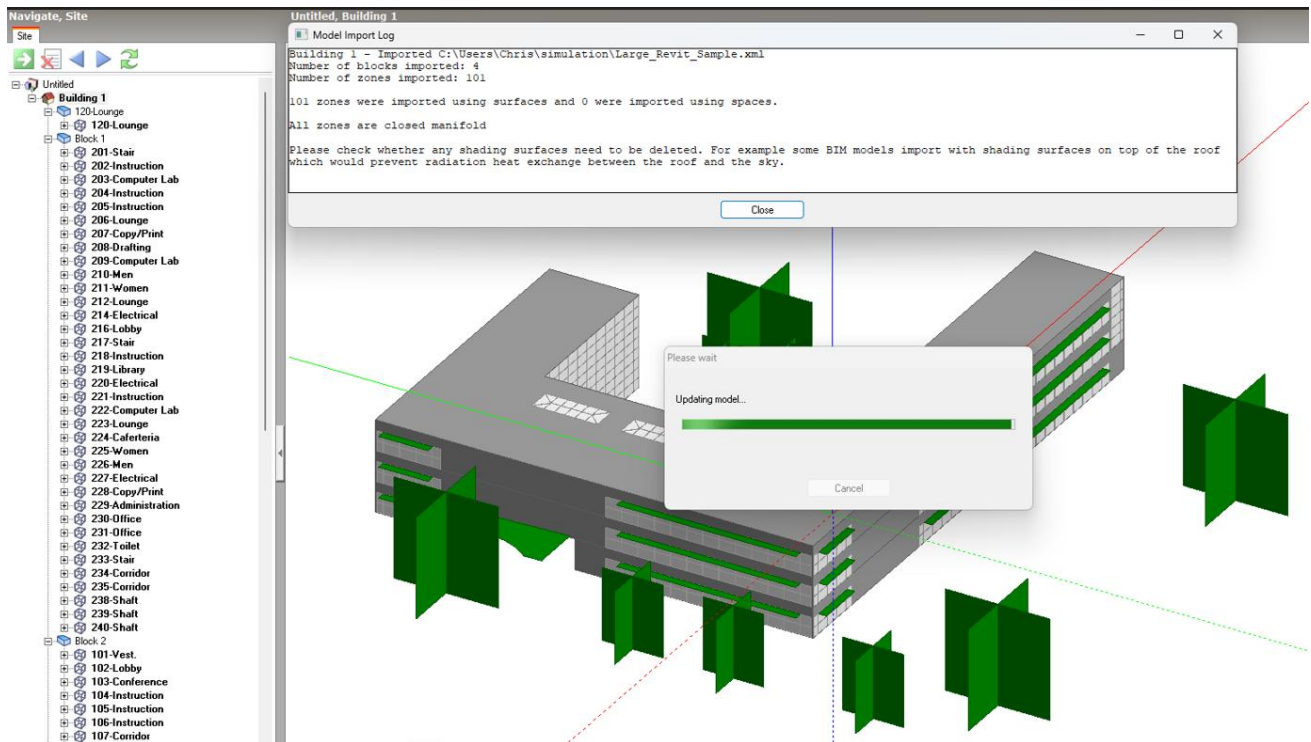
Tip: It is generally best to switch off **DXF snap** in the Drawing options panel when drawing partitions at block level to avoid this problem. Be sure to switch it back on again when you have finished drawing partitions.

Import 3-D Model

DesignBuilder provides interoperability with other software tools through its [gbXML](#) and [IDF](#) import capability. This allows you to import 3-D architectural models created in Revit, Grasshopper, ArchiCAD, Microstation and other BIM systems supporting gbXML data exchange. You can also import EnergyPlus IDF files which can be exported from a number of different energy modelling tools.

Models can be imported from the site or building level using the **Import 3-D models** toolbar icon or use the **File > Import > Import 3-D model** menu command. This opens the **Import 3-D Model** dialog. If you are at site level DesignBuilder will prompt you to create a building first to import the model into.

Unlike the [2-D drawing import](#) which provides only a set of lines for tracing over, importing 3-D models imports the full geometry as blocks with zones, windows, doors and shading surfaces all included. Other construction, glazing and material data can optionally be loaded if it exists in the source model.



Example model imported from Revit into DesignBuilder



Example model imported from ArchiCAD into DesignBuilder

The **Import BIM Model** wizard has two pages which are explained below.

Import 3-D Model

Most of the import controls are the same for gbXML and IDF import.

On the first page you can select the XML or IDF file to import. The file must be in the gbXML or IDF standard data format. Once you have selected the model, press the **Next** button to move to the second page. For large models there will be a delay at this point while the model is loaded to the dialog.

The second page shows a preview of the model along with some controls to control the way the import is to be carried out.

Import Options

View

You can display the preview of the model from various viewpoints. Select from:

- **1-Plan,**
- **2-Left,**
- **3-Right,**
- **4-Back,**

- **5-Front**,
- **6-Axonometric** (3-D view).

Import mode

This option is only available for gbXML imports. Choose whether the importer will base block and zone geometry on the space entities in the source data or on the primary surface elements. The options are:

- **1-Surface** - where the surfaces in the source gbXML data are used to define block geometry. This is the default option as it typically gives the best results for the majority of BIM models imported from Revit and most other BIM applications.
- **2-Space** - where the block geometry is based on the spaces in the source data. This option can give rise to gaps between zones in the DesignBuilder model when (as is usually the case) the spaces do not touch neighbouring adjacent spaces due to the finite thickness of partition walls in the BIM model.

Adjacency tolerance

This is the maximum gap between neighbouring parallel gbXML surfaces that will be interpreted as being an adjacency between 2 spaces. You might enter a value higher than the default 0.01m if the gbXML spaces are separated by large gaps due to thick partitions or ceiling /floor voids being excluded from the gbXML. For example if the largest gap in the between spaces that are to be modelled as adjacent is 0.3m then enter 0.3m here.

The **Adjacency tolerance** can also be updated after the import from the [Advanced tab](#) of the Model options dialog.

Create storey blocks

This option is used to automatically identify stories by grouping adjacent spaces created by extruding from the floor plan with the same upper and lower Z coordinates into single blocks. This has the advantage of making it much easier to later modify internal layouts and it also helps to navigate around large models.

Note: For imported spaces to be grouped into storey blocks, they must all have the same height. To ensure that this is the case, DesignBuilder checks that all non-horizontal surfaces are vertical and all z-coordinates have either the maximum or minimum z-coordinates. Any zones with different ceiling heights will be imported as independent blocks.

Tip: It can sometimes help, especially in lower quality data sets, to import raw surface geometry exactly as defined in the source model. In this case the **Create storey blocks** setting should be switched off and the [Allow open manifold building blocks](#) setting should be switched on.

Import as building blocks

By default this option is selected and the geometric space data will be imported as building blocks ready for use in DesignBuilder modelling. However, you can instead import zone geometry as outline blocks by unchecking this option. The outline blocks give an exact representation of the spaces in the source data and this can be useful for diagnosing any possible problems with geometry. For example if any part of a space intersects with another space then it would not be created as a building block, but it could still be created as an outline block to help you identify and fix any possible issues with intersections in the source model.

Import thermal properties

If this option is selected then DesignBuilder will any load materials, constructions and glazing system thermal properties present in the source data and assign at surfaces and opening level in the DesignBuilder model. In this case, because settings are made at surface and opening levels, the normal inheritance paths used for construction and glazing data will be overridden.

For materials where material thickness is either not specified or is set to 0 a default thickness of 0.05m is applied.

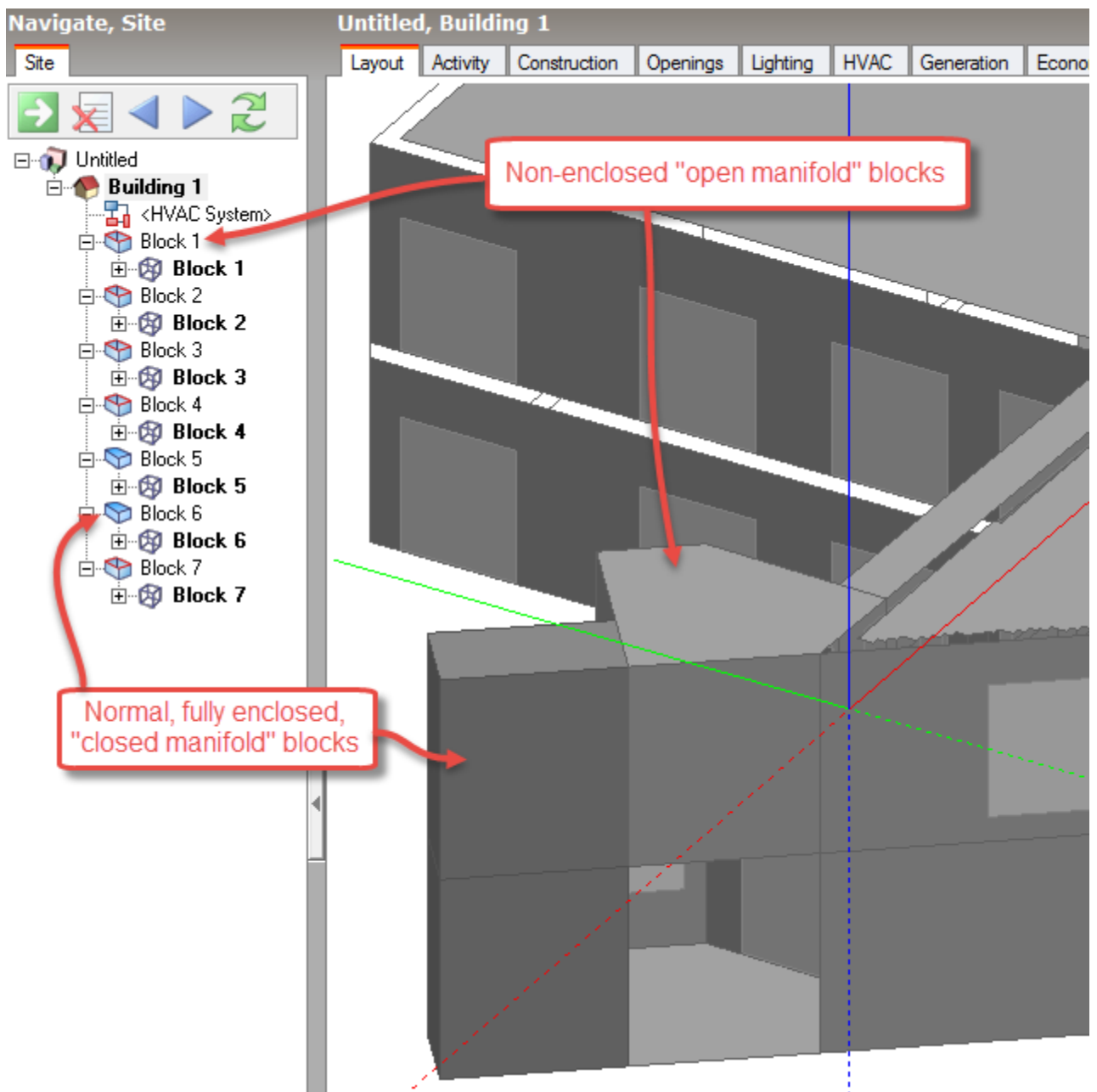
Tip: Since any construction or glazing assignments are set in the DesignBuilder model at surface level, if you need to parametrically adjust construction or glazing surface properties in optimisation, parametric analysis or UA/SA analyses (i.e. they are [design variables](#)), you may need to apply the changes to each surface individually by selecting the appropriate surfaces as [targets](#).

Allow open manifold building blocks

Open manifold (non-enclosed) blocks are a special type of block which can only be created when models are imported from external sources. They have one or more surfaces missing which means that they are not fully enclosed and this has significant consequences on what can be done with the block in the DesignBuilder modeller and in calculations:

- Radiance and CFD calculations cannot be carried out.
- The boolean operations used behind the scenes to manipulate block geometry are not possible and so only limited edit operations may be performed on open manifold blocks. In particular these operations cannot be performed:
 - o [Add surface](#)
 - o [Drag block face](#)
 - o [Mirror](#)
 - o [Divide block using cutting tool](#)
 - o [Boolean geometry operations](#).

Tip: Open manifold blocks are shown in a different shade of grey from normal closed manifold blocks to differentiate them and their icons in the Navigator have a red outline. See screenshot below.



Because Open manifold blocks cannot use all geometry operations, the orientation of their surfaces cannot be reliably determined from the model and DesignBuilder must rely on the orientation data stored with the surface from the source model. This data is not always reliable and so in some cases surfaces are facing in the wrong orientation. The [Flip block surface orientation](#) command can and should be used to check and, if necessary fix, the orientation of surfaces in all open manifold blocks.

Import shading plane surfaces

Shading plane surfaces are specially marked in imported data and so can be treated separately if required. You can choose to import shading surfaces as planes as part of the import or not using this control. Shading surfaces imported into a model will be treated as 2-D flat shading plane surfaces.

Shading Plane EnergyPlus Technical Note

Shading planes are written to EnergyPlus IDF as Shading:Building:Detailed objects since they are considered to be part of the building rather than the site.

Note: Shading plane surfaces do not have any impact on external CFD analyses.

Tip: You can choose whether or not shading planes will be applied to the model using the [Shade Surfaces](#) model option.

Merge co-planar surfaces

If the above Import shading surfaces option is selected then you can choose to either merge any such co-planar surfaces or not.

More information on Importing 3-D models can be found in the [Importing drawing files](#) tutorial

How Does DesignBuilder IDF Import Work?

DesignBuilder [IDF Import](#) is based on a 2 stage process:

1. Convert IDF source data into gbXML.
2. Load the gbXML through the same internal process used for loading gbXML files. More information on that can be found below.

How Does DesignBuilder GbXML Import Work?

The geometric information contained in gbXML includes a collection of surface and space objects which represent the outer and inner volumes of rooms in the parent BIM model respectively. The surface objects are derived from corresponding wall, ceiling, floor etc elements in the BIM and also contain openings for windows and doors. The space objects aren't native to the BIM system but are identified by users through a manual or semi-automatic process. The result of the spatial identification process is a collection of spaces that are separated by gaps that would be filled in the parent BIM model by the BIM primitives such as partitions, slabs, etc.

Internally DesignBuilder includes 2 different ways to import gbXML data:

- **Surfaces** where the gbXML surfaces are used to define the block geometry. In well constructed gbXML data sources this data is used to define the outer block geometry. It avoids the need to see gaps between the blocks caused by partition and internal floor thickness. However if surfaces are missing you may end up with surfaces also missing in the DesignBuilder model leading to open manifold blocks which can't be simulated in daylighting and CFD.
- **Spaces** where surface data is not available the spaces are used as the source of the block geometry.

The DesignBuilder gbXML import works by simply mapping the gbXML space surface geometry to building blocks, one block per zone.

The way to think about the relationship between gbXML "spaces" and "surfaces" is that surfaces define the interlocking zones that make up the whole building model, and the spaces are the inner volume spaces representing the contained air volumes within each zone. So, each internal surface between adjacent zones would generally represent the centre-line between the zones, and the associated record in the gbXML file would reference both zones in the adjacent space field.

Depending on the method used to identify the block geometry they represent:

- In the normal case where DB was able to use surfaces to define the block, the **External measurements Geometry convention template** is applied to give realistic thickness for block surfaces based on the overall thickness of the surface constructions defined in the gbXML if this option was selected.
- Alternatively if DB wasn't able to find enough surface data to define the block but full space data is available then it uses the space *inner* geometry and applies the **Simple Geometry convention template** to give zero thickness for all block surfaces. In this case the building will be made up of a set of blocks

defined based on the gbXML inner spaces, possibly with gaps between them due to the partitions and internal floors.

Note: It is important to understand that in both cases the success of the BIM to DesignBuilder data transfer process depends on how well the spatial identification process has been conducted in the source model. If there are intersecting or missing spaces in a BIM model for example then the model imported into DesignBuilder will generally have corresponding problems.

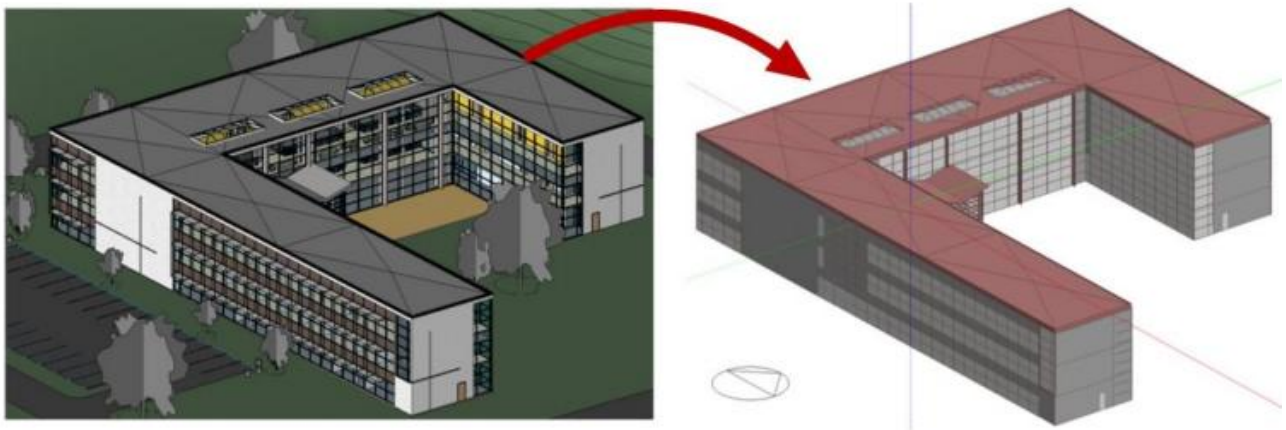
The adjacency between neighbouring surfaces is calculated within DesignBuilder based on the [Adjacency tolerance](#) setting.

Constructions and glazing systems are assigned to surfaces in the gbXML and when the [Import thermal properties](#) option is selected in the import dialog the constructions assigned in the BIM program will be assigned at surface level within the imported model.

Shading surfaces in the gbXML are retained and displayed in the imported model. Other surface objects aren't normally required and are hidden after the import has been completed but can be activated as shading surfaces from the Model options dialog, Drawing tools tab.

Note: For models imported through the gbXML mechanism, the windows will always be custom. This is true even when importing a model that had previously been [exported](#) from a DesignBuilder model that used the default façade mechanism.

The Revit DesignBuilder Workflow



There is a DesignBuilder add-in for all recent versions of Revit, which can help to ease the workflow when transferring BIM models from Revit to DesignBuilder.

Tip: The [DesignBuilder Revit – gbXML Tutorial](#) contains some useful advice on how to use the Revit to DesignBuilder link to best effect.

Installation

The appropriate Revit add-in is installed when you install DesignBuilder. Ideally the version of Revit you wish to use should be installed at the time you install DesignBuilder. However if this is not the case, you can install the DesignBuilder Revit add-in individually after you have installed the version of Revit you wish to use without reinstalling the whole DesignBuilder application. The DesignBuilder Revit add-in can be installed from the [Downloads](#) section of the website. Note that a different add-in is available for each version of Revit. You should ensure that you install the right version. For example if you are using Revit 2021 you should install the **DesignBuilder Add-in For Revit 2021**.

Note: It is important to understand that the quality of the BIM geometry data ultimately imported into DesignBuilder is directly related to the quality of the geometry included in the gbXML itself. This in turn depends on a) the accuracy of the base Revit architectural model and b) the care with which the analytical model settings were set up prior to the gbXML export.

IDF Import



This page brings together some additional information relating to the IDF import option which can be accessed from the "Import 3-D Model" on page 170

The IDF Import tool allows the building geometry and materials and construction data in EnergyPlus IDF model files to be imported into a DesignBuilder model. All EnergyPlus versions from 7.2 forwards are supported.

In summary the process to use IDF Import tool is as follows:

1. Select the **Import 3-D Model** toolbar icon (above), or use the **File > Import > Import IDF file** menu command.
2. On the first page of the **Import 3-D Model** dialog select the **2-IDF** File type if it is not already selected.
3. Select the IDF file to be imported and press the **Next** Button.
4. On the second page make any selections as described on the "Import 3-D Model" on page 170 dialog help page.

The import may take a while especially if the IDF file is an old version and the IDF data needs updating to the latest version.

The IDF Import processor places a log file in the same location as the main DesignBuilder diagnostics file in the form of IDF_Import_YYYY_MM_DD_HH_MM_SS.log. The latest 10 log files are retained. Any errors in updating IDF files to the latest version ahead of the import are included in the log file.

Supported IDF Versions and Model Objects

DesignBuilder v2025.1 can import IDF files for EnergyPlus v25.1 and earlier.

The following model data objects are supported for import:

General

- Site:Location
- Building
- Zone
- GlobalGeometryRules

Detailed Surfaces

- BuildingSurface:Detailed
- Wall:Detailed
- RoofCeilingDetailed
- Floor:Detailed
- FenestrationSurface:Detailed
- Shading:Site:Detailed
- Shading:Building:Detailed
- Shading:Zone:Detailed

Simple Surfaces

- Wall:Exterior
- Wall:Interzone
- Wall:Adiabatic
- Wall:Underground
- Roof
- Ceiling:Interzone
- Ceiling:Adiabatic
- Floor:GroundContact
- Floor:Interzone
- Floor:Adiabatic
- Window
- Window:Interzone
- Door
- Door:Interzone
- GlazedDoor
- GlazedDoor:Interzone

Materials & Constructions

- Material
- Material:NoMass
- Material:AirGap
- Construction
- WindowMaterial:SimpleGlazingSystem
- WindowMaterial:Glazing
- WindowProperty:FrameAndDivider
- WindowMaterial:Gas
- WindowMaterial:GasMixture

Limitations

- As detailed above the import tool supports the main building and shade geometry, materials and constructions but other IDF data is not imported.
- Use of the zone, surface or opening multipliers is not supported. If any of these multipliers are encountered during an import a warning will be recorded in the log file.
- For materials where material thickness is either not specified or is set to 0 a default thickness of 0.05m is applied.

Merge Model Data From File

It is sometime necessary to accommodate changes made by the architect to a BIM model after significant work has already been done on the energy model on a previous iteration of the design. The **Merge model data from file** command allows you to load all of the important settings from the DesignBuilder model you previously worked on into the latest geometry imported from BIM.

In the descriptions below the "old" model refers to the DesignBuilder model you previously worked on whose settings you wish to import, and the "new" model refers to the DesignBuilder model newly imported from the latest BIM model and needing settings to be imported from the old model.

A typical process is as follows:

1. Create a new DesignBuilder model from the updated BIM model by importing gbXML or through the Revit add-in.
2. In the new DesignBuilder model, use the **Merge model data from file** command to merge the attributes and any components, templates and HVAC system from the previous "old" DesignBuilder model.
3. Check the merge has been completed as required. Note that any newly added blocks, zones, surfaces and openings in the updated model will not receive settings from the old model that was merged in.

When the **Merge model data from file** command is selected a dialog is displayed allowing you to select the "old model" file to be merged into the "new model" with some options. When these settings are confirmed DesignBuilder makes the following changes to the new model:

- Blocks, zones and surfaces are matched between the old and the new geometry based on their geometry and position. Where an object is matched all hard set attributes (those shown in red in model data) are copied over from the old to the new model.
- All user-defined components and templates are copied from the old to the new model.
- If a Detailed HVAC system was defined in the old model then it is loaded to the new model (optional merge). It is important to check that HVAC zone groups contain the right HVAC zones after the merge has been completed.
- Any assemblies defined in the old model are merged into the new model (optional merge).
- Any component blocks defined in the old model are merged into the new model (optional merge).
- Openings such as windows, doors etc are merged based on a user-definable selection. The options are:
 - **1-Retain existing openings** where the new openings are kept and the old ones are not copied over.
 - **2-Overwrite all existing openings** where the old openings are copied to the new model replacing all new openings.
 - **3-Merge non-overlapping openings** where the old openings are copied to the new model wherever possible without overlapping with the openings already in the new model.

Note: The old and new models referred to above must each only contain 1 building.

Tip: The Merge model from file system was specifically created to accommodate the needs of energy modellers working with BIM models that are often in a state of flux, however it can be used in any situation where data is to be merged from one model to another, especially when some degree of geometry matching is required.

Import Building From External File

It can sometimes be necessary to import buildings/blocks from another dsb file into a file that contains an existing model. To do that follow the steps below.

1. Open the model where the building geometry is to be imported to.
2. From the DesignBuilder File > Import menu, click on the **Import building from external file** option and select the dsb file containing the building(s) to be imported. When the import is completed you will end up with the original building and the newly imported building(s) from the external file.

Tip: If you wish to merge selected blocks from the imported building(s) to the existing building you can use the [Transfer blocks](#) tool to move the blocks from the imported building to the existing building. Once you have transferred the blocks you wish to keep you can delete the imported building(s) from the Navigator panel.

Note: Bear in mind that components and templates are not imported with the geometry and attributes when using the **Import building from external file** command. This means that if the building in the external file refers to custom user-defined components or templates that are not also included in the destination model, it is likely that component/template selections will be missing or incorrect in the imported building. For example if the model in the external file includes a custom activity template and it is imported to a model

without that template then you are likely to end up with blank settings where you might expect that Activity template to be selected. **In general you should assume that any custom components and templates and their selections will need to be remade in the imported model.**

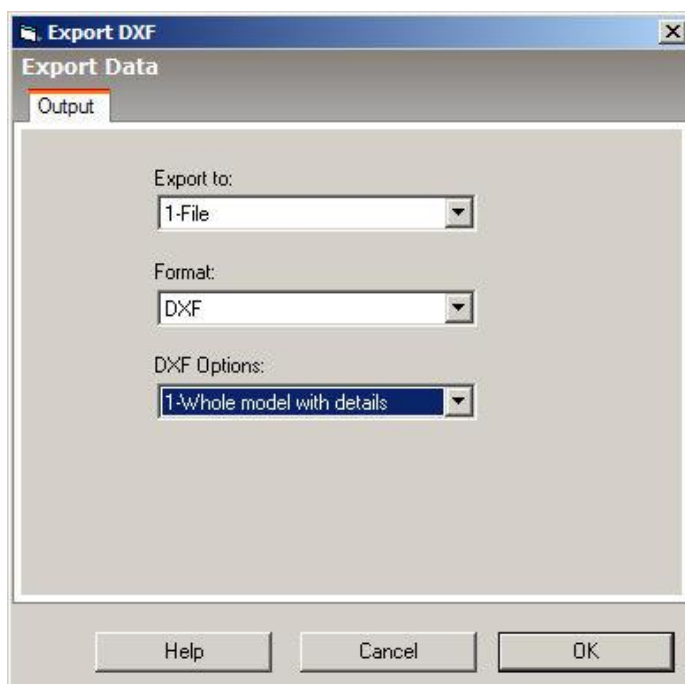
Exporting Geometric Data

The model geometric data can be exported in these formats:

- [3-D DXF](#)
- "Exporting gbXML" on page 182

Exporting 3-D DXF

You can export the geometry of the current building model in 3-D DXF format using the Export 3-D DXF model command from the [File](#) > Export menu. This opens the 3-D DXF Export dialog:



Choose one of two options:

- **1-Whole model with details** - The entire building model is exported including 3-D details such as window frames, 3-D building elements etc. This option can be useful for exporting for visualisation and further processing in other software such as SketchUp (image below)
- **2-Zones** - The building model is exported showing only zones. This option can be useful for visualising the geometry of the model used in simulations.

Note: DesignBuilder DXF output uses surface objects and not lines so be sure to select one of the shaded surface view options in your DXF viewer. Wire frame views will not display the model.



Example 3-D DXF Output - Whole model with details

The above model was created in DesignBuilder, exported in DXF format using the **Whole model with details** option and Imported into SketchUp where details such as textures, trees and people were added.

Note: when importing DXF files into SketchUp be sure to set the units options on the Open dialog to Metres.

Exporting GbXML

You can export the current building model in gbXML format using the **Export gbXML model** command from the [File](#) > Export menu. This opens the gbXML Export dialog. Press OK to confirm and select the name of the gbXML file to be created.

The gbXML file includes full surface geometry, zone lists with names, windows, doors, component block shading devices, constructions, materials and glazing systems. Note that all other data is not exported. No data is stored on activity, lighting, HVAC and model settings.

gbXML files exported from DesignBuilder can be imported into any other software supporting the gbXML standard file format, including DesignBuilder v3 and later.

Tip: It is not generally possible to load models saved in DesignBuilder dsb format into earlier versions, but as an alternative you can export gbXML from a recent version and import that into earlier versions of DesignBuilder supporting the gbXML import. While this method would not retain most model data settings it would allow you to transfer the data listed above.

Working With Buildings (Site Level)

At the site level you can:

- [Add new buildings](#),
- [Copy, delete and change the position](#) of existing buildings.
- Change the [site orientation](#).
- Enter [site model Data](#) including information on the site location, weather data etc.

Add New Building



You can add one or more new buildings from the site level. Each building has its own set of [Model options](#) and is simulated separately but shares [site Model data](#) with the other buildings on the site.

When you select the Add new building option you are presented with the **Add new building** dialog which allows you to make some basic settings:

- [Building](#) - basic setting for the building model
- [Default Data](#) - defaults for Construction, Activity, Glazing, Lighting and HVAC

When you press OK to close the dialog the building is created with the various selected settings and in most cases starts you in Draw block mode so you can start adding geometry straight away. If you are not ready to do this simply press the <Esc> key or right click and select Cancel.

See also the [Starting a new project](#) tutorial

Including Multiple Buildings On The Site

You can add as many buildings to a site as you like, however there are some considerations to bear in mind when more than one building are to be included on a site. These are listed below.

1. You can obtain snaps from other buildings when moving or copying buildings at site level, however it is important to be aware that buildings do not share adjacencies with each other. In other words, you may have positioned Building A so that one of its façades is adjacent to the façade of a neighbouring building, but that does mean that there will be an internal adjacency set up between the 2 buildings - the touching façades of the 2 buildings will be simulated using external surfaces.
2. Snap points from other buildings are not available at building level or below, so if you need to accurately align buildings with each other you may need to go to site level and move them into position.
3. Other buildings can be accounted for in EnergyPlus and Daylighting simulations by selecting the appropriate calculation options. For both [EnergyPlus](#) and [Daylighting](#) the other buildings can cast shadows and provide reflections, but they cannot be simulated within the same calculation.
4. Each building on the site adds to the memory overhead, especially if large amounts of result data are stored. So if buildings and/or their stored result sets are likely to be large, when planning your modelling, you may consider including each building in a separate model and simulating the shading/reflectance effects of neighbouring buildings using component blocks.

Add New Building - Building Tab

Model Type

Type

Select the type of building model you wish to create from the list of options. Your selection will depend on what you want to do with the building model and the amount of data available. Some typical options are:

- **Default** - Recommended default settings for general-purpose modelling.
- **Architect early stage design** - Early design stage analysis of building components, façade and cooling options.
- **Detailed design with HVAC** - If you plan to use the [Detailed HVAC](#) model option then select this option.

- **Parametric building + standard data** - Add a building of standard form (rectangular, L-shaped or courtyard) without needing to draw anything.
- **Calculated natural ventilation** - Airflows are calculated using the EnergyPlus multizone AirflowNetwork based on wind and stack pressure differences.
- **ASHRAE 90.1** - The template to be used for ASHRAE 90.1 models which sets Detailed HVAC and other appropriate model options. This selection is forced for [ASHRAE 90.1 models](#).

Tip: Regardless of the selection you make here, you can fine tune the Model options later after having created the building using the [Model options dialog](#).

Geometry convention template

Select the geometry convention template to be used for this building model. You can easily change this at any level in the model on the Constructions tab under the [Geometry, Areas and Volumes](#) header.

ASHRAE 90.1 building type

If this is an ASHRAE 90.1 model then you can select whether it is a proposed or baseline model. In most cases you will start by creating a proposed model and create the baseline model from this. Options are:

- **1-Proposed** and
- **2-Baseline**.

Add New Building - Default Data Tab

This tab allows you to define the templates used to populate the new building with default Model Data.

Construction

Select the construction template to use as a source of default fabric data for the building. By default DesignBuilder will select the Reference construction template for the current region, or, if a building has been added previously, then the templates selected the last time a building was added are provided as defaults.

Activity

Select the activity template to use as a source of default activity-related data for the building. This will set the following default data:

- Occupancy levels and timing, holidays etc
- Metabolic rates
- Levels of equipment use
- Heating and cooling temperature setpoints
- Illuminance requirements
- Fresh air requirements

Glazing

The glazing template is used as a source of default window, shading and percentage glazing data for the building.

Lighting

The lighting template is used as a source of default lighting and lighting control data for the building.

HVAC

The HVAC template is used as a source of default data for the HVAC equipment to heat and cool the building.

Working At Building Level - Blocks And Solar Collectors

At building level you can:

- [Add Blocks](#) - Building blocks, Component blocks and Outline blocks.
- [Edit Block geometry](#) using a range of different block editing tools.
- [Add Solar collectors](#) - PV and solar hot water.
- "Edit Openings at Building Level" on page 249

The rest of this topic about adding various types of block.

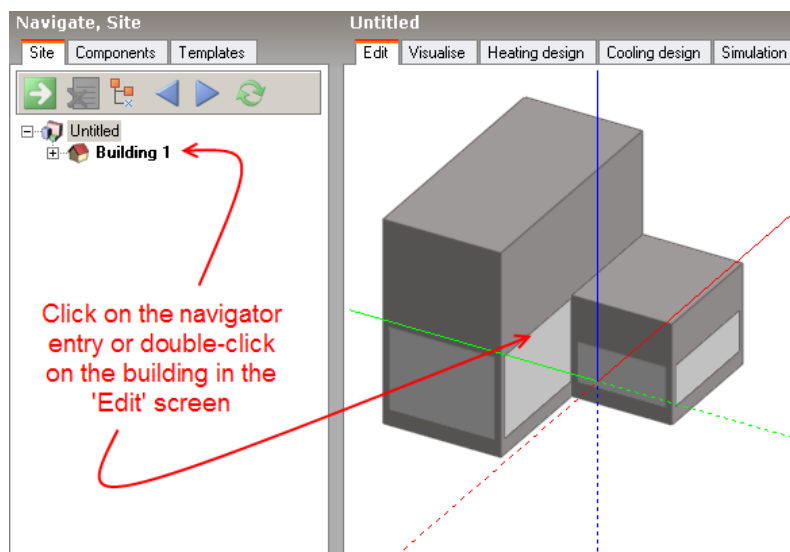
Blocks are the basic elements used to create a 3-D DesignBuilder model. Three types of block can be added to the building:

- [Building Blocks](#) are the mechanism used in DesignBuilder to create 3-D building volumes. They are often referred to in simply as Blocks and are composed of building elements such as walls, floor slabs and roofs . Building blocks may be partitioned internally to form a number of zones.
- [Outline Blocks](#) are used to help in the creation of more complex building block geometry.
- [Component Blocks](#) are used to create visual, ground and shading structures which do not contain zones.

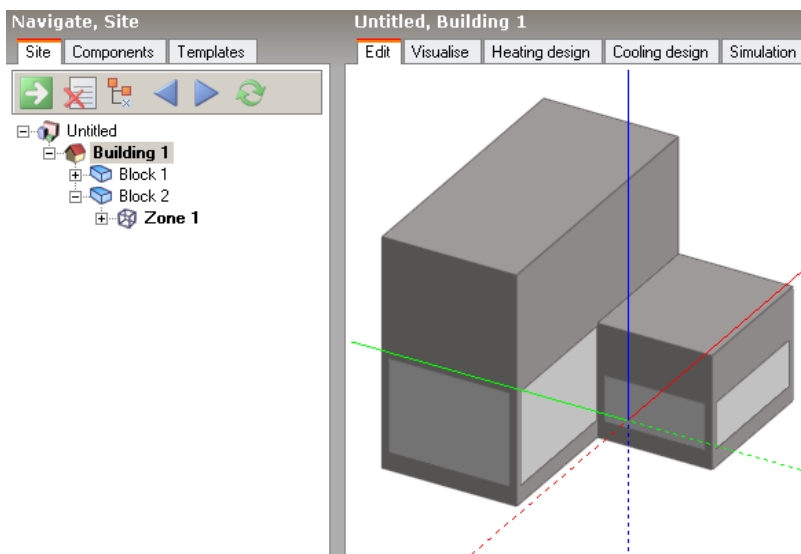
Blocks of all types are created at the Building Level in the same way and are drawn using [perimeters](#) which can be extruded either horizontally, vertically, or in any other direction.

This section describes the various operations involved in [creating](#) and [editing blocks](#) at the building level.

To create or edit a block you must first go to the building level and you can do this by clicking on the building entry in the navigator or if you're at the site level and you want to edit an existing building, you can double-click on the building in the 'Edit' screen.



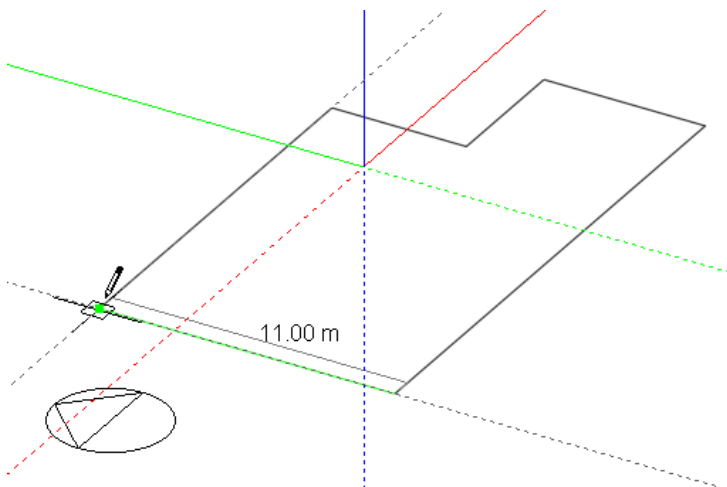
Either operation will take you to the building level:



Adding Blocks

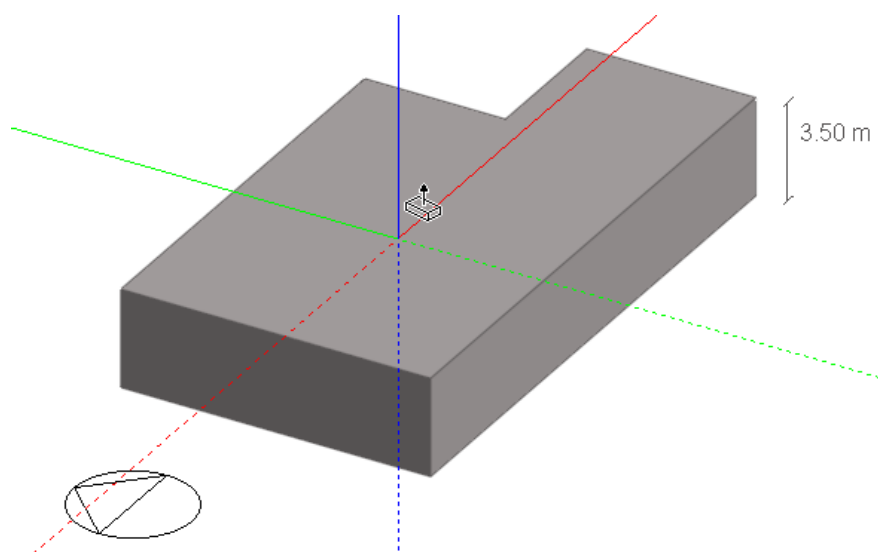


To add or create a new block, first go to the building level (if you are not already there) and click on the **Add new block** tool. If you've just created a new building, the Add new block tool is automatically activated. A block is created by drawing the base perimeter of the block using one of the perimeter shapes such as a polygon. The [dimensions of the block](#) are those measured from the outside, the height of the block is the floor-floor height.

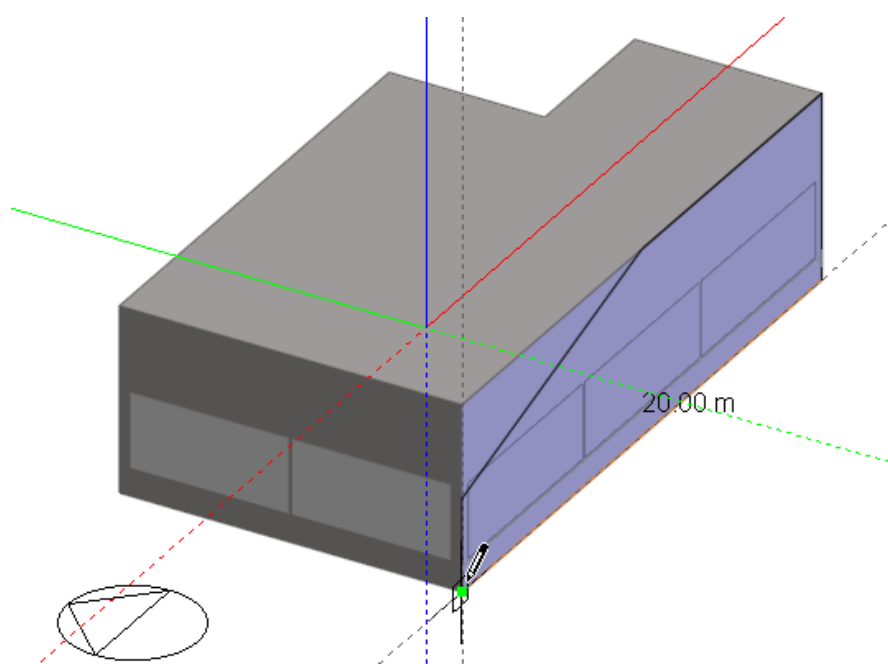


See also these tutorials: [Starting a new project](#), [Blocks and dimensioning protocols](#) and [Adding blocks and using the measure tool](#)

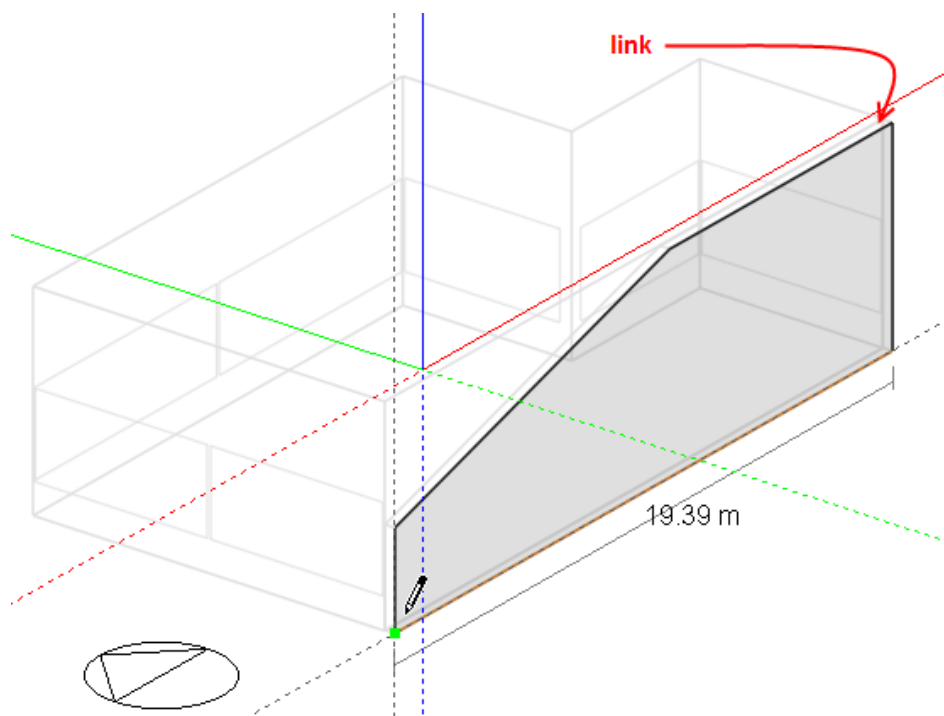
Please refer to [perimeter shapes](#) for details of drawing perimeters. After drawing a perimeter, a block will be automatically extruded by the [Height](#) entry in the 'Drawing options' settings or if the [Auto-complete block](#) setting is switched off, the perimeter can be dragged to form the extrusion. To help in setting the correct extrusion height, the [increment snap](#) setting can be used which enables the drag movement to jump by any defined increment. Alternatively, you can simply type in an extrusion height and press the enter key:



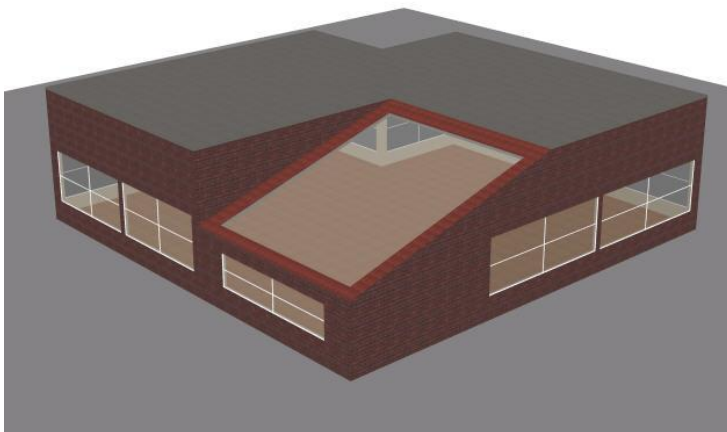
When a block has been placed, additional blocks can be drawn onto any surface of the block to form more complex aggregate blocks. As you move the mouse cursor across the faces of an existing block, each face will change colour from grey to blue to indicate the active drawing plane. You can lock the active drawing plane by pressing the SHIFT key, which allows you to move the cursor off the current face while keeping the plane active. You can also lock the active drawing plane to the ground plane by moving the cursor over the background area and pressing the SHIFT key, this is particularly helpful in avoiding the selection of the wrong plane when drawing a vertically extruded block adjacent to an existing block. Once you've clicked the first data point into the active drawing plane, it becomes locked regardless of whether or not the SHIFT key is pressed until the perimeter has been completed or the tool is cancelled. In the example below the vertical plane shown in blue has been selected, the shape of the block will be drawn in this plane and the volume created by extruding normal to it.



When a block is added to the surface of another block, a geometric [link](#) is automatically created between the two blocks:



A [hole](#) can be added to the mating surface of this link (the [inter-block partition](#)) in order to form one contiguous space:



Note: the [Merge zones connected by holes](#) Model option must be set if you wish the two zones in the 2 blocks to be treated as one zone in the thermal calculations.

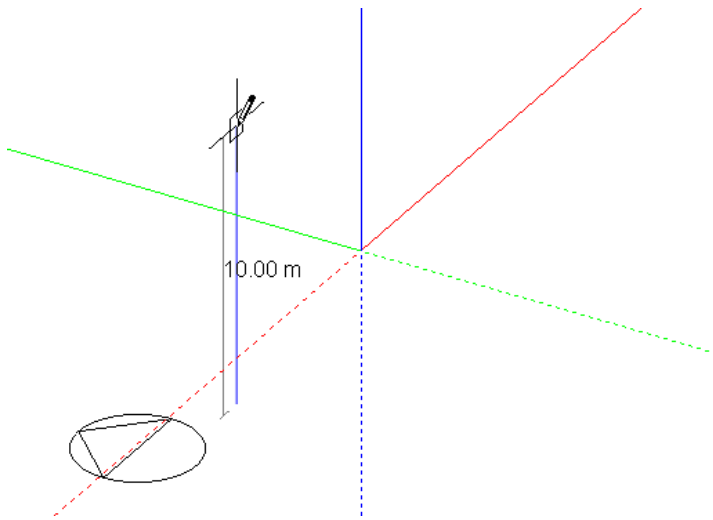
As already illustrated, a block perimeter can be drawn on the ground plane or onto the surface of an existing block in which case the perimeter is extruded normal to the surface on which it is drawn.

Avoiding Indents

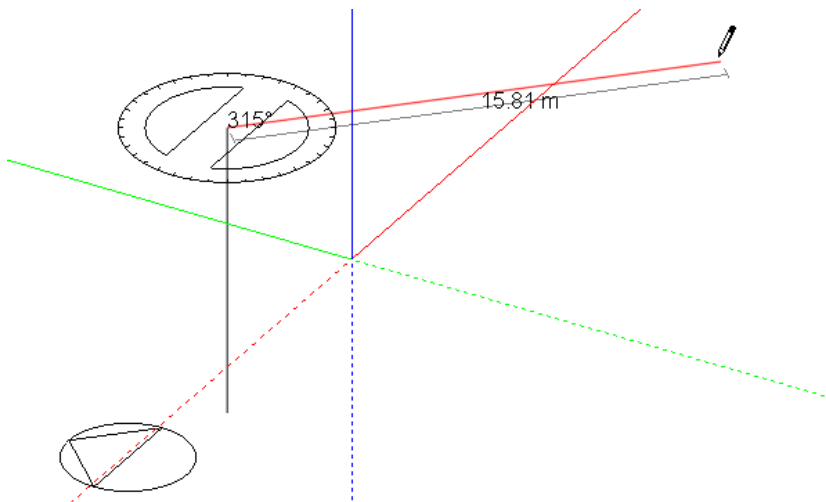
If the perimeter includes indents in such a way that the requested wall thickness is physically impossible and the wall thickness cannot be accommodated then DesignBuilder will still allow the block to be created but the wall thickness is reset to a small value (typically 0.025m). See the wall thickness topic for more on this.

Drawing in a vertical plane without reference to an existing block

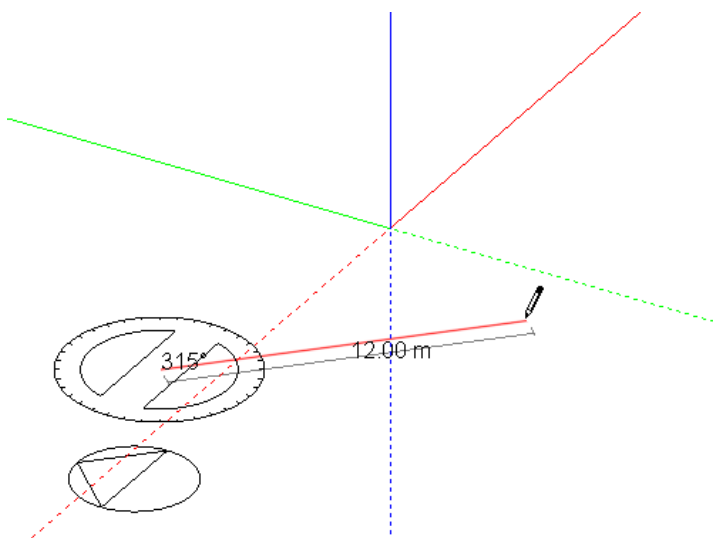
To create horizontal extrusions from a vertical plane, the first perimeter line can be aligned with the Z-axis using the [axis snap](#):



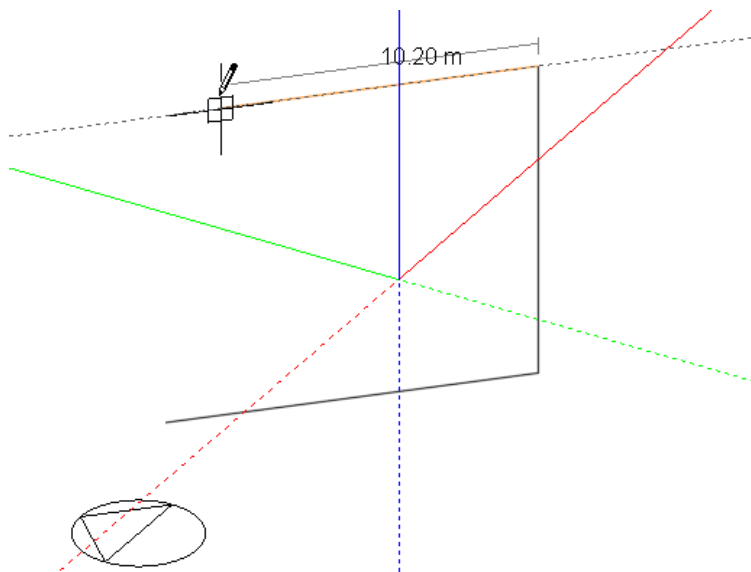
The [protractor](#) can be used to place the second line in order to accurately define the orientation of the plane relative to the base or ground plane:



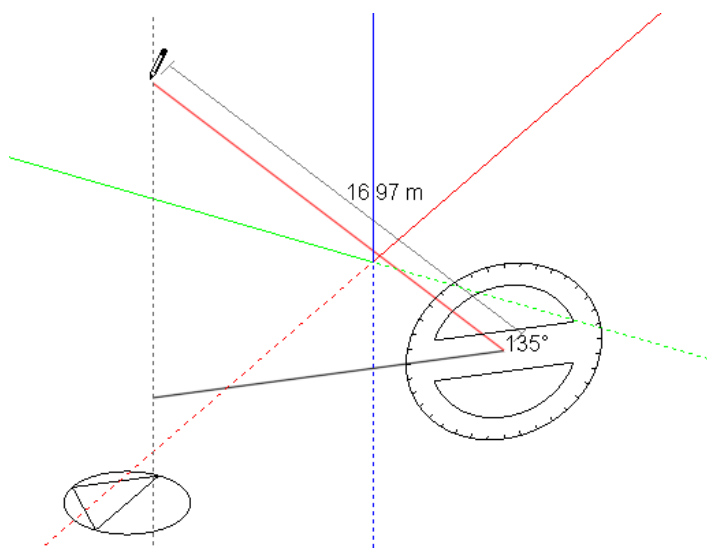
Alternatively, the orientation of the plane can be defined using the first line:



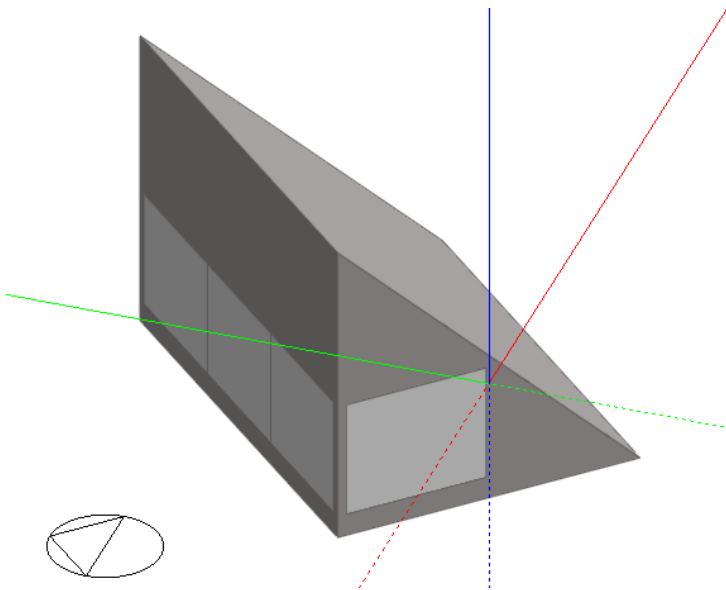
Snapping the second line to the Z-axis then forms the horizontal extrusion:



Once a vertical plane has been defined it will be 'remembered' until the 'Add new block' command is cancelled, so the perimeter can be 'unwound' using the ESC key or the right-click menu 'Undo last point' entry and any perimeter shape drawn into the stored plane:



After drawing the last perimeter line to connect to the perimeter start point, the perimeter can then be dragged to create a horizontally extruded block:



Block Creation Options

The block creation options enable you to control the way a block is drawn:

- "Block and Zone Dimensions" on page 192
- [Block types](#)
- [Block Form](#)
- [Height](#)
- [Auto-complete block](#)
- "Follow Path" on page 217

These options are accessed through the **Drawing options** data panel, which is displayed when you click on the [Add new block](#) tool:

Drawing Options

Tools

Geometry

Block type1-Building block

Form1-Extruded

Height (m)3.50

☒ Auto complete block

Perimeter

Shape1-Polygon

Line type1-Straight line

Thickness (m)0.292

Protractor>>

Direction Snaps>>

Point Snaps>>

Drawing Guides>>

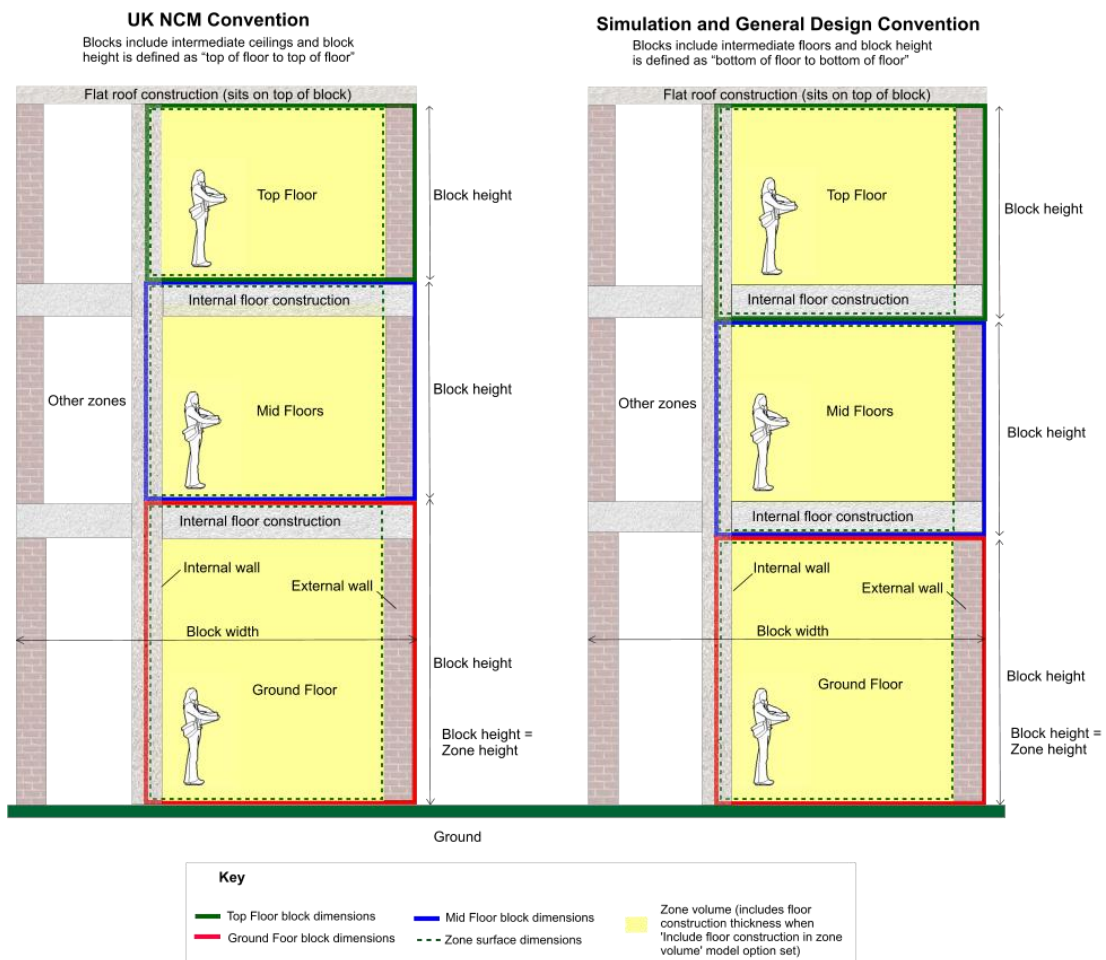
Block And Zone Dimensions

See also: [Geometry Areas and Volumes](#)

The diagram below illustrates how blocks should be defined and how zone volumes and surface dimensions are derived from the parent block. Note that there are 2 alternative dimensioning protocols:

- **UK NCM Convention** which should always be used when using DesignBuilder for assessing buildings through the UK National Calculation Methodology, i.e. for Part-L and Section 6 Building Regulations checks and EPC/BER calculations.
- **Simulation and General Design Convention** which should generally be used for all other applications where an NCM calculation is not required.

Schematic Illustrating Two Alternative DesignBuilder Block Dimensioning Conventions



The main differences between the 2 conventions are:

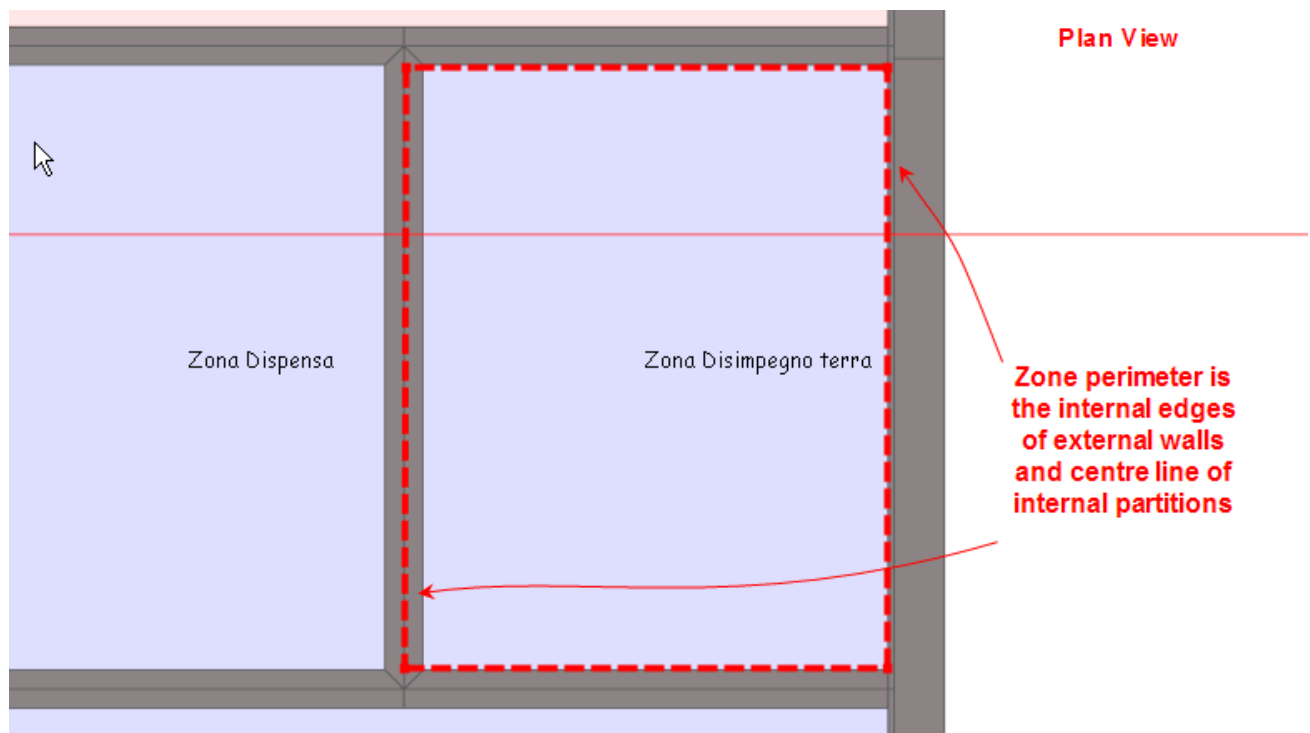
1. **UK NCM Convention** - For ground and intermediate floors, the block height is defined as "top of floor to top of floor" and for top floor blocks with a flat roof the block height is "top of floor to soffit/underside of roof slab". Details on defining building geometry for other less common geometries when using the NCM can be found in the [Non Domestic EPC Conventions for England & Wales](#) document.
2. **Simulation and General Design Convention** - For ground floors the block height is "top of floor to bottom of floor". For intermediate floors, the block height is defined as "bottom of floor to bottom of floor" and for top floor blocks with a flat roof the block height is "bottom of floor to soffit/underside of roof slab".

The important points to note for **both conventions** are:

1. The length/width dimensions of the block are measured from the outside of external walls.
2. Zone height is always the same as block height, i.e. it is unaffected by block wall thickness or thickness of floor constructions.
3. Wall surfaces extend the full height of the block.
4. Flat roofs are placed on top of the block, so when drawing a block with a flat roof you should exclude the [roof](#) construction in the height.
5. External floors are placed under the block, so when drawing a block with an external floor you should exclude the [external floor](#) construction thickness in the height.
6. Zone surface dimensions (shown in green dotted lines above) are measured from inner edges of outer walls and the centre line of partitions.
7. Floor surfaces extend horizontally to the inner surface of external walls and to the centre line of partitions. See below for more on zone/surface dimensions.

The zone **volume** used in ventilation calculations is the actual inside volume (shown in yellow in the diagram above). These inside zone surface dimensions are used to define surface heat transfer areas in simulations and other calculations.

Zone perimeter (plan view)



Other related information

1. See [Constructions](#) topic for information on the meaning of the Construction Model data.
2. Also see [Constructions](#) for how to edit the construction configuration.
3. Block wall thickness for information on how zone inner volumes are created from external block dimensions
4. [Construction and glazing](#) model options for details of volume calculations with the options to include/exclude floor and ceiling volumes from the zone volume used in the energy calculations.
5. And the Model Geometry Example section for a detailed worked example illustrating how surface areas and volumes are calculated.

See also the [Blocks and dimensioning protocols](#) Tutorial

Block Types

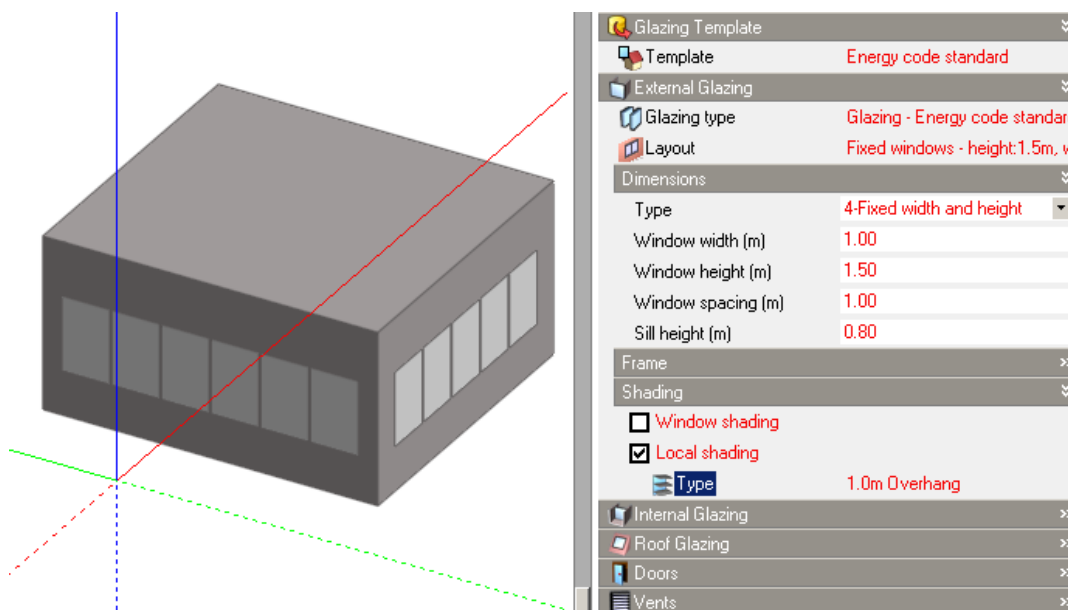
Four types of block are available:

- [Building Blocks](#) are used to draw the model or a section of the model.
- [Outline Blocks](#) are used to help in the creation of building block geometry and do not affect the model.
- [Component Blocks](#) are used to create visual and shading structures which do not contain zones.
- "Assembly Block" on page 205 are assemblies of component blocks, typically used for modelling building details and equipment in CFD or for more complex façade shading structures.

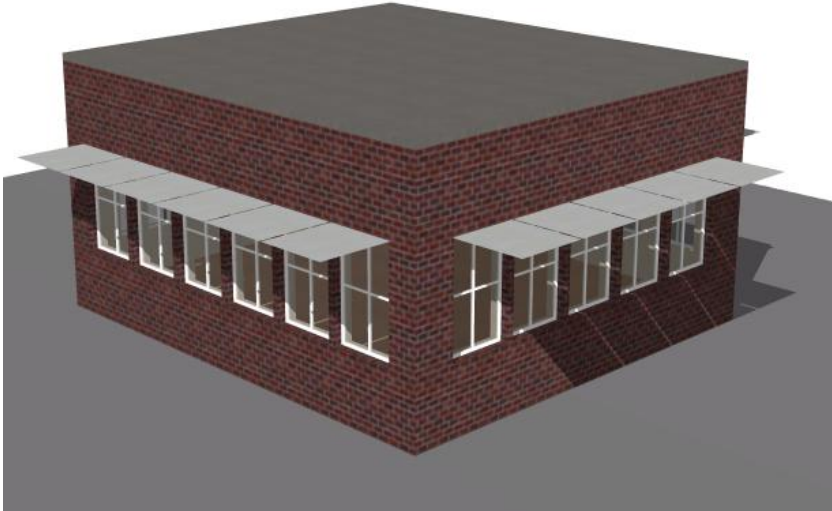
Tip: If you find you have accidentally created a block of the wrong type, you can convert it the required type using the [Convert Blocks tool](#).

Building Block

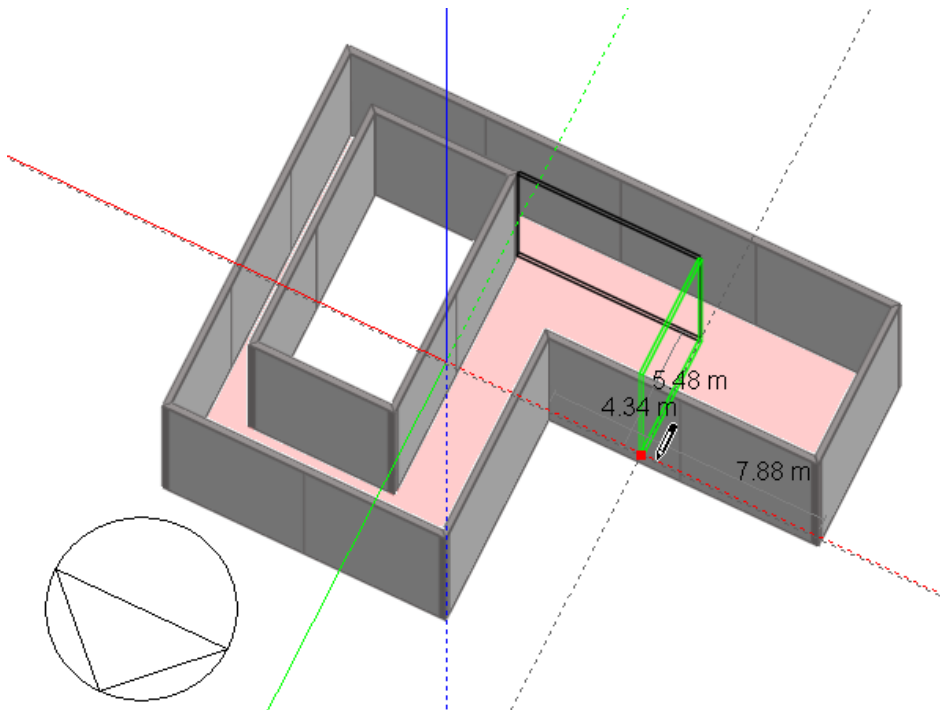
A building block represents the outer shell of the model or part of the model. A finished block comprises a set of building elements, which may include external walls, roofs and floor slabs, the constructions of which are determined from the current building [construction model data](#). Similarly, using [opening model data](#), block facades are automatically created with the required configuration of windows, doors, vents and solar shading devices:



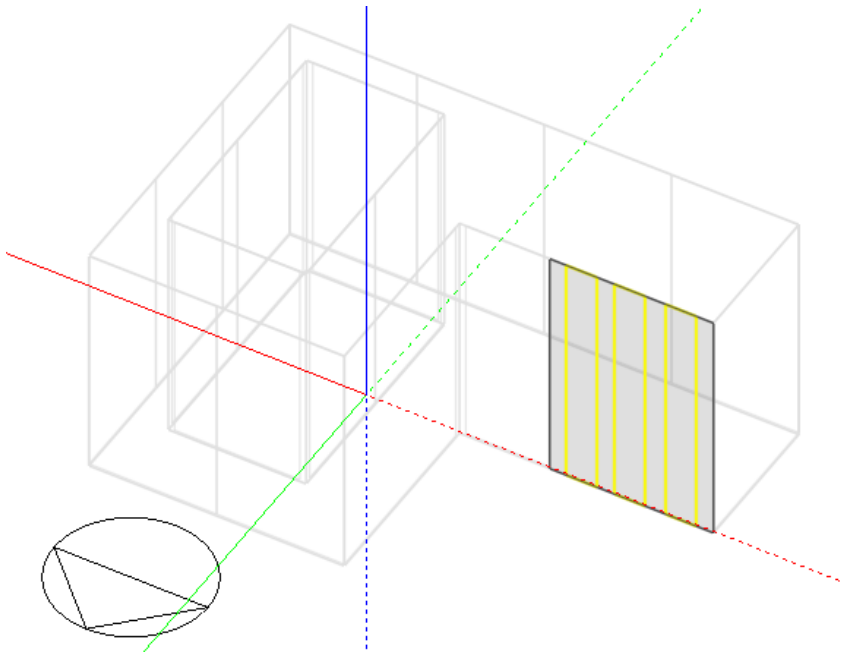
Solar shading devices are displayed in the [visualisation](#) screen:



A block may be divided internally into any number of zones by drawing [partitions](#) and where needed, voids or [courtyards](#) can also be incorporated:



Custom [openings](#) (windows, doors, etc.) can be added to block facades by going to the surface level and using the 'Draw window' family of commands:



Blocks can be [moved](#), [copied](#) and [rotated](#). Blocks can also be [cut](#) or [stretched](#) and block faces can be [dragged](#) to different positions.

Outline Block

An outline block is created and edited in exactly the same way as a building block but is just a 3-D shape without associated building elements such as walls, floors, roofs, etc.

Outline blocks are very useful in the creation of more complicated models. Because they are relatively simple objects, they are much faster to create, copy, move, etc. than conventional building blocks and can be manipulated freely without worrying about intersections. Once the required geometry has been established, outline blocks can be converted to conventional blocks.

Outline blocks can also be used as a temporary device to create drawing planes from which building blocks or other outline blocks can be extruded.

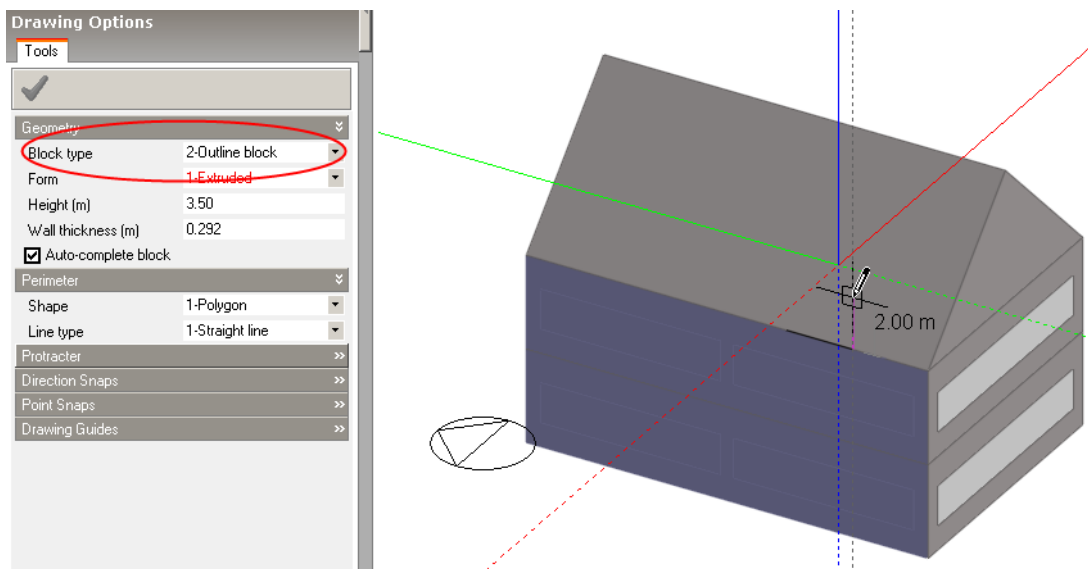
Note: Because Outline blocks are not part of the model (in the same way that [construction lines](#) are not part of the model) they do not appear in the Navigator panel.

You can remove all outline blocks from a model by selecting **Remove all outlines** from the [Edit](#) menu.

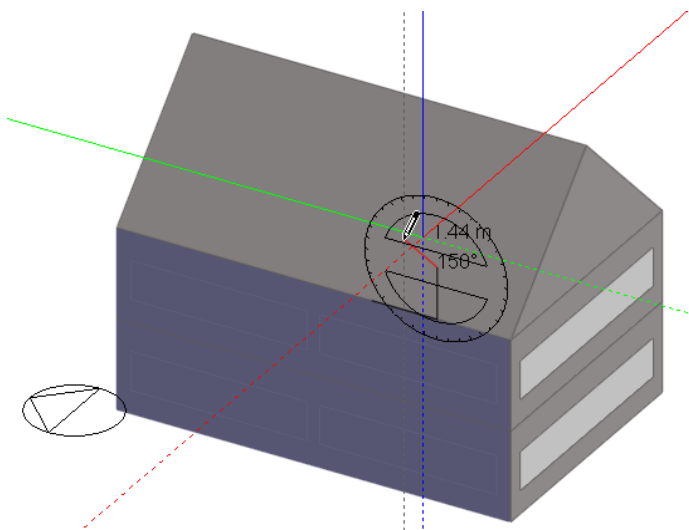
Basic use of outline blocks is covered in the [Outline blocks, Boolean operations](#) tutorial

Using Outline Blocks To Create A Dormer Window

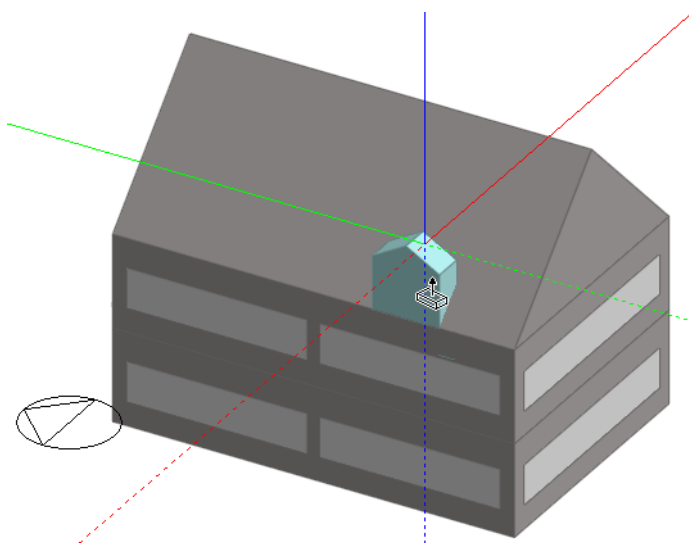
For example, to create a dormer window, first set the [Block type](#) to **2-Outline block** and lock the active drawing plane to the front surface of the first floor block by selecting it and pressing the SHIFT key, then draw the base line of the dormer face perimeter:



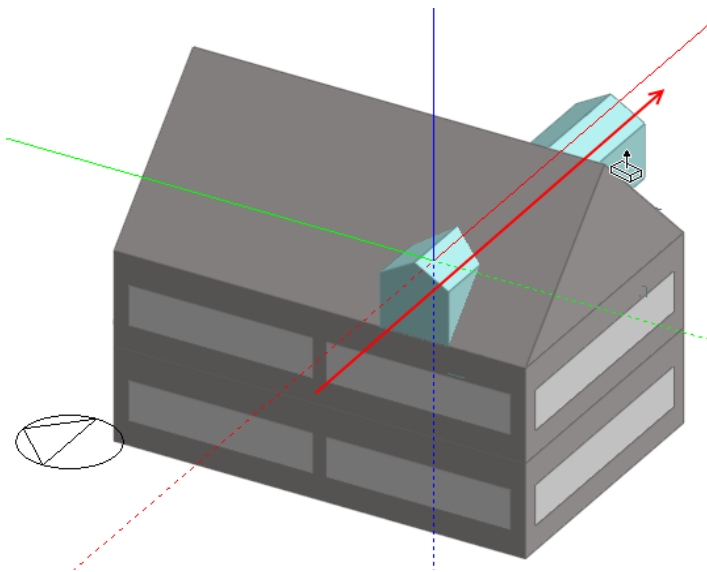
Create the angle of the dormer pitch using the [Protractor](#):



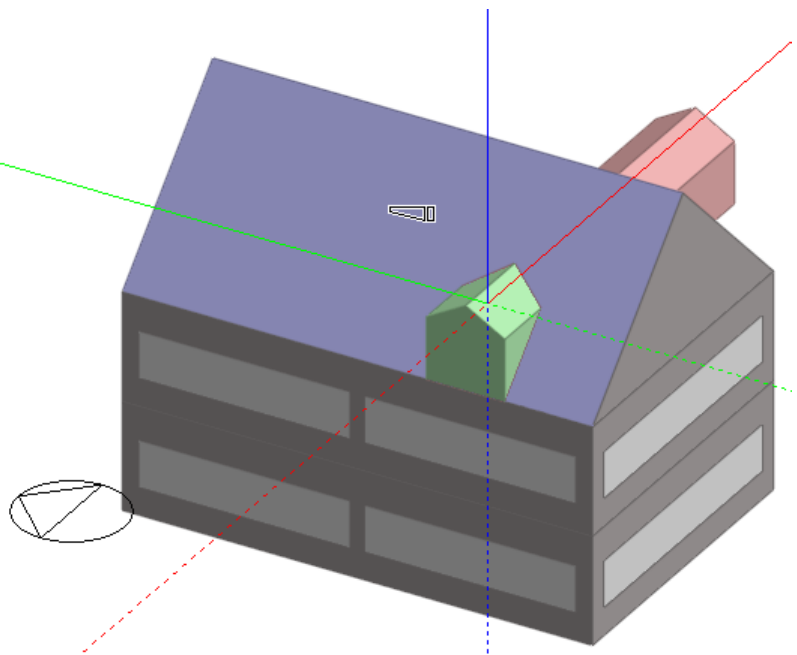
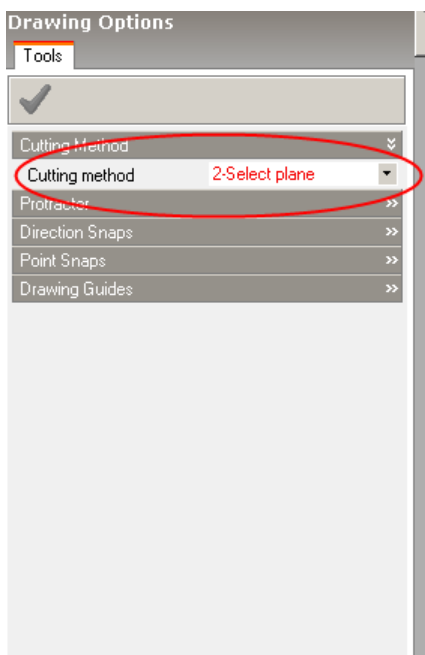
and then complete the perimeter:



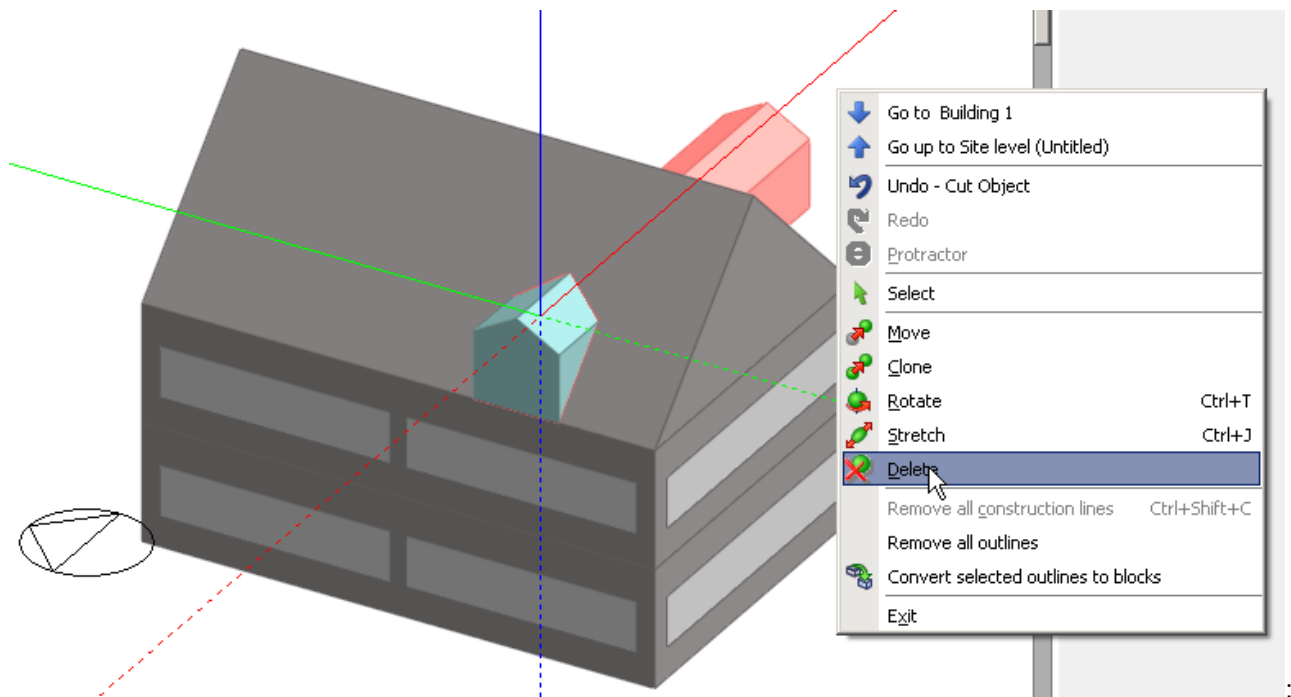
Now drag the dormer perimeter all the way through the main roof block:



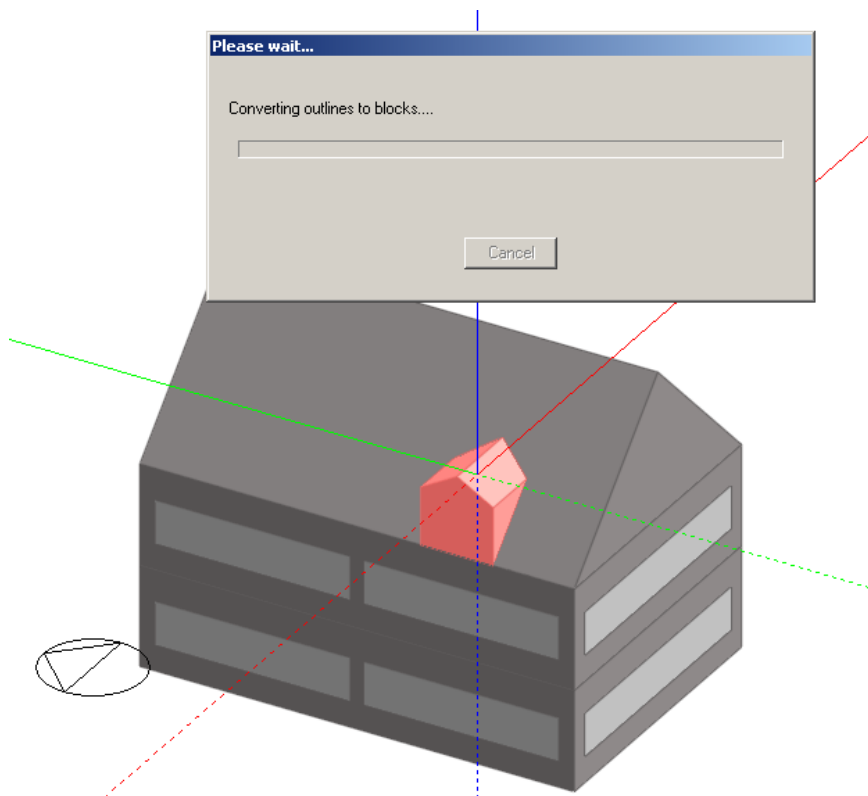
Select the dormer block and click on the [Cut block](#) tool, making sure that the Cutting method on the Drawing options data panel is set to **2-Select plane** and click on the forward facing plane of the main roof block to create a cut in the dormer block in the plane of the roof:



Click again to finish the cut and then select the section of the dormer block that is not required and delete it:



Select the dormer outline block and click on the [Convert outline block](#) tool to complete the process:



The finished block after adding openings:



Drawing dormer roofs is also illustrated in the [Dormer Roof Tutorial](#)

Note: Sometimes when converting outline blocks to component or building blocks you will notice that the new block has a very slightly different size to the original outline block. This can happen when DesignBuilder needs to slightly adjust the size in order to generate a viable zone with 3-D slab surfaces. It is therefore advisable when using outline blocks to create complex geometries to draw and convert the outline blocks individually. This avoids adjacencies between touching outline blocks from getting 'broken' when they are converted 'en-masse'.

Component Block

There are 3 types of component block:

- [1-Standard](#) - used for shading, reflection and visualisation.
- [2-Ground](#) - used for setting ground adjacency, shading, reflection and visualisation.
- [3-Adiabatic](#) - used for setting adiabatic adjacency for modelling adjacent buildings at similar conditions, shading, reflection and visualisation.

Component blocks are drawn in the same way as Building blocks but selecting the [Block type](#) in the Drawing Options panel to be **3-Component block**.

Each Component block has its own Component block material, transmittance schedule and shade/not shade [model data](#). To edit this you must first go to the component block itself by double-clicking on it in the model edit screen or clicking on the entry in the Navigator. Alternatively, component blocks inherit default model data from the building level so if you leave the component block model data in its default state (blue) you can change all model data in one go by making the edit at the building level.

All component blocks cast shadows in [Visualisations](#). In Radiance daylighting calculations component blocks both cast shadows and provide diffuse and/or specular reflection.

Standard Component Block

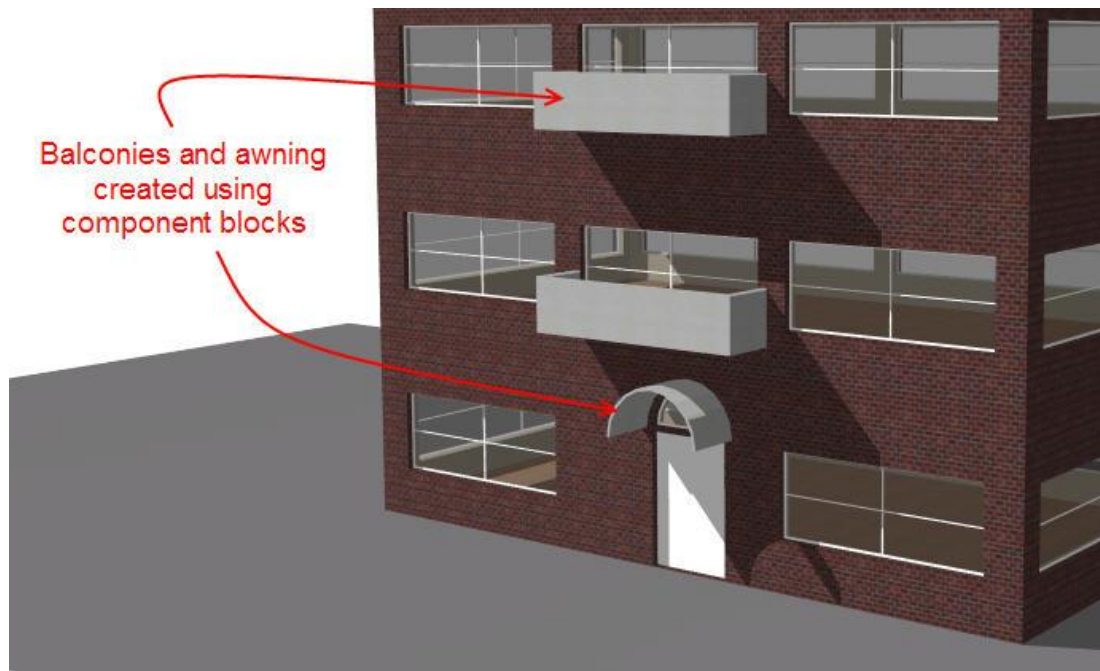
Sometimes it can be useful to create structures such as pillars, shading obstructions and balconies which do not contain zones. **Standard component blocks** can be used for modelling such structures as they are treated simply as shading/reflection surfaces in simulations without any zones.

Standard component blocks are visible in [Visualisations](#), using the texture associated with the [Component block material](#) as set on the [Constructions tab](#) under the Component Block header. They can be used to cast

shadows and reflect solar radiation and light in EnergyPlus simulations by checking the [Component block shades and reflects](#) model data under Component Blocks on the Constructions tab.

Note: In EnergyPlus simulations, component blocks do not shade ground reflected solar radiation and light unless the [Model all external reflections and shading of ground reflected solar](#) calculation option is used.

The image below shows some typical uses of Standard component blocks.



Important Note: Component blocks do not absorb or conduct heat in any way - their only effect on building surfaces in simulations is related to the shading and reflection of short-wave solar radiation and light.

Component blocks are a flexible way to apply shading to any part of the building and can be used to model more complex window shading systems not catered for by the in-built [local shading](#) systems that can be selected on the Openings tab. However there are some issues that should be considered when using component block in this way and these are discussed in the [Component blocks used for local shading](#) reference page.

Note: Standard component blocks can be added at building and block levels, but **component blocks added at block level are not used in EnergyPlus simulations**. They are however used in CFD, Daylighting and Visualisation.

More information can be found in the [Shading using standard component blocks](#) tutorial.

Adding component blocks to assemblies

One or more component blocks can be combined into [assemblies](#) and stored in the [Assembly library](#). Assemblies are used in EnergyPlus simulations for shading in much the same way as standard component blocks. The same rules as described above apply in that they are used at building level in EnergyPlus simulations, and at both building and block level for CFD, Daylighting and Visualisation.

One important advantage of using assemblies over component blocks is that it is possible to edit the size or form of one instance of an assembly and all similar instances are automatically updated as well. This allows a large number of geometries to be edited in once go which can be useful for quickly editing façade geometry for example.

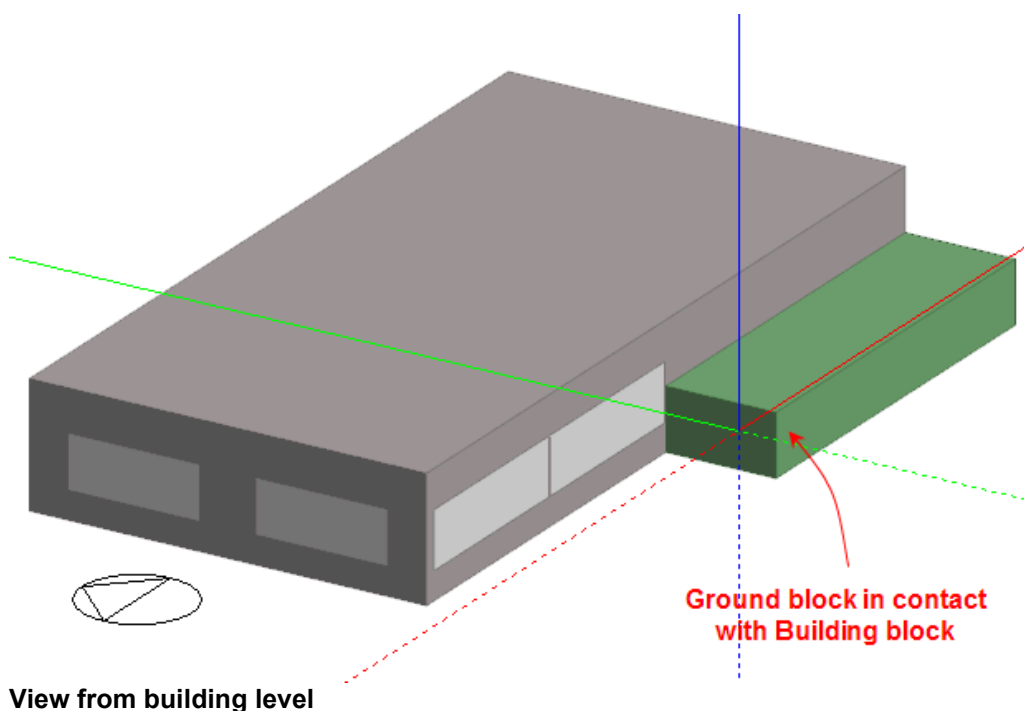
Ground Component Block

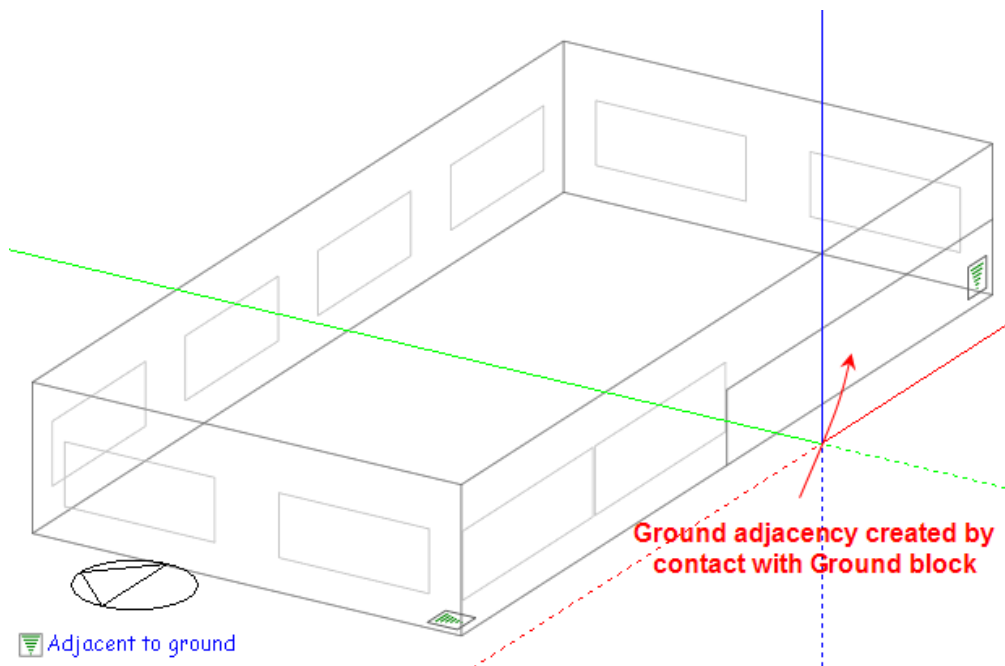
Ground component blocks can be used as a convenient way to set the adjacency of any touching building block surfaces as being 'adjacent to ground'. In the model below the Ground block (shown in green) has been positioned in contact with the building block to create a ground adjacency in the part of the Building block surface in contact with the Ground block.

Ground component blocks are visible in [Visualisations](#), using the texture associated with the [Component block material](#) as set on the [Constructions tab](#) under the Component Block header. They can be used to cast shadows and reflect solar radiation and light in simulations by checking the [Component block shades and reflects](#) model data under Component Blocks on the Constructions tab.

Note: It is important to understand that the ground component block is used only to modify the adjacency of the surfaces touching it to be 'adjacent to ground'. The areas of the zone surfaces that do touch the ground component block are treated as being adjacent to the [monthly ground temperatures](#) defined for the site. **The material associated with the ground component block and the thickness of the block are not used in any way to modify the conduction path of heat from the building to the ground.** To modify the properties of the ground heat transfer, edit the [ground construction](#).

Surfaces with any openings which touch a ground component block will have those openings removed when the component block is placed. Also it is not possible to add openings in parts of a zone surface that touch a ground component block. This applies to sub-surfaces as well as windows, doors, vents and holes.





View from zone level

Ground component blocks are displayed in the visualisation screen using the [Ground texture](#) as set at the site level.

More information on ground modelling can be found in the [Ground Modelling](#) topic.

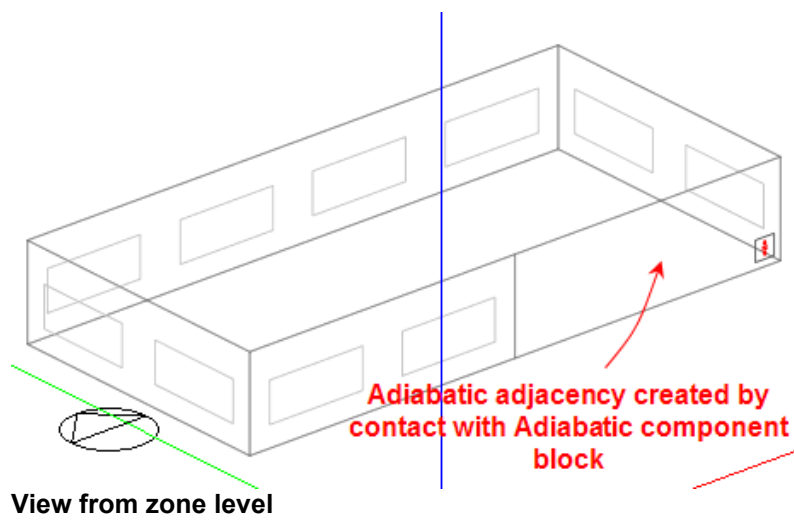
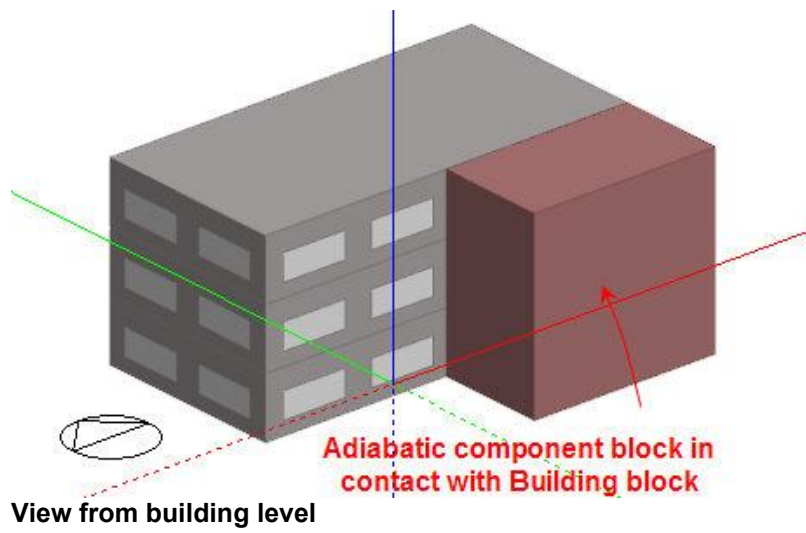
Note: The adjacencies established by contact with ground blocks can be overridden by making settings under [Adjacency](#) in Constructions model data at the surface level.

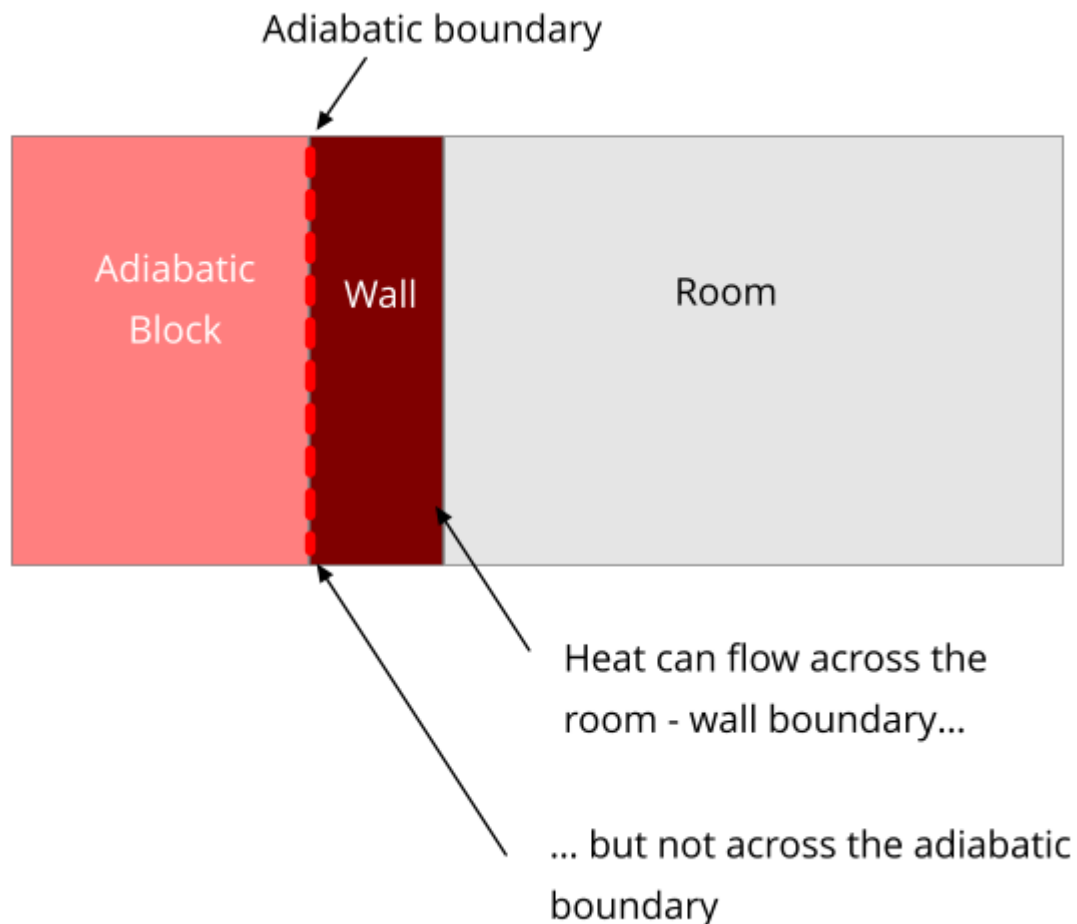
More information on how to work with ground blocks can be found in the [Setting adjacency conditions using ground and adiabatic blocks](#) tutorial.

Adiabatic Component Blocks

Adiabatic component blocks can be used as a convenient way to set the adjacency of any touching building block surfaces to 'adiabatic'. Adiabatic surfaces do not transfer heat beyond their outer surface and are often used for modelling boundaries with other spaces that can be considered to be at roughly the same temperature as the zone. In the model below the Adiabatic block (shown in red) has been positioned in contact with half of the South facade of the building to model the effect of an adjacent building which does not form part of the DesignBuilder model. The touching adiabatic component block creates an adiabatic adjacency in the part of the Building block surface in contact with the Adiabatic block.

Adiabatic component blocks are visible in [Visualisations](#), using the texture associated with the [Component block material](#) as set on the [Constructions tab](#) under the Component Block header. They can be used to cast shadows and reflect solar radiation and light in simulations by checking the [Component block shades and reflects](#) model data under Component Blocks on the Constructions tab.





Effect of Adiabatic boundaries on heat transfer

Note: While no heat is conducted across the outer side of adiabatic surfaces, these surface will still allow the flow of heat at the inside surface into the structure and so act to provide thermal mass to the connected zone. **Therefore simulation results will show heat transfer for adiabatic surfaces.** In thermally massive constructions (e.g. concrete floors) these heat flows can be quite significant over the period of hours and days, but usually less so over monthly intervals.

Note: All openings and sub-surfaces in adiabatic surfaces are ignored in EnergyPlus simulations

More information on how to work with adiabatic blocks can be found in the [Setting adjacency conditions using ground and adiabatic blocks](#) tutorial.

Assembly Block

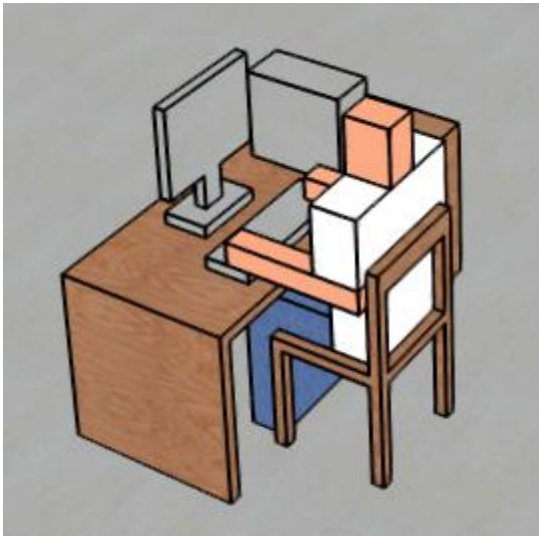
An assembly block (or just assembly) is a collection of (usually connected) [component blocks](#) used to create an object of some sort. Assemblies can be used to create internal objects such as furniture, people, stairs, or external objects such as shading systems, trees or buildings.

Custom assemblies can be created from existing component blocks that you draw and are added to the "Assembly Library" on page 646 before being placed. Once added to the library they can be loaded to the model ([placed](#)) either at building level (external assemblies) or at block level (internal assemblies).

Some typical applications of assemblies are:

1. Creating internal objects with CFD boundary conditions, such as occupants or screens with convective gains spread over their surface.
2. Creating shading devices for EnergyPlus simulations, cooling design calculations and daylighting studies.

In general, assemblies are used and behave very much like [standard component blocks](#). Like component blocks, assemblies are used at building level in EnergyPlus simulations, and at both building and block level for CFD, Daylighting and Visualisation. They can be used to shade and reflect in EnergyPlus simulations and Radiance daylighting calculations and they show up on visualisations using the texture associated with the material selected for the underlying component blocks. If different component blocks within an assembly have different materials selected then the assembly is displayed with corresponding multiple textures as shown below.



One important advantage of using assemblies over component blocks is that it is possible to edit the size or form of one instance of an assembly and all similar instances are automatically updated at the same time. This allows a large number of geometries to be edited in once go which can be useful for quickly editing large amounts of façade geometry.

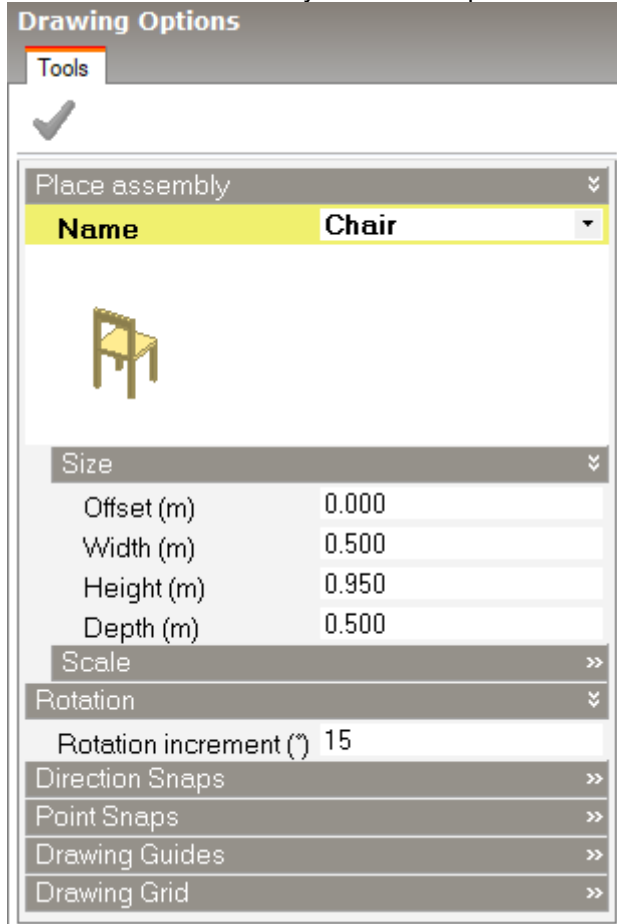
Note: To edit the form of an assembly so as to make the same edit to all other similar assembly instances you must first navigate down to the assembly and edit the form of the underlying component block(s). The [Add surface](#), [Drag face](#) and the [Cutting](#) block edit tools can be used in this case. It is possible to edit individual assemblies without affecting similar instances by making the edit using the [Stretch](#) tool while at building level. This is the only block edit tool that can be used to change the form of an assembly without affecting the other instances.

Placing Assemblies

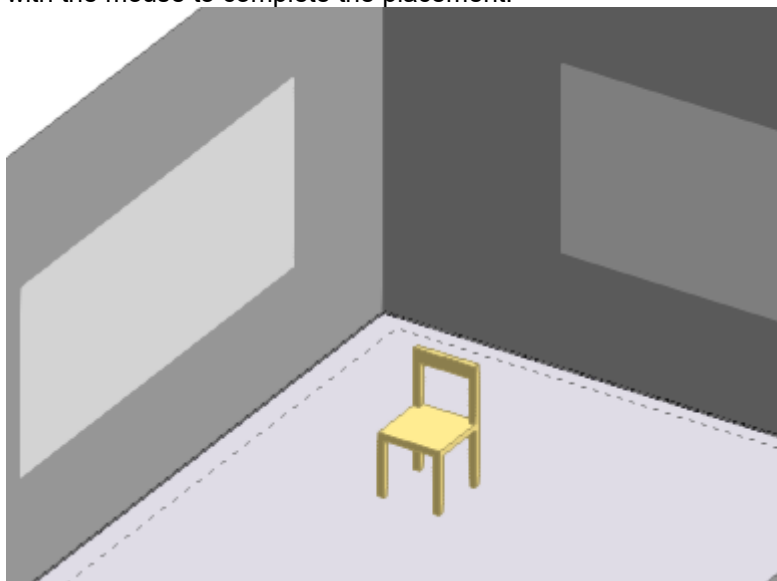
Once an assembly has been added to the library it can be placed either at building or block level. For example to place a chair assembly from the library (DesignBuilder comes with a small assembly library which includes a chair):

1. Navigate to the block where the chair is to be placed.
2. Click on the **Place assembly** icon 
3. The Drawing options panel is displayed in the bottom left of the screen.

4. Select the Chair assembly from the dropdown list of available library assemblies.



5. If you would like to change the dimensions of the chair to be placed (or its offset from the floor plane) you can do so by changing any of the Offset, Width, Height, Depth and X, Y, Z Scale settings in the Drawing options panel.
6. Move the mouse to the location in the block where the chair is to be placed. As you move the mouse around the block different selection planes become available. In this case you will place the chair on the floor. Note that the place where you click on the floor will position the part of the chair that was defined as the [attachment point](#) when the assembly was added to the library.
7. Click to define the location of the assembly attachment point.
8. Now define the orientation of the chair by moving the mouse. It is usually easiest to move the mouse away from the object being placed during this process to give you the finest control. As you move the mouse the rotation of the chair will be adjusted. Once the rotation of the chair is as required click again with the mouse to complete the placement.



9. The Place assembly command remains active allowing you to place more assemblies. To close the command press the <Esc> key.

See also:

- "Assembly Library" on page 646
- "Component Blocks & Assemblies as CFD Boundaries" on page 1922

Block Forms

While drawing a block perimeter, you can select the block form from the Drawing options data panel in the bottom left of the screen. The standard block forms are provided to facilitate the creation of blocks representing specific building geometries. You can select from options 1-4 below:

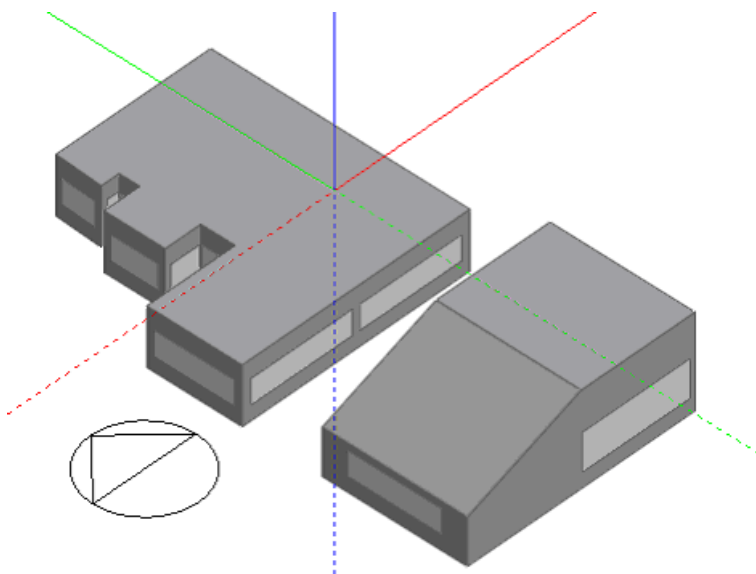
- [1-Extruded](#)
- [2-Sloped wall](#)
- [3-Pitched roof](#)
- [4-Dome](#)
- [\(5-General\)](#)

The **5-General** block form is not selectable but rather is defined internally by DesignBuilder following block geometry operations such as [Cut block](#), Horizontal block extrusion, [Drag face](#) and [Boolean geometry operation](#).

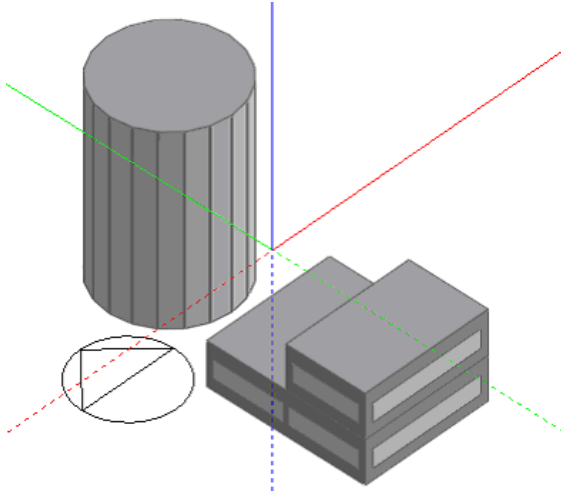
Once a block has been created, it is possible in most cases to change to a new form by navigating to the block and opening the [Model options](#) dialog. When at block level the Model options dialog opens at the [Block tab](#) and there the Block form can in most cases be changed.

Extruded Block Form

The Extruded [block form](#) is the most commonly used geometry. You create an extruded block by drawing a [perimeter](#) using a number of pre-defined shapes and then extruding the perimeter by whatever distance is required. If you use the polygon perimeter [shape](#) you can create extrusions in both horizontal and vertical planes (or even in sloped planes).



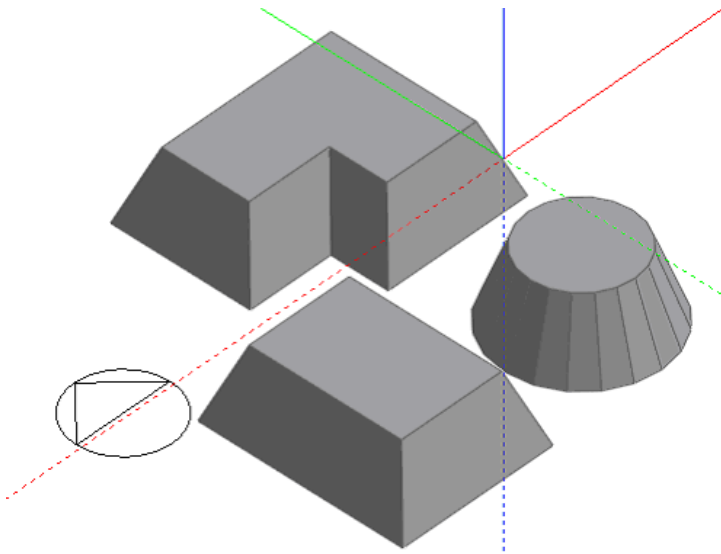
The various methods of drawing general extrusions are explained under [Adding Blocks](#). Circular and rectangular perimeter [shapes](#) are used to create cylindrical and rectilinear prismatic extrusions from horizontal planes:



See also the [Blocks and dimensioning protocols](#) tutorial.

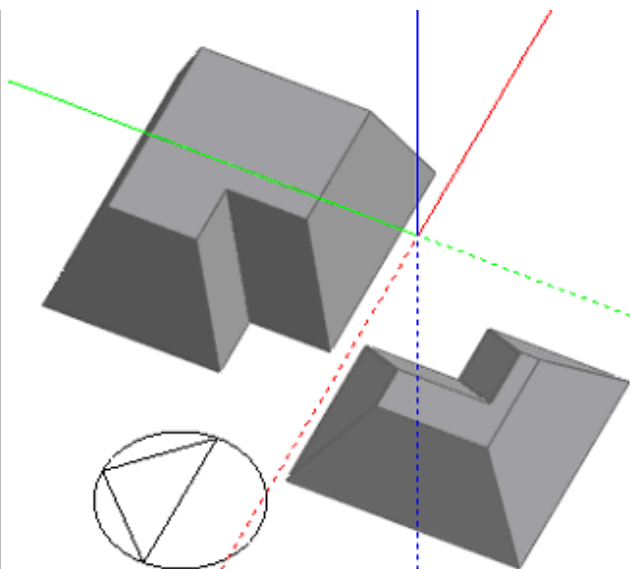
Sloped Wall Block Form

The Sloped wall [block form](#) is used to create blocks on horizontal planes with uniformly sloping walls. Sloped wall blocks can be created using any perimeter [shape](#) (polygon, rectangle or circle):



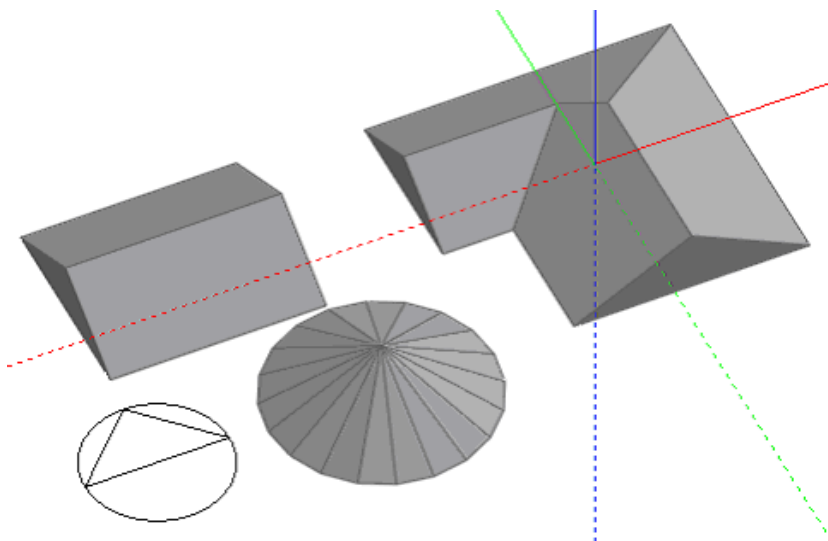
The wall angle is defined using the **Slope** setting in the Drawing options panel and the blocks are automatically truncated by the **Height** setting:

Geometry	
Block type	1-Building block
Type	2-Sloping walls
Slope (°)	50
Height (m)	3.00
Perimeter	
Shape	1-Polygon
Line type	1-Straight line
Thickness (m)	0.292
Protractor >>	
Direction Snaps >>	
Point Snaps >>	
Drawing Guides >>	

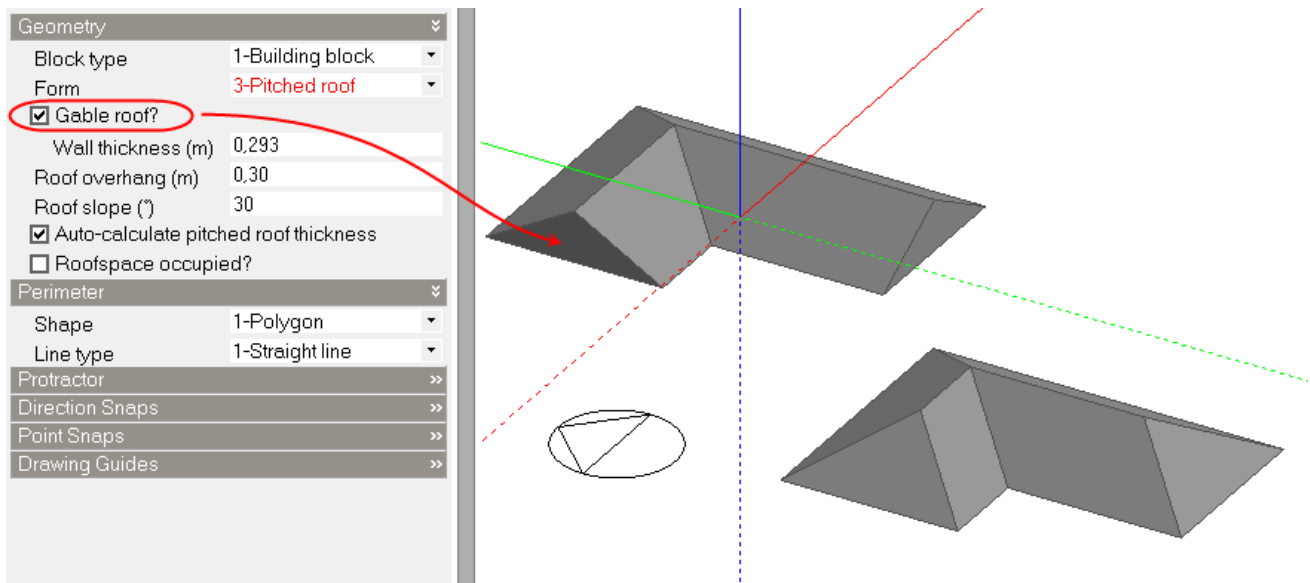


Pitched Roof Block Form

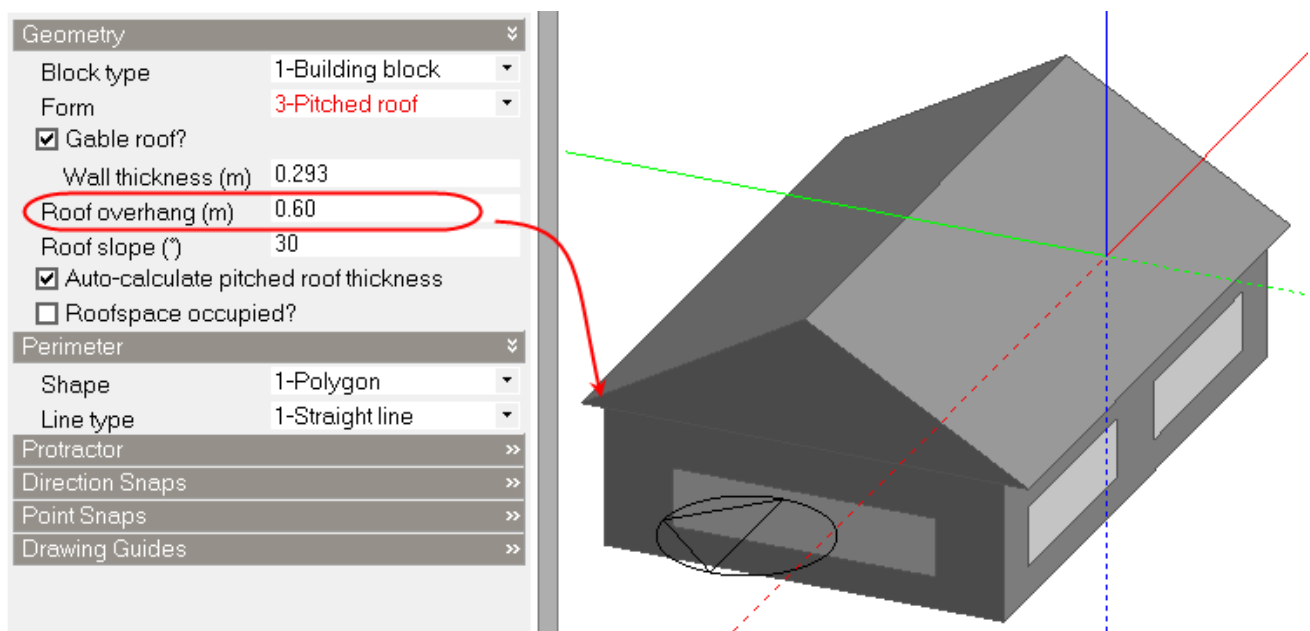
The pitched roof [block form](#) automatically generates a pitched roof from a perimeter drawn on a horizontal plane. Pitched roof blocks can be created using any perimeter shape (polygon, rectangle or circle):



The slope of the roof is defined using the **Roof slope** setting in the Drawing options data panel. A pitched roof can be created with hips or gable ends using the Gable roof setting:



If you draw a pitched roof on top of an existing block, you can control the overhang or the depth of the roof eaves using the **Roof overhang** setting:



See also the [Adding a roof, construction lines](#) tutorial

Voids

Where roofs are to be created on top of conventional blocks that contain voids and the voids are to be automatically incorporated within the pitched roof, the [Automatically include voids from lower blocks](#) Model options switch must be switched on before drawing the roof perimeter.

Pitched Roof Drawing Options

Gable roof

Check this option if the roof is to have vertical gable ends (see above). If this option is not checked then the roof geometry will be a 'hip roof'.

Roof overhang

The extent to which the roof forms an overhang over the zones below. If the [Auto-calculate pitched roof thickness](#) option is checked then the overhang displayed in the model and also the overhang modelled in the simulation will be exactly the value of Roof overhang entered here.

Roof slope

The slope of the roof to be generated measured from horizontal.

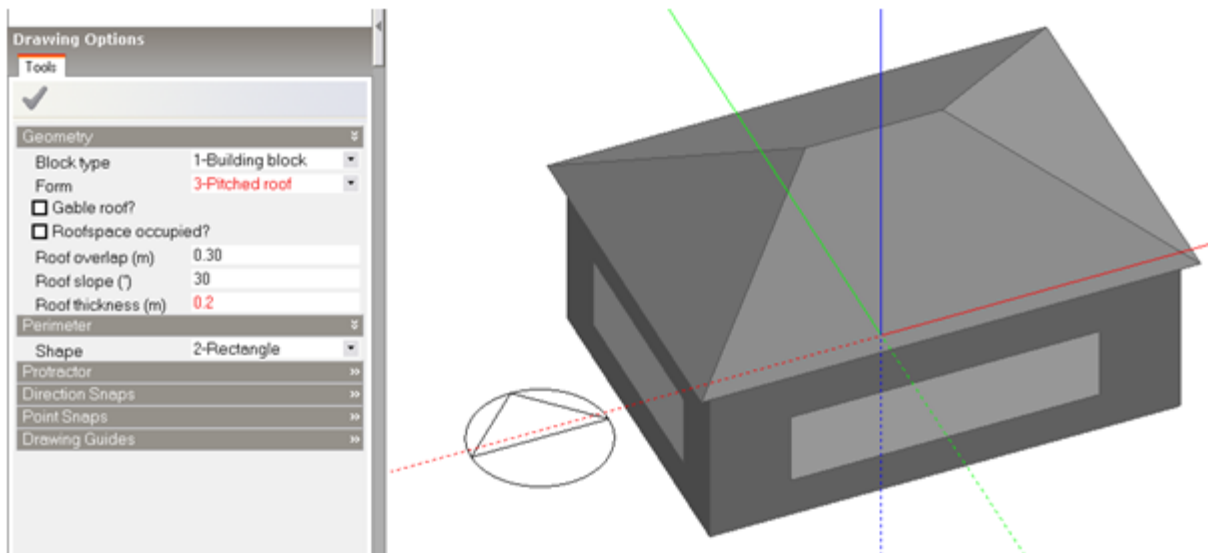
Roofspace occupied

If the roofspace is flagged as being unoccupied (this option is left unchecked) then a series of changes are made to the block attributes:

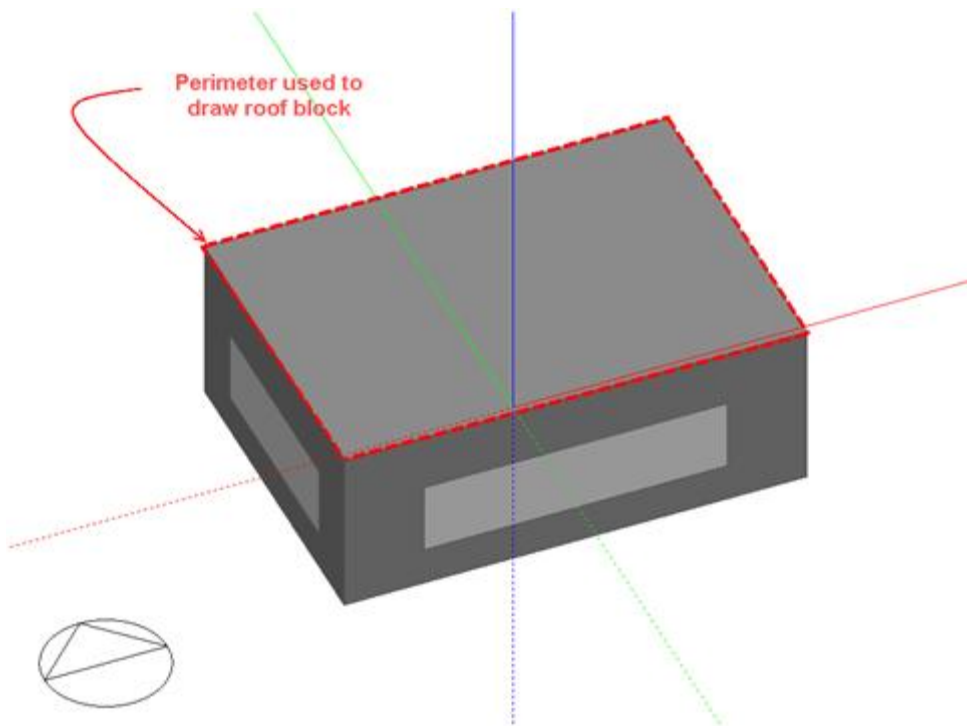
- Any glazing that would otherwise have been included on the gable end walls is removed.
- [Zone type](#) is set to **2-Semi-exterior unconditioned**.
- Infiltration in the roofspace has a default value of 1 ac/h when using [Scheduled](#) natural ventilation or 'Very poor' sealing quality when using [Calculated](#) natural ventilation.

Automatic Pitched Roof Geometry

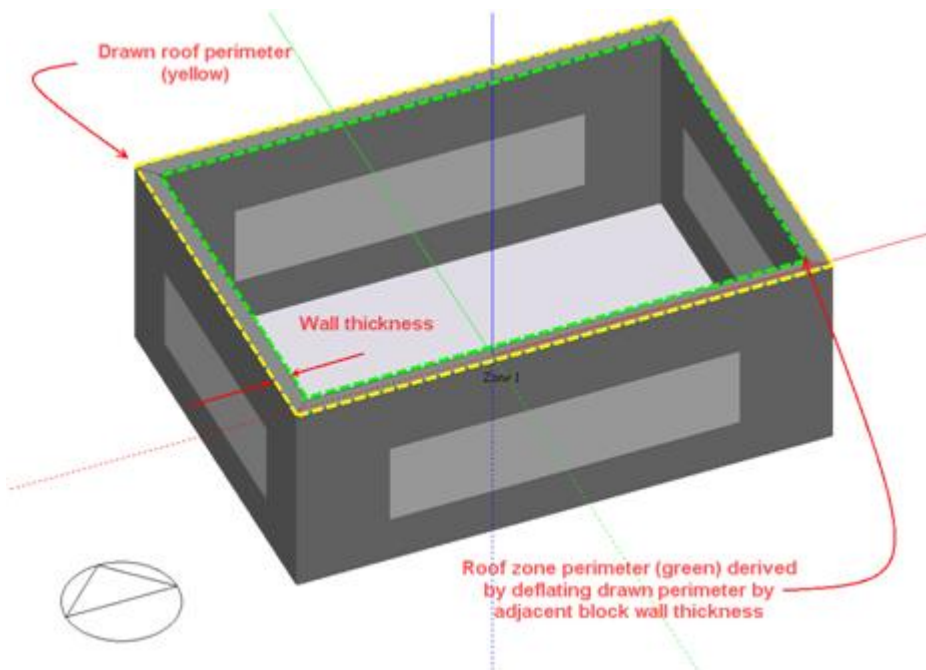
This section describes the automatic DesignBuilder pitched roof geometry generation with reference to an example roof created by drawing a rectangular perimeter with a roof overhang of 0.3m and a roof slab thickness of 0.2m:



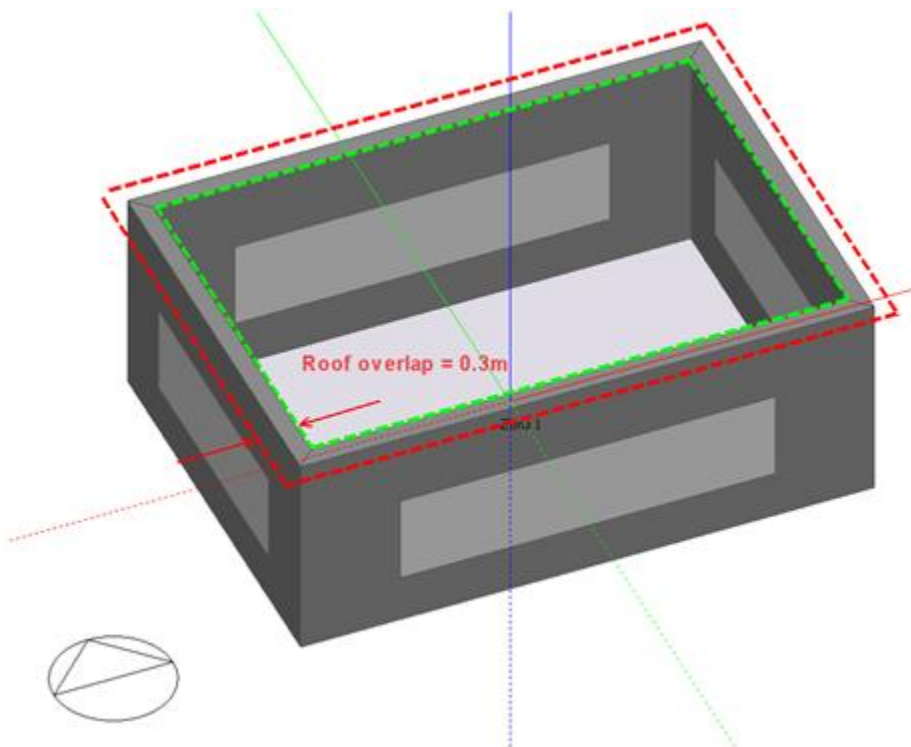
Roof perimeters are drawn using the perimeters of lower adjacent blocks:



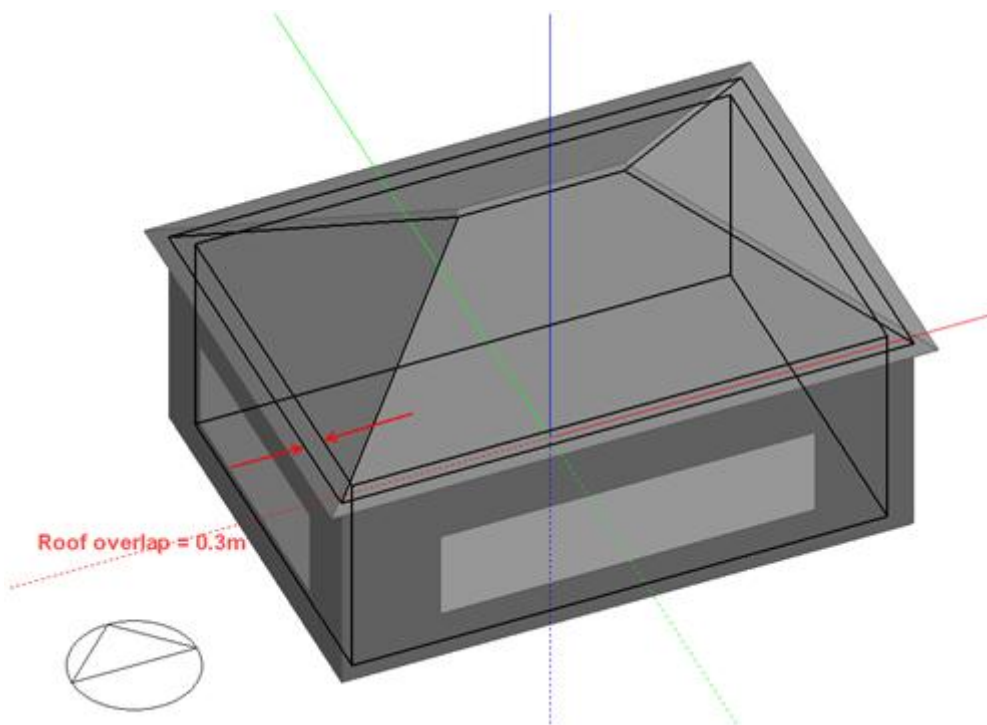
The software first creates the roof zone (this would be the zone written to the EnergyPlus IDF file) by first identifying the wall thickness of the adjacent lower block and then deflating the drawn perimeter by this thickness to obtain the basic roof zone perimeter. The resulting roof zone perimeter is then guaranteed to coincide with the lower adjacent zone perimeter(s) in the absence of a roof overhang:



If a roof overhang is specified, the basic roof zone perimeter is then inflated by the roof overhang:



The roof zone is then completed by identifying the skeleton of the perimeter and applying the specified slope to the resulting segments. The base of the roof zone now overlaps the lower adjacent zone(s) by the specified roof overhang:



Note: In the case where a roof perimeter is drawn on top of several blocks having different wall thicknesses, the software identifies the wall thickness having the longest perimeter and uses that thickness to derive the roof zone perimeter. You should be aware that in this case the specified roof overhang will only be guaranteed where the roof overlaps blocks having the same wall thickness as that identified for the roof zone perimeter.

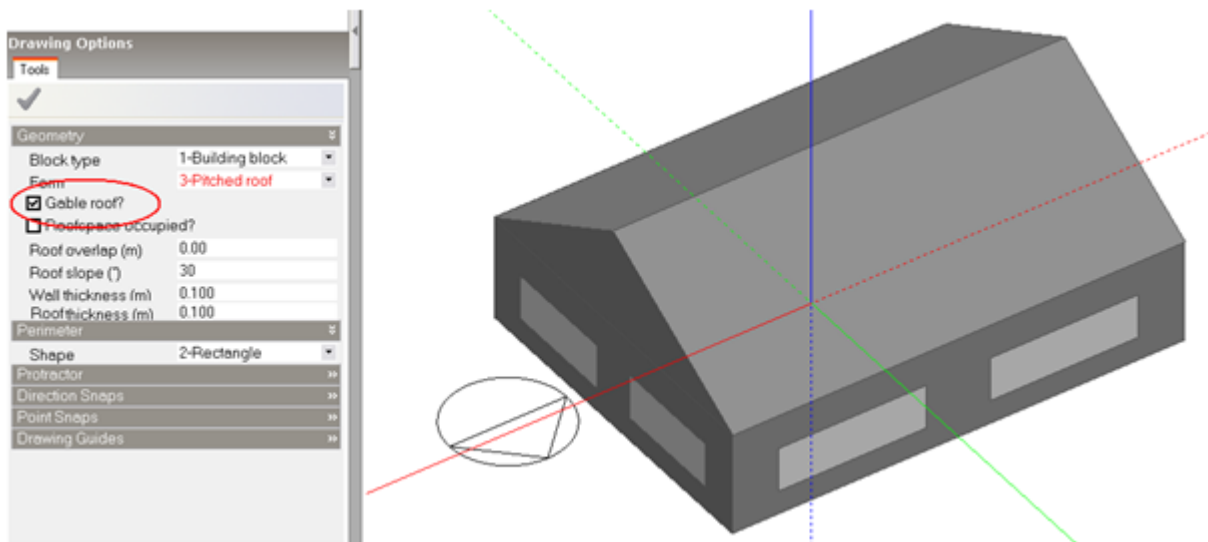
The roof building block slabs are then created by extruding the roof zone surfaces outward along a normal to the surfaces by a distance specified using the Roof Thickness setting.

The perimeter of the resulting roof block will extend beyond the roof zone perimeter by a distance (d in the illustration) determined by the specified roof thickness:

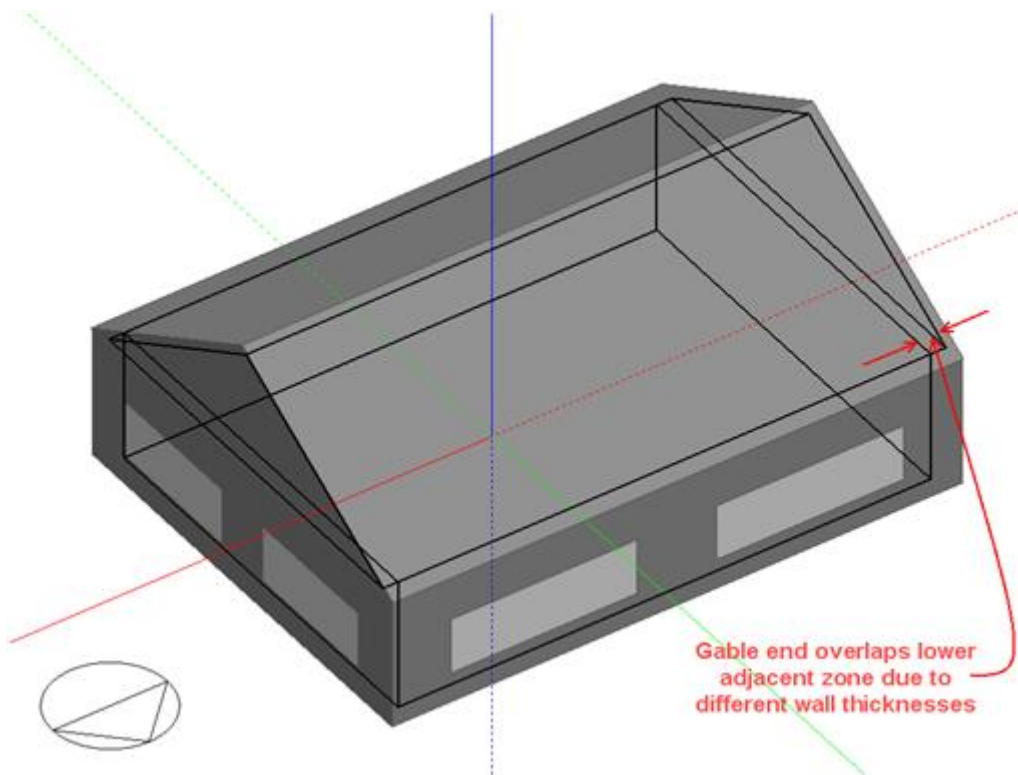
Note: The overhang created by the formation of the roof slabs (d in the above illustration) is only noticeable in the Edit and Visualisation views and will not be written to the IDF file i.e. unlike the roof zone overlap, it has no effect on the EnergyPlus shading calculations.

Gable roofs

If the Gable roof option is selected, all gable end roof segments (isosceles triangles) are generated to fit flush with the lower adjacent block surface:



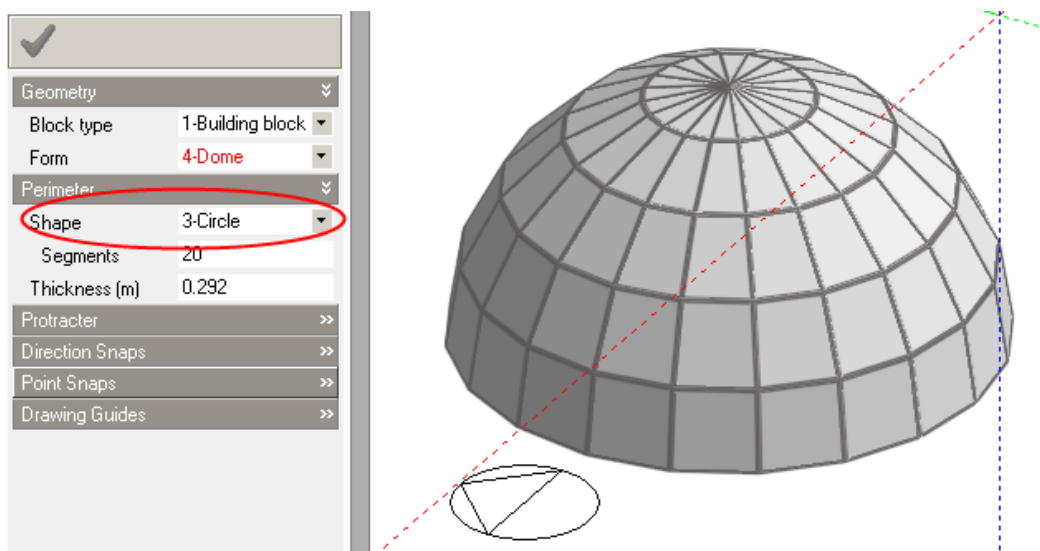
If the specified gable wall thickness is different to the wall thickness of the lower adjacent block, an overlap will be created between the roof zone and the lower adjacent zone, which may affect shading calculations in EnergyPlus:



Note: Pitched roof form blocks are automatically configured so that the external walls are not shown on the Edit screen when at block level. This can help when drawing partitions, positioning assembly blocks etc. The option can be reversed using the Display external walls Block display model option if required.

Dome Block Form

The dome [block form](#) is used to create hemispherical blocks and can only be created using the circular perimeter shape:



More information on using the Dome block form can be found in the [Drawing arcs, circles and domes](#) tutorial

General Block Form

When a block is edited in such a way that it no longer fits with one of the standard block forms ([Extruded](#), [Sloped wall](#), [Pitched roof](#) or [Dome](#)), it is labelled by DesignBuilder with the **5-General** block form.

Blocks having the General block form can still be transformed back to have the Sloped wall or Pitched roof form (but not the Extruded form) on the [Block tab](#) of the Model options dialog.

Note: General form blocks are automatically configured so that the external walls are not shown on the Edit screen when at block level. This can help when drawing partitions, positioning assembly blocks etc. The option can be reversed using the Display external walls Block display model option if required.

Height

After drawing a vertically extruded block perimeter on a horizontal plane, the block will be automatically extruded using the Height setting, if the [Auto-Complete Block](#) option is switched on or the [View Rotation](#) has been set to 'Plan'.

See also the [Blocks and dimensioning protocols](#) tutorial

Auto-Complete Block

After drawing a vertically extruded block perimeter on a horizontal plane, the block will be automatically extruded by the [Height](#) setting if the Auto-Complete Block option is switched on or the [View Rotation](#) has been set to 'Plan'. With this option switched off (or in the case of a non-horizontal drawing plane), the perimeter can be dragged along an axis normal to the perimeter to obtain the required extrusion.

See also the [Blocks and dimensioning protocols](#) tutorial

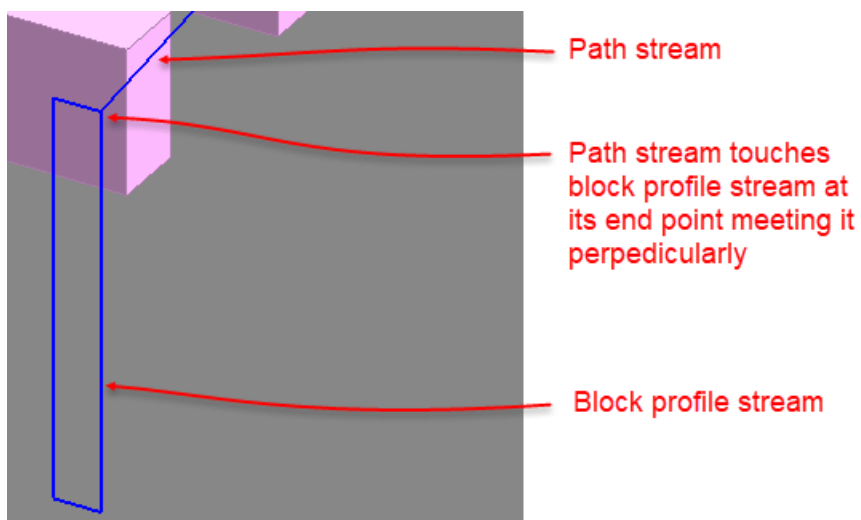
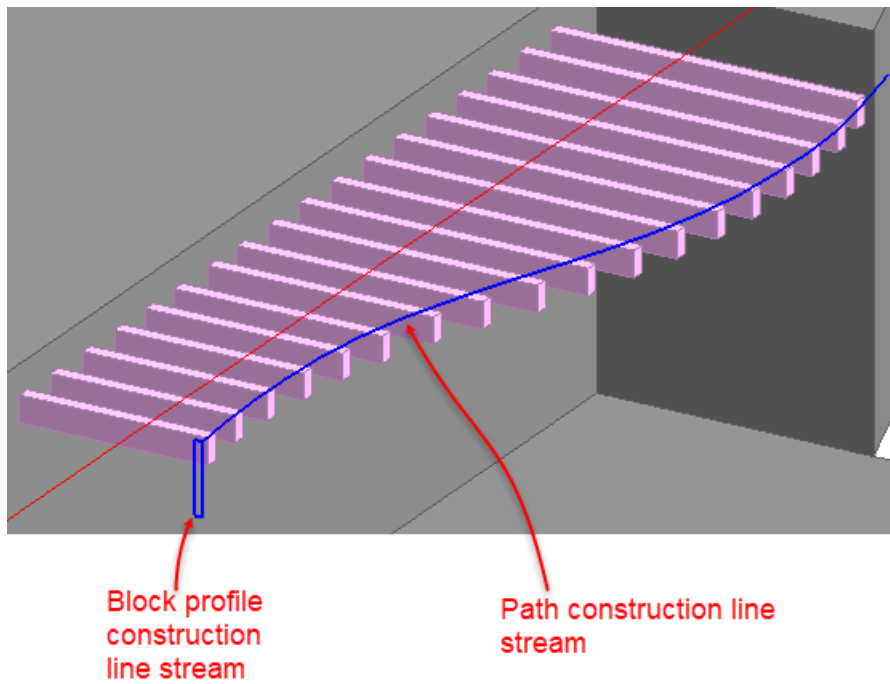
Follow Path

The **Follow path** tool allows a new block to be added to the model by drawing a block profile and extruding it along a path to create the 3D form. Both the block profile and the path are created by drawing [construction line streams](#).

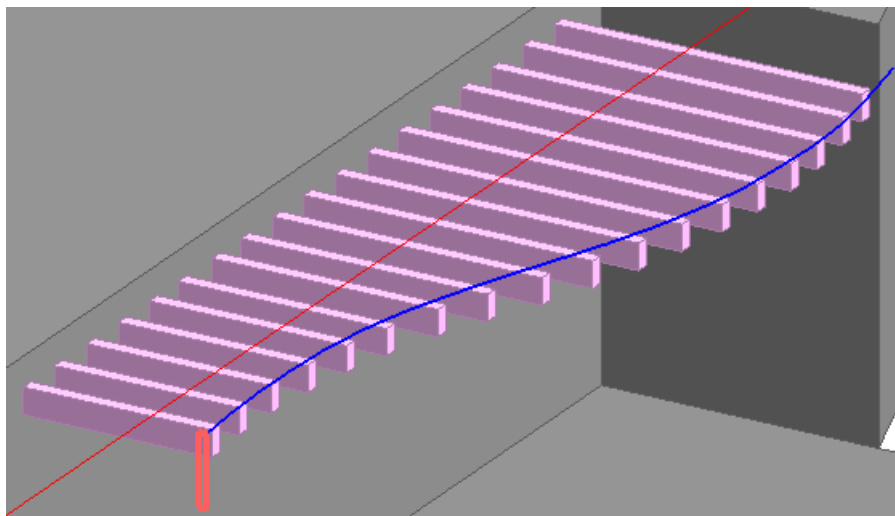
The **Follow path** tool is similar in concept to the [Drag face](#) tool but it can also turn corners and follow any non-intersecting path, so curved block shapes or wrap-around shades can be created.

The process is described below using an example to illustrate:

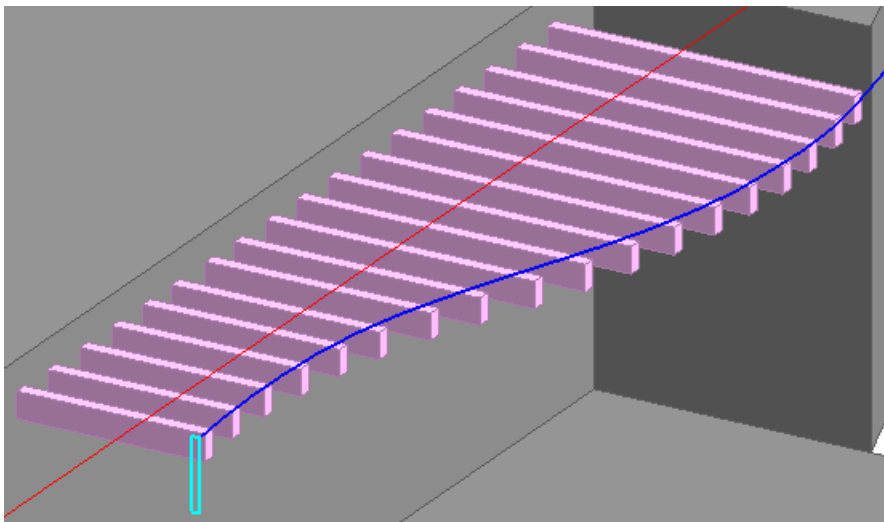
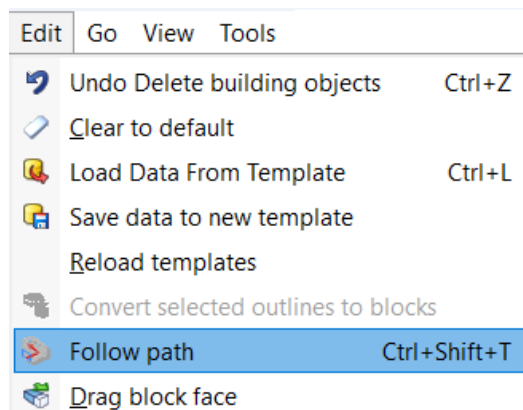
1. Start at building level.
2. Draw a closed loop using a construction line stream to represent the block profile. The closed loop block profile must be orientated vertically.
3. Draw a second construction line stream to represent the path that the block profile will be extruded along. The path stream must touch the block profile stream at its end point and lead away from the face in a perpendicular direction following a horizontal route.



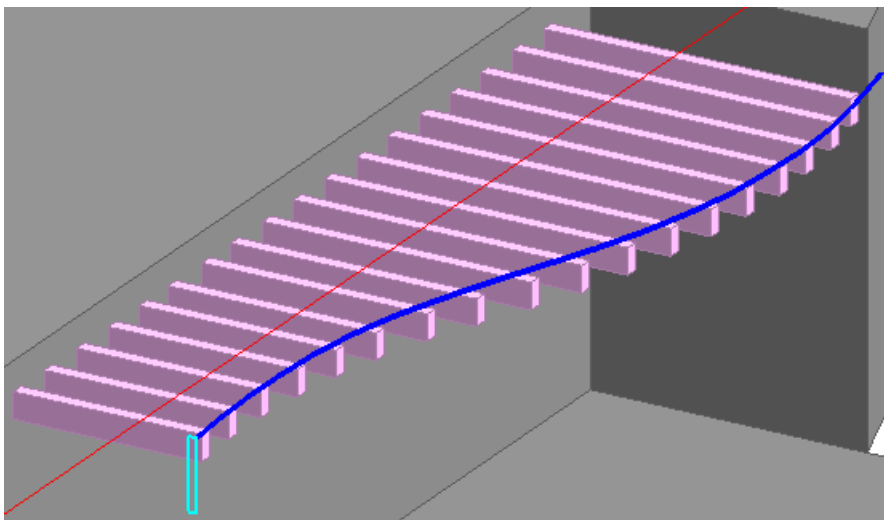
4. Once both the block profile and path streams have been created, select all of the construction lines in the block profile stream. It is usually easiest to do this using a selection rectangle.



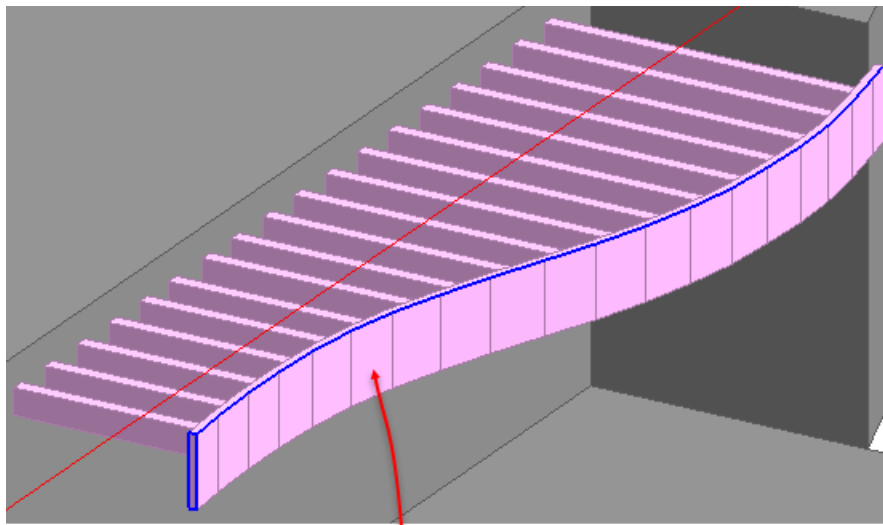
5. Select the **Edit > Follow path tool** menu option or use the <Ctrl + Shift + T> keyboard shortcut. The block profile construction lines will turn from dark blue to light blue (cyan).



6. Select the path construction line stream by clicking on the stream once with the mouse. When selected, the stream of construction lines representing the path will be displayed as a thicker dark blue line.

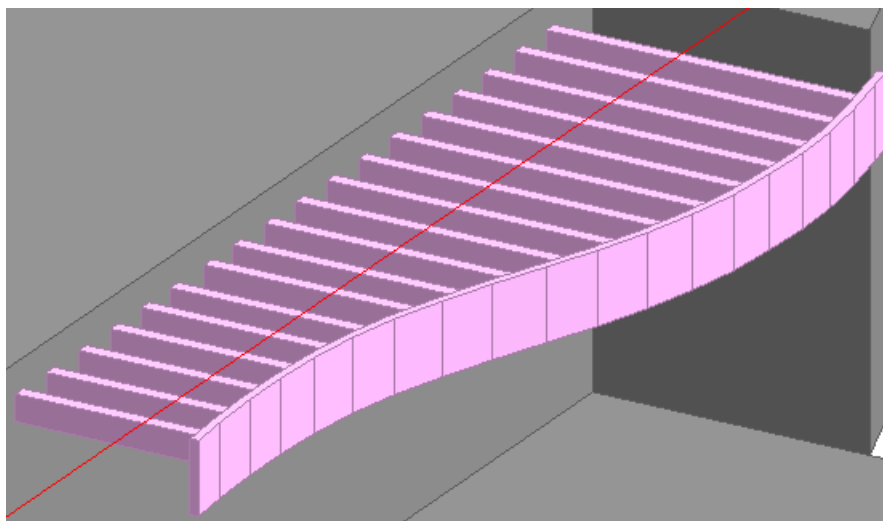


7. Select the type of block to create in the Drawing options panel. In this example the **3-Component block** option is selected.
8. Click with the mouse anywhere on the model to create the new block.



Newly created curved component block

9. Once the block has been created the construction lines are no longer needed and can be deleted.



Tip: Although the current **Follow path** tool only works for vertical closed loop profiles and the path needs to be horizontal, you can rotate the horizontal path extrusion to any required orientation.

Note: **Follow path** requires more than one construction line segment to be present in the path stream, i.e. it cannot just be a straight line.

View a [video showing how to use the Follow Path tool](#) to create a roof shading surface.

Edit Blocks

The geometry and other attributes of blocks can be modified using the following tools:

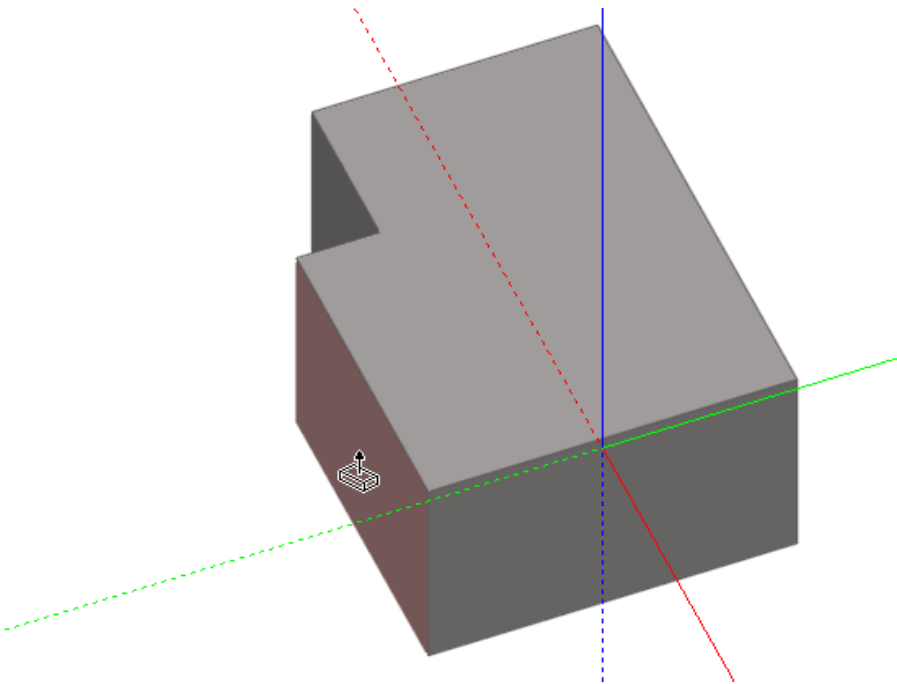
- [Drag block face](#)

- [Divide block using cutting tool](#)
- [Boolean Geometry Operations](#)
- [Edit Block Geometry Dialog](#)
- [Convert Outline Block](#)
- [Transfer Blocks](#)
- "Add Surface" on page 246

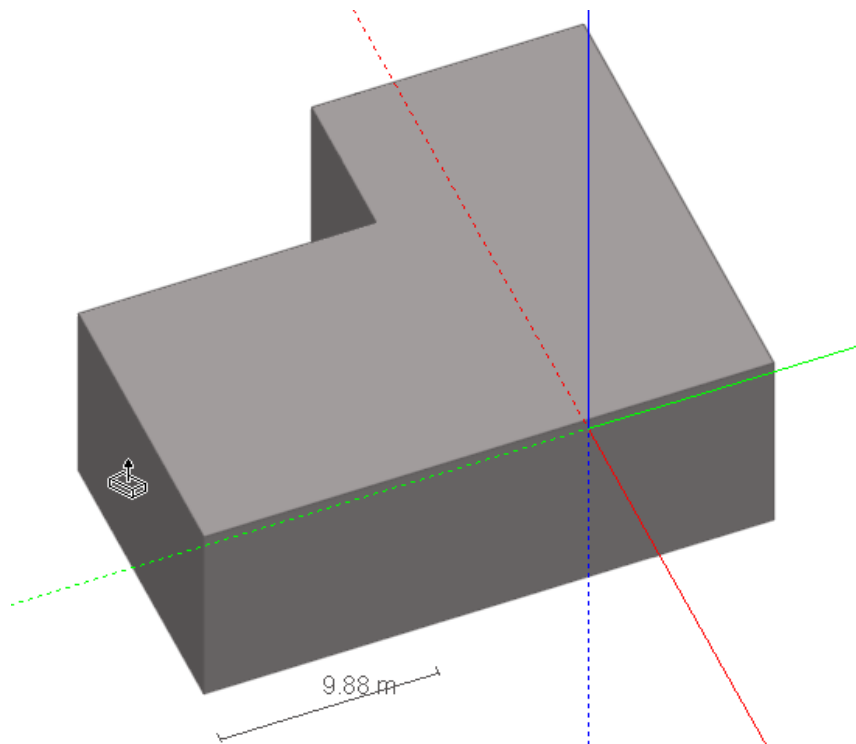
Drag Face



The topology of a block can be edited using the **Drag face** tool which allows you to select a block surface and drag it along an axis, normal to the surface. To edit a block topology first go to the building level and click on the **Drag face** tool. Then move the mouse cursor to the block face that you want to move. As you move the cursor across the block surfaces, each surface will highlight to help in selecting the required surface. When the mouse cursor is over the required surface, click to start dragging it:



Drag the surface using the mouse or key-in the distance to shift the surface by the required amount:



See also the [Stretch, drag face and rotate tools](#) tutorial.

Divide Block Using Cutting Tool



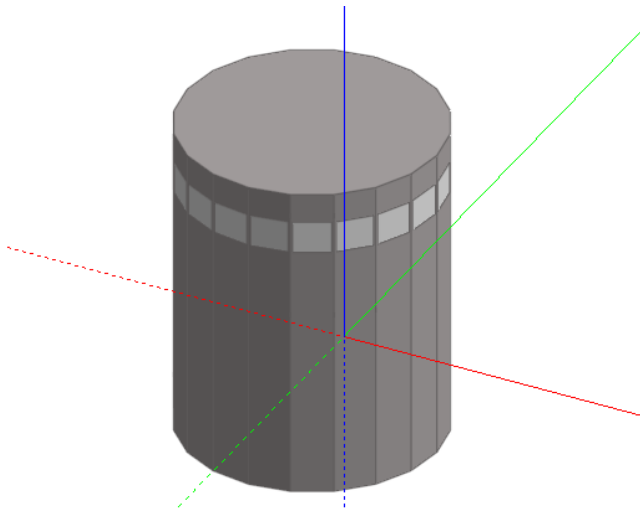
Blocks can be cut (i.e. divided into 2) using the **Divide (Cut) Block** Tool. To cut or divide a block first go to the building level, select the block that you want to modify and click on the Divide block toolbar icon. Blocks are divided using a plane which can be defined using one of 2 methods:

- **1-Draw plane** - you define the plane by clicking on 3 points.
- **2-Select plane** - you define the plane by selecting an existing plane in the model.

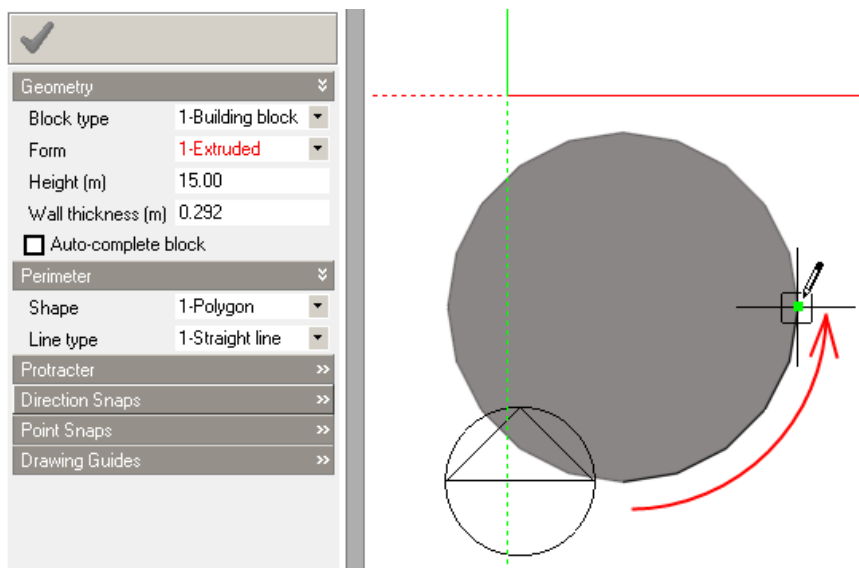
See also the [Cutting blocks, protractor tool](#) tutorial

Draw Plane Method

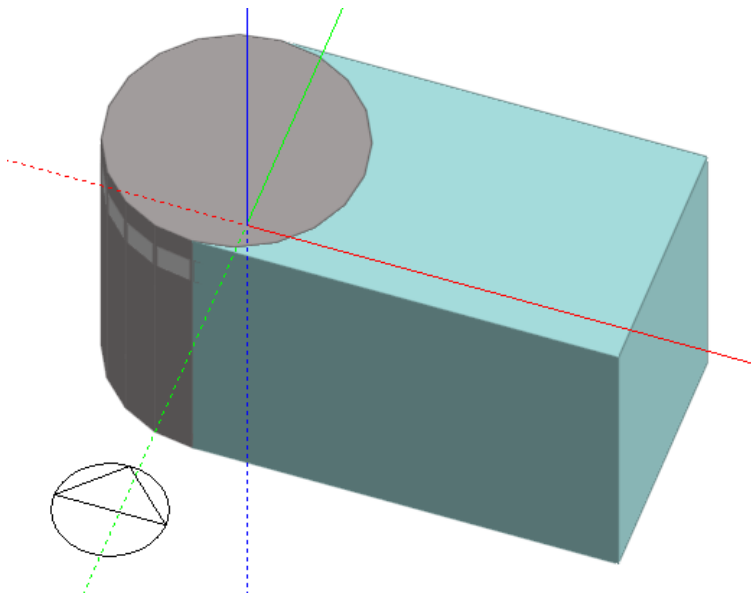
The **Draw plane** cutting method involves drawing two lines which define the plane. For example to cut a pitched roof to fit flush with a rotunda:



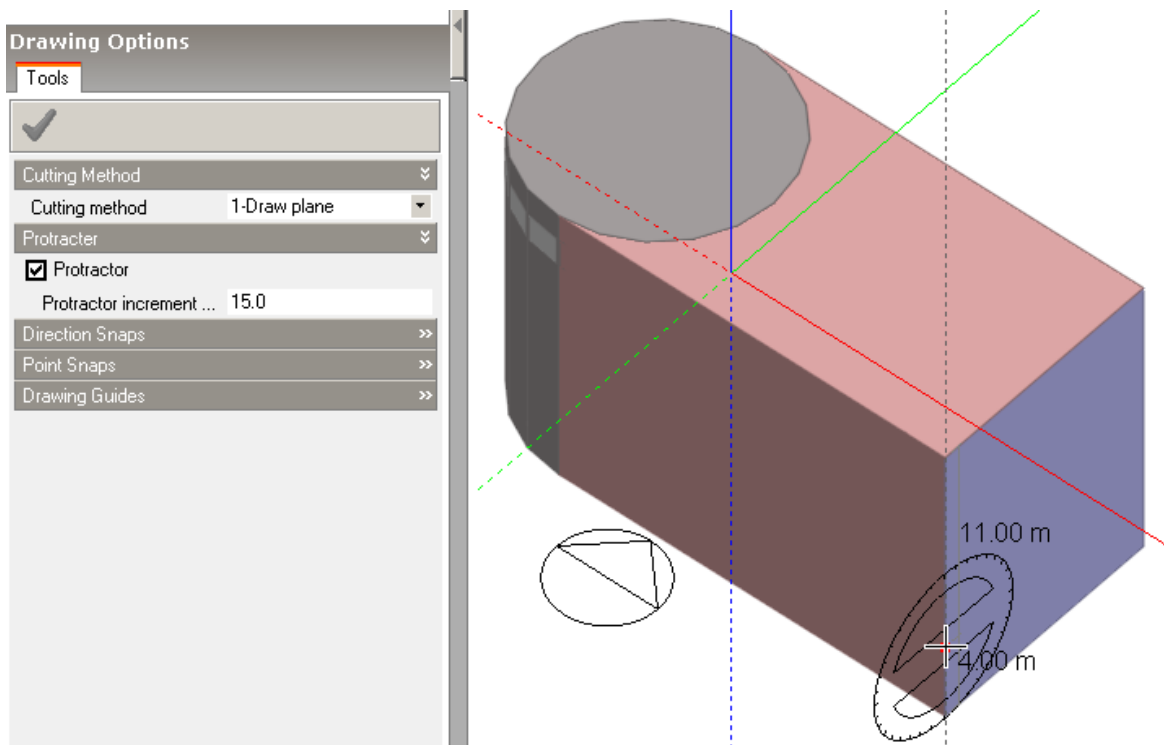
Because the geometry is fairly complicated, it's advisable to use an [outline block](#) to generate the intermediate geometry and then convert the block to a [building block](#) after the editing process is complete. So, from the building level, click on the [Add new block](#) tool, select **2-Outline block** as the block type, set the perimeter shape to **1-Polygon** and the [Height](#) to the same height as the rotunda, then draw a conventional vertical extrusion to fit flush to the rotunda. In order to trace round the base of the rotunda, it's probably easier to set the view rotation to 'Plan', click on to the ground plane and press SHIFT to lock the plane before placing the first perimeter point:



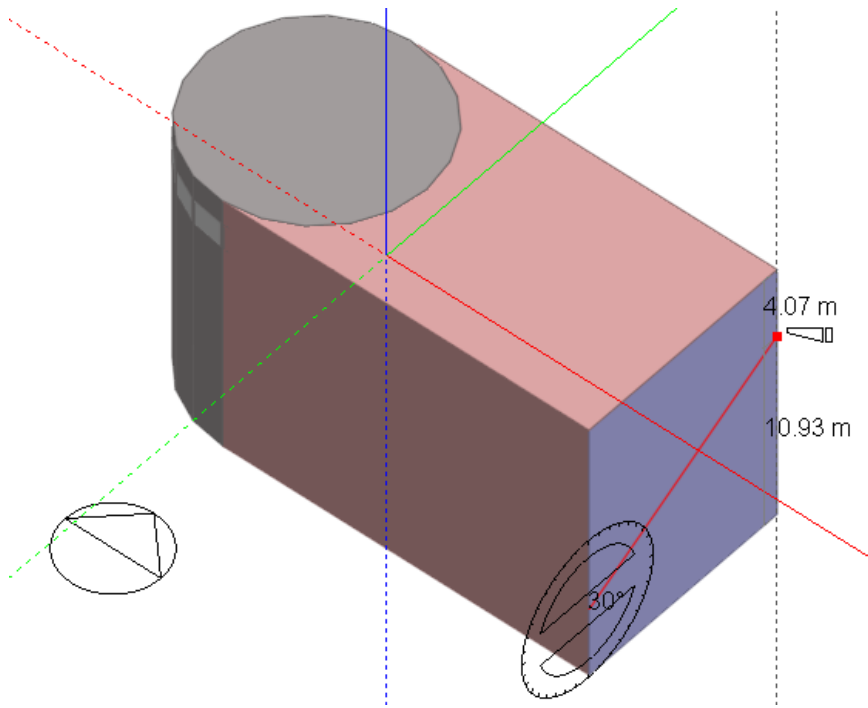
When tracing around a perimeter by snapping from end-point to end-point, it can be helpful to switch off the [direction](#) snaps and the [drawing guides](#) to prevent them from interfering with the point snaps. Completing the perimeter:



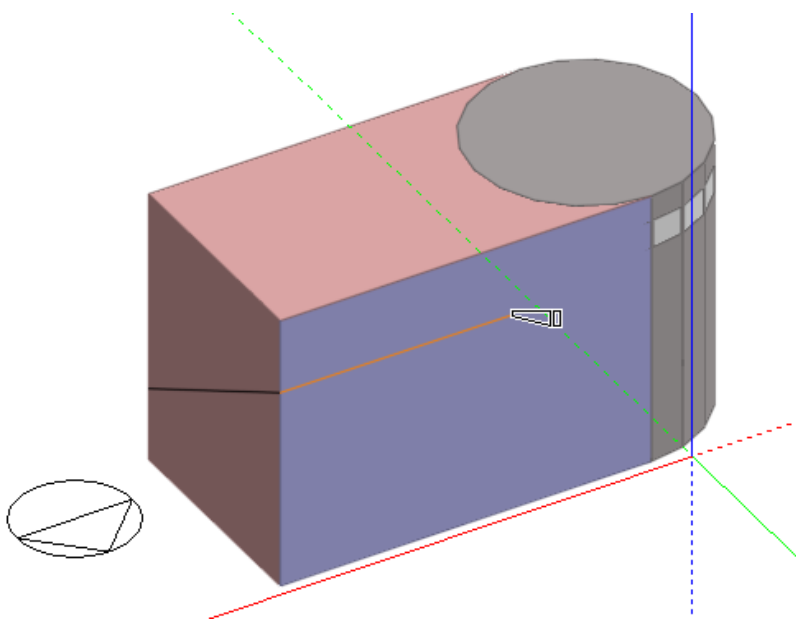
Click on the outline block to select it and then click on the 'Divide block' tool and make sure that the 'Cutting method' is set to **1-Draw plane** in the Drawing Options data panel. Switch the [protractor](#) on and move the cursor over the end face of the selected block, the surface will highlight to indicate that it's become the active drawing plane. Press the SHIFT key to lock the drawing plane and then locate the cursor 4m from the base of the block using [edge snap](#):



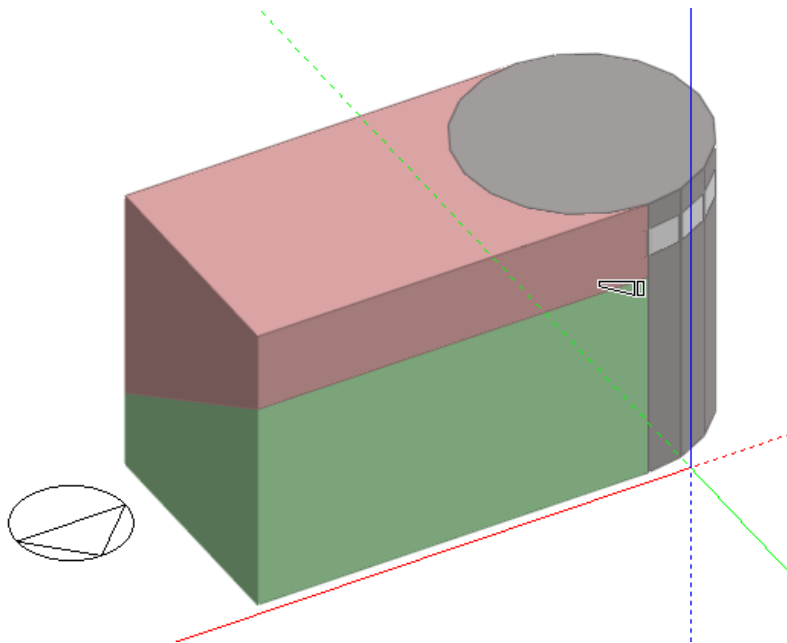
Click to place the beginning of the first cutting plane line, then place the end of the line to intersect the face edge with a pitch angle of 30°:



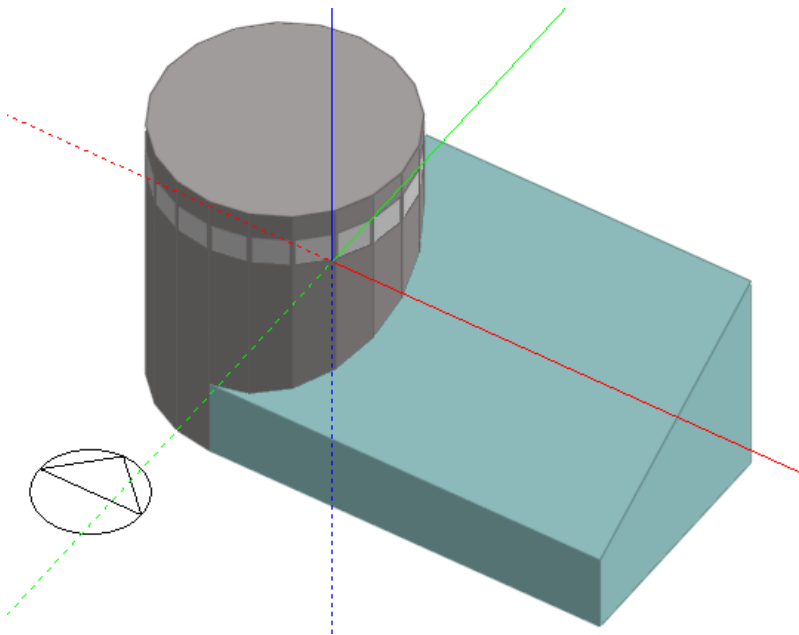
[Orbit](#) to make the back face of the block visible. Move the mouse cursor over the back face and when it highlights, press the SHIFT key to lock the plane. Place the second cutting plane line to snap to the local X-axis of the back face:



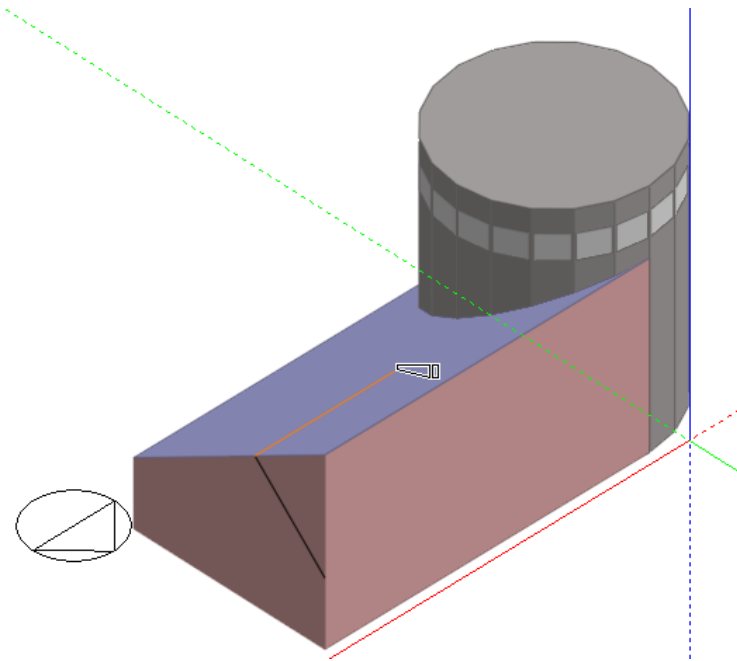
Click to complete the cutting plane. The blocks that will be produced by the cut are displayed in red and green, depending on which side of the cutting plane they lie. At this point, you can choose to cancel the cut by pressing the ESC key or selecting 'Cancel Cut' from the right-click menu.



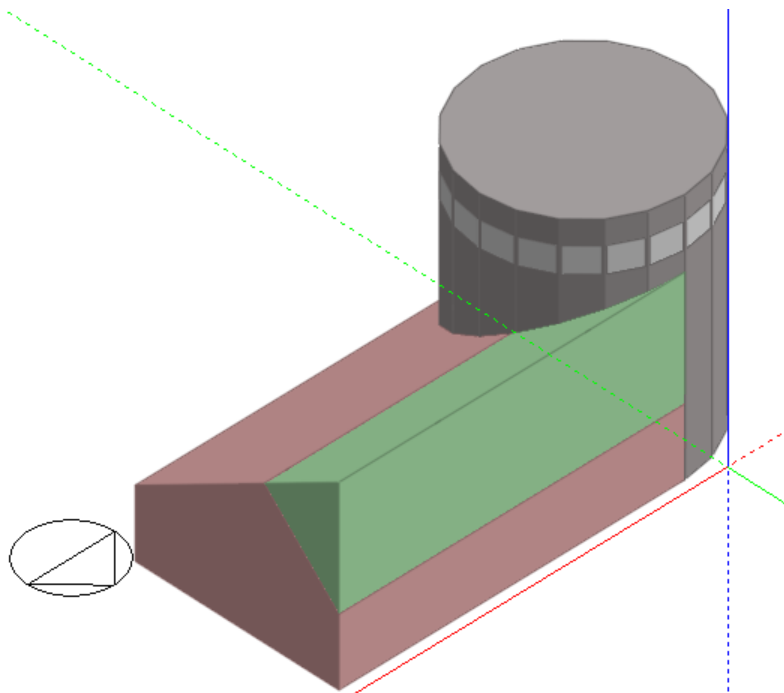
Click again to complete the cut and then select and delete the block that isn't needed:



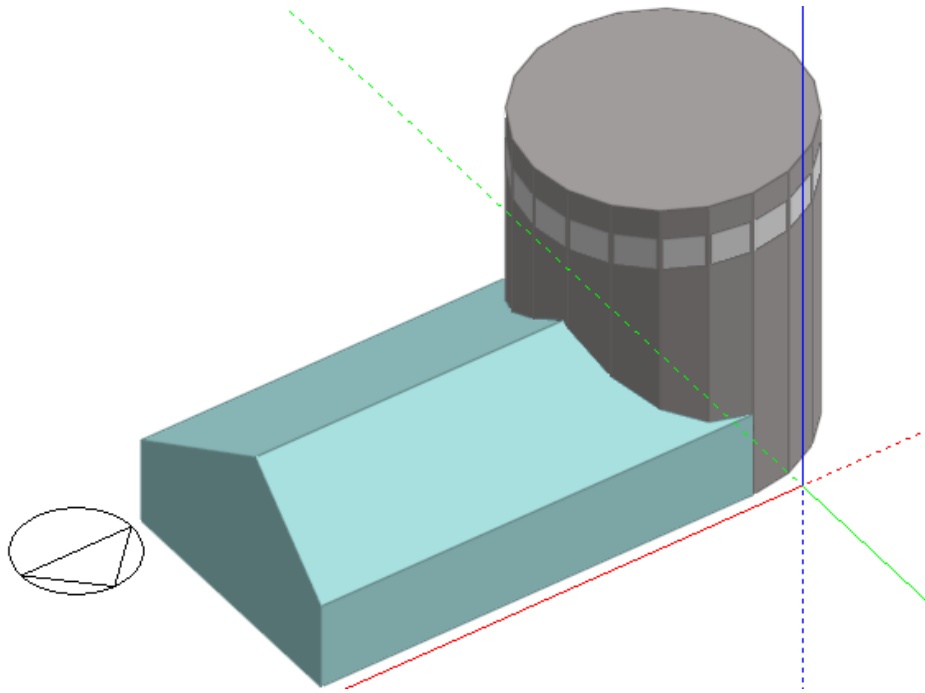
Go through a similar process to create the other pitch. You may need to switch off the [drawing guides](#) and [mid-point snap](#) to prevent them from interfering with the [protractor](#) and [edge snap](#):



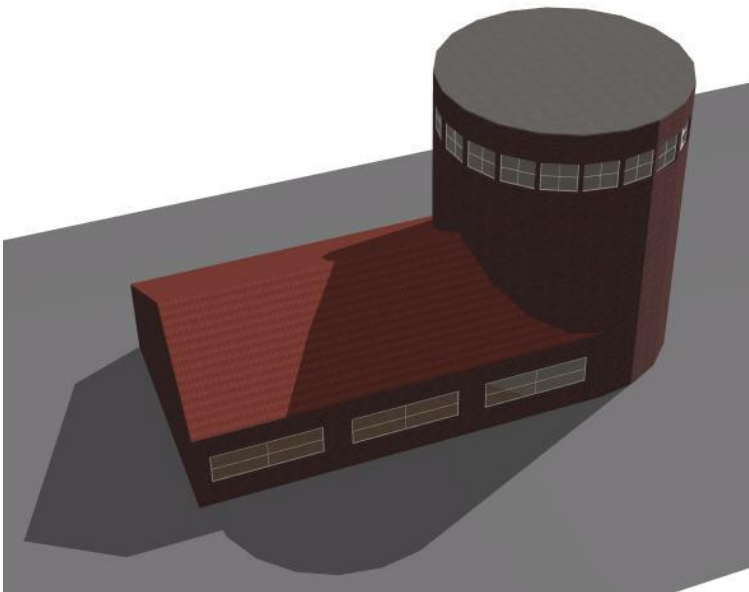
Click to create the cutting plane:



Click again to make the final cut and then select and delete the block that isn't part of the finished structure:

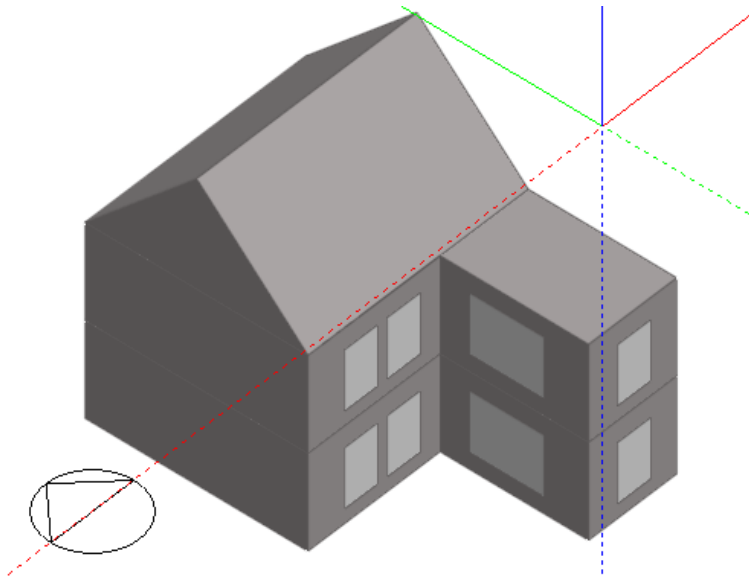


The outline block can then be selected and converted to a building block:

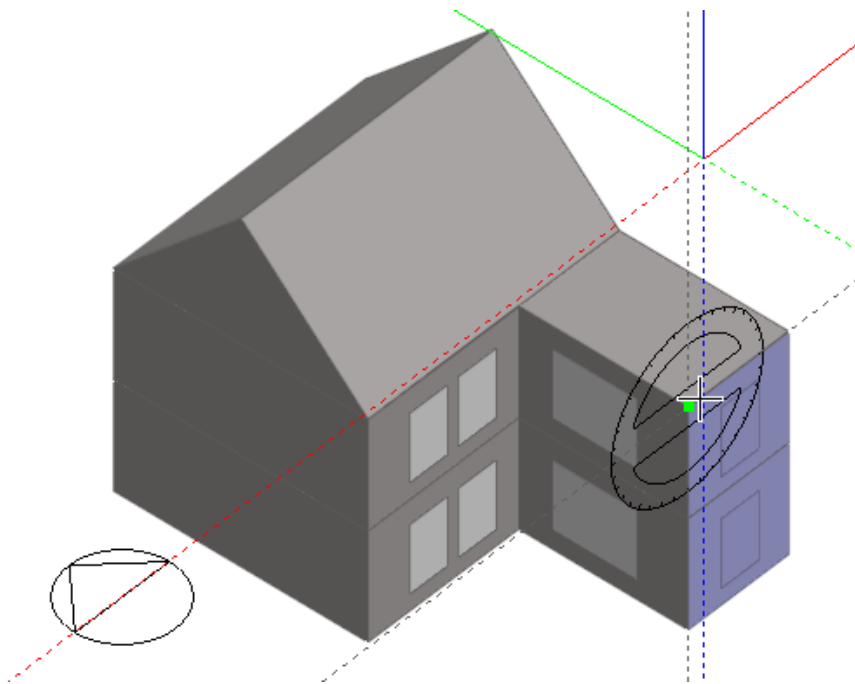


Select Plane Method

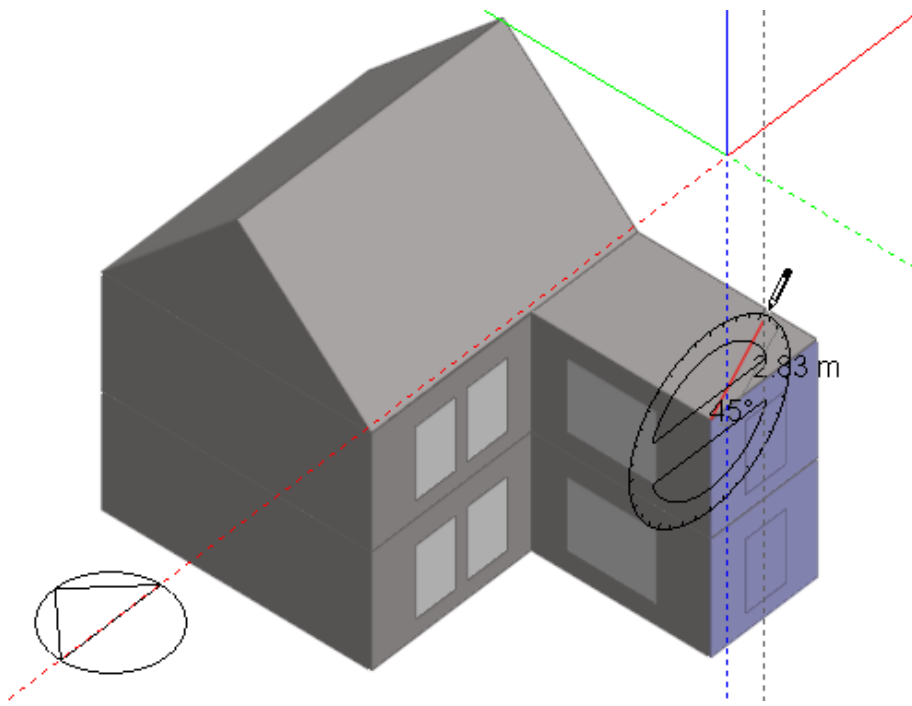
The Select plane method of cutting a block uses an existing plane to define the cutting plane. It can be demonstrated by modelling a roof offshoot:



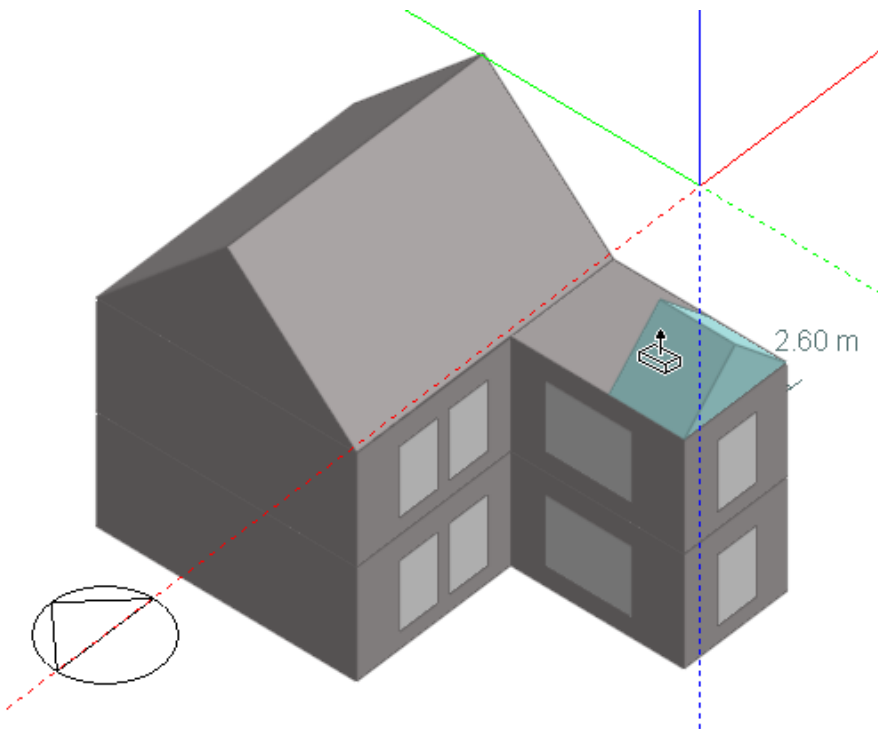
Click on the [Add new block](#) tool, select [2-Outline block](#) as the [block type](#) and set the [perimeter shape](#) to **1-Polygon**. Switch the [protractor](#) on and move the mouse cursor to the back face of the offshoot. When the cursor is over the back face, press the <Shift> key to lock the drawing plane and snap the protractor to the upper edge end-point:



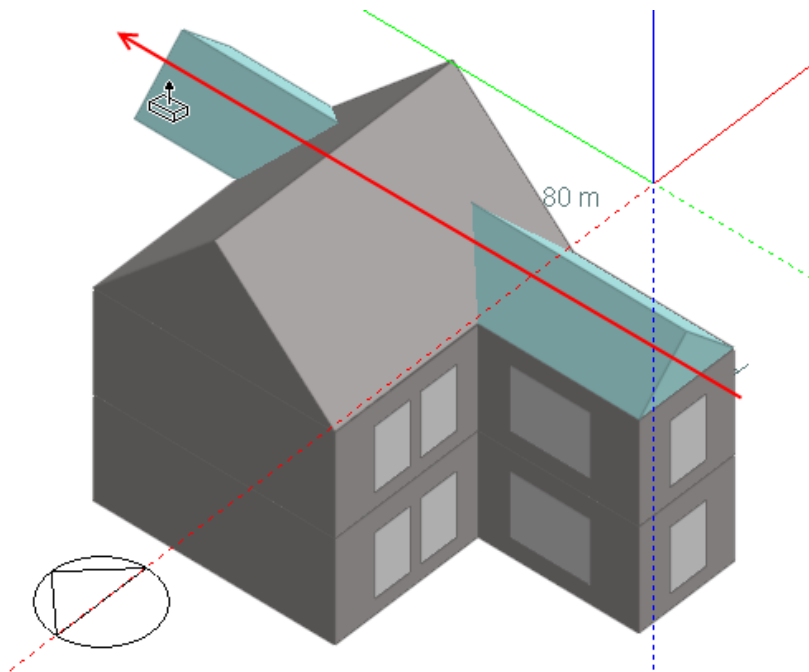
Draw the first perimeter line at a 45° pitch and move the cursor to intersect with the back face upper edge mid-point drawing guide and click to set the end-point:



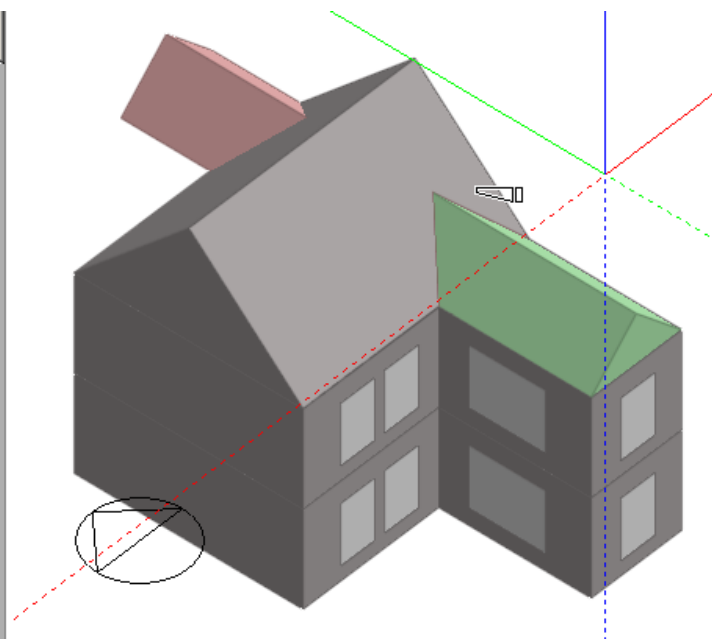
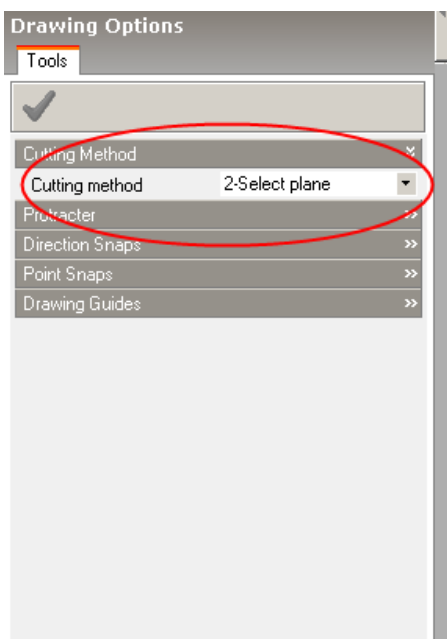
Move the cursor to snap to the other end-point of the back face upper edge and then complete the perimeter:



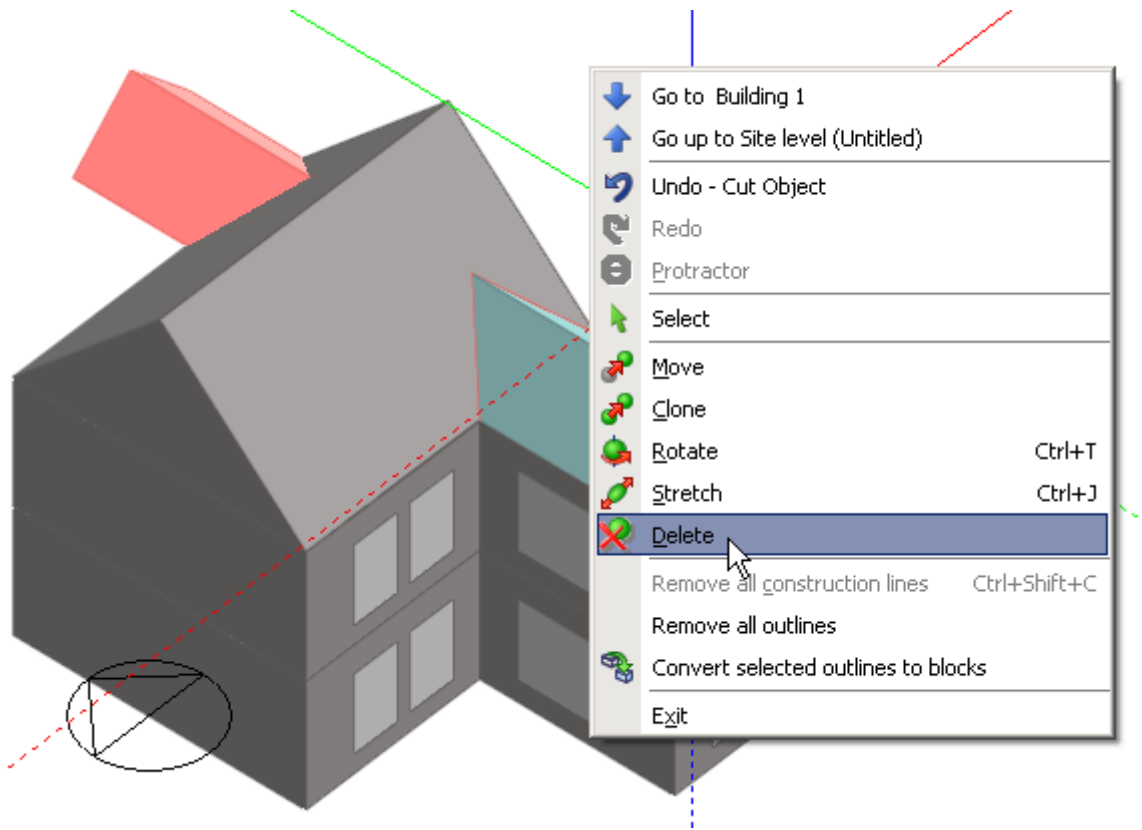
Now drag the outline block face through the main roof block:



Select the offshoot outline block and click on the 'Cut block along plane' tool, making sure that the 'Cutting method' on the 'Drawing options' data panel is set to 'Select plane' and click on the forward facing plane of the main roof block to create a cut in the offshoot block in the plane of the roof:



Click again to finish the cut and then select the section of the offshoot roof block that is not required and delete it:



Select the roof offshoot block and click on the 'Convert selected outline blocks to building blocks' tool to complete the process:



Note: In some circumstances, resulting blocks may not be able to be converted to building blocks due to geometrical limitations. In this case, the block will be displayed in a sepia colour to indicate that it cannot be converted.

Boolean Geometry Operations

DesignBuilder provides some powerful modelling tools for combining blocks in various ways to create new more complex geometric forms. Boolean geometry (otherwise known as [Constructive Solid Geometry](#) or CSG) provides these functions:

- **1-Union**, the most common Boolean tool where 2 or more adjacent blocks are combined into a single new block.
- **2-Intersection**, where a new block is formed where 2 original blocks intersect.
- **3-Subtraction**, in which one block is subtracted from the other to form a single new block.

The Boolean process works in 4 stages:

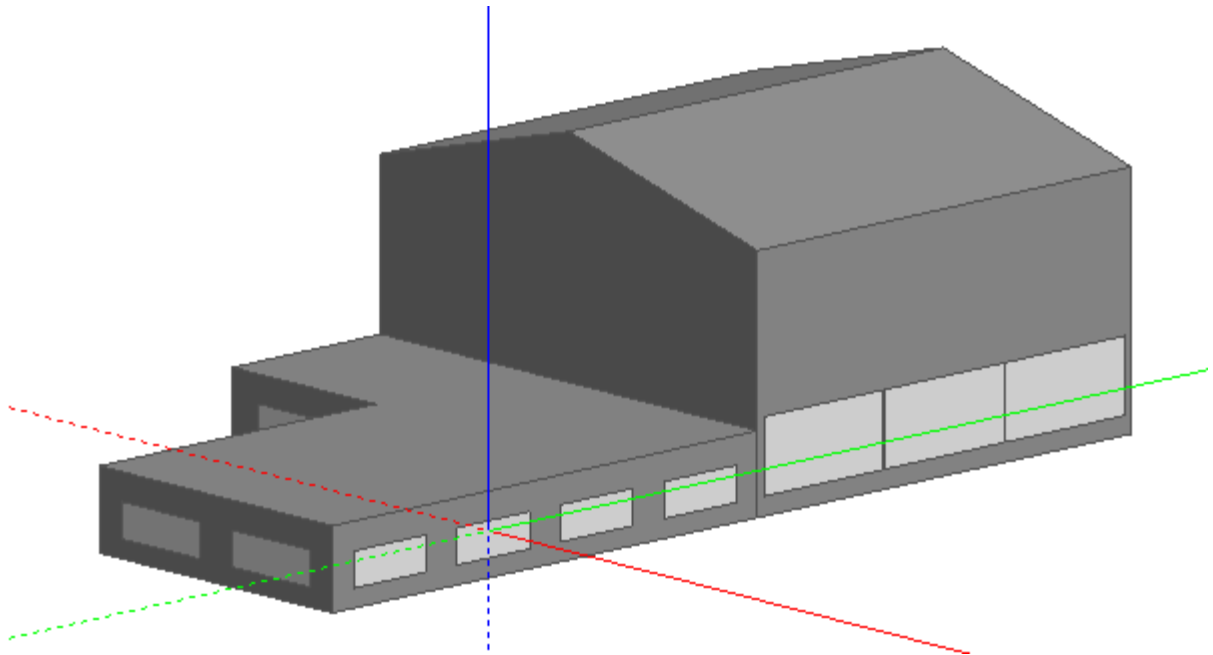
1. Select the blocks involved. The union operation can be applied to any block type including building blocks, component blocks and outline blocks, whereas the Subtraction and Intersection operations can only be applied to outline blocks as they require intersection of the 2 primary blocks. The blocks must both either be of the same type (building block /component block /outline block) or must involve an outline block. The blocks must touch in the case of the Union operation and intersect in the case of the Subtraction and Intersection operations.
2. Click on the Boolean operations toolbar icon. This opens the Drawing options dialog in the bottom left of the screen.
3. Select the operation type in the Drawing options panel in the bottom left of the screen. When 2 or more building blocks are selected Union is the only option so the drop list is greyed out. The blocks involved are shown in brown/purple for the Union and Intersection operations, or one in brown/purple and the other in green for the Subtraction operation. For the Subtraction operation the green block will be used to subtract space from the purple block.
4. Left-mouse click a final time on the model to complete the operation.

Note: The Boolean tools work most effectively when the geometry has been constructed accurately using snaps to ensure that adjacent blocks are positioned correctly. If the model geometry has issues such as adjacent block vertices being very slightly out of alignment, this can lead to Boolean operations being disallowed in some cases.

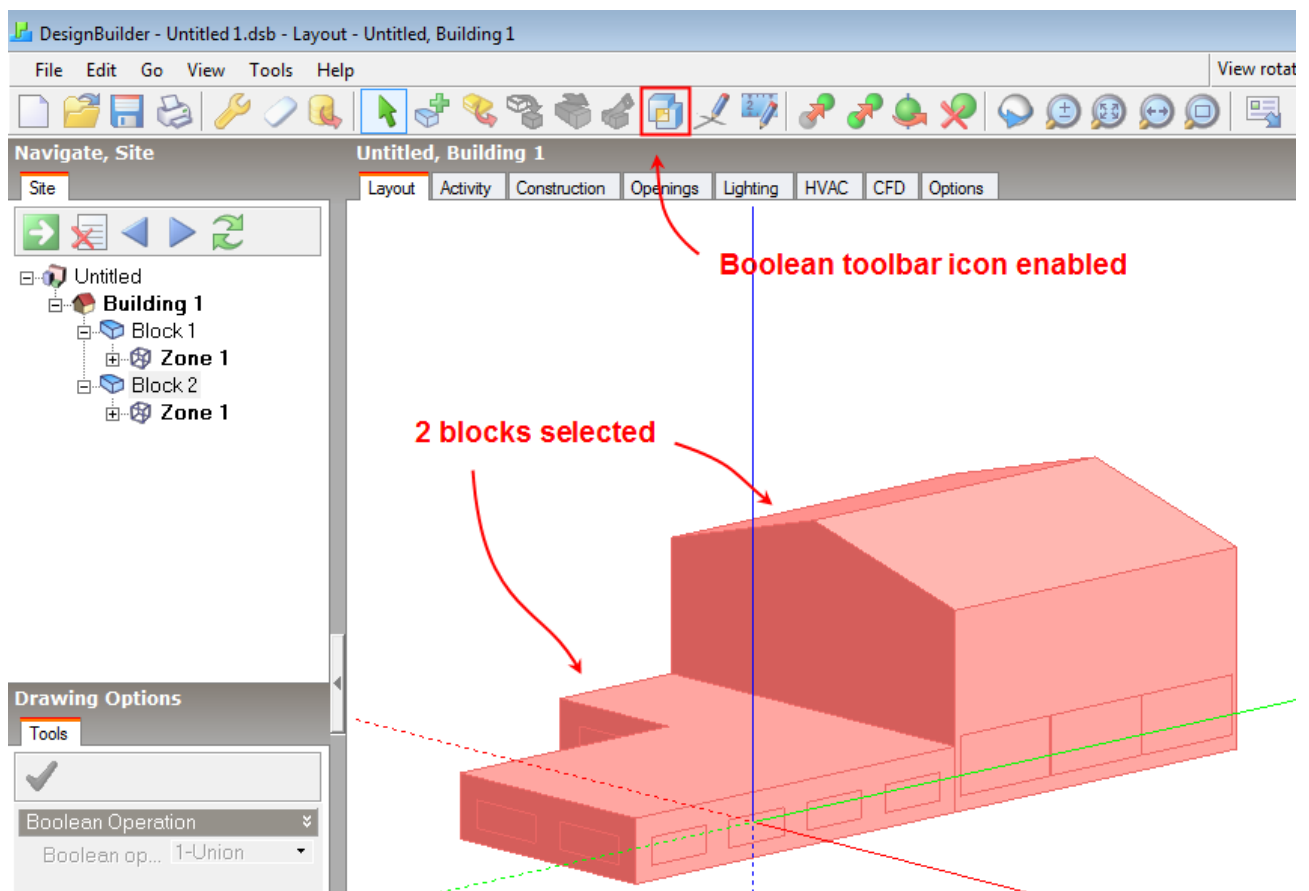
See also the [Outline blocks, Boolean operations](#) tutorial

Using the Boolean Operations - Union Blocks Example

The first step in using the Boolean Union tool is to go to building level and select 2 or more blocks to combine.



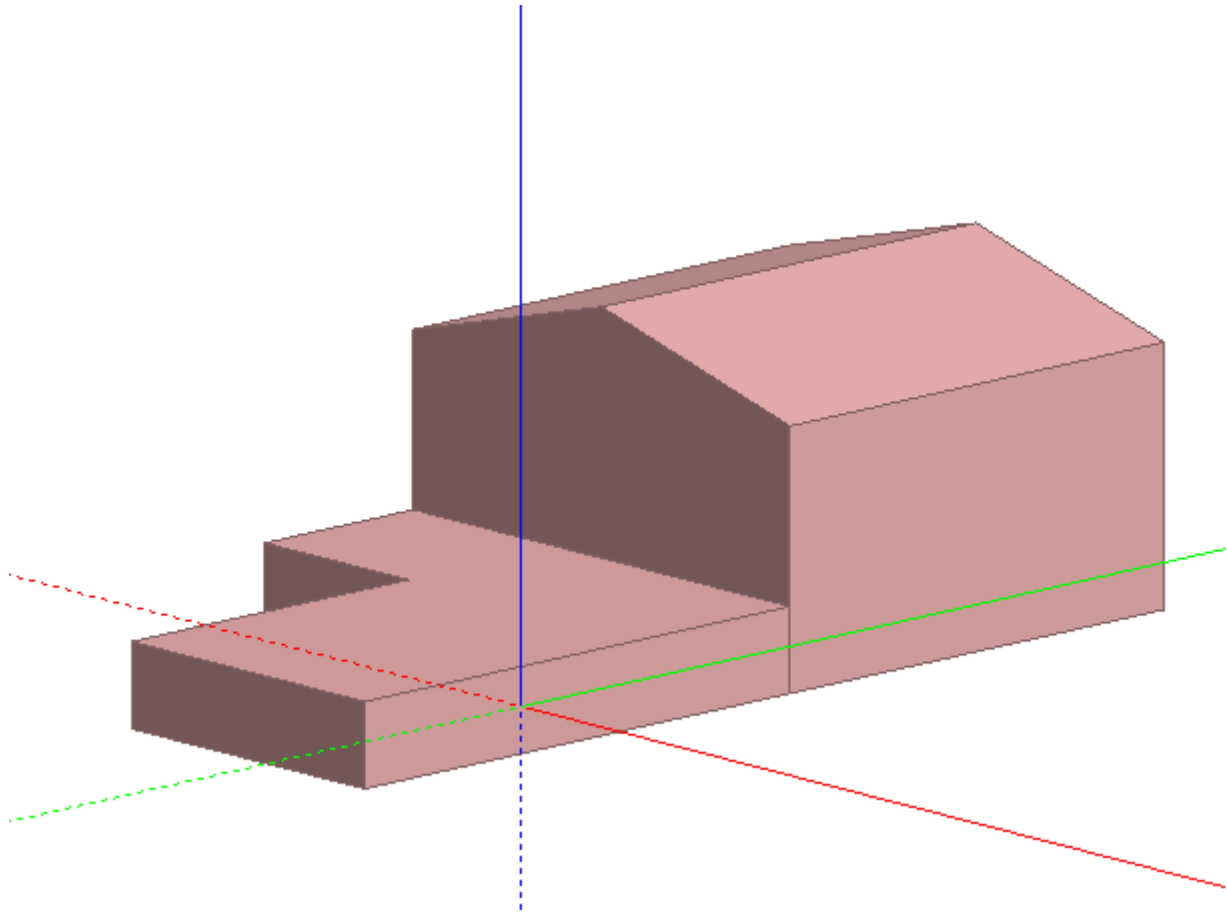
In this example, 2 blocks are to be combined are shown above. Before the operation is started the model consists of a 3.5 m height L-shaped block adjacent to a tall sports hall block.



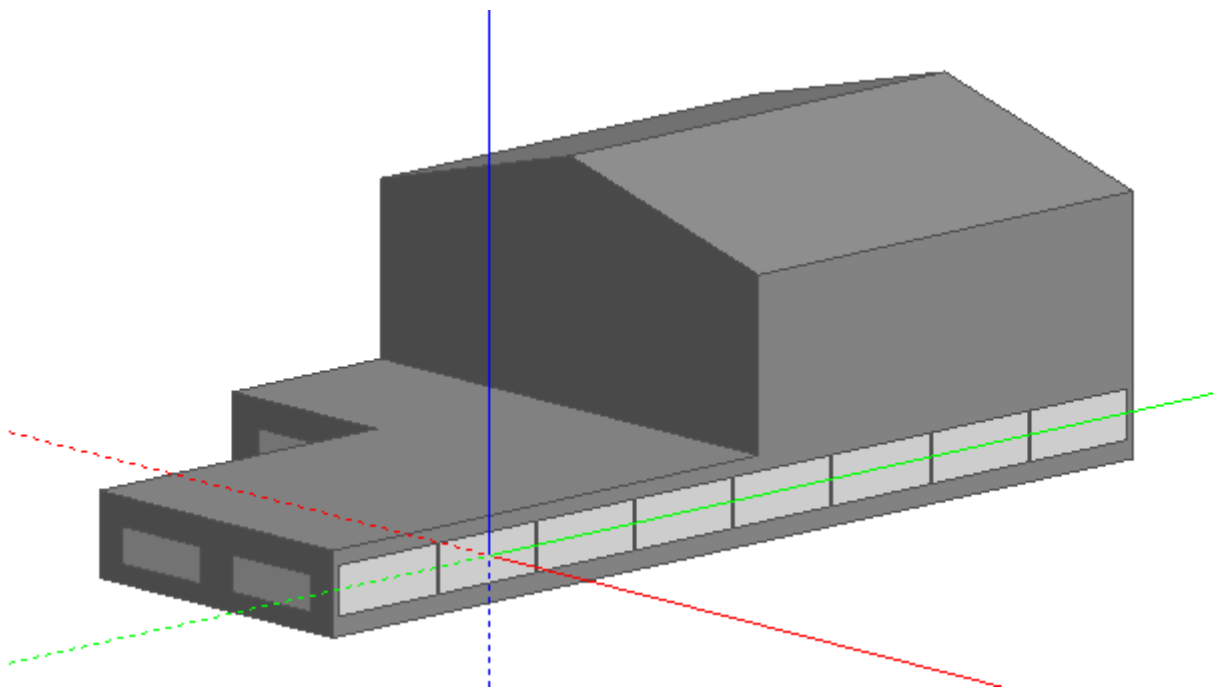
When the 2 blocks are both selected the screen looks as above with the **Boolean** operations toolbar icon enabled.

Clicking the **Boolean** toolbar icon starts the command and the Display options panel is displayed in the bottom left of the screen. On this panel there is a drop list with the 3 Boolean functions available for selection.

Note: In the case where 2 building or component blocks are selected, only the Union operation is allowed. This is because building blocks cannot intersect and the Subtraction and Intersection options do not apply as they require blocks to intersect.



Once the command has been started the 2 blocks will be coloured brown/purple. To complete the operation, click once more on the edit screen with the mouse.



After the final click, the 2 blocks are combined into a single block as shown above.

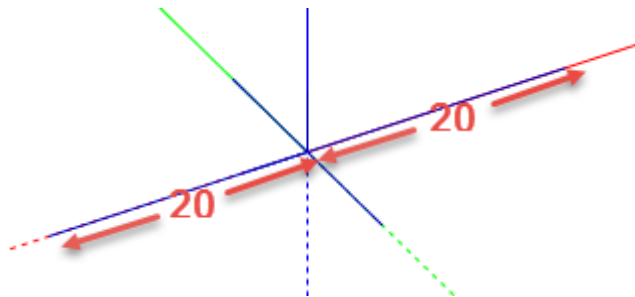
This can be a very useful way to simplify building geometry avoiding use of multiple blocks and easing the creation of zones that may span what were multiple blocks.

Note: When combining adjacent blocks using the Union operation, block boundary walls that used to separate the blocks are not replaced by partitions.

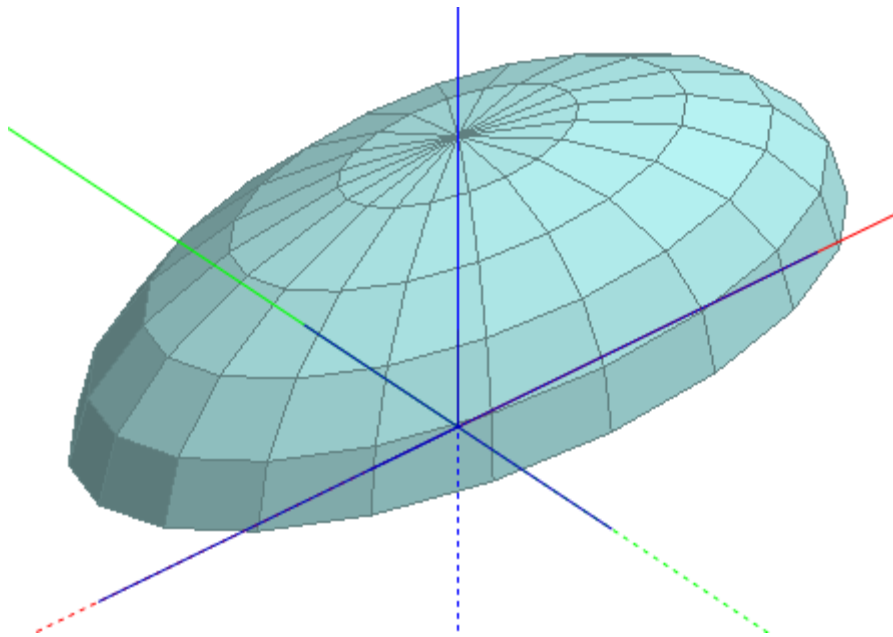
Using the Boolean Operations - Subtraction Example - Nested Glazed Ellipsoids

This example illustrates how to create a complex geometry with one ellipsoid (egg-shaped) block within another.

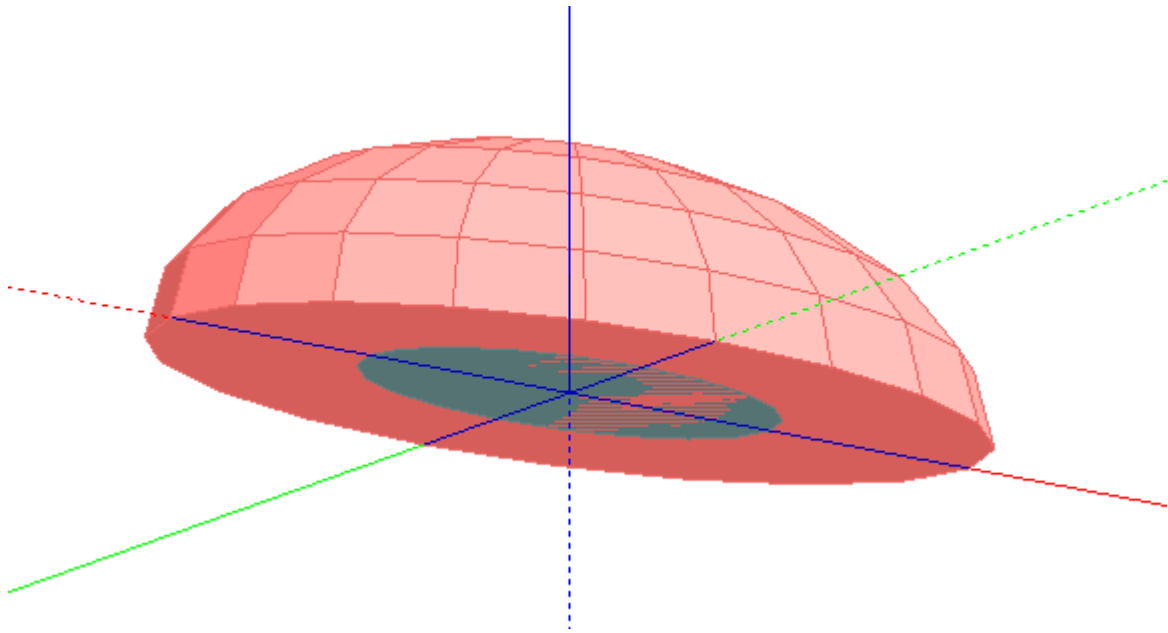
Step 1 - Draw 4 construction lines starting from origin to create a cross centred on the origin 20 x 10m



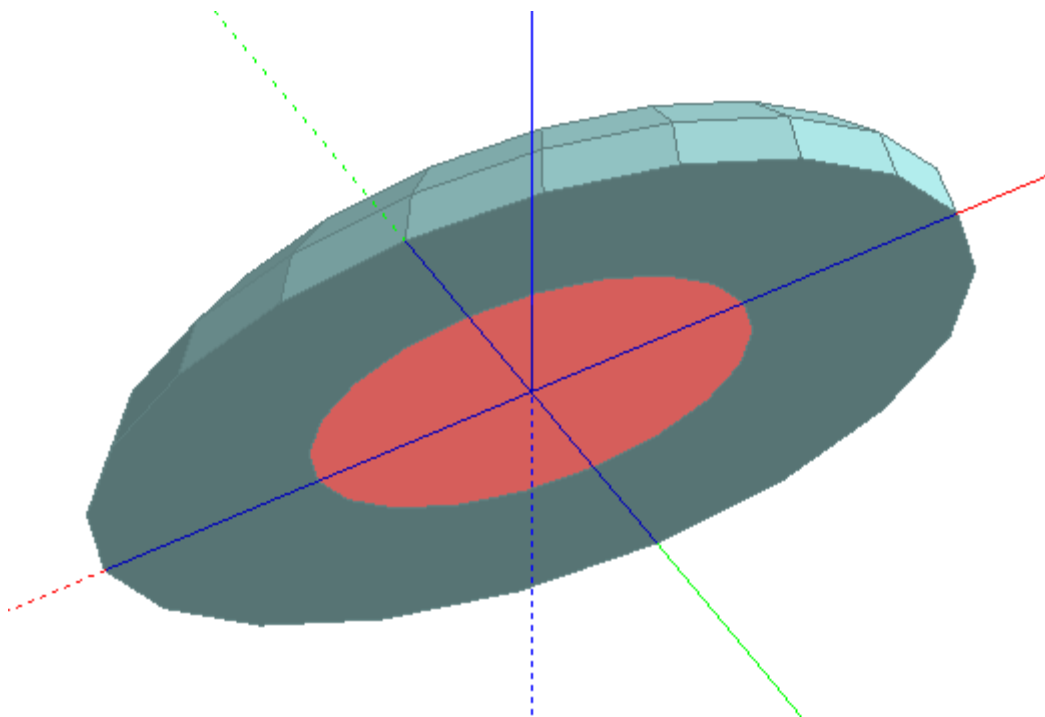
Step 2 - Draw a dome 20m radius centred on the origin, then use the Stretch tool to reduce it's y and z dimensions by 50%. The dimensions should be 40m in the x-direction, 20m in the y-direction and 10m height (z-direction).



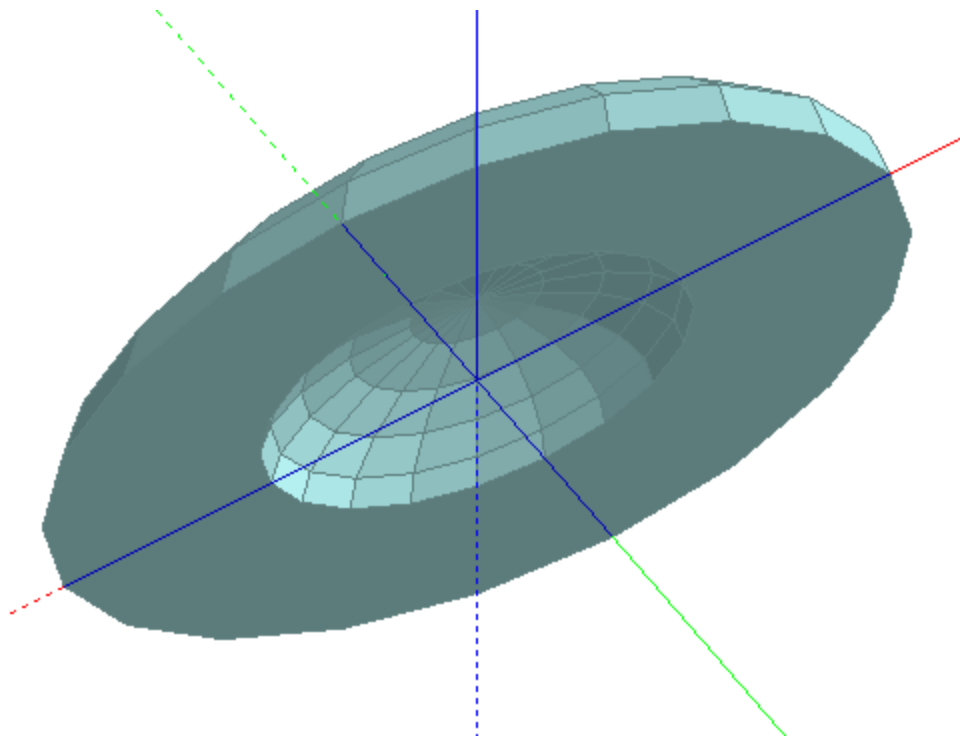
Step 3 - Draw a similar but 2x smaller ellipsoid 10m x 5m x 5m. Move the large ellipsoid on top of the smaller one with the centre of each over the origin as shown below.



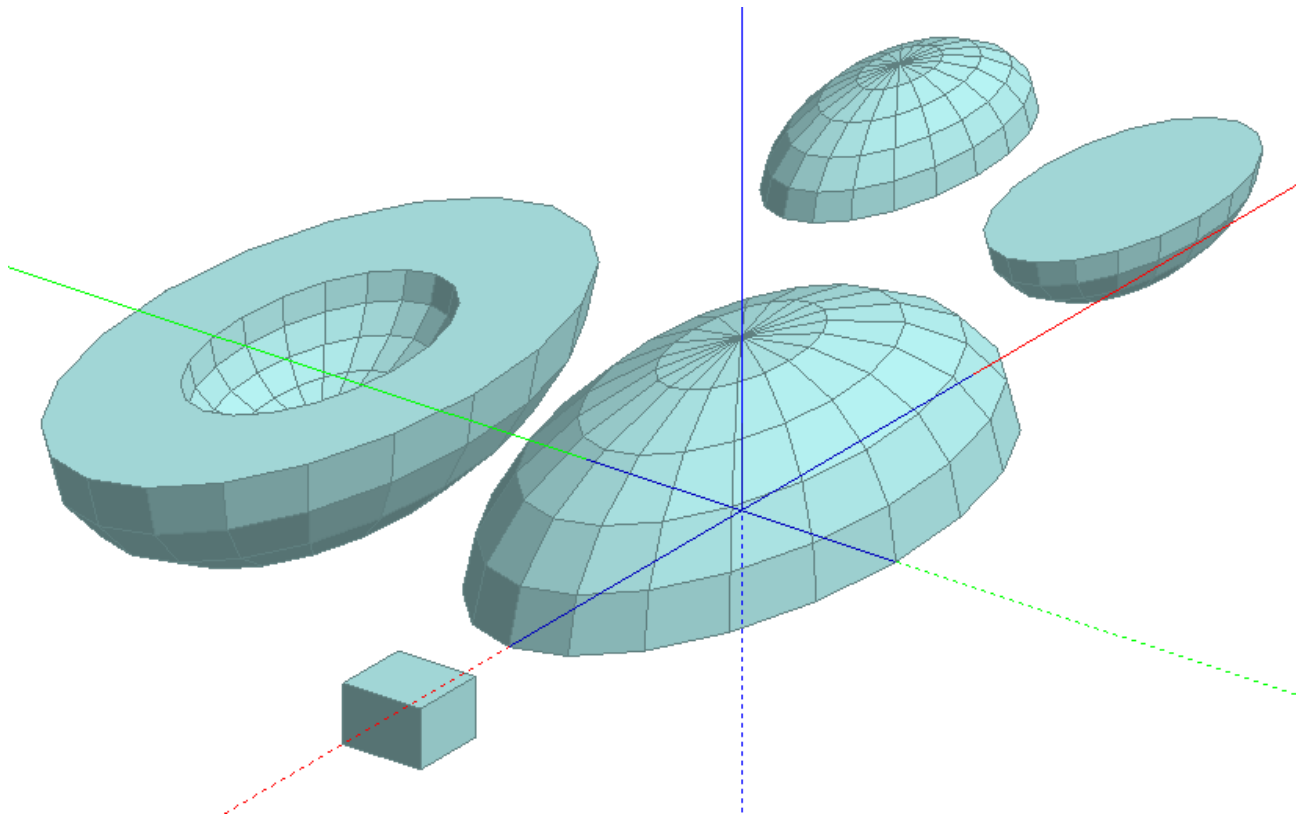
Step 4 - Use the selection tool to rubber band a selection over both blocks, click on the Boolean operations toolbar icon and select the Subtraction option in Drawing options. If necessary click on the Switch subtraction block checkbox to ensure that the smaller inner block is in green (i.e. it is to be used to subtract volume from the larger block). After the Subtraction operation you should see the blocks as below.



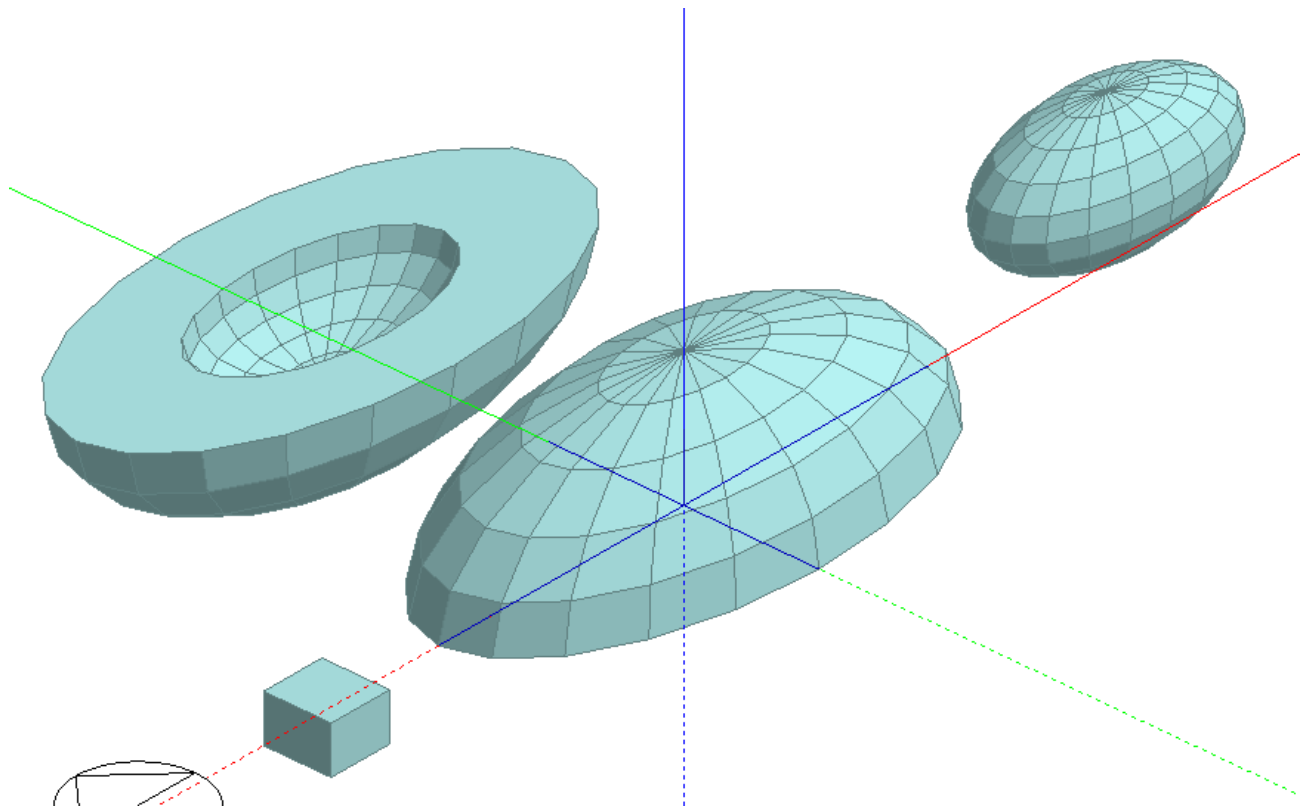
... and with the smaller inner block moved out of the way:



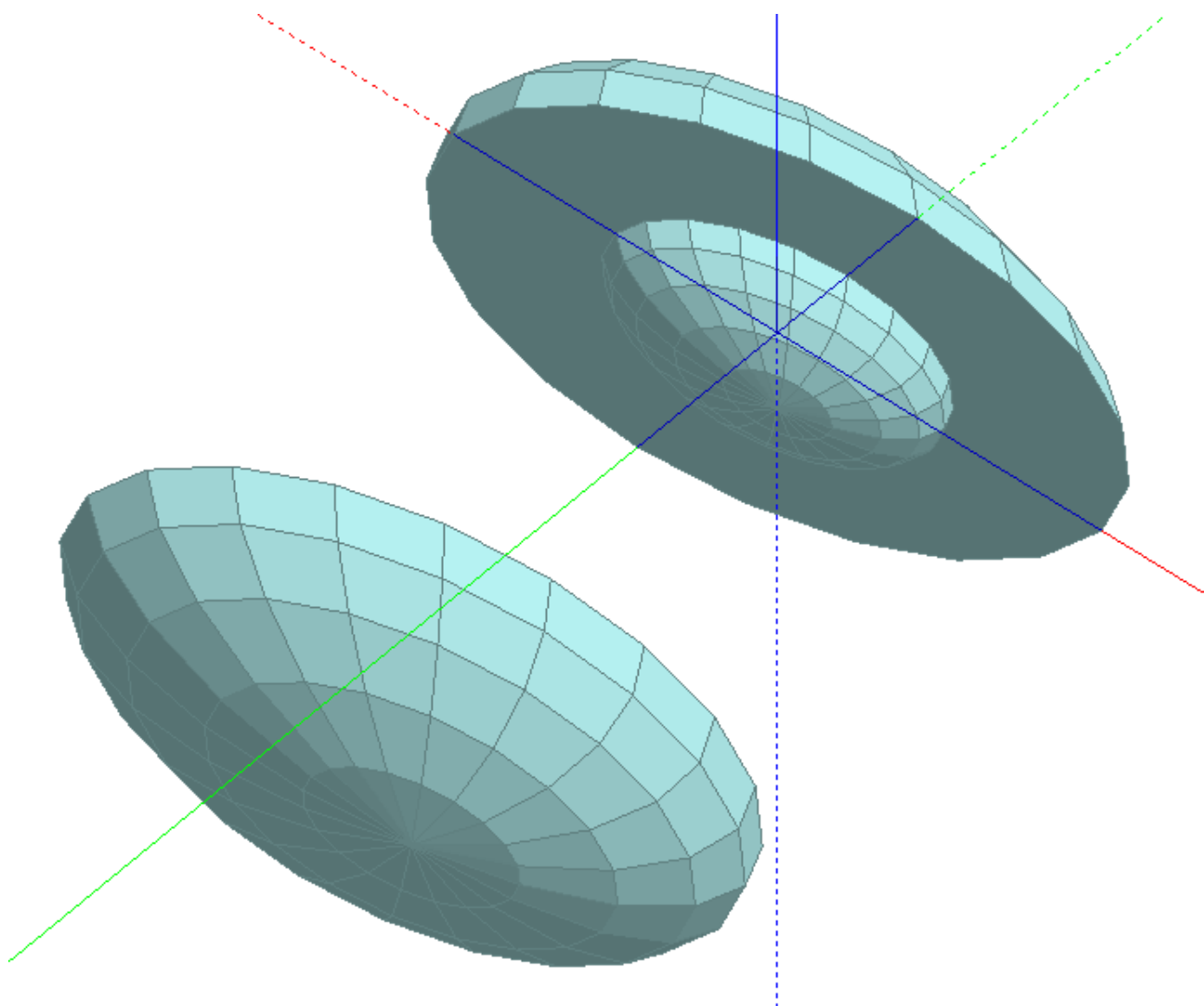
Step 5 - Draw a small rectangular outline block aligned with the x and y axes to give you a vertical plane to allow rotation operations to create the lower blocks. Then copy the larger outer block and rotate the copy by 180° using a vertical plane as the rotation plane. Repeat these steps for the smaller inner outline block.



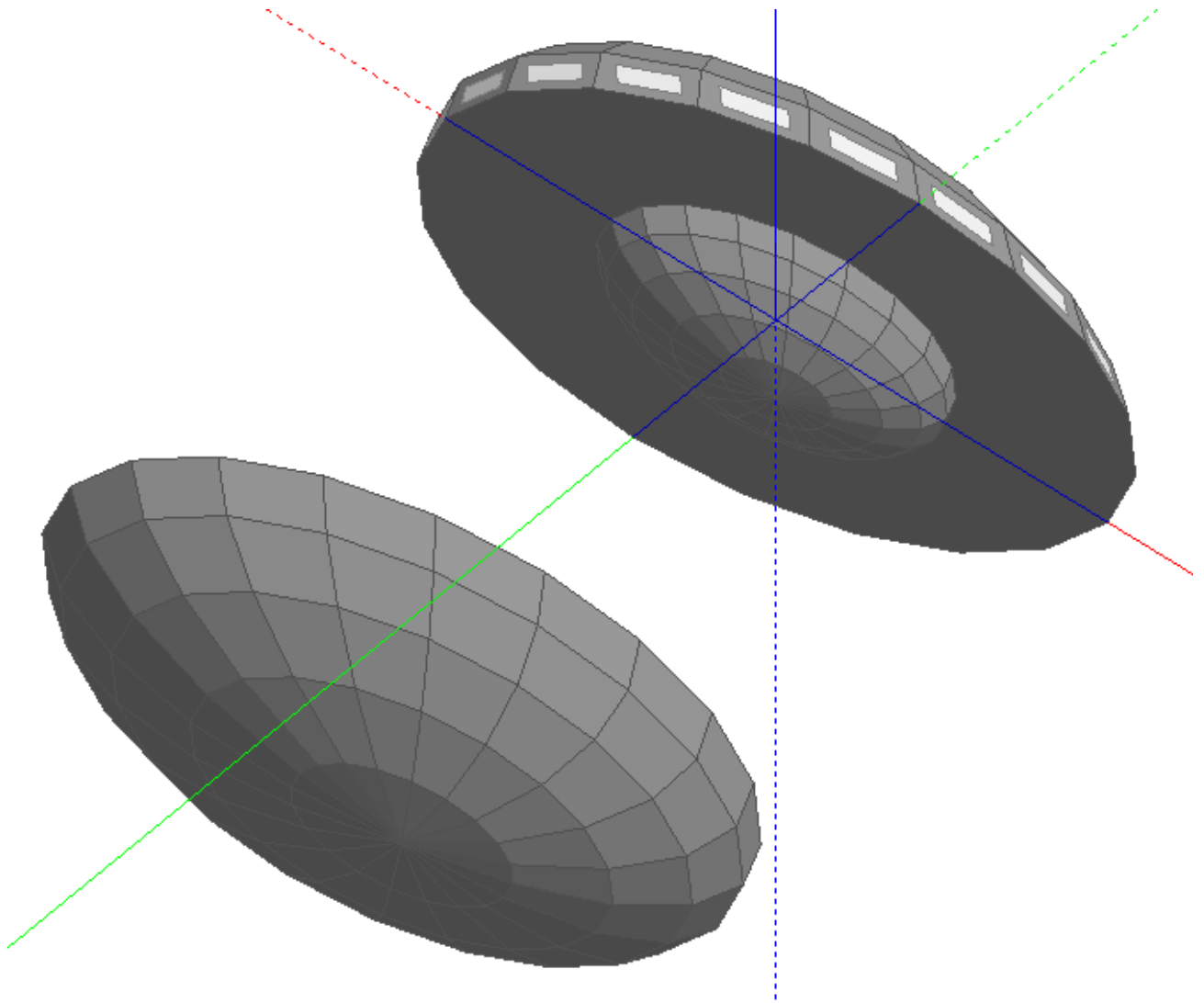
Step 6 - Position and Union the 2 small inner blocks together to create an egg shaped inner block.



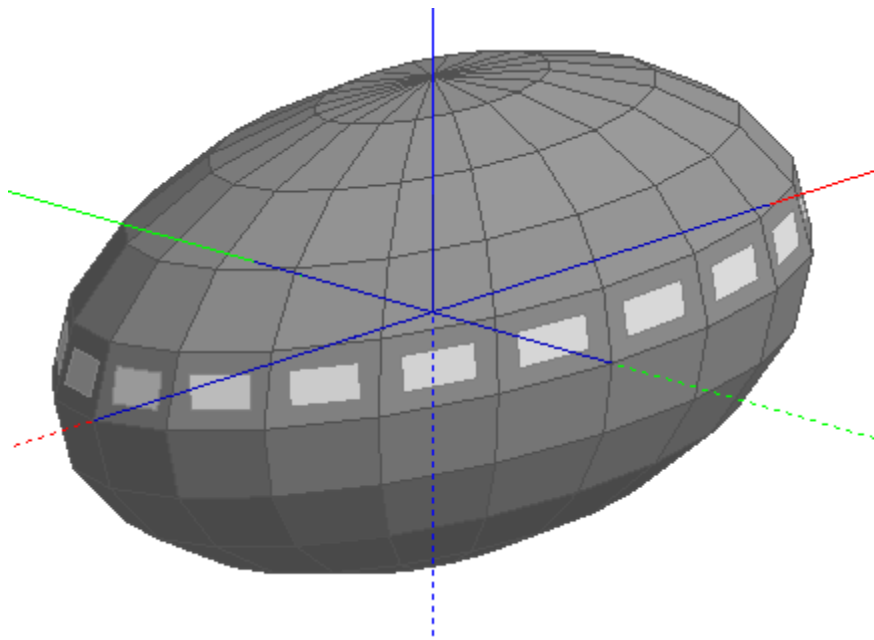
Step 7 - Move the inner outline block into place.



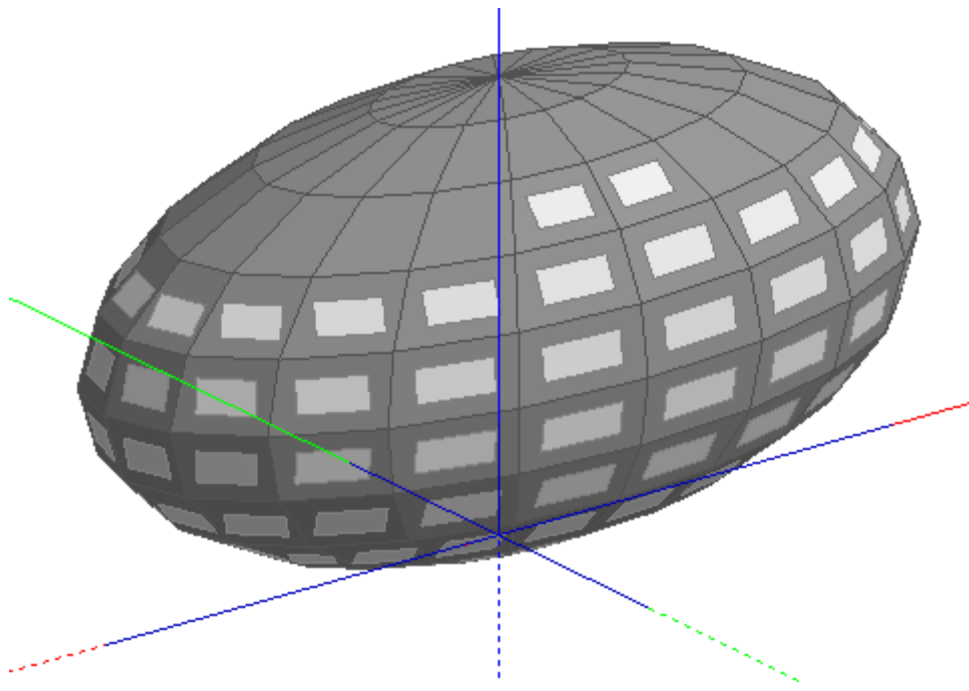
Step 8 - Convert the outline blocks to building blocks



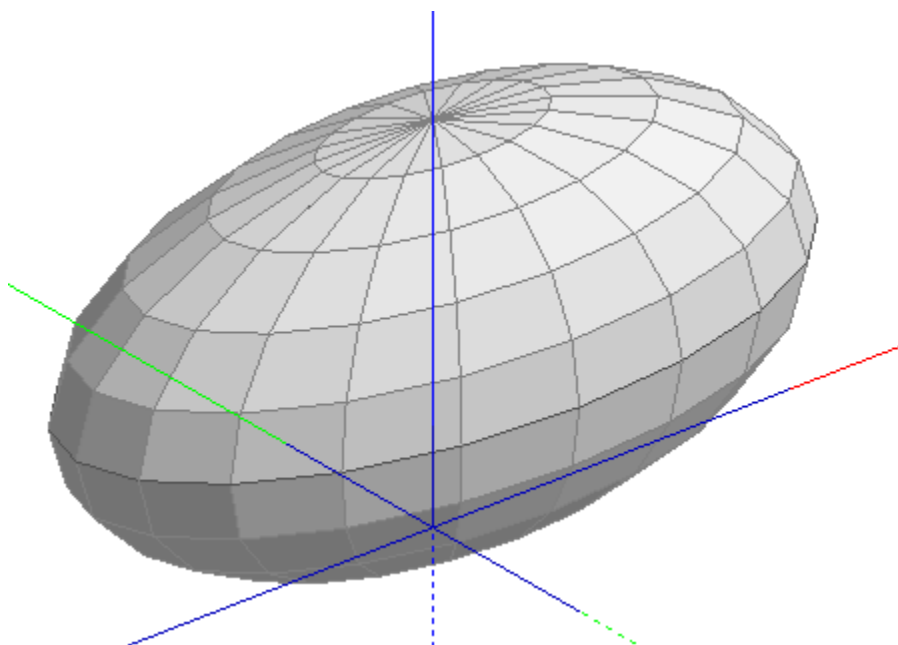
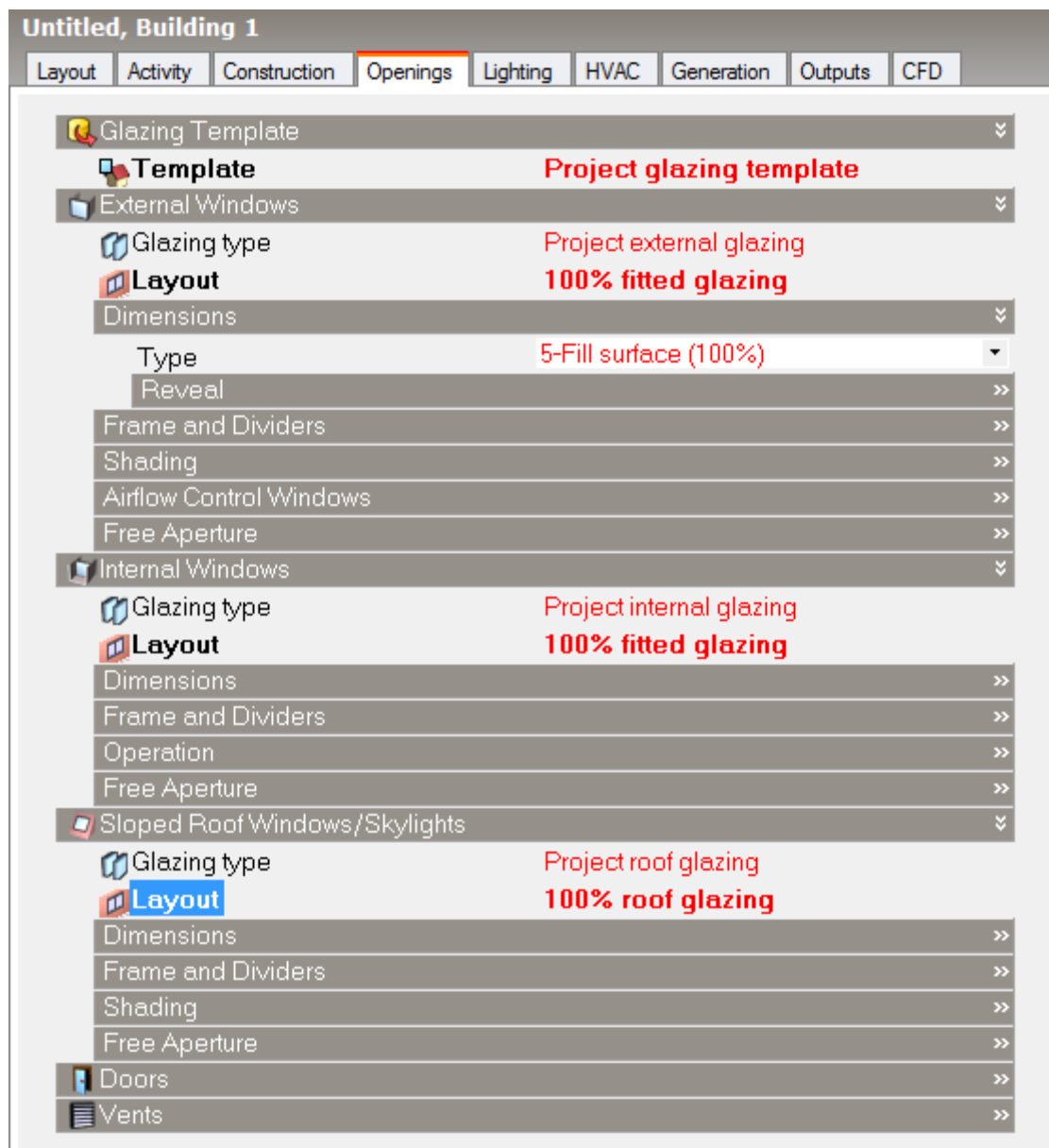
Step 9 - Move the lower outer block into position.



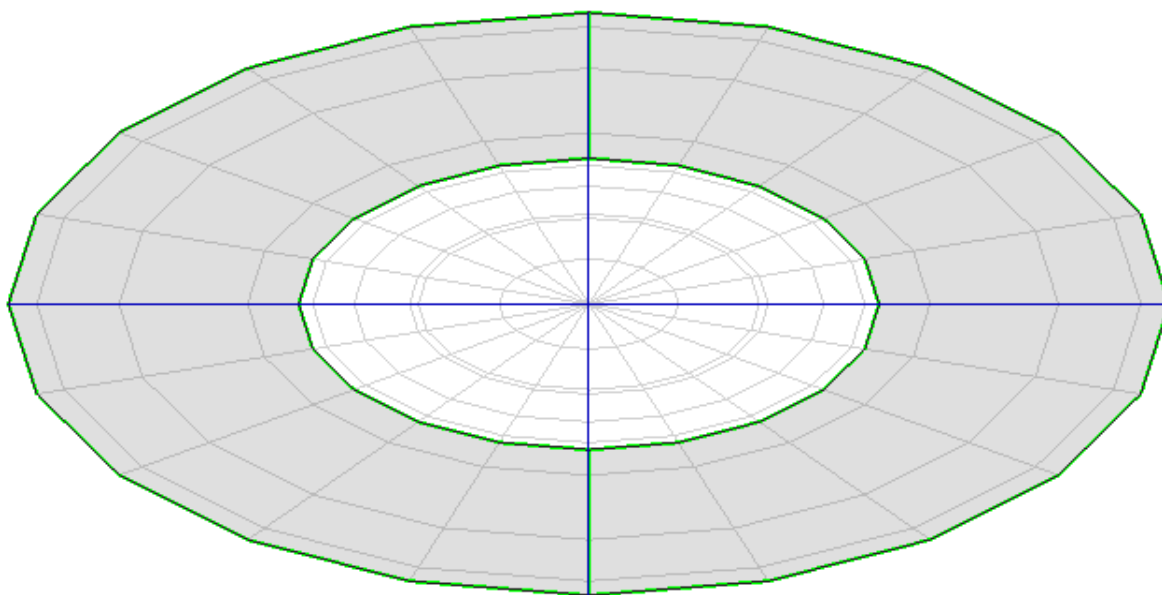
Step 10 - If necessary, move the entire set of 4 blocks above ground by selecting all 3 blocks (rubber band selection) and using the base point of the lower outer block as the reference point and moving it the origin.



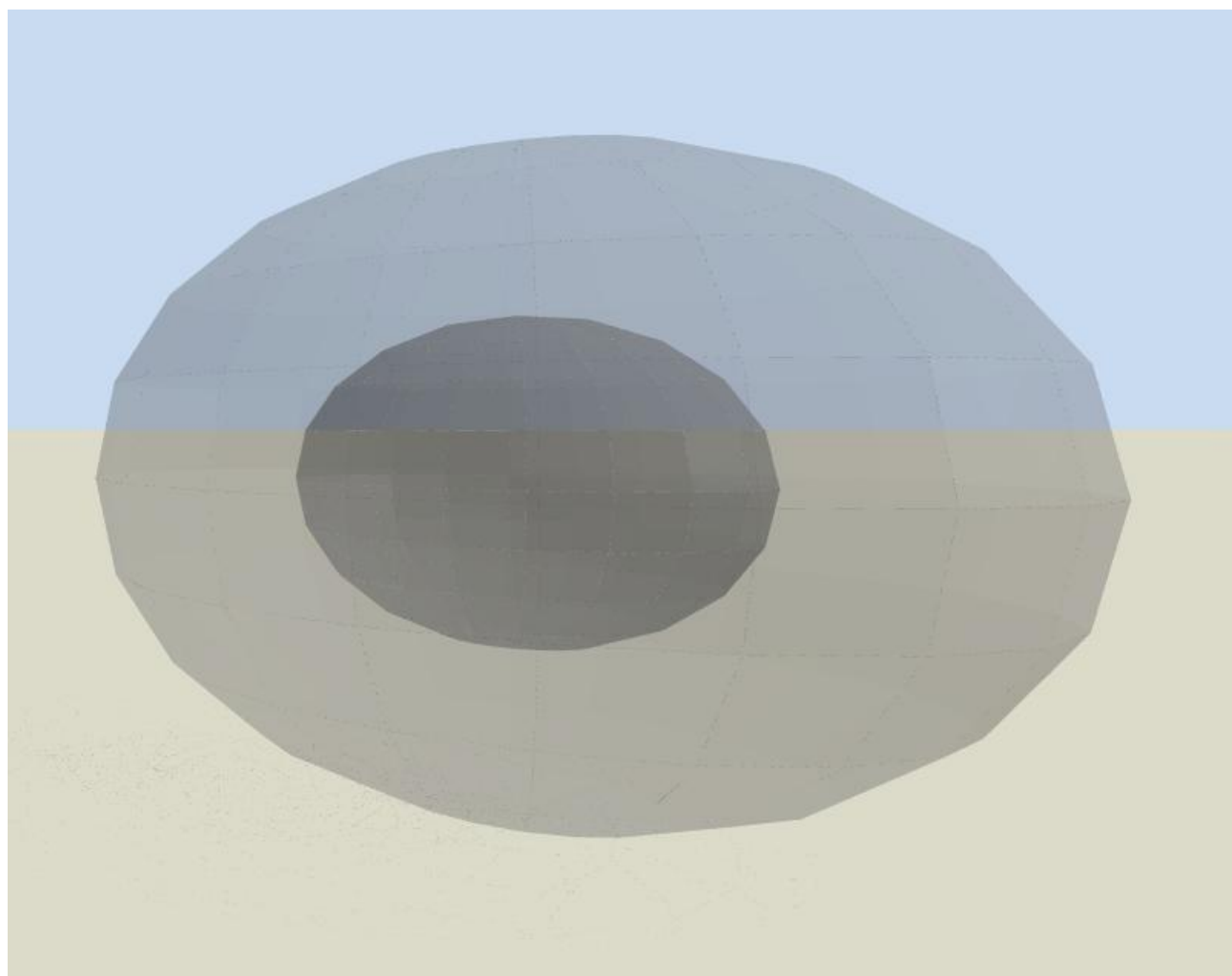
Step 11 - Set 100% glazing for exterior, interior and roof glazing.



Step 12 - Draw 2 holes in the floor of the outer top block. Note it is not possible to union the 2 outer blocks as there would be problem maintaining the inner blocks in this case.



Step 13 - Visualise and check the geometry is as expected.



Edit Block Options



You can edit certain aspects of a building block geometry by clicking on the Model options icon while at the block level. This opens the Model options dialog box opened on the [Block](#) tab.

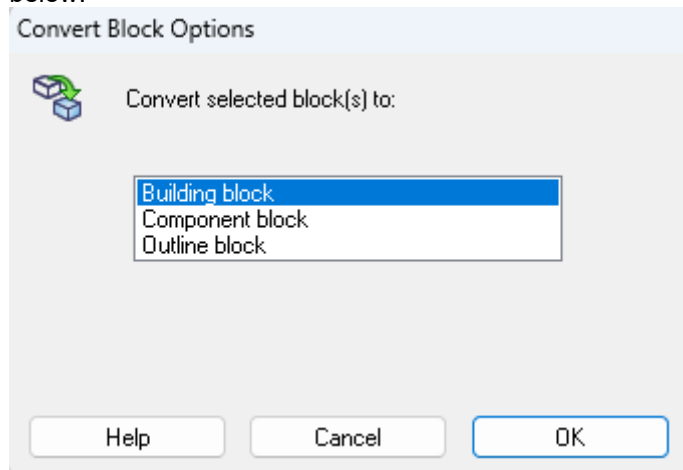
See also the [Program and Model Options](#) tutorial

Convert Blocks



To convert one or more blocks to a different [Block type](#) follow the steps below:

1. Navigate to building level.
2. Select the [Outline](#), [Building](#), or [Component](#) block(s) to be converted by clicking with the mouse.
3. Click on the **Convert selected block** toolbar icon to open the **Convert Block Options** dialog as shown below.



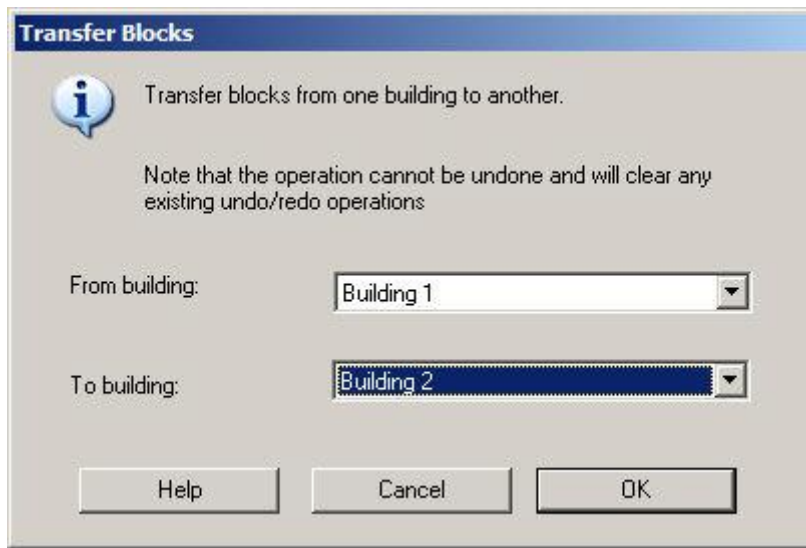
4. Select the type of block to convert the selected blocks to. Choose from the options:
 - [Building block](#)
 - [Component block](#)
 - [Outline block](#)
5. Press OK to complete the operation.

See also the [Outline blocks, Boolean operations](#) tutorial

Transfer Blocks

You can move blocks from one building to another using the Transfer blocks command accessed from the [Edit](#) menu when the site contains more than 1 building. This menu option opens the Transfer blocks dialog below. Select the building containing the blocks to be copied and the building to receive the blocks and Press OK. *All blocks* in the source building will be moved to the destination building.

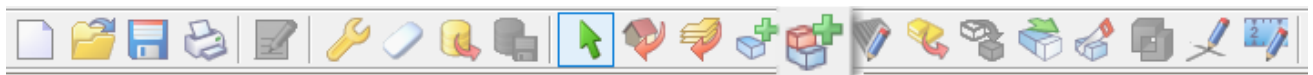
Tip: If you would like to transfer only selected blocks then you could first clone the source building, remove the blocks that are not required from the copy then transfer the remaining blocks to the destination building.



Note: You should check before starting the block transfer that none of the source blocks overlap with blocks in the destination building. Even a small overlap will cause the whole process to fail.

A Transfer block operation cannot be undone and it clears any existing undo/redo information.

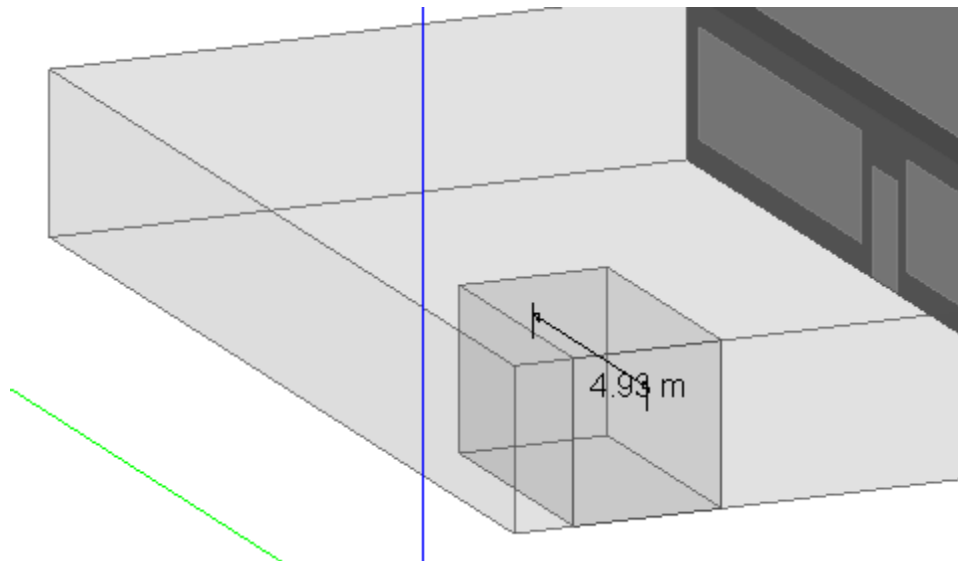
Add Surface



The topology of a block can be edited using the **Add surface** tool which allows you to modify a block surface by drawing a perimeter on an existing block surface and dragging it inwards or outwards along an axis, normal to the surface. It is a special case of the [Drag block face](#) tool which is used to drag a whole face. Add surface allows you to drag a portion of a block surface creating a new block topology.

To use the **Add surface** command:

1. Go to the building level (or block level if you wish to add a surface to a block-level component block).
2. Click on the **Add surface** tool.
3. Move the mouse cursor to the surface that you want to edit and draw the perimeter of the surface to be created.
4. Once the new surface perimeter has been defined use the mouse to start dragging it inwards or outwards. You can key in an exact distance to be used in the drag process. See example screenshot below.



5. When you have the block geometry you need, left mouse click to confirm.

See also the [Add surface, increment snaps](#) tutorial

Adding Solar Collectors

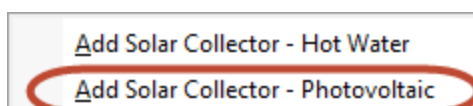


To add or create a new Solar collector follow these steps:

1. First go to the building level (if you are not already there) and click on the **Draw solar collector** toolbar

icon.

2. Select the type of collector from the pop-up menu. For example to add a PV panel, select the **Add solar collector - Photovoltaic** option.



3. Position and draw the Solar collector in the way that you would a block when using the **2-Rectangle shape**. The first click defines one corner (the bottom right say), the second click defines the diagonally opposite corner (top left say) and completes the placement.

The panel is effectively defined in 2-D, i.e. there is no option to define its depth.

You can draw the panel on an existing building block, component block or outline block surface or directly on the ground.

Note: In cases where a solar collector is placed on a building surface (e.g. on a pitched roof surface) it is not modelled as being thermally coupled with the underlying surface by default, but it does cast a shadow like any other block or shading surface.

PV panels can however be thermally coupled to the building using the BIPV tools, specifically using the [Heat transfer integration mode](#) for the Construction used for a surface below the panel.

4. You can edit the properties of the solar collector by first navigating to the Solar collector object by either double-clicking on the graphical object in the Layout screen or single-clicking on the item in the Navigator. For PV solar collectors you can define the panel performance and other properties on the [Construction tab](#) under the [Solar Collector header](#).

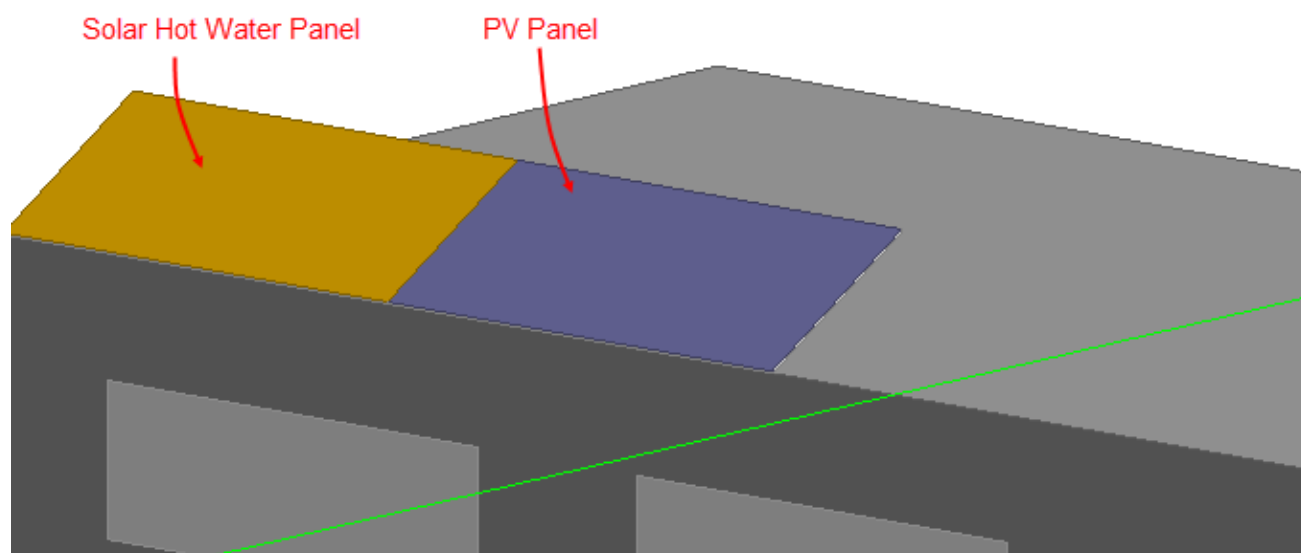
5. The electrical side of the **Photovoltaic** collectors operation is defined on the HVAC tab under the [Generation](#) header.

The hot water, operation and control of **Solar hot water** collectors is defined with the Detailed HVAC data.

Note: PV systems can be modelled when using either Simple or Detailed HVAC but Solar hot water systems require Detailed HVAC.

Panel Active Direction

When placing a solar panel, one face will usually be pointing upwards towards the sky and the other downwards towards the ground or the building. DesignBuilder automatically determines the "active" side as the one pointing upwards. The active face of the panel is displayed in blue for PV panels and in yellow for solar hot water panels, as shown below.



Changing the active direction

In some cases it may be necessary to change the active direction of a solar panel. For example, when the panel is placed vertically, or after it has been rotated to a new slope and the active face is pointing downwards. In such cases you can change the active side of the panel by holding the <Shift> key down and left-clicking with the mouse. This toggles the active face.

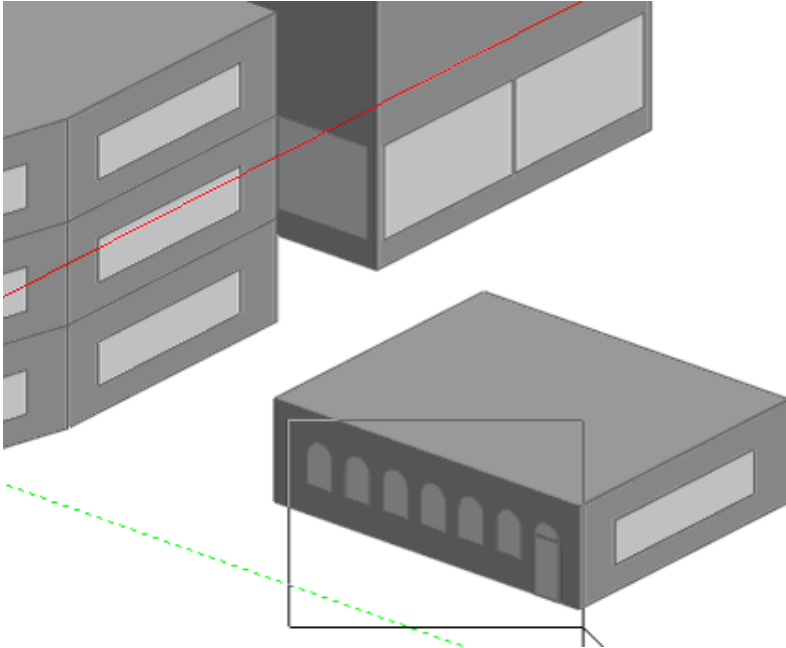
See also:

- "How to Model Photovoltaic (PV) Systems" on page 523

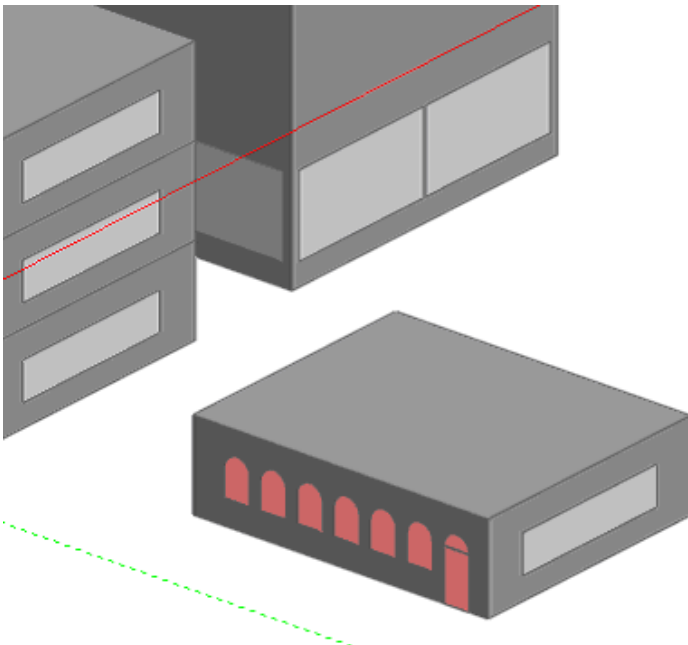
Edit Openings At Building Level

Building block façade openings can be selected at the building level and subsequently [moved](#), [copied](#), [mirrored](#), [stretched](#) or [deleted](#).

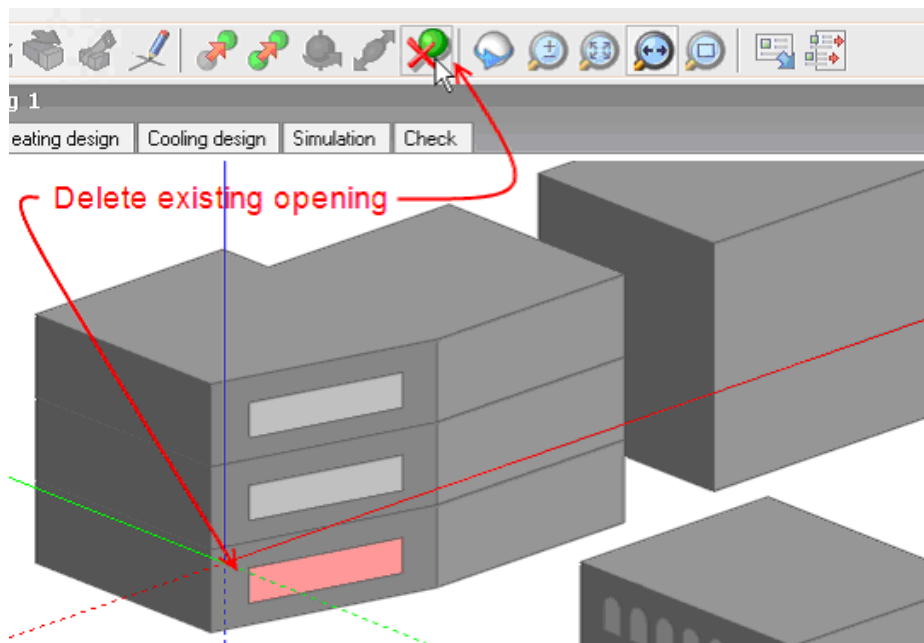
In order to edit openings at the building level you first need to select the openings, either clicking on them using the [Select](#) tool or by dragging a selection window around them:



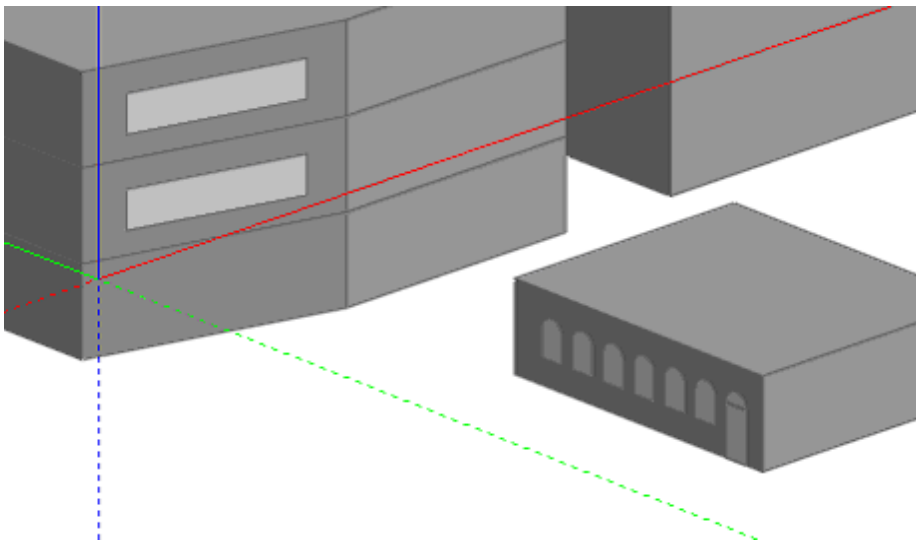
The selected openings are highlighted:



When you have selected an opening, you will see that the [Move selected object\(s\)](#), [Clone selected object\(s\)](#), [Mirror selected object\(s\)](#), [Stretch selected object\(s\)](#) and [Delete selected object\(s\)](#) icons are active and you can select one of these commands. As an example, the arched openings and doorway are to be copied onto the angled façade of the leftmost block. To begin with, you need to delete the existing opening on the angled façade by selecting it and clicking on the [Delete selected object\(s\)](#) icon:

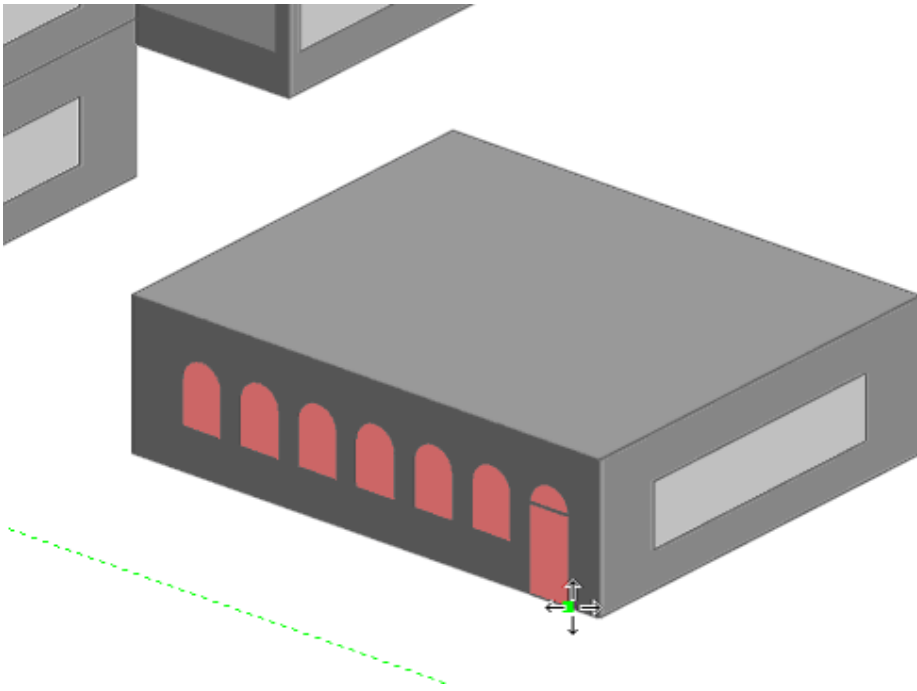


Opening removed:

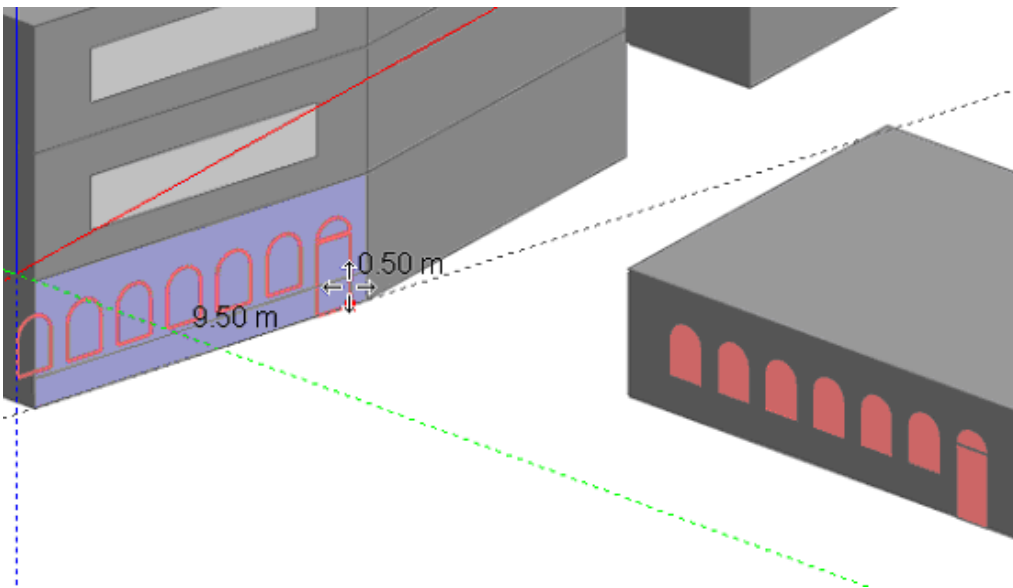


All opening edit operations at the building level automatically convert default openings to custom openings, which is why you need to remove the existing opening.

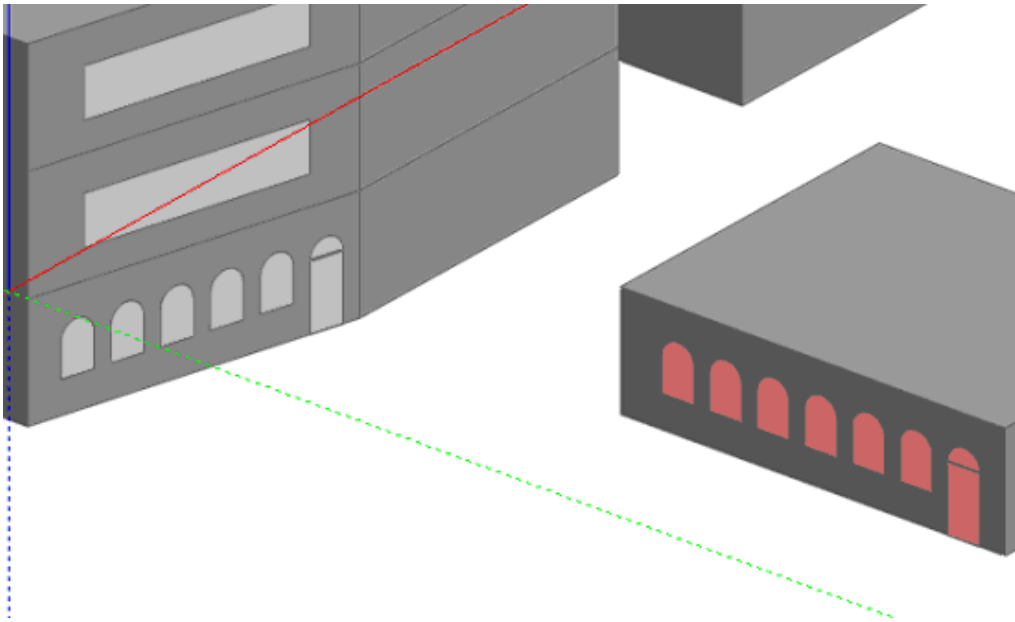
Then in order to copy the openings, you need to select a **handle point** and you can do this by first clicking on the [Clone selected object\(s\)](#) command icon and then clicking on the required handle point - the [endpoint snap](#) will help in selecting the handle point.



First select the handle point, and then position the copy using the various [snaps](#):



As you move the copied openings across each block façade you will see them automatically snap into the façade plane and after positioning the openings, you can then click on the mouse key to add the openings in the required position:



You will see that any copies that won't fit into the selected location are automatically discarded.

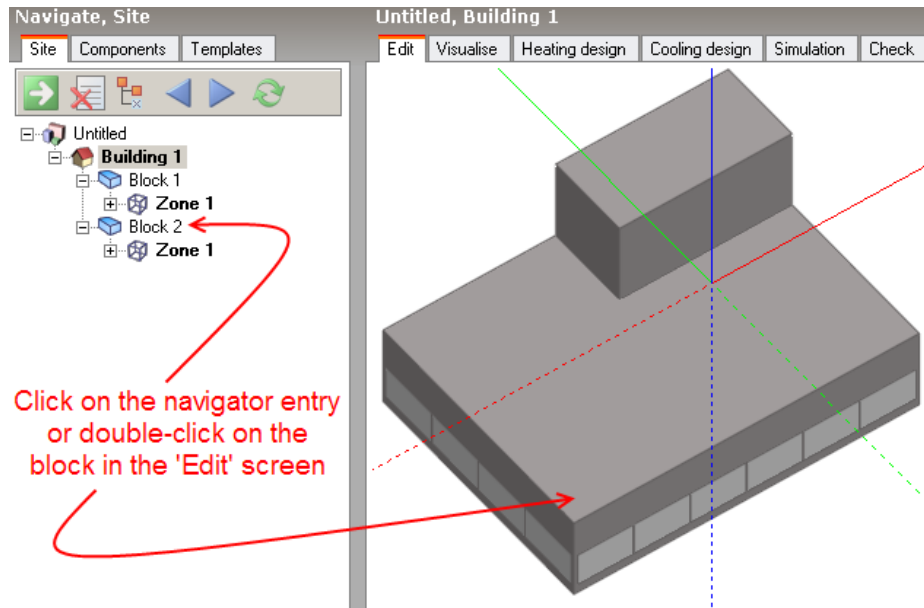
Note: You can select openings on multiple planes, but in this case the openings can only be deleted, not moved or copied.

See also the [Drawing windows and doors](#) tutorial

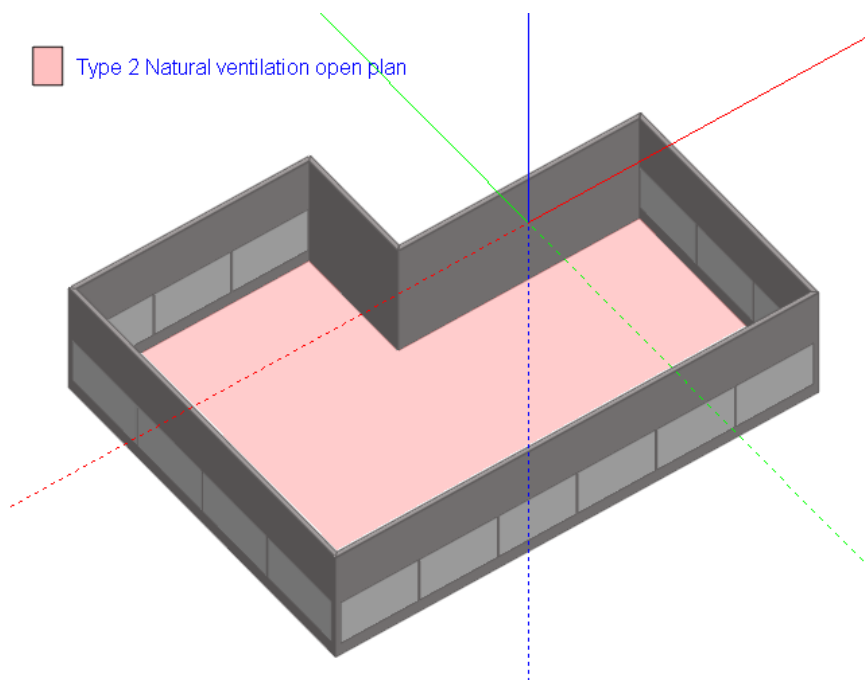
Working At Block Level

This section describes the various operations involved in [partitioning](#) a block to create zones and adding [courtyards](#) to blocks. Before you start to draw partitions or courtyards, you first need to go to the block that you want to edit and you can do this by clicking on the block entry in the navigator or by double-clicking on the block in the 'Edit' screen, if you're at the building level.

Go to block level



Either of these operations will take you to the block:



At block level you can also:

- Add and edit **1-Standard** type [component blocks](#) for use in visualisation, daylighting and CFD analysis.
- Add internal [CFD fans](#) for CFD analysis.

Block Level Options

When drawing construction lines or using the measure tool at block level, some special options become available in the **Drawing Options** panel.

Snap mode

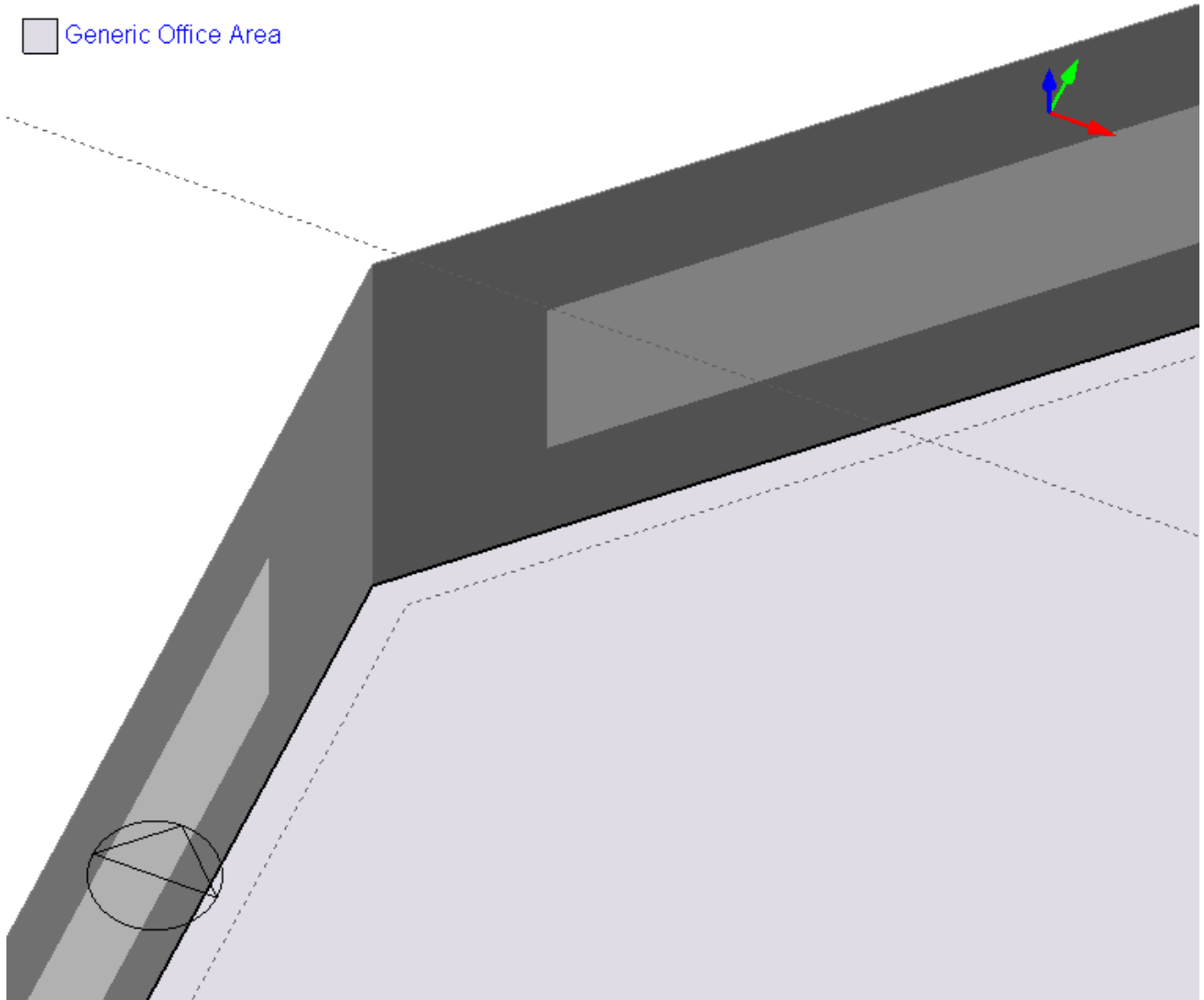
At block level the **snap mode** allows you to specify whether snaps should be made available for the block outer perimeter or for the zone inner surfaces. The options are:

- **1-Block perimeter** where snaps are provided for the block perimeter lines.
- **2-Zone inner surfaces** where snaps are instead provided for the zone inner surfaces.

Display zone inner surfaces

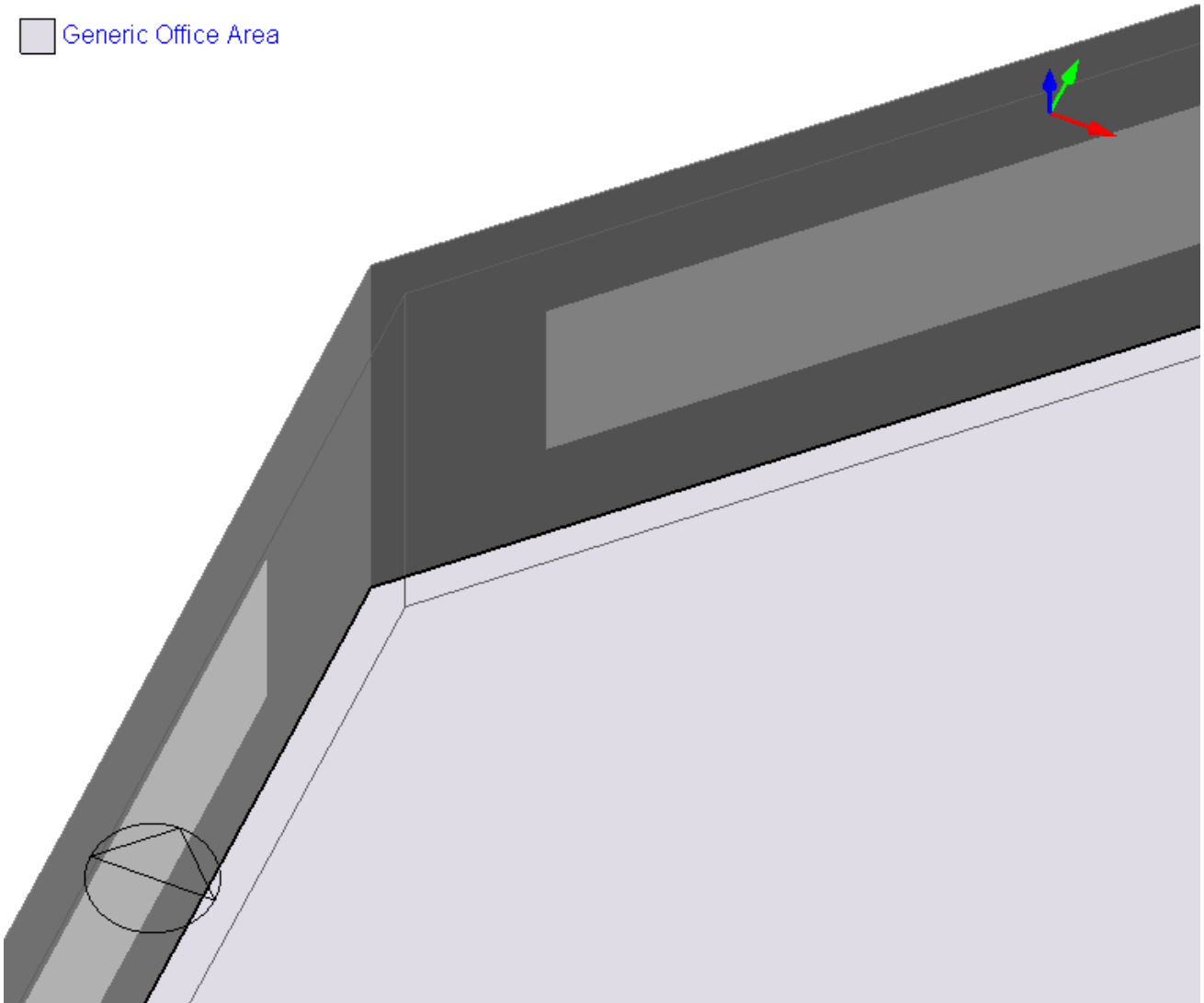
This option can be found under the **Display** header in the **Drawing Options** panel. When checked it ensures that the zone inner surfaces are displayed in addition to the block perimeter surfaces.

☐ Generic Office Area



Display zone inner surfaces OFF

 Generic Office Area



Display zone inner surfaces ON

By default the option is switched off and the zone inner surfaces are only represented by a dotted line on the floor plane. To get a clearer picture of where all of the zone inner surfaces are, you can switch this option on.

Zone Block By Drawing Partitions



Building blocks can be divided into any number of zones by drawing partitions inside the block. To start drawing partitions, click on the 'Draw partitions' tool. You can place partitions freely within a block either free-standing or snapped to an external block wall or existing partition. Partitions can be positioned using the [snaps, drawing guides and protractor](#). A partition doesn't need to be snapped to a wall element, you can drag a partition across a wall and when you click the end-point, it will be automatically trimmed to the wall.

You can enter the thickness of the first partition drawn in the block and subsequent partitions take on the same thickness as the first.

See also the [Partitioning blocks to create zones](#) tutorial

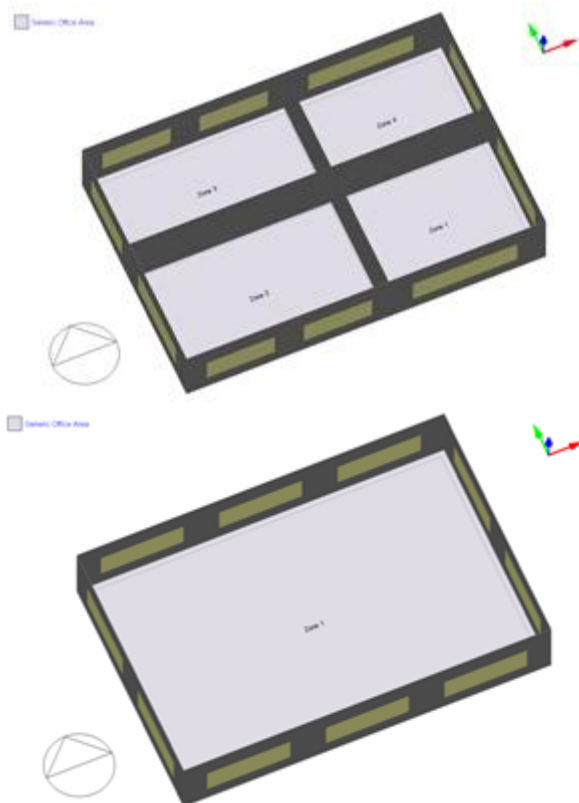
Note for partitioning large models

By default, blocks are automatically re-zoned as soon as complete zone perimeters are formed by drawing partitions. In order to speed up the process of adding partitions, particularly in the case of large and complex blocks, the block zoning process can be postponed until all partitioning for the current block has been completed. The easiest way to do this is to switch off the **Automatic block zoning** option under the **Display** header in the Drawing Options panel while drawing partitions. You also achieve the same effect by opening the Model options dialog and switching off the **Automatic block zoning** option under the **Miscellaneous Edit** header on the **Drawing Tools** tab. When partitioning has been completed, the block can be manually re-zoned by selecting the **Rezone block** command from the **Tools** menu. Partitions drawn with the **Automatic block zoning** option switched off are called **Outline partitions** and are shown in blue.

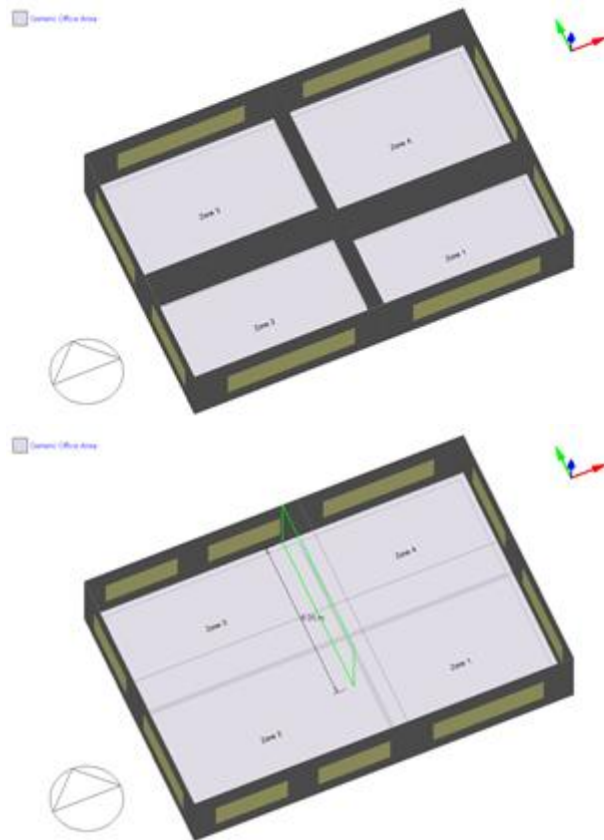
Rezoning blocks

When partitions are moved and/or deleted, DesignBuilder attempts to match the new zone geometry with the previous geometry and, if it is successful, it keeps the previous zone attributes. The matching is done based on overlaps with the floor areas of the old and new zone layouts. After partitions have been moved or deleted the software checks for overlaps in floor areas between new zones and previous zones and if overlaps are found, the attributes are inherited from the previous zones. The exception to this is for detailed HVAC zones where the floor area must match exactly for the new zone to be interpreted as a reworking of a previous zone.

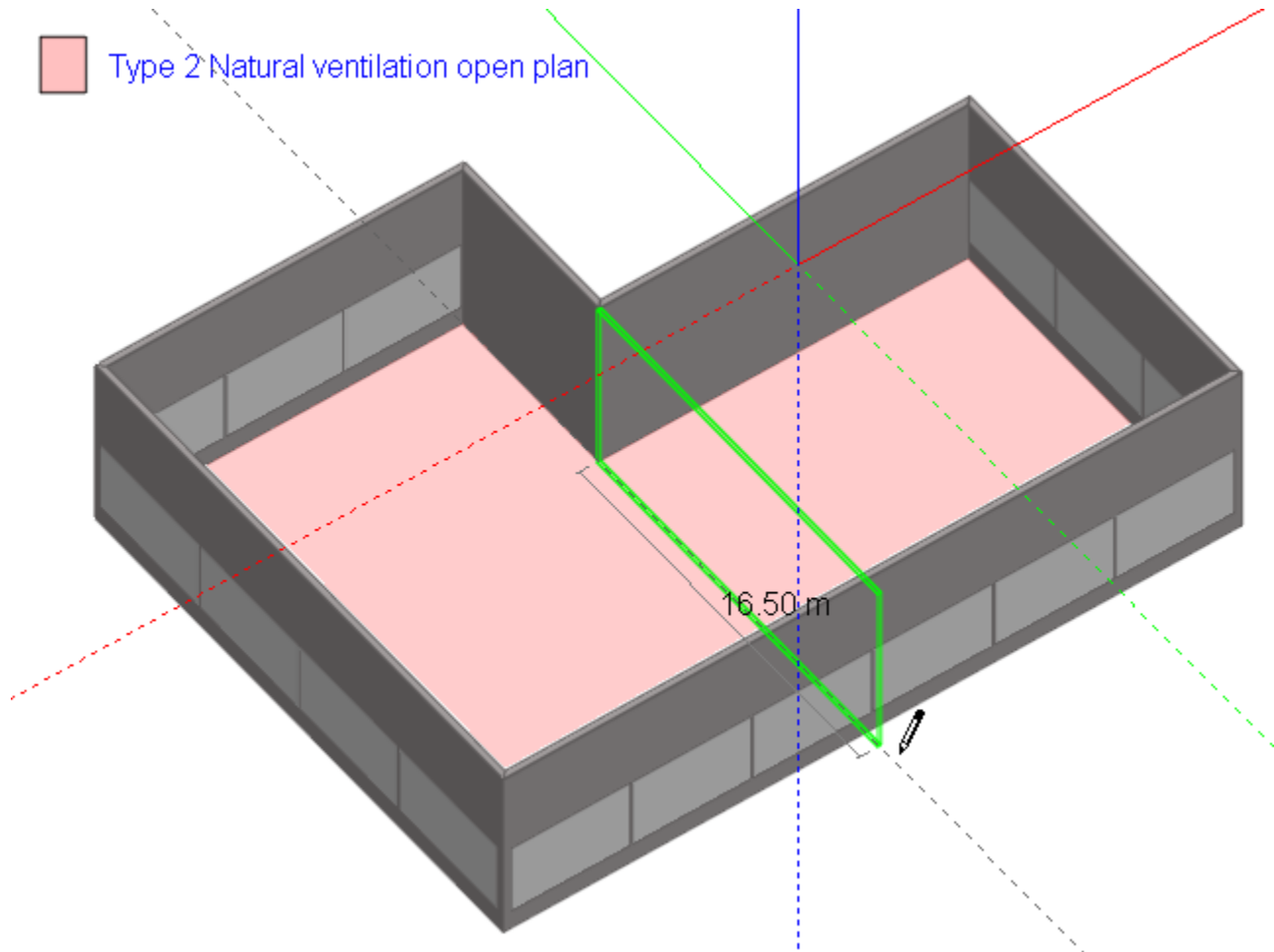
Switching off the **Automatic block zoning** option can be a big help when rezoning an existing block layout when aiming to maintain zone and surface attributes from the previous layout because it allows the timing of the matching process to be controlled. For example, in the following simple example, if all partitions are first deleted (with automatic block zoning turned on), the resulting single zone will inherit all attributes from the previous zone with the largest floor area and so any subsequently partitioned zones will inherit from that single zone:



However, if "automatic block zoning" is turned off prior to deleting the partitions and the block is manually rezoned after adding the new partitions, the four new zones will inherit from the original four zones:

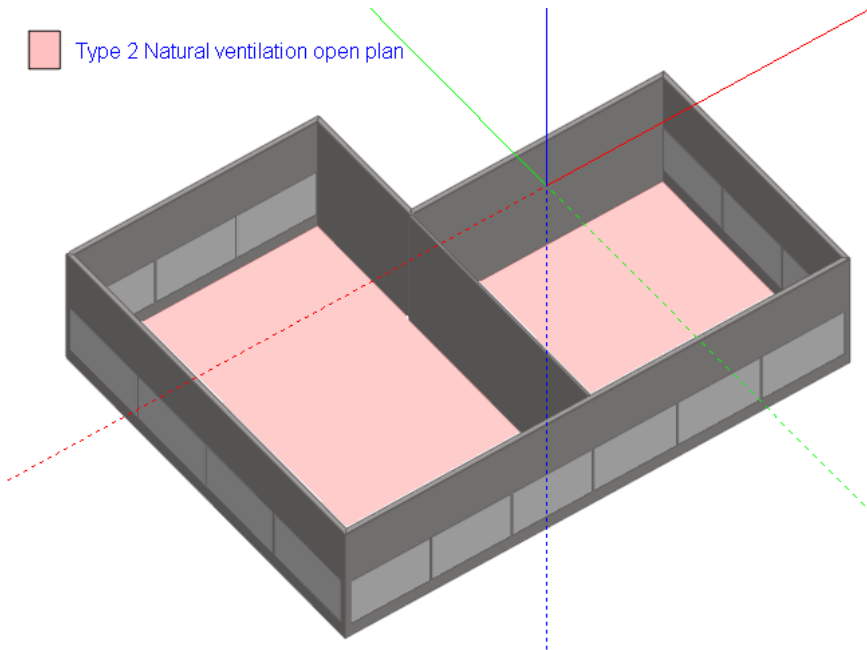


Partition Block Into Zones



Click the partition end-point on the outside of the wall and it's automatically trimmed:

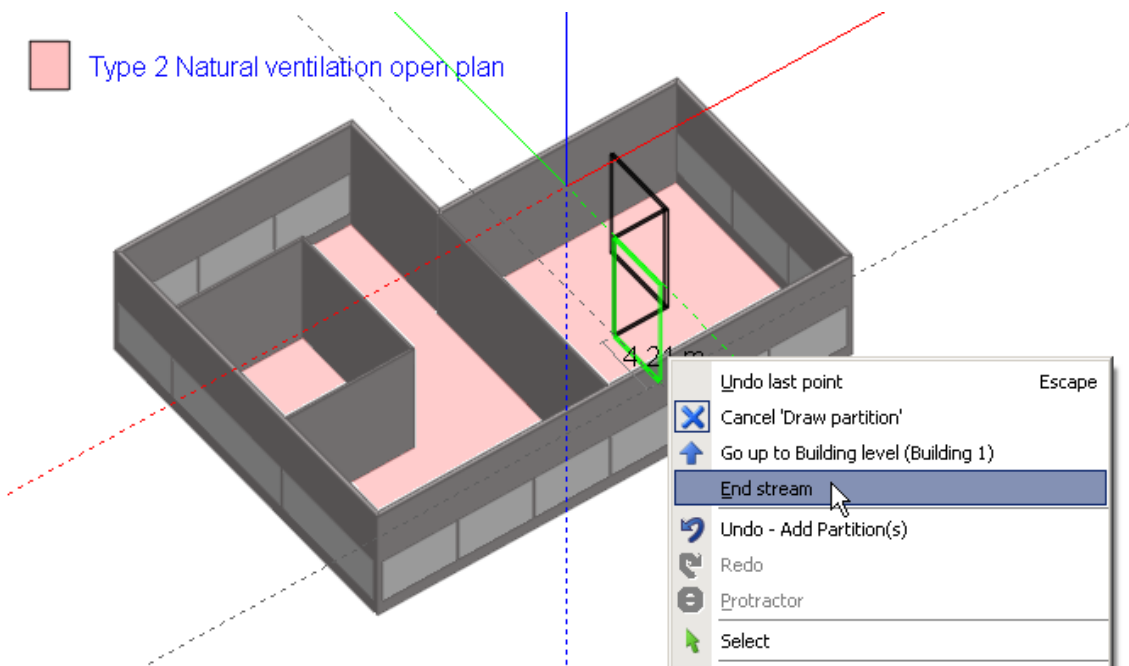
 Type 2 Natural ventilation open plan



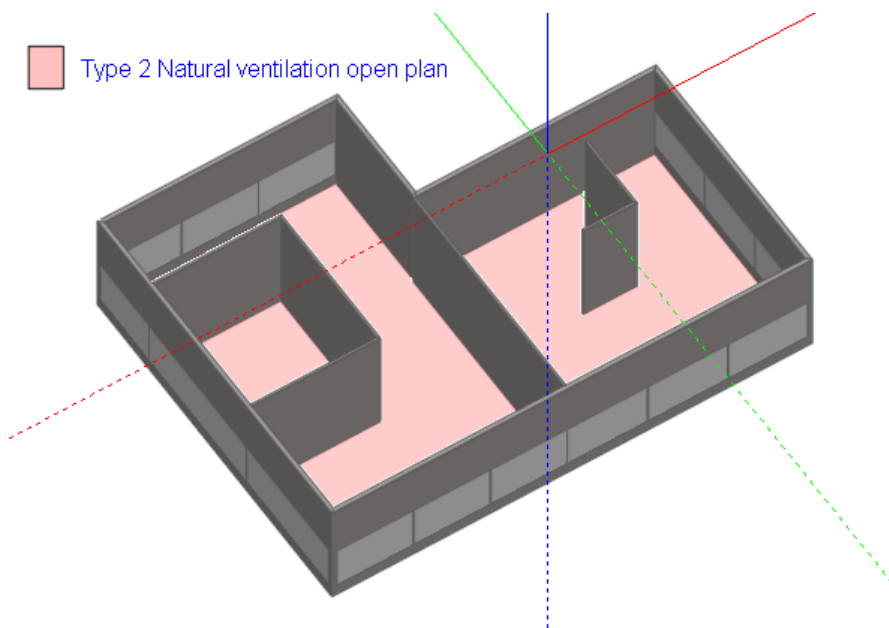
When you draw a partition or a series of partitions (stream), you can undo the last point placed by pressing the ESC key or selecting 'Undo last point' from the right-click menu.

You can also terminate a partition stream anywhere by selecting 'End stream' from the right-click menu (or by pressing the ENTER key):

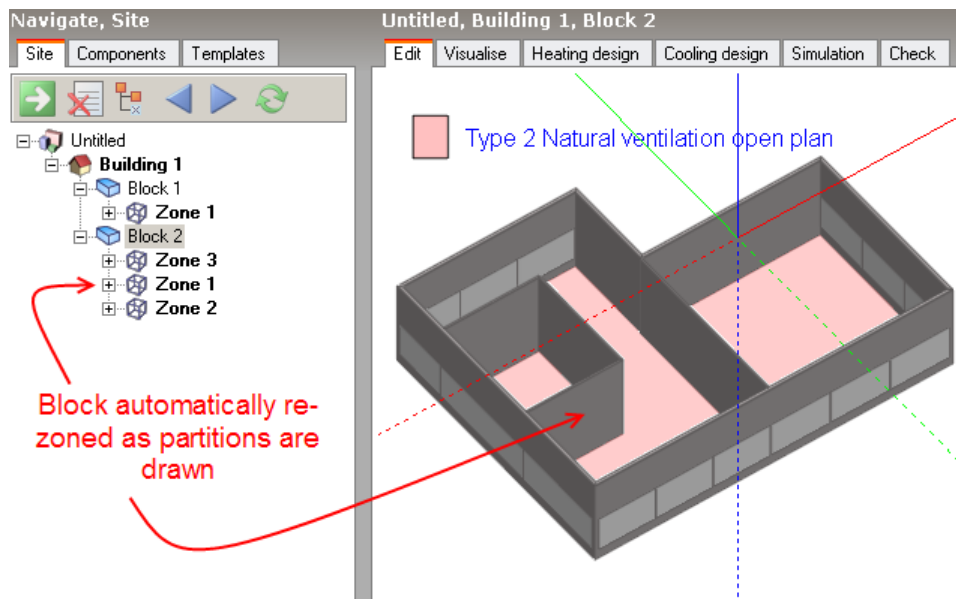
 Type 2 Natural ventilation open plan



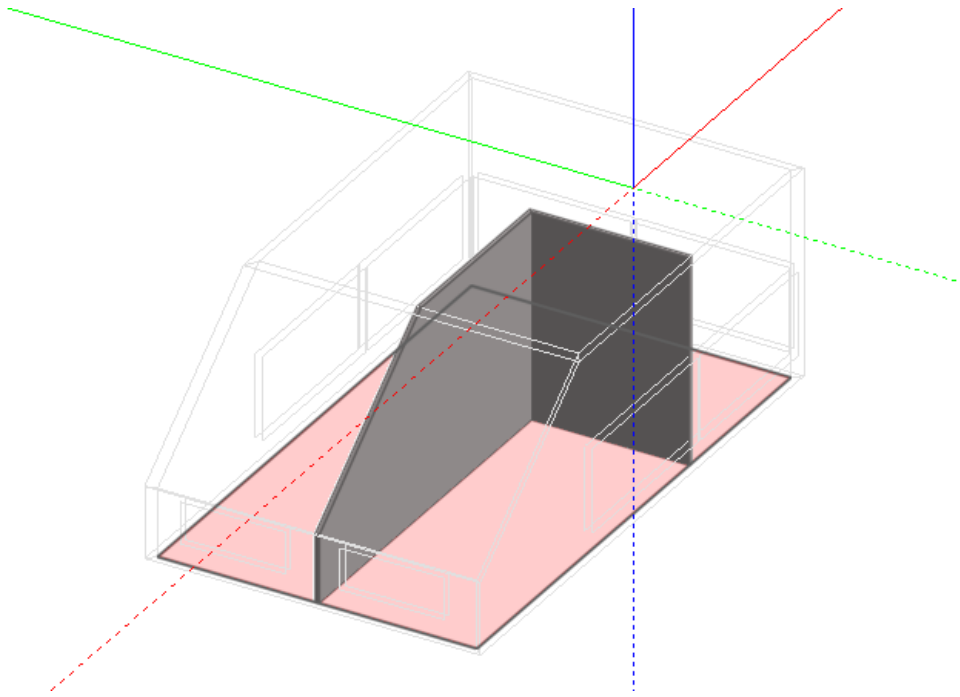
When partitions or partition streams are placed that don't connect with anything at either or both ends, they are called [hanging partitions](#) and can be used to add thermal mass to zones:



Once you've placed partitions that divide existing zones within a block, the block is re-zoned automatically:



When you partition complex blocks rather than simple vertical extrusions, partitions are automatically cut to fit within the block:



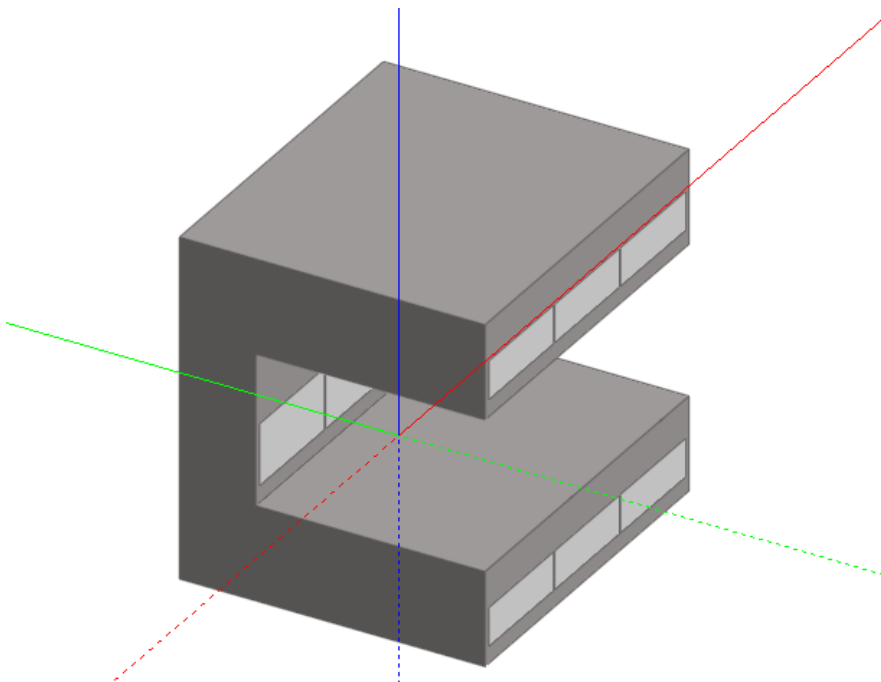
Tip: If the display of the block outer walls prevents you from drawing partitions (for example in a pitched roof block) then you can switch off the [display filled external walls](#) model option.

Deleting Partitions

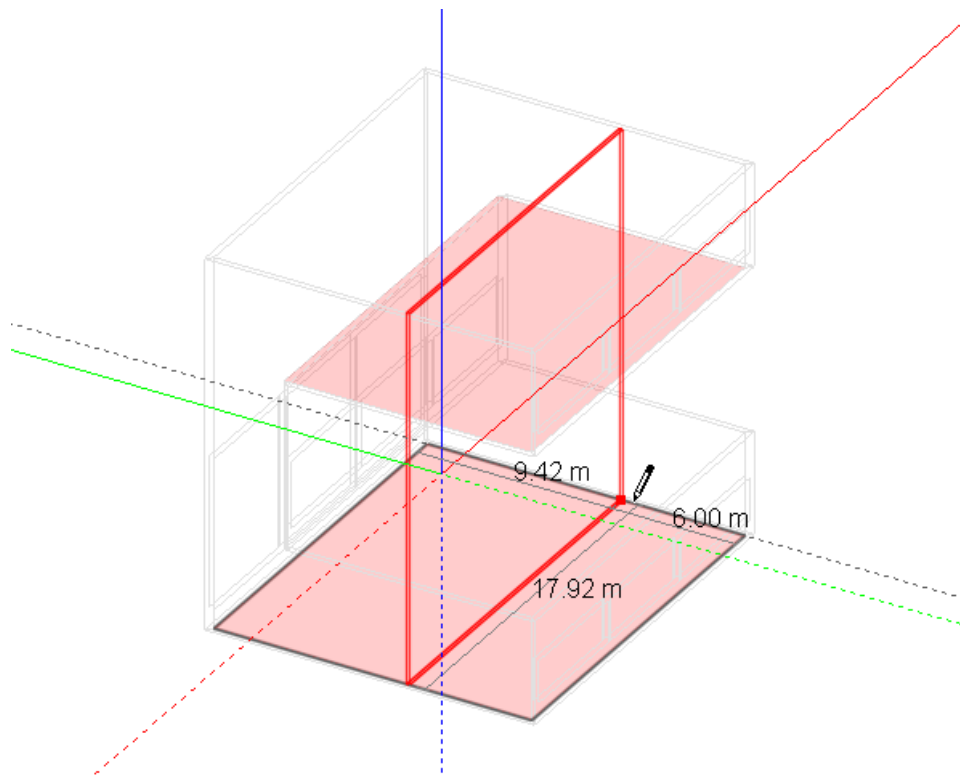
When a partition between 2 zones is deleted, the new zone created from the 2 previous zones will use the attributes from the original zone having the greatest floor area.

Complex Block Geometry

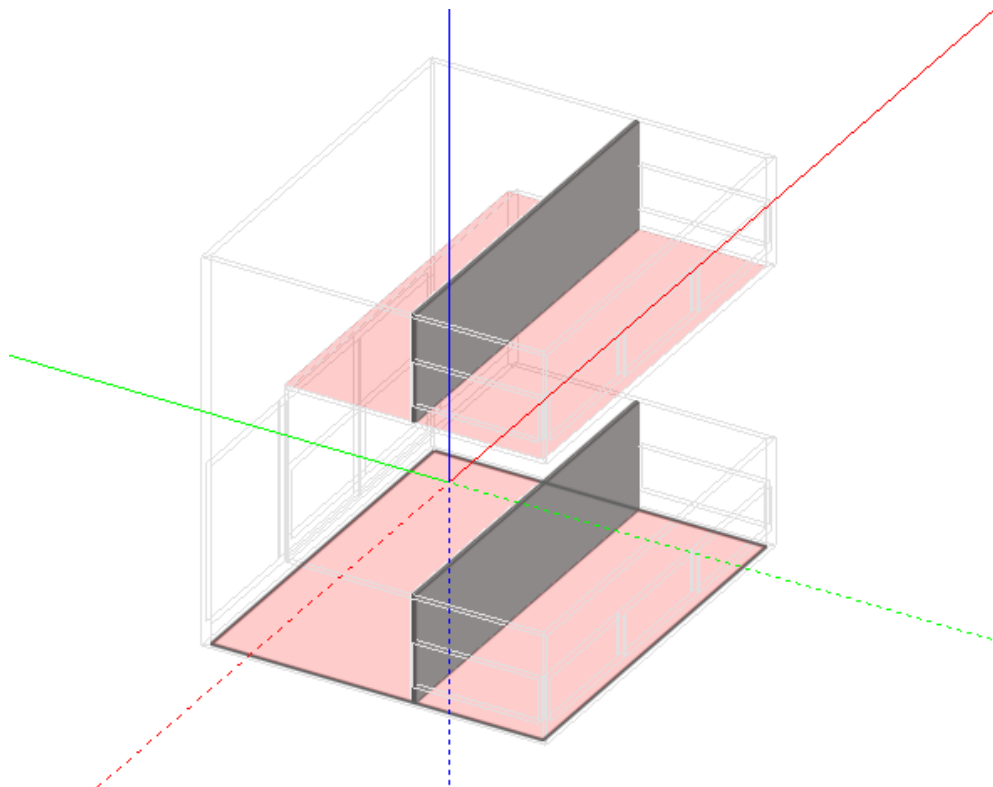
In some cases, if the geometry is complex, it can be possible to create a zone that comprises several geometric bodies which are physically separated from each other. For example, in the case of a horizontally extruded 'C' shape:



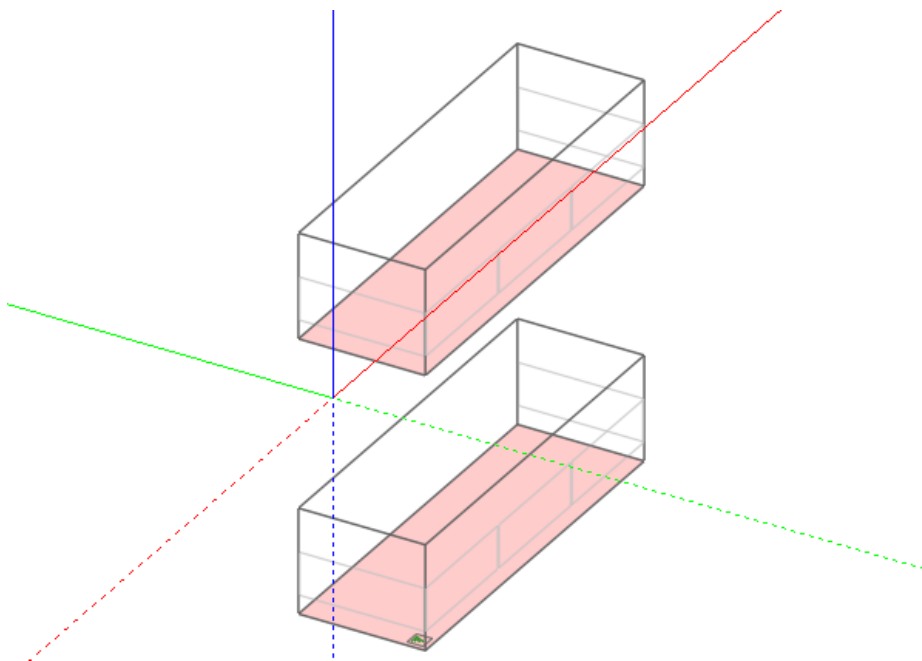
If a partition is drawn as shown:



The partition and resulting zone will be cut to form what appears to be two partitions and two zones:



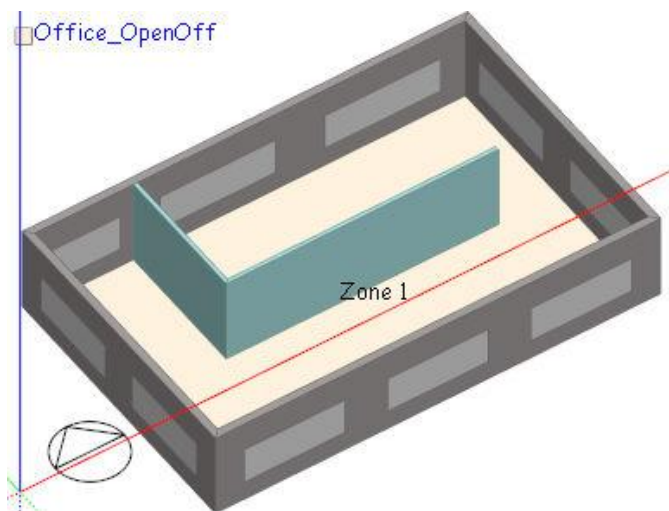
However the two resulting zones are in fact two geometric bodies within one zone:



While this geometry is perfectly acceptable for the default EnergyPlus solar calculations (see Model Options), it cannot be processed by the more rigorous [Full exterior and interior](#) solar distribution option. If this latter option is required, you can either model the geometry as two separate blocks or [cut the block](#).

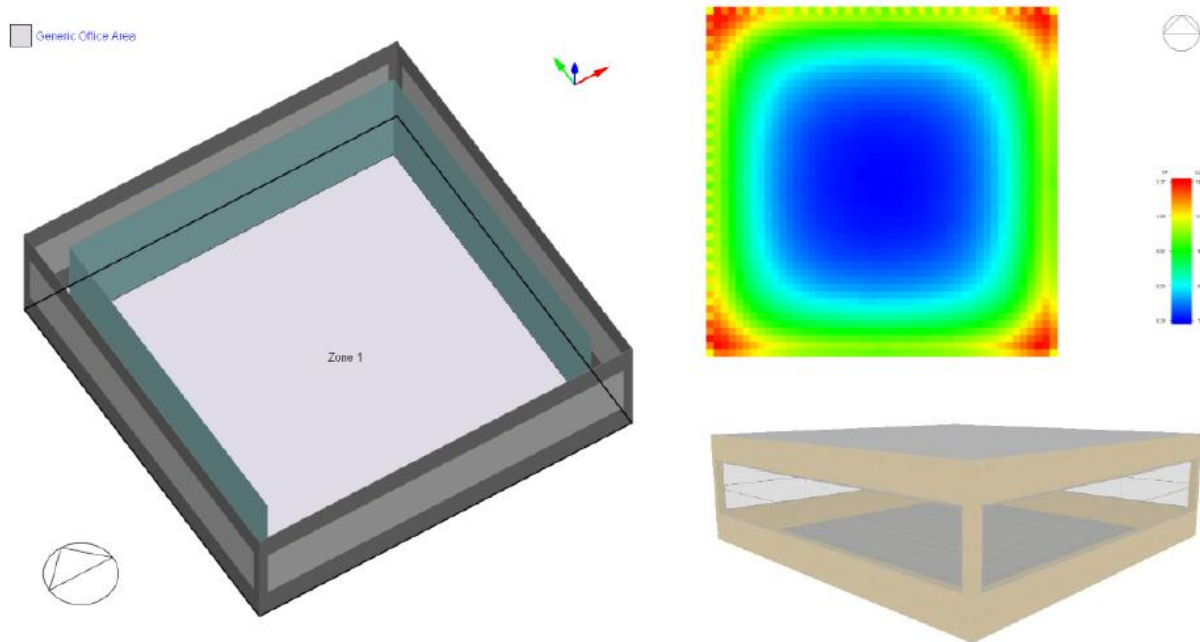
Hanging And Outline Partitions

Hanging partitions are partitions that do not create zones because they do not connect at both ends with other walls and/or partitions. Hanging partitions are created in the same way as ordinary partitions using the [Draw partition](#) tool, but are finished by pressing <Enter> (or the **End partition stream** command from the right click menu) to define the end of a partition stream. They are shown in light blue (see below) to differentiate them from normal and virtual partitions.



Hanging partitions are modelled non-geometrically in EnergyPlus as [Internal thermal mass](#).

Note: Hanging partitions are not included in visualisation, daylighting and CFD studies (see below). You should use internal component blocks to create an obstruction to daylight, CFD air movement or for more realistic visualisations.



Outline partitions

Also if [Automatic block zoning](#) is switched off, all new partitions are treated as outline partitions (shown in blue and do not create zones) until converted to standard partitions by using the [Rezone block](#) command from the [Tools](#) menu.

Virtual Partitions

A virtual partition is a partition between 2 zones which exists purely to sub-divide the space up and has no corresponding wall in the actual building.

Virtual partitions are commonly used for separating perimeter zones from core zones when there is to be different HVAC provision or when carrying out daylighting or solar overheating studies in situations where a large open plan space is subject to a high level of solar gain around the perimeter. Virtual partitions can be placed to create separate perimeter and core zones so that the local effect of the solar gain in the perimeter zone may be calculated. In the absence of virtual partitions, the risk of overheating could be underestimated due to the distribution of solar gain throughout the open plan space.

When drawing partitions you can set the Partition type in the Drawing option panel to **1-Standard** (the default for a standard partition) or to **2-Virtual** to create a virtual partition.

For partitions that have already been created you can navigate to the partition (click on the appropriate item in the Navigator or double-click on the partition object from either zone or block level) and change the partition type between **1-Standard** and **2-Virtual** using the Partition type Drawing option control. This is only possible for partitions within a block and inter-block partitions cannot have their partition type set to **2-Virtual**.

Note: Virtual partitions are modelled using a hole that fills the whole surface area so you cannot draw custom openings on a virtual partition. For details on how DesignBuilder EnergyPlus models virtual partitions see information on [holes](#). The [model options](#) for controlling the effect of holes apply equally to virtual partitions.

See also:

- [Merge zones connected by holes](#)
- [How airflow through holes \(and virtual partitions\) is modelled](#)
- [Natural ventilation options](#)

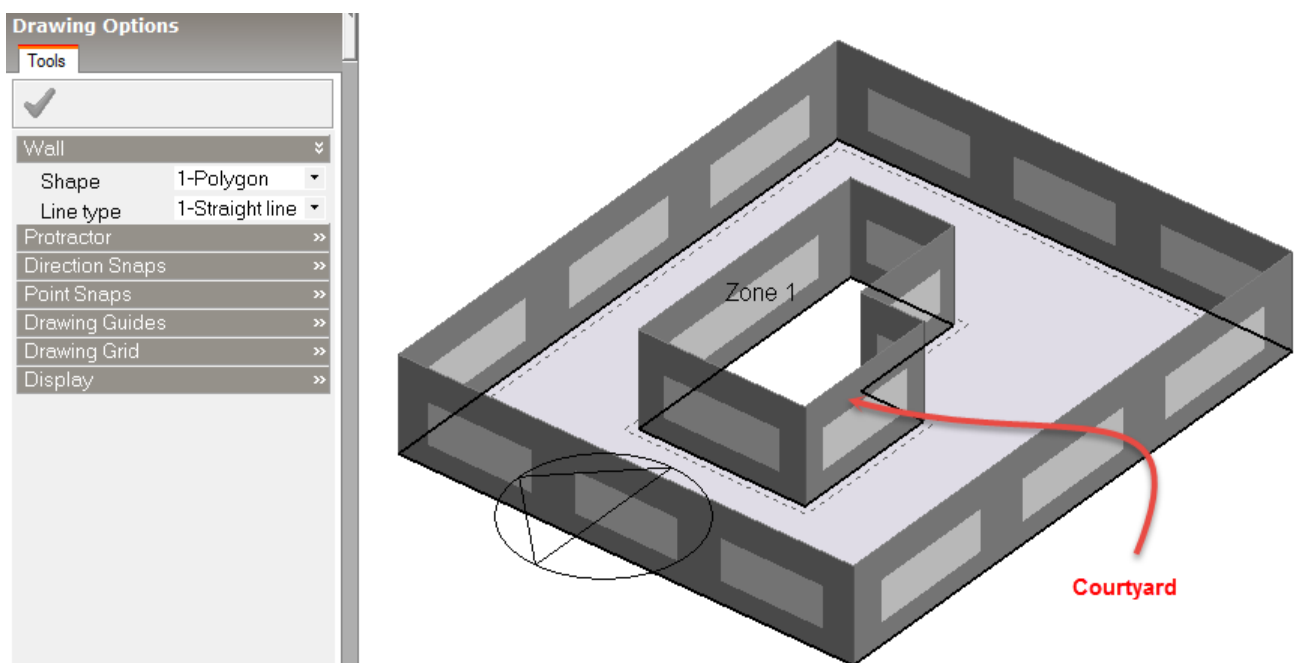
See also the [Partitioning blocks to create zones](#) tutorial

Creating Voids (Courtyard Spaces) Within Blocks



To create an external courtyard within a block, go to the required block by clicking on the navigator entry or double-clicking the block and then click on the **Draw void perimeter** tool. You can select a perimeter [shape](#) and the line type used to draw it in the **Drawing Options** data panel:

After completing the perimeter, a courtyard is automatically generated within the block:



Important note: in simulations, the void space is treated as **exterior** to the building and not as any sort of zone. If you need to create an atrium space whose temperature and other conditions are simulated then avoid use of the **Draw void perimeter** tool and instead model it like any other zone within the block using partitions.

If you would like DesignBuilder to automatically punch voids through new blocks drawn on top of existing blocks having voids, then use the [Automatically include voids from lower blocks](#) model option which can be set on the [Miscellaneous Edit](#) tab of the model options dialog.

Restriction: The void perimeter lines you draw to create a void cannot touch any other external or internal wall, so you can't for example use the void command to take chunks out of the external wall - the void must be wholly contained within the block.

Restriction: You can't create any other objects within a void - the void space is reserved and you will get intersection errors if you try to draw a block in the void space. If you need to do this then you should create a "hard" void using [Boolean tools](#) instead (see below).

Soft And Hard Voids

There are 2 ways to create voids, such as courtyards, in DesignBuilder:

- **"Soft" voids** are the courtyards etc added to a building block using the **Draw void perimeter** tool described above. Soft voids are "soft" because they can be moved and generally edited.
- **"Hard" voids** are created by cutting a hole through a block using a **Boolean** operation.

The main practical differences between "soft" and "hard" voids are that:

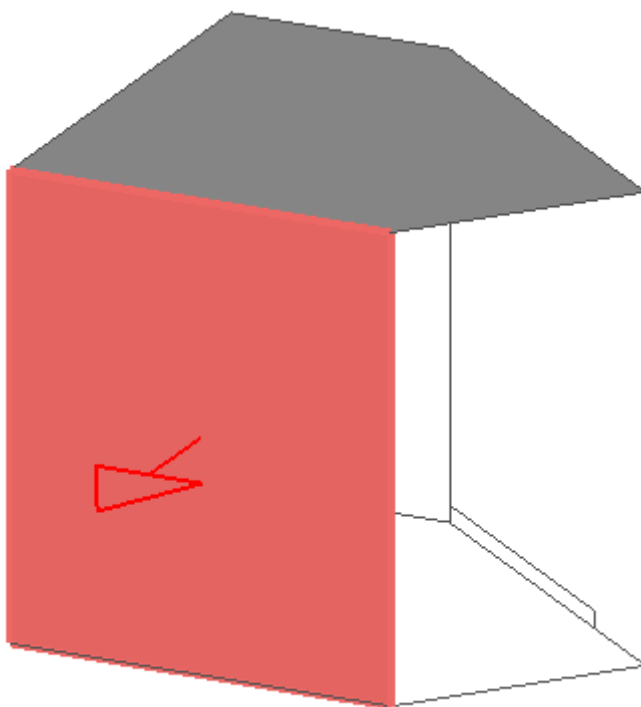
- another block (building block, component block, solar panel, assembly) can be located within a hard void but this is not possible within a soft void, and,
- soft voids can be edited at block level using the delete, copy, move, rotate, etc. commands.

Open Manifold Blocks - Flip Block Surface Orientation

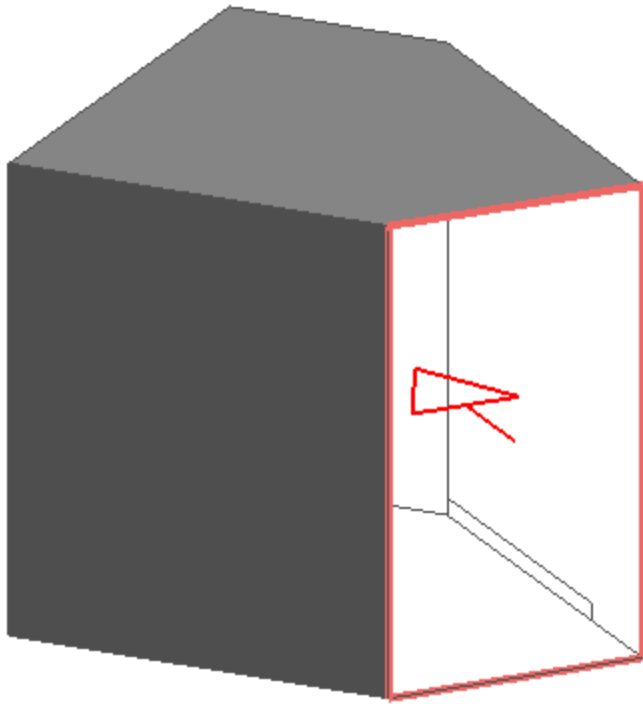
This command applies only to open manifold blocks imported from BIM models. It can be accessed from the Edit screen when at building level when an open manifold block is selected or when at block level for an open manifold block. Using the **Ctrl-F** keyboard shortcut is the fastest way to start the flip block command.

Open manifold blocks cannot use all geometry operations and the surface orientation cannot be reliably determined from the model. DesignBuilder must rely on the orientation data stored with the surface from the BIM model. This data is not always reliable and so in some cases surfaces come through into DesignBuilder facing in the wrong orientation. The **Flip block surface orientation** command can and should be used to check and, if necessary fix, the orientation of surfaces in all open manifold blocks.

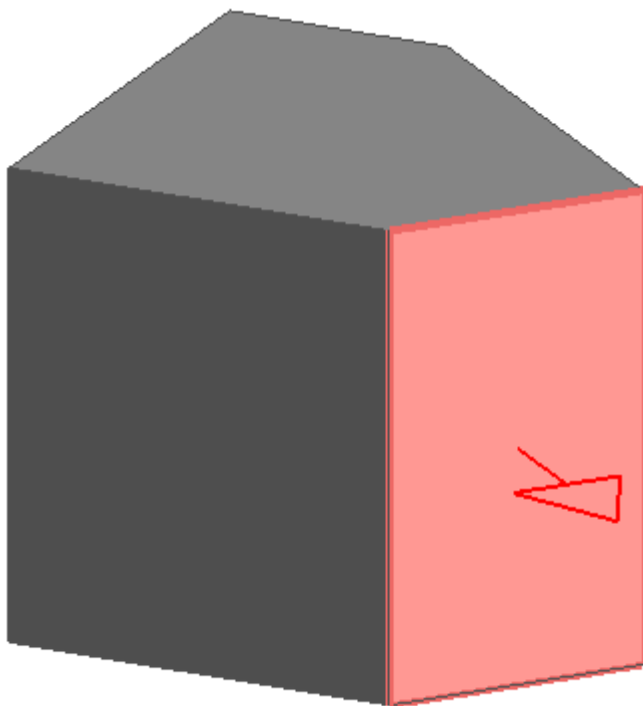
When the Flip Block Surface Orientation command is active you can click on the surfaces in a block to see their orientation. You should see the red arrow pointing **outwards** away from the centre of the block as shown below.



Some surfaces may be facing in the wrong direction (inwards) and will often be seen in white looking from outside the block as shown below.



Clicking on these surfaces shows the arrow pointing inwards. In this case press the **Shift** key while left clicking with the mouse to flip the direction of the surface to make it point outwards. The image below shows the fixed block looks after the Shift mouse click was used to flip the surface orientation.



All such inward facing surfaces must be flipped to ensure an accurate model before using it to run energy simulations. Otherwise inward-facing surfaces will have the wrong orientation in energy simulations.

Note: even when all surfaces in open manifold blocks have been correctly flipped these models cannot be used in Radiance daylighting and CFD calculations which require all blocks to be closed manifold.

CFD Circulation Fans



One or more CFD Circulation Fans (often referred to as CFD fans) can be added at block level to increase air momentum and add/subtract heat during CFD simulations. The supplied flow rates are converted to momentum equation boundary conditions by combining them with the associated cell dimensions.

Example applications are jet fans and desk fans which can both be modelled using the CFD circulation fan to add movement to the air.

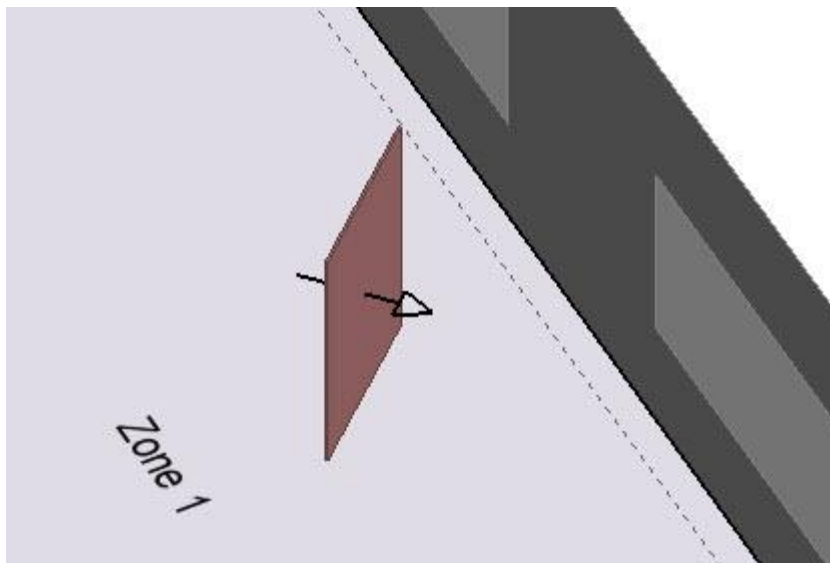
CFD Fans are only used for CFD simulations and have no impact on any other calculations.

Add CFD Circulation Fan

The position and size of a CFD fan is defined by drawing a line (typically in the X-Y plane) and extruding to create a rectangular plane. Air will flow through the fan, perpendicular to the fan plane.

To add a CFD fan:

1. Navigate to the block where the fan is to be placed.
2. Click on the **Add CFD circulation fan** toolbar icon.
3. Move the mouse cursor to the starting position of the fan plane and left click with the mouse to start drawing.
4. Move the mouse to the end position of the fan plane and left click again to complete the base line.
5. Now move the mouse perpendicular to the line to extrude creating a plane. Once you have set the required extrusion length, left click to complete the fan geometry.
6. Once the fan has been drawn, you may want to move the fan to final position (e.g. on to a desk) or rotate it etc. The screenshot below shows a fan that has been moved off the ground (using the [Move](#) command with z-snap) and [rotated](#).



Fans can be drawn onto the block floor, onto component or outline blocks or onto walls of the block.

7. Having now drawn (and possibly repositioned) the fan, the final step is to define boundary conditions for any heat flux added by the fan (e.g. fan heater), the flow rate and the direction.
8. Navigate to the fan either by double clicking on the object in the model view or by clicking on the CFD Fan item in the Navigator and click on the [CFD](#) model data tab.
9. Enter any heat flux added by the fan, the fan flow rate and the direction as shown in the screenshot below.

Internal Analysis Boundary Defaults	
Component Settings	
Heat flux (convective fraction only) ...	100.000
Flow rate (l/s)	50.00000
Flow direction	1-Positive

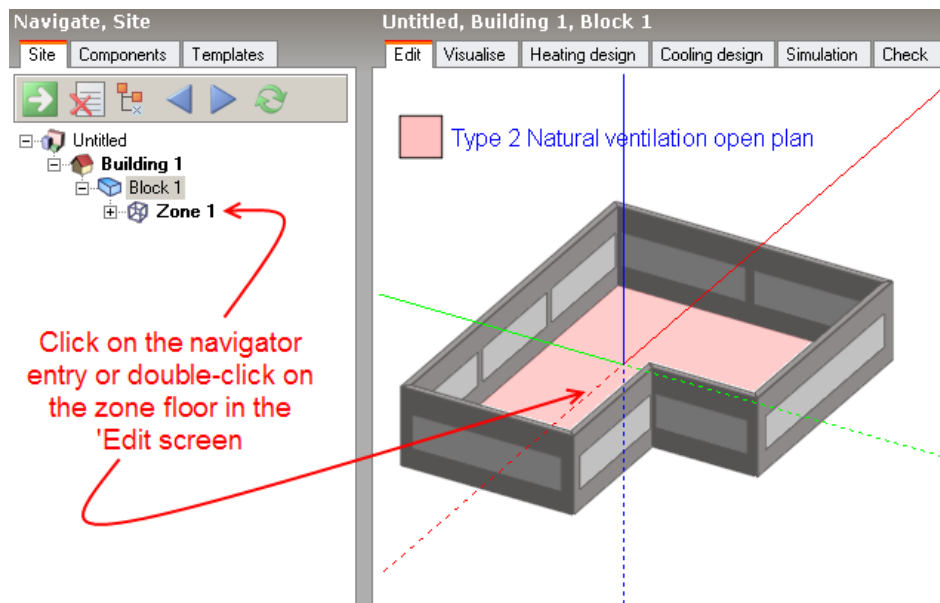
- To change the direction of flow, change the **Flow direction** setting from **1-Positive** to **2-Negative** or vice-versa.

Note: The fan discharge area dictates the fan discharge velocity and consequently the momentum for a given flow rate.

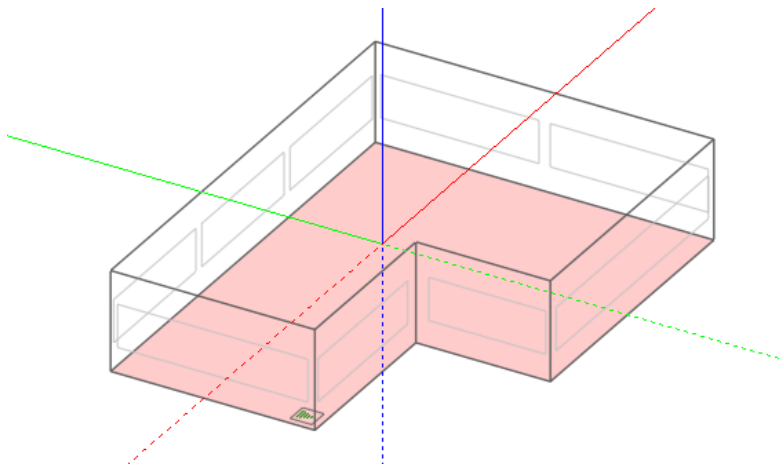
Working Within Zones (Zone Level)

This section describes the operations involved in placing [light sensors](#) within zones. To [place light sensors](#) within a zone, you first need to go to the zone that you want to modify. You can do this from the block level either by clicking on the navigator entry or double-clicking on the zone floor in the 'Edit' screen if you're at the block level.

Go to zone level



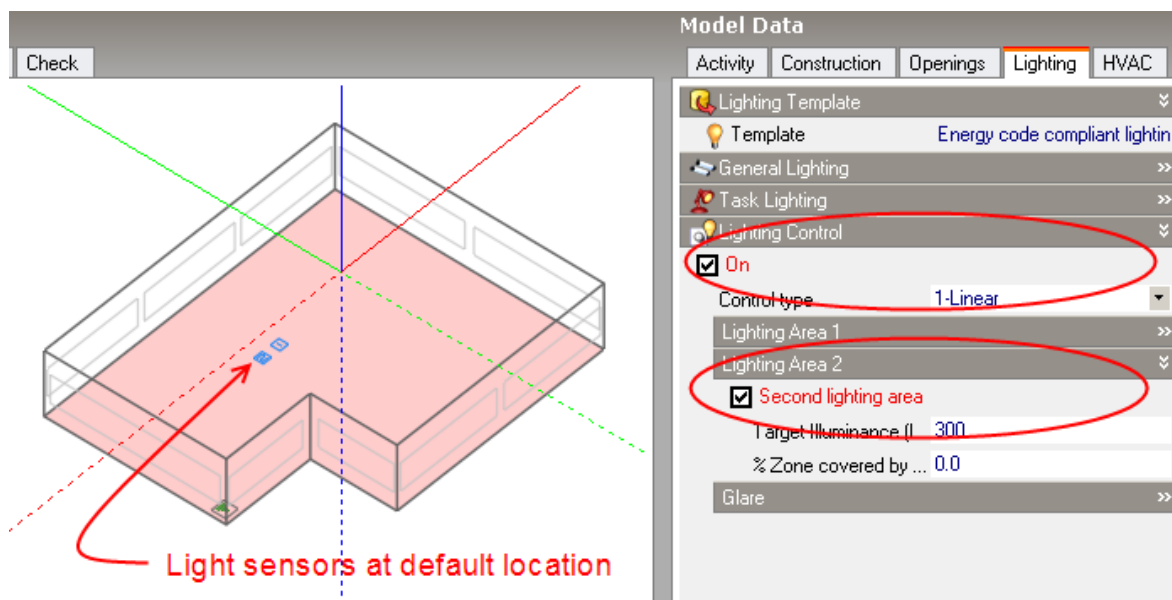
Either of these operations will take you to the zone:



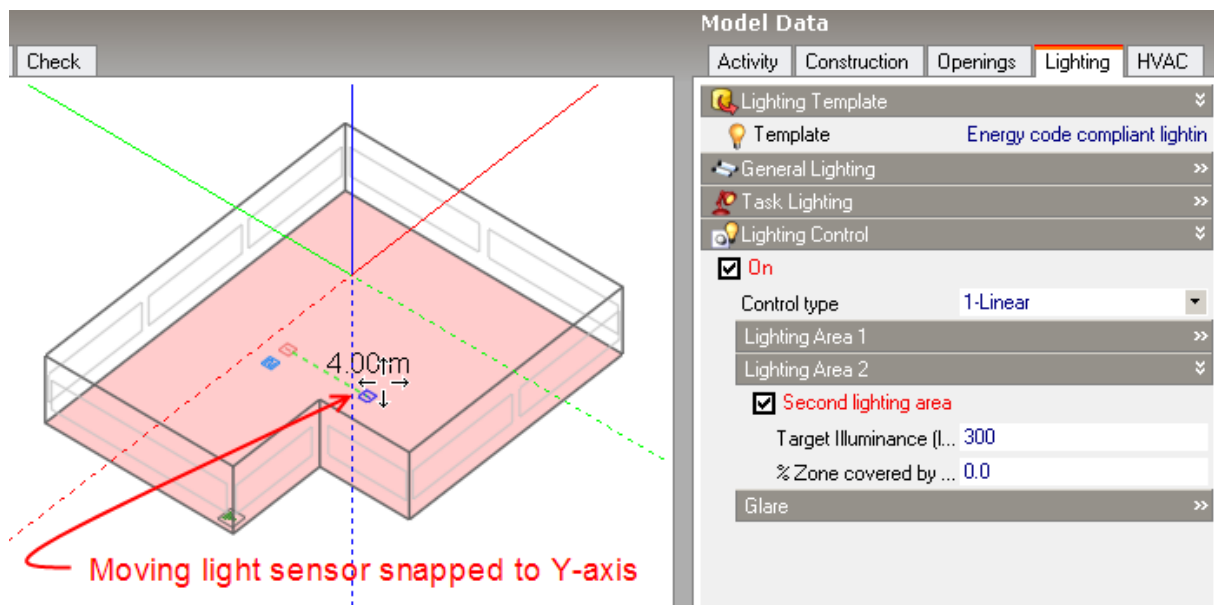
Placing Light Sensors

If you switch zone [Lighting Control](#) on, a light sensor will be displayed as a blue rectangle with the number '1' inside it indicating 'Light sensor 1'. By default Light sensor 1 is positioned in the geometric centre of the zone.

If you now switch on the [Second Lighting Area](#), a second light sensor will be displayed with a number '2' inside it:



To move a light sensor to a different location, move the mouse cursor over the required sensor and click on it to select it and then [Move](#) it to wherever you want within the floor perimeter:



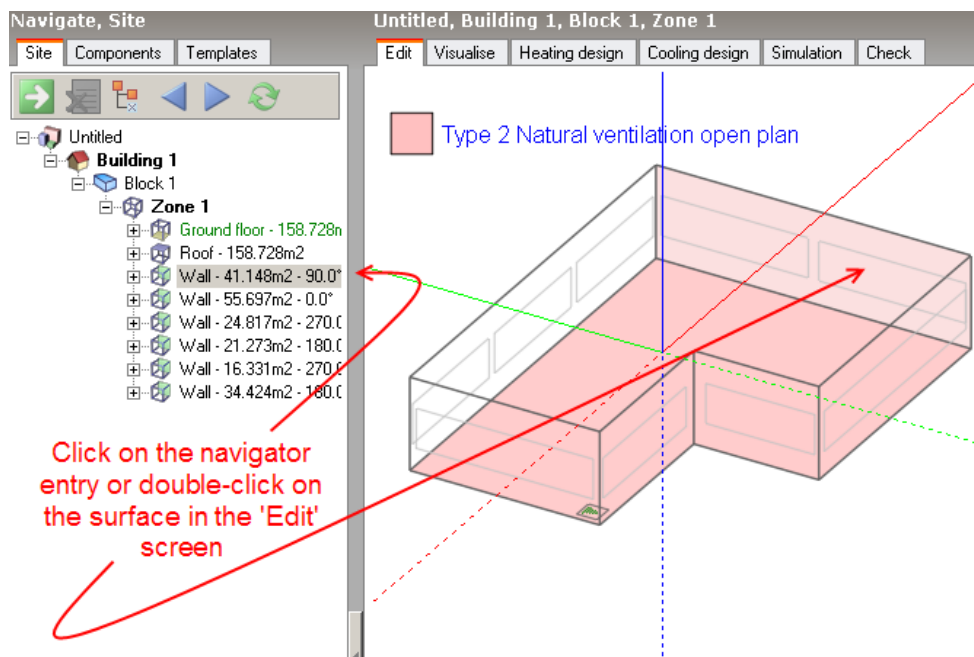
Note: You can set the [working plane height](#) to define the desk height (and hence the z-position of the lighting sensor) on the [Lighting tab](#).

See also the [Lighting data and daylight controls](#) tutorial

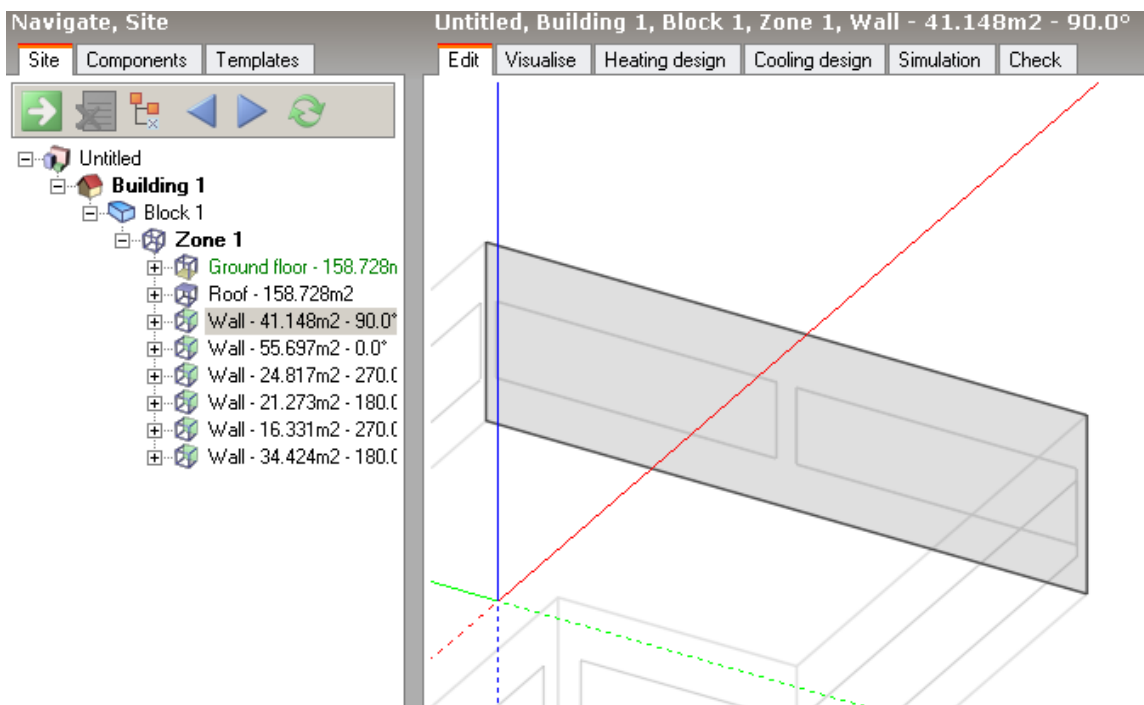
Working With Surfaces (Surface Level)

This section describes the operations involved in creating surface openings and sub-surfaces. To [create openings](#) and sub-surfaces in a zone surface, you first need to go to the surface that you want to modify. You can do this from the zone level either by clicking on the navigator entry or double-clicking on the zone surface in the 'Edit' screen, if you're at the zone level.

Go to surface level

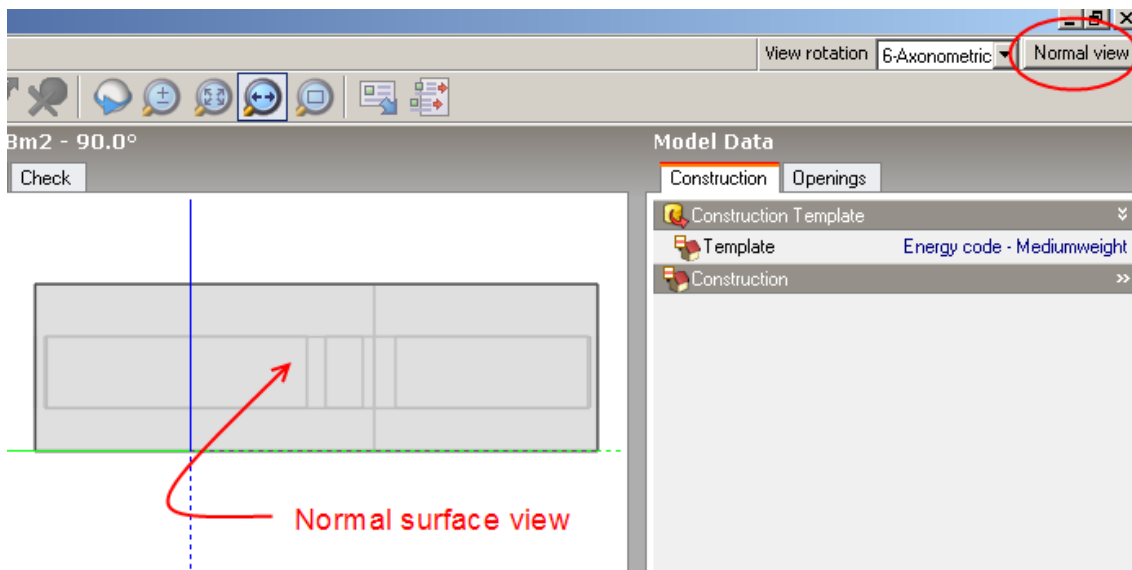


Either of these operations will take you to the surface:



Note: When navigating to surfaces from zone level by clicking on them with the mouse, to access the "backward-facing" surfaces (i.e. surfaces whose *inside* face is shown) you must hold the Shift key down while clicking.

When working with surfaces, switching to [Normal view](#) rotation can be helpful in orientating openings and sub-surfaces:



Openings

You can draw, move, clone (copy), rotate and delete the following types of opening:

- [Window](#)
- [Sub-surface](#)
- [Hole](#)
- [Door](#)

- [Vent](#)

Note that custom openings that have been drawn by hand or copied are displayed in bright colours to differentiate them from (grey) default openings:

- Yellow - windows
- Blue - sub-surfaces
- Green - holes
- Cyan - doors
- Magenta - vents

Note: All openings and sub-surfaces in adiabatic surfaces are ignored in EnergyPlus simulations

See also the [Drawing windows and doors](#) tutorial

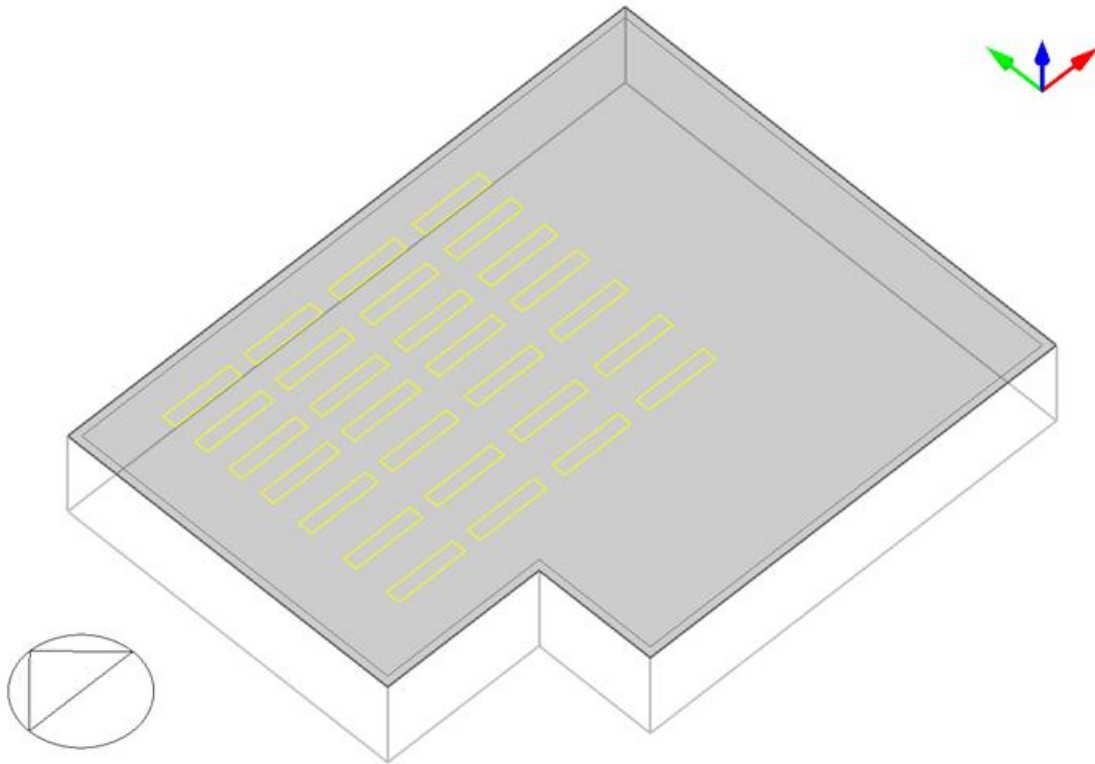
Surface Polygons Generated for EnergyPlus - Technical Description

EnergyPlus requires all surfaces to be convex and it does not support surfaces with holes. But of course real-world buildings can have surfaces of any shape. So when DesignBuilder encounters a concave surface or a surface with one or more holes (not openings, but an actual hard geometric hole) it must break the DesignBuilder surface down in to 2 or more polygons. The purpose of this technical description is to explain how that is done. With an understanding of the process applied users may be able to find ways to configure their model geometry to minimise the number of polygons DesignBuilder needs to generate to meet EnergyPlus' requirements.

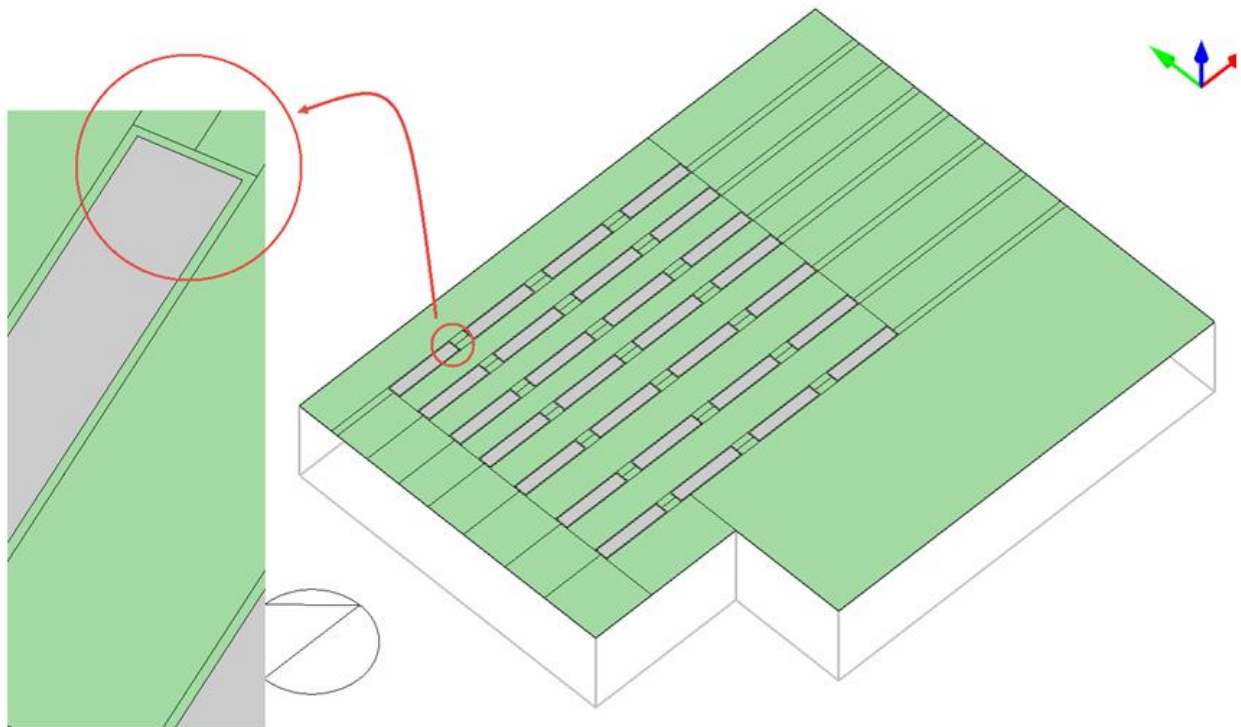
In order to ensure that valid polygons are provided to EnergyPlus, DesignBuilder follows the process below for concave surfaces with windows:

1. Inflates windows to [account for any frames specified](#) and to provide a 1mm margin around the window.
2. Uses the inflated window size to create a parent surface for each window. A similar process is used for other opening types except that doors, vents, sub-surfaces etc do not have frames, so that part of the inflation is not required.
3. Further sub-divides the surface into convex polygons to ensure that all surfaces provided to EnergyPlus are valid.

The screenshot below illustrates an example case of a concave roof surface with many windows:



DesignBuilder breaks this surface/opening configuration down into EnergyPlus polygons as shown below:



Note the rectangular polygons around each window.

Draw Window

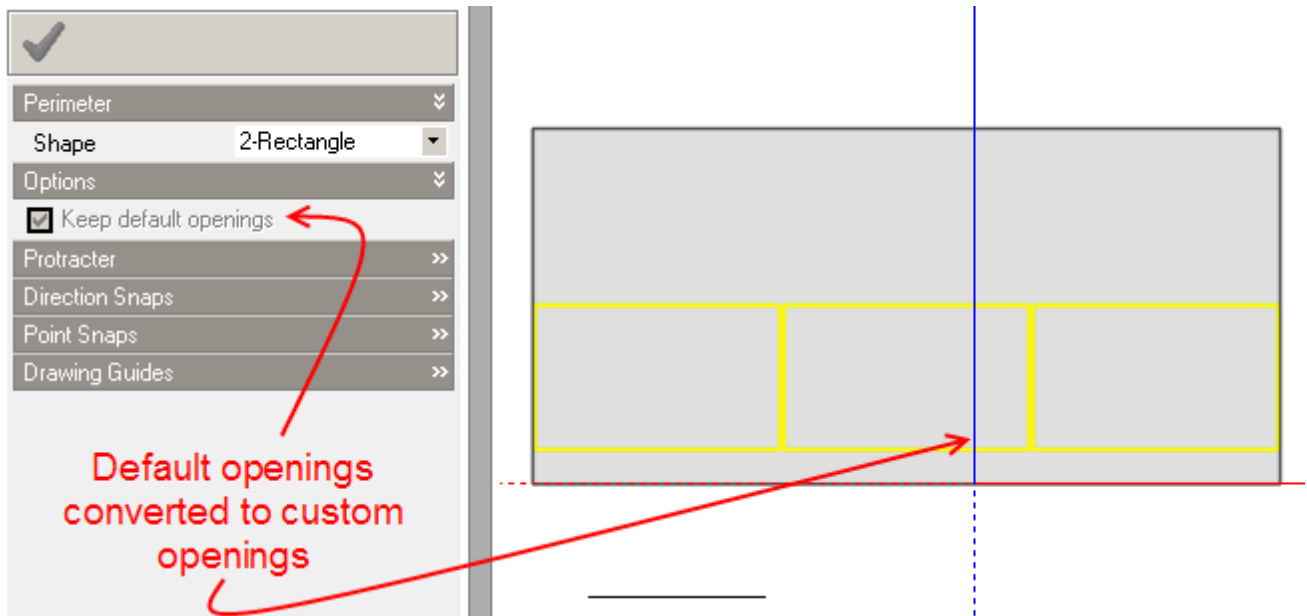


Before activating the **Draw window** tool, you are given the option of keeping any default surface openings through a check box in the Display options dialog. If you choose to keep these openings, they will be converted to custom openings and so will not be subsequently affected by any changes to the façade attribute data and must be deleted or edited manually, or [cleared back to default openings](#).

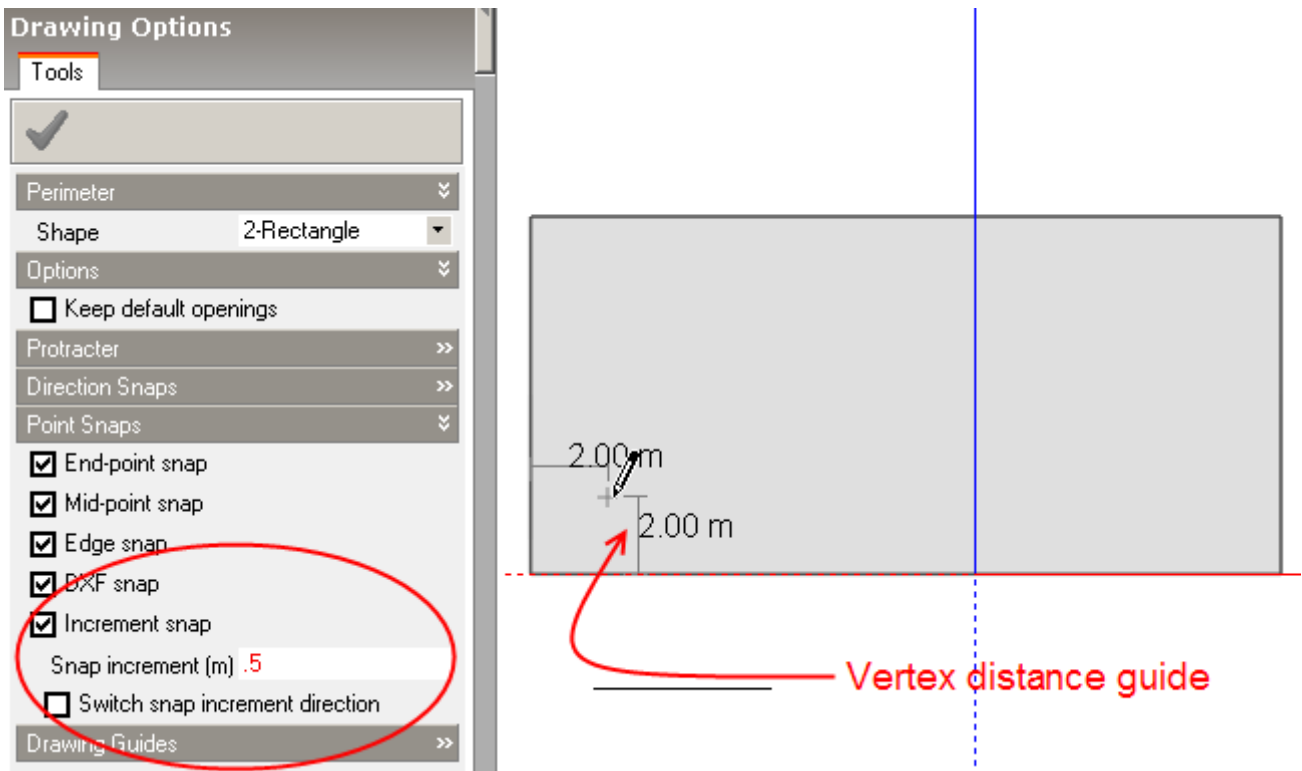
Note: When drawing windows you should draw the whole opening area in the fabric including the space required for the frame. The way that the frame and window areas are provided to EnergyPlus is described in more detail in the "Dimensions (Walls)" on page help topic.

See also the [Drawing windows and doors](#) tutorial

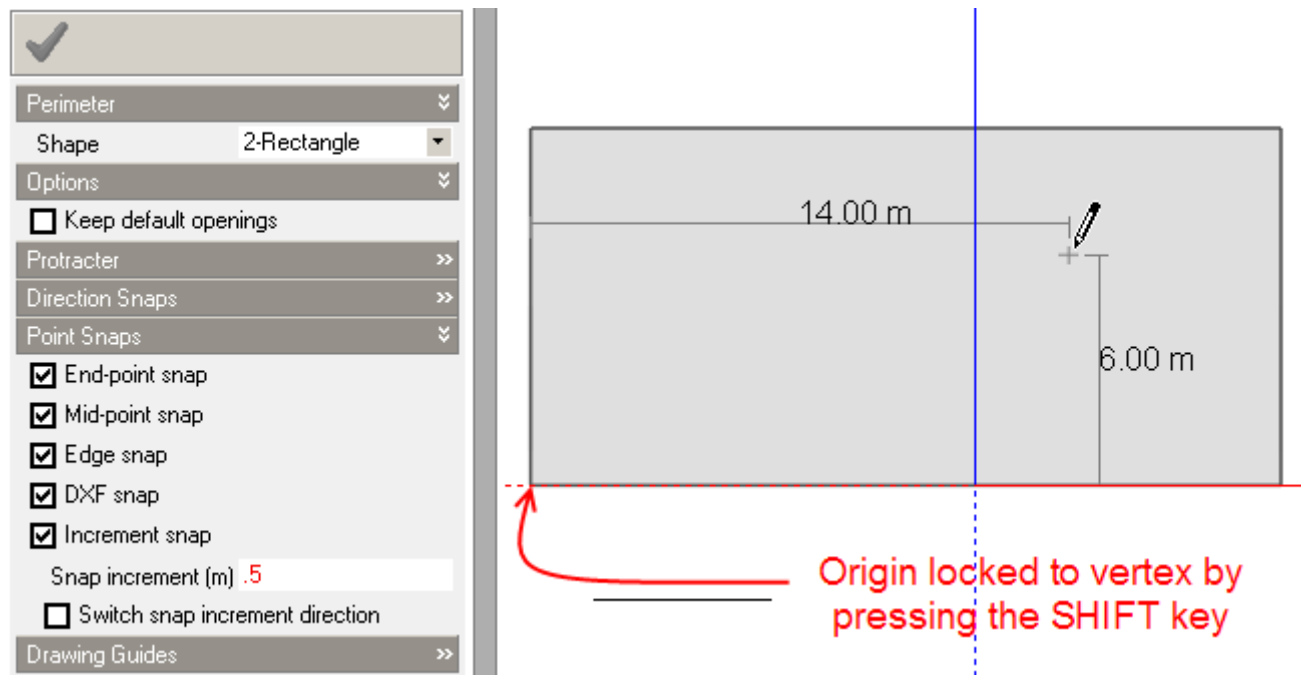
To add a window to a surface, click on the **Draw window** tool. If you have opted to keep the default openings, they will be converted to custom openings:



On the other hand, if you uncheck the **Keep default openings** option, all default openings will be removed. Select the [perimeter shape](#) that you require and then move the mouse cursor to place the window origin. As you move the mouse cursor over the surface, distance guides will be displayed from each of the surface vertices to help in placing the window origin. If the increment snap is switched on, the origin will snap in increments from the vertex that's nearest to the mouse cursor:



Tip: You can lock the window origin to a particular vertex (surface corner) by moving the mouse cursor close to the vertex and pressing the <Shift> key and keeping it pressed until you actually place the window origin. This can be helpful if you want to place a window relative to a vertex at the other end of the surface:

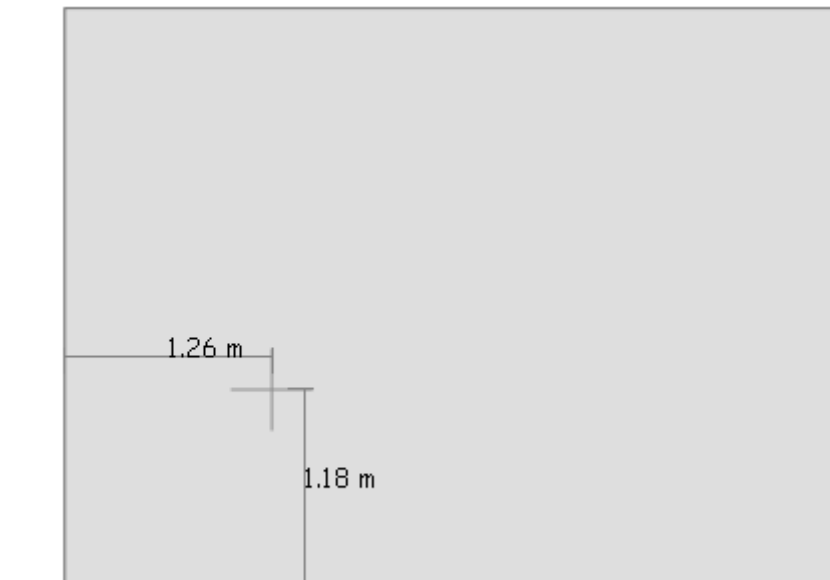


Having placed the window origin, you can then draw the window using the selected perimeter drawing mode, for details of drawing perimeters please refer to the [Perimeters](#) section.

Using Key Ins To Create Openings

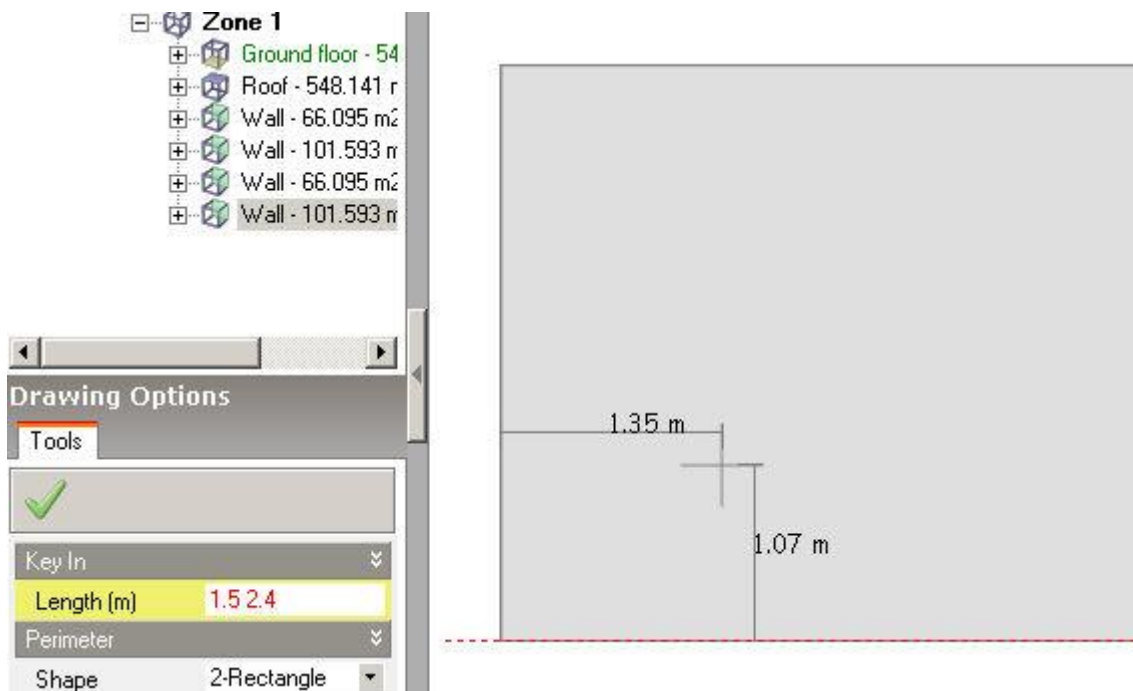
You can type in the precise dimensions and offset from the origin of your window using 'Key Ins'. The procedure is:

1. Start the **Draw window** command as described above.
2. Define the window position relative to an origin by first moving the mouse cursor to define the origin (typically the bottom left corner of the surface)

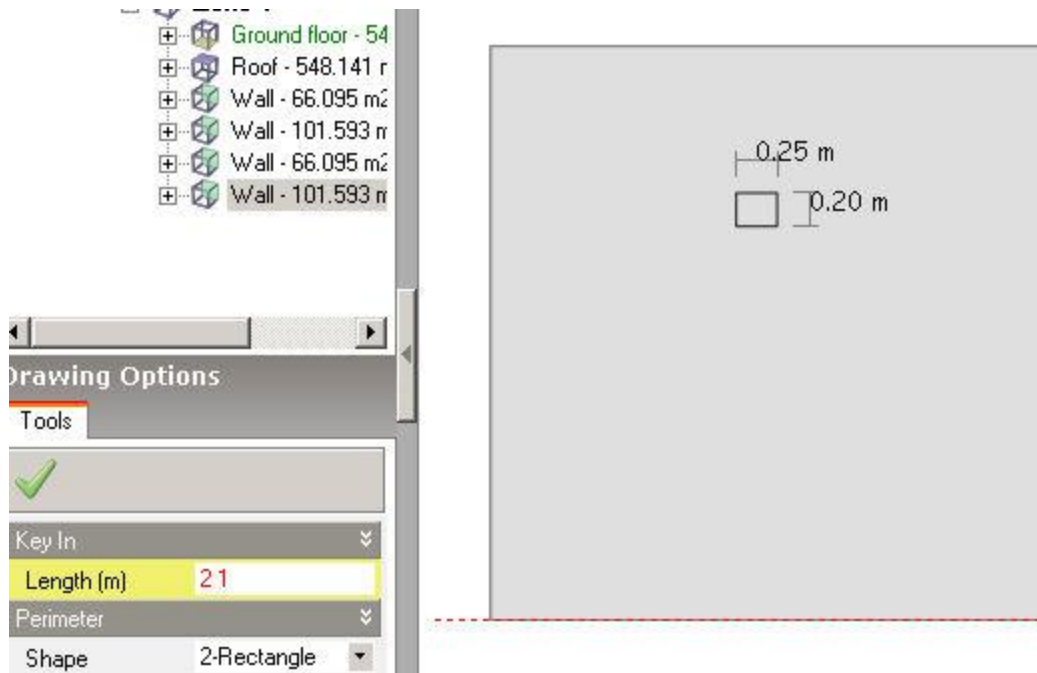


3. Then type in the offset of the window origin (bottom left corner of the window in this example) in the format x y. Note that there is no comma between the x and the y coordinates.

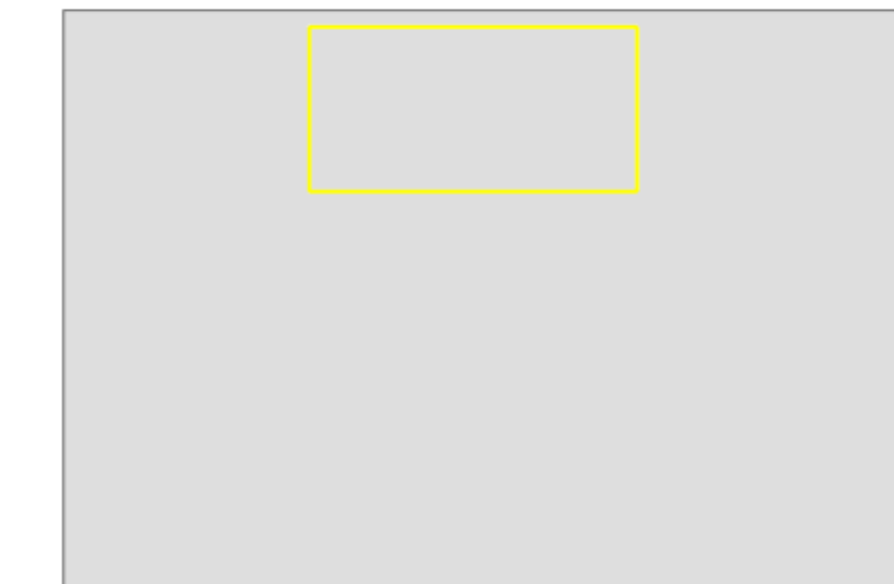
The coordinates that you type in will appear under the **Key In** header in the Drawing Options panel. The convention used when positioning and drawing openings at surface level is that x increases from left to right and y increases from bottom to top. This is true whether viewing the surface from inside or from outside.



4. Then press <Enter> to define the window origin.
5. Next type in the window width followed by a space followed by the window height. Again the coordinates you type in are shown in the Drawing Options panel. In the example below a window is being defined having width 2m and height 1m.



6. Press enter to create the window:



You can use the mouse in conjunction with Key Ins if you prefer. So you could use the mouse to define the window offset from the origin and then use Key Ins to define the window dimensions or vice versa.

Tip: You can also enter negative x and y values when keying in offset and coordinates.

Draw Sub-Surface



For details of using the **Draw sub-surface** tool, please refer to the [Draw window](#) tool.

See the [Sub-Surface Construction](#) topic for information on the construction data used for sub-surfaces and other details.

Draw Door



For details of using the **Draw door** tool, please refer to the [Draw window](#) tool.

Note: In EnergyPlus doors must be rectangular

See also the [Drawing windows and doors](#) tutorial

Draw Hole



For details of using the **Draw hole** tool, please refer to the [Draw window](#) tool.

Holes are usually used for two purposes:

1. For merging zones together. Note that the [Merge zones connected by holes](#) option must be selected for this to work.
2. If adjacent zones aren't merged then holes between them allow air and radiation to pass. See below for details on this.

Holes can also be added in exterior surfaces to allow air and radiation to pass.

Tip: It is possible to draw large holes in surfaces to give the effect of the surface having being removed. This can be a useful technique for creating atrium-type spaces, connecting gable roof blocks to the main roof block and generally merging zones in different blocks.

More information on Drawing holes can be found in the [Drawing holes and merging zones](#) tutorial

The way holes are modelled in EnergyPlus depends on the model options selected:

Airflow

The way that airflow through holes (and virtual partitions) is modelled depends on the [Natural ventilation](#) model option selection. The rules are as follows.

Scheduled Natural Ventilation and Heating and Cooling design calculations	Simulations with Calculated Natural Ventilation
When the Advanced Natural Ventilation Model airflow through openings and Airflow through internal openings model options are checked, holes positioned on internal partition walls are used to mix the air between the two adjacent zones. In this case the airflow through internal holes is calculated based on the Airflow rate per opening area data. External holes have no effect on airflow. This is because scheduled natural ventilation has no way to calculate the airflow through an external hole which would in reality require knowledge of wind speed and direction and other pressure-related data.	Holes are represented using an unobstructed air flow path in the fabric and air flow through them is unaffected by whether natural ventilation is enabled on the HVAC tab and the natural ventilation setpoints and other controls. The airflow through holes is calculated (using the EnergyPlus Airflow Network) based on the pressure difference across the opening and a coefficient of discharge of 0.65.

Radiation and conduction heat transfer

The way that radiation and conduction of heat through holes (and virtual partitions) is modelled depends on the [solar distribution](#) setting, the [solution algorithm](#) and the calculation type as described below.

All exterior holes and when Solar distribution algorithm is Full interior and exterior	Internal holes in simulations when using CTF solution algorithm	Internal holes in heating design and cooling design or when using Finite difference solution algorithm
Modelled as "windows" with "perfectly clear" glazing and so they transmit solar radiation and conduct heat. Note that in this case the conductance U-value for this special type of glazing is 3.12 W/m ² K.	Modelled as Infrared Transparent surfaces in EnergyPlus. The hole will participate in the transfer of visible and solar radiation by doing a "wavelength transformation", making all short wave length radiation that is incident on the surface into long wave length radiation and having it participate in the long wavelength radiant exchange.	Modelled as an opaque sub-surfaces with a standard "Holes and vents" construction, so no radiation transfer takes place between the 2 zones, only conduction plus any airflow.

Draw Vent



For details of using the **Draw vent** tool, please refer to the [Draw window](#) tool.

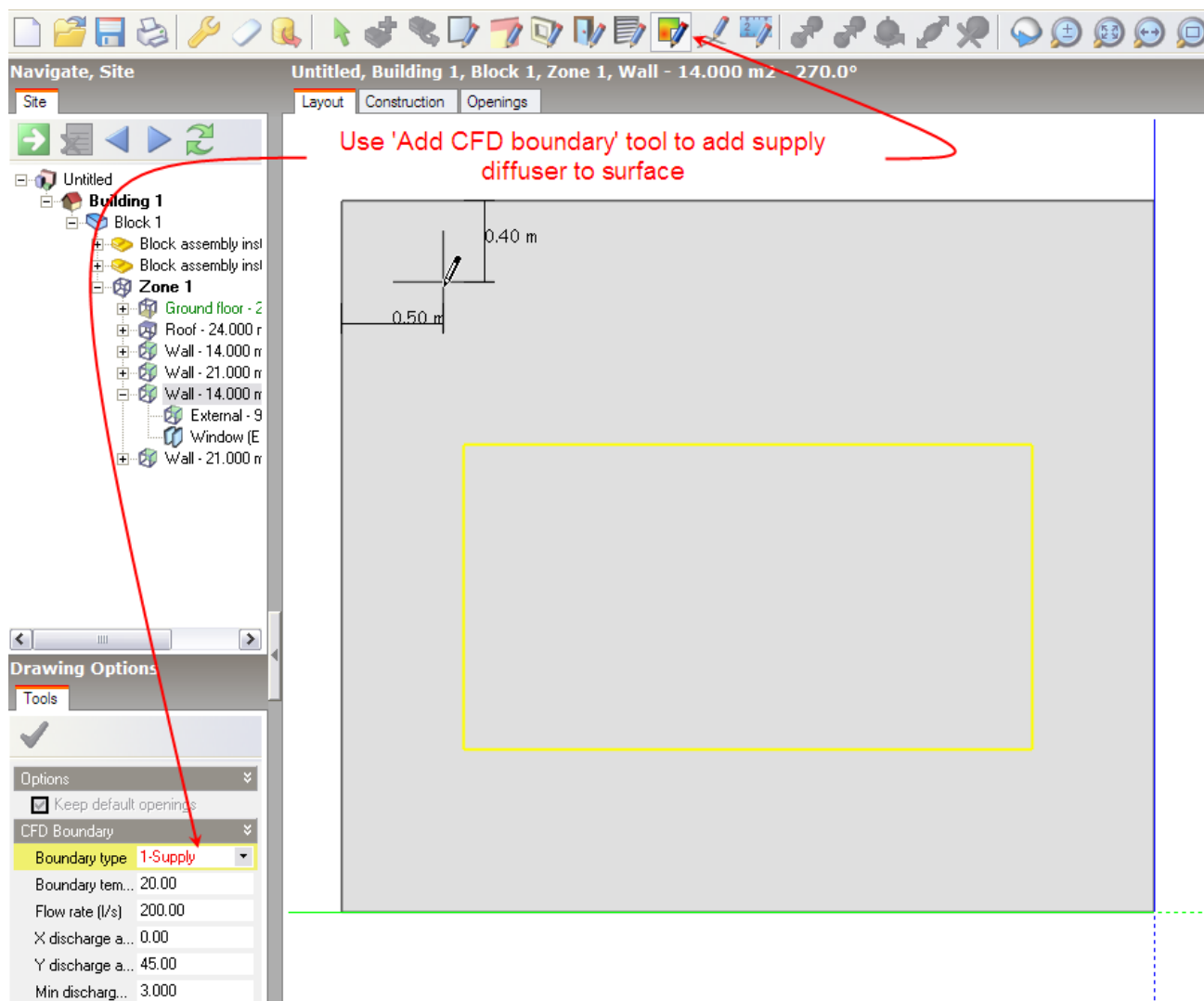
Add CFD Boundary



When at surface level you can add a CFD boundary in much the same way as a window or any other opening. Please refer to the instructions for the [Draw window](#) tool for details on the process of drawing the CFD boundary.

Note: An important difference with placing CFD supply and extract boundaries compared with other openings such as windows, is that their edges must not coincide with parent surface edges. The CFD boundary should be separated from the surface edges by a distance of at least 0.1 m.

You should also ensure that there are at least three grid cells between a supply boundary and an obstruction (such as a wall, component or assembly), otherwise, during simulation, the flow can 'lock' and the mass residual will not converge.



After selecting the **Add CFD boundary** tool, the boundary settings are displayed on the Drawing options data panel. You can select the required boundary type from the **Boundary type** drop list. The following boundary types are available for surfaces of all orientations:

- **1-Supply** (general purpose HVAC supply diffuser)
- **4-Extract** (HVAC extract grille)
- **5-Temperature** (surface temperature patch). the heat transfer is calculated in one of two ways depending on the [Surface heat transfer](#) option on the CFD Calculation options dialog selected. If option **2-User defined** is selected, the heat transfer is calculated by combining the specified surface transfer coefficient with the cell surface area and the delta T between the surface and the adjacent cell. If the [Surface heat transfer](#) option is set to **1-Calculated**, the surface heat transfer coefficient is calculated using a wall function.

- **6–Flux** (surface heat flux patch) - the calculation scheme incorporates this flux directly within the cells adjacent to the surface at which the flux is specified. The heat flux is incorporated after multiplying it by the surface area of the cell.

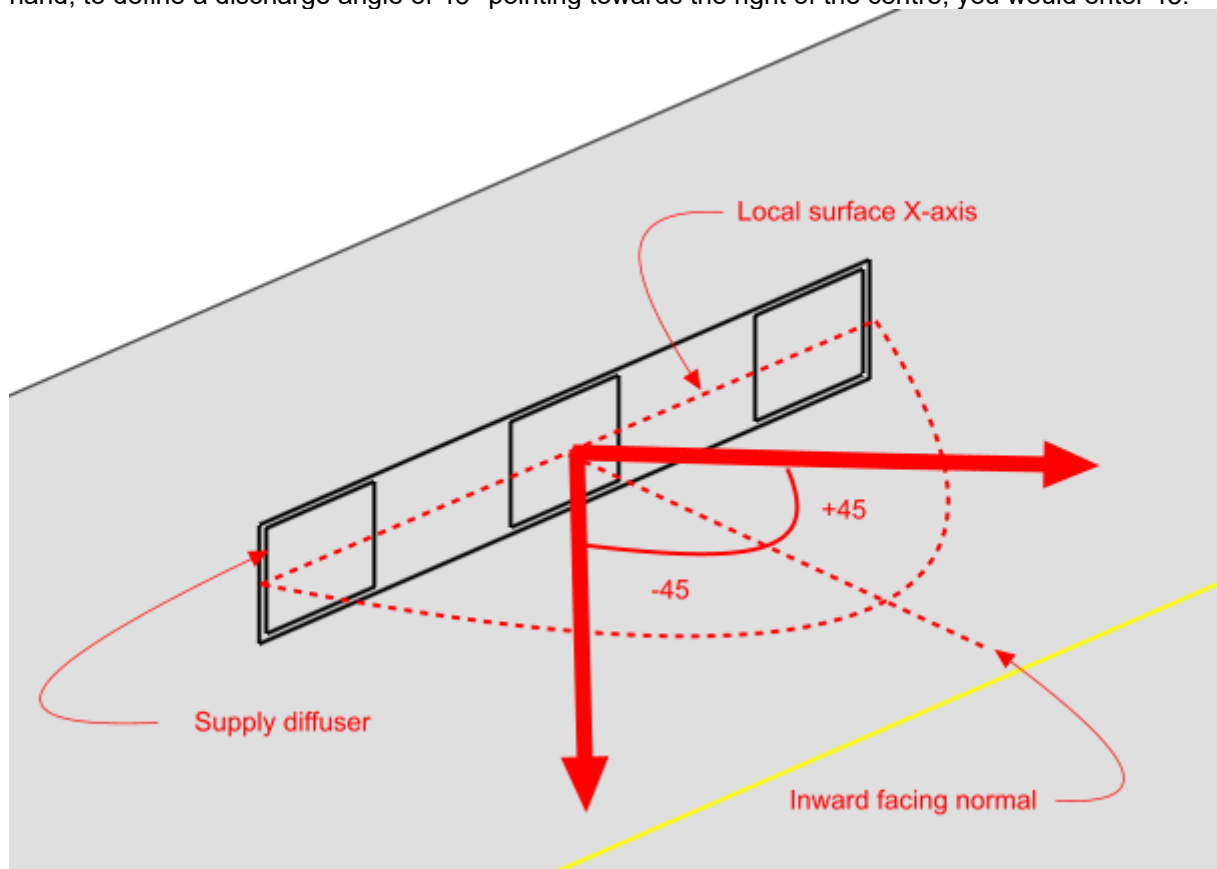
The following additional boundary types for multi-directional diffusers are available for ceiling and other downward-facing surfaces only:

- **2–Four-way** (four-way supply diffuser)
- **3–Two-way** (two-way supply diffuser)

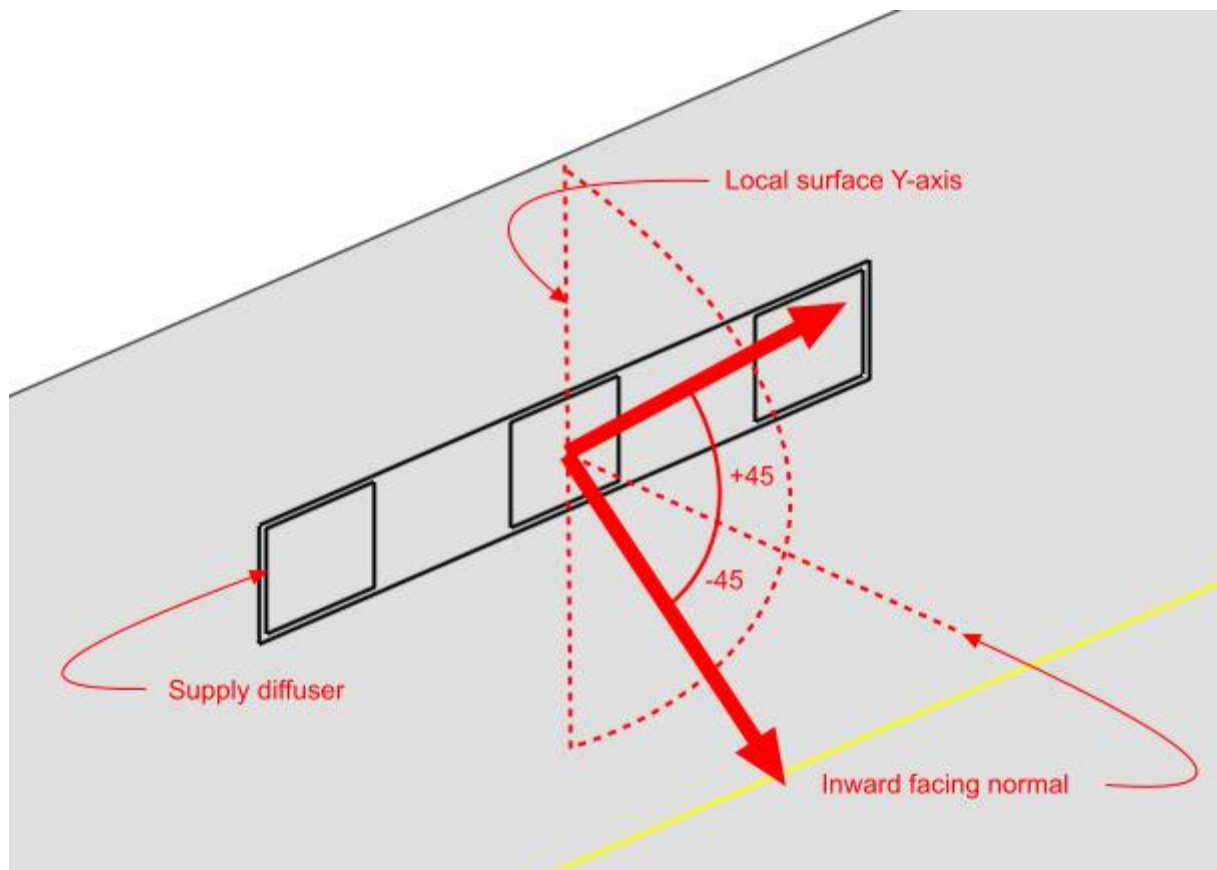
Various settings are available depending on the boundary type:

1-Supply:

- **Boundary temperature** - the temperature of the air entering the space.
- **Flow rate** - the supply volume flow rate.
- **X-discharge angle (°)** - the discharge angle between the local surface X-axis and an inward facing normal to the surface. At the surface level, taking a normal view from the inside of the zone to the surface, the X-axis discharge angle is positive between the normal and the positive X-axis (i.e. the axis pointing towards the right) and negative between the normal and the negative X-axis (i.e. the axis pointing towards the left). For example to define a discharge angle of 45° pointing towards the left of the centre of a diffuser, looking at it from the inside of the zone, you would enter –45. On the other hand, to define a discharge angle of 45° pointing towards the right of the centre, you would enter 45:



- **Y-discharge angle (°)** - the discharge angle between the local surface Y-axis and an inward facing normal to the surface. At the surface level, taking a normal view from the inside of the zone to the surface, the Y-axis discharge angle is positive between the normal and the positive Y-axis (i.e. the axis pointing upwards) and negative between the normal and the negative Y-axis (i.e. the axis pointing downwards). For example to define a discharge angle of 45° pointing downwards from the centre of a diffuser, looking at it from the inside of the zone, you would enter –45. On the other hand, to define a discharge angle of 45°, pointing upwards from the centre, you would enter 45:



- **Min discharge velocity** is the minimum required velocity at the face of the diffuser. The minimum discharge velocity is an important setting in that the diffuser supply velocity in conjunction with the diffuser discharge angle(s) can determine the flow pattern in a space.

The following paragraph only applies to orthogonal diffusers only. If the minimum discharge velocity cannot be achieved by combining the specified flow rate with the face area of the diffuser as drawn, supply diffusers are created using a number of elements, the areas of which are determined by combining the specified volume flow rate together with the minimum discharge velocity. The maximum linear dimension of each component element is currently 'hard-set' at 0.2m. If the minimum discharge velocity is set to a very small value (e.g. 0.01m/s), a single element will be created spanning the entire area of the diffuser as drawn and the actual discharge velocity will be determined from the geometry of the diffuser face and the supply volume (i.e. $v=V/A$). The actual discharge velocity is displayed for diffusers on the CFD data tab at the opening level.

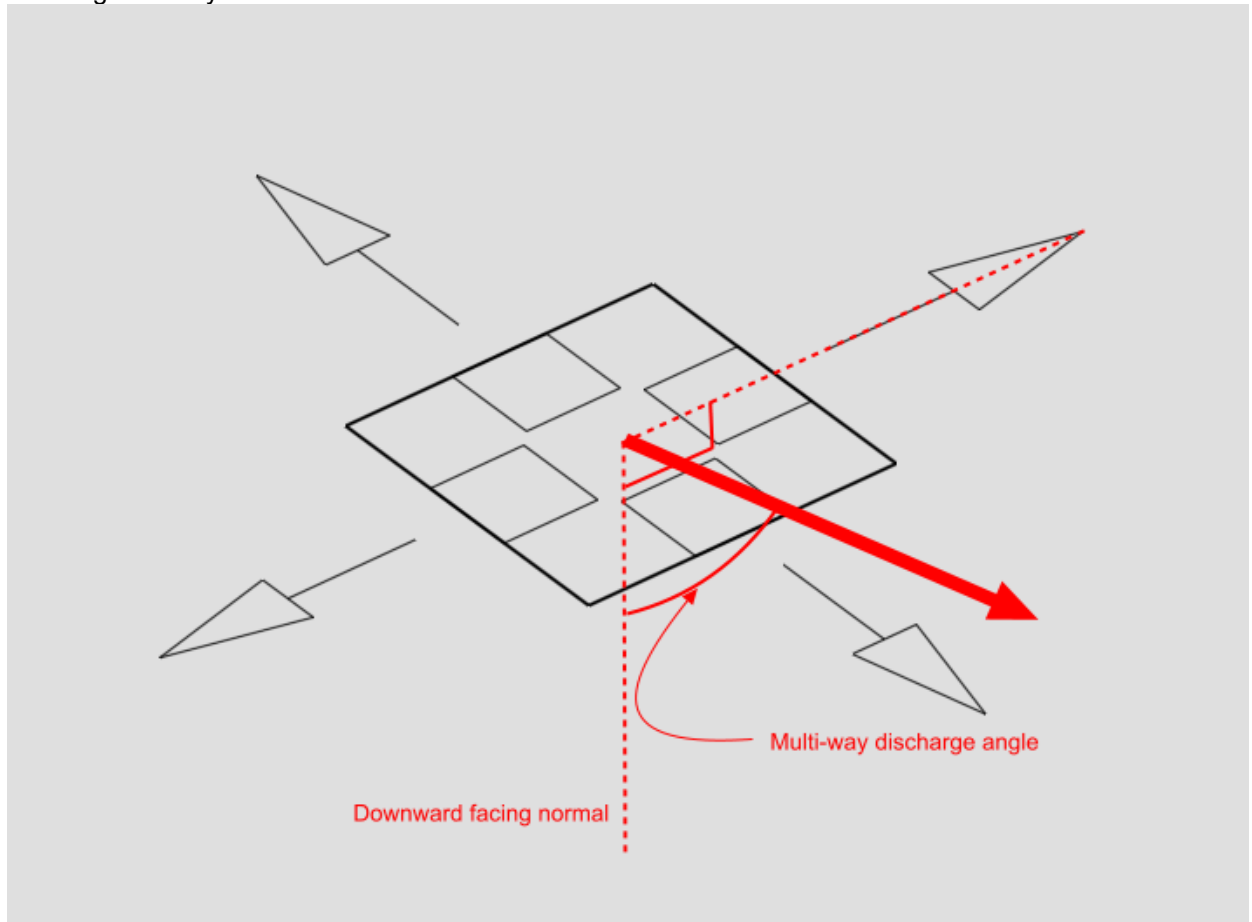
Note: In the case of **non-orthogonal** supply diffusers, the minimum discharge velocity can only be guaranteed if the diffuser has no tilt in the z-dimension (i.e. it has a non-orthogonal azimuth but zero tilt).

Note: Supply diffuser discharge angles only apply to orthogonal diffusers, i.e. to diffusers aligned with the grid. For diffusers placed on non-orthogonal diffusers the discharge angles simulated will be incorrect.

2-Four-way

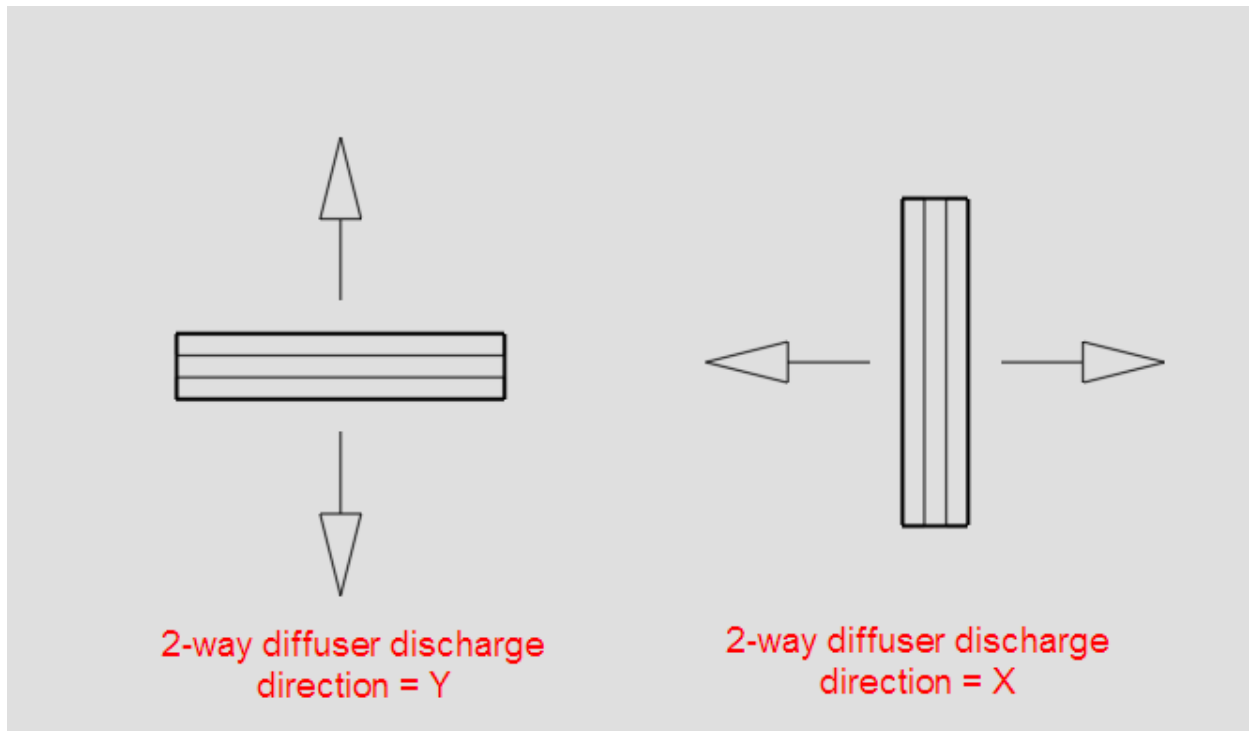
- **Boundary temperature** - temperature of the air entering the space.
- **Flow rate** - the supply flow rate.
- **Multi-way diffuser discharge angle (°)** - the discharge angle is the angle between the downward pointing normal and the diffuser jet.
- **Minimum discharge velocity** - the minimum required velocity at the face of the diffuser. Four-way diffusers are created using four separate elements, one located at each edge of the diffuser, the area

of each being determined by combining the specified volume flow rate together with the minimum discharge velocity.



3–Two-way

- **Boundary temperature** - the temperature of the air entering the space.
- **Flow rate** - enter the supply flow rate.
- **Multi-way diffuser discharge angle (°)** - the angle between the downward pointing normal and the diffuser jet (see illustration in 'Four-way' section).
- **2-way diffuser discharge direction** - the local surface axis along which the diffuser is to discharge.
- **Minimum discharge velocity** - the minimum required velocity at the face of the diffuser. Two-way diffusers are created using two separate elements, one on either side of the diffuser along the discharge axis, the area of each being determined by combining the specified volume flow rate together with the minimum discharge velocity.



4–Extract

- **Flow rate** - the extract flow rate.

5–Temperature

- **Boundary temperature** - temperature for the patch.

6– Flux

- **Heat flux** - heat flux for the patch.

Note: You can add all types of CFD boundary conditions to zone surfaces, but only supply and extract CFD boundaries can be added to component block surfaces.

Edit Surfaces

At the surface level, you can select custom openings and [move](#), [clone](#) (copy), [delete](#) and [rotate](#) them.

Note: Default openings cannot be selected and manipulated in this way.

You can also clear custom openings their default state using the [Clear Data to Default](#) command.

Undo/Redo

Undo/Redo commands accessed from [Edit](#) menu

To undo a geometric operation, select Undo from the 'Edit' menu (or press <Control-Z>) and similarly to redo a geometric operation, select Redo (or press <Control-R>). Because many of the geometric operations are dependent on specific model levels, you may need to undo or redo navigation operations before you get to the required geometric operation, e.g. 'Undo - Go to surface', etc.

Note that Undo/Redo applies only to geometric operations and *changes to Model Data cannot be undone*.

Rebuild

Rebuild command accessed from [Tools](#) menu

Sometimes the adjacencies within DesignBuilder models can become corrupt. The **Rebuild** command recalculates the zone, surface and adjacency geometry from the base building elements.

See also the [Setting adjacency conditions using ground and adiabatic blocks](#) tutorial

Rezone Block

Rezone block command accessed from [Tools](#) menu

Use the Rezone block command to refresh the zoning in the current block. The command converts any [outline](#) partitions in a block to standard partitions and updates zones due to any partitions that may have deleted. The command should be used when the Automatic block zoning model option is switched off. When the option is switched on (and it has always been switched on during the life of a model) then you will not need to use this command.

Model Data

Model Data is the data for the [site](#) on the tabs:

- [Location](#)
- [Legislative region and insulation standards](#)

and data for [building model](#) on the tabs:

- [Activity \(usage\)](#)
- [Constructions](#)
- [Openings](#) (windows, doors, vents, holes, sub-surfaces)
- [Lighting](#)
- [HVAC](#)
- "On Site Electricity Generation" on page 521 (on site electricity generation)
- "Economics" on page 546
- "Miscellaneous" on page 636
- "CFD" on page 640

The Model Data tabs are shown in the [Edit Screen - User Interface](#) help topic.

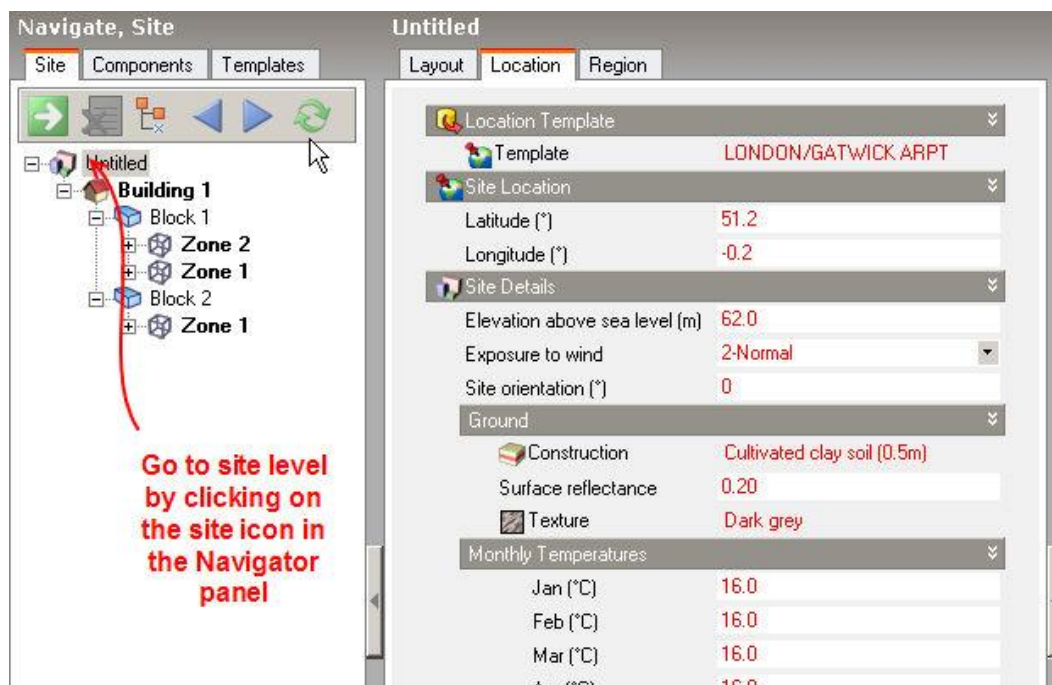
At the point of the building is created, building model default data is initially loaded at the building level from templates. This is inherited from the building to blocks to zones to surfaces and to openings. This default model data can be overwritten by making edits on the Model Data tabs on the [Edit screen](#).

See the [DesignBuilder User Interface – Model Data: Site level, model data tabs, data hierarchy and inheritance](#) tutorial

Site Data

The **site data** represents information about the site including its location, the conditions of the ground, weather, the legislative region and the maximum allowed U-values for new buildings. All buildings on the site use the same site model data.

To edit site model data, go to site level and click on the Location or Region tabs (below).



You can easily change the location of the site by making a selection for the [Location template](#) on the [Location tab](#) or by using the [Load data from template](#) command.

Alternatively you can open up the headers on the [Location](#) and [Region](#) tabs to edit the data.

See also the [Site level data and weather files](#) tutorial

Location

Location tab on model data at [Site level](#)

Use this screen to edit the location of the site, its weather data, the daylight savings and energy codes which apply.

You can load a location from the database by clicking on Template line under the Location Template header then on the '...' at the right of the line.



Select location template

When you make a selection from the list of [location templates](#) data from selected template will be loaded to the model.

Alternatively you can use the [Load data from template](#) tool.

You can enter data on:

- [Site location](#),
- [Site details](#),
- [Time and daylight saving](#),
- [Simulation weather data](#),
- [Winter design weather](#) and
- [Summer design weather](#).

See also the [Site level data and weather files](#) tutorial

Site Location

[Location tab](#) on model data at [Site level](#)

You can set the latitude/longitude of the site for cooling design calculations as well as a number of site characteristics.

Latitude

The latitude (in degrees) of the site. By convention, North Latitude is represented as positive; South Latitude as negative. Minutes should be represented in decimal fractions of 60. (15' is 15/60 or .25)

Longitude

The longitude (in degrees) of the site. By convention, East Longitude is represented as positive; West Longitude as negative. Minutes should be represented in decimal fractions of 60. (15' is 15/60 or .25).

Note: the latitude, longitude and time zone location data will only affect cooling design and Radiance illuminance calculations. For EnergyPlus and annual Daylighting simulations, the position of the sun is defined by the data in the [hourly weather file](#) header.

ASHRAE Climate Zone

The climate zone as defined by ASHRAE and loaded from the Location template can be overridden by selecting from this dropdown list This allows you to deal with situations where the default climate zone does not agree with the value in ASHRAE 90.1 Standard Appendix D. This is an important setting for ASHRAE 90.1 models as it defines the baseline constructions, glazing etc. to be used for the building.

See also the [Site level data and weather files](#) tutorial

Site Details

- "Site Details - General" on page 288
- "Site Details - Site Height Variation" on page 290
- "Site Details - Ground" on page 291
- "Horizon" on page 311
- "Site Details - Mains Water Temperature" on page 311
- "Site Details - Precipitation + Irrigation" on page 312
- "Site Details - Outdoor Air CO2 and Contaminants" on page 314

See also the [Site level data and weather files](#) tutorial

Site Details - General

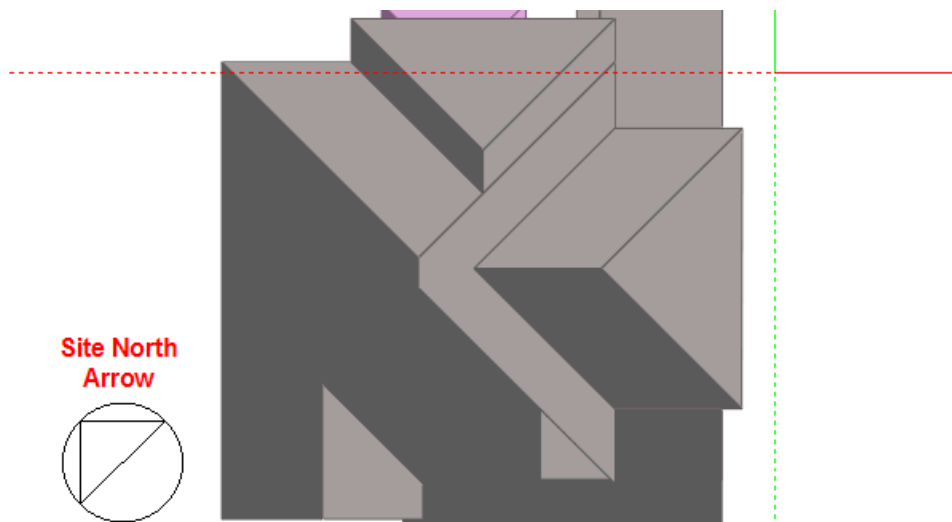
[Location tab](#) on model data at [Site level](#)

Elevation above sea level

This data defines the elevation of the site relative to sea level (in m or ft) and is passed directly to EnergyPlus for use in pressure-related calculations. It is also used to calculate [air density](#) within DesignBuilder when calculating volumetric flow rates from cooling loads.

Site orientation

The site orientation is the angle of the site plan view relative to North. For example enter 45 to rotate the building(s) clockwise by 45° relative to North (screenshot example below). This is the quickest and most reliable way to rotate all the buildings on the site relative to North and allows you to keep your x and y axes aligned with the building.



North is indicated by the direction of the North arrow in the sketch plan view (above).

Warning: While it is also possible to rotate building geometry using the [Rotate selected objects](#) command, once this has been done it is more difficult to make further geometric edits because, for rotations other than 90, 180 or 270°, the x and y axes will be out of alignment with the building. Also, whole-model rotations increase the risk of geometric issues in the model. You are therefore strongly advised to use the **Site orientation** setting instead.

Tip: You can navigate to building level and use the [Measure](#) tool to measure the angle between 2 lines on a floor plan to calculate the Site orientation.

Implementation Details

The **Site orientation** defined here is accounted for in the EnergyPlus IDF input data by rotating the geometry of all site and building objects. Because DesignBuilder handles all rotation itself, it is possible for it to accurately apply additional *building* rotations in combination with the *site* rotation entered here. Building rotations are applied to all building objects (not site objects) about the geometric centre of the building in the x-y plane.

Building rotation applies to a) LEED baseline auto-rotation and b) parametric analysis, optimisation, uncertainty and sensitivity analysis when the [Building rotation](#) variable is selected.

Exposure to wind

The site's terrain affects how the wind hits the building – as does the building height. In addition, the external conduction method usually has its own parameters for the calculation. The options available are:

- **1-Sheltered** - maps to the EnergyPlus "City " setting for towns, city outskirts and the centre of large cities.
- **2-Normal** - maps to the EnergyPlus "Suburbs" setting for "rough, wooded country, suburbs".
- **3-Exposed** - maps to the EnergyPlus "Country" setting for "flat, open country"

When the [Calculated](#) Natural ventilation option is set, the exposure to wind model data affects the [pressure coefficients](#) used . It also affects the calculation of U-values (though these are not used in EnergyPlus simulations).

When the [Scheduled](#) Natural ventilation option is set, the exposure to wind model data affects infiltration when using one of the options normalised to exposed façade area, i.e. **2-m3/h-m2 at 50 Pa** and **3-m3/h-m2 at 4 Pa**. It acts through the **wind exposure class** as explained in the Technical notes under [Infiltration rate](#).

Important note: When the wind exposure setting is changed, the outside convective heat transfer coefficient, outside surface resistance and U-value for each user-defined construction in the [Constructions](#) database is recalculated because the wind exposure affects these values. This is necessary to ensure that the info panel data remains up to date and for local calculations such as SBEM which rely on U-values being up to date.

Site Details - Site Height Variation

[Location tab](#) on model data at [Site level](#)

The Site height variation data is used to specify the local variation in atmospheric properties at the site and should be used only if you require advanced control over the height-dependent variations for wind speed and temperature. The coefficients set here are used by EnergyPlus to calculate the local variation in atmospheric properties as a function of height above ground. Outdoor air temperature decreases with height, while wind speed increases with height. The local outdoor air temperature and wind speed are calculated separately for all zones and surfaces, and optionally for outdoor air nodes for which a height has been specified. With the default inputs, wind speed falls significantly at heights lower than the weather station measurement height, and temperature increases slightly. The algorithms for this calculation are in the Engineering Reference. There are necessary correlations between the entries for this object and the [Exposure to wind](#) setting.

Atmospheric Variables at Two Different Heights Above Ground Level.

Variable	1.5 m	284 m	Absolute Diff	Percent Diff
Air Temperature	15°C	13.15°C	1.85°C	12.3%
Barometric Pressure	101,325 Pa	97,960 Pa	3,365 Pa	3.3%
Wind Speed	2.46 m/s	7.75 m/s	5.29 m/s	215%

Note that using this object overrides the wind speed profile coefficients implied by the Terrain field of the Building object. The wind speed profile coefficients can be estimated from the table above (see Site:WeatherStation) or calculated beforehand using more sophisticated techniques such as CFD modeling of the site terrain.

Use site height variation

Check this checkbox to apply custom variation in atmospheric properties at the site.

Wind speed profile exponent

The wind speed profile exponent for the terrain surrounding the site. The exponent can be estimated from the table above or calculated beforehand using more sophisticated techniques, such as CFD modeling of the site terrain. Note that using this object overrides the wind speed profile coefficients implied by the Terrain field of the Building object.

Wind speed profile boundary layer thickness

The wind speed profile boundary layer thickness (in m or ft) for the terrain surrounding the site. The boundary layer can be estimated from the table above or calculated beforehand using more sophisticated techniques, such as CFD modeling of the site terrain. Note that using this object overrides the wind speed profile coefficients implied by the Terrain field of the Building object even if this field is left blank. This field can be set to zero to turn off all wind dependence on height.

Air temperature gradient coefficient

The air temperature gradient coefficient (in K/m or °F/ft) is a research option that allows the variation in outdoor air temperature to be defined as a function of height above ground. The real physical value is 0.0065 K/m. This field can be set to zero to turn off all temperature dependence on height.

Site Details - Ground

[Location tab](#) on model data at [Site level](#)

Ground Modelling

Ground modelling method

DesignBuilder provides 4 different ways to model the heat transfer between the ground, the external environment and the building. Select the method to be used from the options below.

- **1-Standard** the default method where a set of 12 fixed [Ground monthly temperatures](#) are used to define the temperature on the outside boundary of building ground adjacent surfaces. Conceptually this is the simplest method since it is based on 1-D ground heat transfer, however it requires prior knowledge of the ground temperature just below the building for accurate calculations of the heat transfer between the ground and the building. Because these temperatures are rarely known in practice, this method can give rise to significant errors and it is not recommended in cases where accurate ground modelling is required.
- **2-Kiva Basic** where a single [Kiva foundation](#) object is used to represent heat transfer to, from and within the ground. This option is ideal for accurate simulations of smaller buildings such as dwellings where a single Kiva foundation object can realistically be used to represent the ground and the connection with the building. It requires only a small amount of extra data entry compared with the **1-Standard** option to define the ground properties etc, but in return it provides a significant improvement in ground modelling accuracy. See below for more details.
- **3-Ground domain** a more detailed method than the previous 2 options where EnergyPlus [ground domain](#) objects are used to represent the ground, basement and foundation configuration and adjacency to the building. See below for more details.
- **4-Kiva Full** the most accurate and flexible ground modelling option where EnergyPlus' Kiva system is used to perform two-dimensional finite difference heat transfer calculations to, from and within the ground. See below for more details on this option.

Note: For heating and cooling design calculations, the **1-Standard** method is always used regardless of the Ground modelling method selected.

Standard Method

This is the default method. You can find detailed information on how to use the **1-Standard** ground modelling method in the "Ground Modelling - Standard Method" on page 296 section.

Kiva Basic Method

Kiva enables detailed modelling of the ground while requiring a minimal amount of data to be entered. The **Kiva Basic** option provides an easy introduction to Kiva ground simulations and is ideal for modelling smaller buildings where a single [Kiva foundation](#) object can realistically be used to represent the ground, foundations, insulation and its adjacencies to the building. Note that the Ground monthly temperatures, Ground deep temperatures, Shallow and Deep ground temperatures described below are not used in Kiva simulations.

When the **2-Kiva Basic** Ground modelling method option is selected, 2 more items are revealed below: **Kiva foundation settings** and **Kiva foundation**. These are the only additional settings that needs to be entered to define the ground modelling, adjacencies etc.

DesignBuilder automatically calculates the fraction of floor perimeters that are exposed to outside based on the model surface adjacencies.

Note: This option is exactly equivalent to the **4-Kiva Full** option when the [Kiva adjacency](#) checkbox is checked at building level and the [Kiva foundation](#) selected at building level with no further overriding settings made at block, zone and surface levels.

Kiva foundation settings

Select a [Kiva foundation settings](#) component that defines the soil thermal and surface properties, the extent of the domain and some calculation options.

Kiva foundation

Select a single [Kiva foundation](#) object to define the extent and properties of any insulation and details of the walls and footings.

Note: These settings are automatically applied to *all* ground adjacent surfaces in the model when using the **2-Kiva Basic** ground modelling method.

Ground Domain Method

The EnergyPlus Ground domain system provides a relatively quick and easy way to access the advanced ground modelling capabilities of the Slab and Basement preprocessor tools which in older versions of EnergyPlus had to be run as external utilities with a manual process required to feed the outputs from the preprocessors as inputs to EnergyPlus. See the [EnergyPlus Auxiliary Programs](#) help for more information on these legacy utilities.

The way that the temperature boundary conditions for the simulations are handled is defined on the [Ground Undisturbed Temperatures](#) tab of the Ground domain dialog. Calculations are based on the hourly weather data and soil properties. Note that the Ground monthly temperatures, Ground deep temperatures, Shallow and Deep ground temperatures described below are not used in Ground domain simulations.

Select the **3-Ground domain** option to activate the ground domain model. This reveals an option to select the number of ground domain objects to use in the study. The number entered here will enable the display of that number of ground domain objects as shown below.



Ground Modelling	
Ground modelling mechanism	3-Ground domain
Number of ground domains	1
Ground domain 1	Default Slab

You will normally need to create new [Ground domain objects](#) to define the exact situation for your model.

Note: Ground domains are not used in any heating and cooling design calculations. In this case surfaces adjacent to ground and [assigned with ground domain objects](#) use the [Ground monthly temperatures](#) as the other side.

Note: You must connect the ground domains defined and selected here to ground adjacent surfaces to complete the domain definition. Wall surfaces that are underground can be selected as basements and any floor that has any ground contact can be defined as having either slab or basement contact.

Tip: Domain-surface connection settings can be made at any of building, block, zone and surface levels. Setting data at building, block or zone level provides a quick way to associate multiple ground adjacent surfaces to a ground domain.

Kiva Full Method

Kiva enables detailed modelling of the ground while requiring a minimal amount of data to be entered. The **Kiva Full** option provides access to the full EnergyPlus Kiva ground capability which allows accurate simulations of the ground, foundations, insulation and its adjacencies to the building. It is an improvement on the ground domain method in terms of accuracy, flexibility and execution speed. Note that the Ground monthly temperatures, Ground deep temperatures, Shallow and Deep temperatures described below are not used in Kiva simulations.

DesignBuilder automatically calculates the fraction of floor perimeters that are exposed to outside based on the model surface adjacencies.

When the **4-Kiva Full** Ground modelling method option is selected, the [Kiva foundation settings](#) must also be selected here at site level. Also each ground-adjacent surface has an option to allow it to be selected as having an adjacency with a [Kiva foundation](#). You should normally check the option and select the appropriate Kiva foundation.

Kiva foundation settings

Select the [Kiva foundation settings](#) component that defines the soil thermal and surface properties, the extent of the domain and some calculation options.

Surface

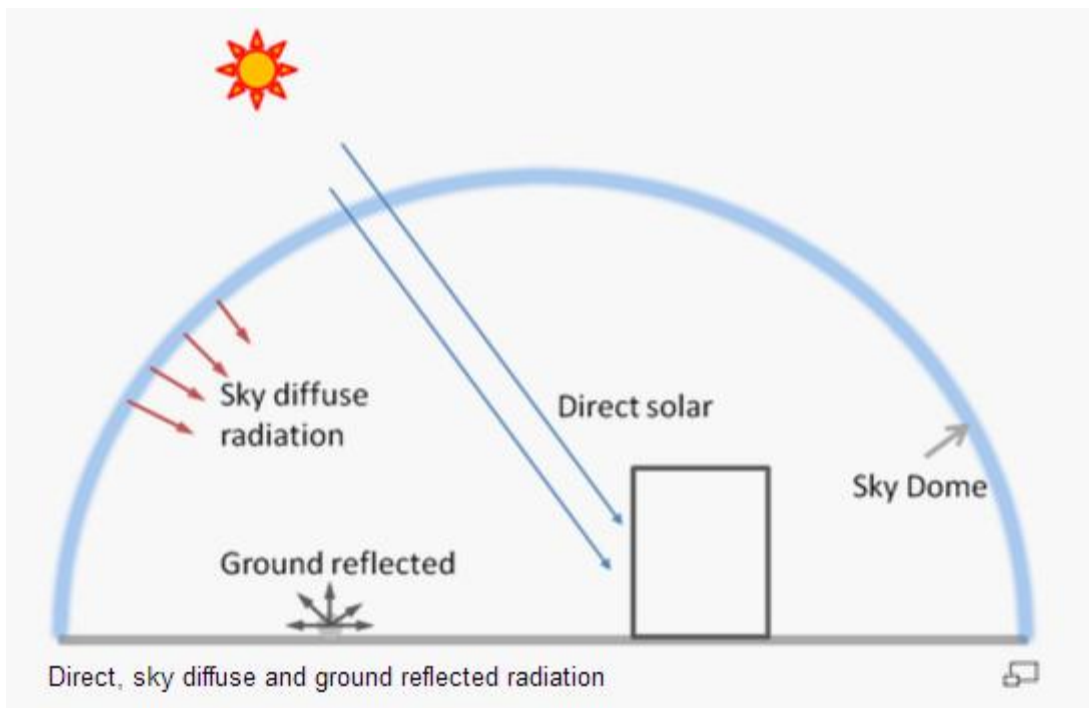
Ground texture

Select the texture to be used to represent the ground in the rendered view on the [Visualisation](#) screen.

Surface solar and visible reflectance

This is a decimal number between 0.0 and 1.0 and is used to characterize the average reflectivity of the ground throughout the year. Ground reflectance data is used to calculate the ground reflected solar radiation. This fractional amount is used in the following equation:

$$\text{GroundReflectedSolar} = (\text{BeamSolar} \times \text{COS}(\text{SunZenithAngle}) + \text{DiffuseSolar}) \times \text{GroundReflectance}$$



Reproduced from the [IBPSA BEM Book](#).

Important Note: Reflection from the ground is always calculated even if the [Model all external reflections and shading of ground reflected solar](#) option is not selected, in which case the ground plane is considered unobstructed, i.e., the shadowing of the ground by the building itself or by component blocks is ignored. In other words, shadowing of ground-reflected radiation by obstructions such as the building itself and component blocks is only taken into account if the [Model all external reflections and shading of ground reflected solar](#) option is set.

The ground surface reflectance is also used to quantify the reflection of visible light in the [Daylighting](#) calculations.

Some typical ground reflectance values are shown for various surface types below.

Surface Type	Surface Reflectance
Water (large angle of incidences)	0.07
Coniferous forest (winter)	0.07
Bituminous and gravel roof	0.13
Dry bare ground	0.20
Weathered concrete	0.22
Green grass	0.26
Dry grassland	0.2 to 0.3
Desert sand	0.4
Light building surfaces	0.6

Ground Reflectance of Foreground Surfaces from ASHRAE 2009 HOF, Source: Adapted from Thevenard and Haddad (2006).

The default ground reflectance is 0.2.

Snow reflected solar modifier

A number between 0.0 and 10.0 which is used to modify the basic ground surface reflectance when snow is on the ground. Note that the value of $\text{GroundReflectance}_{\text{Used}}$ (below) must be ≤ 1 .

$$\text{GroundReflectance}_{\text{Used}} = \text{GroundReflectance} \times \text{Modifier}_{\text{Snow}}$$

During simulations, the ground is considered to be snow-covered when the SnowDepth data in the hourly weather file is > 0 .

From the IBPSA BEM Book:

Site exposure	Reflectance of snow-covered ground
Typical city centre	0.2
Typical urban site	0.4
Typical rural site	0.5
Isolated rural site	0.7

The default modifier is 2 for both solar and daylight radiation, giving an overall default reflectance for snow covered ground of $0.2 \times 2 = 0.4$, suitable for a typical urban site.

Snow reflected daylight modifier

A number between 0.0 and 10.0 which is used to modify the basic ground surface reflectance when snow is on the ground. Note that the value of $\text{DaylightingGroundReflectance}_{\text{Used}}$ (below) must be ≤ 1 .

$$\text{DaylightingGroundReflectance}_{\text{Used}} = \text{GroundReflectance} \times \text{Modifier}_{\text{Snow}}$$

During simulations, the ground is considered to be snow-covered when the SnowDepth data in the hourly weather file is > 0 .

Ground Monthly Temperatures

There are 12 fields allowing you to represent the mean ground temperature for each month of the year. These temperatures are used as the outside surface temperature for surfaces [adjacent to ground](#).

Note that the depth at which these temperatures apply depends on the thickness of the ground construction (see above) added to the main floor/wall construction.

Caution: It is generally not appropriate to use the "undisturbed" ground temperatures calculated by the weather converter and provided in the header of EnergyPlus hourly weather files as these values are too extreme for the soil under typical conditioned buildings. A reasonable default value of **2°C less than the average monthly indoor building temperature** is appropriate for large buildings. For smaller buildings, the ground temperatures will be somewhere between that value and undisturbed ground temperatures.

More information about determining appropriate ground temperatures is given in [Ground Modelling](#) and in the EnergyPlus Auxiliary Programs document.

Ground Deep Temperatures

There are 12 fields allowing you to represent the undisturbed deep ground temperature for each month of the year. These temperatures are used when the **1-Deep** [Reference ground temperature](#) option is used in a **15-Follow ground temperature** Setpoint manager.

Tip: Undisturbed deep ground temperatures can usually be obtained from the [epw weather file header](#).

Ground Shallow Temperatures

There are 12 fields allowing you to represent the undisturbed shallow ground temperature for each month of the year. These temperatures are used when the **2-Shallow** [Reference ground temperature](#) option is used in a **15-Follow ground temperature** Setpoint manager.

Tip: Undisturbed shallow ground temperatures can usually be obtained from the [epw weather file header](#).

FCFactor Ground Temperatures

Building energy code and standards like ASHRAE 90.1, 90.2 and California Title 24 require the underground wall constructions and slabs-on-grade or underground floors not to exceed certain maximum values of [C-factor and F-factor](#), which do not specify detailed layer-by-layer materials for the constructions. If using the normal layer by layer definition method for ground constructions, users would need to create a pseudo wall or floor construction to match the thermal performance such as thermal mass effect and U-factor, and rely on the EnergyPlus Basement and Slabs tools to generate the monthly ground temperatures.

A simplified approach has therefore been provided to create equivalent constructions and model the ground heat transfer through underground walls and ground floors for the building energy code compliance calculations. The approach is to create constructions based on the user defined C or F factor with two layers: one concrete layer (0.15 m thick) with thermal mass, and one fictitious insulation layer with no thermal mass. This is all done internally within EnergyPlus and the user simply needs to define C-factor or F-factor values.

There are 2 ways to define the ground temperatures to be used with F-factor and C-factor calculations which are defined through making one of the settings for the FCFactorMethod ground temperature source:

- **1-Hourly weather file** where the monthly values of ground temperature come from the header of the selected .epw hourly weather file. Note if this option is selected then the header in the selected epw file must include at least one set of monthly ground temperatures. DesignBuilder will read the first set of ground temperatures provided in the header which by convention is the closest to the surface.
- **2-Model data** in which case the 12 monthly temperatures are defined below.

Ground Modelling - Standard Method

How It Works

When using the simplest **1-Standard** ground modelling method, heat is transferred between zones and the ground through one-dimensional ground floor surface elements. The 2-D and 3-D effects in the ground heat conduction that occur in reality are not treated by default. EnergyPlus uses ground construction data to model the storage of heat in the ground and it can accurately model heat transfer processes over several days (so it can model the effect of say a warm week followed by a cool week) but it cannot directly model long-term ground thermal storage effects (periods measured in months).

Tip: [Other more advanced methods](#) are also provided for more accurate 2D/3D ground modelling.

EnergyPlus applies a [constant temperature boundary condition](#) for each month on the lower surfaces of ground adjacent elements.

There are 3 ways to create an adjacency to ground in DesignBuilder:

1. Using DesignBuilder's [Auto adjacency detection](#) where ground adjacency is set for all surfaces at or below ground level. By default, all external surfaces which are at or below ground level are considered to be [adjacent to ground](#). For surfaces that are partially above and partially below ground level ($z=0$), only the parts that are below $z=0$ are modelled with ground adjacency.
2. Using [Ground component blocks](#) - any surfaces (or part of surfaces) touching a ground component block are considered to be adjacent to ground.
3. Using the [Adjacency](#) model data on the Construction tab at block, zone and surface levels where external surfaces can be defined as being adjacent to ground by changing the Adjacency model data on the [Construction tab](#) from **1-Auto** to **3-Adjacent to Ground**. If a surface is below ground level but is not adjacent to ground you can select **2-Not adjacent to ground**.

Ground Temperatures

Monthly [ground temperatures](#) are defined at the site level under [Site details](#).

Modellers generally prefer to define the ground temperature just below the slab and exclude the earth layers from the model. This has the advantage of simplicity and clarity and is the approach recommended by EnergyPlus developers.

A contributing factor is that EnergyPlus cannot model very thick constructions so it is necessary to use less thick constructions (2m or smaller) combined with some assumptions about temperatures at about half a metre or less below the floor.

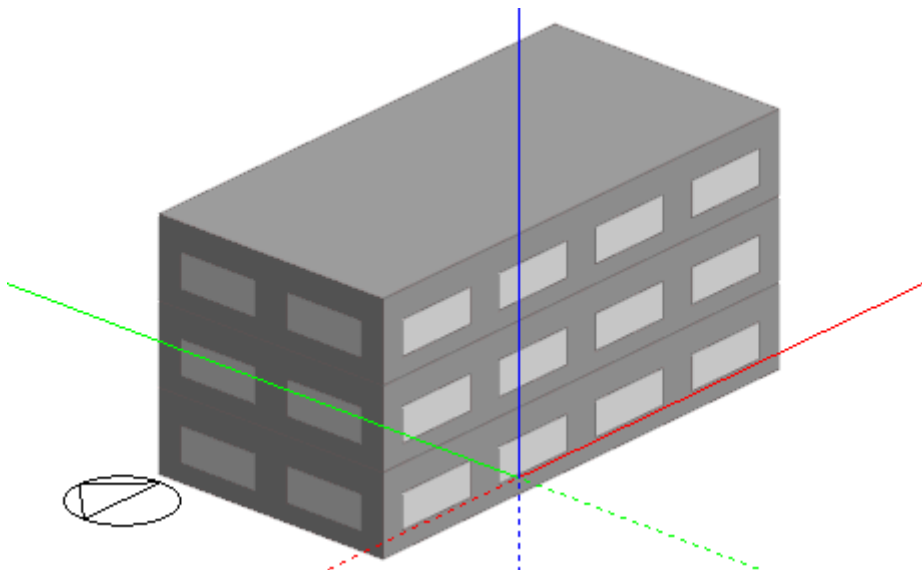
Note: You should not use the ground temperatures in the [EnergyPlus weather file header](#) because these are for undisturbed sites. The temperature beneath buildings is significantly affected by the building itself. The EnergyPlus documentation recommends using ground temperatures of 2°C below average internal temperatures for large commercial buildings (where perimeter heat loss is relatively less important). *Note that this temperature should be applied directly below the slab and should not include ground material.*

Example Application - Basement Or Cellar Under Inclined Ground Slope

This example illustrates how to create a simple 3 storey rectangular building model with:

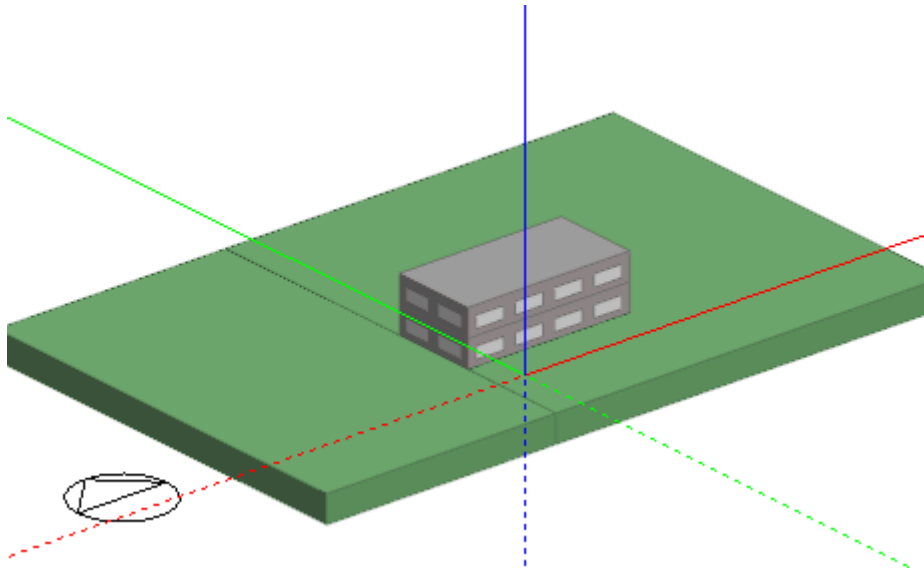
- Basement which is fully below ground and
- Ground floor is partially buried under an inclined ground plane.

Step 1



Draw 3 blocks on the $z=0$ ground plane. The lowest block will be the basement (completely buried), the middle block will be the ground level block, partially buried and the top block will be the second level. So far only the bottom surface is adjacent to ground because it sits on the $z=0$ plane.

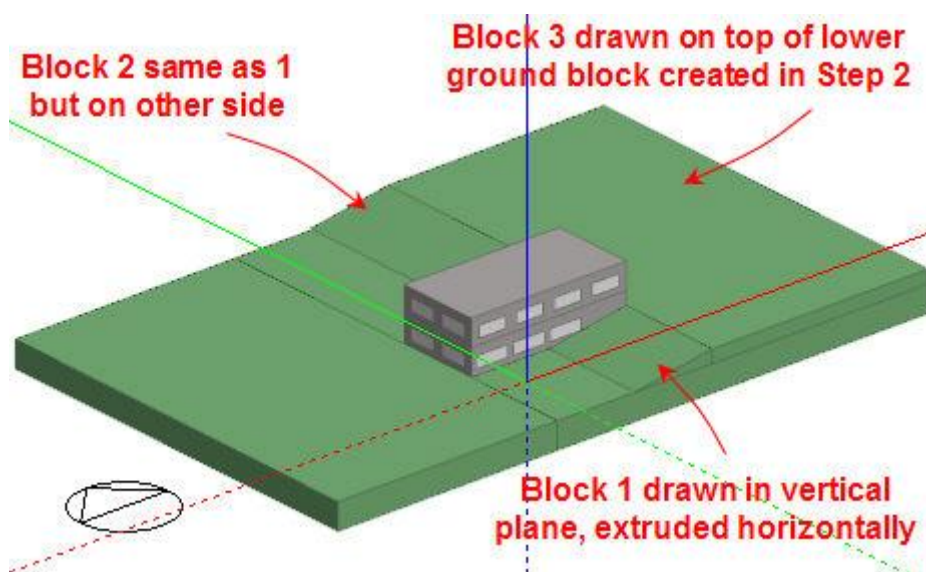
Step 2



Draw 2 ground component blocks around the basement building block to give it's surfaces ground adjacency.

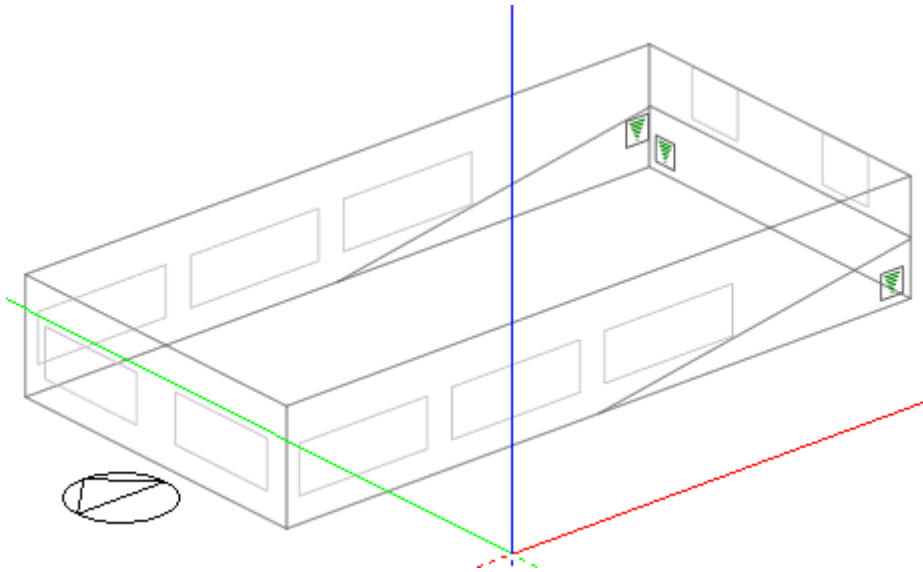
Note this step could be avoided by moving the whole building down so that the basement is fully below ground. To do this go to building level, select all the blocks, select the move command, click on one of the bottom corners of the middle block to create a reference point then click on a blank part of the screen to move the reference point down to $z=0$.

Step 3



Draw three new [ground component blocks](#) to create the sloping ground and the upper ground levels.

Step 4

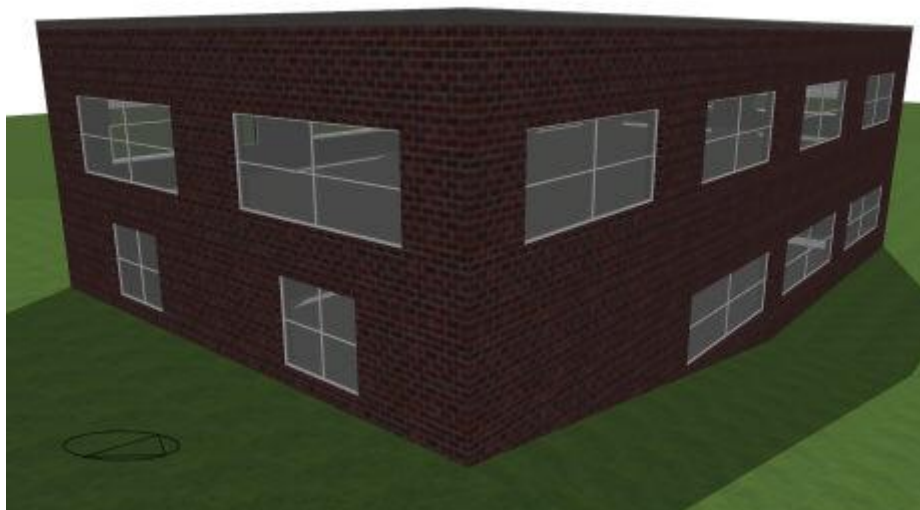


The zone of the middle block is shown above. Notice how 3 of the surfaces have been split into external and ground adjacencies (ground adjacencies are flagged by green ground symbols). Note also that the portions of the surfaces touching the ground component block cannot have windows but DesignBuilder attempts to set up the correct % glazing for the remaining external parts of the surface.

Step 5

The ground component blocks probably do not need to shade this model as they are mainly below all surfaces that can receive solar radiation/light, but if you are modelling reflections you will probably need to include the reflection of solar radiation and light from the ground blocks during simulations. In other cases you may prefer to switch off the generation of the component blocks shading/reflection surfaces to speed simulations. Do this on the Constructions model data tab under [Component Block](#).

Step 6



The finished model. The texture of Ground component blocks is defined by the [material](#) set under Component blocks on the Constructions tab.

Kiva Summary

This summary applies when the [Ground modelling method](#) has been set to **2-Kiva Basic** or **4-Kiva Full**.

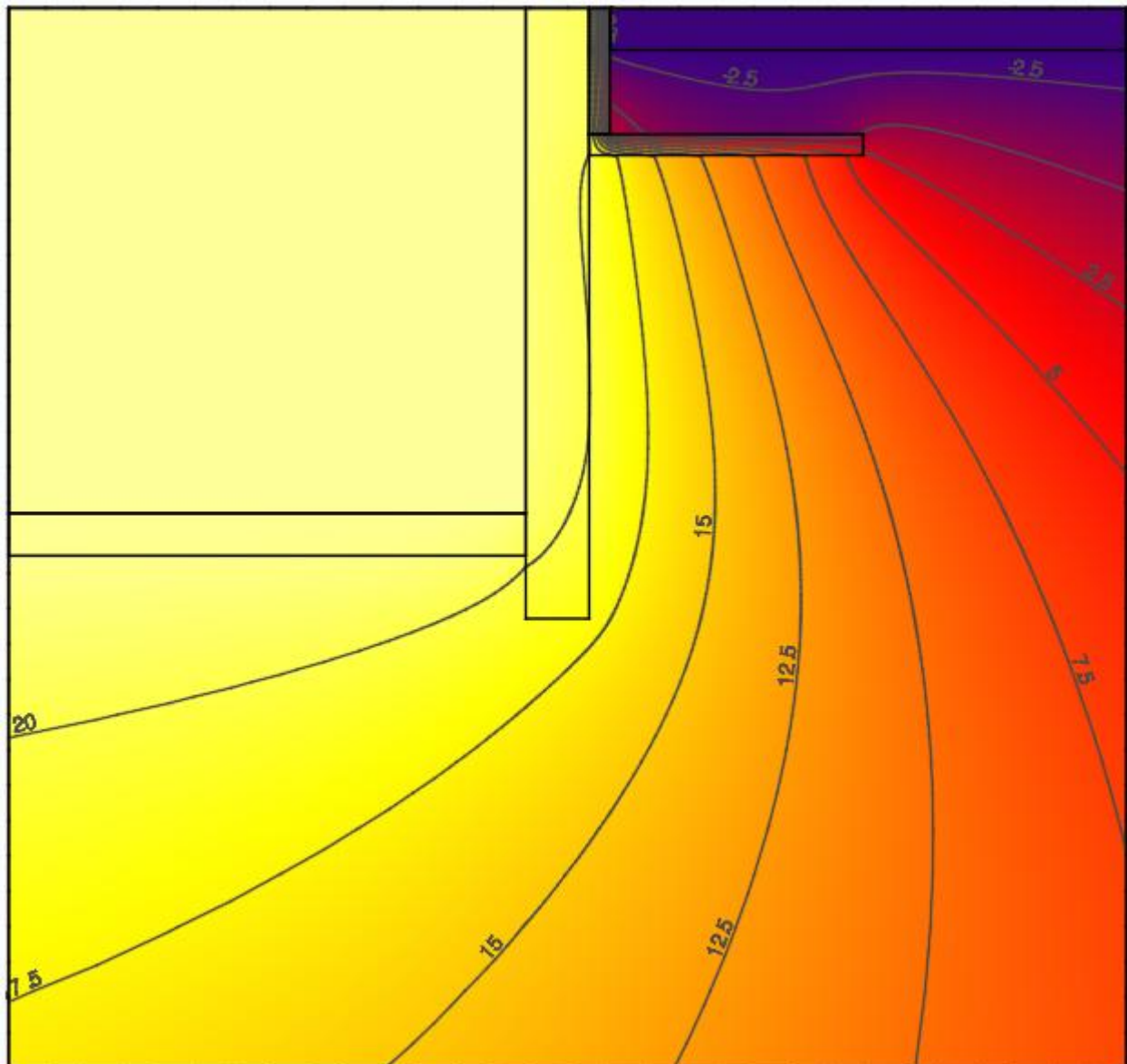
[Kiva™](#) is an open source foundation heat transfer calculation tool developed by Big Ladder Software. It is the product of research by Neal Kruis where he proved that accurate foundation heat transfer calculations can be performed quickly without any noticeable loss of accuracy relative to a mesh-independent, fully three-dimensional simulation.

Within EnergyPlus, Kiva is used to perform a set of two-dimensional finite difference heat transfer calculations. Each [Kiva foundation](#) is represented by a single floor and optionally a basement wall. Any ground-adjacent walls linked to the Kiva Foundation are mapped to a single representative wall in the two-dimensional context using an area weighted average for any non-uniform boundary conditions among the walls.

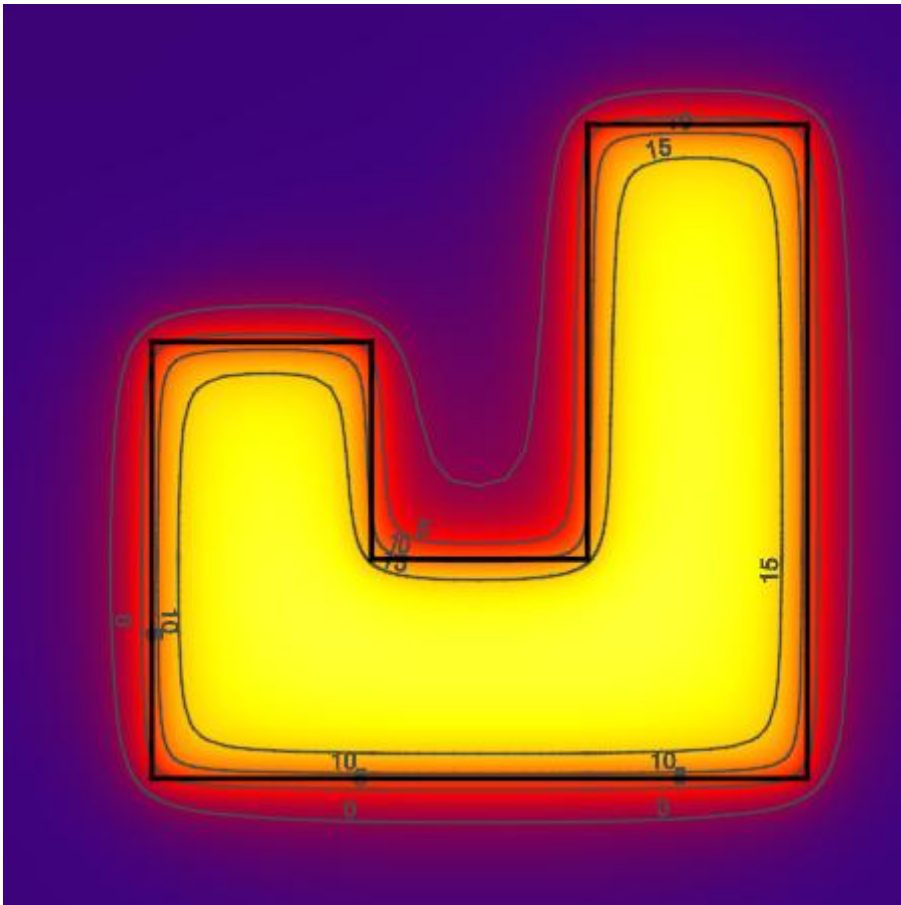
Kiva uses these EnergyPlus boundary conditions to calculate the convective heat gains and surface temperatures for the floor and wall surfaces associated with a single [Kiva foundation](#) object:

- weather data,
- solar position, and
- zone temperatures (from previous timestep),
- zone radiation (solar, IR, etc.)

Because Kiva performs multi-dimensional finite difference calculations, the associated surfaces do not use the same [Solution Algorithm](#) (e.g. CTF, Finite difference etc) as the rest of the model.



Example profile output from standalone Kiva



Example plan output from standalone Kiva

Two-Dimensional Approximation

The two-dimensional approximation method employed by Kiva relies on knowing the footprint shape, area, and exposed perimeter of each instance. The appropriate footprint shape, area, and exposed perimeter for each instance will be defined within the context of the overall geometry of the Foundation surfaces.

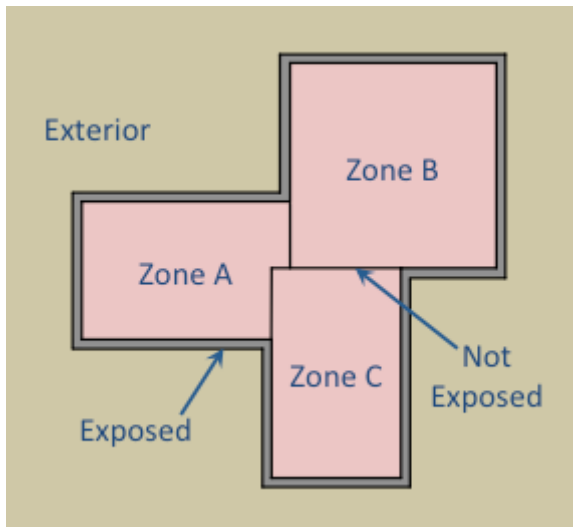
The general method is to define the width of the floor (the distance from the symmetry plane to the wall interior), w in the two-dimensional context as:

$$A/P_{\text{exp}}$$

where:

A is the area of the foundation footprint, and,

P_{exp} is the exposed perimeter of the foundation, that is the perimeter of the foundation footprint that is in contact with the ground, as opposed to being in contact with an internal space, see image below.



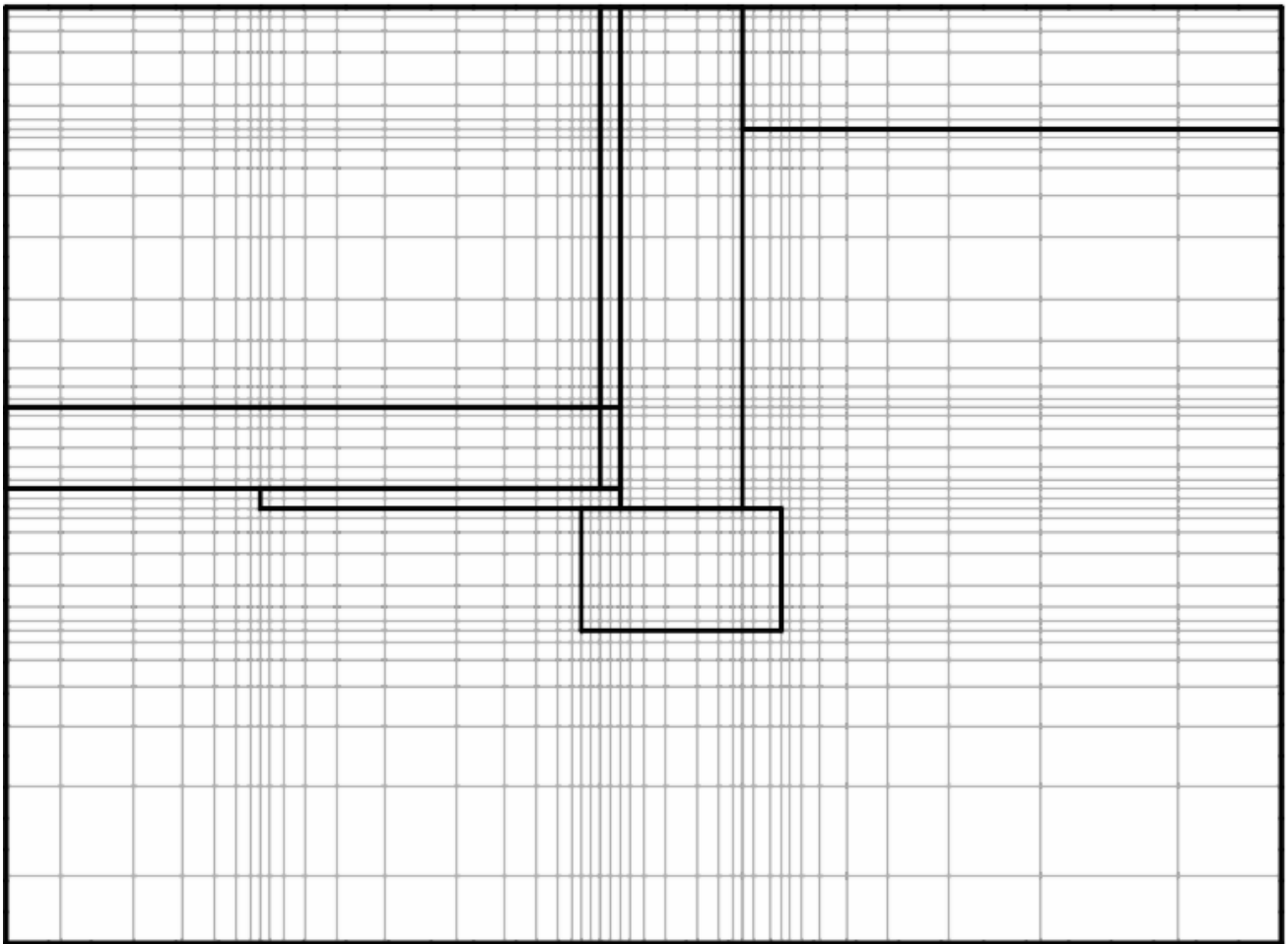
Note: One of the key inputs for each EnergyPlus Kiva Foundation is the “Exposed foundation perimeter”, i.e. the length of the foundation perimeter edge that is adjacent to the ground. DesignBuilder calculates this behind the scenes based on the model geometry. This is one of the advantages of using DesignBuilder for your Kiva simulations.

Kiva also has the capability to adjust this width to account for concave foundation footprint shapes. This adjustment is based on the boundary layer adjustment method described by Kruis and Krarti (2017). The approach adjusts the exposed perimeter to account for interactions in heat flow within concave corners and narrow gaps between two exposed edges.

This approach allows for accurate representation of building foundation heat transfer without performing three-dimensional calculations. Because the two-dimensional context is symmetric, the domain can be divided in half to further reduce the number of calculations.

Numerical Calculations

Kiva automatically discretises the two-dimensional domain into rectangular cells. The size of each cell is defined by the [Kiva foundation settings](#) object’s **Minimum cell dimension** and **Maximum Cell Growth Coefficient**. The **Minimum cell dimension** defines the smallest possible dimension of a cell within the domain. Cells along a block boundary start at this size and grow geometrically away from the boundary according to the “**Maximum cell growth coefficient**”. This is evident from the diagrams below which show the discretisation for a single foundation at close and far perspectives, respectively.



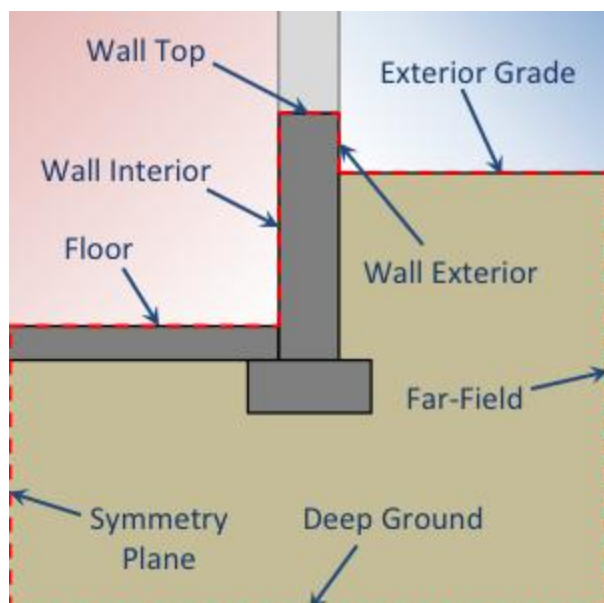
Example generated discretisation near foundation perimeter



Example generated discretisation of full domain

The discretised partial differential equations are solved using the Alternating Direction Implicit (ADI) finite difference time stepping scheme. This scheme provides relatively fast calculations with stable results as demonstrated by Kruis and Krarti (2015).

Boundary Conditions



Boundary conditions in Kiva's two-dimensional context

Symmetry Plane: Zero heat flux in the horizontal direction.

Wall Top: Zero heat flux in the vertical direction (assumes heat transfer through the exterior wall above is one-dimensional in the horizontal direction).

Deep Ground: Either constant temperature or zero vertical heat flux, depending on user input. Deep ground depth may be automatically calculated based on water table estimates using a method defined by Williams and Williamson (1989).

Far-Field: Zero heat flux in the horizontal direction. If this boundary is sufficiently far from the building, this will result in an undisturbed ground temperature profile.

Interior Surfaces:

- Convection is calculated according to the [6-TARP](#) method.
- Long and short wave radiation is passed from EnergyPlus radiant exchange and interior solar distribution algorithms. Note: Kiva uses area weighted averages to define the radiation incident on walls in the two-dimensional context.

Exterior Surfaces:

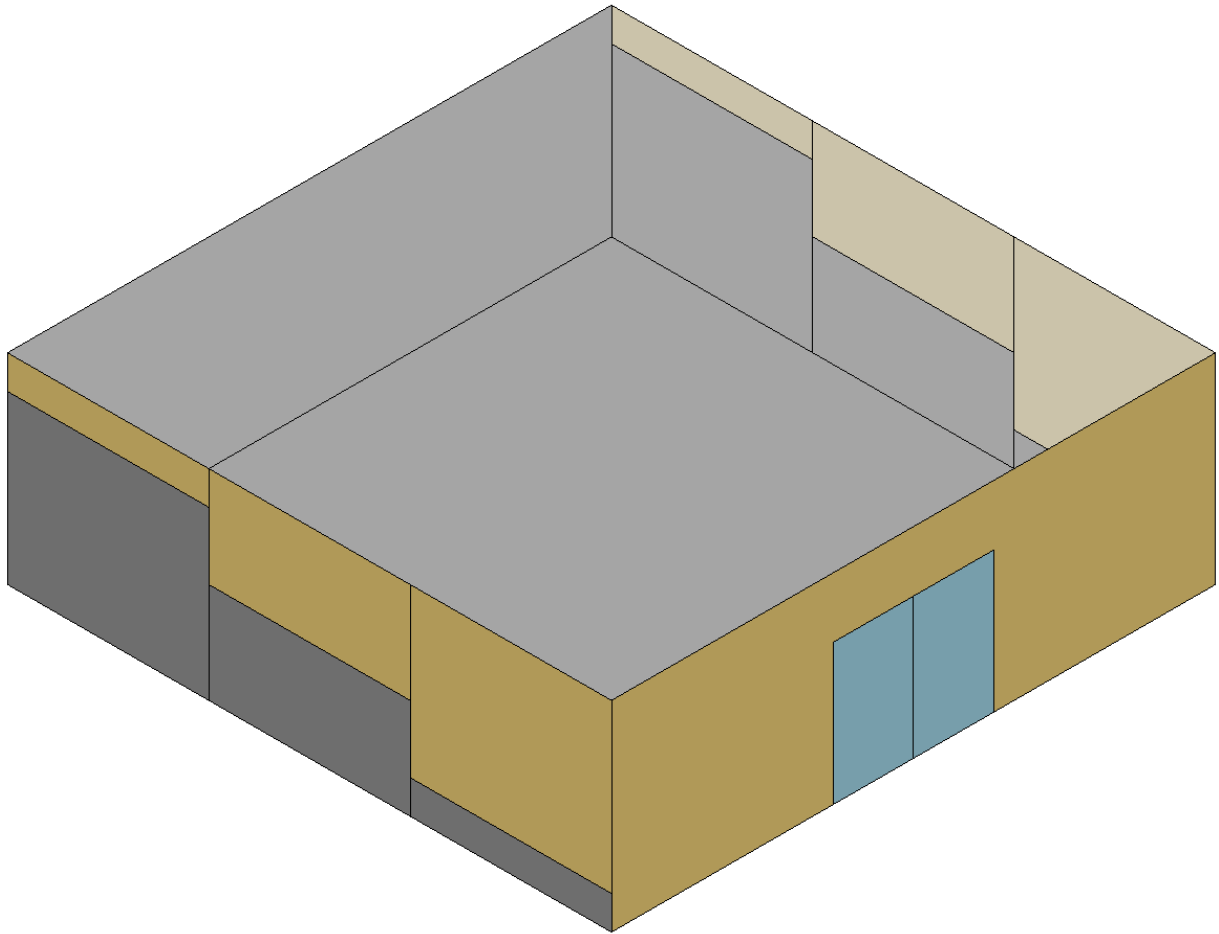
- Convection is calculated according to the [6-DOE-2](#) method. Wind speeds along the exterior grade are calculated at the roughness height.
- Exterior long wave radiation is calculated using the same algorithms used for other EnergyPlus surfaces. Note: there is no explicit radiant exchange between the ground and building surfaces.
- Exterior solar incidence is uniform along the exterior grade surfaces. No shading is taken into account. Solar incidence along the wall exterior

Multiple Kiva Instances

In some cases, a single Foundation boundary condition might require multiple Kiva instances.

Multiple Zones: If the surfaces in several zones reference the same [Kiva foundation](#) object, each zone will internally be calculated using separate Kiva instances, one per EnergyPlus floor surface.

Walk-Out Basements: Walkout basements are defined by using walls of different heights all referencing the same [Kiva foundation](#) object.

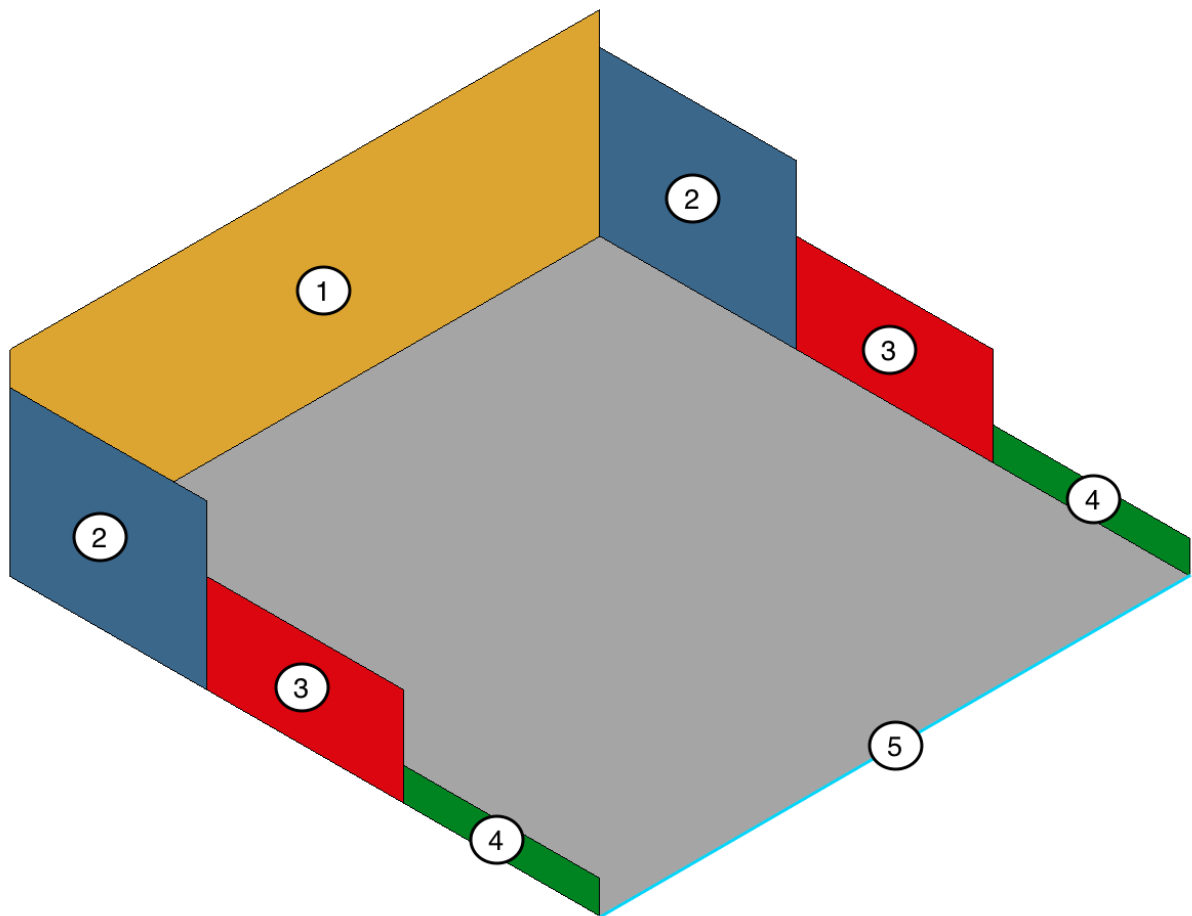


Walkout basement surfaces (in grey) all reference the same Foundation:Kiva object

A separate Kiva instance will be run internally for any walls with different average heights associated with the same [Kiva foundation](#) object.

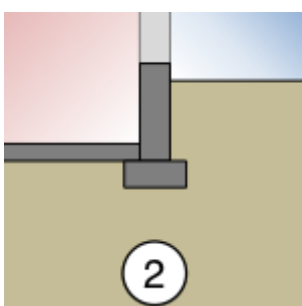
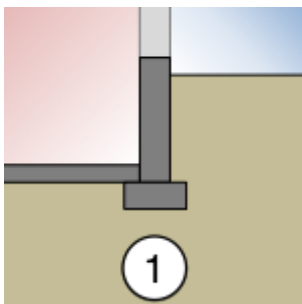
For non-rectangular surfaces, EnergyPlus uses the average height of each basement wall surface. A separate Kiva instance is created for each wall (or group of walls) with the same average height.

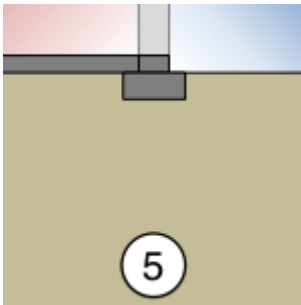
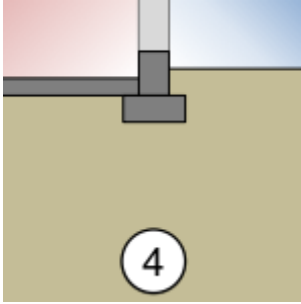
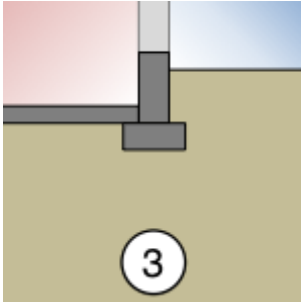
The diagram below shows the grouping of walls by height based on the basement illustrated above, including the portion that is only a slab.



Walkout basement Kiva instances (one for each wall height)

The resulting five two-dimensional contexts will look like the figures below.





Each Kiva instance with a different wall height will calculate different heat fluxes, convective coefficients and surface temperatures for both the wall and the floor. The heat flux through the associated floor will be weighted according to the fraction of the total exposed perimeter, $P_{exp,tot}$, represented by each segment of different height. The total heat flux through the walkout basement floor is:

$$\dot{q} = \sum_i^{N_{segs}} \frac{P_{exp,i}}{P_{exp,tot}} \cdot h_i (T_{\infty} - T_{floor,i})$$

The weighted average convective coefficient for the walkout basement floor surface is:

$$\bar{h} = \sum_i^{N_{wall,segs}} \frac{P_{exp,i}}{P_{exp,tot}} \cdot h_i$$

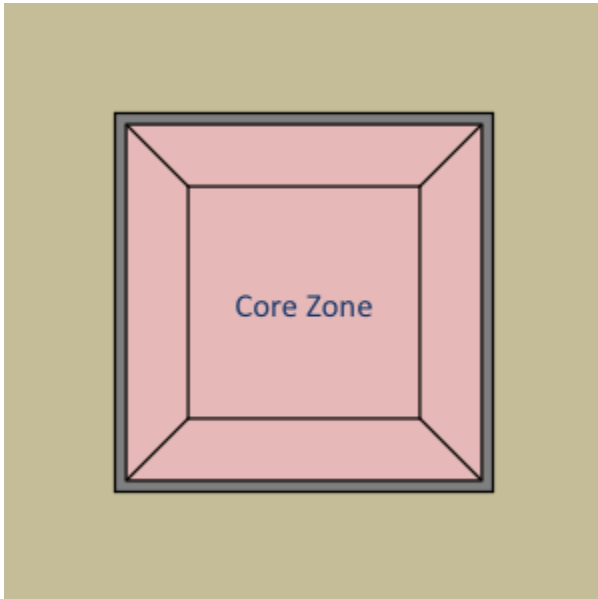
The weighted average temperature for the floor surface is:

$$\bar{T}_{floor} = T_{\infty} - \dot{q} / \bar{h}$$

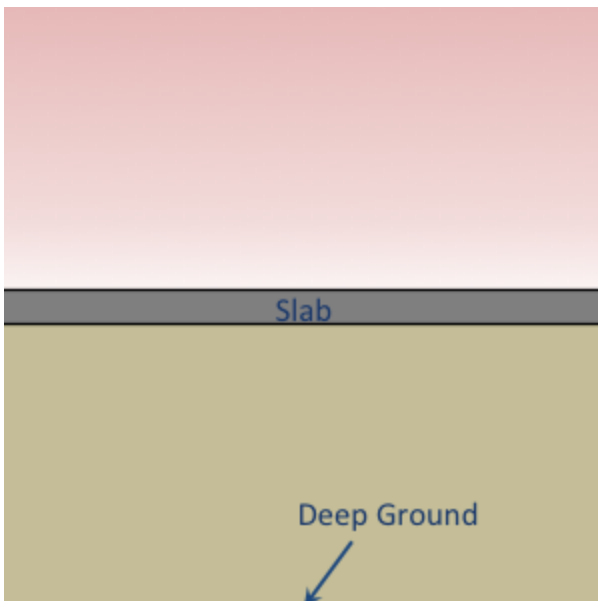
Multiple Floor Surfaces: If a floor has multiple constructions (e.g., carpeted and bare) each surface must reference a separate [Kiva foundation](#) object, or be combined into a single equivalent construction.

Core Zone Slabs

Because core zones have no exposed perimeter, they are assumed to exchange heat only with the deep ground boundary condition. This is calculated using a one-dimensional finite difference formulation. The associated Kiva instance will use only the description of the slab and the deep ground boundary condition to define the heat flux through the surface.



Core zone (no exposed perimeter)



Core zone one-dimensional context

Initialisation And Warm-Up

Summary

EnergyPlus uses an accelerated initialisation, which starts with the Kiva [Steady State](#) method. This solution scheme initialises the domain temperatures using the first timestep's boundary conditions in a steady-state finite difference solution, giving an initial condition temperature variation across the domain. EnergyPlus then runs a few months of accelerated (long) timesteps before holding the ground state constant for warm-up days to ensure that the warmup simulations don't erase the thermal history of the ground.

Detail

The traditional “warm-up” (aka preconditioning) period in EnergyPlus (of repeating a single day) presents some challenges for foundation heat transfer calculations:

- As the ground can have time constants on the order of years, a single day is simply not long enough to adequately capture the thermal history of the ground.
- Any repetition of a single day would erase any pre-calculated thermal history and likely take much longer to converge.

Instead, Kiva instances are initialized independently from the rest of the simulation using the accelerated initialisation method developed by Kruis (2015). This method analyses the weather file and simulates long timestep (in the order of weeks or months) calculations using an implicit numerical scheme. These long timesteps allow Kiva to capture a long term history of the ground without running the entire building model.

The initialisation of the ground domain can either use a [user-defined value](#) for the temperature of adjacent buildings, or the indoor air temperatures can be inferred (as they are not yet calculated by EnergyPlus). In the latter case, when a thermostat is assigned to a zone with Kiva foundation surfaces, the setpoint (or a weighted average of heating and cooling setpoints depending on the outdoor temperature) is used. Alternatively, for zones without thermostats, a constant 22°C indoor temperature is assumed.

HVAC Autosizing

In EnergyPlus HVAC autosizing calculations, the state of the Kiva ground domain is approximated by running a few long timestep simulations (in the order of months) using some basic assumptions about the state of the building. EnergyPlus holds the ground constant in this initial state during warmup but allows it to vary during the autosizing calculations themselves. This process helps to ensure reasonable convergence times.

Kiva Limitations

Some limitations in the current implementation of Kiva are:

1. Only floors and walls can connect to the ground through Kiva foundation boundary conditions.
2. At least one floor surface must reference each Kiva foundation object.
3. To model 2 stories at -1 and -2 levels using Kiva requires a workaround since when using Kiva for a zone in EnergyPlus, that zone must have one floor surface touching ground for Kiva to calculate heat transfer. There are 2 ways to do this:
 - a. Keep the zones stacked, but use Kiva for only the -2 level zone and change the boundary condition for the walls of the -1 level zone to use standard ground modelling, i.e. uncheck the [Kiva adjacency](#) checkbox for these wall surfaces. This is necessary because a Kiva wall from upper zone can't be connected to the floor of the zone below and Kiva walls must be connected to a Kiva floor.
 - b. [Merge the two zones](#) together to remove the ceiling for the -2 level zone and floor for the -1 level zone and modelling that surface as "Internal Thermal Mass" on page 361 instead.The second option would generally be the easiest and most accurate choice.
4. Kiva ground modelling is not compatible with Fanger PMV temperature control in E+ v9.4. The bug is fixed in E+ v9.6 and later.
5. Outputs for surfaces with a Foundation boundary condition type will include all opaque surface output variables except:
 - o Surface Outside Face variables (since there is no “Outside Face”).
 - o Surface Heat Storage variables (since this definition depends on an “Outside Face”).
 - o Surface Internal Source Location Temperature (Kiva doesn't handle internal sources yet, but this is a possible future enhancement).
6. Kiva is not compatible with F-Factor constructions used in Kiva ground adjacent surfaces. This means that you should not use Kiva for LEED and ASHRAE 90.1 Appendix G simulations.
7. Kiva is not compatible with constructions with [Simple resistance](#) (R-value) material layers in Kiva ground adjacent surfaces.
8. Kiva is not compatible with constructions with an internal source (e.g. underfloor heating) used in Kiva ground adjacent surfaces. You can use surfaces with an internal source in any other surface in Kiva simulations though. See [EnergyPlus issue #6576](#).

9. The Kiva ground domain is not used by Detailed HVAC [Ground heat exchangers](#) which have their own representation of the ground temperature field, typically based on G-function data for [vertical ground heat exchangers](#).

Validation

Kiva has been tested against the BESTEST Ground coupled cases with accuracy within 3% of the reference solutions (Kruis and Krarti, 2015).

References

- [1] N. Kruis and M. Krarti, "KivaTM: A Numerical Framework for Improving Foundation Heat Transfer Calculations," Journal of Building Performance Simulation, vol. 8, no. 6, pp. 449-468, 2015.
- [2] N. Kruis, "Development and Application of a Numerical Framework for Improving Building Foundation Heat Transfer Calculations," Ph.D. Dissertation. University of Colorado, 2015.
- [3] N. Kruis and M. Krarti, "Three-dimensional accuracy with two-dimensional computation speed: using the KivaTM numerical framework to improve foundation heat transfer calculations," Journal of Building Performance Simulation, vol. 10, no. 2, pp. 161-182, 2017.
- [4] T. Williams and A. Williamson, "Estimating Water-Table Altitudes for Regional Ground-Water Flow Modeling, U.S. Gulf Coast," Ground Water, vol. 27, no. 3, pp. 333-340, 1989.

Horizon

[Location tab](#) on model data at Site level

You can add a horizon to your model to include the effects of terrain or more distant tall buildings on the shading of the simulated building. Horizons are modelled by defining a horizon elevation angle for each direction (azimuth) around the building.

The horizon data is used in Cooling design calculations and EnergyPlus Simulations, but not in Daylighting simulations and Visualisations.

Note: To include the shading effect of *nearby* buildings you should generally use component blocks to represent those buildings for most accurate modelling. This is more accurate because their proximity to the modelled building will give rise to different horizon elevations for different parts of the building. The horizon system on the other hand would not capture this as it generates distant shading objects (100m away from the building by default) which for most buildings will provide an approximately constant horizon for all parts of the building.

Include horizon in the model

Check this option to include the effects of terrain such as mountains, hills or more distant buildings.

Horizon component

Select the [horizon component](#) that defines the elevation of the horizon for each azimuth angle around the building.

Site Details - Mains Water Temperature

[Location tab](#) on model data at Site level

The Water Mains Temperature data is used to calculate water temperatures delivered by underground water main pipes. The mains temperatures are used as default, make-up water temperature inputs for the [Water](#)

[heater](#). The mains temperatures are also used in the water systems objects to model the temperature of cold water supplies.

Note: This data is only used when using Detailed HVAC. For Simple HVAC a single annual average mains water temperature is defined under the [DHW](#) header on the HVAC tab.

Water mains temperatures are a function of outdoor climate conditions and vary with time of year. A correlation has been formulated to predict water mains temperatures based on two weather inputs:

- Average annual outdoor dry-bulb air temperature.
- Maximum difference in monthly average outdoor air temperatures.

These values can be easily calculated from annual weather data using a spreadsheet or from the ".stat" file available with the EnergyPlus weather files at www.energyplus.gov. Monthly statistics for dry-bulb temperatures are shown with daily averages. The daily averages are averaged to obtain the annual average. The maximum and minimum daily average are subtracted to obtain the maximum difference. For more information on the water mains temperatures correlation, see the EnergyPlus Engineering Document.

Alternatively, the water mains temperatures can come from a schedule. This is useful for measured data or when water comes from a source other than buried pipes, e.g., a river or lake.

Calculation method

Select from the options:

- **1-Schedule** where the water mains temperature is scheduled.
- **2-Correlation** where the water mains temperature is calculated from the average outdoor air temperature and range of monthly outdoor air temperatures.

Schedule

If the Calculation method is **1-Schedule**, the water mains temperatures are read from the schedule referenced by this field. Note that the schedule temperatures must be in °C even when using IP units.

Annual average outdoor air temperature

If the Calculation method is **2-Correlation**, this field is used in the calculation as the annual average outdoor dry-bulb air temperature (in °C or °F).

Maximum difference In monthly average outdoor air temperatures

If the calculation method is **2-Correlation**, this field is used in the calculation as the maximum difference in monthly average outdoor air temperatures (in Δ°K or Δ°F).

Site Details - Precipitation + Irrigation

[Location tab](#) on model data at **Site level**

Precipitation

[Location tab](#) on model data at **Site level**

The Site precipitation data is used to describe the amount of precipitation on all **Green roof** surfaces over the course of the simulation. Precipitation includes both rain and the equivalent water content of snow. Precipitation is not yet described well enough in the usual building weather data file. So this object is used to provide the data using Schedule objects that define rates of precipitation in meters per hour. A set of schedules for site precipitation have been developed for USA weather locations and are provided with EnergyPlus in the data set called "PrecipitationSchedulesUSA.idf." The user can develop schedules however

they want. The schedules in the data set were developed using EnergyPlus' weather file (EPW) observations and the average monthly precipitation for the closest weather site provided by NOAA. EPW files for the USA that were based on TMY or TMY2 include weather observations for Light/Moderate/Heavy rainfall, however most international locations do not include these observations. The values were modeled by taking the middle of the ranges quoted in the EPW data dictionary. The assumed piecewise function is shown below.

Amount	Light	Medium	Heavy
m/h	0.0125	0.052	0.1

The values were inserted on hour by hour basis for the month based on the observations. Then each month was rescaled to meet the average precipitation for the month based on the 30-year average (1971-2000) provided by the NOAA/NCDC. Therefore, the flags in the EPW file match the precipitation schedules for the USA. Note that summing the average monthly precipitation values will not give you the average yearly precipitation. The resulting value may be lower or higher than the average yearly value. Once the typical rainfall pattern and rates are scheduled, the Site Precipitation data provides a method of shifting the total rainfall up or down for design purposes. Wetter or drier conditions can be modelled by changing the Design Annual Precipitation although the timing of precipitation throughout the year will not be changed.

Design annual precipitation

Magnitude of total precipitation for an annual period to be used in the model. Value selected by the user to correspond with the amount of precipitation expected or being assumed for design purposes. The units are in meters. This field works with the following two fields to allow easily shifting the amounts without having to generate new schedules. Units are m/yr or ft/yr

Precipitation schedule

The schedule that describes the rate of precipitation. The precipitation rate schedule is analogous to weather file data. However, weather files for building simulation do not currently contain adequate data for such calculations. Therefore, EnergyPlus schedules are used to enter the pattern of precipitation events. The values in this schedule are the average rate of precipitation in meters per hour. The integration of these values over an annual schedule should equal the nominal annual precipitation.

Nominal annual precipitation

Magnitude of annual precipitation associated with the rate schedule. This value is used to normalize the precipitation.

Site Green Roof Irrigation

The Site Roof irrigation data is used to describe the amount of irrigation on all **Green roof** surfaces over the course of the simulation. You can define time-varying irrigation data using a maximum irrigation value and a schedule. There is also an option to apply some intelligence to the irrigation system to avoid over-watering the green roof.

Irrigation type

Defines the way irrigation water is supplied. Select from:

- **1-Scheduled** - water is provided using the maximum irrigation rate and schedule modifier (below).
- **2-Smart** - as Scheduled but overrides the irrigation to be off when the earth is 30% or more saturated with water.

Max irrigation rate

Maximum rate of irrigation per hour. Units are m/h or ft/h.

Irrigation schedule

The schedule defining the operation of the irrigation system. A schedule value of zero means the irrigation is switched off and a value of 1 means that the **Max irrigation rate** is used.

Site Details - Outdoor Air CO2 And Contaminants

[Location tab](#) on model data at [Site level](#)

The Outdoor air CO2 and generic contaminants data at site level provides a way to select which contaminant types will be simulated. Although carbon dioxide is not normally considered to be an indoor contaminant it is used as an indicator of indoor air quality in buildings. From the simulation point of view EnergyPlus treats carbon dioxide as a type of contaminant. In addition to carbon dioxide, a generic contaminant type model is available.

These options should be selected if you wish to model CO2 or contaminant concentration levels in the simulation, either to monitor zone air quality or if [DCV control](#) is being used.

Note: DesignBuilder automatically selects the **CO2 concentration** and **Generic contaminant concentration** options below as appropriate if you have one of the IAQP DCV systems requiring CO2 or generic contaminant tracking selected in a Detailed HVAC model.

Simulate CO2 concentration

Check this option if simulation of CO2 concentration levels are to be performed.

CO2 concentration schedule

Select the schedule that defines outdoor air carbon dioxide level values in units of ppm. The default schedule defines outdoor constant CO2 concentration at 400ppm based on NOAA ESRL sources.

Simulate contaminant concentration

Check this option if simulation of contaminant concentration levels are to be performed.

You can enter details on zone [contaminant generation and removal](#) on the Activity tab.

Contaminant schedule

Select the schedule that defines outdoor air contaminant level values in units of ppm.

Time And Daylight Saving

[Location tab](#) on model data at [Site level](#)

Time zone

Select the [Time zone](#) for the site location. The time zone defines both the standard time meridian and information on daylight saving. Generally you should leave this as the default for the location.

You can override daylight savings to be off by unchecking the **Use daylight saving** model data.

DesignBuilder models the effect of daylight saving time where the clocks are moved forward one hour in the summer months in most non-equatorial countries. The timing of this is handled automatically using the [Time zones database](#) when you load data from one of the [Location templates](#) supplied with the program.

There are some factors you should be aware of:

- In EnergyPlus, summer Daylight saving works by adding the number of hours specified in the Time zone data (usually 1 hour) to the local mean time to get the time to determine schedule operation. Because [Simulation results](#) are displayed in **local mean time**, during the summer when daylight saving time is in use, occupancy and equipment etc will appear to start 1 hour earlier in the day than it does in

the winter. To help you pinpoint this, DesignBuilder prints the times when daylight saving starts and ends on the graph.

- When comparing results with and without Daylight saving you will notice:
 - Solar radiation data has exactly the same timing with and without daylight saving.
 - Occupancy, equipment (and other aspects of building operation controlled by schedules) will start 1 hour earlier with daylight saving.
- [Visualisations](#) use local mean time when setting the position of the sun for site shading analysis. Again you can account for daylight saving time by adding 1 hour to the time during summer months to see shading for local clock time.

See also:

- [Hourly and Sub-Hourly Results Times](#)

Start/End of Winter/Summer

These settings are loaded from the [Location template](#) and cannot be edited in Model data.

For Northern hemisphere locations you should normally expect to see data like:

- Start of Winter = Oct
- End of Winter = Mar
- Start of Summer = Apr
- End of Summer = Sep

For Southern hemisphere locations you should normally expect to see data like:

- Start of Winter = Apr
- End of Winter = Sep
- Start of Summer = Oct
- End of Summer = Mar

Note: It is important that the selected months cover the entire year.

Simulation Weather Data

[Location tab](#) on model data at [Site level](#)

Hourly weather data

Select the hourly weather data from the list of Hourly weather data components. These components reference the epw file that will be actually used in the simulation.

DesignBuilder uses EnergyPlus epw format hourly weather data for:

- [EnergyPlus simulations](#) - note that heating and cooling design calculations use simpler design weather data.
- [Daylighting annual simulations](#) - in this case the weather data is used to provide Radiance with hourly direct and diffuse illuminance data for the site.

There is no direct link between hourly weather data and the ASHRAE design values on the [Winter](#) and [Summer](#) design data tabs.

You can review, edit and convert weather using the [Weather Data Analysis dialog](#).

You can also [add your own hourly weather data](#).

See also the [Site level data and weather files](#) and [Hourly weather data](#) tutorials.

Day of week for start day

For flexibility, the day of week indicated in the weather file can be overridden by this field's value. Select from:

- **1-Sunday**,
- **2-Monday**,
- **3-Tuesday**,
- **4-Wednesday**,
- **5-Thursday**,
- **6-Friday**,
- **7-Saturday**,
- **8-Use weather file** - use the weather file to decide the first day of the week.

This value will be used as the Start day of the week for the simulation. The default is **8-Use weather file** where the day of week comes from the weather file.

Note: There is an option to [specify](#) the [Start year](#) of the simulation on the Simulation Calculation options dialog. When the start year is specified it overrides this setting and the year will define the day of the week of the start day.

Winter Design Weather Data

[Location tab](#) on model data at [Site level](#)

Heating design and simulation autosizing simulations use simplified winter design weather based on a worst case design day.

The weather data used in [Heating design calculations](#) is as follows:

- **Minimum outside dry-bulb temperature** - the winter design outside air temperature.
- Co-incident **Wind speed**. This data is used to help calculate external convective heat transfer coefficients.

In addition for zone heating load autosizing this additional data is used:

- Coincident **Wind direction** is used in Calculated natural ventilation autosizing calculations (along with the wind speed above).

The degree of 'confidence' of the design data can be selected, i.e. the probability that the design data will occur in practice based on conditions encountered over recent years. Select either 99.6 or 99% confidence (i.e. 0.4 or 1% chance of more extreme winter weather occurring).

See also the [Site level data and weather files](#) and [Heating Design Calculations](#) tutorials

Sizing Period

When running simulations, some design features may benefit by using a longer design weather period for HVAC autosizing or loads calculations. Longer design periods may be set up by allowing the hourly weather file to be used in the sizing calculations.

Autosize method

Choose the design HVAC autosizing method to use for simulations from the following options:

- **1-Design day**, the default option where data for a single winter design day is used based on the settings described above.
- **2-Weather file days** where you can select the period to be used from the weather file for sizing.
- **3-Weather file condition type** where you can select from some standard per-defined types of period such as **3-Winter extreme**.

When **2-Weather file days** is selected, enter the From **Start day** and **Month** and the To **Start day** and **Month** as shown in the screenshot below.

Sizing Period	
Autosize method	2-Weather file days
From	
Start day (°)	1
Start month	Dec
To	
End day (°)	7
End month	Dec

Period selection

When the **3-Weather file condition type** Autosize method is selected you must choose the Weather file condition type from the following list.

- **1-Summer extreme**
- **2-Summer typical**
- **3-Winter extreme** (default setting)
- **4-Winter typical**
- **5-Autumn typical**
- **6-Spring typical**
- **7-Wet season**
- **8-Dry season**
- **9-No wet season**
- **10-No dry season**
- **11-Tropical hot**
- **12-Tropical cold**

Note: Not all weather files have all of the valid choices so this option is not completely generic. For example there may be extreme cold periods in some weather files but extreme wet periods (tropical) in others.

Summer Design Weather Data

[Location tab](#) on model data at [Site level](#)

Cooling design and simulation autosizing simulations use simplified summer design weather based on worst-case design days.

The weather data used in [cooling design calculations](#) is as follows:

- Maximum outside dry-bulb temperature - the maximum dry bulb air temperature over the day
- Minimum outside dry-bulb temperature - the minimum dry bulb air temperature (night time)

- Wet-bulb temperature at the time of the maximum dry-bulb temperature.

In addition for zone cooling load simulation autosizing this additional data is used:

- Dry bulb temperature range modifier type to help define the daily profile of the outside air temperature
- Wind speed and direction for use in Calculated natural ventilation autosizing calculations.

The degree of 'confidence' of the design data can be selected, i.e. the probability that the design data will occur in practice based on conditions encountered over recent years. This risk can be specified based on dry or wet-bulb temperatures, for 99.6, 99 and 98% confidence (i.e. 0.4, 1 or 2% chance of more extreme weather occurring).

See also the [Site level data and weather files](#) and [Cooling Design Calculations](#) tutorials

Sky

Solar model indicator

Choose the method to be used for solar modelling in the calculation of the solar radiation in the summer design day(s):

- **1-ASHRAE Clear Sky**, the default model described in ASHRAE HOF 2005 Chapter 31, can be used to estimate hourly clear-day solar radiation for any month of the year in U.S. or similar temperate climates in the northern hemisphere. EnergyPlus calculations extend the clear sky application to both northern and southern hemispheres. Calculations use a [Clearness](#) value as part of their calculations. The Clear Sky model usually over estimates the amount of solar radiation available to the building.
- **2-ASHRAE Tau**, aka the ASHRAE Revised Clear Sky Model is introduced in the ASHRAE 2009 HOF as a revised clear sky model based on location-specific optical depths for direct and diffuse radiation. These values are tabulated by month for all 5564 locations in the ASHRAE design data that accompanies the 2009 HOF. This data is included with EnergyPlus installations in C:\EnergyPlusV8-9-0\PreProcess\WeatherConverter\ASHRAE_2013_Monthly_DesignConditions.csv. The Taub and Taud design values for each month of the year are provided in the first 24 data columns. Studies done as part of ASHRAE research projects show that the revised tau model produces more physically plausible irradiance values than does the traditional clear sky model. In particular, diffuse irradiance values are more realistic. Taub and Taud values can be entered as described below.
- **3-Zhang Huang**, developed for initial use in modelling typical meteorological years for China weather sites, this model seems to be good for other locations as well. Using total cloud cover, dry-bulb temperature, relative humidity, and wind speed as the independent variables, the total (global horizontal) solar radiation is estimated through a simple equation. This is the same model used to estimate the global solar radiation value when it is absent from the source data in the Weather Converter. Calculations use a [Clearness](#) value as part of their calculations.
- **4-Schedule**, allows you to enter custom Beam solar day and Diffuse solar day scheduled values for the day's profile (below).

Beam solar day schedule

Select your own custom day profile of beam solar values (Wh/m²) when the [Solar model indicator](#) is selected as **4-Schedule**. These values will replace the calculated values during design day processing. Only [Day schedules](#) can be selected here.

Diffuse solar day schedule

Select your own custom day profile of diffuse solar values (Wh/m²) when the [Solar model indicator](#) is selected as **4-Schedule**. These values will replace the calculated values during design day processing. Only [Day schedules](#) can be selected here.

ASHRAE clear sky optical depth for beam irradiance (Taub)

The Optical depth for beam radiation must be entered when the [Solar model indicator](#) is selected as **2-ASHRAE Tau**. Taub and Taud values are tabulated by month for 5564 locations worldwide on the CD that accompanies the ASHRAE 2009 HOF and in ASHRAE_2013_Monthly_DesignConditions.csv that is installed with EnergyPlus.

ASHRAE clear sky optical depth for diffuse irradiance (Taud)

The Optical depth for diffuse radiation must be entered when the [Solar model indicator](#) is selected as **2-ASHRAE Tau**. Taub and Taud values are tabulated by month for 5564 locations worldwide on the CD that accompanies the ASHRAE 2009 HOF and in ASHRAE_2013_Monthly_DesignConditions.csv that is installed with EnergyPlus.

Sky clearness

The sky “clearness” index for the summer design day and must be entered when the [Solar model indicator](#) is selected as **1-ASHRAE clear sky** or **3-Zhang Huang**. Along with the solar position as defined by the Location information and the date, this value helps define the solar radiation for each hour of the design day. Clearness may range from 0.0 to 1.2, where 1.0 represents a clear sky at sea level. Values greater than 1.0 may be used for high altitude locations.

Note: This “sky clearness” does not have the same meaning as the EnergyPlus output variable “Site Daylighting Model Sky Clearness”.

Weather Data Modifiers

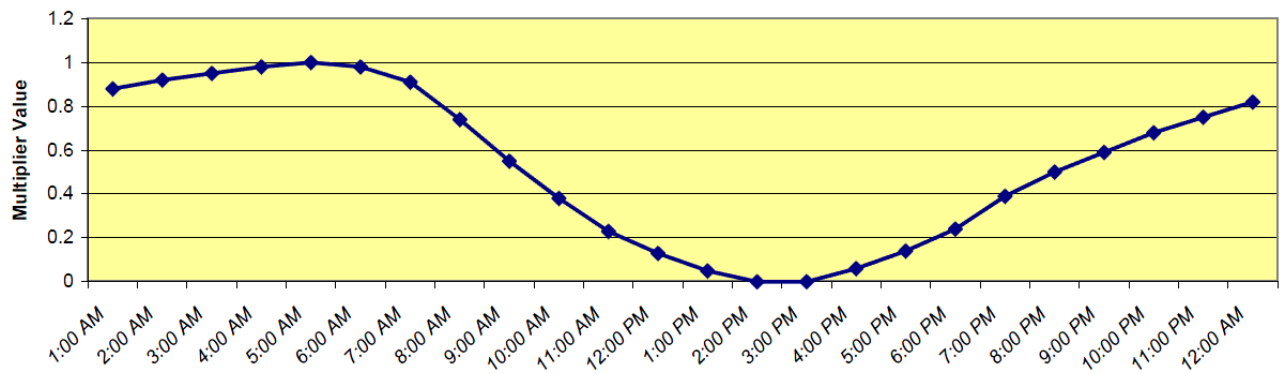
The daily temperature profile used in the cooling design calculations is calculated from the maximum and minimum values using a modified sinusoidal curve and assuming that the maximum temperature occurs at 15:00 and the minimum at 5:00. You can change this behaviour by using the data described below.

Dry-bulb temperature range modifier type

DesignBuilder provides some options to give more control over the dry-bulb temperatures used for the design days:

- **1-Default multipliers** - where EnergyPlus automatically generates the outside dry-bulb temperature profile based on the data provided in ASHRAE 2009 HOF.
- **2-Multiplier schedule** - where you can define a day schedule that specifies a multiplier applied to the daily temperature range to create the proper dry-bulb temperature range profile for your design day. This works as described below.
- **3-Temperature profiles** - allows you to use a Dry-bulb temperature profile schedule to define the dry-bulb temperature for the design day.

Important note: When using multiple summer design months these options are applied identically for each month. They are therefore generally of more use for single design months, i.e. when **1-Single design month** is selected as the [Design temperature period](#).



When using the **1-Default multipliers** option, the multipliers are taken from the ASHRAE 2009 HOF. More specifically, EnergyPlus creates an air temperature for each timestep by using the entered maximum dry-bulb temperature in conjunction with the entered daily range and the above multiplier values. The actual equation used is shown below:

$$T_{\text{current}} = T_{\text{max}} - T_{\text{range}} \times T_{\text{multiplier}}$$

where:

T_{current} = Air temperature of current hour of day
 T_{Max} = User supplied Max Dry-bulb Temperature
 T_{range} = The daily temperature range ($T_{\text{max}} - T_{\text{min}}$)
 $T_{\text{Multiplier}}$ = Range multiplier as shown on the above graph

The range multiplier values represent typical conditions of diurnal temperatures (i.e. the low temperature for the day occurring about 5:00 AM and the maximum temperature for the day occurring about 15:00. Note that EnergyPlus does not shift the profile based on the time of solar noon as is optionally allowed in ASHRAE procedures.

ASHRAE research indicates that dry-bulb and wet-bulb temperatures typically follow the same profile, so EnergyPlus can use the default profile to generate humidity conditions.

Dry-bulb temperature range modifier schedule

If you would like to override the default ASHRAE profile you can select the **2-Multiplier schedule** option for the [Dry-bulb temperature range modifier type](#) setting. This shows allows a Day schedule to be selected to define the range multipliers for each time of the cooling design day.

Dry-bulb temperature profile schedule

Select the Day schedule to be used to represent the dry-bulb temperature for the design day.

Note: The temperature values in the schedule must be defined in °C even when SI units are selected.

Humidity condition type

This option allows the method to be used to define humidity to be defined. It is only made available when the Dry-bulb temperature range modifier type is set to **3-Temperature profiles**. Select from:

- **1-Wet-bulb**, the default option where a wet-bulb temperature at the time of maximum temperature is used. The as defined using the coincident wet-bulb temperature further down the data sheet.
- **2-Relative humidity**, where the hourly variation of relative humidity is defined using the [Humidity condition schedule](#).

Humidity condition schedule

Select the Day schedule which defines the relative humidity in %.

Wind Data

Wind speed

This is the wind speed to be used for calculating the exterior convective heat transfer coefficients and for use (only in Calculated natural ventilation autosizing simulations) for calculating the exterior wind pressure along with the surface [wind press coefficients](#).

Wind direction

Used only for only in autosizing simulations of Calculated natural ventilation models for calculating the exterior wind pressure along with the surface [wind press coefficients](#).

Sizing Period

When running simulations, some design features may benefit by using a longer design weather period for HVAC autosizing or loads calculations. Longer design periods may be set up by allowing the hourly weather file to be used in the sizing calculations.

Autosize method

Choose the design HVAC autosizing method to use for simulations from the following options:

- **1-Design day** - the default option where data for one or more summer design days is used based on the settings described above.
- **2-Weather file days** where you can select the period to be used from the weather file for sizing.
- **3-Weather file condition type** where you can select from some standard per-defined types of period such as **1-Summer extreme**.

When **2-Weather file days** is selected, enter the From **Start day** and **Month** and the To **Start day** and **Month** as shown in the screenshot below.

Sizing Period	
Autosize method	2-Weather file days
From	
Start day (°)	17
Start month	Aug
To	
End day (°)	23
End month	Aug

Period selection

When the **3-Weather file condition type** Autosize method is selected you must choose the Weather file condition type from the following list.

- **1-Summer extreme** (default setting)
- **2-Summer typical**
- **3-Winter extreme**
- **4-Winter typical**
- **5-Autumn typical**
- **6-Spring typical**
- **7-Wet season**
- **8-Dry season**
- **9-No wet season**
- **10-No dry season**
- **11-Tropical hot**
- **12-Tropical cold**

Note: Not all weather files have all of the valid choices so this option is not completely generic. For example there may be extreme cold periods in some weather files but extreme wet periods (tropical) in others.

Design Temperature Period

The design month for cooling design and autosizing simulations is not always clear from the outset and indeed is often different for different parts of the same building. You can therefore select up to 12 design months to be used in cooling design and autosizing simulations.

You can choose to enter design data for either 1 or multiple months:

- **1-Single design month** where a single set of worst case design data is used for cooling design and autosizing simulations. In this case the design data is typically for the month of July in the Northern hemisphere and January in the Southern hemisphere.
- **2-Multiple design months** where design data for more than one month is used for cooling design and autosizing simulations. In this case the range of months to be used are selected on either the Cooling design calculation options or Model options dialog and the design data to be used for each month is specified here on the Location tab.

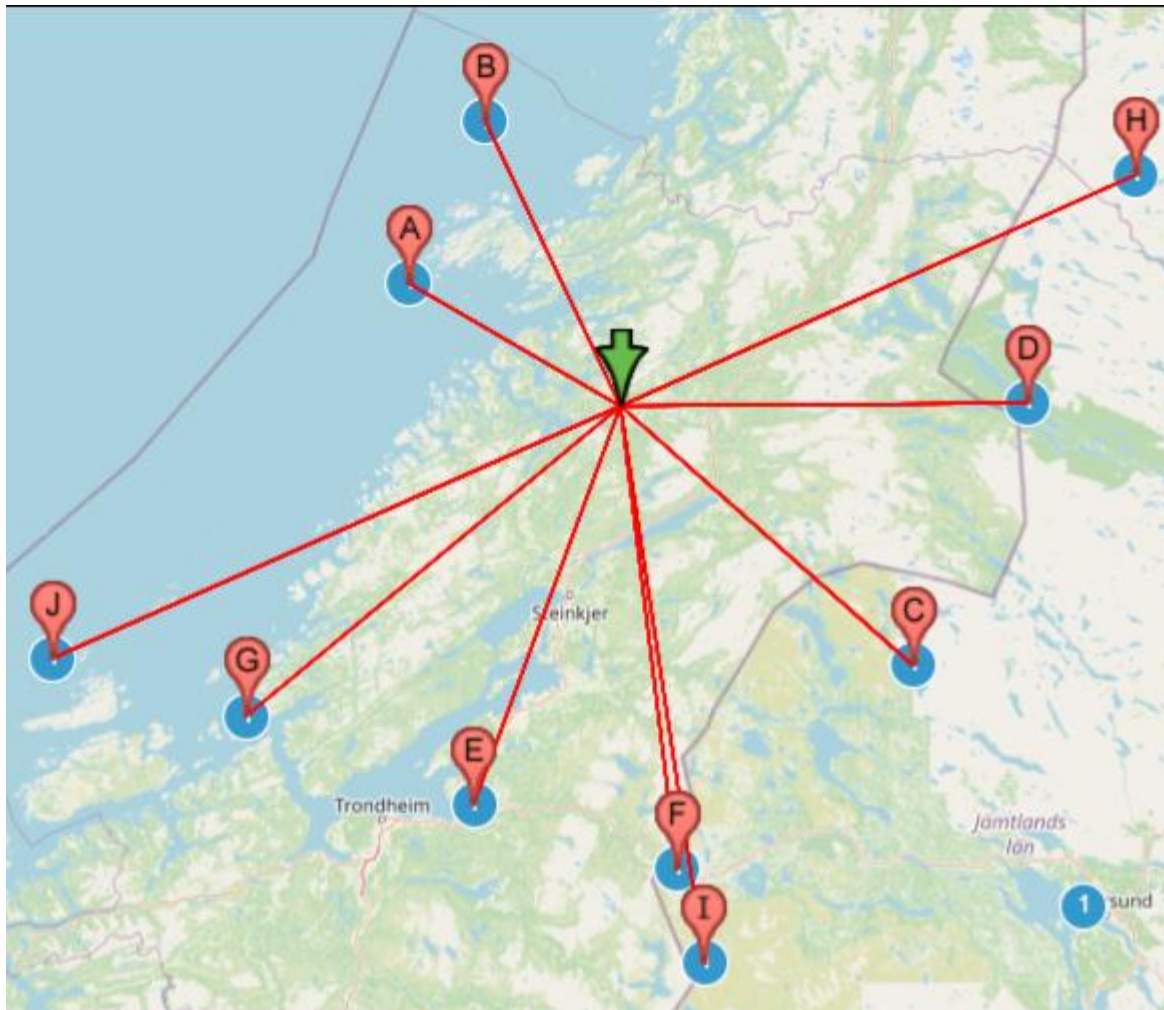
Tip: The **2-Multiple design months** option is the safest choice as the peak cooling month may vary across the building depending on the amount of glazing and shading on each façade. For example, for buildings in the Northern Hemisphere, large areas of glazing on south-facing façades may cause the peak design month to be delayed until August or even September when the sun is lower in the sky.

Note: The details of the design day(s): day of the month, the month range and day of week to be used for the Cooling design analysis can be defined on the Model options and [Cooling design](#) dialogs.

ASHRAE Summer Design Data

DesignBuilder includes a wide range of design data suitable for use with both the multiple and single design month options for over 6,400 locations around the world. The data supplied is all based on the ASHRAE 2013 design data set and is licensed from ASHRAE. It is included in the [Location templates](#) database so the appropriate data is loaded to the model when you select the location of your project.

If your precise project location is not included in the ASHRAE database, or if you need to access design data from more recent ASHRAE handbooks, there is a free utility at ashrae-meteo.info/v2.0 which identifies the nearest stations to your project location with distances and provides design data for each station.



Yearly Design Temperatures

When the Design temperature period is selected as **1-Single design month** the following data can be entered for a single month.

Dry-bulb and mean coincident wet-bulb data

- **Max dry-bulb temperature** - the maximum dry bulb air temperature over the day.
- **Coincident wet-bulb temperature** - the wet-bulb temperature at the time of the maximum dry-bulb temperature.
- **Min dry-bulb temperature** - the minimum dry bulb air temperature (night time). The template values have been calculated by DesignBuilder by subtracting the 0.4% Monthly Design Dry Bulb Temperature with the Mean Daily Dry Bulb Temperature Range Coincident with 5% Design Dry Bulb Temperature.

Wet-bulb and mean coincident dry-bulb data

- **Max wet-bulb temperature** - the maximum wet-bulb air temperature over the day.
- **Coincident dry-bulb temperature** - the dry-bulb temperature at the time of the maximum wet-bulb temperature.
- **Min dry-bulb temperature** - the minimum dry-bulb air temperature (night time). The template values have been calculated by DesignBuilder by subtracting the 0.4% Monthly Design Dry Bulb Temperature with the Mean Daily Dry Bulb Temperature Range Coincident with 5% Design Dry Bulb Temperature.

Monthly Design Temperatures

When the Design temperature period is selected as **2-Multiple design months** then the following data can be entered for 12 months.

Dry-bulb and mean coincident wet-bulb data

- **Monthly Design Dry-Bulb Temperatures** - the maximum dry bulb air temperatures over the design day for each month.
- **Monthly Coincident Wet-Bulb Temperatures** - the wet-bulb temperatures at the time of the maximum dry-bulb temperature.
- **Monthly Minimum Dry-Bulb Temperatures** - the minimum dry bulb air temperature (night time).

Wet-bulb and mean coincident dry-bulb data

- **Monthly Design Wet-Bulb Temperatures** - the maximum wet-bulb air temperatures over the design day for each month.
- **Monthly Coincident Dry-Bulb Temperatures** - the dry-bulb temperatures at the time of the maximum wet-bulb temperature.
- **Monthly Minimum Wet-Bulb Temperatures** - the minimum wet-bulb air temperature (night time).

Legislative Region And Insulation Standards

Region tab on model data at [Site level](#)

The data on the tab is related to the [Legislative Region](#) and associated insulation standards of the site.

Legislative region

A Legislative region in DesignBuilder is typically a whole country or a region, province or state within a country which has its own building control powers. The legislative region contains information on:

- Local energy codes and building regulations
- [Fuel emission factors](#) for a range of fuels.

Include Fuel Emissions

You can optionally request fuel emissions output data from EnergyPlus simulations for each fuel individually. Outputs are requested individually because a large number of pollutants are involved for each fuel. When this data is requested the pollution caused by each fuel type is reported in the eso file and a summary is provided in the Summary output file.

Include Fuel Emissions	
☒	Electricity
☒	Natural gas
☐	Fuel oil 1
☐	Fuel oil 2
☐	Coal
☐	Gasoline
☐	Propane
☐	Diesel
☒	Other fuel 1
☐	Other fuel 2

The fuel emission factors used in EnergyPlus' emission calculations comes from [Emissions factor](#) selection on the [Emissions](#) tab of the [Legislative regions template dialog](#). Inputs are only included in EnergyPlus input data for the fuels selected here.

Energy Codes/insulation Standards

[Region tab on model data at Site level](#)

You can select the Mandatory energy code in force as well as three additional levels of reference insulation standards to represent buildings with 'Uninsulated', 'Stock average' and 'Best practice' construction standards.

This data is used:

- To identify the mandatory energy code in force in the region. When one of the ASHRAE 90.1 codes are selected this has a significant knock on impact on the rest of the model as described in the [ASHRAE 90.1 Modelling](#) section.
- To set the insulation levels used when the 'Pre-design' Construction and glazing Model option is selected.
- When calculating asset ratings.
- Defines the [Fuel emissions factors](#) for the region.

Note: If you change any of this data, the insulation standards of the constructions in the <System> category will be automatically updated.

Building Model Data

Building Model Data can be entered at building, block, zone and surface levels (but not at site level). The set of data shown will depend on the level in the hierarchy (building, block etc) and on the Model options selected.

Building Model Data is laid out on the tabs:

- [Activity](#)
- [Construction](#)
- [Openings](#)
- [Lighting](#)
- [HVAC](#)
- [On Site Electricity Generation Data](#)
- [Economics](#)
- [Options](#)
- [CFD](#)
- [Detailed Equipment](#)

Each of these categories of data has a template selection at the top of the tab allowing rapid loading of the data from template.

Model data colour

Default data is [inherited](#) from the level above in the hierarchy and is shown in [blue](#), while 'user data', i.e. data you have explicitly set (or loaded from template) is shown in [red](#). You can clear user data back to its default state so it inherits from the level above using the [Clear data to default](#) command.

See also [Component](#) and [Template list item colours codes](#)

Tip: Use DesignBuilder defaults wisely! Take care to enter the minimum amount of data to get the most from the inheritance system. For example if all zones in a particular block have the same activity called say 'Office work', you should set the activity at block level and not multiple times at zone level. Equally if all the blocks in the building have the same activity, you should set the data at the building level and allow the blocks to inherit the data from the building. By keeping the amount of user data to a minimum, you will find it quicker to make changes at a later date. Clearly it is much faster to change a few Model Data items at building and block level than it is to make multiple changes at zone or surface levels.

See also section on [Model Data Hierarchy & Data Inheritance](#).

Activity Model Data

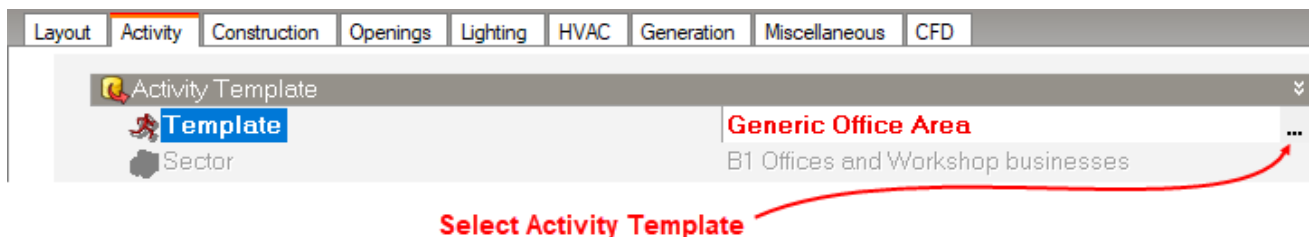
Activity tab in model data

See also the [Activity data templates and time schedules](#) tutorial

Data on the Activity tab allows you to define the activity (usage) of the zones including information on:

- [Zone Type \(block and zone level only\)](#)
- [Zone Multiplier](#)
- [Include zone](#) [Ref-1310383514](#)
- [Floor Areas and Volumes](#)
- [Occupancy](#)
- "Contaminant Generation and Removal" on page 336
- [Holidays \(building level only\)](#)
- [Environmental control](#)
- [Domestic Hot Water \(DHW\)](#)
- [Computer gains](#)
- [Office equipment gains](#)
- [Miscellaneous gains](#)
- [Catering gains](#)
- [Process gains](#)

You can load generic activity data by clicking on the Template browse control under the Activity Template header and then by clicking on the '...' at the right of the line (below).



When you make a selection from the list of [Activity templates](#), data from selected template will be loaded to the model.

Alternatively you can use the [Load data from template](#) tool.

Note that the appropriate [Sector](#) is loaded with the activity (screenshot above). The Sector is for information purposes only and does not affect the simulations.

All Activity data (apart from Holidays) is used to generate simulation data at the **Zone level**. Holiday data is used at **Building level**.

See also the [Activity data templates and time schedules](#) tutorial

Zone Type

[Activity tab](#) in model data under Activity Template header

The zone type can be set at block or zone level on the Activity tab. The options depend on whether you are working on an ASHRAE 90.1 model or not. **If it is not an ASHRAE 90.1 model** then the Zone type can be one of:

- **1-Standard** - the zone is within the main envelope and is usually (but not necessarily) occupied and heated and/or cooled. This zone type also covers zones within the main building envelope that are unoccupied and not directly heated or cooled such as offices that are temporarily unoccupied may come into this category.
- **2-Semi-exterior unconditioned** - the zone is unoccupied and is neither heated nor cooled and lies outside the main building envelope. Examples of unconditioned spaces are roofspaces, sunspaces, crawlspaces, car parks etc.
- **3-Cavity** - the zone is a narrow sealed vertical cavity such as the glazed cavity within a double façade or a Trombe wall. This algorithm calculates the convection coefficients for such a cavity based on the ISO 15099 standard.
- **4-Plenum** - the zone is a plenum which is unoccupied and has no heating, cooling, or mechanical ventilation (though of course air can flow through it to meet the needs of the zones it serves).

Otherwise if it is an **ASHRAE 90.1 model** then the following options are available.

- **1-Conditioned** - the zone is within the main envelope and is usually (but not necessarily) occupied and heated and/or cooled. This zone type also covers zones within the main building envelope that are unoccupied and not directly heated or cooled such as offices that are temporarily unoccupied may come into this category.
- **2-Unconditioned** - the zone is unoccupied and is neither heated nor cooled and lies outside the main building envelope. Examples of unconditioned spaces are roofspaces, sunspaces, crawlspaces, car parks etc. Note: that setting a zone to have 2-Unconditioned zone type does not *ensure* that it has no heating and cooling. You must ensure that through the HVAC system definition.
- **3-Cavity** - the zone is a narrow sealed vertical cavity such as the glazed cavity within a double façade or a Trombe wall. This algorithm calculates the convection coefficients for such a cavity based on the ISO 15099 standard.
- **4-Plenum** - the zone is a plenum which is unoccupied and has no heating, cooling, or mechanical ventilation (though of course air can flow through it to meet the needs of the zones it serves).
- **5-Semiheated** - the zone is heated to a low level only. **This option is only available for ASHRAE 90.1 models.** In exceptional cases where an ASHRAE 90.1 model has been converted to non-ASHRAE 90.1 model, any zones with the **5-Semiheated** zone type are treated as if **1-Standard** had been selected.

Zone types are also described in the [Partitioning blocks to create zones](#) tutorial

Note: Zones having one of the **2-Semi-exterior unconditioned**, **3-Cavity** and **4-Plenum** zone types do not have their floor area included in the building total floor area in the summary reports.

2-Semi-exterior unconditioned

When the Zone type is set to **2-Semi-exterior unconditioned**, the program asks you 'Do you want to load default unconditioned data including Activity, Lighting and HVAC templates?' If you reply 'Yes' then the zone is set as unoccupied by loading **<None>** Activity, HVAC and Lighting template data.

Also [Semi-exposed constructions](#) will be used instead of standard partitions for adjacencies with 'Unconditioned' zones. For example the wall between an occupied office and a car park would use the Semi-exposed construction, whereas the wall between two occupied offices would use Internal partition construction.

2-Unconditioned

When the Zone type is set to **2-Unconditioned**

3-Cavity

When the Zone type is set to **3-Cavity**, the program asks you 'Do you want to load default 'cavity' data including appropriate convection coefficients, calculation options and Activity, HVAC and Construction templates'. If you reply 'Yes' then the following changes are made:

- The zone is set as unoccupied by loading **<None>** Activity, HVAC and Lighting template data.
- Suspended floors and ceilings are switched off.
- The **5-Cavity** [Inside convection algorithm](#) is set for Cooling design, Heating design and simulation calculations.
- The **3-Full interior and exterior** [Solar distribution algorithm](#) is set for Cooling design and simulation calculations.

You can find out more about [Trombe wall simulations](#) in the [Modelling Issues](#) section.

4-Plenum

When the Zone type is set to **4-Plenum**, the program asks you 'Do you want to load default plenum data including appropriate convection coefficients, calculation options and Activity and HVAC templates?'. If you reply 'Yes' then the following changes are made:

- The zone is set as unoccupied by loading **<None>** Activity, HVAC and Lighting template data.
- Suspended floors and ceilings are switched off.
- The **4-Ceiling diffuser** [Inside convection algorithm](#) is set for Cooling design, Heating design and simulation calculations.

You can find out more about modelling plenums under HVAC.

Unconditioned zone type (ASHRAE 90.1 only)

In ASHRAE 90.1 models, when the **2-Unconditioned** Zone type is selected you must define whether the unconditioned zone has normal levels of airtightness, or it is outside the main envelope. The options are:

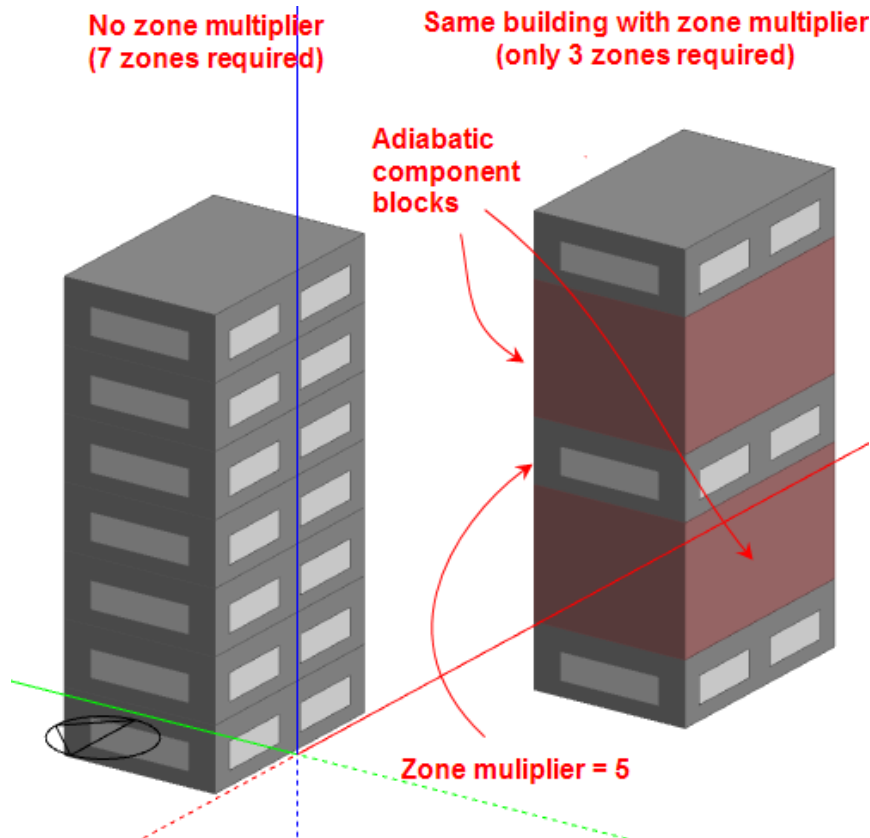
- **1-General** for unconditioned zones having normal levels of airtightness.
- **2-Ventilated crawlspace/attic etc** for zones outside the main envelope such as attics, crawlspaces and car parking which are unsealed and have no provision for airtightness.

Note: Zones having one of the **2-Unconditioned** zone types and **2-Ventilated crawlspace/attic etc** Unconditioned zone ventilation do not have their floor area included in the building total floor area in the summary reports.

Zone Multiplier

[Activity tab](#) in model data under [Activity Template](#) header

The Zone multiplier data allows you to reduce the size of your model in cases where there are similar zones by specifying that certain zones are repeated and so only need to be simulated once. A typical use is for multi-storey buildings with identical (or very similar) floors, for example, a hotel where there is a standard room layout on each floor that is repeated many times. The image below is taken from the [Zone Multiplier Example.dsb](#) file provided with the software and illustrates how the zone multiplier mechanism works.



The basic 7 storey building on the left above doesn't use the zone multiplier and, since each storey has one zone, the whole building contains 7 zones. The equivalent building on the right has been simplified by using the zone multiplier. The single mid-storey zone has a zone multiplier of 5 which means that during the simulation the zone is treated as if there were 5 of them. In particular the energy consumption and HVAC airflow rates are multiplied by 5 so the results for the reduced building match approximately with that of the non-reduced model.

The top floor and ground floor zones cannot be combined using the zone multiplier because they have roof and ground adjacencies respectively and so must be retained in the reduced model. However the 5 mid-floor zones all have the same operation and adjacencies and so can be combined.

Note: The [Zone multiplier](#) data is only used at zone level to multiply up zones. The settings at building and block level are there only to allow zone defaults to be set for multiplying zones quickly and easily, they do not multiply blocks or buildings.

See also the [Activity data templates and time schedules](#) tutorial.

Note: The **Zone multiplier** is not currently used in Daylighting calculations and reporting.

Adiabatic Component Blocks

Note the use of the [adiabatic component blocks](#). These have various functions:

- Set the adjacency to be adiabatic for the underside of the top floor zone, the top and underside of the mid-floor zone and the top of the ground floor zone.
- Help to position the mid-floor representative zone in a reasonable representative position to ensure that a good average wind speed will be applied.
- Ensures correct shading in a more complex model. Shadows will be correctly cast by 'missing zones' and also shading received by the representative mid-floor zone will be representative of the shading received by all of the actual mid-floor zones.
- Improves the look of the reduced model to match that of the full model.

For other ways to simplify models see [Simplification](#) on the Advanced tab of the Model options.

Include Zone

[Activity tab](#) in model data under **Activity Template header**

Include zone in thermal calculations

By default all zones are included in simulations and heating and cooling design calculations, but some zones can be excluded through use of this checkbox if necessary. For example, you may wish to exclude a zone or series of zones from the analysis to focus on a smaller part of the building such as a single shop within a shopping mall, and in this case you should uncheck this option on the Activity tab for zones to be excluded as appropriate.

During the simulation:

- Surfaces between included and excluded zones are modelled as being **adiabatic**, i.e. heat is transferred into the wall mass as if the temperature of the excluded zone is the same as that of the included zone.
- Depending on the setting of the [Include shading from excluded zones in simulation](#) model option, surfaces from excluded zones are either represented simply as shading surfaces to ensure that their impact on the solar gains and daylighting in the included zone(s) is still accounted for, or they are ignored altogether.

Note: In Calculated natural ventilation models you should be careful when excluding zones because they could play an important role in the transfer of airflow from one part of the building to another and so could have a significant effect on the natural ventilation in included zones. For example an excluded zone could block the airflow from one included zone to another. Equally, by removing large parts of the model, you run a risk of altering the stack flow regime, neutral pressure level etc in the building airflow model.

Tip: If you need to focus on a just few zones in a large model it is usually easiest to uncheck this option at building level and include only the zone(s) of interest. Another quick way to change the include/exclude status of multiple zones in one session is to use the [Model data grid edit](#) tool.

See also the [Activity data templates and time schedules](#) tutorial

Include zone in Radiance daylighting calculations

Zones which are not regularly occupied, or which for any other reason should not be included, can be excluded from daylighting calculations by unchecking this option. This option can be used when carrying out a LEED, BREEAM or Green Star daylighting credit calculation to avoid including zones that are not regularly occupied in the building assessment.

From the LEED v3 documentation:

Regularly occupied spaces in commercial buildings are areas where people sit or stand as they work. In residential applications these spaces include all living and family rooms and exclude bathrooms, closets, or other storage or utility areas. In schools, they are areas where students, teachers, or administrators are seated or standing as they work or study.

Floor Areas And Volumes

[Activity tab](#) in model data at zone and building levels only

When viewed at zone level the data on this tab refers to the current zone and when viewed at building level the data refers to the whole building. With one unusual [exception](#), all data under this heading is read-only.

Building Level

Building rotation

This is the rotation of the building relative to the site in degrees. It is available at building level only and will be zero in most cases.

Occupied floor area

The floor area of occupied zones (in m2 or ft2). This is the sum of the floor areas for zones having [1-Standard Zone type](#).

Occupied volume

The volume of occupied zones (in m3 or ft3). This is the sum of the zone volumes for zones having [1-Standard Zone type](#).

Unoccupied floor area

The floor area of unoccupied zones (in m2 or ft2). This is the sum of the floor areas for zones *not* having [1-Standard Zone type](#).

Unoccupied volume

The volume of unoccupied zones (in m3 or ft3). This is the sum of the volume of zones *not* having [1-Standard Zone type](#).

Note: The zone floor areas are calculated based on the selected [Zone floor area calculation method](#). They are not necessarily simply the sum of the floor surface areas listed in the Navigator. Likewise zone volumes are based on the [Zone volume calculation method](#).

Tip: To ensure you are viewing the most up to date building areas and volumes you can refresh the navigator using the rightmost icon at the top of the navigator panel.

Zone Level

Floor area

The floor area of the zone (in m2 or ft2). It cannot be edited.

Zone volume

The volume of the zone (in m3 or ft3). For normal closed manifold zones the zone volume is calculated from the zone geometry based on the [Zone volume calculation method](#). For [open manifold](#) zones the zone volume data comes initially from the BIM software but it can be overwritten here if required.

Inner surface mode

The Inner surface mode is a read-only value, visible at zone level only, indicating how the zone inner surfaces, and hence the zone floor area and volume, have been derived. It can have one of 4 values:

- **Deflation**, where finite thicknesses have been defined for at least one element surrounding the zone and the inner volume has been successfully created by deflating the parent zone. Volumes and surface areas are obtained from the inner zone geometry. This is the most accurate method.
- **Approximate**, the same as Deflation above but the inner volume fails to be created for one of a number of technical reasons (vertices having more than three surfaces associated with them etc.). In this case zone volume and surface areas are calculated using approximate methods.
- **Coincident** where all zone surfaces have zero thickness and so the inner volume coincides with the parent zone. In this case volumes and surface areas are obtained from the outer zone geometry.
- **Open manifold** for zones in [open manifold](#) blocks loaded from gbXML. In this case all zone surfaces have zero thickness and the surface areas are calculated from the parent zone surfaces. Because the zones are open-manifold their volumes cannot be calculated from the surface geometry and instead are read directly from the gbXML file and stored with the zone data. Open-manifold **Zone volume** settings can be overwritten by the user if required as described above.

Occupancy

[Activity tab](#) in model data

The Occupancy model data defines the number people in the space and the times of occupancy. This data is used in Simulations and Cooling design calculations together with the Metabolic heat output of the occupants and [Holidays](#) to calculate the heat input to the building.

Occupancy method

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for occupancy here on the Activity tab. Choose from the options:

- **1-People/area** - the default option where occupancy is defined as occupancy density (in people / m2 or people / ft2).
- **2-Area per person** - is sometimes preferred as an inverse occupancy density. Note that when switching between this option and occupancy density the definitions on the Activity tab zone level are translated accordingly. This is not the case when the number of people option below is selected.
- **3-Number of people** - used when detailed zone by zone survey or design data is available. Unless you need to enter specific known occupancy gains zone by zone, this option is best avoided as it increases the amount of work involved

Note: When using the **3-Number of people** option, the number of people in each room is entered zone by zone as values inherited from building and block levels will typically not be correct.

Depending on the Occupancy method selected either on model options or above, the occupancy gains can be defined in one of 3 ways, as **Occupancy density**, **Floor area per person** or **Number of people** as described below.

Occupancy density

When the [Occupancy method](#) is set to **1-Occupancy density** you can set the number of people per unit floor area. It is usually most convenient to load this data from the Activity template.

Floor area per person

When the [Occupancy method](#) is set to **2-Area per person** you can set the floor area per person, i.e. the reciprocal of the occupancy density above. You must check the **Occupied** checkbox first.

Number of people

When the [Occupancy method](#) is set to **3-Number of people** you can enter the number of people in the zone directly. In this case note that the value entered at block and building level does not normally inherit very effectively, especially in buildings with zones of very different floor areas. In this case enter the number of people for the most common type of zone at building level. Unless you need to enter specific numbers of people in each zone this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Radiant fraction

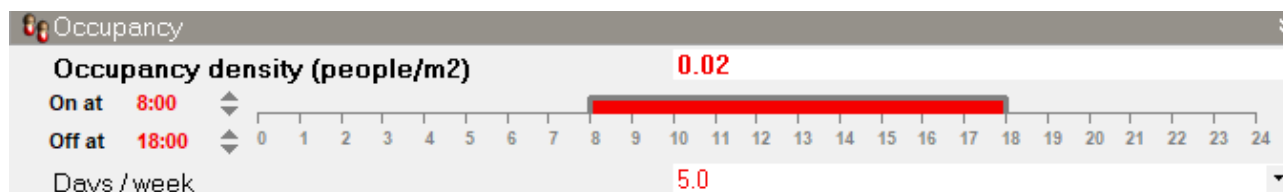
The radiant fraction of occupant heat emissions is hard set to 0.3.

Timing

The way that Occupancy is scheduled depends on the [Timing](#) model data option as described below.

Typical workday timing model option

When the [Typical workday](#) Timing model option is set, the occupancy times are controlled by setting **On** and **Off** values and the number of days in the working week.



In above example there are 0.02 people per m2 between 8am and 6pm and there are no people outside these times and at weekends.

Note: The Occupancy **Days / week** is set at the building level only and is used for all other Typical workday schedules in the model apart from Mechanical ventilation, and Heating and Cooling operation which have their own Days / week data. Valid values are 1-7.

Schedule timing model option

When the Schedule [Timing](#) model option is set, the occupancy times are controlled by a [Schedule](#). The metabolic gains for the zone are multiplied by the value of the Schedule at each time step in the simulation so you can define periods of part-occupancy using Schedule values less than 1. Use values of 0 to indicate that the space is unoccupied.

Occupancy	
Occupancy density (people/m2)	0.0200
Schedule	TM59_Kitchen_Occ

In above example number of people per m2 is calculated as $0.02 \times \text{Schedule value}$.

Occupancy latent gains

When the [Gain definition level](#) option is set to **2-Zone by zone in model data**, select the Occupancy latent gains method. There are 2 ways for the sensible-latent split of occupancy gains to be defined:

- **1-Dynamic calculation** - where the latent fraction of occupancy gains is calculated within EnergyPlus at runtime based on internal temperatures and metabolic rate. In this case the ratio of sensible to total occupancy gain will reduce as the internal temperature in the zone increases and at temperatures much above 30°C all of the occupancy gain will be latent (occupants sweating).
- **2-Fixed fraction** - where the latent fraction of occupancy gains is constant regardless of internal conditions and defined on the Activity tab (below).

Note: The total occupancy gain (sensible + latent) is not affected by this option, only the split of sensible vs latent gain.

Latent fraction

Occupancy latent gains account for moisture from people in the building due to both perspiration (sweating) and from exhaled moisture. The way that latent gains are calculated depends on the [Occupancy latent gains](#) model option.

When this option is set to **2-Fixed fraction** then you can enter the latent fraction here. A typical default value is 0.5. Alternatively if this setting is left as the default **1-Dynamic calculation** option then the latent fraction is calculated within EnergyPlus at runtime based on internal temperatures and metabolic rate and no additional setting is required here.

Metabolic

Select the level of activity by selecting a [metabolic rate](#) component based on the level of activity of people using the space.

The metabolic rate determines the amount of heat gain per person in the zone *under design conditions*. This value is modified during simulations based on a correlation to account for variations in space temperature. *This should be borne in mind when analysing occupant sensible heat output in the Cooling design and Simulation results screens.* Metabolic rate has units Watts per person and represents the total heat gain per person including convective, radiant, and latent heat. An internal algorithm is used to determine what fraction of the total is sensible and what fraction is latent. Then, the sensible gain is divided into radiant and convective portions.

Metabolic rate data can be found in the ASHRAE Handbook of Fundamentals and in CIBSE Guide A.

DesignBuilder default metabolic data is derived from the ASHRAE values and is for adult males having typical surface area of 1.8m². For women multiply the adult male value by 0.85 and for children multiply by 0.75.

The **metabolic factor** accounts for people of various sizes. Enter 1.00 for men, 0.85 for women, 0.75 for children or you can use an average value if there is a mix of sizes.

Note: If the [Occupancy latent gains](#) model option is set to **Dynamic calculation** then the sensible/latent split for occupancy gains is affected by the internal temperature and relative humidity. With high internal temperatures, people cool themselves largely by evaporation (sweating) and sensible occupancy gains can be very low or even zero as the internal temperature approaches that of the human body.

Background

Heat is generated in the human body by oxidation at a rate called the metabolic rate. This heat is dissipated from the body surface and respiratory tract by a combination of radiation, convection and evaporation. The relative proportions of sensible and latent heat from people is a complex function of the metabolic rate and the environmental conditions.

EnergyPlus uses a polynomial function to divide the heat gain into sensible and latent portions. More details can be found in the EnergyPlus EngineeringDoc.pdf documentation.

CO2 generation rate

Enter the carbon dioxide generation rate per person in m3/s-W or (ft3/min)/(Btu/hr). The total carbon dioxide generation rate for the zone is:

Number of People * People Schedule * People Activity * Carbon Dioxide Generation Rate.

The default value is 3.82E-8 m3/s-W (obtained from ASHRAE Standard 62.1-2007 value at 0.0084 cfm/met/person over the general adult population). The maximum value can be 10 times the default value.

Clothing

Clothing schedule definition

You can define the clothing levels of the occupants using two Clothing schedule definition options:

- **1-Generic summer and winter clothing** where the level of clothing insulation is entered as 2 fixed values one for Summer and the other for Winter. Start and end days for each period are defined under the "Time and daylight saving" on page 314 header at site level.
- **2-Schedule based clothing** which allows application of a custom schedule to define level of clothing insulation

This data is used when generating [comfort output](#).

Clothing reduces the body's heat loss and is classified according to its insulation value. The unit normally used for measuring clothing's insulation is the Clo unit. The more technical unit is m²C/W is also used frequently (1 Clo = 0.155 m²C/W). The Clo value can be calculated by adding the Clo value of each individual garment. The insulation value for individual garments can be found in ISO 7730.

Clo-Values for Different Items of Clothing and Ensembles

Clothing	Clo-Value
Naked	0.0
Briefs	0.06
T-shirt	0.09
Bra and panties	0.05
Long underwear	
upper	0.35
lower	0.35
Shirt	
White, short sleeve	0.14
heavy, long sleeve	0.29
Add 5% for tie or turtleneck	

Skirt	0.22-0.70
Trousers	0.26-0.32
Sweater	0.20-0.37
Socks	0.04-0.10
Light summer outfit	0.3
Working clothes	0.8
Typical indoor winter clothing combination	1.0
Heavy business suit	1.5

Note 1: Clo-values are additive, so one can calculate the clo-value for a person wearing a T-shirt and light socks $(0.09 + 0.04) = 0.13$. (Adapted from ASHRAE Fundamentals and "Technical Review of Thermal Comfort," Bruel and Kjaer, No. 2, 1982.)

Note 2: For equatorial regions where there is no clear 'summer/winter' weather pattern you should generally use a **2-Schedule based clothing** definition and adjust the clothing schedule values.

Comfort Radiant Temperature Calculation

This field specifies the type of Mean Radiant Temperature (MRT) calculation the user wishes to use for the thermal comfort model. There are currently two options:

- **1-Zone averaged** where the standard MRT is used in comfort calculations.
- **2-Surface with largest window** which calculates the radiant temperature for a person close to the largest window in each zone. The MRT used in the thermal comfort calculations in this case is actually the average of the inside surface temperature of the largest window and the zone averaged MRT. The surface temperature alone is not used because the maximum view factor from a person to any flat surface is roughly 0.5. Actually the largest window surface gets slightly more weighting than 50% since the surface selected is still a part of the zone average MRT calculation. This simplification is made to avoid the need to specify view factors and the exact location of the person. If a zone has no windows then the **1-Zone averaged** option is used instead.

Note: This selection only affects comfort calculations and will not affect the [Radiant Temperature](#) reported by DesignBuilder which remains a zone-averaged value regardless.

Air Velocity

Air velocity schedule

The [thermal comfort calculations](#) carried out by EnergyPlus to obtain the [comfort outputs](#) require the velocity of the air close to the occupants .

You can select a schedule for each zone to define the time-varying velocity of the air in the space in m/s. The default schedule gives a constant air velocity of 0.137 m/s, but higher values might be appropriate for rooms with mixing fans or high volumes of HVAC air delivery, for example.

Note: For TM59 simulations you should set the air velocity to 0.1 m/s.

Contaminant Generation And Removal

[Activity tab](#) in model data

DesignBuilder offers 4 different ways to model contaminant generation and removal within each zone of the building. EnergyPlus allows only one type of contaminant to be modelled in a simulation (in addition to CO2 which is handled separately).

Note: The [Simulate contaminant concentration](#) site level option must be selected for any of the zone settings to take effect in the simulation and actually generate/remove contaminants.

Tip: The results of the contaminant simulation are not displayed in the built in results on the Simulation screen, but the zone air contaminant concentration for each zone can be viewed when using the [Results Viewer](#).

Contaminant generation/removal

Check this checkbox if you would like to model contaminant generation and/or removal for the current zone. When this option is checked the rest of the data described below becomes available for editing.

Contaminant simulation method

The contaminant simulation methods available for selection are:

- **1-Constant** where contaminant is generated/removed at a constant design rate multiplied by a time-varying schedule
- **2-Cutoff model** where the contaminant generation rate is based on the "cutoff concentration" model
- **3-Decay source** where the contaminant generation rate is based on the "decay source" model
- **4-Deposition rate sink** where the contaminant removal from a zone stops when the sink concentration level is higher than the zone air concentration level.

Different **Contaminant simulation method** settings can be made in each zone.

Constant

The **1-Constant** option allows you to specify the generic contaminant generation rate and removal rate coefficient in the zone. The associated fraction schedules allow the magnitude of sources and sinks to vary with time. The object is equivalent to the combination of the constant coefficient model and the burst source model defined in the sources and sinks element types of CONTAM 3.0.

The basic equation used to calculate generic contaminant source and sink for the constant model is given below:

$$S_f(t) = G_f(t) * F_G - R_f(t)C_f(t) * 1.0E - 6 * F_R$$

where:

S_f = Contaminant generic contaminant source strength [m3/s]
 G_f = Generic contaminant generation rate [m3/s]
 F_G = Fraction value from the source fraction schedule at a given time [dimensionless]
 R_f = Generic contaminant effective removal rate [m3/s]
 F_R = Fraction value from the sink fraction schedule at a given time [dimensionless]
 C_f = Generic contaminant concentration value at a given previous time step [ppm]

Design generation rate

Enter the full contaminant design generation rate (in m3/s or ft3/min). The design generation rate is the maximum *volume* of generic contaminant expected at design conditions. The design value is modified by the schedule fraction in the Generation schedule described below for each timestep in the simulation.

Note: When only the *mass* generation rate is available, the rate must be converted to a volume flow rate. In this case, use the mass flow rate divided by the vapour density of the generic contaminant.

Generation schedule

Select the schedule that modifies the maximum design generation rate (G_i). This fraction between 0.0 and 1.0 is noted as F_G in the above equation.

Design removal coefficient

This setting defines the full volumetric contaminant design removal coefficient (in m³/s or ft³/min). The design removal rate is the maximum amount of generic contaminant expected at design conditions times the generic contaminant concentration in the same zone. The design value is modified by the schedule fraction in the Removal schedule described below.

Removal schedule

Select the schedule that modifies the maximum design generation rate (R_i). This fraction between 0.0 and 1.0 is noted as F_R in the above equation.

Cutoff Model

The **2-Cutoff model** option allows the contaminant generation rate to be specified based on the cutoff concentration model. The basic equation used to calculate generic contaminant source for the pressure driven constant model is given below:

$$S_f(t) = \begin{cases} G_f(t) * F_G * \left(1 - \frac{C_f(t)}{C_{cutoff}}\right) & C_f < C_{cutoff} \\ 0 & C_f \geq C_{cutoff} \end{cases}$$

where:

S_f = Generic contaminant source strength [m³/s]

G_f = Generic contaminant generation rate [m³/s]

F_G = Fraction value from the source fraction schedule at a given time [dimensionless]

C_{cutoff} = Cutoff concentration at which emission ceases [ppm]

C_f = Generic contaminant concentration value at a given previous time step [ppm]

Maximum generation rate

Enter the maximum contaminant design generation rate (in m³/s or ft³/min). The design generation rate is the maximum amount of generic contaminant expected at design conditions. The design value is modified by the Generation schedule fraction (below).

Note: When the *mass* generation rate is available, the rate must be converted to a volume flow rate. Use the mass flow rate divided by the vapour density of the generic contaminant.

Generation schedule

Select the schedule that modifies the maximum design generation rate (G_i). This fraction between 0.0 and 1.0 is noted as F_G in the above equation.

Cutoff concentration

Enter the contaminant cutoff concentration level where the source ceases its emission in ppm.

Decay Source

The **3-Decay source** option allows you to specify the contaminant generation rate based on the decay source model. The basic equation used to calculate generic contaminant source for the decay source model is given below:

$$S_f(t) = G_f(t)F_G \exp(-t/t_c)$$

where:

S_f = Generic contaminant source strength [m3/s]

G_f = Initial generic contaminant generation rate [m3/s]

F_G = Fraction value from the source fraction schedule at a given time [dimensionless]

t = Time since the start of emission [second]

t_c = Decay time constant [second]

Initial emission rate

Enter the initial generic contaminant design emission rate (in m3/s in ft3/min). The generation is controlled by a schedule, as defined in the next field. Contaminant emission begins when the schedule changes from a zero to a non-zero value (between zero and one). The initial emission rate is equal to the schedule value times the initial generation rate.

Schedule

Select the schedule that modifies the maximum design emission rate (G_f). This fraction between 0.0 and 1.0 is noted as F_G in the above equation. When the value is equal to 1, the generation rate is used and time is reset to zero. When the value is equal to zero, the schedule value is ignored in the equation.

Decay time constant

The decay time constant (in s) the time at which the generation rate reaches 0.37 of the original rate.

Note: The variable t , time since the start of emission, will be reset to zero, when a new run period starts, or the generation schedule value is equal to zero.

Deposition Rate Sink

The **4-Deposition rate sink** option allows you to specify the generic contaminant removal rate from a zone. The object is equivalent to the deposition rate sink model defined in CONTAM 3.0 sources and sinks element types.

The deposition rate model provides for the input of a sinks' characteristic in the familiar term of deposition rate in a zone. The removal stops when the sink concentration level is higher than the zone air concentration level. The deposition rate model equation is:

$$S_f(t) = k_d V_z * C_f(t) * 1.0E - 6 * F_R m$$

Where:

$S_f(t)$ = Removal rate at time t [m3/s]

k_d = Deposition rate [1/T]

V_z = Zone volume [m3]

m = Element multiplier [dimensionless]

$C_f(t)$ = Concentration of generic contaminant at the previous time step [ppm]

F_R = Schedule or control signal value at time t [dimensionless]

Deposition Rate

This field specifies the deposition rate to the zone (in m3/s or ft3/min).

Schedule

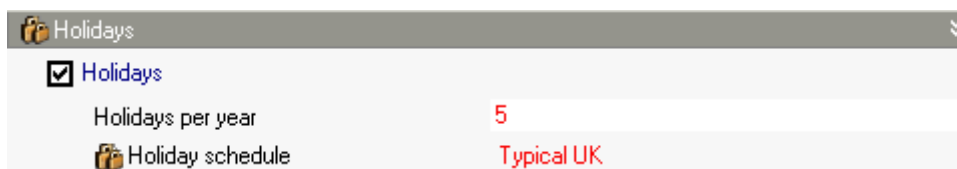
Select the schedule that modifies the maximum design removal rate (S_i). This fraction between 0.0 and 1.0 is noted as F_R in the above equation.

Holidays

[Activity tab](#) in model data (building level only)

You can define the dates of holidays to include the effect of building shutdown during certain periods of the year. Do this by selecting the Holidays option under the Holidays header on the Activity tab **at building level**. Holidays defined at building level in this way are used for the whole simulation. The holiday data is used for Simulations only (not Heating/Cooling Design Calculations) and allows you to set the [Holiday Schedule](#) and the number of days of holiday per year. The [possible holiday dates](#) are defined in the Holiday schedule and the actual holidays to be used in the simulation are selected from the top of the list based on the **Holidays per year** data entered.

Note that you must have at least as many days defined in the holiday schedules as you request here. If there are insufficient holidays in the holiday schedule you will receive an error message to this effect prior to a simulation.



With the data in the above screenshot there will be 5 holidays and the dates are given by the first 5 holiday dates set in the 'Typical UK' Holiday schedule.

See also the [Timings, schedules and profiles](#) tutorial

The way holiday definition works depends on the [Timing](#) model option:

Compact Schedules

When the [Timing](#) model option is Schedules you can use [Compact schedules](#) to define holiday dates and schedule values. The value of Compact schedule output during holidays can be defined explicitly using the Holidays day type following **For**:

For example:

```
SCHEDULE:COMPACT,  
Bedroom_Cool,  
Temperature,  
Through: 31 Dec,  
For: Weekdays SummerDesignDay WinterDesignDay,  
Until: 05:00, 0.5,  
Until: 09:00, 1,  
Until: 24:00, 0.5,  
For: Weekends,
```

```

Until: 05:00, 0.5,
Until: 24:00, 1,
For: Holidays AllOtherdays,
Until: 24:00, 0;

```

The Holiday data in this [Compact schedule](#) indicates that the cooling is switched off on holidays.

7/12 Schedules

Holiday operation is not defined explicitly and holidays use the same data as Sundays

Typical Workday

Holidays use the same data as Sundays

Environmental Control

[Activity tab](#) in model data

The heating and cooling setpoint temperatures, minimum fresh air per person and illuminance requirements are related to the Activity or Usage of the zone and are accessed by opening the Environmental Comfort group header.

Heating Setpoint Temperatures

Heating setpoint temperature

Defines the ideal temperature (i.e. the setting of the heating thermostat) in the space when heating is required. Its meaning depends on the [Temperature control](#) calculation option. When using **1-Air temperature** control the **Heating setpoint temperature** corresponds to the air temperature in the space and when it is set to set to **2-Operative temperature** it is the operative temperature $(MAT+MRT)/2$.

Heating setback setpoint temperature

Some buildings require a low level of heating during unoccupied periods to avoid condensation/frost damage or to prevent the building becoming too cold and to reduce peak heating requirements at startup. Enter the setpoint temperature to be used at night-time, weekends and other holidays during the heating season. Its meaning depends on the [Temperature control](#) calculation option (above).

Note: when using the **Typical workday** timing option, this setback temperature set point is only applied during unoccupied periods such as night-time and holidays during the heating season. It is not applied during the summer (if heating is switched off during the summer.)

Note: if the setback temperature set does not appear to affect results when using a [Compact schedule](#), this may be because the **Zone setpoint heating schedule** selected uses only 0 and 1, not the 0.5 required to access setback temperature. See [Defining setpoint temperature schedules](#) for more on this.

Cooling Setpoint Temperatures

Cooling setpoint temperature

Defines the ideal temperature (i.e. the setting of the cooling thermostat) in the space when cooling is required. Its meaning depends on the [Temperature control](#) calculation option. When using **1-Air temperature** control the **Cooling setpoint temperature** corresponds to the air temperature in the space and when it is set to set to **2-Operative temperature** it is the operative temperature $(MAT+MRT)/2$.

Note: the cooling setpoint temperature must be higher than the heating setpoint temperature.

Cooling setback setpoint temperature

Some buildings require a low level of cooling during unoccupied periods to prevent the building becoming too hot and to reduce the startup cooling load the next morning. Enter the setpoint temperature to be used at night-time, weekends and other holidays during the cooling season. Its meaning depends on the [Temperature control](#) calculation option (above).

Note: when using the **Typical workday** timing option, this setback temperature set point is only applied during unoccupied periods such as night-time and holidays during the cooling season. It is not applied during the winter (if cooling is switched off during the winter.)

Note: if the setback temperature set does not appear to affect results when using a [Compact schedule](#), this may be because the **Zone setpoint cooling schedule** selected uses only 0 and 1, not the 0.5 required to access setback temperature. See [Defining setpoint temperature schedules](#) for more on this.

Humidity Setpoint Temperatures

When using some Simple and Detailed HVAC systems you can control humidity. In the case of AHU-type systems the humidity in a control zone is controlled. The zone humidification and dehumidification setpoints can be defined here. Note that for systems without humidity control these setpoints are ignored.

RH Humidification setpoint

The zone humidistat relative humidity setpoint for humidification (0 to 100%).

RH Dehumidification setpoint

The zone humidistat relative humidity setpoint for dehumidification (0 to 100%).

Comfort PMV Setpoints

When using Detailed HVAC and the [Temperature control](#) option is set to **4-Fanger comfort** and the [Detailed HVAC Activity data](#) option is set to **1-Simple HVAC**, the zones are heated and cooled using the Fanger PMV heating and cooling comfort setpoints defined below as setpoints. When using the **2-Detailed HVAC** [Detailed HVAC Activity data](#) option, schedules of Fanger PMV comfort are defined on the [HVAC zone](#) dialog instead.

The [Thermal comfort calculator tool](#) can be used to derive PMV values based on the known environmental conditions.

Heating PMV setpoint

The PMV used as the heating setpoint when the heating operation schedule on the HVAC tab has value 1.

Heating PMV setback

The PMV used as the heating setback when the heating operation schedule on the HVAC tab has value 0.5.

Cooling PMV setpoint

The PMV used as the cooling setpoint when the cooling operation schedule on the HVAC tab has value 1.

Cooling PMV setback

The PMV used as the cooling setback when the cooling operation schedule on the HVAC tab has value 0.5.

Ventilation Setpoint Temperatures

Natural ventilation indoor min temperature control

If you wish to use natural ventilation to cool the building/block/zone down to a particular setpoint temperature and then shut off when the temperature goes below that setpoint then check this box. This allows you to define the **Natural ventilation cooling setpoint temperature** described below.

Natural ventilation min temperature control applies to both Calculated and Scheduled natural ventilation.

Tip: To ensure that natural ventilation occurs regardless of internal temperature (e.g. to use nat vent provide fresh air) then uncheck this option.

Min temperature definition

There are 2 ways to define the Natural ventilation min temperature:

- **1-By value** - use this option when the setpoint does not change with time in which case you can enter a fixed setpoint temperature.
- **2-By schedule** - use this option when the setpoint changes with time in which case you can select a schedule of time-varying temperature setpoints.

Natural ventilation min temperature

This is the fixed indoor temperature (in °C or °F) below which ventilation is shut off. The control is visible when the **1-By value** option is selected for **Min temperature definition**. It can be thought of as the cooling setpoint temperature which controls the activation of natural ventilation. If the inside air temperature is greater than this setpoint temperature (and the natural ventilation operation schedule is on) then [natural ventilation](#) can take place. The way this works depends on the [Natural ventilation model option](#).

Note that when using the [Calculated natural ventilation](#) model option windows and vents are only opened when:

- The inside air temperature is above this cooling setpoint temperature, and
- The inside air temperature is greater than the outside air temperature, and
- The operation schedule allows ventilation.

Also, the data under the [Calculated Natural Ventilation](#) header on the [Advanced tab](#) of the [Model Options](#) dialog can modify ventilation control behaviour.

When using the [Scheduled natural ventilation](#) model option the **Natural ventilation delta T** (below) can prevent natural ventilation from occurring if the outside air temperature is too high.

Natural ventilation min temperature schedule

The schedule which defines the time-varying indoor temperature (in °C) below which ventilation is shut off. The control is visible when the **2-By schedule** option is selected for **Min temperature definition**. It can be thought of as the time-varying cooling setpoint temperature which controls the activation of natural ventilation.

See also [Defining setpoint temperature schedules](#).

Natural ventilation indoor max temperature control

If there is a maximum zone air temperature above which the natural ventilation should shut off then check this option. This upper temperature limit is intended to avoid overheating a space and/or unnecessarily increasing cooling loads.

Normally when using natural ventilation for cooling this option will be left unchecked as it would be unusual to place an upper limit on the zone temperature for operation of such a system.

Indoor max temperature control option is used only with Scheduled natural ventilation and is not available for Calculated natural ventilation.

Max temperature definition

There are 2 ways to define the Natural ventilation max temperature:

- **1-By value** - use this option when the setpoint does not change with time in which case you can enter a fixed setpoint temperature.
- **2-By schedule** - use this option when the setpoint changes with time in which case you can select a schedule of time-varying temperature setpoints.

Natural ventilation max temperature

This is the fixed indoor temperature (in °C or °F) above which ventilation is shut off. The control is visible when the **1-By value** option is selected for **Max temperature definition**.

Natural ventilation max temperature schedule

The schedule which defines the time-varying indoor temperature (in °C) above which ventilation is shut off. The control is visible when the **2-By schedule** option is selected for **Max temperature definition**.

See also [Defining setpoint temperature schedules](#).

Mechanical ventilation control

The mechanical ventilation cooling setpoint temperature and delta T values described below control the activation of mechanical ventilation when using [Simple HVAC](#) together with the **1-Room ventilation Mechanical ventilation method**.

Mechanical ventilation cooling setpoint temperature

The mechanical ventilation cooling setpoint temperature controls the activation of mechanical ventilation when using [Simple HVAC](#) together with the **1-Room ventilation Mechanical ventilation method**. If the inside air temperature is greater than this setpoint temperature (and the mechanical ventilation operation schedule is on) then mechanical ventilation can take place.

This data and the corresponding delta T data described below are not used in simulations when working with the Simple HVAC **2-Ideal loads** option or with Detailed HVAC and are never used in Heating and Cooling design calculations.

Mechanical ventilation max natural ventilation in-out delta T

The mechanical ventilation Max int-out temp difference is used for [Scheduled mechanical ventilation](#) only and is the temperature difference between the indoor and outdoor air dry bulb temperatures below which ventilation is shut off. This is to allow ventilation to be stopped if the temperature outside is too warm and could potentially heat the space. It is not related to preventing drafts.

For example, if you specify a maximum delta temperature of 2°C, ventilation is assumed to be available if the outside air temperature is at least 2°C cooler than the zone air temperature. If the outside air dry bulb temperature is less than 2°C cooler or warmer than the indoor dry bulb temperature, then ventilation is automatically turned off. The values for this field can include negative numbers. This allows ventilation to occur even if the outdoor temperature is above the indoor temperature. The Delta Temperature control works in the code in the following way:

if $(IndoorTemp - OutdoorTemp) < \text{Max ventilation in-out delta } T$ then ventilation is **not allowed**.

For example:

Value of Mechanical ventilation Max in-out delta T	Effect
0	Mechanical ventilation only operates when $OutdoorTemp < IndoorTemp$
-50	Mechanical ventilation unrestricted by Inside-Outside delta T
2	Mechanical ventilation only operates when $OutdoorTemp + 2 < IndoorTemp$

Tip: Enter a large negative number to allow ventilation even if the outdoor temperature is greater than the indoor temperature. This useful for uncontrolled mechanical ventilation or as a way to estimate the effect of required ventilation air for load calculations.

See also [Defining setpoint temperature schedules](#).

Minimum Fresh Air

Minimum fresh air data is defined on the Activity tab because the fresh air requirement of the people in the zone depends on the activity (use) of the space.

Fresh air

The **Minimum fresh air per person** defines the fresh air requirements of each person in the zone and is used to generate appropriate mechanical outside air distribution rates when the mechanical ventilation [Outside air definition methods](#) on the [HVAC tab](#) is set to any of:

- **2-Min fresh air per person** where the fresh air flow rate is calculated as the **Fresh air (l/s/person)** x People / m² x Zone floor area (m²)
- **4-Min fresh air (Sum per person + per area)** where the fresh air flow rate is the sum of 'per person' and 'per area' fresh air requirements
- **5-Min fresh air (Max per person and per area)** where the fresh air flow rate based on 'per person' is compared with 'per area' and the maximum is used.

This data is also used to calculate *natural* ventilation outside air flow rates into the zone when the natural ventilation [Flow type](#) on the HVAC tab is set to **2-Min fresh air per person**.

Mech vent per area

The **Mech vent per area** data defines the fresh air requirements of the zone per floor area and is used to calculate mechanical outside air distribution rates when the mechanical ventilation [Outside air definition methods](#) on the [HVAC tab](#) is set to any of:

- **3-Min fresh air (Per area)**, where the fresh air flow rate is calculated as the **Mech vent per floor area** (l/s/m²) x Zone floor area (m²)
- **4-Min fresh air (Sum per person + per area)**,

- **5-Min fresh air (Max per person and per area).**

This data is not currently used for natural ventilation and is only used if one of the above mechanical ventilation [Outside air definition methods](#) has been used.

CO2/Contaminant Setpoints

If the zone has its fresh air delivery controlled using the **4-IAQP** DCV method and the **1-Simple HVAC Detailed HVAC Activity data** model option is set then you must enter the CO2 setpoint and minimum setting here on the Activity tab. Likewise if the **7-IAQP generic contaminant** option is selected then the generic contaminant setpoint and minimum setting must be entered here.

If the **2-Detailed HVAC Detailed HVAC Activity data** model option is set then this data is entered on the [HVAC Zone dialog](#) using schedules.

CO2 concentration setpoint

Enter the constant CO2 concentration setpoint in ppm.

Min CO2 concentration

Enter the minimum allowed CO2 concentration in ppm.

Max CO2 concentration

Enter the maximum allowed CO2 concentration in ppm. It is used to calculate design CO2 concentration level in the proportional ventilation control.

The **Min CO2 concentration** and **Max CO2 concentration** settings are only used when the [System outdoor air method](#) is set to one of:

- **5-Proportional control based on design occupancy**
- **6-Proportional control based on occupancy schedule**
- **9-Proportional control based on design outdoor air rate**

Generic contaminant concentration setpoint

Enter the constant generic contaminant concentration setpoint in ppm. The value should be between 0 and 1.

Lighting

Target illuminance

The desired lighting level (in lux) at the first daylighting sensor. This is also the lighting level that would be produced at this sensor position at night if the overhead electric [General lighting](#) were operating at full input power. Recommended values depend on type of activity; they may be found, for example, in the Lighting Handbook of the Illuminating Engineering Society of North America and in CIBSE A Guide. A typical value for general office work (excluding computer terminals) is 500 lux.

Default display lighting density

This data allows you to define the typical density of display lighting for the activity. The value is used as a default on the Lighting tab for the Task and Display lighting density to ensure that the display lighting density is appropriate for the activity of the zone.

When the [Lighting gain units](#) model option is set to **1-Power density** and a lighting template is loaded, this value is copied directly from the Activity tab to the corresponding [Task and Display Gain](#) value on the Lighting tab.

The Default display lighting density comes from the activity rather than from the lighting template because the gain is more likely to be a function of the activity than of the type of lighting system installed.

Domestic Hot Water (DHW)

[Activity tab](#) in model data

DHW Consumption rate

In DesignBuilder the amount of DHW consumed is considered to be a function of the activity in the zone. For example, there is a demand assumed to arise from the occupants of an office for activities such as washing hands and washing up cups. This demand is associated with the office rather than the toilet or tea room. Thus the demand from each space needs to be assigned to a DHW system even if the system itself is not present in the space.

The effect of hot water consumption (hot water taps, showers etc) on building energy use can be included in your thermal model by switching on [DHW](#) on the HVAC tab.

The data on this tab defines the hot water consumption rate in each zone as l/day/m² data (gal/day/ft in IP units). **The DHW consumption data is only used if DHW is selected on the HVAC tab.**

The DHW consumption rate is the amount of water used over a whole day per floor area. The consumption should be entered for the day of the year having the most consumption. For example in an office it might be the consumption over a typical work day. You should *not* enter an average rate over the year including reduced rates on weekends etc.

Note: If you have DHW consumption data in **l/person/day** format you can convert to the **l/m²-day** format used in DesignBuilder by multiplying it by [occupant density](#) (people/m²).

With Simple HVAC, the [hot water generation system](#) is defined under [DHW](#) on the HVAC tab.

Note: This DHW consumption data is also used in Detailed HVAC with the default **1-Simple HVAC Detailed HVAC Activity data** setting.

Computer Gains

[Activity tab](#) in model data

Check the **Computers > On** checkbox to model the heat emissions and fuel consumption due to computer gains and other related IT equipment in the simulation.

Units

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for internal gains here on the Activity tab. Choose from the options:

- **1-Power density** where zone equipment gains are entered as a power density, i.e. as a power per zone floor area in W/m² or W/ft². This option is usually the most convenient method for most general purpose modelling as the data is normalised by area and so can be inherited from levels above in the model hierarchy and can be effectively loaded from Activity template.
- **2-Absolute zone power** where zone equipment gains are entered as a power value in W. This option is usually used when detailed zone by zone survey or design data is available. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases

the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Gain

Set the design level of energy consumption due to computers. Depending on the [Equipment gain units](#) model option setting, the gains data is entered either as **Absolute zone power** (in W) or as **Power density** (in W/m² or W/ft²).

The fuel is assumed to be *electricity*, all gains are assumed to be emitted to the space.

Schedule

Select the schedule that governs the timing of gains in the zone. The level of gains are multiplied by the time-varying factor in this schedule to give the gain for each timestep in the simulation.

Radiant Fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of long-wave radiant heat being given off by computer equipment in a zone. The radiant fraction will be multiplied by the total energy consumed by electric equipment and (1-Fraction lost) to give the amount of long wavelength radiation gain from computer equipment in a zone. A value of zero means that the heat gain is entirely convective in which case all the heat is transferred to the air node in simulations. A value of 1 means that the gain is entirely radiative in which case all the heat is distributed to the inside surfaces. A typical value is somewhere between 0.1 and 0.2.

If the [Internal gains operate with occupancy](#) model option is not selected you can also set the operation schedule.

Office Equipment Gains

[Activity tab](#) in model data

Check the **Office Equipment > On** checkbox to include office equipment internal gains in the simulation.

Units

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for internal gains here on the Activity tab. Choose from the options:

- **1-Power density** where zone equipment gains are entered as a power density, i.e. as a power per zone floor area in W/m² or W/ft². This option is usually the most convenient method for most general purpose modelling as the data is normalised by area and so can be inherited from levels above in the model hierarchy and can be effectively loaded from Activity template.
- **2-Absolute zone power** where zone equipment gains are entered as a power value in W. This option is usually used when detailed zone by zone survey or design data is available. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Gain

Set the design level of energy consumption due to office equipment. Depending on the [Equipment gain units](#) model option setting, the gains data is entered either as **Absolute zone power** (in W) or as **Power density** (in W/m² or W/ft²).

The fuel is assumed to be *electricity*, all gains are assumed to be emitted to the space.

Schedule

Select the schedule that governs the timing of gains in the zone. The level of gains are multiplied by the time-varying factor in this schedule to give the gain for each timestep in the simulation.

Radiant Fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of long-wave radiant heat being given off by office equipment in a zone. The radiant fraction will be multiplied by the total energy consumed by electric equipment to give the amount of long wavelength radiation gain from office equipment in a zone.

A value of zero means that the heat gain is entirely convective in which case all the heat is transferred to the air node in simulations. A value of 1 means that the gain is entirely radiative in which case all the heat is distributed to the inside surfaces. a typical value is somewhere between 0.1 and 0.4.

If the [Internal gains operate with occupancy](#) model option is not selected you can also set the operation schedule.

Miscellaneous Gains

[Activity tab](#) in model data

Check the **Miscellaneous > On** checkbox to include any Miscellaneous gains in the simulation.

Units

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for internal gains here on the Activity tab. Choose from the options:

- **1-Power density** where zone equipment gains are entered as a power density, i.e. as a power per zone floor area in W/m² or W/ft². This option is usually the most convenient method for most general purpose modelling as the data is normalised by area and so can be inherited from levels above in the model hierarchy and can be effectively loaded from Activity template.
- **2-Absolute zone power** where zone equipment gains are entered as a power value in W. This option is usually used when detailed zone by zone survey or design data is available. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Gain

Set the design level of energy consumption due to miscellaneous equipment. The actual heat gain to the space may be reduced through the **Fraction lost** data (below).

Depending on the [Equipment gain units](#) model option setting, the gains data is entered either as **Absolute zone power** in W or as **Power density** in W/m² or W/ft².

Schedule

Select the schedule that governs the timing of gains in the zone. The level of gains are multiplied by the time-varying factor in this schedule to give the gain for each timestep in the simulation.

Fuel

The fuel can be selected from a drop down list. Note that these fuels are currently mapped to a shorter list in the results as follows:

Fuel in Model Data	Mapped to Fuel in Output
--------------------	--------------------------

1-Electricity from grid	Electricity
2-Natural gas	Gas

Fraction lost

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of 'lost' heat being given off by the equipment in a zone. The number specified in this field will be multiplied by the total energy consumed by equipment to give the amount of heat which is 'lost' and does not impact the zone energy balances. This might correspond to electrical energy converted to mechanical work or heat that is vented to the atmosphere.

Latent fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterize the amount of latent heat given off by miscellaneous equipment in a zone. The number specified in this field will be multiplied by the total energy consumed to give the amount of latent energy produced by the miscellaneous equipment. This latent energy affects the moisture balance within the zone.

Radiant Fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of long-wave radiant heat being given off by miscellaneous equipment in a zone. The radiant fraction will be multiplied by the total energy consumed by electric equipment and (1-Fraction lost) to give the amount of long wavelength radiation gain from miscellaneous equipment in a zone. A value of zero means that the heat gain is entirely convective in which case all the heat is transferred to the air node in simulations. A value of 1 means that the gain is entirely radiative in which case all the heat is distributed to the inside surfaces. a typical value is somewhere between 0.1 and 0.5.

If the [Internal gains operate with occupancy](#) model option is not selected you can also set the operation schedule.

CO2 generation rate

This numeric input field, available only when the fuel is selected as **2-Natural gas**, specifies carbon dioxide generation rate with units of m3/s-W or (ft3/min)/(Btu/hr). The default value of 0.0 assumes the equipment is fully vented to outdoors.

In the absence of better information, a value of 3.45E-8 m3/s-W can be used which assumes the equipment is not vented to outdoors. This value is converted from natural gas CO2 emission rate at 11.7 lbs CO2 per therm, the CO2 emission rate provided by U.S. Energy Information Administration. The maximum value for this input field is 3.45E-7 m3/s-W.

Calculation

The fuel input to the equipment ultimately appears as heat that contributes to zone loads. In the simulation this heat is divided into four different fractions. Three of these are given by the input fields Latent fraction, Radiant fraction and Fraction lost. The convected fraction, defined as the fraction of the heat from electric equipment convected to the zone air, is calculated by the program as:

$$F_{\text{convected}} = 1.0 - (\text{Latent fraction} + \text{Radiant fraction} + \text{Fraction lost})$$

You will get an error message if Fraction Latent + Fraction Radiant + Fraction Lost exceeds 1.0.

Catering Gains

[Activity tab](#) in model data

Check the **Catering > On** checkbox to include any cooking gains and other catering activities in the simulation.

Units

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for internal gains here on the Activity tab. Choose from the options:

- **1-Power density** where zone equipment gains are entered as a power density, i.e. as a power per zone floor area in W/m² or W/ft². This option is usually the most convenient method for most general purpose modelling as the data is normalised by area and so can be inherited from levels above in the model hierarchy and can be effectively loaded from Activity template.
- **2-Absolute zone power** where zone equipment gains are entered as a power value in W. This option is usually used when detailed zone by zone survey or design data is available. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Gain

Set the design level of energy consumption due to catering equipment. The actual heat gain to the space may be reduced through the **Fraction lost** data (below).

Depending on the [Equipment gain units](#) model option setting, the gains data is entered either as **Absolute zone power** in W or as **Power density** in W/m² or W/ft².

Schedule

Select the schedule that governs the timing of gains in the zone. The level of gains are multiplied by the time-varying factor in this schedule to give the gain for each timestep in the simulation.

Fuel

The fuel can be selected from a drop down list. Note that these fuels are currently mapped to a shorter list in the results as follows:

Fuel in Model Data	Mapped to Fuel in Output
1-Electricity from grid	Electricity
2-Natural gas	Gas

Fraction lost

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of 'lost' heat being given off by the catering equipment in a zone. The number specified in this field will be multiplied by the total energy consumed by equipment to give the amount of heat which is 'lost' and does not impact the zone energy balances. This might correspond to cooker heat that is vented to the atmosphere through an extract hood.

Latent fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterize the amount of latent heat given off by catering equipment in a zone. The number specified in this field will be multiplied by the total energy consumed to give the amount of latent energy produced by the catering equipment. This latent energy affects the moisture balance within the zone.

Radiant fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of long-wave radiant heat being given off by catering equipment in a zone. The radiant fraction will be multiplied by the total energy consumed by electric equipment and (1-Fraction lost) to give the amount of long wavelength radiation gain from catering equipment in a zone. A value of zero means that the heat gain is entirely convective in which case all the heat is transferred to the air node in simulations. A value of 1 means that the gain is entirely radiative in which case all the heat is distributed to the inside surfaces. a typical value is somewhere between 0.1 and 0.5.

If the [Internal gains operate with occupancy](#) model option is not selected you can also set the operation schedule.

CO2 generation rate

This numeric input field, available only when the fuel is selected as **2-Natural gas**, specifies carbon dioxide generation rate with units of m3/s-W or (ft3/min)/(Btu/hr). The default value of 0.0 assumes the equipment is fully vented to outdoors.

In the absence of better information, a value of 3.45E-8 m3/s-W can be used which assumes the equipment is not vented to outdoors. This value is converted from natural gas CO2 emission rate at 11.7 lbs CO2 per therm, the CO2 emission rate provided by U.S. Energy Information Administration. The maximum value for this input field is 3.45E-7 m3/s-W.

Calculation

The fuel input to the equipment ultimately appears as heat that contributes to zone loads. In the simulation this heat is divided into four different fractions. Three of these are given by the input fields Latent fraction, Radiant fraction and Fraction lost. The convected fraction, defined as the fraction of the heat from electric equipment convected to the zone air, is calculated by the program as:

$$F_{\text{convected}} = 1.0 - (\text{Latent fraction} + \text{Radiant fraction} + \text{Fraction lost})$$

You will get an error message if Fraction Latent + Fraction Radiant + Fraction Lost exceeds 1.0.

Process Gains

[Activity tab](#) in model data

Check the **Process > On** checkbox to include any process gains in the simulation. Process gains model the heat emissions and fuel consumption due to any manufacturing or industrial process which is part of the business activity.

Units

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for internal gains here on the Activity tab. Choose from the options:

- **1-Power density** where zone equipment gains are entered as a power density, i.e. as a power per zone floor area in W/m2 or W/ft2. This option is usually the most convenient method for most general purpose modelling as the data is normalised by area and so can be inherited from levels above in the model hierarchy and can be effectively loaded from Activity template.
- **2-Absolute zone power** where zone equipment gains are entered as a power value in W. This option is usually used when detailed zone by zone survey or design data is available. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Gain

Set the design level of energy consumption due to process activity. The actual heat gain to the space may be reduced through the **Fraction lost** data (below).

Depending on the [Equipment gain units](#) model option setting, the gains data is entered either as **Absolute zone power** in W or as **Power density** (in W/m2 or W/ft2).

Tip: You can enter **negative process gains** for modelling the cooling effect from refrigeration units such as those found in supermarkets where the refrigerator doors may open and cause cool air to enter the zone. Please note that **for any such negative gains, the associated fuel consumption is not accounted for.**

This is because the EnergyPlus fuel type must be set to “None” for negative gains regardless of the fuel type actually selected in the user interface.

Schedule

Select the schedule that governs the timing of gains in the zone. The level of gains are multiplied by the time-varying factor in this schedule to give the gain for each timestep in the simulation.

Fuel

The fuel can be selected from a drop down list. Note that these fuels are currently mapped to a shorter list in the results as follows:

Fuel in Model Data	Mapped to Fuel in Output
1-Electricity from grid	Electricity
2-Natural gas	Gas

Fraction lost

This field is a decimal number between 0.0 and 1.0 and is used to characterise the amount of 'lost' heat being given off by the process equipment in a zone. The number specified in this field will be multiplied by the total energy consumed by equipment to give the amount of heat which is 'lost' and does not impact the zone energy balances. This might correspond to electrical energy converted to mechanical work or heat that is vented to the atmosphere.

Latent fraction

This field is a decimal number between 0.0 and 1.0 and is used to characterize the amount of latent heat given off by process equipment in a zone. The number specified in this field will be multiplied by the total energy consumed to give the amount of latent energy produced by the process equipment. This latent energy affects the moisture balance within the zone.

Radiant fraction

This is a decimal number between 0.0 and 1.0 and is used to characterise the amount of long-wave radiant heat being given off by process equipment in a zone. The radiant fraction will be multiplied by the total energy consumed by electric equipment and (1-Fraction lost) to give the amount of long wavelength radiation gain from process equipment in a zone. A value of zero means that the heat gain is entirely convective in which case all the heat is transferred to the air node in simulations. A value of 1 means that the gain is entirely radiative in which case all the heat is distributed to the inside surfaces. a typical value is somewhere between 0.1 and 0.5.

If the [Internal gains operate with occupancy](#) model option is not selected you can also set the operation schedule.

CO2 generation rate

This numeric input field, available only when the fuel is selected as **2-Natural gas**, specifies carbon dioxide generation rate with units of m³/s-W or (ft³/min)/(Btu/hr). The default value of 0.0 assumes the equipment is fully vented to outdoors.

In the absence of better information, a value of 3.45E-8 m³/s-W can be used which assumes the equipment is not vented to outdoors. This value is converted from natural gas CO₂ emission rate at 11.7 lbs CO₂ per therm, the CO₂ emission rate provided by U.S. Energy Information Administration. The maximum value for this input field is 3.45E-7 m³/s-W.

Calculation

The fuel input to the equipment ultimately appears as heat that contributes to zone loads. In the simulation this heat is divided into four different fractions. Three of these are given by the input fields Latent fraction, Radiant fraction and Fraction lost. The convected fraction, defined as the fraction of the heat from electric equipment convected to the zone air, is calculated by the program as:

$$F_{\text{convected}} = 1.0 - (\text{Latent fraction} + \text{Radiant fraction} + \text{Fraction lost})$$

You will get an error message if Fraction Latent + Fraction Radiant + Fraction Lost exceeds 1.0.

Construction Model Data

Construction tab in model data

DesignBuilder uses [construction](#) components to model the conduction of heat through walls, roofs, ground and other opaque parts of the building envelope. Constructions can be selected on the Constructions model data tab to define the thermophysical and visual properties of the various internal and external surface elements in the building. From the Construction model data tab, you can access data on:

- [Construction](#)
- [Airtightness](#)
- "Construction Cost" on page 406

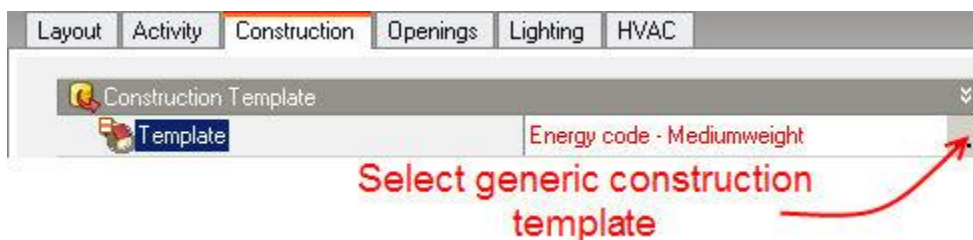
Open the headers to enter construction details where they are different from those loaded from the template. Any changes you make that are different from the template data originally loaded shown in bold red.

See also the [Construction data](#) and [Create and edit constructions](#) tutorials

Templates

General Construction and glazing

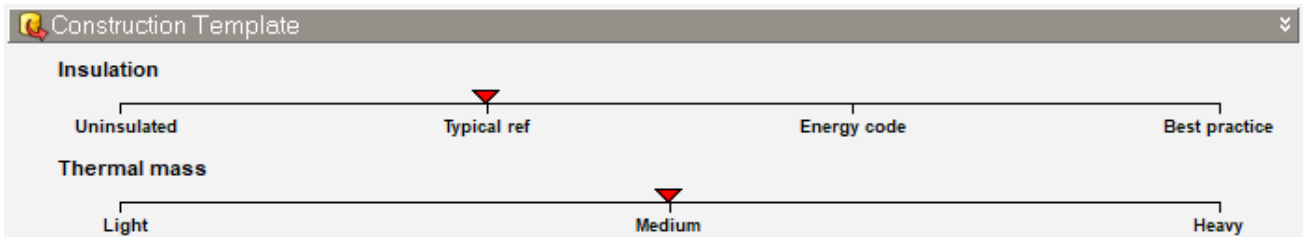
When you have the [General](#) Construction and glazing model option set, you can load generic Construction data by clicking on the Template option under Construction Template and then by clicking on the '...' at the right of the line (below).



When you make a selection from the list of [Construction templates](#) data from selected template is loaded to the model.

Pre-design Construction and glazing

Alternatively when you have the [Pre-design](#) Construction and glazing model option set, generic constructions can be loaded using the Insulation and Thermal mass sliders as shown below. In this case you are restricted to the 4x3 construction templates from the <System> Construction templates category identified by the combination of Insulation and Thermal mass selected on the sliders. For example in the screenshot shown below data will be loaded from the 'Energy code - Mediumweight' template.



As you change the slider settings DesignBuilder loads an appropriate set of constructions based on the rules explained below.

The various Insulation options take their insulation performance from the site level Energy code settings, which in turn default to settings based on the Region selected at the top of the Region tab.

When using the Pre-design Construction and Glazing model option ...



Tip: You can read about how to use and control these system "Early Design" Construction templates on the "Legislative Region Templates - Standard Insulation" on page 1510 help page.

Or in either case you can use the [Load data from template](#) tool.

Constructions

[Construction tab in model data](#)

From the Construction tab, you can open the group header boxes to edit detailed makeup of the construction of walls, roofs, floors, ceilings, partitions etc. used in the building.

The [default/inheritance](#) system used in DesignBuilder allows you to define building constructions quickly and easily by [loading data from templates](#) and by making global settings at building, block and zone levels.

Construction data:

- [Sub-Surface Construction](#) - construction details of sub-surfaces (opaque surfaces within surfaces)
- [Internal Thermal Mass](#) - how to define additional thermal mass
- [Adjacency](#) - defining special adjacency conditions
- [Geometry Areas and Volumes](#)
- [Surface Convection](#) - defining special surface convection options
- [Component Block](#) - construction and surface properties of component blocks
- [Solar Collector](#)
- [Linear Thermal Bridges at Junctions](#)
- [Baffle](#)
- "Ground Modelling" on page 392

Note: data on the [Openings tab](#) can be used to define the layout of the facade.

View/Edit construction details

To inspect the details of a selected [construction](#), click on the icon to the left of the appropriate data entry box ([Learning mode](#) only). Alternatively double-click the icon to view/edit the data in a dialog.

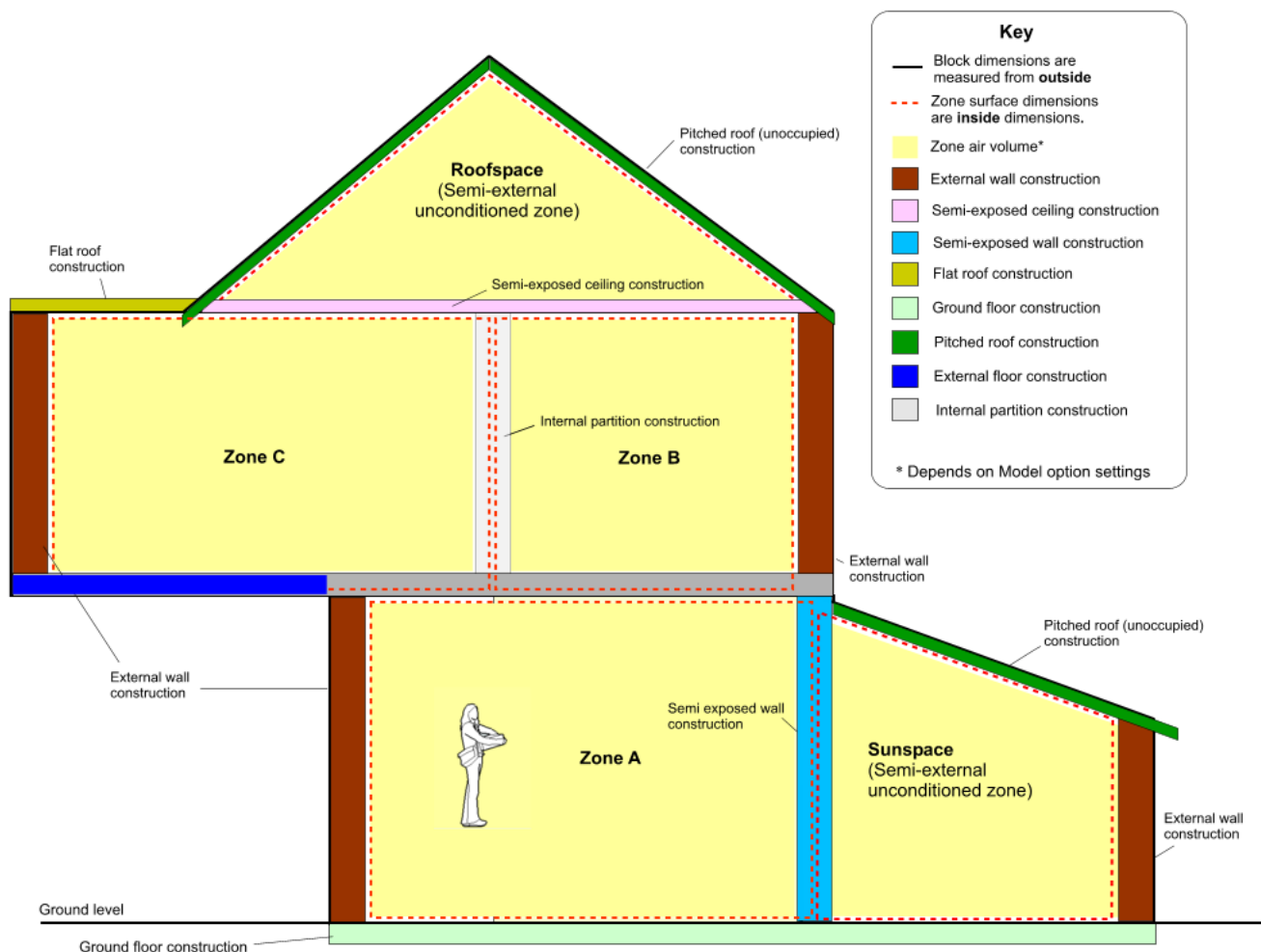
Note: Please be aware that **all settings made on the Constructions Model data tab are defaults, (unless you are working at the surface level)**. Model data at the surface level (inherited from the zone above or directly set) defines the thermal and visual characteristics of the surface in the model.

Constructions Data

[Construction tab](#) in model data under [Construction header](#)

Schematic Diagram Showing Construction Locations

The schematic below shows the locations of various surface types and the constructions that will be applied to them. It also shows block and zone dimension definitions.



Note: If you are using the NCM for EPCs, Part-L/Section 6 etc calculations you should refer to the [Block and Zone Dimensions](#) page for a more detailed definition of block heights.

External walls

The external wall construction defines the construction of walls adjacent to outside. The definition of walls is given by the roof-wall transition angle model option. In cold climates this construction will generally include an insulating layer.

Below grade walls

The Below grade wall construction is used for walls that are adjacent to ground. If there are no below grade (underground) walls in the model then just ignore this data.

Flat roof

Flat roof constructions apply to external horizontal surfaces. The construction used to represent the flat roof should include any slabs, insulation and suspended ceiling layers. If there are no flat roof surfaces, just ignore this data.

Pitched roof (occupied)

The Pitched roof (occupied) construction applies to external sloped surfaces when they appear in [occupied zones](#). In cold climates this construction will generally include an insulating layer. If there are no pitched roof surfaces, just ignore this data.

The definition of 'pitched roof' is given by the roof-wall transition angle model option.

Pitched roof (unoccupied)

The Pitched roof (unoccupied) construction applies to external sloped surfaces when they appear in [Semi exterior unconditioned zones](#). This construction will generally not include any insulation layers. If there are no pitched roof surfaces, just ignore this data.

The definition of 'pitched roof' is given by the roof-wall transition angle model option.

Internal partitions

The internal partition construction defines the construction of internal partitions (walls used to divide blocks into zones) and inter-block partitions (internal walls shared with other blocks). Internal partitions are used where both zones are occupied. When one of the zones is occupied and the other is not the **Semi-exposed walls** construction data is used instead (below). If there are no partitions then this data can be ignored.

The construction for the partition should represent all the layers in the actual internal wall. The partition between the zones is represented by a single Internal partition construction and all its layers should all be defined in the same construction.

Changes in model data settings at the surface level are reflected as equivalent changes in data for the partition on the surface representing other side.

Note: this data is not available at zone level to avoid ambiguity when different settings are made in adjacent zones. Surface level partition settings are generally inherited from block level, however settings for *inter-block* partition surfaces are inherited from the building level. For details on these rules see: "Zone Level Data for Internal Surfaces" on page 698.

Semi-Exposed

Semi-exposed walls

The semi-exposed wall construction applies to walls between fully occupied zones and [Semi-exterior unconditioned zones](#). They will normally include some insulation. DesignBuilder determines which of the zones is relatively 'outside' and uses this information to correctly order the material layers in the wall. The material layer defined in the construction as being 'outermost' (first-in list) will be the one adjacent to the semi-exterior unconditioned zone. If there are no semi-exposed wall surfaces, just ignore this data.

Note: this data is not available at zone level to avoid ambiguity when different settings are made in adjacent zones. Surface level semi-exposed wall settings are generally inherited from block level, however settings for *inter-block* semi-exposed walls are inherited from the building level. For details on these rules see: "Zone Level Data for Internal Surfaces" on page 698.

Semi-exposed ceiling

The semi-exposed ceiling construction is used between occupied zones and [Semi-exterior unconditioned zones](#) (such as roofspace/attic/loft spaces) above. The construction should include any slabs, insulation and suspended ceiling layers. DesignBuilder determines which of the zones is relatively 'exterior' and uses this information to correctly order the material layers in the roof. If there are no semi-exposed ceiling surfaces, just ignore this data.

Note: the semi-exposed ceiling data is defined from the occupied zone side only. If you navigate to the other side of the surface (the semi-exterior unconditioned zone floor) the data is not available for access.

Semi-exposed floor

The semi-exposed floor construction is used between occupied zones and [Semi-exterior unconditioned spaces](#) below. The construction should include any slabs, insulation and raised floor layers. If there are no semi-exposed floor surfaces, just ignore this data.

Note: This data is accessed from the occupied zone side only and cannot be accessed from the ceiling surface of the semi-exterior unconditioned zone below.

External door

Select the construction to be used to represent external doors.

Note: Doors use opaque constructions and so if the door you wish to model is transparent then it should be modelled using a window. If only a portion of the door is glazed then the door could be modelled as part door and part window.

Internal door

Select the construction to be used to represent internal doors.

Note: this data is not available at zone level to avoid ambiguity when different settings are made in adjacent zones. Surface level internal door settings are generally inherited from block level, however settings for *inter-block* internal doors are inherited from the building level. For details on these rules see: "Zone Level Data for Internal Surfaces" on page 698.

Floors

Ground floor

The ground floor construction is used between internal zones and the ground. The [site ground temperatures](#) are applied to the outside of all surfaces adjacent to ground. If there are no ground-adjacent floor or wall surfaces this data can be ignored.

Note: Constructions of type **6-F-Factor** can be selected here for both ASHRAE 90.1 and non-ASHRAE 90.1 models, however for surfaces with zero exposed perimeter the behaviour depends on whether it is an ASHRAE 90.1 model. For ASHRAE 90.1 models the **Basement ground floor** construction (below) is used instead. For non-ASHRAE 90.1 models an error message will be generated.

Basement ground floor

Used for ASHRAE 90.1 models only, the **Basement ground floor** construction is applied to ground adjacent floors whose zone has no external perimeter, which normally means it is a basement. The [site ground temperatures](#) are applied to the outside of these basement ground floor surfaces. If there are no basement ground floor surfaces in the model, this data can be ignored.

Note: The Basement ground floor is provided to help distinguish slab-on-grade floors from other floors. It is normally outside the scope of slab-on-grade floor definition described in ASHRAE 90.1 and is not regulated by the 90.1 Standard. In DesignBuilder, the baseline model uses the same construction for Basement ground floor as that used in the proposed model, typically a standard layered floor construction (as opposed to an F-Factor construction normally used for slab on grade floors).

External floor

External floors are floors that are adjacent to outside air, e.g. the floor of a cantilevered part of the building or where for any other reason there is no ground or other zone below. The External floor construction should include any slabs, insulation and raised floor layers. If there are no external floor surfaces, just ignore this data.

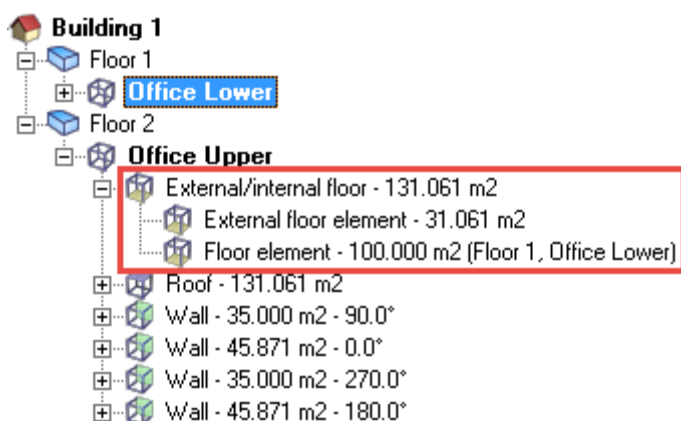
Internal floor

The internal floor construction defines the makeup of floors between occupied zones including any slabs, air gaps, raised floors and suspended ceilings. It will not usually contain insulation. Where the zone above is unoccupied the **semi-exposed ceiling** construction is used instead. Likewise where the zone below is unoccupied the **Semi-exposed floor** construction is used instead. If there are no internal floor surfaces, just ignore this data.

Navigator Use - Surface Level

When working with constructions it is often useful to view and make settings at surface level as this is the data that is actually used in calculations. Building, block and zone construction data is used as defaults for surface level data.

On the Navigator panel, for each surface, you will see one or more adjacency items listed below and to the right as shown in the highlighted region of the screenshot below. Clicking on any of the highlighted items in the navigator list will take you to the same surface level.



Important Note: DesignBuilder does not provide direct access to adjacency level data in the editor. For example, in the above screenshot, when navigating to the surface level by clicking on the "External/internal floor - 131.061 m2" item you will be working at the same surface level and with the same data as if you had navigated to one of the adjacency items below, e.g. "External floor element - 31.061 m2". You can set the construction type of both the external and internal floors from the surface level.

Zone Statistics

Zone Volume

The zone volume is calculated for all zone geometries and by default, the volume is the actual air volume of the space excluding the volume of floor and ceiling constructions but including half the volume of partition walls. The volume of the floor constructions can be optionally subtracted from the zone volume by using the [Zone volume calculation](#) Model options.

Other related information

1. [Block and Zone Dimensions](#) is an introductory section explaining how to enter blocks and the relationship with zones.
2. Also see [Constructions](#) for how to edit the construction configuration.
3. And the Model Geometry Example section for a detailed worked example illustrating how surface areas and volumes are calculated.

Sub-Surface Constructions

[Construction tab](#) in model data under [Construction header](#)

Sub-surfaces are used to define areas of a surface having a different construction to that of the main surface construction. Typical examples include cold-bridging elements such as window lintels and opaque panels in a lightweight façade. You can define different sub-surface construction types for external walls, internal partitions and pitched roofs. This makes it easier to set up defaults at building, block, zone or surface level.

Sub-surfaces are modelled to the same level of detail as standard surfaces including the effect of thermal mass but not including any 2-D or 3-D heat conduction effects due to the interface between the sub-surface and the background parent surface.

To **add** a sub-surface, go to the surface you wish to work on, click on the Layout tab and select the [Draw sub-surface](#) command.

You can also define default constructions for doors under the Sub-Surfaces header.

Sub-Surfaces	
 Walls	Project wall sub-surface construction
 Internal	Project internal wall sub-surface construction
 Roof	Project roof sub-surface construction
 External door	Project external door
 Internal door	Project internal door

The sub-surface constructions defined here are applied to sub-surfaces placed on the various base surface types as follows.

Walls

Any [sub-surfaces](#) created in a wall or ground floor will have this construction.

Internal

Any [sub-surfaces](#) created in any inside surface (partitions, internal floor etc) will have this construction.

Note: You can set Internal sub-surface defaults at building, block and surface levels. The Internal sub-surface option is not available at zone level to avoid possible ambiguity with sub-surfaces placed on internal walls between 2 different zones which could otherwise potentially have different settings.

Roof

Any [sub-surfaces](#) created in any pitched or flat roof will have this construction.

External door

Defines the construction of external doors.

Internal door

Defines the construction of internal doors.

Note: You can set Internal door defaults at building, block and surface levels. The Internal door option is not available at zone level to avoid possible ambiguity with doors placed on internal walls between 2 different zones which could otherwise potentially have different settings.

Internal Thermal Mass

[Construction tab in model data under Construction header](#)

DesignBuilder uses the thermal characteristics of the constructions for each of the walls, floors, roofs, partitions etc in each zone and accounts for the thermal mass in the simulations. You can also include additional thermal mass to account for partitions within a zone, furniture and any other mass which will affect the dynamic thermal response of the zone. There are two ways you can define additional internal thermal mass in DesignBuilder:

1. Drawing [hanging partitions](#) (i.e. partitions which do not meet other walls at both ends),
2. Entering a non-zero internal thermal mass exposed area on the Constructions tab under the [Internal Thermal Mass](#) header at zone level.

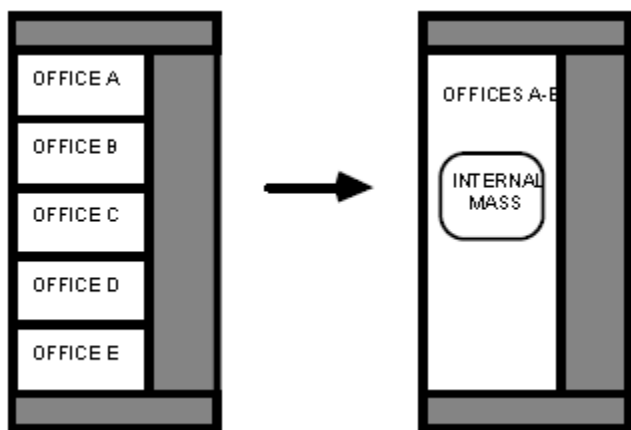
Construction

Thermal mass defined using these methods is lumped together for each zone and modelled in EnergyPlus in a 'non-geometric' way. In both cases, the associated construction is defined under the [Internal Thermal Mass](#) header on the Constructions tab.

Internal thermal mass defined in this way is used to specify the construction and area of items within the space that are important to heat transfer calculations but not necessarily important geometrically. For example, furniture within the space – particularly for large spaces. It can also be used for internal walls that are not needed for solar distribution, when [Solar distribution](#) is set to **3-Full interior and exterior** or to represent all interior walls when solar is distributed to the floors only with [Solar distribution](#) set to **2-Full exterior**.

Example

When zoning an office building, five west-facing offices have been combined into one zone. All of the offices have interior walls made of the same materials. As shown in the figure below, this zone may be described with 5 exterior walls and 11 internal walls or 1 exterior wall and 1 internal mass. Note that fewer surfaces will speed up the EnergyPlus calculations.



Example

A five-story building has the same ceiling/floor construction separating each of the levels. Zones that are on floors 2 through 4 may be described using a single piece of internal mass to represent both the floor and ceiling. The construction for this internal mass would be identical to the ceiling/floor construction that would be used to describe separate surfaces.

Exposed Area

The exposed area is the surface area of the internal mass. The area that is specified must be the entire surface area that is exposed to the zone. If both sides of a wall are completely within the same zone, then the area of both sides must be included when describing that internal wall.

Zone capacitance multiplier (Research Option)

This data can be used at building level only to control the effective storage capacity of the air in all zones in the building. Capacitance multipliers of 1.0 indicate the capacitance is that of the (moist) air in the volume of the specified zone. This multiplier can be increased if the zone air capacitance needs to be increased for stability of the simulation or to allow modelling higher or lower levels of damping of behaviour over time. The multipliers are applied to the base value corresponding to the total capacitance for the zone's volume of air at current zone (moist) conditions.

Typical values will be in the range 1 to 20.

Tip: The Zone capacitance multiplier can also be used to model the increase of air capacity due to lightweight thermal mass which could be considered to be mixed with the air. Examples include furniture, papers and any other items which might increase the effective thermal mass of the building zone response. For example, work at LBNL (Leveraging Zone Air Temperature Data to Improve Physics-Based Energy Simulation of Existing Buildings, by Sang Hoon Lee, Tianzhen Hong) suggests:

"The internal mass validation study using the DOE reference energy models (Deru et al. 2011) shows that a multiplier of 8.0 reflects a typical office configuration. An internal mass multiplier of 3 to 6 is recommended for light offices, 6 to 10 for typical offices, and 10 to 15 for heavy mass office configurations."

Adjacency

Construction tab in model data under Construction header

The adjacency of individual surfaces can be set to be either adjacent to ground or adiabatic.

The available options are:

- **1-Auto** - the adjacency of the surface is determined automatically by DesignBuilder based on its position. This is the default setting where external surfaces below the ground plane are considered to be adjacent to ground and external surfaces above the ground plane are considered to be adjacent to outside conditions.
- **2-Not adjacent to ground** - the surface is not adjacent to ground even if it is situated on or below the ground plane.
- **3-Adjacent to ground** - the surface is adjacent to ground even if it is above the ground plane or it is an internal surface.
- **4-Adiabatic** - the surface is adiabatic. This means that heat is not transferred across in its external surface. Adiabatic surfaces are frequently used in thermal modelling to represent surfaces which are between two zones at similar conditions. Note that thermal mass is still modelled in Adiabatic surfaces because heat can still be transferred into the mass through the internal surface. Adiabatic surfaces are often used for modelling the boundary between the actual building model and any [adjacent buildings](#) which are not to be modelled.

You can determine which surfaces are adiabatic or adjacent to ground by looking for the corresponding surface adjacency markers in the Edit Screen at zone and surface levels:



Adiabatic



Adjacent to ground

Tip: The recommended way to set ground and adiabatic adjacencies is to use ground or adiabatic [component blocks](#). Or, if for any reason that isn't practical and you need to set multiple surface adjacencies you can use the [Model data grid view](#).

Adiabatic surfaces

Adiabatic surfaces are modelled as being adjacent to the same conditions as the current zone and always use internal constructions:

- Adiabatic external floors use [Internal floor construction](#).
- Adiabatic flat roofs use the *reversed* [Internal floor construction](#).
- Adiabatic external wall surfaces use the [Internal partition construction](#).

Note: All openings and sub-surfaces in adiabatic surfaces are ignored in EnergyPlus simulations

Exclude this surface area from total zone floor area?

When modelling situations such as offices space within large warehouse zones where the ceiling at the top of the offices is not usable floor area in the warehouse it may be useful to exclude the area of any unused floor surfaces from the floor area of the zone above. An example of this situation is shown on the [Warehouse with Office](#) page.

For internal floors such as the unused 'floor' on top of the office space you can check the surface level **Exclude this surface area from total zone floor area?** checkbox to ensure the floor is not used in the calculation of zone and building floor area and in the calculation of internal gains where defined per m2 or per ft2.

Note this surface setting should *not* be made for the floor of office zone itself, which of course is usable floor area, but for the 'floor' of the zone above (typically a warehouse or other large volume space) which is not usable floor area.

Another situation where this option can be used is where holes have been cut in a floor surface to remove it, so upper and lower zones can be [merged](#). In this case, any small remaining border in the cut floor surface should ideally be removed from the floor area calculation for the combined zone.

Geometry Areas And Volumes

DesignBuilder provides a great deal of flexibility in the way that block and zone geometry is defined and passed through to the various calculations. There are various options including building blocks drawn using external measurements while providing correct internal zone geometry for floor area and zone volume calculations derived from actual surface thickness. It is also possible to use a simpler approach where the block and zone dimensions drawn are exactly the ones used in calculations and the thickness of the surfaces are not considered in the analysis.

Geometry convention template

Select the template which best fits the way you would like to define the building geometry. Templates provided include:

- **Simple** - where the zone surfaces, floor areas and volumes used in calculations are all derived directly from the drawn block and zone dimensions. If blocks are drawn using internal measurements then the model would be considered to be using an internal convention and if blocks are drawn using outer dimensions then the model would be using the external convention. Likewise to obtain a centre-line convention simply draw blocks to the centre-line between inner and outer dimensions. This is the only option provided in most other energy simulation tools.
- **External measurements** - the default option for simulation where blocks are drawn using outer dimensions and the surfaces used in thermal calculations are derived from the zone outer geometry and air volumes and floor areas are derived from the zone inner geometry. Note that this option works well for EnergyPlus simulations but is not compatible with the UK NCM which requires internal surface dimensions to be used in heat transfer calculations.
- **External measurements to top of roof** - like **External measurements** but zone geometry extends to the top of flat roofs.
- **DesignBuilder pre-v4 compatibility** - surfaces are derived in the same way as in v3 and earlier where a single "Block wall thickness" setting was used to define the thickness of all external wall surfaces in the block. For v3 and earlier models loaded to v4 and later the old "Block wall thickness" attribute is used to set the [External wall thickness](#) at block level.
- **External measurements, internal zone geometry** - blocks are defined using external measurements and surfaces dimensions, zone volume and floors areas are all derived from the inner zone geometry.
- **UK NCM external perimeter** - blocks are drawn using external measurements and simulations use internal measurements for surfaces, floor area and zone volume. The thickness of external wall and pitched roof constructions is used to define the inner zone surfaces, volume etc. Use this template for NCM work where the model has been or is to be defined using external block measurements. It may be useful in cases where a model previously drawn using external measurements is to be used or when importing from BIM where surfaces represent the external building geometry.

- **UK NCM internal perimeter** - blocks are drawn using internal measurements and simulations use those internal measurements for surfaces, floor area and zone volume. Use this template for NCM work as the easiest option on projects where it is possible to draw or import internal building geometry.

The usual inheritance mechanism applies so loading the required template at building level defines the geometry convention for the whole building. Settings can be made at building level only but [Fixed surface thicknesses](#) can be defined down to surface level and [Void depths](#) can be defined down to Zone level.

The Geometry convention template can also selected when creating a [new building](#).

Tip: After making any changes to the model which could affect the geometry including construction type selection as well as the settings described on this page, you should manually refresh the Navigator by

pressing the Refresh  Navigator toolbar icon. This will ensure that the Navigator is synchronised with the model.

Zone geometry and surface areas

This setting defines whether the geometric surface data used in the calculation (heat transfer etc) is based on the inner or outer zone volume dimensions. Options are:

- **1-Inner** - where the surface dimensions and areas used in thermal calculations are derived from the zone inner geometry. This is obtained from the outer zone geometry and "deflated" by the thickness of each surface and any floor or ceiling voids. For each type of surface the thickness comes from either a) the construction thickness or b) an overriding value set under [Fixed Surface Thickness](#) header. **You should carefully check the notes below when using this option.**
- **2-Outer** - where the surface dimensions and areas used in thermal calculations are derived from the zone outer geometry, which for a single zone block is exactly equal to the block outer dimensions.

Note 1: In cases where the **1-Inner volume** option is selected and non-zero surface thickness settings are being used, any component block shading devices placed adjacent to the block at building level will not be exactly adjacent to the corresponding zone inner surfaces used in the simulations which are offset inwards from the block outer geometry. The resultant gap will allow solar radiation to pass into the windows which will not be apparent by looking at the model at building level or on the visualisation screen. You are therefore advised to avoid using the combination of **1-Inner volume** geometry, non-zero surface thickness and external component blocks used as local shading devices. This issue does not arise with the [local shading](#) mechanism which ensures that overhangs, sidefins and louvre blades are offset with the window.

Note 2: In cases where the **1-Inner volume** option is selected and non-zero surface thickness settings are being used, for some zone geometries, the inner volume cannot be generated and in this case the zone outer surfaces will be used instead. This issue is discussed in more detail below under [Special Conditions](#).

Zone volume calculation method

This setting dictates whether internal or external measurements are used to calculate the zone volume is required to calculate air volumes etc.

- **1-Inner** - where the zone volume used in thermal calculations is derived from the zone inner geometry.
- **2-Outer** - where the zone volume used in thermal calculations is derived from the zone outer geometry.

Note: Unless otherwise specified by your regional modelling code, the **1-Inner volume** setting should generally be used for calculating zone volume to allow it to account for the effects of surface thickness and any floor and ceiling voids included.

Zone floor area calculation method

This setting dictates whether internal or external measurements are used to calculate the zone floor area. The options are:

- **1-Inner** - where the zone volume used in thermal calculations is derived from the zone inner geometry.
- **2-Outer** - where the zone volume used in thermal calculations is derived from the zone outer geometry.

The floor area calculated from the method selected here is used for:

- The Zone floor area used for calculating occupancy and other internal gains from the user-defined per floor area values.
- Occupied and Unoccupied "Floor Areas and Volumes" on page 331 reported at building level on the Activity tab.
- Floor area values for general reporting, e.g. in UK Part-L and EPC reports.

Note: Unless otherwise specified by your regional modelling code, the **1-Inner volume** setting should generally be used for calculating zone floor area.

Tip: For zones with walls that have a finite thickness, this floor area will not be the same as the total area of the floor surfaces listed in the Navigator, whose geometry is based on the [Zone geometry and surface areas](#) method.

Window to wall ratio method

This setting is used to specify whether or not the outside wall surface or inside wall surface is to be used to calculate the available area for glazing. This option is only visible when the [Zone geometry and surface areas](#) option is set to **2-Outer**. Options are:

- **1-Inner**, where the glazing layout for default (non-custom) façades are calculated based on the available internal area of external walls. For example if there is a non-zero height ceiling void (using the [Ceiling void depth](#) attribute) then the wall area seen from inside will be less than the area seen from outside, and this lower inside area is used in the calculation of the window size. Consider a case where the WWR is set to 100%. In this case the occupants inside the building will see external walls as being fully glazed, but people viewing from outside will see the areas of the outside wall where the ceiling voids are as opaque. Another example of how this setting affects façade layouts is that the sill height of default glazing is defined with reference to the top of the internal floor not the base of the block. Note that custom openings are unaffected by this setting.
- **2-Outer**, where the whole of the outer wall surface is used in calculating the window sizes. When this option is set and a 100% WWR is specified people viewing from outside will see a fully glazed building. In general it is more obvious how this setting works since there is only one wall area to consider.

Important Note: When the **Window to wall ratio method** is set to use the **2-Outer** wall surface, rendered views on the Visualisation, Daylighting and CFD screens will default to a single skin display. Therefore all surface elements including walls, partitions, floors, ceilings and roofs are displayed without thickness, and floor and ceiling voids are not displayed. This allows full size windows to be rendered accurately. Special treatment is required for this option because windows can occupy the whole surface and in that case it wouldn't be possible to show the extent of the windows together with the element thicknesses.

Also, because daylighting calculations use the same elements as seen on the Visualisation screen, the results will be based on the geometry seen there. So depending on the geometry of your model, the **2-Outer** option may not be ideal if you are planning to use the model to run daylighting calculations.

Fixed Surface Thicknesses

Each surface can have its geometric thickness calculated from the construction selected but only when the checkbox for the associated surface type (list below) is unchecked. If the thickness is to be overridden then check the appropriate checkbox and enter the fixed surface thickness. Settings can be made from building level right down to surface level giving considerable flexibility over the model geometry.

In most cases you should simply select the [Geometry convention template](#) - you shouldn't normally need to override the surface thickness settings below.

When at zone level, the fixed surface thickness for internal surfaces are not displayed due to the ambiguity involved if the 2 zones involved in the connecting internal surface have different settings. This applies to Internal partitions, Internal floor, Semi-exposed wall, Semi-exposed floor and Semi-exposed ceiling data.

Note: The surface thickness does not affect the thermal properties of the wall/floor/roof etc (U-value, thermal mass etc). Surface thermal properties are based purely on the construction data specified for the surface.

External wall

If the external wall surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the wall thickness (in m or in).

Below grade wall

If the below grade wall surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the wall thickness (in m or in).

Internal partition

If the partition surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the wall thickness (in m or in).

Ground floor

If the ground floor surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the floor thickness (in m or in).

Note: In most cases ground floor blocks are measured from ground level and the ground slab construction is not included in the volume of the building so this setting is normally overridden to be zero.

Basement ground floor

The basement ground floor is the construction used for underground ground floors (i.e. the floor is located at a height lower than $z=0$). If the basement ground floor surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the floor thickness (in m or in).

External floor

If the external floor surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the floor thickness (in m or in).

Internal floor

If the internal floor surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the floor thickness (in m or in).

Semi-exposed wall

If the semi-exposed wall surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the wall thickness (in m or in).

Semi-exposed floor

If the semi-exposed floor surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the floor thickness (in m or in).

Semi-exposed ceiling

If the semi-exposed ceiling surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the ceiling thickness (in m or in).

Flat roof

If the flat roof surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the flat roof thickness (in m or in).

Pitched roof

If the pitched roof surface thickness is to be overridden (i.e. not derived from construction thickness) then check the checkbox and enter the pitched roof thickness (in m or in).

Considerations For CFD

DesignBuilder [CFD](#) can include multiple zones in a single domain, but in this case it is important to ensure that all internal partitions and floors/ceilings separating zones in the domain have finite (non-zero) thickness. Any internal surfaces in the domain with zero thickness will be ignored in the analysis and so will not present a barrier to air flow.

Considerations For Parametric Analysis And Optimisation

When running parametric or optimisation studies in which wall or other surface thickness are changed as part of the parametric variations, there can be other significant knock-on impacts on the model geometry including varying floor area (and so internal gains, ventilation etc) and zone volume (and so ventilation and infiltration changes). To avoid running into these issues you can fix surface thickness for the relevant part of the model. For example if wall insulation thickness is being varied in a parametric analysis and you don't wish these changes to impact on zone floor area and volume then you could apply a constant External wall thickness.

Tip: the easiest way to avoid any such issues would be to use the "Simple" Geometry conventions template at the outset.

Void Depths

Ceiling void depth

The depth of the ceiling void (in m or in). This is used, in addition to any surface thickness applied to the ceiling/roof surface to reduce the size of the zone inner volume and surfaces areas to account for the ceiling void.

When using the [1-Inner](#) Zone geometry and surface areas option, ceiling voids will reduce the height of zone wall surfaces used in the simulation. Also when using the [1-Inner](#) Zone volume calculation method option, ceiling voids will reduce the zone volume for air flow and heat exchange calculations.

Note: If the ceiling/roof construction already includes an air gap to define the ceiling void and its thickness is being used to define the inner volume then you should enter 0 here. Likewise, if the ceiling void is already modelled using its own zone then enter 0.

Floor void depth

The depth of the floor void (in m or in). This is used, in addition to any surface thickness applied to the floor surface to reduce the size of the zone inner volume and surfaces areas to account for the floor void.

When using the **1-Inner** Zone geometry and surface areas option, floor voids will reduce the height of the zone wall surfaces used in the simulation. Also when using the **1-Inner** Zone volume calculation method option, floor voids will reduce the zone volume for air flow and heat exchange calculations.

Note: If the floor construction already includes an air gap for the floor void and its thickness is being used to define the inner volume then you should enter 0 here. Likewise, if the floor void is already modelled using its own zone then enter 0.

Technical: EnergyPlus Zone Floor Area and Volume Checking

When writing out zone floor area and volume settings to the EnergyPlus input data, DesignBuilder uses the geometry convention settings described above. This ensures that the real usable floor area and actual zone air volume are used in the simulation. In many cases these values will be less than the values that EnergyPlus estimates when carrying out its input checking which is based only on the 2-D surfaces that make up the zone geometry. You will therefore often see warnings generated in the error file such as:

```
** Warning ** Entered Zone Volumes differ from calculated zone volume(s).
** ~~~ ** ...use Output:Diagnostics,DisplayExtraWarnings; to show more details on individual zones.
```

You can safely ignore these warnings.

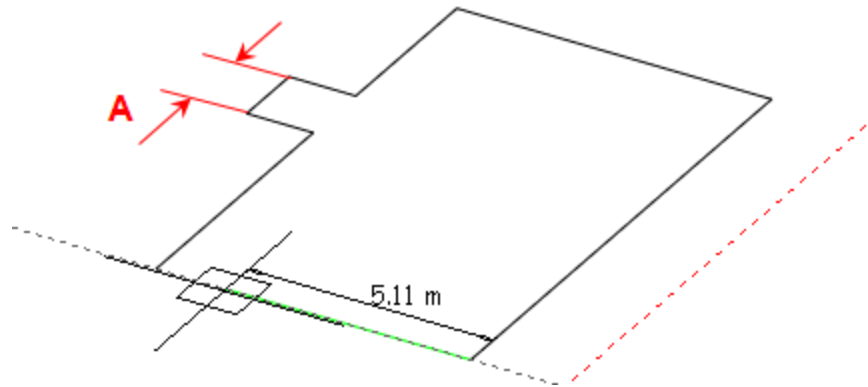
Special Conditions

Inner geometry cannot be generated

When using **1-Inner Zone geometry and surface areas** option, DesignBuilder creates an "inner volume" by moving all zone surfaces inwards using the construction thicknesses (or fixed thicknesses where specified). This process is called "deflation". In some cases, however the deflation process is not possible and the inner volume cannot be generated. The two main reasons for this failure are:

1. When the resulting inner volume geometry will not physically fit, often due to the surface thickness being too great. Examples of this situation include small kinks in the surface geometry with dimension of 2 x the surface thickness or less (image below) and zones where there are acute angle corners and thick wall constructions.

If dimension A is less than 2 x the wall thickness (however that is calculated) then the zone inner volume cannot be calculated



2. Zones where one or more zone outer geometry vertex has 4 or more surfaces adjacent.

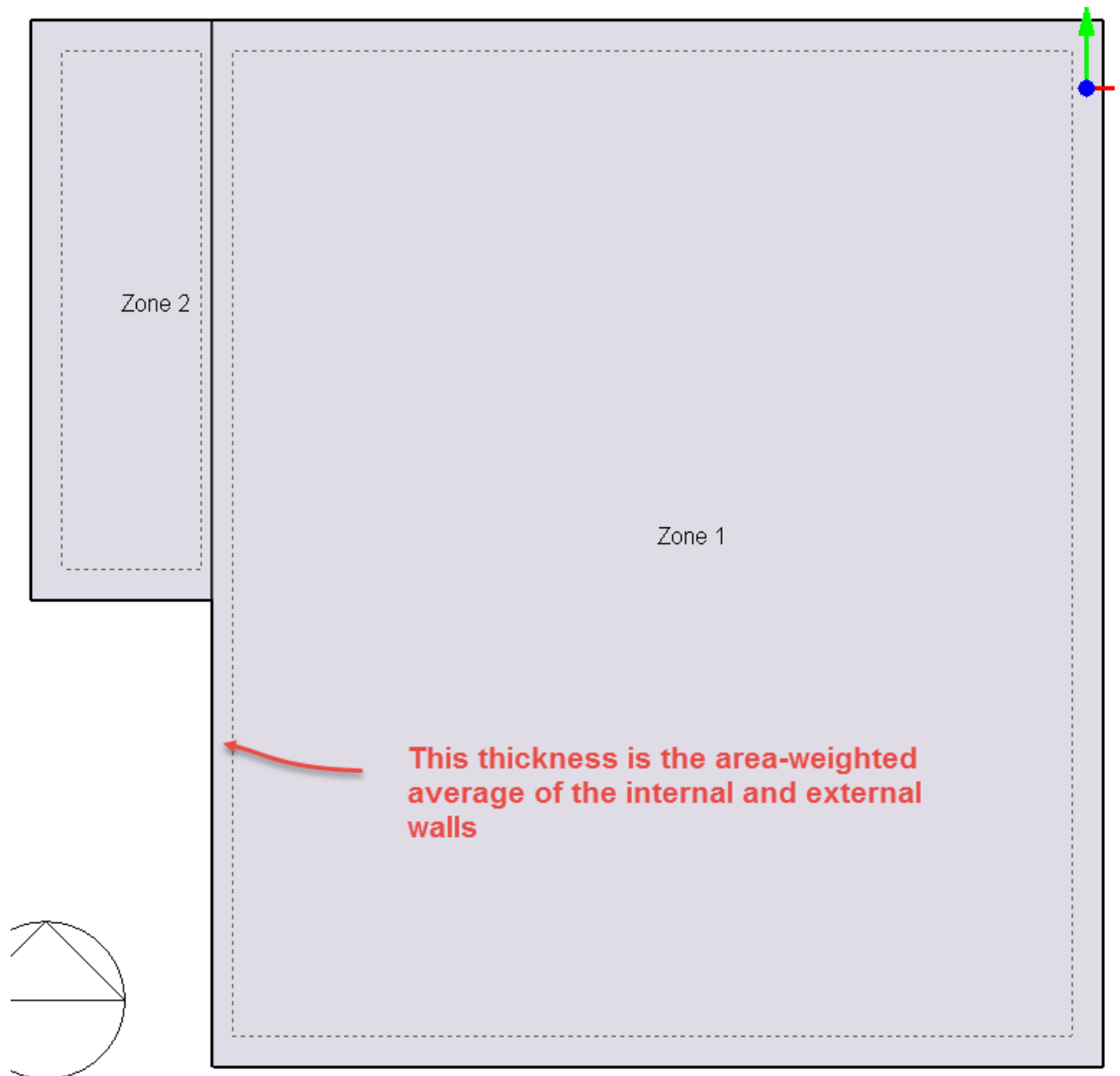
If the deflation process fails, the software uses an alternative mechanism in which the zone *outer* geometry is used for all zone thermal surfaces. Although DesignBuilder is unable to generate full inner zone geometry for these zones it can still calculate zone inner volumes and floor areas fairly accurately using a simpler algorithm based on the outer zone geometry and surface thicknesses.

Zone inner geometry is calculated zone by zone and so if the deflation process fails for 1 zone in a block, this should not affect the process in other zones in the block.

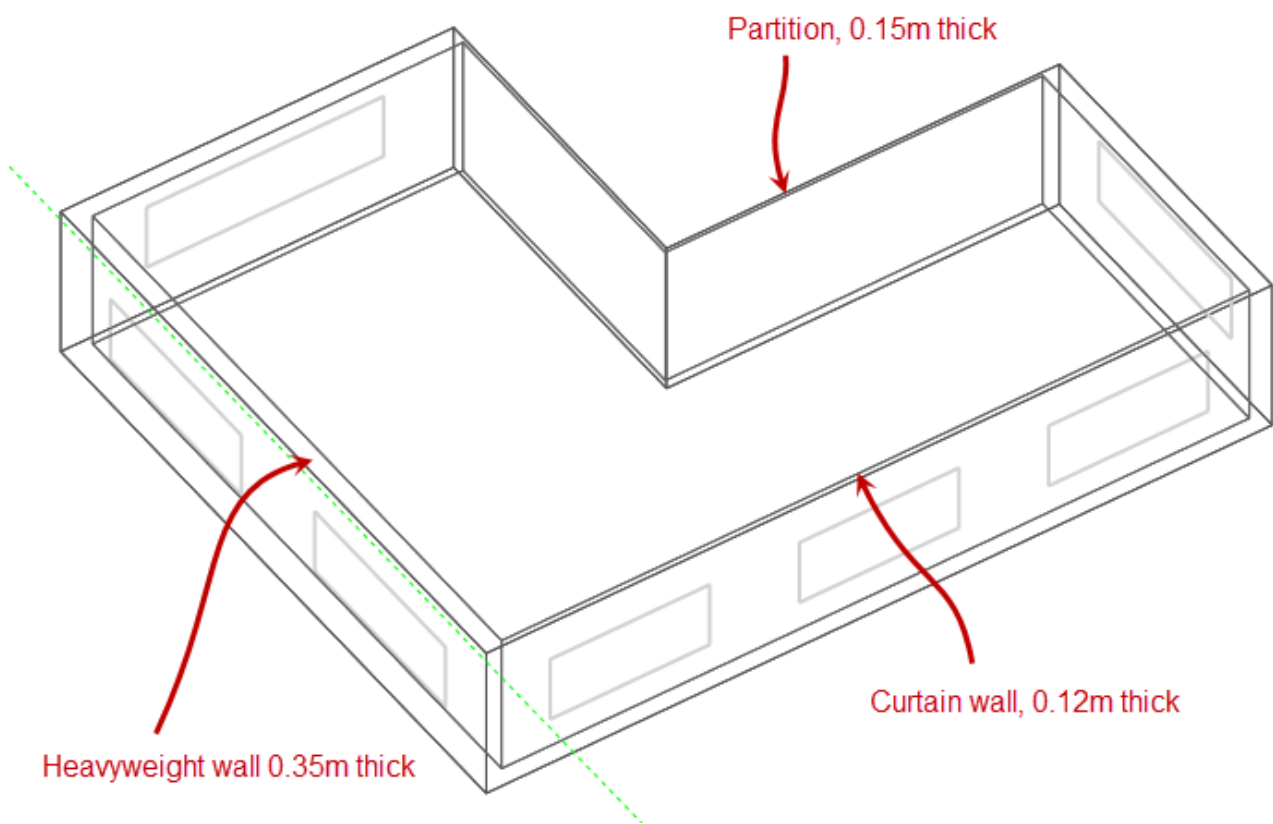
Wall thickness is weighted average

In cases where a surface is made up of more than one adjacency, each using a different construction, DesignBuilder must use an area-weighted average surface thickness. For example:

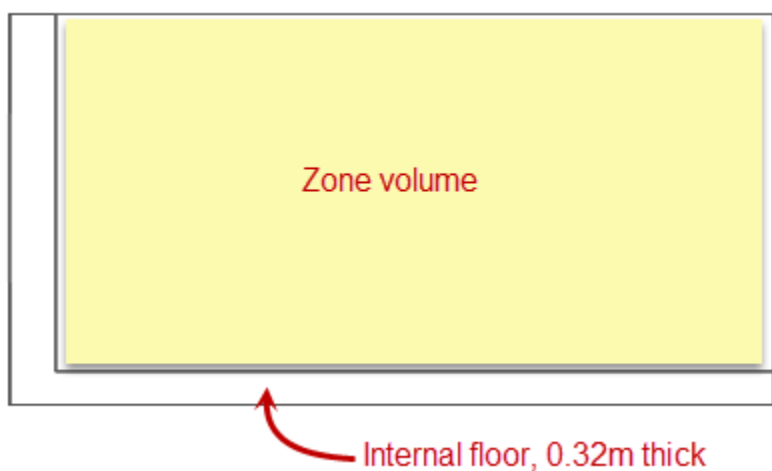
- When a wall surface has two different thickness, e.g. internal and external. This case is illustrated below.
- When the ceiling surface has two different thickness (flat roof and internal floor).



Geometry Convention Examples



External measurements, perspective view

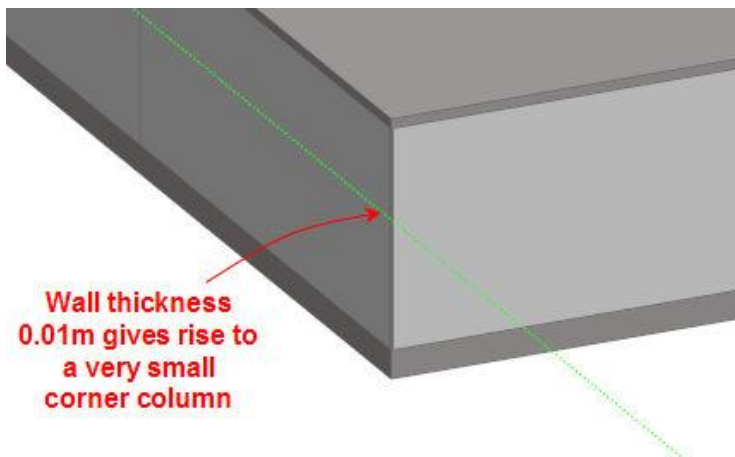


External measurements, section view

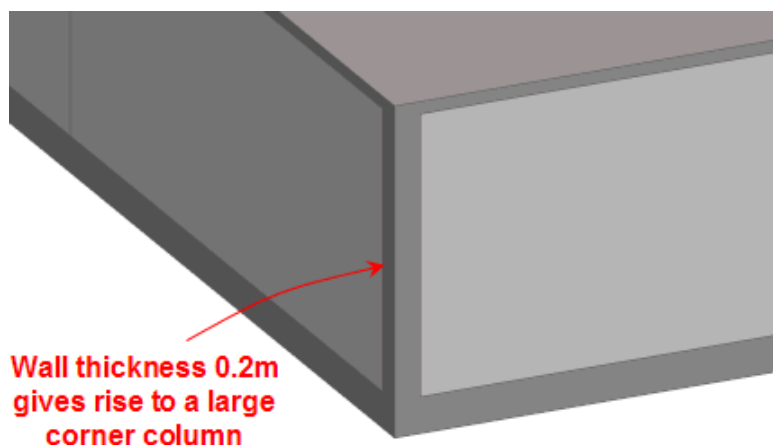
Curtain Glazing - Example

When modelling continuous glazing for curtain walls, atria, greenhouses etc. it is often necessary to model glazing joining at the corners. It is not possible for windows to literally meet in this way because EnergyPlus requires all openings (windows, doors etc) to be placed on a parent surface with at least a small gap between the opening and the parent surface edge. But by using a low value of wall surface thickness you can give the appearance of adjacent windows as shown below.

A small wall thickness, below, allows the glazing to cover most of the wall giving good results (below).



A larger wall thickness, below, requires more area to cater for the walls joining at the corners and the glazing does not meet 'round the corner' giving poor results (below).



Surface Convection

[Construction tab](#) in model data under [Construction header](#)

You can select the inside and outside surface convection algorithms in the Model data on the Construction tab under the Surface Convection header. This allows you to make different selections in different parts of the building. Further details of the algorithms are provided in the [EnergyPlus Engineering reference document](#).

Tip: You can override the inside and outside convection algorithm selection described below by setting fixed values on the [Surface tab of the Construction component dialog](#).

Inside convection algorithm

You can select from 6 main EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. Unless you have good reason to do so you are advised to use the default TARP convection algorithm:

- **1-AdaptiveConvectionAlgorithm** -This advanced option provides a dynamic selection of convection models based on conditions. Beausoleil-Morrison (2000, 2002) developed a methodology for dynamically managing the selection of h_c equations called adaptive convection algorithm. The algorithm is used to select among the available h_c equations for the one that is most appropriate for a given surface at a given time. As Beausoleil-Morrison notes, the adaptive convection algorithm is intended to be expanded and altered to reflect different classification schemes and/or new h_c equations. The adaptive convection algorithm implemented in EnergyPlus for the inside face has a total of 45 different

categories for surfaces and 29 different options for h_c equation selections. The tables provided in the Engineering document summarise the categories and the default assignments for h_c equations.

- **2-Simple** - The simple convection model uses constant coefficients for different heat transfer configurations, using the criteria to determine reduced and enhanced convection. The coefficients are taken directly from Walton (1983). Walton derived his coefficients from the surface conductances for $\epsilon=0.90$ found in the ASHRAE Handbook (1985) in Table 1 on p. 23.2. The radiative heat transfer component was estimated at $1.02 \times 0.9 = 0.918$ BTU/h-ft²-F and then subtracted off. Finally the coefficients were converted to SI units to yield the values below. For a vertical surface: $h_c = 3.076$ For a horizontal surface with reduced convection: $h_c = 0.948$ For a horizontal surface with enhanced convection: $h_c = 4.040$ For a tilted surface with reduced convection: $h_c = 2.281$ For a tilted surface with enhanced convection: $h_c = 3.870$.
- **3-CIBSE** - applies constant heat transfer coefficient derived from traditional CIBSE values.
- **4-Ceiling diffuser** - a mixed and forced convection model for ceiling diffuser configurations. The model correlates the heat transfer coefficient to the air change rate for ceilings, walls and floors. The ceiling diffuser algorithm is based on empirical correlations developed by Fisher and Pedersen (1997). The correlation was reformulated to use the room outlet temperature as the reference temperature. The correlations are shown below. For Floors: $h_c = 3.873 + 0.082 \times ACH^{0.98}$, For ceilings: $h_c = 2.234 + 4.099 \times ACH^{0.503}$ and for Walls: $h_c = 1.208 + 1.012 \times ACH^{0.604}$.
- **5-Cavity** - This algorithm was developed to model convection in a "Trombe wall zone", i.e. the air space between the storage wall surface and the exterior glazing. (See the later sections on Passive and Active Trombe Walls below for more information about Trombe walls.) The algorithm is identical to the convection model (based on ISO 15099) used in Window5 for convection between glazing layers in multi-pane window systems. The use of the algorithm for modelling an unvented Trombe wall has been validated against experimental data by Ellis (2003). This algorithm gives the convection coefficients for air in a narrow vertical cavity that is sealed and not ventilated. This applies both to the air gap in between panes of a window or to the air gap between the Trombe wall glazing and the inner surface (often a selective surface). These convection coefficients are really the only difference between a normal zone and a Trombe zone. See also note below.
- **6-TARP** - based on variable natural convection based on temperature difference from ASHRAE algorithms. This is the same as the old "Detailed" Inside convection algorithm provided in earlier versions of DesignBuilder. ***It is the default option for new models in v3.0.0.085 and later.***

The **5-Cavity** Inside convection algorithm is not available at the surface level.

Paraphrased note from EnergyPlus developers on the 5-Cavity option: "The Trombe wall convection coefficients only make sense for a zone. They are specific coefficients calculated for a narrow enclosed space. The two major walls of a Trombe wall zone are so close together that the convection patterns for the two walls actually interact. If they are close enough they can fight each other and totally stagnate the convection cell in the space. This is not free-boundary convection such as that found in a typical room. Therefore, it does not make sense to apply these coefficients to a single surface. The algorithm analyses the zone to figure out which are the two major surfaces and then sets the coefficients on those surfaces. The other minor surfaces receive negligible convection."

The following inside convection options are also available for advanced users:

- 7-ASHRAEVerticalWall
- 8-WaltonUnstableHorizontalOrTilt
- 9-WaltonStableHorizontalOrTilt
- 10-FisherPedersenCeilingDiffuserWalls
- 11-FisherPedersenCeilingDiffuserCeiling
- 12-FisherPedersenCeilingDiffuserFloor
- 13-AlamdariHammondStableHorizontal
- 14-AlamdariHammondUnstableHorizontal
- 15-AlamdariHammondVerticalWall
- 16-KhalifaEq3WallAwayFromHeat
- 17-KhalifaEq4CeilingAwayFromHeat
- 18-KhalifaEq5WallNearHeat
- 19-KhalifaEq6NonHeatedWalls
- 20-KhalifaEq7Ceiling
- 21-AwbiHattonHeatedFloor

- 22-AwbiHattonHeatedWall
- 23-BeausoleilMorrisonMixedAssistedWall
- 24-BeausoleilMorrisonMixedOpposingWall
- 25-BeausoleilMorrisonMixedStableFloor
- 26-BeausoleilMorrisonMixedUnstableFloor
- 27-BeausoleilMorrisonMixedStableCeiling
- 28-BeausoleilMorrisonMixedUnstableCeiling
- 29-FohannoPolidoriVerticalWall
- 30-KaradagChilledCeiling
- 31-ISO15099Windows
- 32-GoldsteinNovoselacCeilingDiffuserWindow
- 33-GoldsteinNovoselacCeilingDiffuserWalls
- 34-GoldsteinNovoselacCeilingDiffuserFloor

To avoid discontinuities in surface heat transfer rate calculations, all correlations are extrapolated beyond the lower limit of the data set (3 ACH) to a natural convection limit which is applied during the hours when the system is off. These models are explained in greater detail in the EnergyPlus Engineering Reference Document.

Outside convection algorithm

Substantial research has gone into the formulation of models for estimating the exterior convection coefficient. Since the 1930's there have been many different methods published for calculating this coefficient, with much disparity between them (Cole and Sturrock 1977; Yazdanian and Klems 1994). Unless you have good reason to do so you are advised to use the default DOE-2 convection algorithm. You can select from 7 main outside convection algorithms:

- **1-AdaptiveConvectionAlgorithm** - This advanced convection algorithm provides a dynamic selection of convection models based on conditions. This algorithm has a structure that allows for finer control over the models used for particular surfaces. The algorithm for the outside face was developed for EnergyPlus but it borrows concepts and its name from the research done by Beausoleil-Morrison (2000, 2002) for convection at the inside face (see [above](#)). The adaptive convection algorithm implemented in EnergyPlus for the outside face is much simpler than that for the inside face. The surface classification system has a total of 4 different categories for surfaces that depend on current wind direction and heat flow directions. However in other ways it is more complex in that the equation is split into two parts and there are separate model equation selections for forced and natural convection.
- **2-SimpleCombined** - The simple algorithm uses surface roughness and local surface wind speed to calculate the exterior heat transfer coefficient. This is a combined heat transfer coefficient that *includes radiation to sky, ground, and air*. The correlation is based on Figure 1, Page 25.1 (Thermal and Water Vapor Transmission Data), 2001 ASHRAE Handbook of Fundamentals.

Note: selecting the **SimpleCombined** option at *building level* on the Constructions tab under [Surface Convection](#) or on the Calculation options or Model options dialogs has a special meaning. In this case the outside convective selection covers both convection and radiation combined. This means for example that any hard-set outside convective heat transfer coefficients set on the [Surface properties](#) tab of the Construction dialog will actually be used as a *combined* outside convective plus radiation coefficient.

- **3-CIBSE** - applies constant heat transfer coefficients depending on orientation, derived from traditional CIBSE values.
- **4-ASHRAEVerticalWall** - Identical to the DOE-2 option.
- **5-TARP** - TARP, or Thermal Analysis Research Program, is an important predecessor of EnergyPlus (Walton 1983). Walton developed a comprehensive model for exterior convection by blending correlations from ASHRAE and flat plate experiments by Sparrow et. al. In older versions of EnergyPlus, prior to version 6, the "TARP" model was called "Detailed." The model was re-implemented in version 6 to use Area and Perimeter values for the group of surfaces that make up a facade or roof, rather than the single surface being modelled.
- **6-DOE-2** - The DOE-2 convection model is a combination of the MoWiTT and BLAST Detailed convection models (LBL 1994). ***It is the default option for new models.***

- **7-MoWiTT** - The MoWiTT model is based on measurements taken at the Mobile Window Thermal Test (MoWiTT) facility (Yazdanian and Klems 1994). The correlation applies to very smooth, vertical surfaces (e.g. window glass) in low-rise buildings. The MoWiTT algorithm may not be appropriate for rough surfaces, high-rise surfaces, or surfaces that employ movable insulation.

The following outside convection options are also available for more advanced users:

- 8-WaltonUnstableHorizontalOrTilt
- 9-WaltonStableHorizontalOrTilt
- 10-AlamdariHammondStableHorizontal
- 11-AlamdariHammondUnstableHorizontal
- 12-FohannoPolidoriVerticalWall
- 13-NusseltJurges
- 14-McAdams
- 15-Mitchell
- 16-BlockenWindard
- 17-Emmel
- 18-ClearRoof

Note: When the surface is wet (i.e. it is raining and the surface is exposed to wind) then the convection coefficient appears in results as a very large number (1000) and the surface is exposed to the Outdoor Wet Bulb Temperature rather than the Outdoor Dry Bulb Temperature.

As well as being able to define convection options on the Model data on Construction tab under the [Surface Convection](#) header as described above, building level inside and outside convection algorithm settings can be made on:

- Model options dialog ([Heating design](#), [Cooling design](#), [Simulation](#) tabs)
- Calculation options dialogs ([Heating design](#), [Cooling design](#), [Simulation](#))

You cannot make changes to individual surface convection settings where the surface is an internal partition.

All of the above algorithms are described fully in the [EnergyPlus Engineering Reference](#).

Note: the equivalent inside and outside surface convection options in the Calculation and Model options dialogs control **building default values** (equivalent of making a setting in Model data at building level) and any changes made at block, zone or surface levels in the Model data will override these default settings.

Component Block

[Construction tab](#) in model data under [Construction header](#)

[Component blocks](#) can be used to include shading and reflection effects of objects such as local shading devices, awnings, adjacent buildings and trees. They can also be used to define ground and adiabatic adjacencies. The component block model data allows you to define the component block type, whether the component block shades and reflects in simulations, transmittance of the shading material and also a schedule for modelling variations in transmittance at different times of the day, seasons etc. For example they can be used for modelling deciduous trees which shade more in the summer when they have leaves than they do in winter when the leaves have fallen.

Component block type

You can change the type of the component block. Select from:

- [1-Standard](#) - used for shading, reflection and visualisation.
- [2-Ground](#) - used for setting ground adjacency, shading, reflection and visualisation.

- **3-Adiabatic** - used for setting adiabatic adjacency, shading, reflection and visualisation.

For more information on the Component block types see [Component Block](#) under [Working with Blocks \(Building Level\)](#)

Shades and reflects

Switch this option on to include the component block as a shading/reflection surfaces in the simulations, or to include the component block in an LCA or cost analysis.

Shading surfaces have the following effects in an EnergyPlus simulation:

- Shade direct beam solar radiation.
- Reflect solar radiation onto the building (if you select [Model all external reflections and shading of ground reflected solar](#) in the calculation options).
- Shade diffuse solar and long-wave radiation from the sky.
- Shade diffuse solar radiation from the ground (if you select [Model reflections](#) in the calculation options).

Note: Solar and light reflection from the ground is included even if the solar [Model reflections](#) option is not used, but if reflections are not calculated the ground plane is considered unobstructed, i.e., the shadowing of the ground by the building itself or by component blocks is ignored. Shadowing of the ground by component blocks is only taken into account if the [Model reflections](#) option is used. Conclusion: Switch **Model reflections** on for the most accurate shading calculations.

Note: all component block types cast shadows in the [visualisations](#).

Level

This setting allows you to associate component blocks with either the building or the site, depending on whether they are in some way part of the building or a fixed part of the site layout. Most of the time you won't need to change it, but for some simulations it can be important to attach component blocks to the site rather than the building. For example if you are trying to optimise a building's orientation using the parametric or optimisation tools, or using the ASHRAE 90.1 automated baseline or UK DSM tools, you should leave component blocks that represent building structures such as shades and canopies set to the default building level. However for component blocks that represent *fixed* site objects such as other buildings or site topography, you should set these to be site level objects to ensure that they are not rotated with the building as part of any parametric variations.

Select whether the component block is considered to be part of the building or the site, choosing from:

- **1-Building** - the default option where the component block is considered to be part of the building and so is rotated with the building as part of parametric and optimisation studies where the [Building orientation](#) is varied. Typically, any component blocks used to model building shading, or other building objects such as steps, should be given this assignment. Building Level component blocks are also used in LCA analyses.
- **2-Site** - the component block is considered to be part of the fixed wider site, perhaps representing an existing building or a tree and so is not rotated with the building as part of parametric and optimisation studies.

Note: It is especially important to make the correct setting here for UK DSM and ASHRAE 90.1 Appendix G PRM buildings because any **1-Building** level standard component blocks are removed from the auto-generated notional and reference/baseline buildings. This is done based on the assumption that they represent shading devices that must be excluded to be in keeping with code rules. If the component block is intended to represent another building or other site object make sure to use the **2-Site** setting.

Tip: Any component blocks that need to remain in contact with building including ground component blocks must use the **1-Building** option. This may not be obvious because you may consider the ground to be part

of the site, but the important point to remember is that ground component blocks are used to create ground adjacencies for the building surfaces and so they should be considered as part of the building.

Material

The component block material is used to define the [solar and visible reflectance](#) of the block surfaces in Cooling design calculations and Simulations using real weather data when the [model reflections](#) calculation option is selected. It is also used for [LCA calculations](#).

Important Note: Component blocks do not absorb or conduct heat in any way - their only effect on building surfaces in simulations is related to the shading and reflection of short-wave solar radiation and light. In cases where the component block touches the building, the material associated with the component block and the thickness of the block are not used in any way to modify the conduction of heat.

Note: Although component blocks are displayed in DesignBuilder as solid objects, non-flat component blocks are modelled in EnergyPlus as a group of surfaces and it is the reflectance and transmission of each of these surfaces that is being described here. The solar flux incident upon a building façade will be attenuated by each successive shading object located on a path between the façade and the sun position.

Flat surface position

If the component block is determined to be flat (based on the [Maximum flat surface thickness](#) threshold accessible on the Advanced tab of the Model options dialog, you can control how the single surface used to represent a flat component block in simulations is defined. Choose from:

- **1-Upper surface**, where the uppermost of the 2 largest component block surfaces is used.
- **2-Lower surface**, where the lowermost of the 2 largest component block surfaces is used.
- **3-Midway surface**, where a surface midway between the lowermost and uppermost large component block surfaces is used in simulations.
- **4-All surfaces**, where flat components are represented using all surfaces like non-flat component blocks.

Maximum Transmittance

Warning: For *flat, partially transmitting* component blocks you must ensure that the blocks are either rectangular or triangular in shape. More complex shaped *flat, partially transmitting* component blocks will not transmit solar radiation and light as expected.

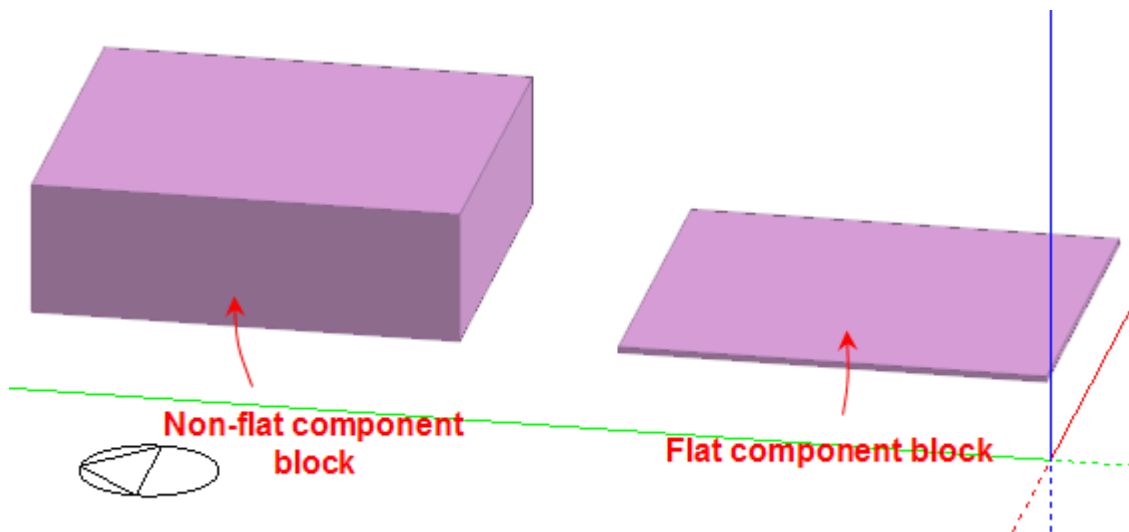
Flat, partially transmitting component blocks must be rectangular or triangular in shape to ensure correct solar and light transmission.

This is the maximum solar transmittance of the entire component block to be used in EnergyPlus simulations. The transmittance of the component block during the simulation is calculated by multiplying the maximum transmittance by the time varying schedule value:

$$\text{Transmittance} = \text{Max Transmittance} \times \text{Schedule value}$$

You will generally want component blocks to provide 100% shading for 100% of the time. In this case you can leave the **Maximum transmittance** and **Transmittance schedule** in their default states of 0 and **On** respectively. If you wish the component block to transmit solar and visible radiation you can define the extent and timing of the transmission. The component block transmittance is calculated during the simulations as the **Maximum transmittance** multiplied by the value of the transmittance schedule below. The maximum transmittance can have any value between 0 and 1. Note that if the component block is [flat](#) (i.e. it consists of 2 large surfaces and several smaller 'edge' surfaces) the block is represented in the simulations by a single shading surface. In this case the surface used will depend on the value of the [Flat surface position](#) above.

Tip: If your component block has a constant transmittance you can set the transmittance schedule below to **On** and set the constant transmittance for the block here.



For non-flat component blocks every surface is included in the simulation as a shading surface and the solar flux incident upon a building facade will be attenuated by each successive shading object located on a path between the façade and the sun position. The shading transmittance schedule you define is therefore modified to ensure that the overall transmittance of the block is the Maximum Transmittance value multiplied by the schedule you selected. This is done assuming that the component block is convex and that therefore solar rays can only pass through a maximum of 2 surfaces.

Note: In the IDF shading surface data generated for each non-flat component block surface, the transmittance is set to the square root of the Maximum Transmittance value you entered to achieve this.

Non-convex, non-flat component blocks will provide more shading than you specified during times when solar rays pass through more than 2 surfaces.

Transmittance schedule

The component block transmittance schedule defines any time-varying transmittance of the component block surfaces (for EnergyPlus simulations only). By default, the component block is fully opaque for the whole year. The transmittance schedule can contain any value between 0 and 1. The transmittance schedule can be used to allow for seasonal transmittance change, such as deciduous trees that have a higher transmittance in winter than in summer. The table "Modifying Characteristics of Various Species of Trees" in "Landscape Architectural Graphic Standards"• by Leonard J. Hopper, 2007 contains some data on tree solar transmission.

Transmittance based on time of day can also be used - a movable awning, for example, where the transmittance is some value less than 1.0 when the awning is in place and is 1.0 when the awning is retracted.

The following assumptions are made in the shading surface transmittance calculation:

- Both sides of the shading surface have the same transmittance properties.
- The transmittance is the same for both beam and diffuse solar radiation.
- Beam solar transmittance is independent of angle of incidence on the shading surface.
- Beam radiation incident on a shading surface is transmitted as beam radiation with no change in direction, i.e., there is no beam-to-diffuse component.
- If two shading surfaces with non-zero transmittance overlap, the net transmittance is the product of the individual transmittances. Inter-reflection between the shading surfaces (and between the shading surfaces and the building) is ignored.

- For the [daylighting](#) calculation the shading surface's visible transmittance is assumed to be the same as its solar transmittance.
- Shading devices are assumed to be opaque to long-wave radiation no matter what the solar transmittance value is.

Note: Shading devices only shade solar radiation when the sun is up, which is automatically determined by EnergyPlus from latitude, time of year, etc. So you only need to account for the time-varying transmittance of the shading device in the transmittance schedule, for times whether the sun is up. In other words for times when the sun is not up, it doesn't matter what value of transmittance is contained in the schedule.

Radiance daylighting

Component blocks allow semi-transparent transmission in Radiance daylighting calculations as well as in EnergyPlus. However the Maximum transmittance and Transmittance schedule described above are not used for Radiance daylighting simulations. Instead, the Radiance material properties defined on the Surface properties tab of the Materials dialog are used.

Solar Collector Model Data

[Construction tab](#) in model data under Solar Collector header

Once a solar collector surface has been placed you can define its properties on the Constructions tab under the Solar Collector header.

Solar collector type

You can change the type of solar collector using the drop list. Choose from:

- **1-Solar hot water**,
- **2-Photovoltaic** (PV)

Depending on the type of solar collector selected you will see different data below. For [Solar hot water collectors](#) the only data that can be edited here is the material used to represent the panel in rendered views on the Visualisation tab. All of the rest of the data is defined with the [HVAC Solar collector](#) component and its host loop.

When the **2-Photovoltaic** option is selected further configuration and performance data for the PV panels can be entered as shown in the screenshot below.

Solar Collector	
Solar collector type	2-Photovoltaic
Cost (GBP/m2)	600.000
☒ Shades and reflects	
Level	1-Building
Material	Bitumen Felt
Flat surface position	1-Upper surface
Photovoltaic Options	
Performance type	1-Simple
Performance model	Copy of PV Scheduled Efficiency = 0.15
Heat transfer integration mode	1-Decoupled

Cost

You can define the cost of the collector per area. The units will be the unit of currency currently selected on the Program options dialog per area (e.g. GBP/m2 or USD/ft2).

Costs for the electric distribution can be included separately on the [Load centre](#) dialog.

Level

This setting allows you to associate solar panels with either the building or the site, depending on whether they are attached to the building or a fixed part of the site layout. Most of the time you won't need to change it, but for some simulations it can be important to attach panels to the site rather than the building. For example if you are trying to optimise the orientation of the solar collector using the parametric or optimisation tools, or using the ASHRAE 90.1 automated baseline or UK DSM tools, you should leave panels that are attached to the building structures set to the default building level. However for panels that are *fixed* site objects, you should set these to be site level objects to ensure that they are not rotated with the building as part of any parametric variations.

Select whether the component block is considered to be part of the building or the site, choosing from:

- **1-Building** - the default option where the solar collector is considered to be part of the building and so is rotated with the building as part of parametric and optimisation studies where the [Building orientation](#) is varied.
- **2-Site** - the solar collector is considered to be part of the fixed wider site, perhaps as part of a ground-mounted array and so is not rotated with the building as part of parametric and optimisation studies.

Material

This material is used only to define the texture to represent the Solar collector in rendered views on the Visualisation screen.

Flat surface position

Although solar panels are represented as special types of 3-D component blocks in DesignBuilder modeller, they are simulated as flat (based on the [Maximum flat surface thickness](#) threshold accessible on the Advanced tab of the Model options dialog). You can control how the single surface used to represent a flat solar panel in simulations is defined. Choose from:

- **1-Upper surface**, where the uppermost of the 2 largest solar panel surfaces is used in simulations.
- **2-Lower surface**, where the lowermost of the 2 largest solar panel surfaces is used in simulations.
- **3-Midway surface**, where a surface midway between the lowermost and uppermost large surfaces is used in simulations.
- **4-All surfaces**, this option should not be selected for solar collectors. It is only made available because the same list is used for component blocks. If this option is selected, DesignBuilder will reset it at the solar collector level to **1-Upper surface** at the point of simulation input generation.

Photovoltaic Options

Performance type

Select from:

- **1-Simple**, and
- **2-Equivalent One-Diode**.

Depending on the selection you will be able to select from either a simple or equivalent one-diode definition of the panel.

Tip: PV panels are included in the electrical generation side of the model by specifying them on the [Generator list](#) tab of the Electric load centre dialog when one of the d.c. [Buss types](#) is selected.

Performance model

Choose the component to be used to define the performance of the collector. When the **1-Simple** performance type is selected, select from the list of previously defined [Simple Photovoltaic components](#), or when **2-Equivalent One-Diode** is selected then choose from one of the [Equivalent One-Diode components](#).

Heat transfer integration mode

The PV model allows for different ways of integrating with other EnergyPlus heat transfer surfaces and models and calculating cell temperature. Choose from:

- **1-Decoupled**, where no adjustments are made to account for energy extracted in the form of electricity and the cell temperature of modules in the array is computed based on an energy balance relative to NOCT conditions. Specifically, the cell temperature of the PV is modelled using the method from Duffie and Beckman (1991) for estimating cell temperature.

$$T_c = T_a + \frac{\left(1 - \frac{\eta_c}{\tau\alpha}\right)}{\left(\frac{G_T \tau\alpha}{U_L}\right)}$$

This equation is based upon the standard NOCT (Nominal Operating Cell Temperature) measurements to compute the module temperature T_c at each timestep. The NOCT temperature ($T_{c,NOCT}$) is the operating temperature of the module with a wind speed of 1 m/s, no electrical load, and a certain specified insolation and ambient temperature. The values for insolation, $G_{T,NOCT}$ and ambient temperature $T_{a,NOCT}$ are usually 800 W/m² and 20° C. η_c is the conversion efficiency of the module, which varies with ambient conditions. $\tau\alpha$ is a user-defined constant.

- **2-Decoupled Ulleberg Dynamic**, where the cell temperature is calculated based on a dynamic model developed by Ulleberg in which the cell temperature is a function of the previous cell temperature and the thermal capacity of the PV module material.

$$T_{cell}|_t = T_{ambient} + T_{cell}|_{t-1} - T_{ambient} * e^{\frac{-U_L \Delta t}{Cap}}$$

Note: The above decoupled PV solar panels shade solar radiation from any other building surfaces below so they can be used to model PV shading, awning and the reduced solar gain on roofs where there is a ventilated cavity below the panel. They will be less accurate from the thermal point of view for building integrated PV panels where there is a significant thermal solar gain through the panel and into the building fabric.

Note: Building Integrated PV (BIPV) panels with closer thermal links with the building fabric can be modelled directly on the [Construction](#) and [Glazing](#) dialogs by selecting the appropriate categories. In this case further **Heat transfer integration modes** are allowed.

Modules in parallel

This field is the number of series-wired strings of PV modules that are in parallel to form the PV array. The product of this field and the next field should equal the total number of modules in the array.

The **Modules in parallel** and **Modules in series** data are only used for **2-Equivalent One-Diode** PV arrays.

Modules in series

This field is the number of modules wired in series (on each string) to form the PV array. The product of this field and the previous field should equal the total number of modules in the array.

For more details on the algorithms used for simulation of PV panels see the [EnergyPlus Engineering Reference](#).

See also:

- "How to Model Photovoltaic (PV) Systems" on page 523

PV Generator - Simple

PV Generator Simple component describes a simple model of PV panels that may be useful for early phase design analysis. It provides direct access to the efficiency with which PV panels convert incident solar radiation to electricity and does not require arrays of specific modules to be defined. The full geometric model for solar radiation is used, including shading and reflections to determine the incident solar on the panel. This model is intended to be useful for design purposes to quickly get an idea of the levels for annual production and peak power. The model can also accept arbitrary conversion efficiencies and does not require actual production units be tested to obtain performance coefficients.

Name

This field is the name of the PV array. The name is only used as an identifier.

Fraction of surface area with active solar cells

This field is the user defined fraction for the area of surface named in the parent PV object that will have active PV cells on it. The area actually covered with solar cells will be the net area of the surface (gross area less any sub-surfaces like windows) times the fraction entered here. This fraction includes the difference between PV module area and active cells within it and any losses for how closely packed modules can be arranged on surface. The value should be between 0.0 and 1.0.

Conversion efficiency input mode

This field is used to determine how the thermal efficiency is input. There are two choices

- **1-Fixed** where a constant value for PV electrical conversion efficiency is used.
- **2-Scheduled** where the PV electrical conversion efficiency values are defined in a schedule.

Value for cell efficiency if fixed

This field specifies the efficiency with which solar incident energy is converted to electricity. Efficiency = (electrical power generated [W])/(power of incident solar[W]). These efficiency values are dimensionless and should be between 0.0 and 1.0.

Efficiency schedule

Select the [schedule](#) which specifies dimensionless efficiency values between 0.0 and 1.0. This could be used for example to vary the efficiency to match results computed with more detailed models.

Rated electric power

This numeric field contains the nominal electric power output to be requested from the panel. It is normally equal to the rated power output of the generator (in W or Btu/hr). This value is used only for supervisory control and generator dispatch; the actual power output for each time step is determined by the generator models. This value affects how much a generator is loaded (i.e., requested electric power output) and can also impact the operation of an [electric storage unit](#) if one is connected to the associated [Electric load centre](#).

Note: The **Rated electric power** is used for sizing the battery storage and inverters and does not define the maximum power output available from the panel.

For more details on this input and its relationship with electric storage refer to [Electrical Storage](#) in the EnergyPlus Engineering Reference.

Availability schedule

Select the generator availability schedule. A schedule value of 0 indicates the generator is not available, while a schedule value > 0 indicates that the generator is available to operate.

PV Solar Collector Performance - Equivalent One Diode

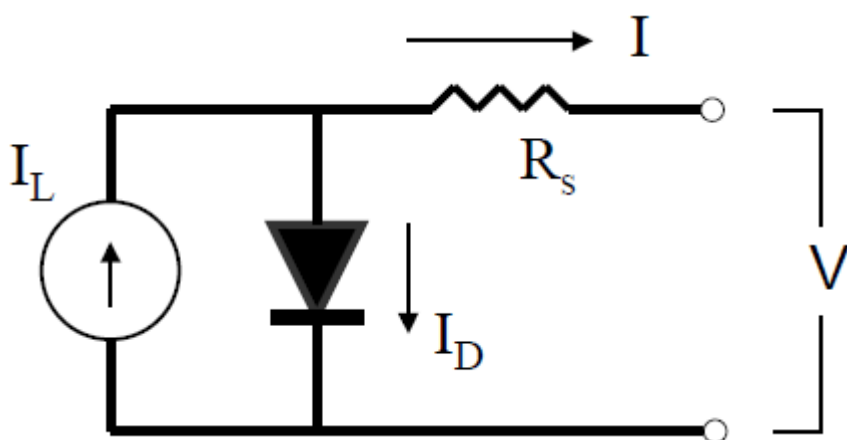
This object describes the performance characteristics of Photovoltaic (PV) modules to be modelled using an equivalent one-diode circuit. This model is also known as the 4- or 5-parameter TRNSYS model for photovoltaics.

Note: The number series and parallel-wired of Equivalent One Diode modules to be included in the model are defined on the Constructions tab under the [Solar Collector](#) header.

Technical

Mathematically speaking, the EnergyPlus PV module employs equations for an empirical equivalent circuit model to predict the current-voltage characteristics of a single module. This circuit consists of a DC current source, diode, and either one or two resistors. The strength of the current source is dependent on solar radiation and the IV characteristics of the diode are temperature-dependent. The results for a single module equivalent circuit are extrapolated to predict the performance of a multi-module array.

The module employs a “four-parameter” equivalent circuit to model crystalline (both mono and poly) PV modules developed at the University of Wisconsin – Madison [2]. The values of these parameters cannot normally be obtained directly from manufacturers’ catalogs. However, the PV module will automatically calculate them from commonly available data. The PV module also includes an optional incidence angle modifier correlation to calculate how the reflectance of the PV module surface varies with the angle of incidence of solar radiation.



Equivalent circuit in the four parameter model

More detailed technical information can be found in the EnergyPlus Engineering Reference.

Name

This field contains the unique name for the photovoltaic module performance data. The name is used as an identifier.

Cell type

This field is used to describe the type of technology used in the PV module. There are two options available:

- **1- Crystalline silicon** and
- **2-Amorphous silicon.**

Cells in series

The number of individual cells wired in series to make up a single module. The typical number for a 12V crystalline silicon PV module is 36.

Active area

This field is the active area of the PV module in (m² or ft²).

Transmittance absorptance product

This field indicates the transmittance-absorptance product at normal incidence angles for the PV modules. If the τ_a product is positive, that value will be used for all angles of incidence. If the value specified is negative, then the magnitude of the given value will be used for normal incidence and the IAM modifier correlation will be used for all other angles.

Semiconductor bandgap

This field is the semiconductor bandgap for the PV material. The bandgap for silicon is 1.12eV (electron volts).

Shunt resistance

This field is the shunt (parallel) resistance (in Ω) in the single diode electrical model of the PV. The shunt resistance is effectively infinite for crystalline silicon based PV modules and is finite for thin film and exotic metal modules.

Reference temperature

This field is the ambient temperature (in K or °F) at reference conditions. The value is usually 298K

Reference Insolation

This field is the radiation level (in W/m² or Btu/h-ft²-°F) at reference conditions. The value is usually 1000 W/m².

Module heat loss coefficient

This field is the heat loss coefficient (in W/m².K or Btu/h-ft²-°F) for the array. The heat loss coefficient is dependent upon measures taken to actively or passively promote airflow over the array surface. The heat loss coefficient value is used only if the Integration and Cell Temperature Mode is set to "Decoupled Ulleberg Dynamic."

Total heat capacity

This field is the heat capacity (in J/m².K or Btu/ft²-F) of the modules in the array. It describes the module's ability to store incident solar radiation internally. Such energy storage is manifested as a temperature increase in the modules that is considered to be undesirable. The total heat capacity value is used only if the Integration and Cell Temperature Mode is set to [2-Decoupled Ulleberg Dynamic](#).

Rated electric power

This numeric field contains the nominal electric power output to be requested from the panel. It is normally equal to the rated power output of the generator (in W). This value is used only for supervisory control and generator dispatch; the actual power output for each time step is determined by the generator models. This value affects how much a generator is loaded (i.e., requested electric power output) and can also impact the operation of an [electric storage unit](#) if one is connected to the associated [Electric load centre](#).

Note: The **Rated electric power** is used for sizing the battery storage and inverters and does not define the maximum power output available from the panel.

For details in this input and its relationship with electric storage refer to [Electrical Storage](#) in the EnergyPlus Engineering Reference.

Availability schedule

Select the generator availability schedule. A schedule value of 0 indicates the generator is not available, while a schedule value > 0 indicates that the generator is available to operate.

Current

Short circuit current

This field is the short circuit current (in A) for an individual module in the PV array at reference conditions.

Module current at maximum power

This field is module current (in A) at the maximum power point and reference conditions.

Temperature coefficient of short circuit current

This field accounts for the fact that the module short circuit current is temperature dependent. The coefficient is given in A/K.

Voltage

Open circuit voltage

This field is the open circuit voltage (in V) for an individual module in the PV array at reference conditions.

Module voltage at maximum power

This field is module voltage (in V) at the maximum power point and reference conditions.

Temperature coefficient of open circuit voltage

This field accounts for the fact that the module open circuit voltage is temperature dependent. The coefficient is given in V/K.

Nominal Operating Cell Temperature

Nominal operating cell temperature test ambient temperature

This field is the ambient temperature (in K or °F) from the Nominal Operating Cell Temperature (NOCT) test. The value is usually 293 K

Nominal operating cell temperature test cell temperature

This field is the cell temperature (in K or °F) from the Nominal Operating Cell Temperature (NOCT) test.

Nominal operating cell temperature test insolation

This field is the insolation level (in W/m² or Btu/h-ft²) from the Nominal Operating Cell Temperature (NOCT) test. The value is usually 800 W/m².

Linear Thermal Bridges At Junctions

[Construction tab in model data under Construction header](#)

You can model linear thermal bridging at junction such as wall to wall corners, wall to floor edges etc using the settings under the **Linear Thermal Bridges at Junctions** header on the Construction tab. These settings are used at zone level and apply to all surface junctions in the zone. Defaults are provided at building and block levels.

The default Psi values in DesignBuilder are the based on BRE IP 1/06 values degraded by the greater of 0.04 W/mK or 50%.

More details on this and how the data relates to the UK NCM can be found in the NCM modelling guide at <https://www.uk-ncm.org.uk/>.

Linear Thermal Bridging at Junctions	
☒ Specify Psi Values	
Psi Values Involving Metal Cladding	
Roof-Wall (W/m-K)	0.420
Wall-Ground floor (W/m-K)	1.730
Wall-Wall (corner) (W/m-K)	0.380
Wall-Floor (not ground floor) (W/m-K)	0.040
Lintel above window or door (W/m-K)	1.910
Sill below window (W/m-K)	1.910
Jamb at window or door (W/m-K)	1.910
Psi Values NOT Involving Metal Cladding	
Roof-Wall (W/m-K)	0.180
Wall-Ground floor (W/m-K)	0.240
Wall-Wall (corner) (W/m-K)	0.140
Wall-Floor (not ground floor) (W/m-K)	0.110
Lintel above window or door (W/m-K)	0.450
Sill below window (W/m-K)	0.080
Jamb at window or door (W/m-K)	0.090

Note: The linear bridging defined by these Psi values is used in Simulations, SBEM and DSM calculations as well as in Heating design and Cooling design calculations.

Specify Psi Values

If you would like to include the effect of thermal bridging in the zone/block/building then check this option and enter the Psi values in the cells below.

Note: Negative psi values can be entered, however if the total zone linear heat conduction rate is calculated as negative it is not included in the model (since EnergyPlus has no concept of negative conduction). In this case a corresponding reduction in surface area must be included manually elsewhere in the zone. The required negative conductance is written out in W/K to the IDF file as a comment.

Reporting

Visualisation

You can obtain a 3D visualisation of the linear thermal bridges applied to the model from the [Model Data tab](#) of Visualisation screen. This enables a visual inspection of the thermal bridges at specific geometric junctions (like wall-to-wall corners) and helps clarify how the various classes of linear bridging are applied to the building geometry.

Heat balance

Note that the bridging component of the heat transfer is not displayed separately in the Heat Balance results displayed on the DesignBuilder results screens when the option is activated. However the total bridging heat transfer for the zone is added into the **Walls** output on the Heating and Cooling design calculations and in Simulation results.

You can obtain a report of the sum of the linear thermal bridging heat conduction for each zone by viewing results using the Results Viewer. To do this look for **Surface Inside Face Conduction Heat Transfer Rate <IDFZoneName> _LINEAR BRIDGE** reports in the Results Viewer report list panel.

EnergyPlus Summary Reports

When checking the **Envelope Summary > Opaque Exterior** tabular data in the [Summary report](#) generated by EnergyPlus, please bear in mind that the small fictitious linear thermal bridging surface areas used in the EnergyPlus linear bridging calculations are included in the reported wall areas. These areas are added to the North-facing surfaces, and the effect on wall areas depends on the Psi values used for the linear thermal bridges.

Psi Values Including Metal Cladding

The Psi values under this header are linear transmittances for junctions where there is metal cladding in a construction on one or more sides of the junction. Metal cladding is important for thermal bridging calculations as junctions involving it tend to create a higher heat flow than those without.

Units are W/m-K or Btu-in/h-ft²-F, i.e. heat flux per length of junction per unit difference in inside to outside temperature.

Definition of Metal cladding

Constructions involving metal cladding are roof or wall systems where metal forms an integral part of the construction, such as metal twin skin systems where the insulation is located between the metal skins and where the metal skins are typically in the range 0.4 mm to 1.2 mm. Cladding with z-spacers would come into this category as would composite metal panel systems.

If the metal is simply used as an external shield against weather, such as a rainscreen, this is not classed, for the purposes of calculations as "metal cladding".

Metal cladding systems are divided into two broad categories, these being:

- built up metal cladding systems involving rail and bracket or z-spacer systems with insulation within the panels
- composite panel metal cladding systems with insulation inside the panels

You can define whether a construction includes metal cladding on the [Constructions](#) dialog.

Junction type

Roof-Wall

Wall-Ground

Wall-External floor

Description

The average Psi value for junctions between external walls and roofs

The average Psi value for junctions between external walls and ground floors

The average Psi value for junctions between external walls and external floors

Wall-Wall (corner)	The average Psi value for external wall corners. Note that the minimum angular threshold to define a wall-wall corner can be defined under the Advanced header .
Wall-Internal Floor	The average Psi value for junctions between external walls and internal floors. See also Linear bridges shared between 2 zones below.
Wall-External Floor	The average Psi value for junctions between external walls and external floors
Lintel above window or door	The linear transmittance created by lintels above windows and doors
Sill below window	The linear transmittance created by sills below windows
Jamb at window or door	The linear transmittance created by window and door jambs

Psi Values NOT Including Metal Cladding

Definitions for Psi values not including metal cladding are as above.

Technical Note - How it Works

DesignBuilder calculates a total linear bridging transmittance for each zone by summing the bridging length for each of the 14 categories and multiplying each by the user-defined Psi value for that category. The total bridging length is calculated based on the zone outer dimensions.

You must check that you have set the Psi values according to the convention set.

The total zone linear bridging transmittance is included in the EnergyPlus model using a single **WallExterior** surface per zone with fixed interior and exterior convection coefficients of 3 and 25 W/m²K for the interior and exterior sides of the surface respectively. Visible, solar and thermal heat transfer is effectively zero through use of zero of absorptance values. The material layer used in the wall has a resistance of 0.6267 m²K/W. Note that the overall resistance of the wall construction (**LinearBridgingConstruction**) including the layer and the film convective resistances is approximately 1 m²K/W, i.e. a conductance of 1 W/m²K.

The area of each **WallExterior** surface is calculated so that its total conductance (W/K) is the same as that of the sum of the zone linear bridges. To achieve that the height of each surface is set to 1m and **the total zone linear bridging conductance is written to the Length field**. This results in the user-specified linear bridging conductance being applied in the simulation at all timesteps.

The linear bridging **WallExterior** surfaces are located below the building to avoid interfering with shading calculations.

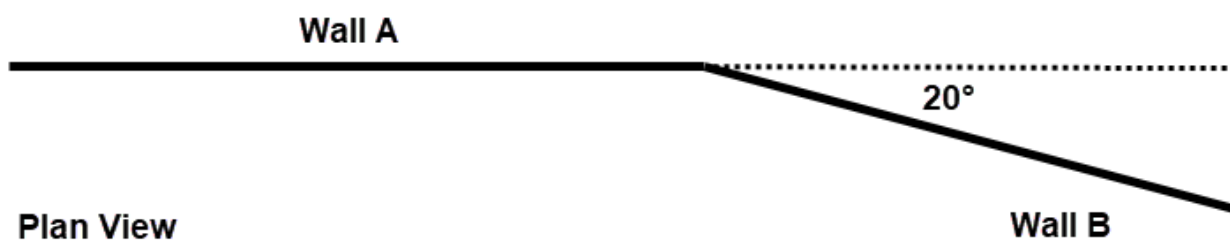
Note: When checking the "Envelope Summary, Opaque Exterior" tabular data in the Summary report generated by EnergyPlus, please bear in mind that these fictitious surface areas are included. This can cause a small inaccuracy in reported wall areas.

Advanced

Angular threshold for Wall-Wall corners

This threshold defines the minimum angular deviation from coplanar that is considered to be a corner in the calculation of Wall-Wall linear thermal bridging.

For example if the threshold is entered as 22.5° (the default) and 2 adjoining external walls meet with a difference of 20° relative to coplanar (see schematic below) then no thermal bridge will be included for this corner. However, if the angular difference between the 2 walls was 30° then a thermal bridge would be included.



Linear bridges shared between 2 zones

In cases where a linear bridge is shared between two zones, for example the bridge between an external wall and an interior floor, any additional thermal conductance caused by such bridges is shared between the two zones involved. Put another way, thermal bridging lengths and Psi values cover both sides, and half the heat transfer is applied on each side. You must take this into account when entering Psi values for such bridges.

Baffle

[Construction tab](#) in model data under [Construction header](#)

You can model a multi-skin exterior heat transfer surface by making settings on the Constructions tab under the **Baffle** header at surface level. Baffle components consist of an outside baffle surface which is slightly detached from the main wall or roof forming a cavity which exchanges air with the outside environment.

Baffles are typically used to model naturally ventilated rainscreens and/or solar shading of exterior surfaces.

The constructions used to define the underlying wall/roof should reflect the construction of just the underlying surface. The data described here is used to describe the decoupled baffle layer and the characteristics of the cavity and openings for natural ventilation.

Tip: If the baffle covers only part of a surface, then surfaces without the baffle can be modelled as sub-surfaces which do not have natural vented cavities.

Tip: You can enter corresponding data for natural vented cavities on the Construction template dialog. If you need to apply settings to a large number of surfaces you may find the fastest way to do this is to use the [Load data from template](#) dialog, selecting the surfaces which have a baffle on the Target tab and selecting the Construction template with the appropriate natural vented cavity and construction settings.

Technical Notes

1. The corresponding EnergyPlus object is **SurfaceProperty:ExteriorNaturalVentedCavity**.
2. The heat capacity of the outer baffle is ignored in the simulation since it is much lower than that of the underlying mass surface.
3. The model involves predicting the rates that ambient air moves in and out of the cavity. Accurate modelling of these air flows would be extremely challenging and so the models provided through this object are simplistic engineering models based on discharge coefficients that are sensitive to wind and buoyancy effects. The accuracy depends on the values for, and applicability of, the discharge coefficients and unfortunately little research is available to help characterize these. The models should be considered rudimentary and the user is encouraged to explore different values for the coefficients in attempts to bound the importance of natural ventilation for the cavities. See the [Engineering Reference](#) for more details.

Area fraction of openings

This field is used to enter an area fraction for what part of the baffle consists of openings. The area of the openings will set to the product of this field and the sum of the area of the underlying surfaces.

Thermal emissivity of exterior baffle material

This field is used to enter the thermal emissivity of the baffle. This surface property is for longwave infrared radiation. The property is used for both sides of collector. Most painted materials have an emissivity of 0.9.

Solar absorptivity of exterior baffle

This field is used to enter the solar absorptivity of the baffle. This surface property is for shortwave, solar radiation. The property is used for the front side of the baffle that faces the environment. Darker colors have a higher absorptivity. While black is the highest performance, other colors might be used to match the colour scheme of the rest of the façade.

Height Scale for buoyancy-driven ventilation

This field is used to enter a nominal height scale (m) for prediction of ventilation induced by buoyancy. This value () is defined as the height from the midpoint of the lower opening to the neutral pressure level. Increasing the value will increase the ventilation rate due to buoyancy.

Effective thickness of cavity

This field is used to enter a nominal gap thickness (m) for the collector. If the baffle is corrugated, use the average depth. This distance value is only used when the collector is near horizontal to determine a length scale in the vertical direction for buoyancy calculations. For example, if the collector is mounted on a flat roof, its tilt-adjusted height is zero and the program will use this gap thickness as a length scale rather than the height from the previous field.

Ratio of actual to projected surface area

This field is used to enter a factor that accounts for the extra surface area resulting from an uneven baffle surface. Corrugations may be present to help stiffen the baffle or ventilated roofing tiles may have more surface area for convection heat transfer than the underlying surface. The projected surface area is obtained by the program from the (flat) underlying surfaces. If the baffle is flat then this ratio is 1.0. If the baffle is corrugated, then this ratio will be greater than one with a typical value might be 1.165.

Roughness of exterior surface

This field is used to describe the relative roughness of the baffle material which influences the exterior convection coefficient. Select from:

- 1-Very rough
- 2-Rough
- 3-Medium rough
- 4-Medium smooth
- 5-Smooth
- 6-Very smooth

Effectiveness for perforations with respect to wind

This field is used to enter a value for the coefficient used to determine natural air exchanges from wind. Wind will cause exterior air to move in and out of the cavity. C_v is an arbitrary coefficient used to model the effectiveness of openings and depends on opening geometry and the orientation with respect to the wind. C_v should probably be in the range 0.05 to 0.65. Increasing C_v will increase the amount of natural ventilation. The following equation shows how C_v is used in the program to predict the volumetric flow rate due to wind:

$$\dot{V}_{\text{wind}} = C_v A_{\text{in}} U_{\infty}$$

Discharge coefficient for openings with respect to buoyancy driven flow

This field is used to enter a value for the coefficient used to determine natural air exchanges from buoyancy. Stack or buoyancy effects will cause exterior air to move in and out of the cavity. C_d is an arbitrary discharge coefficient that depends on the geometry of the opening. C_d should probably be in the range 0.1 to 1.0. Increasing C_d will increase the amount of natural ventilation. The following equations show how C_d is used in the program to predict the volume flow rate due to buoyancy:

$$\dot{V}_{\text{thermal}} = C_D A_{\text{in}} \sqrt{2g \Delta H_{\text{NPL}} (T_{a,\text{cav}} - T_{\text{amb}}) / T_{a,\text{cav}}} \quad (\text{if } T_{a,\text{cav}} > T_{\text{amb}})$$

$$\dot{V}_{\text{thermal}} = C_D A_{\text{in}} \sqrt{2g \Delta H_{\text{NPL}} (T_{\text{amb}} - T_{a,\text{cav}}) / T_{\text{amb}}} \quad (\text{if } T_{\text{amb}} > T_{a,\text{cav}} \text{ and baffle is vertical})$$

where ΔH_{NPL} is the value input into the field above for the height scale for buoyancy-driven ventilation.

Ground Modelling

[Construction tab](#) in model data under [Construction header](#)

Kiva

If the site level [Ground modelling method](#) is set to **4-Kiva Full** then options are made available on the Constructions tab for surfaces in contact with the ground and also at building, block and zone levels for setting defaults for ground-adjacency surfaces.

Kiva adjacency

If you are at surface level and the current surface has an adjacency to a [Kiva foundation](#) then check this checkbox. If you are at building, block or zone level then you can use this checkbox to set the default if the majority of the ground adjacent surfaces in that object have an adjacency with a Kiva foundation.

Tip: If all (or most) ground-adjacent floors and walls in the building are to be linked to Kiva foundations then you should check this option at building level and select the most commonly used Kiva foundation (below).

Kiva foundation

If you are at surface level and the above Kiva adjacency checkbox is checked then select the appropriate [Kiva foundation](#). If you are at building, block or zone level then you can use this checkbox to set the default Kiva foundation.

Ground Domain

If the site level [Ground modelling method](#) is set to **3-Ground domain** then an option will be available on the Constructions tab for surfaces in contact with the ground and also at building, block and zone levels for setting defaults.

Define ground domain

Select the type of [ground domain](#) that the surface is in contact with from the list:

- **1-No ground domain** - the surface is not in contact with a ground domain. Instead the conditions on the other side of the surface are defined using the default ground temperature mechanism using [Ground monthly temperatures](#).
- **2-Slab** - the horizontal surface is in contact with a slab ground domain object.
- **3-Basement** - the underground surface is in contact with a basement ground domain object.

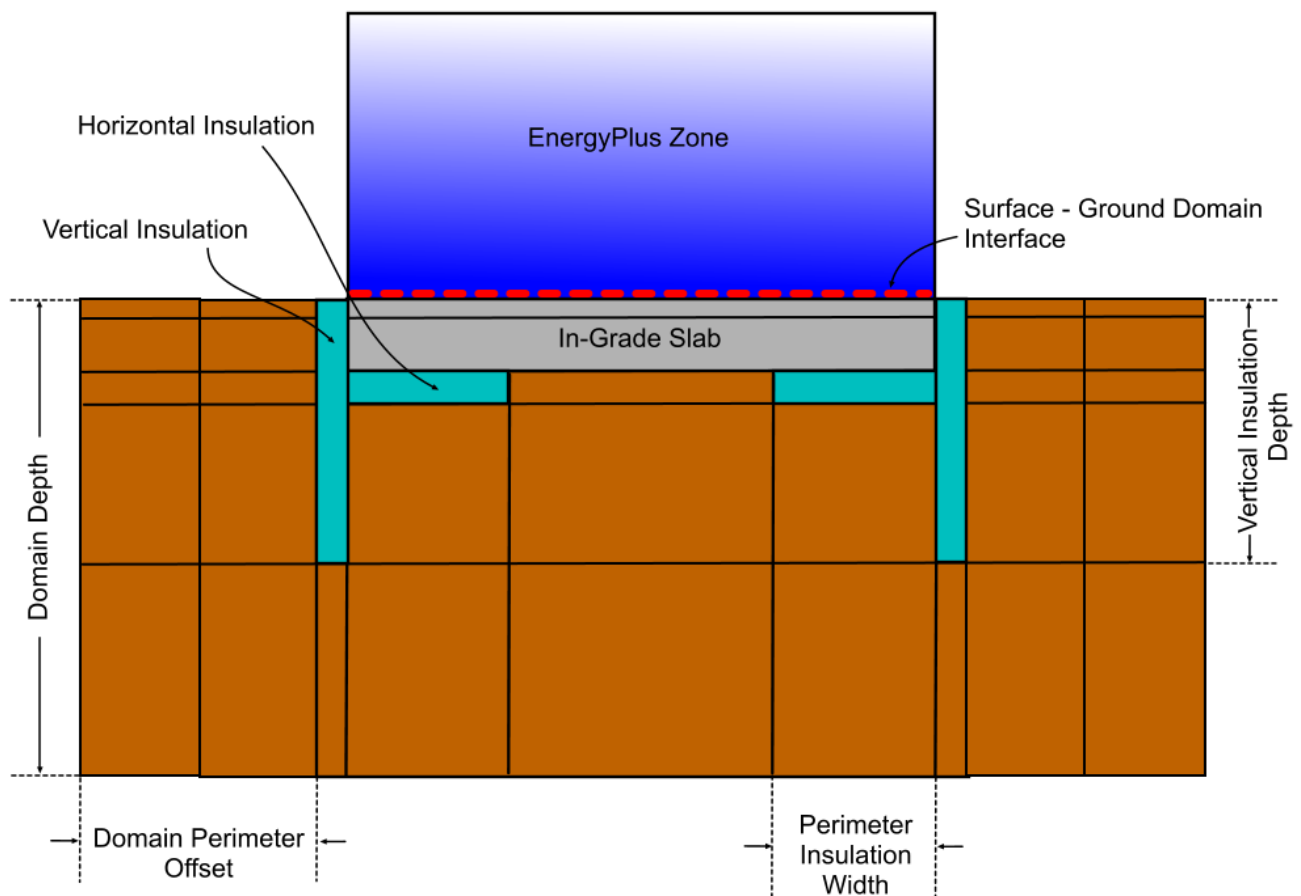
Ground domain

Select the [ground domain](#) in contact with this surface from the drop list of available domains.

Tip: Domain-surface connection settings can be made at any of building, block, zone and surface levels. Setting data at building, block or zone level provides a quick way to associate multiple ground adjacent surfaces to a single ground domain.

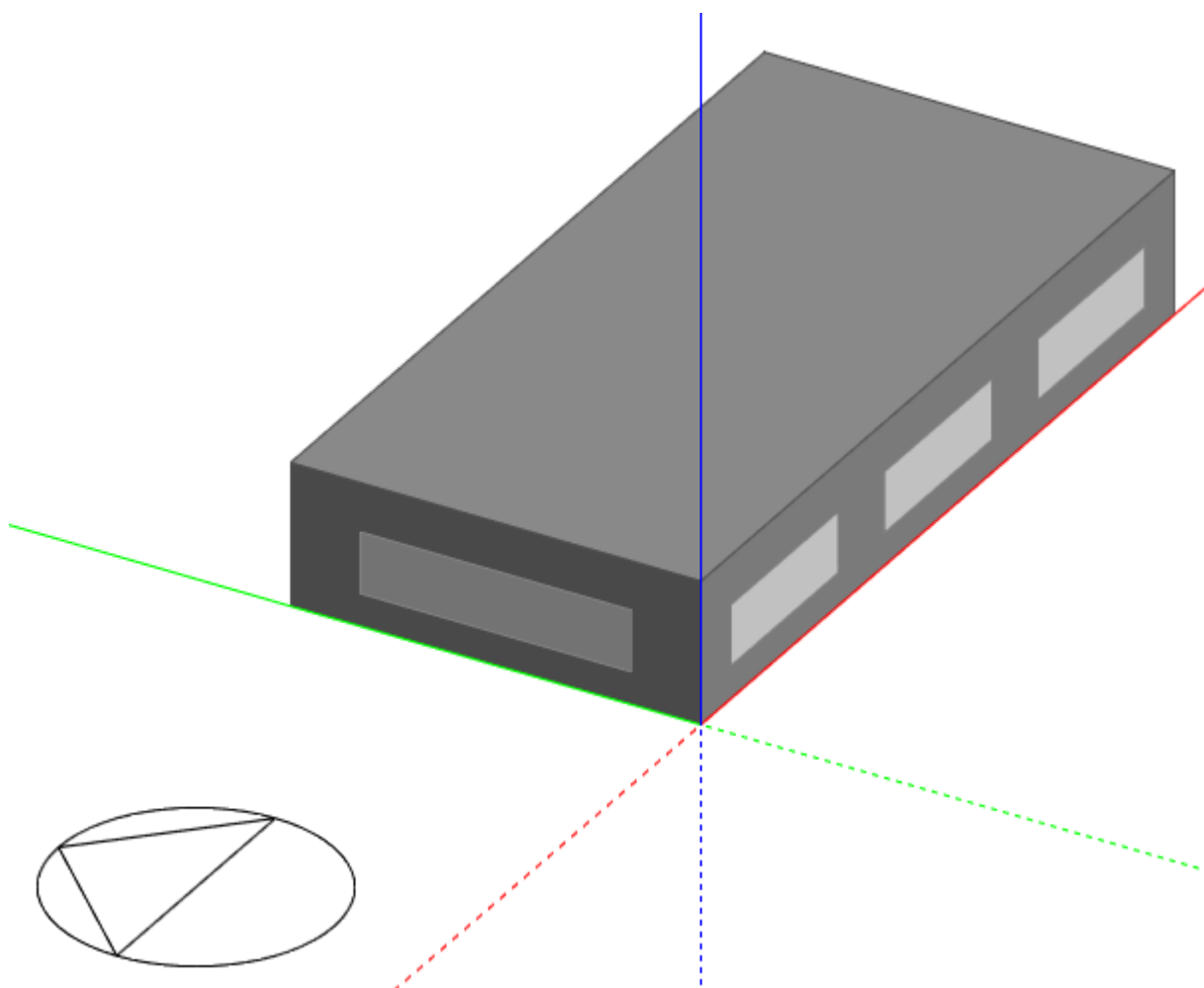
Ground Domain Slab Tutorial

This tutorial takes you through the basic steps of adding a single [ground domain](#) to an existing single zone model. It illustrates the process involved in adding a ground slab with vertical and horizontal insulation. The objective is to model a case like that illustrated schematically below.



Data Entry

1. Create a single zone model 10m x 20m with the longest dimension on the E-W axis as shown below. Use default settings throughout including the default London Gatwick location.



2. The first step in setting up a ground domain is to navigate to site level and on the Location tab, open the [Site Details](#) header and also the Ground and [Ground Modelling](#) headers.

3. Select the **3-Ground domain** [Ground modelling method](#).

4. To model a single ground slab set the **Number of ground domains** to 1 and copy the "Default Slab" Ground domain to create an editable copy for working with. Name it "Project Slab" and select it. Your screen should now look something like the screenshot below.

Location Template		▼
Template		LONDON/GATWICK ARPT
Site Location		>>
Site Details		▼
Elevation above sea level (m)	62,0	
Exposure to wind	2-Normal	▼
Site orientation (°)	0,0	
Site Height Variation		>>
Ground		▼
Surface		>>
Ground Modelling		▼
Ground modelling method	3-Ground domain	▼
Number of ground domains	1	▼
Ground domain 1	Project Slab	

5. Now edit the "Project Slab" Ground domain, making the following changes:

- On the [General](#) tab:

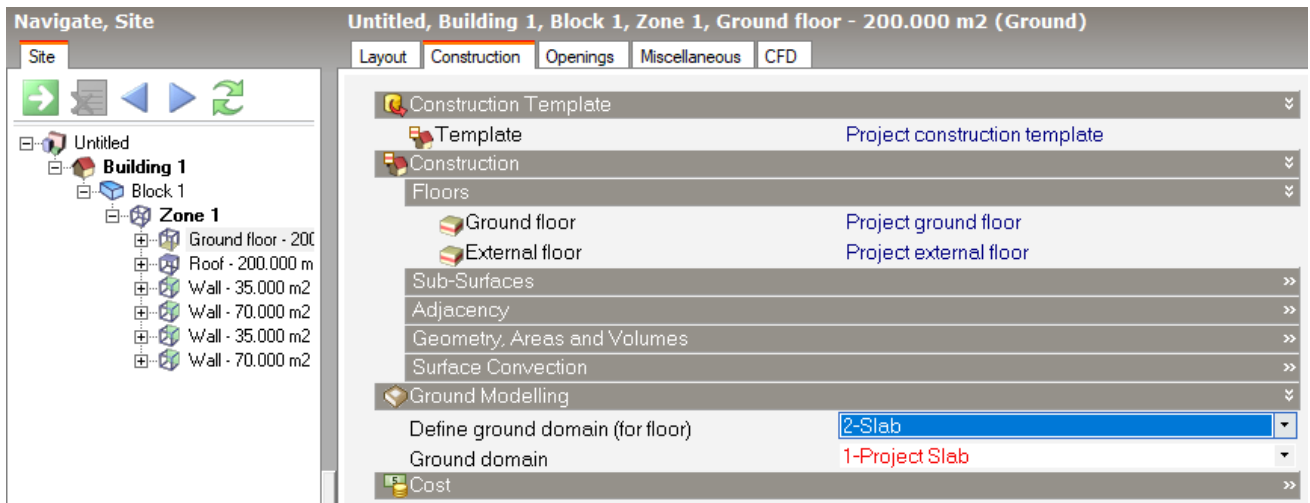
- Set **Slab location** to **2-In grade**.
 - Set **Slab material** to "Cast Concrete (Dense)"
 - Set Slab **Thickness** to 0.1m
 - Set the **Aspect ratio** to 2, calculated as the aspect ratio of the 20:10 length:width aspect ratio of the building perimeter.
 - Set **Perimeter offset** to 5m.
- b. On the [Soil](#) tab, select the Soil material to be "Earth, Common".
 - c. On the [Ground Undisturbed Temperatures](#) tab set the **Ground undisturbed temperature type** to **1-Finite difference**. This allows EnergyPlus to calculate the undisturbed temperature boundary conditions for the Ground domain and no ground temperature data needs to be entered.
 - d. On the [Insulation](#) tab, check the **Include horizontal insulation** checkbox.
 - e. Under that checkbox:
 - Select the horizontal insulation **Material** to be "EPS Expanded Polystyrene (Heavyweight)".
 - Set the **Thickness** to be 0.1m.
 - Set the **Extent** to be **2-Perimeter**, i.e. the slab is only insulated around its perimeter. Note that this setting may not be realistic for a new build in the UK, but it illustrates a powerful capability of the Ground domain slab modelling capability. The effect of using **1-Full** extent is also tested in the next section.
 - Set the **Perimeter insulation width** to 1m.
 - f. Also check the **Include vertical insulation** checkbox, then:
 - Select the vertical insulation **Material** to be "EPS Expanded Polystyrene (Heavyweight)".
 - Set the **Thickness** to be 0.1m.
 - Set the **Depth** to 1m.

Data on the Insulation tab should now look like the screenshot below:

Ground Modelling	
General Soil Ground Undisturbed Temperatures Insulation	
Horizontal Insulation	
☒ Include horizontal insulation	
Material	EPS Expanded Polystyrene (Heavyweight)
Thickness (m)	0.10000
Extent	2-Perimeter
Perimeter insulation width (m)	1.000
Vertical Insulation	
☒ Include vertical insulation	
Material	EPS Expanded Polystyrene (Heavyweight)
Thickness (m)	0.10000
Depth (m)	1.000

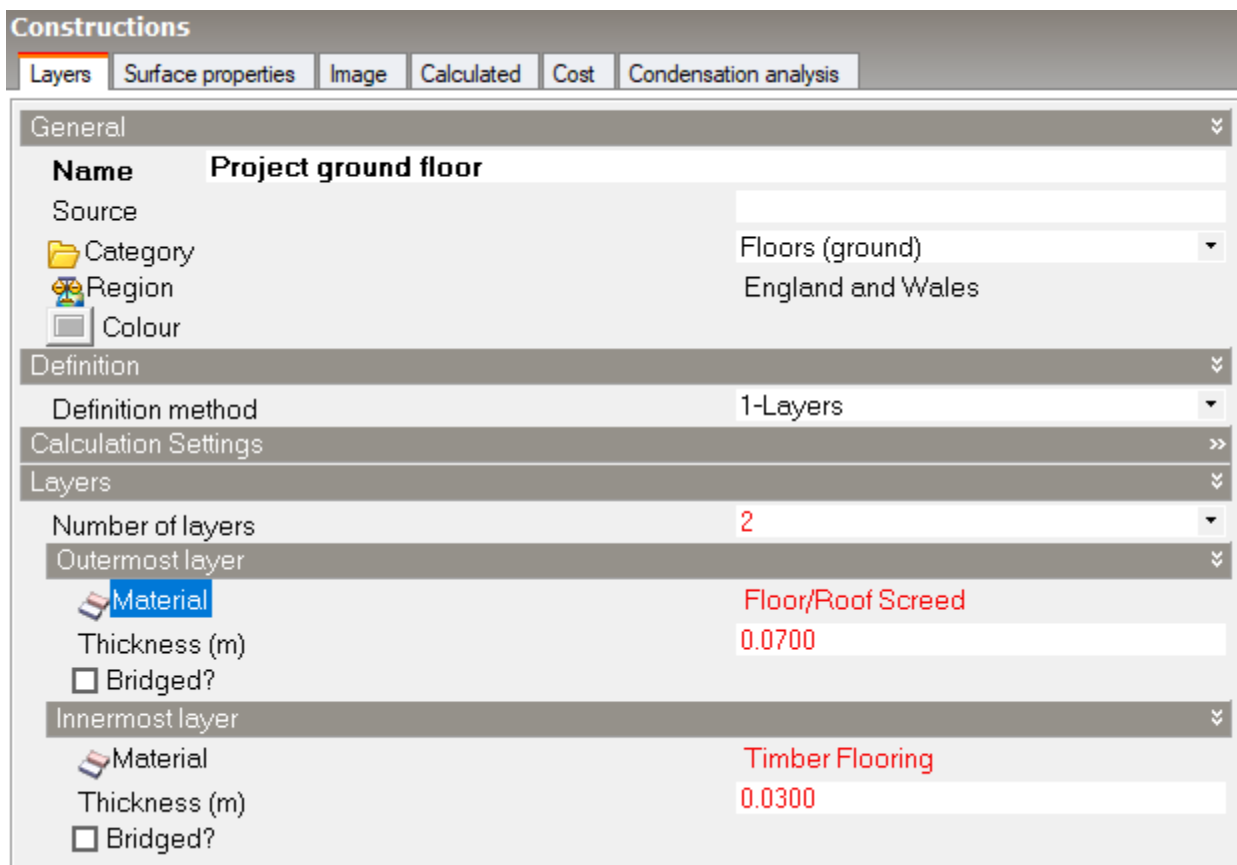
The meaning of the various thicknesses, depths and offsets defined above can be seen in the schematic diagram at the [top](#) of this article.

6. Navigate to the Ground floor surface, open the **Construction** tab and open the **Ground Modelling** header.
7. Click on **Define ground domain (for floor)** and select **2-Slab**.
8. To associate the Project Slab ground domain that you have defined above with the ground floor surface, set the **Ground domain** to **1-Project slab**. The Constructions tab for the ground surface should now look like the screenshot below.



Note: Because this is a single zone model with a single ground floor, you could equally make this setting at building, block or zone level. For a larger model whose ground domain definition is the same for all ground floors you may prefer to make this setting at building level.

9. Because we are using the **2-In-grade** ground domain option, the ground floor construction must not include the slab, insulation or earth, so the default ground floor construction must be updated to reflect this rule. Edit the default "Project ground floor" construction by removing the insulation and cast concrete layers. The construction dialog should look like the screenshot below:



That completes the main part of the model set up and you are now ready to run simulations.

Simulations

We will run 5 annual simulations to help us understand the impact of some key changes to the model. Please read all of this section before running any simulations.

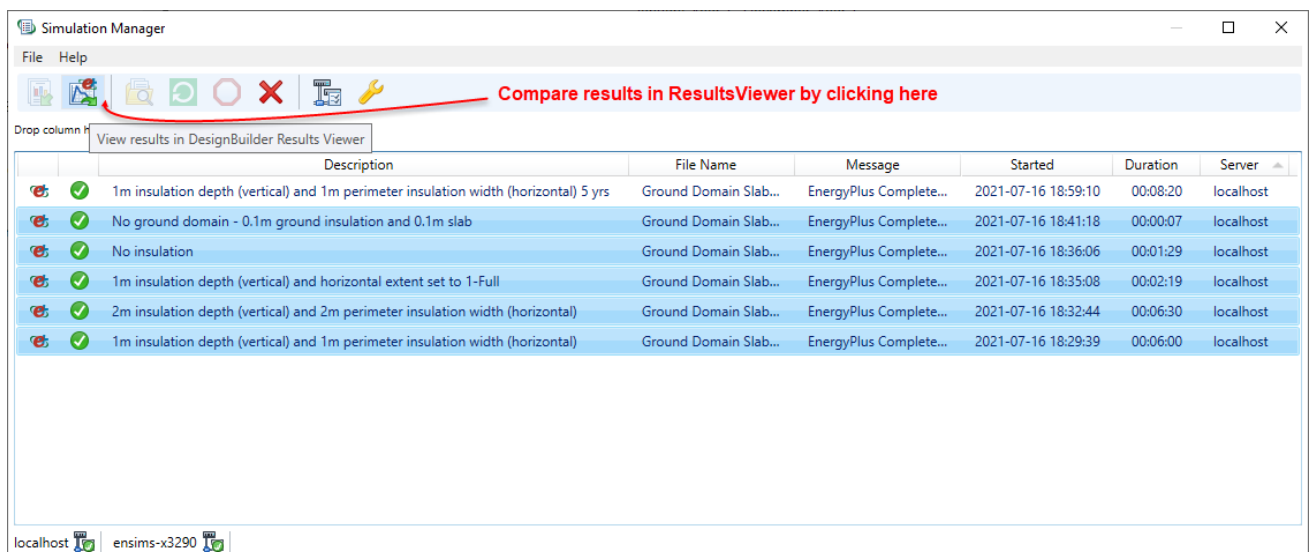
- 1m insulation depth (vertical) and 1m perimeter insulation width (horizontal), the configuration defined above.
- 2m insulation depth (vertical) and 2m perimeter insulation width (horizontal). To set this up open the "Project Slab" Ground domain dialog, click on the Insulation tab and change both the horizontal **Perimeter insulation width** and vertical **Depth** to 2m.
- The same configuration as defined for simulation 1, but with the horizontal insulation extent set to **1-Full**.
- No insulation. To set this up open the "Project Slab" Ground domain dialog, click on the Insulation tab and uncheck the **Include horizontal/vertical insulation** checkboxes.
- No ground domains, reverting to the standard monthly ground temperature approach. To set this up:
 - Select the **1-Standard** Ground modelling method option to simulate the ground floor heat flow in 1-D using an adjacency to the site level [Ground monthly temperatures](#).
 - Add 0.1m of "EPS Expanded Polystyrene (Heavyweight)" insulation and 0.1m "Cast Concrete (Dense)" layers to the ground construction that were removed in step 9 of the Data Entry section above to ensure that we roughly compare like with like.

Ensure that the [Surface heat transfer](#) output option is checked before running any simulations so that ground heat transfer data is generated to help us compare the 5 options.

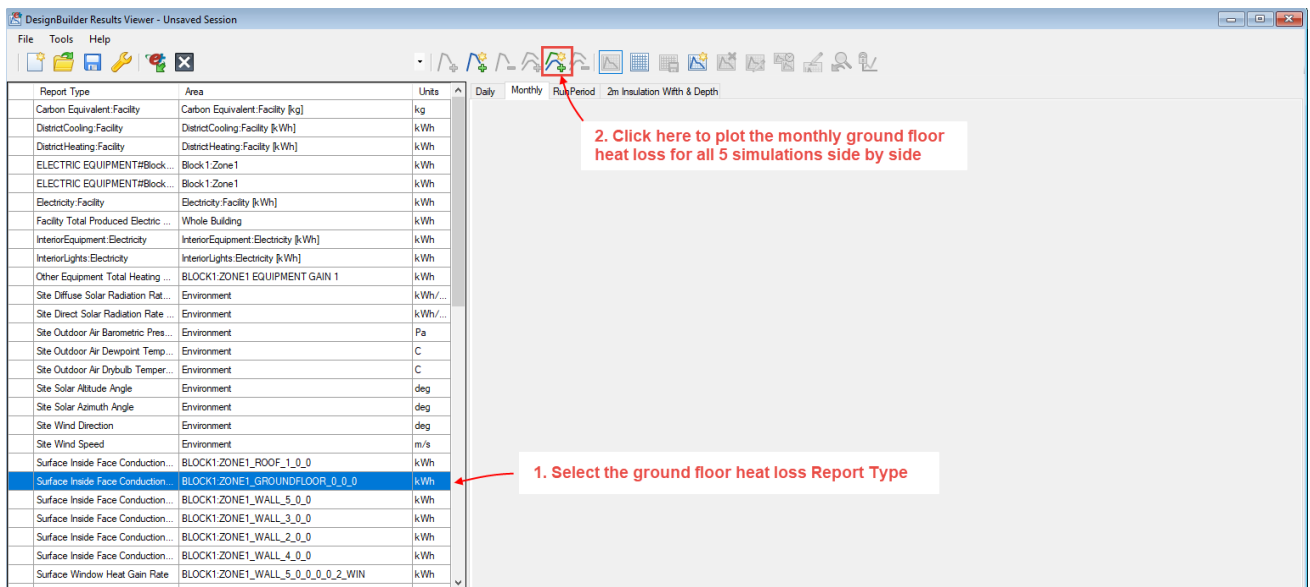
Use the [Simulation Manager](#) to run the simulations and give the simulations appropriate descriptions to ensure that results are labelled correctly when we view them in the ResultsViewer.

Viewing Results

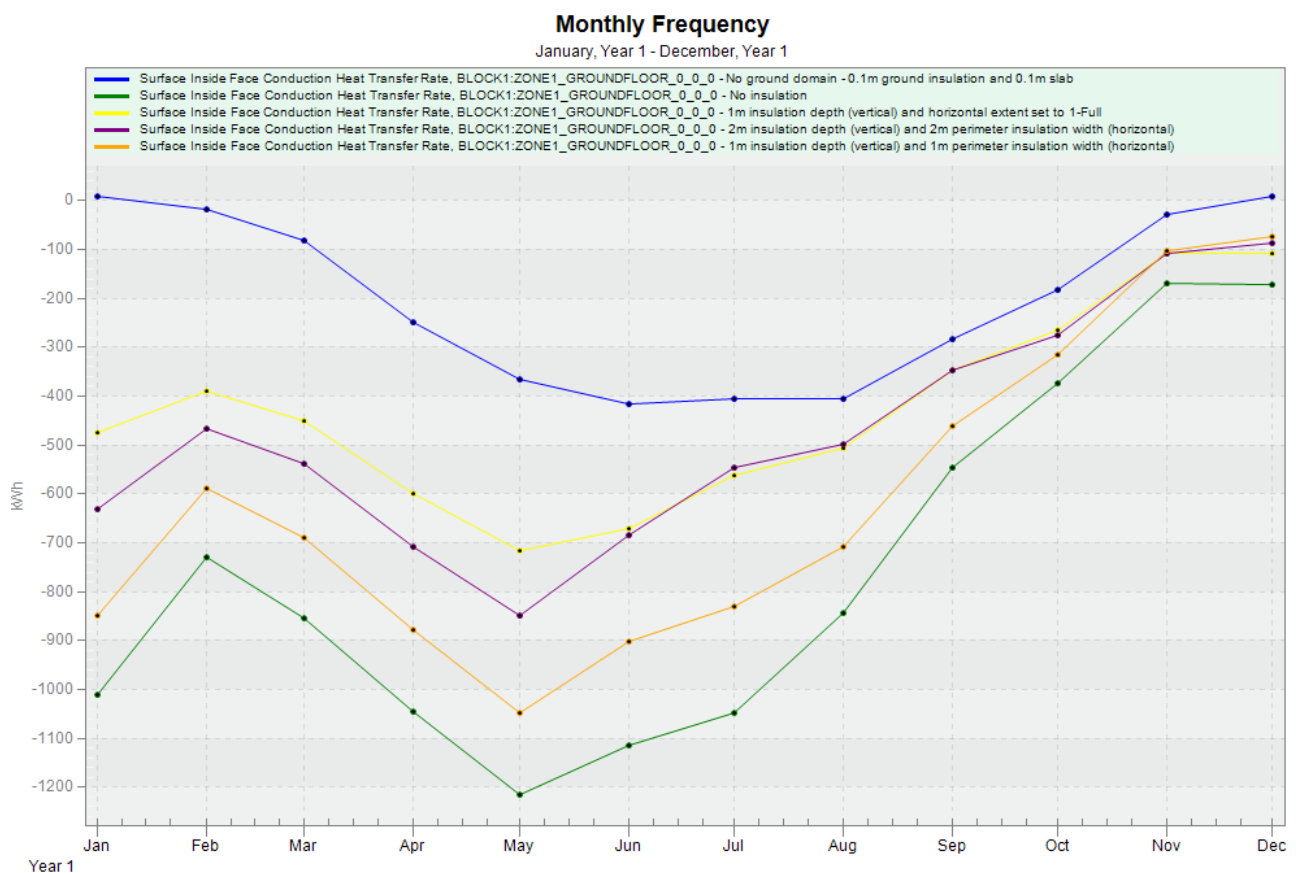
When you have finished running the 5 simulations, open the Simulation Manager and select the 5 simulation result sets. Then click on the toolbar icon to compare the results in ResultsViewer as shown below.



In ResultsViewer, with the results loaded, click on the **Monthly** tab, select the ground floor heat loss Report Type and click on the **Add selected variables to new graph for all datasets** toolbar icon as shown below.



You should see results similar to those below.



Notes on the Results

1. The heat loss results from the case with most extensive insulation: "1m insulation depth (vertical) and horizontal extent set to 1-Full" gives the lowest heat loss over the year (yellow line).
2. Replacing the full horizontal insulation with 2m perimeter insulation gives the next lowest floor heat loss (purple line).
3. Reducing the perimeter insulation further to 1m gives the next lowest heat loss (orange line).
4. Removing all insulation increases heat loss as expected (green line).
5. For the no ground domain case, the results are (perhaps not surprisingly) very different from the other simulations that do use the ground domain. The uninsulated ground floor is directly exposed to the default constant 18°C monthly ground temperatures and heat flows are much lower.

- Results for Jan are *very* different from those for December which suggests that the 1 year simulation we used here was not enough to ensure correct initialisation of the ground domain temperature field. This is addressed in the next section.

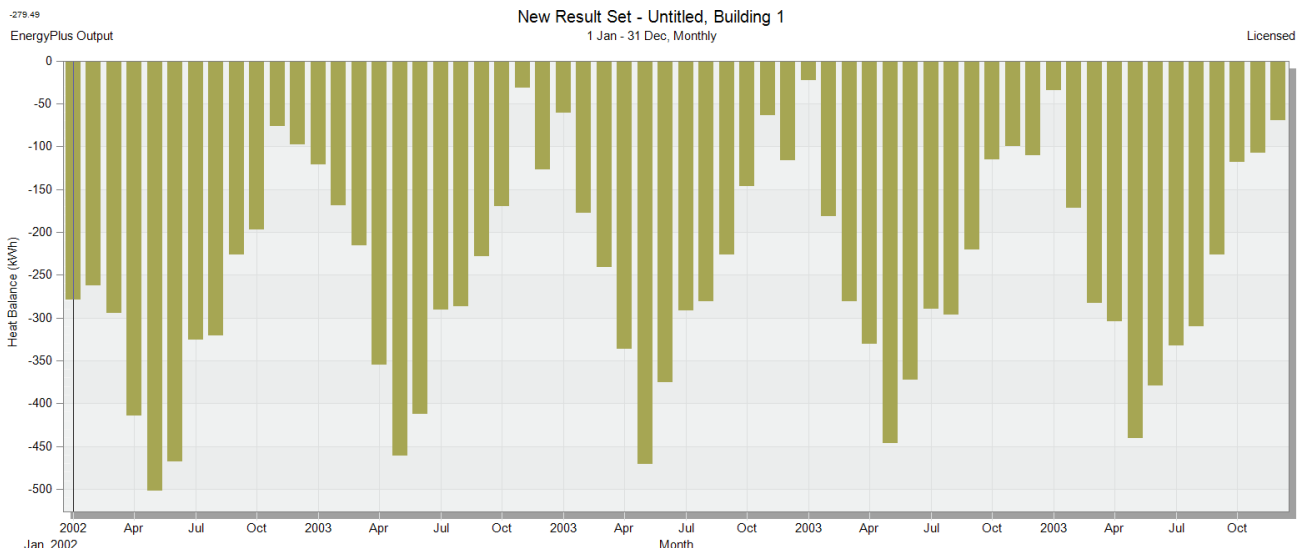
The tutorial demonstrates how spending a few minutes setting up a slab ground domain allows us to model the ground heat loss in a much accurate way than using the basic default 1-D method.

Simulation Using Multiple Years

Ground domains can take a long time to establish a realistic temperature field so, for the best accuracy, you should run simulations for more than one year. DesignBuilder allows any number of years to be simulated by checking the **Run simulation for multiple years** option on the **Simulation calculation options** dialog and entering the number of years.

The next part of the tutorial shows you how to run multi-year simulations and check results. Follow the steps below.

- Ensure that the model is configured per the original settings, case 1 under the [Simulations section](#): 1m insulation depth (vertical) and 1m perimeter insulation width (horizontal), then run a simulation for 5 years, as described above. This takes longer than you might expect for this simple building because the ground domain is being simulated in 2-D over a 5-year period.
- When the simulation has finished load the results into DesignBuilder.
- Navigate to building level, select the **2-Monthly** interval and **5-Fabric and ventilation** data and Show as **1-Graph** options. If you also deselect the other fabric and ventilation data using the checkboxes on the Detailed tab of the Display options panel, you should see an output similar to the one below.



Note: The important learning point from this analysis is that when only running a 1 year ground domain simulation, the ground heat loss in the first month of the simulation is much higher than in subsequent years due to the inadequately formed ground temperature field in the first year. Running simulations for multiple years and only using results for the last year gets around this issue by ensuring a more accurate initialisation of the ground temperature field.

Airtightness

[Construction tab](#) in model data

There are two ways of defining air tightness depending on the Natural ventilation model option:

- [Scheduled](#)
- [Calculated](#)

In both cases infiltration can be switched off at building, block or zone level.

Airtightness (Scheduled Natural Ventilation)

[Construction tab](#) in model data under [Airtightness header](#)

Model infiltration

Infiltration is included by default but you can switch infiltration off by unchecking the **Model infiltration** checkbox.

Infiltration rate

With the Scheduled natural ventilation option set, the design infiltration rate (rate of entry of unintentional air from outside through cracks, holes and through the porosity of the fabric) is assumed to be constant throughout the simulation though it can vary with time through a [schedule](#) and it can also be made to vary according to simple rules based on temperature difference between inside and outside and wind speed.. The design infiltration rate can be set in one of 4 different units depending on the setting of the [Infiltration units](#) model option:

1. **ac/h** - the default option where infiltration is defined in air changes per hour (ac/h), i.e. zone volume per hour under standard operating pressures. An input value is entered equivalent to the actual anticipated air changes due to infiltration during normal operation under typical pressures, not under pressure test conditions. It will operate according to its schedule (below). Note that if you input a value of 0.6 (and assuming a schedule value of 1) then in the outputs you will see 0.6 +/- a small amount as the air temperature (and hence density) varies slightly.
2. **m3/h-m2 at 50 Pa** infiltration is defined as flow rate in m3/h per unit exposed surface area, when the pressure difference between inside and outside is 50 Pa. Unit commonly used in the UK. Note that this is a rate under pressure test conditions and therefore the actual rate under normal operating conditions (at much lower pressures) will be a lot lower.
3. **m3/h-m2 at 4 Pa** infiltration is defined as flow rate in m3/h per unit exposed surface area, when the pressure difference between inside and outside is 4 Pa. Units commonly used in France and Belgium. DesignBuilder calculates the actual resultant value under Standard Pressure of 1atm or 101325 Pa using formulae from EN12831 (section 7.2.2):
4. **n50 - ac/h at 50 Pa** - infiltration is defined in air changes per hour (ac/h), i.e. zone volume per hour, when the pressure difference between inside and outside is 50 Pa. Unit commonly used in Europe. When this option is selected there are options to override the default EN 12831 wind exposure and building height coefficients to account for national standards. See below under [Coefficient Override options for "n50 - ac/h at 50 Pa"](#).

Technical Notes

This note describes the methods used to convert entered infiltration values described above to the m3/s value provided to EnergyPlus. The equation used depends on the units selected as follows.

2-m3/h-m2 at 50 Pa

$ac/h = 2 \text{ (m3/hr-m2 at 50 Pa) (Exposed_Area) (EI) (Epsilon) / (Zone_Volume)}$

3-m3/h-m2 at 4 Pa

$ac/h = 2 \text{ (m3/hr-m2 at 4 Pa) (Exposed_Area) (EI) (Epsilon) / (0.18566355 (Zone_Volume))}$

4-n50 - ac/h at 50 Pa

$ac/h = 2 \text{ (n50 - ac/h at 50 Pa) (EI) (Epsilon)}$

The air flow rate in m³/s at standard pressure provided to EnergyPlus is calculated from the ac/h calculated above using:

$$\text{m}^3/\text{s} = (\text{ac}/\text{h}) (\text{Zone_Volume}) / 3600$$

where **Zone_Volume** is the air volume of the space calculated based on the [Zone volume calculation method](#) selected under "Geometry Areas and Volumes" on page 364

The **Exposed_Area** value is the sum of the gross area of the zone exterior surfaces, excluding any ground and adiabatic surfaces. That includes external walls, flat and pitched roofs, external floors and all external windows. It does not include ground floors, semi-exposed or adiabatic surfaces.

Epsilon is the building height class and accounts for height of building. It is calculated by DesignBuilder based on the building height using EN12831 rules:

epsilon:

- 0-10m = 1
- > 10-30m = 1.2
- > 30m = 1.5

The **EI** wind exposure class [coefficient] is determined based on the façade exposure of each zone and the [Exposure to Wind](#) setting according with the table from EN12831 below.

Exposure to Wind	Exposure description	Heated zone no façade exposed to outside	Heated zone with single façade exposed	Heated zone with multiple façades exposed
3-Exposed	Open site (sea shores, open country...)	0.01	0.03	0.05
2-Normal	Medium sheltered site (towns, city periphery)	0.01	0.02	0.03
1-Sheltered	Well sheltered site (city centre)	0.01	0.01	0.02

Note that the above EN 12831 Epsilon and EI values can be overridden for the **4-n50 - ac/h at 50 Pa** option. [See below.](#)

Coefficient Override options for "n50 - ac/h at 50 Pa"

Override wind exposure coefficient

Some national standards require a fixed value of wind exposure coefficient to be used instead of the default from EN 12831. To override the default, check this checkbox and enter the required coefficient in the text box below.

Wind exposure coefficient

Enter the wind exposure coefficient to override the default value from EN 12831.

Override height coefficient

Some national standards require a fixed value of building height coefficient to be used instead of the default from EN 12831. To override the default, check this checkbox and enter the required coefficient in the text box below.

Height coefficient

Enter the height coefficient to override the default value from EN 12831.

Schedule

The [schedule](#) that modifies the design infiltration rate. The schedule should contain fractions between 0.0 and 1.0 and is used for Cooling design and Simulations (when the Scheduled natural ventilation option is set) but not for Heating design calculations which use a constant design infiltration rate. The exact usage of this schedule (F_{schedule}) is shown in the equation in the section below.

Note: Both the **Infiltration rate** and **Schedule** are only used in Simulations when the Scheduled natural ventilation model option is set. However this data is still used when Calculated natural ventilation is set for Cooling design calculations (Infiltration rate and schedule) and Heating design calculations (Infiltration rate only).

DeltaT And Wind Speed Coefficients

During the simulation, the infiltration flow of outside air into the zone is calculated for each timestep from the equation below:

$$\text{Infiltration} = (I_{\text{design}})(F_{\text{schedule}})[A + B |(T_{\text{zone}} - T_{\text{outside}})| + C (\text{Wind Speed}) + D (\text{Wind Speed})^2]$$

The coefficients A, B, C and D can be defined by entering data for the 4 fields described below.

By default A is 1 and B, C and D are 0 so that the infiltration rate is calculated simply as the entered design infiltration flow rate value multiplied by the schedule value (0 - 1), unmodified by temperature difference or wind effects. However if you need to model the impact of inside to outside temperature difference or wind speed in a simple way then you can enter appropriate values for terms, A-D as described below.

More advanced infiltration calculations are possible using [Calculated natural ventilation](#) through the EnergyPlus AirflowNetwork model for natural infiltration driven by wind when the HVAC system does not operate and/or driven by wind and forced air for times when the HVAC system operates. Exfiltration (the leakage of zone air to the outside) is generally handled better as zone exhaust air in the zone equipment description.

Technical Notes

The question of typical values for these infiltration coefficients is subject to debate. Ideally, one should do a detailed analysis of the infiltration situation and then determine a custom set of coefficients using methods such as those laid out in Chapter 26 of the ASHRAE Handbook of Fundamentals. As previously noted, the defaults are 1,0,0,0 which gives a constant volume flow of infiltration under all conditions.

BLAST (one of the EnergyPlus predecessors) used the following values as defaults: 0.606, 0.03636, 0.1177, 0. These coefficients produce a value of 1.0 at 0°C deltaT and 3.35 m/s (7.5 mph) wind speed, which corresponds to a typical summer condition. At a winter condition of 40°C deltaT and 6 m/s (13.4 mph) wind speed, these coefficients would increase the infiltration rate by a factor of 2.75.

In DOE-2 (the other EnergyPlus predecessor), the air change method defaults are (adjusted to SI units) 0, 0, 0.224 (wind speed), 0. With these coefficients, the summer conditions above would give a factor of 0.75, and the winter conditions would give 1.34. A wind speed of 4.47 m/s (10 mph) gives a factor of 1.0.

The source of the BLAST defaults is noted in the BLAST documentation as:

“Empirical equation and the coefficient default were determined from ASHRAE journal articles and other data on the effects of outdoor weather conditions.”

The source of the DOE-2 defaults is based on examining the infiltration relationships described in the ASHRAE Handbook of Fundamentals.

The EnergyPlus example files use all of the above, the BLAST defaults in some (e.g., GeometryTest), the DOE-2 defaults in some (e.g., 5ZoneAirCooled), and the EnergyPlus defaults in some (e.g., LgOffVAVDetCoil).

Constant

The constant term, A, in the above equation, set to 1 by default.

Temperature

The temperature difference term, B, in the above equation, set to 0 by default.

Velocity

The velocity term, C, in the above equation, set to 0 by default.

Velocity squared

The velocity squared term, D, in the above equation, set to 0 by default.

Airtightness (Calculated Natural Ventilation)

[Construction tab](#) in model data under [Airtightness header](#)

Model infiltration

Infiltration is included by default but you can switch infiltration off by unchecking the **Model infiltration** checkbox. This may be useful if you are carrying out a calculated natural ventilation summertime simulation of a large building to reduce the number of cracks in the model and so speed simulations.

Note: if you switch off infiltration when Calculated natural ventilation is active, you should make sure that the [Natural ventilation](#) option on the HVAC tab is switched on to ensure that sufficient flow paths exist. Otherwise EnergyPlus may generate errors.

Calculated Natural Ventilation

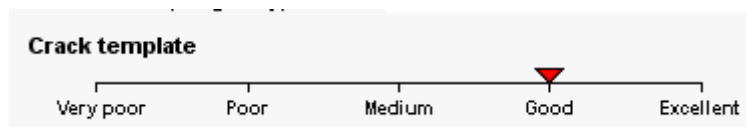
When [Calculated](#) natural ventilation is active, the Airtightness is defined through a crack template. Depending on the [Airtightness method](#) model option there are 2 different ways to define the airtightness of the building:

- **1-Template slider.**
- **2-Crack template.**

These options are used as follows.

Template slider

Airtightness is defined through a five point scale:



The five settings correspond to 5 pre-defined [Crack templates](#).

Crack template

When the **2-Crack template** [Airtightness method](#) is used, airtightness is defined through a specific selection of a template which can be user-defined. With this method it is still possible to select the pre-defined crack templates referred to by the **Template slider** method.

Infiltration airflow calculations

When using Calculated natural ventilation there are 2 main categories of uncontrolled infiltration airflow:

- Airflow through the surface itself which could be caused by cracks or by general fabric porosity.
- Cracks between windows, vents and doors and the main wall or roof surface.

The data that allows DesignBuilder to generate these airflow paths is stored in Crack templates. For example, take a 20m² external wall with a single 2m x 1m window. If the **Good** crack template is selected for the surface then the flow paths used in the simulations will be as follows:

1. Main surface infiltration 0.000040 kg/s.m² @ 1Pa, which for the 18m² (20m² - 2m²) wall = 0.0072 kg/s @ 1Pa.
2. In the **Good** Crack template, cracks between windows and walls have infiltration 0.000060 kg/s.m @ 1Pa. Our window has perimeter 6m so the overall crack property used in the simulation would be 0.00036 kg/s @ 1Pa.

The screenshots below indicate the source of the above Crack template values.

Cracks Data

General | **Openings** | Walls | Floors/Ceilings | Roofs

Windows ▾

External ▾

Flow coefficient (kg/s.m crack @ 1Pa) 0.000060

Flow exponent 0.70

Internal ▾

Flow coefficient (kg/s.m crack @ 1Pa) 0.000600

Flow exponent 0.70

Doors ▾

External ▾

Flow coefficient (kg/s.m crack @ 1Pa) 0.000600

Flow exponent 0.70

Internal ▾

Flow coefficient (kg/s.m crack @ 1Pa) 0.020000

Flow exponent 0.70

Vents ▾

External ▾

Flow coefficient (kg/s.m crack @ 1Pa) 0.004000

Flow exponent 0.66

Internal ▾

Flow coefficient (kg/s.m crack @ 1Pa) 0.003000

Flow exponent 0.60

External window crack data in Crack template dialog

Cracks Data

General Openings **Walls** Floors/Ceilings Roofs

Walls

Internal

Flow coefficient (kg/s.m2 @ 1Pa) 0.002000

Flow exponent 0.75

External

Flow coefficient (kg/s.m2 @ 1Pa) 0.000040

Flow exponent 0.70

External wall crack data in Crack template dialog

Note: As well as being included in standard simulations, infiltration from Calculated natural ventilation is also included in Detailed HVAC autosizing calculations and also in Simple HVAC sizing when using the [2-EnergyPlus Simple HVAC autosizing method](#).

Scheduled Natural Ventilation + Heating And Cooling Design Calculations

Constant rate

Enter the maximum rate of infiltration, which is modified by the multiplier fraction factors in the Schedule select below. The units for the infiltration depend on the [Infiltration units](#) Model data setting. This value and the associated schedule below are used for Cooling design calculations as well as simulations when using Scheduled natural ventilation.

Note: for zones that are below ground level you should set the infiltration rate to zero if there is no external air entering through cracks/porosity. Such zones shouldn't receive any infiltration during the simulation when calculated natural ventilation is set because the "external" surfaces will be adjacent to ground, but this does need to be set manually for heating and cooling design purposes.

Schedule

Select the schedule which defines any time-varying of the infiltration. The default schedule is **On** giving a constant infiltration rate based on the Constant rate entered as described above.

Construction Cost

[Construction tab in model data](#)

Additional cost data can be defined for each zone and surface using the controls under the Cost header on the Constructions model data tab.

Tip: The [Cost data](#) help topic provides an overview of the places in DesignBuilder where costs are entered.

Sub-Structure cost

In general the sub-structure cost is the cost of all works below ground level. The sub-structure cost data entered here is used at zone level and is multiplied by the zone sub-structure surface area, which is calculated as the sum of all ground floor areas in the zone, i.e. the sum of all floors with a ground adjacency.

Note: Additional substructure cost is included in below grade surfaces and their constructions.

Structure

Select the structure type to define the cost of the building structural elements per floor area. This data is used zone by zone when calculating total building structural costs. Structure costs appear in the report, depending on the [cost model](#) used, as follows:

- Basic: Structure costs
- UNIFORMAT II: **B20 Exterior enclosure** Group Element report section
- NRM1: Super Structure

Cost Of Internal Finishes

Walls

Enter the cost of internal finishes for all walls (external and internal partitions) in cost per surface area. This should cover the cost of paint, wallpaper or any other material or process not already covered in the wall construction costs.

Floors

Enter the cost of internal finishes for all floors (external, ground and internal) in cost per surface area. This should cover the cost of paint, polish or any other material or process not already covered in the floor construction costs.

Ceilings

Enter the cost of internal finishes for all ceilings (internal and underside of external roof surfaces) in cost per surface area. This should cover the cost of paint, polish or any other material or process not already covered in the ceiling/roof construction costs.

Openings Model Data (Windows, Doors, Vents, Holes, Sub-surfaces)

[Openings tab in model data](#)

The term **opening** is used in DesignBuilder to describe any opening in the main building fabric. These are modelled as exceptions to the main surface construction. There are five types of opening:

- Windows
- Sub-surfaces (i.e. opaque elements within the surface that have different properties from the main construction. For example lintels and lightweight panels can be defined as sub-surfaces. Data for these is held on the [Construction tab](#))
- Holes
- Doors
- Vents

The layout, operation and construction data for these (apart from sub-surfaces) can be set on the Openings tab.

See also the [Glazing and openings](#) and [Drawing windows and doors](#) tutorials

Templates

You can load generic Opening data by clicking on the Template option under Glazing Template and then by clicking on the '...' at the right of the line (below).



When you make a selection from the list of [Glazing templates](#), data from selected template will be loaded to the model.

You can also open the:

- [External Glazing](#)
- [Internal Glazing](#)
- [Roof Glazing](#)
- [Doors](#)
- [Vents](#)

headers to enter opening details where they are different from those loaded from the template.

Note: Once you have created custom openings on any particular surface, changes made to facade layout on this tab will not affect the layout for the surface.

All Openings data is used to generate simulation data at the **Surface level**.

You can also override default [Pressure Coefficient](#) data when the Calculated Natural ventilation option is set and you are at surface level.

External Glazing

[Openings tab](#) in model data

The data under the External Glazing header applies to all glazing lying on external walls. You can enter information on:

- [Dimensions](#)
- [Frame](#)
- [Shading](#)
- [Operation](#)

Dimensions (Walls)

[Openings tab](#) in model data under External Glazing and Internal Glazing headers

You can use settings such as **Window to wall %** (WWR) and **Window height**, [Window spacing](#) and [Sill height](#) to easily change the amount of glazing and its layout on the walls. The way this data is used for each façade type is described below.

Facade types

There are a number of standard façade types:

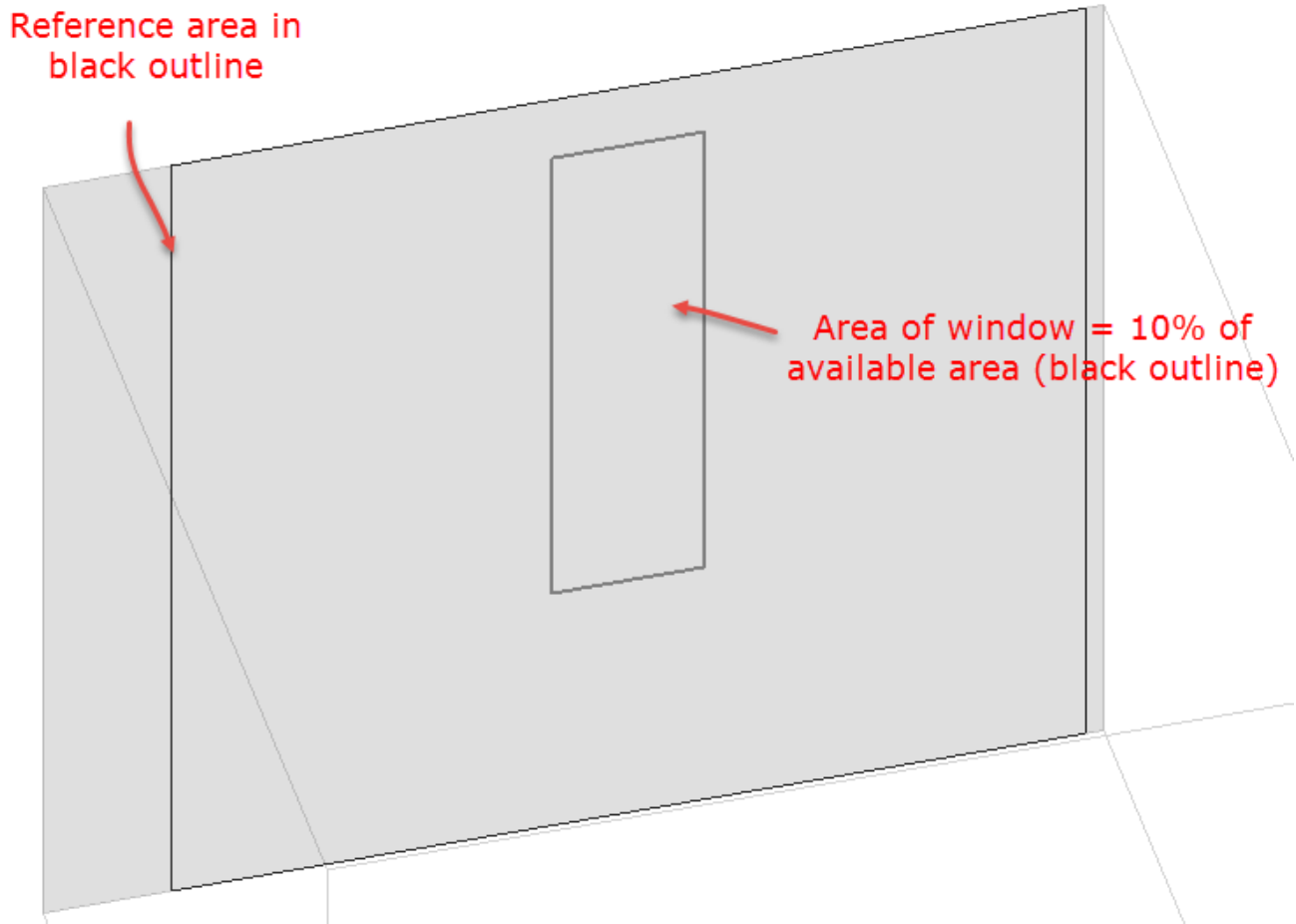
- **0-None** - there is no glazing.
- **1-Continuous horizontal** - glazing is generated in a continuous horizontal strip using Sill height, Window spacing and Percentage glazing.
- **2-Fixed height** - glazing is generated with a fixed height at the specified Sill height and Window spacing, and width is calculated based on the Percentage glazing. This option uses **Window to wall %** but prioritises window height.
- **3-Preferred height** - glazing is generated using the window height data and the Percentage glazing, but the Window height may be adjusted to achieve the required **Window to wall %**. This option uses Window height, Window spacing and Sill height but prioritises **Window to wall %**.
- **4-Fixed width and height** - windows have fixed width and height.
- **5-Fill surface (100%)** - the entire surface is filled with glazing and there is **no frame**.

Window to Wall %

Often referred to as the Window to Wall Ratio (WWR) this setting is a fundamental parameter in DesignBuilder used to control the proportion of glazing on building façades. It can be used to change the amount of glazing for all openings at or below the current level in the building hierarchy. For example changing the WWR at building level changes the size of all windows in the model not overridden by other settings lower in the hierarchy.

When using the **2-Outer volume** [Zone geometry and surface areas](#) option (typically one of the "External measurements" convention templates) it is important to understand that the reference surface area used when calculating the window area from the % value supplied is based on the [Window to wall ratio method](#) which is accessed from the Constructions tab under the "Geometry Areas and Volumes" on page 364header.

The image below illustrates a wall in a model using the "External measurements" geometry convention template where a window has been created using the **3-Preferred height** façade option with a 10% glazing ratio. You can see the black outline showing the available area where the window could possibly be sited. Areas outside this outline to the left and right are taken up by corners with adjacent walls. The % glazing calculation uses the black outline area as the reference area. In an extreme case where 100% glazing is specified the whole of this area would be filled with window, but of course the sides of the surface where there is a corner with an adjacent surface cannot contain a window and so do not take part in the calculation - even though they will form part of the surface used in the simulation.



Tip: If you need to ensure that the % window areas used in the simulation exactly match the % you define here then you should consider using the [Simple Geometry convention template](#) which ensures that 100% of each surface is available for containing windows by using zero surface thickness for all surfaces.

Note: If any custom windows are drawn on a surface, the automatic (aka default) façade mechanism described on this page using Window to Wall % (WWR), window width, height, spacing etc is not applied any more. The façade mechanism can be reinstated using the [Clear data to default](#) option. For more information see expanding text below.

Window to Wall %, Custom Facades and Parametric Tools

The behaviour of the WWR % setting changes significantly when custom windows are introduced. This section explains how WWR % operates in DesignBuilder and its interaction with custom windows, especially in the context of using parametric tools.

How WWR % Functions in DesignBuilder

- **Default Façade Mechanism**
WWR % automatically sizes and places windows on building façades based on the specified ratio. This setting can be applied at building, block, zone, and surface levels.
- **Parametric Tools:**
WWR % is commonly used as a variable in parametric studies to evaluate the impact of different glazing ratios on building performance.

Impact of Using Custom Windows

- **Override of Default Mechanism:**
When you manually draw or size windows (custom windows) on a surface, the automatic façade mechanism—including WWR %, window width, height, and spacing—is disabled for that surface.
- **WWR % Not Applicable:**
For surfaces with custom windows, changes to the WWR % setting will not affect those windows. Only the manually defined windows are considered in calculations.
- **Restoring WWR % Control:**
To re-enable WWR % control, use the “Clear data to default” option. This removes custom windows and reinstates the automatic façade mechanism.

Practical Implications for Parametric Calculations

- **If your parametric study relies on varying WWR %:**
 - WWR % will only affect surfaces without custom windows.
 - Custom windows remain fixed in size and position, regardless of WWR % changes during parametric runs.
- **To ensure WWR % is applied throughout your model:**
 - Avoid drawing custom windows, or clear them before running your parametric analysis.
 - Use the default façade settings to allow WWR % to control window area and placement.

Window width

The width of the windows including any frame (in m or ft).

Window height

The height of the windows including any frame (in m or ft).

Window spacing

The spacing between the each window on the façade (in m or ft). The window spacing is the centre to centre spacing between windows, not the gap between windows.

Sill height

This is the height of the base of the window from the base of the block (in m or ft).

Note 1: You can control the configuration of the building façades in greater detail by [drawing individual "custom" windows](#) at the surface level or by copying previously drawn windows at building level. In either of these cases glazing, vent and door façade layout model data is ignored.

Note 2: Flat roofs do not have default glazing set up using Roof glazing layout model data. To create openings on a flat roof you should go to the surface level and add them there.

Note 3: When the surface is **non-rectangular**, even with the 2 façade type options that prioritise Window to wall % (**3-Preferred height** and **1-Continuous horizontal**), the **Window to wall %** will not be perfectly maintained. For these surfaces, DesignBuilder calculates a rectangle that can accept the windows and starts from one end of the rectangle adding windows using the Window spacing data until no more fit into the rectangle. This results in an approximately correct looking façade but only prioritises **Window to wall %** in the rectangle, not accounting for the areas outside the bounding rectangle. For relatively large values of **Window spacing** on non-rectangular surfaces, no glazing may be generated. For example you may find that gable window surfaces have no default windows generated and it is often easiest to [draw a window](#) in this case.

Outside reveal depth

The **outside reveal depth** gives the extent to which the glazing is offset into the wall. As shown in the [diagram on the Frame and Dividers page](#) it is the distance from the exterior surface of the wall to the glazing outer surface (in m or ft).

Note: The **outside reveal depth** is used in EnergyPlus, Visualisations and Daylighting calculations and (unlike [Inside reveal depth](#)) is available regardless of whether the frame and dividers are included.

Note: EnergyPlus models outside reveals through the co-ordinates of the window, which are offset into the zone so the window and the base surface don't lie on the same plane. DesignBuilder provides window co-ordinates to EnergyPlus in this way and there is no specific data item in the IDF data to define it.

See also [Inside reveal depth](#).

EnergyPlus Window Dimensions - Technical Information

The **FenestrationSurface:Detailed** window data written to the EnergyPlus IDF input file will usually have a smaller area compared with the gross window area defined in DesignBuilder. EnergyPlus window sizes and positions are defined for the glazing area and the frame is extended from the window position to the parent wall/roof surface. DesignBuilder therefore accounts for the [frame](#) which is included in the window area defined in DesignBuilder but excluded from the EnergyPlus IDF window area. In cases where the frame option is switched off in DesignBuilder the frame area in the IDF data will exactly match the area set in DesignBuilder.

See also:

- "Glazing Assemblies" on page 418
- [Surface Polygons Generated for EnergyPlus - Technical Description](#)

Frame And Dividers

[Openings tab](#) in model data under **External Glazing, Internal Glazing and Roof Glazing** headers

If window frames and dividers are being modelled separately from the glazed part then the properties of the frame and dividers can be entered under the Frame and Dividers header on the Openings tab. A discussion on the methods that can be used for modelling windows and frames in "Glazing Assemblies" on page 418.

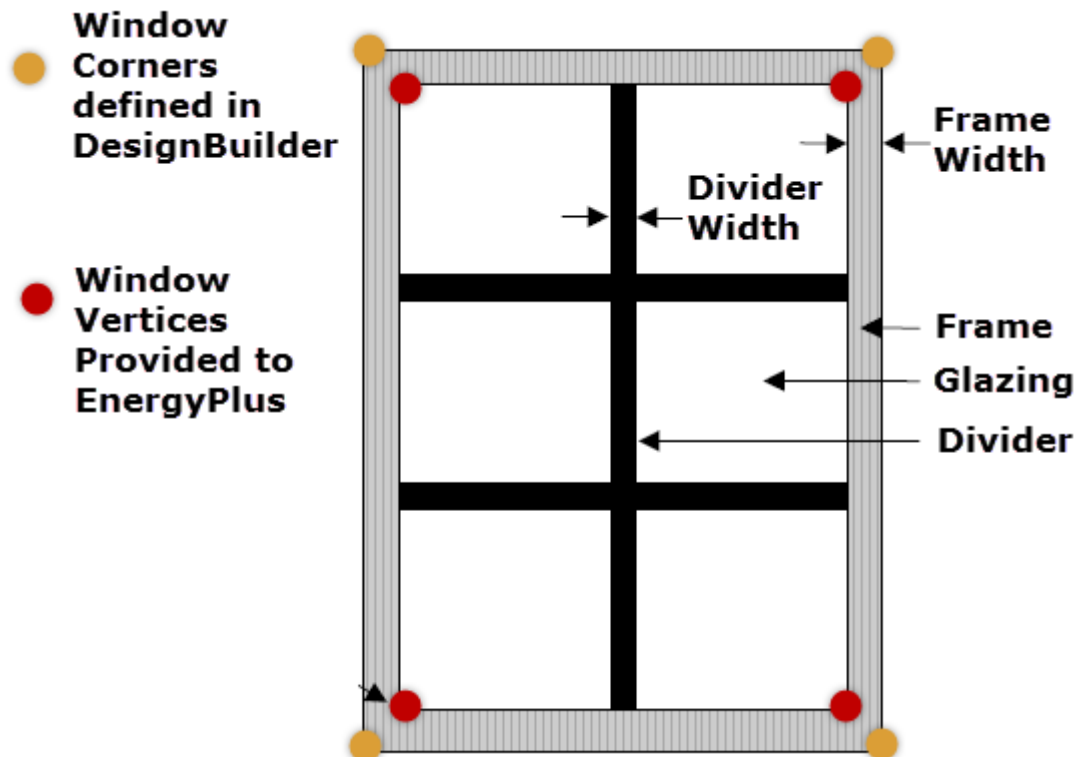
Important Note: Frames and dividers are only applied to *rectangular* windows (default and custom) and not to other shaped windows. They are also not applied when using the **Fill surface (100%)** facade type. If you are unsure about whether frames and dividers will be applied to a window you can use the [Visualisation rendered view](#) to check.

A frame surrounds the glazing in a rectangular window (schematics below). It is assumed that all frame characteristics, such as width, conductance and solar absorptance, are the same for the top, bottom and side elements of the frame. If the frame elements are not the same then you should enter area-weighted average values for the frame characteristics.

Has a Frame/Dividers

Check the **Has a Frame/Dividers** checkbox to enter details of the frame and/or dividers. In this case you can enter:

- The construction of the frame and dividers,
- The number of horizontal dividers and vertical dividers,
- Frame width and divider width (see diagrams below for definition).
- Inside reveal and sill details



[EnergyPlus window dimensions - technical information](#)

For **external windows on walls** the following dimensions in the direction of the glazing normal can also be entered:

- [Outside reveal depth](#)
- Inside reveal depth
- Inside sill depth
- Outside divider projection
- Inside divider projection
- Outside frame projection
- Inside frame projection
- Divider glass edge-centre conduction ratio
- Frame glass edge-centre conduction ratio
- Divider type

The definition of the above dimensions is shown in the diagrams.

Note: If the glazing does not have a frame or dividers or if the whole [glazing assembly](#) properties are being modelled directly using the **2-Simple** glazing [definition method](#) then you should uncheck the **Has a frame/dividers** box.

Reveal

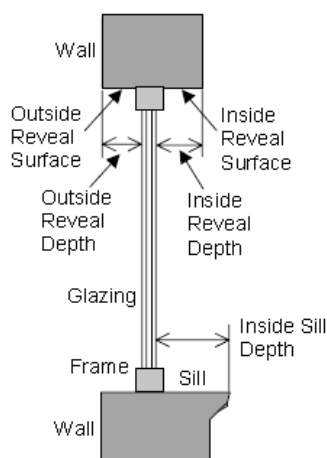
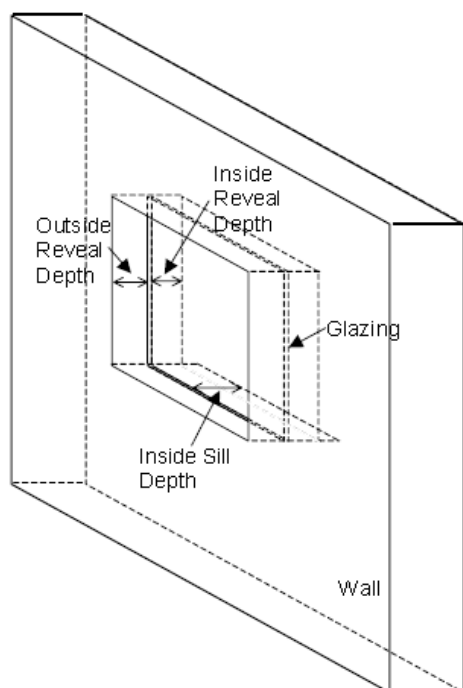
Reveal surfaces are associated with the setback of the glazing from the outside/inside surfaces of the wall they are located in. If the depth and solar absorptance of these surfaces are specified, the program will calculate the reflection of beam solar radiation from these surfaces. The program also calculates the shadowing (onto the window) of beam and diffuse solar radiation by outside reveal surfaces.

The following fields specify the properties of the window reveal surfaces (reveals occur when the window is not in the same plane as the base surface). From this information and from the geometry of the window and the sun position, the program calculates beam solar radiation absorbed and reflected by the top, bottom, right and left sides of outside and inside window reveal surfaces. In doing this calculation, the shadowing on a reveal surface by other reveal surfaces is determined using the orientation of the reveal surfaces and the sun position.

It is assumed that:

- The window is an exterior window (EnergyPlus does not consider reveals for interior windows).
- The reveal surfaces are perpendicular to the window plane.
- If an exterior shade, screen or blind is in place it shades exterior and interior reveal surfaces so that in this case there is no beam solar on these surfaces.
- If an interior shade or blind is in place it shades the interior reveal surfaces so that in this case there is no beam solar on these surfaces.
- The possible shadowing on inside reveal surfaces by a window divider is ignored.
- The outside reveal surfaces (top, bottom, left, right) have the same solar absorptance and depth. This depth is not input here but is automatically determined by the program—from window and wall vertices—as the distance between the plane of the outside face of the glazing and plane of the outside face of the parent wall.
- The inside reveal surfaces are divided into two categories: (1) the bottom reveal surface, called here the "inside sill;" and (2) the other reveal surfaces (left, right and top).
- The left, right and top inside reveal surfaces have the same depth and solar absorptance. The inside sill is allowed to have depth and solar absorptance values that are different from the corresponding values for the other inside reveal surfaces.
- The inside sill depth is required to be greater than or equal to the depth of the other inside reveal surfaces. If the inside sill depth is greater than zero the depth of the other inside reveal surfaces is required to be greater than zero.
- The reflection of beam solar radiation from all reveal surfaces is assumed to be isotropic diffuse; there is no specular component.
- Half of the beam solar reflected from outside reveal surfaces is goes towards the window; the other half goes back to the exterior environment (i.e., reflection of this outward-going component from other outside reveal surfaces is not considered).
- The half that goes towards the window is added to the other solar radiation incident on the window. Correspondingly, half of the beam solar reflected from inside reveal surfaces goes towards the window, with the other half going into the zone. The portion going towards the window that is not reflected is absorbed in the glazing or is transmitted back out into the exterior environment.
- The beam solar that is absorbed by outside reveal surfaces is added to the solar absorbed by the outside surface of the window's parent wall; similarly, the beam solar absorbed by the inside reveal surfaces is added to the solar absorbed by the inside surface of the parent wall.
- The net effect of beam solar reflected from outside reveal surfaces is to increase the heat gain to the zone, whereas the effect of beam solar reflected from inside reveal surfaces is to decrease the heat gain to the zone since part of this reflected solar is transmitted back out the window.
- If the window has a frame, the absorption of reflected beam solar by the inside and outside surfaces of the frame is considered. The shadowing of the frame onto interior reveal surfaces is also considered.

The schematics below show how DesignBuilder reveal data is used to describe real glazing systems.



FAQ: Why does DesignBuilder require both Inside and Outside reveal depth data when wall thickness is available to calculate one reveal depth from the other?

Although it is possible for inside reveal depth to be calculated from the outside reveal depth and the [wall thickness](#), DesignBuilder allows both Inside and Outside reveal depths to be defined giving greater flexibility for situations where it is not possible to enter the wall thickness as accurately as one might like. For example the wall thickness may be entered as zero for simplified modelling. In such cases having direct control over the reveal depth data is an advantage.

Inside reveal depth

The **inside reveal depth** is the distance (in m or ft) from the innermost surface of the window to the interior wall surface as shown in the diagram above. This setting is used for EnergyPlus simulations but not for Visualisations and Radiance which use the [Outside reveal depth](#) to define the position of the window within the wall.

Note: **Inside reveal depth** data is provided to EnergyPlus in the same object as the frame and divider data, so when the [Has a frame/dividers](#) option is unchecked this data is not provided to EnergyPlus and the equivalent of a 0 inside reveal depth will be simulated.

See also [Outside reveal depth](#).

Inside sill depth

The **inside sill depth** is the distance (in m or ft) from the innermost surface of the window to the innermost end of the sill as shown in the diagram above.

Note: **Inside sill depth** data is provided to EnergyPlus in the same object as the frame and divider data, so when the [Has frame/dividers](#) option is unchecked this data is not provided to EnergyPlus and the equivalent of a 0 inside sill depth will be simulated.

Frame

Construction

Select the construction used to define the thermal properties of both the frame and dividers. In particular the frame construction is used to derive these EnergyPlus frame-related fields:

Frame conductance

The effective thermal conductance of the frame (in W/m²-K or Btu/h-ft²-F) measured from inside to outside frame surface *excluding surface air films* and taking frame and window geometry as associated 2-D conduction effects into account. Ideally the frame conductance should be obtained from THERM or some other 2-D calculation. DesignBuilder calculates the surface to surface conductance simply based on the material layers in the Frame construction. If you need to include particular 2-D conduction effects for the frame conductance then you should adjust the thickness or material properties of the layers of the frame construction.

Note: For metal frame windows DesignBuilder will calculate a very high surface to surface U-value for the frame construction, but the value that will be provided to EnergyPlus is limited to an maximum value of 500 W/m²k. This is in line with the limits used in the LBNL WINDOW program. This will have negligible difference on simulation as the corresponding thermal resistance is effectively zero in both cases (relative to the inside and outside film resistances).

Frame solar absorptance

The solar absorptance of the frame is derived from the surface properties of the *outermost* material layer. The value is assumed by EnergyPlus to be the same on the inside and outside of the frame and to be independent of angle of incidence of solar radiation. If solar reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Frame visible absorptance

The visible absorptance of the frame is derived from the surface properties of the *outermost* material layer. The value is assumed to be the same on the inside and outside of the frame and to be independent of angle of incidence of solar radiation. If visible reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Frame thermal hemispherical emissivity

The thermal emissivity of the frame is derived from the surface properties of the *outermost* material layer and is assumed the same on the inside and outside.

Frame width

The width of the frame elements when projected onto the plane of the window (in m or ft). It is assumed that the top, bottom and side elements of the frame have the same width. If not, an average frame width should be entered such that the projected frame area calculated using the average value equals the sum of the areas of the frame elements.

Frame outside projection

The amount by which the frame projects outward from the outside surface of the window glazing (in m or ft). If the outer surface of the frame is flush with the glazing then enter 0. This value is used to calculate shadowing of frame onto glass, solar absorbed by frame, IR emitted and absorbed by frame, and convection from frame.

Frame inside projection

The amount by which the frame projects inward from the inside surface of the window glazing (in m or ft). If the inner surface of the frame is flush with the glazing then enter 0. This value is used to calculate solar absorbed by frame, IR emitted and absorbed by frame, and convection from frame.

Frame conductance

The effective thermal conductance of the frame measured from inside to outside frame surface (no air films) and taking 2-D conduction effects into account. Obtained from WINDOW or other 2-D calculation.

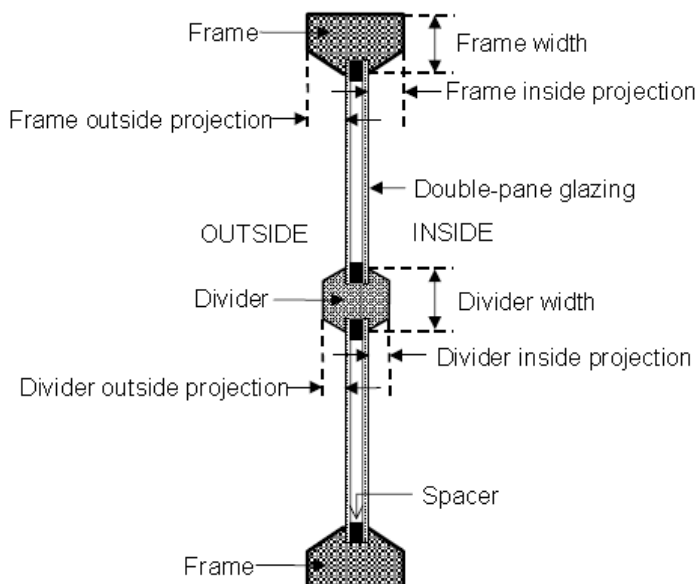
Frame glass edge centre conduction ratio

The glass conductance near the frame (excluding air films) divided by the glass conductance at the centre of the glazing (excluding air films). Used only for multi-pane glazing constructions. This ratio is greater than 1.0 because of thermal bridging from the glazing across the frame and across the spacer that separates the glass panes. Values can be obtained from WINDOW for the user-selected glazing construction and frame characteristics.

Dividers

Dividers split the glazing up into separate lites. It is assumed that all divider elements have the same characteristics. If not, area-weighted average values should be used.

The schematic below show how DesignBuilder divider data is used to describe real glazing systems.

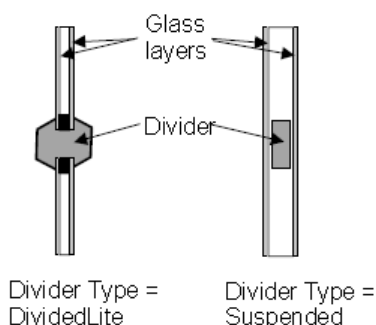


Divider type

The type of divider can be selected from these options:

- **1-Divided lite** means that the divider elements project out from the outside and inside surfaces of the glazing and divide the glazing into individual lites. For multi-pane glazing, this type of divider also has between-glass elements that separate the panes.

- **2-Suspended** is applicable only to multi-pane glazing. It means that the divider is suspended between the panes. (If there are more than two glass layers, the divider is assumed to be placed between the two outermost layers.)



Divider width

The width of the divider elements when projected onto the plane of the window (in m or ft). It is assumed that the horizontal and vertical divider elements have the same width. If not, an average divider width should be entered such that the projected divider area calculated using the average value equals the sum of the areas of the divider elements.

Horizontal dividers

The number of divider elements parallel to the top and bottom of the window.

Vertical dividers

The number of divider elements parallel to the sides of the window.

Divider outside projection

The amount by which the divider projects out from the outside surface of the window glazing (in m or ft). For Divider Type = Suspended, Divider Projection = 0.0. Used to calculate shadowing of divider onto glass, solar absorbed by divider, IR emitted and absorbed by divider, and convection from divider.

Divider inside projection

The amount by which the divider projects inward from the inside surface of the window glazing (in m or ft). If the inner surface of the divider is flush with the glazing, enter 0.0. Used to calculate solar absorbed by divider, IR emitted and absorbed by divider, and convection from divider.

Divider glass edge-centre conduction ratio

The glass conductance near the divider (excluding air films) divided by the glass conductance at the centre of the glazing (excluding air films). Used only for multi-pane glazing constructions. This ratio is greater than 1.0 because of thermal bridging from the glazing across the divider and across the spacer that separates the glass panes. Values can be obtained from the WINDOW program for the user-selected glazing construction and divider characteristics.

Divider properties defined by the Frame construction

The [Construction](#) is used to define the thermal properties of both the frame and dividers. In particular the frame construction is used to derive these EnergyPlus divider-related fields:

Divider conductance

The effective thermal conductance of the divider (in W/m²-K or Btu/h-ft²-F) measured from inside to outside divider surface *excluding surface air films* and taking 2-D conduction effects into account. Obtained from the LBNL WINDOW program or other 2-D calculation. DesignBuilder calculates the surface to surface conductance simply based on the material layers. If you need to include particular 2-

D conduction effects for the frame/divider conductance then you should adjust the thickness or material properties of the layers of the frame construction.

Divider solar absorptance

The solar absorptance of the divider is derived from the surface properties of the *outermost* material layer. The value is assumed to be the same on the inside and outside of the divider and to be independent of angle of incidence of solar radiation. If solar reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Divider visible absorptance

The visible absorptance of the divider is derived from the surface properties of the *outermost* material layer. The value is assumed to be the same on the inside and outside of the divider and to be independent of angle of incidence of solar radiation. If visible reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Divider thermal hemispherical emissivity

The thermal emissivity of the divider, assumed to be the same on the inside and outside.

Note 1: If you do not wish to model the shading/reflection effect of the reveal/frame/divider then simply set the projection/depth to zero.

Note 2: In the case where a [window blind](#) is applied [mid-pane](#), any [dividers](#) that may have been specified for the window will not be applied in the model. EnergyPlus will generate a warning to this effect.

Glazing Assemblies

By default EnergyPlus models glazing and frames separately through separate window and frame objects. This topic discusses some of the issues related to modelling overall glazing assemblies including the glazed window and opaque frame elements.

There are two general ways to model glazing assemblies in DesignBuilder:

- **Data for glass and frame are entered separately** where "glass only" properties are defined in the [glazing](#) component selected and frame data is defined on the Openings tab under [Frame and Dividers](#). Note that when using this method EnergyPlus is not able to calculate an overall assembly U-value from separate window and frame data. If this output is required it can be done either approximately by adding the parallel conductance of the window and frame and taking the reciprocal or in more detail in a tool such as WINDOW.
- **Data for whole glazing assembly is entered directly**. This is done using the **2-Simple** glazing [definition method](#) and entering whole assembly U-value, solar and light transmission data in the [glazing](#) component selected on the Openings tab. In this case, to avoid double-counting the impact of the frame, it is important to remove the frame on the Openings tab for all windows using the whole assembly glazing system by unchecking the [Has frame/dividers](#) checkbox. If you only have access to "glass only" solar and visible transmission data then these can be multiplied by 1 minus the frame fraction to calculate the whole assembly solar and visible transmission values.

See the "Dimensions (Walls)" on page 408 topic for details on how glazing dimensions are written to EnergyPlus IDF.

Shading

[Openings tab](#) in model data under **External Glazing, Internal Glazing and Roof Glazing** headers

You can select two types of shading for external windows under the Shading header on the Openings tab:

- [Window shading](#) (blinds, curtains etc on wall and roof surfaces).
- [Local shading](#) (overhangs, louvres, sidefins on external wall surfaces only)

These types of shading can be used individually or combined.

Other shading mechanisms supported by DesignBuilder are:

- [Component blocks](#)
- Assemblies

Window and Local shading are both covered in the [Shading via louvres, overhangs, and side fins](#) tutorial

Window Shading

[Openings tab](#) in model data under **External Glazing, Internal Glazing and Roof Glazing** headers

Window shading can be included to reduce solar gains and increase resistance to heat conduction through windows. To include blinds, transparent insulation or electrochromic glazing check the **Window shading** checkbox. Then select the window blind component, the position of the shading device relative to the window and the way that the window shading is controlled.

Note: Window shading is not included in Heating design calculations.

Note: In [Radiance daylighting](#) calculations, Window shading is controlled using the [Annual Window shading](#) and [Point in time Window shading](#) settings on the Calculation options dialog. Depending on the setting selected, Window shading settings on the Openings tab may or may not be applied.

Note also that in Radiance, electrochromic and thermochromic glazing cannot dynamically respond to daylight levels. Instead, their performance is simulated using static properties based on the selected Glazing type. If Window shading is enabled and an electrochromic glass is selected, the daylighting simulation uses its darkest state.

Type

Select the [Window blind](#) component type. There are 5 categories of window blinds:

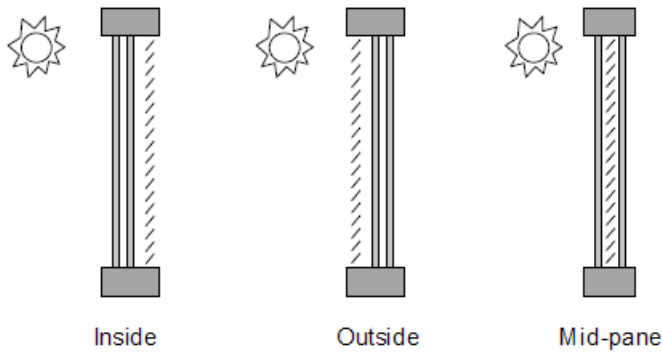
- [Slat](#) - slatted blinds whose transmission properties vary with solar position.
- [Shade](#) - assumed perfectly diffusing.
- [Electrochromic switchable](#) - if you select a blind of this category the Position (below) must be **4-Switchable**.
- [SageGlass electrochromic](#) - multi-state electrochromic glazing systems.
- [Transparent insulation](#) - Deprecated.

Position

The window shading devices can be positioned in one of four ways:

- **1-Inside** - the window shading devices is positioned inside the zone.
- **2-Mid-pane** - the window shading device is positioned between the inner pane and the second pane.
- **3-Outside** - the shading devices positioned outside.

- **4-Switchable** - select this option for [electrochromic glazing](#) in which case the outer pane is switched based on the shading control (below). This option is only available when a Window shading component whose category has been set to **Electrochromic switchable** has been selected as the Type (above).



Note: When selecting the **2-Mid-pane** position option you must ensure that there is sufficient gap between the innermost and second pane in the [Glazing](#) window gas layer to accommodate the blind [Slat width](#).

Control type

Control data specifies how the shading device is controlled. It determines whether the shading device is “on” or “off.” For [blinds and shades](#), when the device is “on” it is assumed to cover all of the window except its frame; when the device is “off” it is assumed to cover none of the window (whether “on” or “off” the shading device is assumed to cover none of the wall that the window is on). For [switchable glazing](#), “on” means that the glazing is in the fully-switched state and “off” means that it is in the unswitched state; for example, for electrochromic glazing, “on” means the glazing is in its darkest state and “off” means it is in its lightest state.

The choices for **Shading control type** are the following. *Equivalent EnergyPlus control types* are shown in italics. If a setpoint is applicable its units are shown in parentheses.

- **1-Always on** (*AlwaysOn*) - Shading devices are always activated.

The following six control types are used primarily to reduce zone cooling load due to window solar gain.

- **2-Daylight** (*MeetDaylightIlluminanceSetpoint*) - Used only with Position = **4-Switchable** in zones with [daylighting controls](#). In this case the transmittance of the glazing is adjusted to meet the [daylight illuminance set point](#) for the main photoelectric sensor used to control electric lighting according to daylight availability. This type of control assures that there is just enough daylight transmitted through windows with electrochromic shading to meet the daylighting requirements in a zone, and no more, thus reducing the cooling load. Note that the [daylight illuminance set point](#) is specified under [Environmental Control](#) on the Activity tab and that there is no option to apply separate setpoints for electrochromic shading and electric lighting control.
- **3-Schedule** - Shading operation is defined by a time only through a schedule. When the schedule has value 1 then shading operates, otherwise it does not.
- **4-Solar** (*OnIfHighSolarOnWindow*) - Shading is active if beam plus diffuse solar radiation incident on the window exceeds the [Solar setpoint](#).
- **5-Glare** (*OnIfHighGlare*) - Shading is on if the total daylight glare index at the zone’s first daylighting sensor from all of the exterior windows in the zone exceeds the maximum glare index specified in the daylighting input for zone.

Note: Glare control requires Lighting control to be switched on in the zone.

- **6-Outside air temp** (*OnIfHighOutsideAirTemp*) - Shading is on if outside air temperature exceeds [Outside air temperature setpoint](#).
- **7-Inside air temp** (*OnIfHighZoneAirTemp*) - Shading is on if zone air temperature in the previous time step exceeds the [Inside air temperature setpoint](#).
- **8-Cooling** (*OnIfHighZoneCooling*) - Shading is on if zone cooling rate in the previous time step is non-zero.

The following two control types can be used to reduce zone cooling load. They are applicable to any Shading type but are most appropriate for interior or exterior blinds, interior or exterior shades with low insulating value, or switchable glazing.

- **9-Day cooling and solar + night** (*OnNight/OnDayIfCoolingAndHighSolarOnWindow*) - Shading is on at night. Shading is on during the day when the solar radiation incident on the window exceeds the [Solar setpoint](#) and the zone cooling rate in the previous time step is non-zero.
- **10-Day cooling and solar** (*OffNight/OnDayIfCoolingAndHighSolarOnWindow*) - Shading is off at night. Shading is on during the day when the solar radiation incident on the window exceeds the [Solar setpoint](#) and if the zone cooling rate in the previous time step is non-zero.

The following three control types can be used to reduce zone heating load during the winter by reducing window conductive heat loss at night and leaving the window unshaded during the day to maximize solar gain. They are applicable to any Shading type but are most appropriate for interior or exterior shades with high insulating value ("movable insulation"). "Night" means the sun is down and "day" means the sun is up.

- **11-Night outside low air temp** (*OnNightIfLowOutsideTemp/OffDay*) - Shading is on at night if the outside air temperature is less than [Outside air temperature setpoint](#) and schedule, if specified, allows shading. Shading is off during the day.
- **12-Night inside low air temp** (*OnNightIfLowInsideTemp/OffDay*) - Shading is on at night if the zone air temperature in the previous time step is less than the [Inside air temperature setpoint](#) and schedule, if specified, allows shading. Shading is off during the day.
- **13-Night heating** (*OnNightIfHeating/OffDay*) - Shading is on at night if the zone heating rate in the previous time step is greater than zero and schedule, if specified, allows shading. Shading is off during the day.

The following two control types can be used to reduce zone heating and cooling load. They are applicable to any Shading type but are most appropriate for translucent interior or exterior shades with high insulating value ("translucent movable insulation").

- **14-Night outside low air temp + day cooling** (*OnNightIfLowOutsideTemp/OnDayIfCooling*) - Shading is on at night if the outside air temperature is less than the [Outside air temperature setpoint](#). Shading is on during the day if the zone cooling rate in the previous time step is non-zero.
- **15-Night heating + day cooling** (*OnNightIfHeating/OnDayIfCooling*) - Shading is on at night if the zone heating rate in the previous time step is non-zero. Shading is on during the day if the zone cooling rate in the previous time step is non-zero.
- **16-Horizontal solar** (*OnIfHighHorizontalSolar*) - Shading is on if total (beam plus diffuse) horizontal solar irradiance exceeds the [Solar setpoint](#) and schedule, if specified, allows shading.
- **17-Outdoor air temp + Solar on window** (*OnIfHighOutdoorAirTempAndHighSolarOnWindow*) - Shading is on if the outside air temperature exceeds the [Outside air temperature setpoint](#) and if the solar radiation incident on the window exceeds the [Solar setpoint](#).
- **18-Outdoor air temp + Horizontal solar** (*OnIfHighOutdoorAirTempAndHighHorizontalSolar*) - Shading is on if the outside air temperature exceeds the [Outside air temperature setpoint](#) and if the horizontal solar radiation on the window exceeds the [Solar setpoint](#).

Solar setpoint

The solar setpoint can be defined when one of these shading control types has been selected:

- **4-Solar**
- **9-Day cooling and solar + night**
- **10-Day cooling and solar**
- **16-Horizontal solar**
- **17-Outdoor air temp + Solar on window**
- **18-Outdoor air temp + Horizontal solar**

When the solar radiation (in W/m² or BTU/h-ft²) is higher than the setpoint entered here and any other required condition are met (control types 9, 10, 17 and 18) and the schedule allows shading operation then the window shading is activated.

Inside air temperature setpoint

The Inside air temperature setpoint can be defined when one of these shading control types has been selected:

- **7-Inside air temp**
- **12-Night inside low air temp**

For cooling control types, when the inside air temperature (in °C or °F) is *higher* than the setpoint entered here and the schedule allows shading operation, the window shading is activated. For heating control types the logic is reversed and shading is activated when the temperature is *below* this setpoint.

Outside air temperature setpoint

The Outside air temperature setpoint can be defined when one of these shading control types has been selected:

- **6-Outside air temp**
- **11-Night outside low air temp**
- **14-Night outside low air temp + day cooling**
- **17-Outdoor air temp + Solar on window**
- **18-Outdoor air temp + Horizontal solar**

When the outside air temperature (in °C or °F) is higher than the setpoint entered here and the schedule allows shading operation, the window shading is activated.

Slat Angle Control

Where a [Window shading](#) system from the **Slatted blinds** category is selected, the angle of the slats must be defined using the settings below.

Slat angle control type

The angle of the blind slats can be controlled using 3 options:

- **1-Fixed slat angle**, the default option where the angle of the slats is fixed to the [Slat angle](#) set on the Window shading dialog for the whole simulation.
- **2-Scheduled slat angle** where the angle of the slats is defined by a schedule selected on the **Slat angle schedule** below.
- **3-Block beam solar** where the slats are automatically orientated to be perpendicular to the direction of the suns rays.

Slat angle schedule

The schedule defines the angle of the slats for every timestep in the simulation. This setting is only available when the **2-Scheduled slat angle** [Slat angle control type](#) is set. See Window shading dialog for the definition of [Slat angle](#).

Note: the operation of the blinds is controlled independently of the slat angle.

Operation

Operation schedule

For all shading control options apart from **Glare** and **Always on**, the shading can only be active when this schedule value is 1. Select the schedule **On** if the blind is always activated or if the only control is one of the others listed above.

Limitations

- If two daylight zones share an interior window, neither zone can have an exterior window whose shading device does glare control (i.e. [Control type](#) is **5-Glare**).
- If two daylight zones share an interior window, neither zone can have an exterior window with daylight control (i.e. [Control type](#) is **2-Daylight**).

Local Shading

[Openings tab](#) in model data under **External Glazing headers**

To include overhangs, sidefins and/or louvres check the **Local shading** checkbox. This shows the [Local shading component](#) data. DesignBuilder comes with a small database of Local shading devices, but if you cannot find a suitable component for your model, you can create a custom local shading component by clicking on the **Add new** button from the Selection dialog or from the info panel when using Learning mode.

Note: If you have repeated windows having same size, glazing type and window shading on a surface but different Local shading settings, you should switch off the [Lump similar windows on surface](#) model option. This is because the window lumping mechanism does not account for local shading and so windows may be lumped inappropriately in this case.

See also the [Shading via louvres, overhangs, and side fins](#) tutorials

Note: the DesignBuilder Local shading devices allow a range of different overhangs, side fins and louvres to be defined, but if you need to model a shading device not covered by the standard mechanism, you can draw custom shading devices using [Component blocks](#) or Assembly blocks.

Airflow Windows

[Openings tab](#) in model data under **External Glazing header**

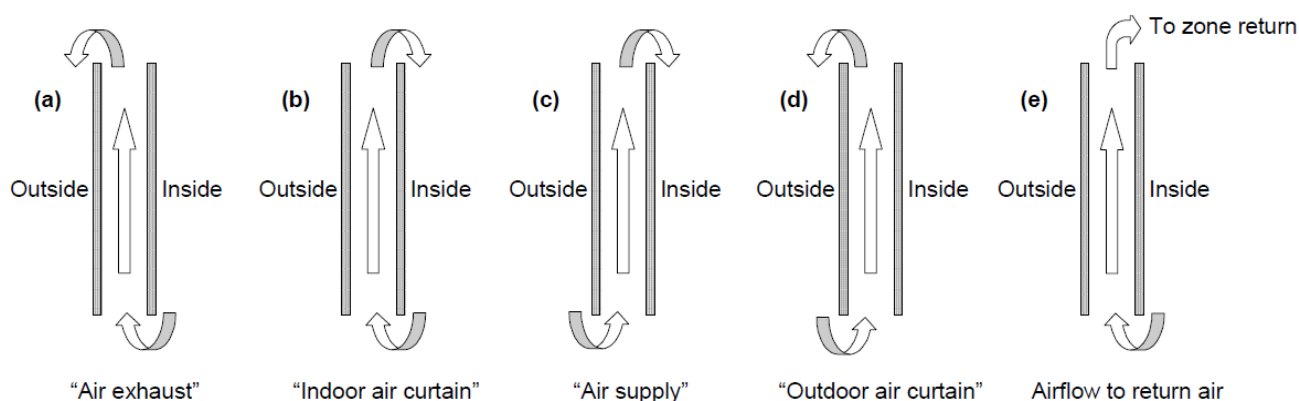
Airflow windows can be modelled by checking the Airflow windows option on the Openings tab. This option can be used to specify the control mechanism for windows in which forced air flows in the gap between adjacent layers of glass. Such windows are called “airflow windows.” They are also known as “heat-extract windows” or “climate windows.”

A common application is to reduce the zone load by exhausting indoor air through the window. In the cooling season this picks up and expels some of the solar heat absorbed by the window glass (and by the between-glass shade or blind, if present). In the heating season this warms the window, reducing the heat loss from the window. A side benefit is increased thermal comfort. This is because the inside surface of the window will generally be cooler in summer and warmer in winter.

Limitations

- Windows must have two or three glass layers, i.e., double or triple-glazing. For triple-glazing the airflow is assumed to be between the two inner glass layers.
- An error will result if the gas in the airflow gap is other than air.
- If an airflow window has a between-glass shade or blind, the gas in the gap on either side of the shade or blind must be air.

The surface output variable **Surface Window Gap Convective Heat Transfer Rate** gives the heat picked up (or lost) by the gap airflow.



Gap airflow configurations for airflow windows:

- (a) **Air exhaust window:** Airflow Source = **1-Indoor air**, Airflow Destination = **2-Outdoor air**;
 (b) **Indoor air curtain window:** Airflow Source = **1-Indoor air**, Airflow Destination = **1-Indoor air**;
 (c) **Air supply window:** Airflow Source = **2-Outdoor air**, Airflow Destination = **1-Indoor air**;
 (d) **Outdoor air curtain window:** Airflow Source = **2-Outdoor air**, Airflow Destination = **2-Outdoor air**;
 (e) **Airflow to Return Air:** Airflow Source = **1-Indoor air**, Airflow Destination = **3-Return air**.

Based on "Active facades" Version no. 1, Belgian Building Research Institute, June 2002.

Note: Although Airflow control windows are defined under the **External Glazing** header (which normally holds data for wall glazing only) Airflow control window settings apply to *all* windows, including rooflights.

Airflow control

Check this option to include airflow control for external windows.

Airflow source

The source of the gap airflow. The choices are:

- **1-Indoor air** - Indoor air from the window's zone is passed through the window.
- **2-Outdoor air** - Outdoor air is passed through the window.

Airflow Destination

This is where the gap air goes after passing through the window. The choices are:

- **1-Indoor air** - the gap air goes to the indoor air of the window's zone.
- **2-Outdoor air** - the gap air goes to the outside air.
- **3-Return-air** - the gap air goes to the return air for the window's zone. This choice is allowed only if Airflow Source = 1-Indoor air. If the return air flow is zero, the gap air goes to the indoor air of the window's zone. If the sum of the gap airflow for all of the windows in a zone with **Airflow destination = 3-Return air** exceeds the return airflow, then the difference between this sum and the return airflow goes to the indoor air.

The diagram above shows the allowed combinations of **Airflow source** and **Airflow destination**. The allowed combinations of **Airflow source** and **Airflow destination** are:

- IndoorAir to OutdoorAir
- IndoorAir to IndoorAir
- IndoorAir to ReturnAir
- OutdoorAir to IndoorAir
- OutdoorAir to OutdoorAir

Maximum flow rate

The maximum value of the airflow (in m³/s per m of glazing width or ft³/min-ft). The value is typically 0.006 to 0.009 m³/s-m (4 to 6 ft³/min-ft). The airflow can be modulated through the Airflow multiplier schedule (below).

The fan energy used to move the air through the gap is generally very small and so is ignored.

Airflow multiplier schedule

The name of a [schedule](#) with values between 0.0 and 1.0. The timestep value of the airflow is **Maximum flow rate** times the schedule value.

Window Operation And Free Aperture

[Openings tab](#) in model data under **External Glazing**, **Internal Glazing** and **Roof Glazing** headers

Operation

The operation of windows can be defined by entering data under the **Operation** header. This data is only shown:

- When [Natural ventilation](#) is activated on the HVAC tab at the current level and Calculated natural ventilation is selected, or,
- For Internal glazing or both Calculated and Scheduled natural ventilation.

Schedule definition

Select the method to be used to define the timing of the window operation. Options are:

- **1-Follow occupancy** - the window can be opened when the zone is occupied and not when it is not occupied.
- **2-Custom schedule** - the operation of the window is controlled by a specific schedule which is displayed when this option is selected.

Operation schedule

When the **Schedule definition** method (above) has been set to **2-Custom schedule** the schedule to be used to define window opening times can be selected. It defaults to the occupancy schedule based on the most recently loaded [Activity template](#).

Control option (internal windows only)

The control option allows you to define how *internal windows* behave in natural ventilation systems and whether they operate purely based on a schedule or, when using Calculated natural ventilation, whether they are linked to the main natural ventilation controllers defined on the HVAC tab. This data is not available for external windows. The options are:

For **Calculated** natural ventilation:

- **1-Constant** - the size of the internal window airflow opening (when open) is constant and the opening of internal windows is controlled entirely by schedule, i.e. it is not affected by any of the natural ventilation controls on the HVAC tab. In particular the [Internal control mode](#) setting on the HVAC tab is not used.
- **2-Use Internal control mode** - the internal windows are controlled using natural ventilation [Internal control mode](#) settings on the HVAC tab in combination with the operation schedule described below.

Note: When using Calculated natural ventilation and the **1-Constant** control option is selected, airflow through internal windows will occur (when enabled by the operation schedule) regardless of whether natural ventilation is [activated](#) on the HVAC tab. The **2-Use Internal control mode** option should only be selected when natural ventilation has been [activated](#) on the HVAC tab.

For **Scheduled** natural ventilation:

- **1-Control by schedule** - the internal windows are controlled entirely by schedule and are not affected by any of the natural ventilation controls on the HVAC tab.
- **2-Closed** - the internal windows are always closed and do not allow any airflow to be exchanged.

Note: When using Scheduled natural ventilation and the **1-Control by schedule** control option is selected, airflow through internal windows will occur (when enabled by the operation schedule) regardless of whether natural ventilation is [activated](#) on the HVAC tab.

Operation schedule

The schedule that specifies when window venting is available. A zero or negative schedule value means window venting is not allowed. A value greater than zero means venting can occur if other venting control conditions (specified by natural ventilation [Control Mode](#)) are satisfied.

Note: Fractional availability schedules do not restrict the venting in any way. When the schedule value > 0 the window can open, when it is 0 the window is closed.

An exception to this rule is for [CIBSE TM59:2026](#) simulations when using the [Restrict venting factor by schedule](#) option which, when set, does allow time-varying window aperture restriction to be defined through this schedule.

This schedule allows you to turn off venting at certain times of the day (at night, for example), of the week (on weekends, for example), or of the year (during the winter, for example).

If used with natural ventilation [Control Mode](#) = **4-Constant**, the ventilation rate is constant only when this schedule allows venting; otherwise the ventilation rate is set to zero. If Ventilation Control Mode = **1-NoVent**, this schedule has no effect.

The windows are 'open' when:

$T_{\text{zone_air}} > T_{\text{setpoint}}$ AND $T_{\text{zone_air}} > T_{\text{outside_air}}$ AND the schedule value = 1

The ventilation rate is calculated based on the pressure difference across the opening calculated from wind and stack pressure effects using equations described in [Natural Ventilation Modelling](#).

Free Aperture

The size and position of window opening airflow paths can be defined by entering data under the **Free Aperture** header. This data is used in both:

- a. EnergyPlus Calculated natural ventilation Airflow network simulations and
- b. CFD simulations

Note: While free aperture data is shown when the Scheduled natural ventilation model option is selected, this data is not used in EnergyPlus simulations. In this case the data is only made available to allow it to be used for CFD boundary conditions.

Opening position

The position of the opening in the window can be selected from

- **1-Top** - the aperture is in the top of the window. Air flow in and out will flow through the top of the window.
- **2-Bottom** - the aperture is in the bottom of the window. Air flow in and out will flow through the bottom of the window.
- **3-Right** - the aperture is in the right of the window (looking from outside). Air flow in and out will flow through the right of the window.
- **4-Left** - the aperture is in the left of the window (looking from outside). Air flow in and out will flow through the left of the window.

Windows should be rectangular to have an airflow opening in them and the position of the opening on the window can be defined by selecting from the above list. Options **3-Right** and **4-Left** will give identical simulation results as the EnergyPlus Airflow Network only accounts for the z-position of openings. The difference is useful for CFD simulations though.

Note: Non-rectangular windows (or rotated rectangular windows) can still be used as airflow openings. In this case for Airflow Network opening size purposes, EnergyPlus treats triangular windows as if they were rectangular with height and width based on the maximum dimensions in each direction.

% Glazing area opens

The % Glazing area opens data allows you to define the openable window "free area" as a percentage of the window area. For example enter 20 if the window free area is 20% of the total glazed window area. In simple terms this is the fraction of the window that can be opened, but it should also take into account the effect of any obstructions and in practice a more correct definition for the free area is the "cross-sectional (measurable, geometric) area".

The "free area" is multiplied by the [discharge coefficient](#) to give the "effective area" for the Airflow Network calculations within EnergyPlus.

Note: When calculating the **% Glazing area opens** you should divide the openable window free area by the total glazed area of the window, excluding the frame. The expandable section below under [Opening Areas For Fully Opening Windows with Frames](#) provides advice on how to deal with fully opening windows with large frames to avoid under-representing airflow.

Tip: A good rule of thumb to approximate of the free area for a top hung window is the bottom rectangular "opening area" plus 2 x the triangular side opening area. For closely spaced windows the side triangles

should be reduced to account for the reduced effective opening area. A similar calculation can be made for side and bottom hung windows.

A couple of good reference for how to measure or otherwise obtain this data are:

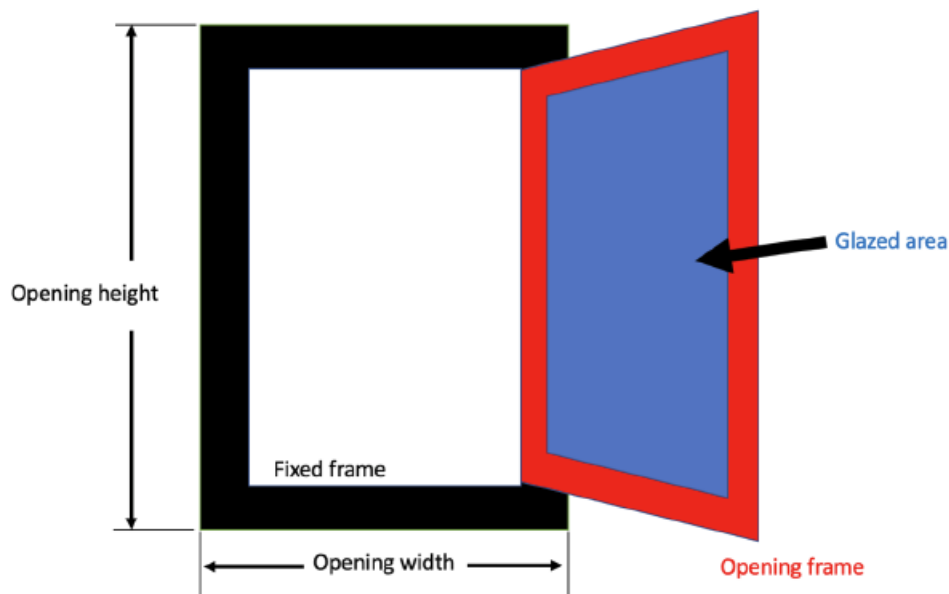
- [A review of ventilation opening area terminology](#) by, Benjamin M. Jones, Malcolm J. Cook, Shaun D. Fitzgerald, Christopher R. Iddon., 2015.
- "Air of Credibility" in the CIBSE Journal May 2016 (same authors as above).

Note: if [modulation](#) is switched on (it isn't by default) the glazing opening areas defined here are multiplied by a factor to account for external conditions in EnergyPlus Airflow Network simulations when using Calculated natural ventilation.

Tip: In some cases it is best to switch off the [Lump similar windows on surface](#) model option when using Calculated natural ventilation. Particular cases where this is important are a) when there are windows of the same size and type but different **% Glazing area opens** values on the same surface and b) where the air flow rate through individual windows is required, e.g. when preparing boundary conditions for a linked EnergyPlus CFD study.

Opening Areas For Fully Opening Windows with Frames

Because EnergyPlus only uses the glazed area for window airflow calculations, for windows with significant frames that fully open, a correction needs to be applied to avoid under-representing the airflow in the simulation.



Considering the example in the image above, when drawing a glazed opening such as a window or patio door, the opening height and width are entered including the frame. The width of the frame, which includes any fixed parts (black) and opening parts (red) is entered as a single frame width figure and DesignBuilder subtracts this to give the glazed area.

However, because only the glazed area is used for airflow calculations, the airflow through the opening would be under-represented if we simply entered **% Glazing area opens** to be 100% without further correction. The wider the frame, the bigger the issue. For example, a typical uPVC glazed door might have a frame width of 0.14m, of which 0.04m is the fixed frame (black) and 0.1m is part of the opening area (red) along with the glazing. For a 2.1m high by 0.9m wide glazed door, the free area opening size when fully

open (subtracting the black fixed frame area) would be 1.66 m², which is 1.47 times the glazed only area in blue (1.13 m²) used in the airflow network ventilation calculations.

This issue can have significant implications for airflow modelling in some overheating assessments. It is not possible to correct for it by increasing the glazed area (either by increasing the opening size or reducing the frame size) because that would affect solar gains.

In our example case of the large glazed door, with 100% openable area there is no scope to increase the **% Glazing area opens** beyond 100%. **The recommended workaround is to modify the [discharge coefficient](#) to account for the missing frame area in the airflow model.** In our example, multiplying the default discharge coefficient for the window (0.65) by 1.47 (actual ventilation free area divided by glazed only area) gives a discharge coefficient of 0.95 which can be entered to compensate for the under representation of opening size.

Credit: Stephen Porritt for this workaround and the image.

Internal Window operation in CFD

Like other internal openings, internal windows cannot have airflow specified to define air movement in CFD simulations. Because CFD is for a snapshot in time and so does not use schedules, if you have loaded simulation results for a time where an internal window was closed *due to the schedule or some control setting*, to ensure that you see the same behaviour in CFD you can set the **% Glazing area opens** to zero to zero in a copy of the simulation model.

Discharge coefficient

The constant discharge coefficient for the window (C_d) is used for Calculated natural ventilation only. The value must be between 0.0 and 1.0. Separate values can be entered for internal, external and roof windows.

For open doors and holes, a discharge coefficient can be entered on the Advanced tab of the model options dialog under the [Natural Ventilation](#) header.

Theory explaining the discharge coefficient

From the IEA Annex 20 document [Air flow Through Large Openings in Buildings](#).

The EnergyPlus Airflow Network model is based on the following empirical powerlaw relationship between the flow and the pressure difference across a crack or opening in the building envelope:

$$Q = C(\Delta P)^n$$

The volumetric flow rate, Q [m³/s], is a simple function of the pressure drop, ΔP [Pa], across the opening. A common variation of the powerlaw equation is:

$$F = C(\Delta P)^n$$

where the mass flow rate, F [kg/s], is a simple function of the pressure drop. A third variation is related to the orifice equation:

$$Q = C_d A \sqrt{\frac{2\Delta P}{\rho}}$$

where:

C_d = discharge coefficient, and
 A = orifice opening area.

Theoretically, the value of the flow exponent, n , should lie between 0.5 and 1.0. Large openings are characterized by values very close to 0.5, while values near 0.65 have been found for small crack-like openings.

Suggested values for discharge coefficient

The literature suggests a large range of different values for discharge coefficient (symbol C_d often used in orifice equations). For example IEA Annexe 20 Air flow Through Large Openings in Buildings indicates that discharge coefficients reported in the literature vary between 0.3 and 0.8 and that it is not understood what causes the differences.

Other sources recommend a value close to 0.6. For example the [CONTAMW 2.0 User Manual](#) states that the *discharge coefficient, is related to the dynamic effects and is typically close to 0.6 for an orifice and slightly higher for other openings in buildings.*

ASHRAE propose the following correlation based on interzone temperature differences:

$$C_d = 0.4 + 0.0045 \delta T$$

for the range δT 0.5 to 40.

Given other uncertainties in natural ventilation calculations (wind pressure coefficients, effective areas of real-world openings and crack flows etc), using a discharge coefficient between 0.60 and 0.65 should provide sufficient accuracy.

Scheduled Natural Ventilation

Airflow through individual *external* windows, vents, doors and holes are not modelled explicitly with [Scheduled](#) natural ventilation because the natural ventilation airflow from outside is defined using the total [natural ventilation flow rate](#) and [operation](#) on the [HVAC](#) tab. *No operation data or free aperture needs to be entered on the Openings tab for external windows in this case.*

Note: Natural Ventilation can be [overridden to be off](#) on the HVAC tab for both Calculated and Scheduled model options.

Internal Glazing

[Openings tab](#) in model data

The data under the Internal Glazing header applies to all glazing lying on internal partition walls. You can enter information on:

- [Dimensions](#)
- [Frame](#)
- [Operation](#)

The data is as described previously for [External glazing](#) except that [shading](#) is not permitted for internal glazing.

Note: Data for internal windows is not available at zone level as explained in "Zone Level Data for Internal Surfaces" on page 698.

Roof Glazing

[Openings tab](#) in model data

The data under the Roof Glazing header applies to all glazing lying on external roof surfaces. You can enter information on:

- [Dimensions](#)
- [Frame](#)
- [Shading](#)
- [Operation](#)

Note: flat roofs do not have default glazing set up using Roof glazing layout model data. To create openings on a flat roof you should go to the surface level and add them there.

Dimensions (Roofs)

[Openings tab](#) in model data under Sloped Roof Windows / Skylights headers

Notes: The settings for roof window dimensions are similar to those for external walls with some important exceptions:

- The façade settings apply only to *sloping* roofs. **Flat roofs are not covered by this data.** To create windows in flat roofs you should draw them at surface level or clone them at building level from other façades. The reason flat roofs do not use the façade mechanism is that there is no top and bottom to set up sills, height vs width etc.
- Only the **0-None**, **4-Fixed width and height** and **5-Fill surface (100%)** façade options are available. There is therefore no way to set up windows using a % glazing (WWR) on roof surfaces.
- Reveals are not provided for roof surfaces.

Facade types

There are 3 façade types available for sloped roof surfaces:

- **0-None** - there is no glazing.
- **4-Fixed width and height** - windows have fixed width and height.
- **5-Fill surface (100%)** - the entire surface is filled with glazing and there is **no frame**.

Window width

The width of the windows including any frame (in m or ft).

Window height

The height of the windows including any frame (in m or ft).

Window spacing

The spacing between the each window on the façade (in m or ft). The window spacing is the centre to centre spacing between windows, not the gap between windows.

Sill height

This is the height of the base of the window from the base of the block (in m or ft).

Frame And Dividers

[Openings tab](#) in model data under **External Glazing, Internal Glazing and Roof Glazing** headers

If window frames and dividers are being modelled separately from the glazed part then the properties of the frame and dividers can be entered under the Frame and Dividers header on the Openings tab. A discussion on the methods that can be used for modelling windows and frames in "Glazing Assemblies" on page 418.

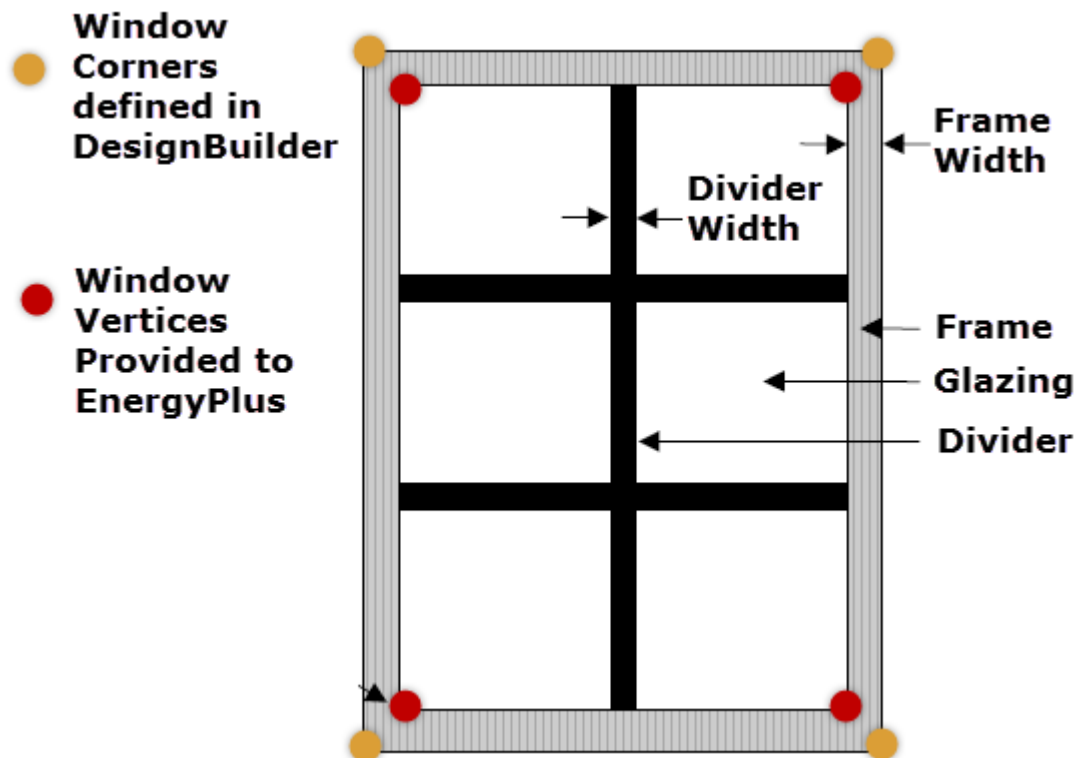
Important Note: Frames and dividers are only applied to *rectangular* windows (default and custom) and not to other shaped windows. They are also not applied when using the **Fill surface (100%)** facade type. If you are unsure about whether frames and dividers will be applied to a window you can use the [Visualisation rendered view](#) to check.

A frame surrounds the glazing in a rectangular window (schematics below). It is assumed that all frame characteristics, such as width, conductance and solar absorptance, are the same for the top, bottom and side elements of the frame. If the frame elements are not the same then you should enter area-weighted average values for the frame characteristics.

Has a Frame/Dividers

Check the **Has a Frame/Dividers** checkbox to enter details of the frame and/or dividers. In this case you can enter:

- The construction of the frame and dividers,
- The number of horizontal dividers and vertical dividers,
- Frame width and divider width (see diagrams below for definition).
- Inside reveal and sill details



[EnergyPlus window dimensions - technical information](#)

For **external windows on walls** the following dimensions in the direction of the glazing normal can also be entered:

- [Outside reveal depth](#)
- Inside reveal depth
- Inside sill depth
- Outside divider projection
- Inside divider projection
- Outside frame projection
- Inside frame projection
- Divider glass edge-centre conduction ratio
- Frame glass edge-centre conduction ratio
- Divider type

The definition of the above dimensions is shown in the diagrams.

Note: If the glazing does not have a frame or dividers or if the whole [glazing assembly](#) properties are being modelled directly using the **2-Simple** glazing [definition method](#) then you should uncheck the **Has a frame/dividers** box.

Reveal

Reveal surfaces are associated with the setback of the glazing from the outside/inside surfaces of the wall they are located in. If the depth and solar absorptance of these surfaces are specified, the program will calculate the reflection of beam solar radiation from these surfaces. The program also calculates the shadowing (onto the window) of beam and diffuse solar radiation by outside reveal surfaces.

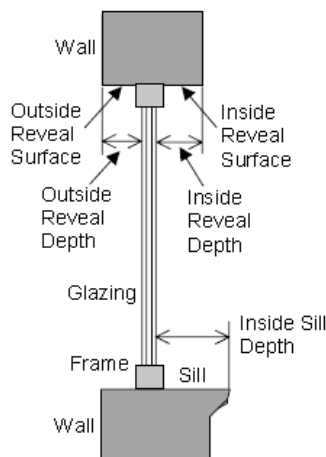
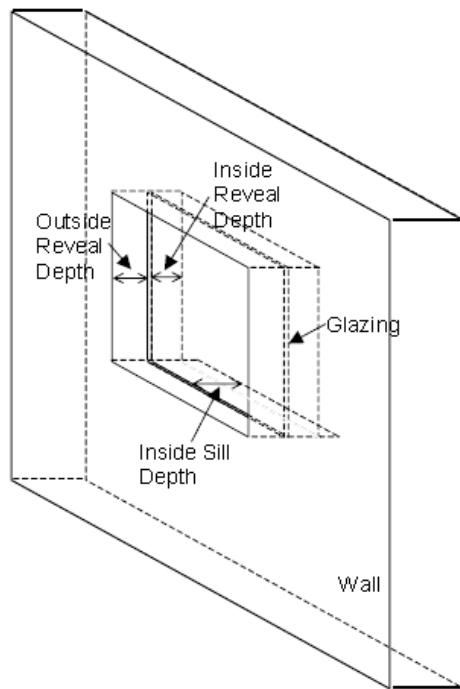
The following fields specify the properties of the window reveal surfaces (reveals occur when the window is not in the same plane as the base surface). From this information and from the geometry of the window and the sun position, the program calculates beam solar radiation absorbed and reflected by the top, bottom, right and left sides of outside and inside window reveal surfaces. In doing this calculation, the shadowing on a

reveal surface by other reveal surfaces is determined using the orientation of the reveal surfaces and the sun position.

It is assumed that:

- The window is an exterior window (EnergyPlus does not consider reveals for interior windows).
- The reveal surfaces are perpendicular to the window plane.
- If an exterior shade, screen or blind is in place it shades exterior and interior reveal surfaces so that in this case there is no beam solar on these surfaces.
- If an interior shade or blind is in place it shades the interior reveal surfaces so that in this case there is no beam solar on these surfaces.
- The possible shadowing on inside reveal surfaces by a window divider is ignored.
- The outside reveal surfaces (top, bottom, left, right) have the same solar absorptance and depth. This depth is not input here but is automatically determined by the program—from window and wall vertices—as the distance between the plane of the outside face of the glazing and plane of the outside face of the parent wall.
- The inside reveal surfaces are divided into two categories: (1) the bottom reveal surface, called here the "inside sill;" and (2) the other reveal surfaces (left, right and top).
- The left, right and top inside reveal surfaces have the same depth and solar absorptance. The inside sill is allowed to have depth and solar absorptance values that are different from the corresponding values for the other inside reveal surfaces.
- The inside sill depth is required to be greater than or equal to the depth of the other inside reveal surfaces. If the inside sill depth is greater than zero the depth of the other inside reveal surfaces is required to be greater than zero.
- The reflection of beam solar radiation from all reveal surfaces is assumed to be isotropic diffuse; there is no specular component.
- Half of the beam solar reflected from outside reveal surfaces is goes towards the window; the other half goes back to the exterior environment (i.e., reflection of this outward-going component from other outside reveal surfaces is not considered).
- The half that goes towards the window is added to the other solar radiation incident on the window. Correspondingly, half of the beam solar reflected from inside reveal surfaces goes towards the window, with the other half going into the zone. The portion going towards the window that is not reflected is absorbed in the glazing or is transmitted back out into the exterior environment.
- The beam solar that is absorbed by outside reveal surfaces is added to the solar absorbed by the outside surface of the window's parent wall; similarly, the beam solar absorbed by the inside reveal surfaces is added to the solar absorbed by the inside surface of the parent wall.
- The net effect of beam solar reflected from outside reveal surfaces is to increase the heat gain to the zone, whereas the effect of beam solar reflected from inside reveal surfaces is to decrease the heat gain to the zone since part of this reflected solar is transmitted back out the window.
- If the window has a frame, the absorption of reflected beam solar by the inside and outside surfaces of the frame is considered. The shadowing of the frame onto interior reveal surfaces is also considered.

The schematics below show how DesignBuilder reveal data is used to describe real glazing systems.



FAQ: Why does DesignBuilder require both Inside and Outside reveal depth data when wall thickness is available to calculate one reveal depth from the other?

Although it is possible for inside reveal depth to be calculated from the outside reveal depth and the [wall thickness](#), DesignBuilder allows both Inside and Outside reveal depths to be defined giving greater flexibility for situations where it is not possible to enter the wall thickness as accurately as one might like. For example the wall thickness may be entered as zero for simplified modelling. In such cases having direct control over the reveal depth data is an advantage.

Inside reveal depth

The **inside reveal depth** is the distance (in m or ft) from the innermost surface of the window to the interior wall surface as shown in the diagram above. This setting is used for EnergyPlus simulations but not for Visualisations and Radiance which use the [Outside reveal depth](#) to define the position of the window within the wall.

Note: **Inside reveal depth** data is provided to EnergyPlus in the same object as the frame and divider data, so when the [Has a frame/dividers](#) option is unchecked this data is not provided to EnergyPlus and the equivalent of a 0 inside reveal depth will be simulated.

See also [Outside reveal depth](#).

Inside sill depth

The **inside sill depth** is the distance (in m or ft) from the innermost surface of the window to the innermost end of the sill as shown in the diagram above.

Note: **Inside sill depth** data is provided to EnergyPlus in the same object as the frame and divider data, so when the [Has frame/dividers](#) option is unchecked this data is not provided to EnergyPlus and the equivalent of a 0 inside sill depth will be simulated.

Frame

Construction

Select the construction used to define the thermal properties of both the frame and dividers. In particular the frame construction is used to derive these EnergyPlus frame-related fields:

Frame conductance

The effective thermal conductance of the frame (in W/m²-K or Btu/h-ft²-F) measured from inside to outside frame surface *excluding surface air films* and taking frame and window geometry as associated 2-D conduction effects into account. Ideally the frame conductance should be obtained from THERM or some other 2-D calculation. DesignBuilder calculates the surface to surface conductance simply based on the material layers in the Frame construction. If you need to include particular 2-D conduction effects for the frame conductance then you should adjust the thickness or material properties of the layers of the frame construction.

Note: For metal frame windows DesignBuilder will calculate a very high surface to surface U-value for the frame construction, but the value that will be provided to EnergyPlus is limited to an maximum value of 500 W/m²k. This is in line with the limits used in the LBNL WINDOW program. This will have negligible difference on simulation as the corresponding thermal resistance is effectively zero in both cases (relative to the inside and outside film resistances).

Frame solar absorptance

The solar absorptance of the frame is derived from the surface properties of the *outermost* material layer. The value is assumed by EnergyPlus to be the same on the inside and outside of the frame and to be independent of angle of incidence of solar radiation. If solar reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Frame visible absorptance

The visible absorptance of the frame is derived from the surface properties of the *outermost* material layer. The value is assumed to be the same on the inside and outside of the frame and to be independent of angle of incidence of solar radiation. If visible reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Frame thermal hemispherical emissivity

The thermal emissivity of the frame is derived from the surface properties of the *outermost* material layer and is assumed the same on the inside and outside.

Frame width

The width of the frame elements when projected onto the plane of the window (in m or ft). It is assumed that the top, bottom and side elements of the frame have the same width. If not, an average frame width should be entered such that the projected frame area calculated using the average value equals the sum of the areas of the frame elements.

Frame outside projection

The amount by which the frame projects outward from the outside surface of the window glazing (in m or ft). If the outer surface of the frame is flush with the glazing then enter 0. This value is used to calculate shadowing of frame onto glass, solar absorbed by frame, IR emitted and absorbed by frame, and convection from frame.

Frame inside projection

The amount by which the frame projects inward from the inside surface of the window glazing (in m or ft). If the inner surface of the frame is flush with the glazing then enter 0. This value is used to calculate solar absorbed by frame, IR emitted and absorbed by frame, and convection from frame.

Frame conductance

The effective thermal conductance of the frame measured from inside to outside frame surface (no air films) and taking 2-D conduction effects into account. Obtained from WINDOW or other 2-D calculation.

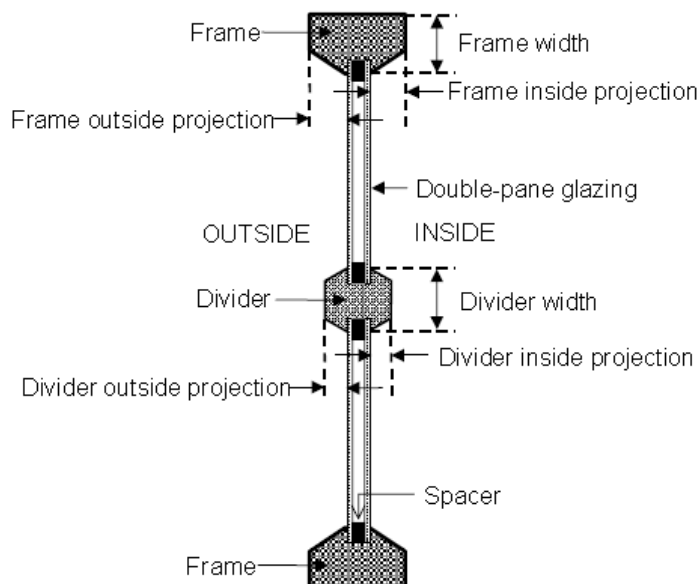
Frame glass edge centre conduction ratio

The glass conductance near the frame (excluding air films) divided by the glass conductance at the centre of the glazing (excluding air films). Used only for multi-pane glazing constructions. This ratio is greater than 1.0 because of thermal bridging from the glazing across the frame and across the spacer that separates the glass panes. Values can be obtained from WINDOW for the user-selected glazing construction and frame characteristics.

Dividers

Dividers split the glazing up into separate lites. It is assumed that all divider elements have the same characteristics. If not, area-weighted average values should be used.

The schematic below show how DesignBuilder divider data is used to describe real glazing systems.

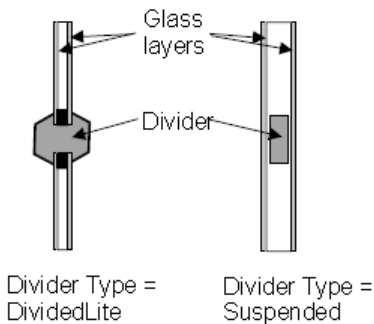


Divider type

The type of divider can be selected from these options:

- **1-Divided lite** means that the divider elements project out from the outside and inside surfaces of the glazing and divide the glazing into individual lites. For multi-pane glazing, this type of divider also has between-glass elements that separate the panes.

- **2-Suspended** is applicable only to multi-pane glazing. It means that the divider is suspended between the panes. (If there are more than two glass layers, the divider is assumed to be placed between the two outermost layers.)



Divider width

The width of the divider elements when projected onto the plane of the window (in m or ft). It is assumed that the horizontal and vertical divider elements have the same width. If not, an average divider width should be entered such that the projected divider area calculated using the average value equals the sum of the areas of the divider elements.

Horizontal dividers

The number of divider elements parallel to the top and bottom of the window.

Vertical dividers

The number of divider elements parallel to the sides of the window.

Divider outside projection

The amount by which the divider projects out from the outside surface of the window glazing (in m or ft). For Divider Type = Suspended, Divider Projection = 0.0. Used to calculate shadowing of divider onto glass, solar absorbed by divider, IR emitted and absorbed by divider, and convection from divider.

Divider inside projection

The amount by which the divider projects inward from the inside surface of the window glazing (in m or ft). If the inner surface of the divider is flush with the glazing, enter 0.0. Used to calculate solar absorbed by divider, IR emitted and absorbed by divider, and convection from divider.

Divider glass edge-centre conduction ratio

The glass conductance near the divider (excluding air films) divided by the glass conductance at the centre of the glazing (excluding air films). Used only for multi-pane glazing constructions. This ratio is greater than 1.0 because of thermal bridging from the glazing across the divider and across the spacer that separates the glass panes. Values can be obtained from the WINDOW program for the user-selected glazing construction and divider characteristics.

Divider properties defined by the Frame construction

The [Construction](#) is used to define the thermal properties of both the frame and dividers. In particular the frame construction is used to derive these EnergyPlus divider-related fields:

Divider conductance

The effective thermal conductance of the divider (in W/m²-K or Btu/h-ft²-F) measured from inside to outside divider surface *excluding surface air films* and taking 2-D conduction effects into account. Obtained from the LBNL WINDOW program or other 2-D calculation. DesignBuilder calculates the surface to surface conductance simply based on the material layers. If you need to include particular 2-

D conduction effects for the frame/divider conductance then you should adjust the thickness or material properties of the layers of the frame construction.

Divider solar absorptance

The solar absorptance of the divider is derived from the surface properties of the *outermost* material layer. The value is assumed to be the same on the inside and outside of the divider and to be independent of angle of incidence of solar radiation. If solar reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Divider visible absorptance

The visible absorptance of the divider is derived from the surface properties of the *outermost* material layer. The value is assumed to be the same on the inside and outside of the divider and to be independent of angle of incidence of solar radiation. If visible reflectance (or reflectivity) data is available, then absorptance is equal to 1.0 minus reflectance (for opaque materials).

Divider thermal hemispherical emissivity

The thermal emissivity of the divider, assumed to be the same on the inside and outside.

Note 1: If you do not wish to model the shading/reflection effect of the reveal/frame/divider then simply set the projection/depth to zero.

Note 2: In the case where a [window blind](#) is applied [mid-pane](#), any [dividers](#) that may have been specified for the window will not be applied in the model. EnergyPlus will generate a warning to this effect.

Roof Shading

[Openings tab](#) in model data under Roof Glazing headers

You can select one type of shading for external roof windows:

- [Window shading](#) (blinds, curtains etc walls and roofs).

Local shading is not available for roof windows.

Window Shading

[Openings tab](#) in model data under External Glazing, Internal Glazing and Roof Glazing headers

Window shading can be included to reduce solar gains and increase resistance to heat conduction through windows. To include blinds, transparent insulation or electrochromic glazing check the **Window shading** checkbox. Then select the window blind component, the position of the shading device relative to the window and the way that the window shading is controlled.

Note: Window shading is not included in Heating design calculations.

Note: In [Radiance daylighting](#) calculations, Window shading is controlled using the [Annual Window shading](#) and [Point in time Window shading](#) settings on the Calculation options dialog. Depending on the setting selected, Window shading settings on the Openings tab may or may not be applied.

Note also that in Radiance, electrochromic and thermochromic glazing cannot dynamically respond to

daylight levels. Instead, their performance is simulated using static properties based on the selected Glazing type. If Window shading is enabled and an electrochromic glass is selected, the daylighting simulation uses its darkest state.

Type

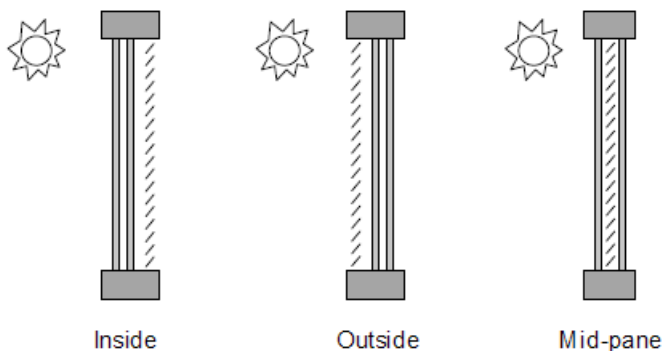
Select the [Window blind](#) component type. There are 5 categories of window blinds:

- [Slat](#) - slatted blinds whose transmission properties vary with solar position.
- [Shade](#) - assumed perfectly diffusing.
- [Electrochromic switchable](#) - if you select a blind of this category the Position (below) must be **4-Switchable**.
- [SageGlass electrochromic](#) - multi-state electrochromic glazing systems.
- [Transparent insulation](#) - Deprecated.

Position

The window shading devices can be positioned in one of four ways:

- **1-Inside** - the window shading device is positioned inside the zone.
- **2-Mid-pane** - the window shading device is positioned between the inner pane and the second pane.
- **3-Outside** - the shading device is positioned outside.
- **4-Switchable** - select this option for [electrochromic glazing](#) in which case the outer pane is switched based on the shading control (below). This option is only available when a Window shading component whose category has been set to **Electrochromic switchable** has been selected as the Type (above).



Note: When selecting the **2-Mid-pane** position option you must ensure that there is sufficient gap between the innermost and second pane in the [Glazing](#) window gas layer to accommodate the blind [Slat width](#).

Control type

Control data specifies how the shading device is controlled. It determines whether the shading device is “on” or “off.” For [blinds and shades](#), when the device is “on” it is assumed to cover all of the window except its frame; when the device is “off” it is assumed to cover none of the window (whether “on” or “off” the shading device is assumed to cover none of the wall that the window is on). For [switchable glazing](#), “on” means that the glazing is in the fully-switched state and “off” means that it is in the unswitched state; for example, for electrochromic glazing, “on” means the glazing is in its darkest state and “off” means it is in its lightest state.

The choices for **Shading control type** are the following. *Equivalent EnergyPlus control types* are shown in italics. If a setpoint is applicable its units are shown in parentheses.

- **1-Always on** (*AlwaysOn*) - Shading devices are always activated.

The following six control types are used primarily to reduce zone cooling load due to window solar gain.

- **2-Daylight** (*MeetDaylightIlluminanceSetpoint*) - Used only with Position = **4-Switchable** in zones with [daylighting controls](#). In this case the transmittance of the glazing is adjusted to meet the [daylight illuminance set point](#) for the main photoelectric sensor used to control electric lighting according to daylight availability. This type of control assures that there is just enough daylight transmitted through windows with electrochromic shading to meet the daylighting requirements in a zone, and no more, thus reducing the cooling load. Note that the [daylight illuminance set point](#) is specified under [Environmental Control](#) on the Activity tab and that there is no option to apply separate setpoints for electrochromic shading and electric lighting control.
- **3-Schedule** - Shading operation is defined by a time only through a schedule. When the schedule has value 1 then shading operates, otherwise it does not.
- **4-Solar** (*OnIfHighSolarOnWindow*) - Shading is active if beam plus diffuse solar radiation incident on the window exceeds the [Solar setpoint](#).
- **5-Glare** (*OnIfHighGlare*) - Shading is on if the total daylight glare index at the zone's first daylighting sensor from all of the exterior windows in the zone exceeds the maximum glare index specified in the daylighting input for zone.

Note: Glare control requires Lighting control to be switched on in the zone.

- **6-Outside air temp** (*OnIfHighOutsideAirTemp*) - Shading is on if outside air temperature exceeds [Outside air temperature setpoint](#).
- **7-Inside air temp** (*OnIfHighZoneAirTemp*) - Shading is on if zone air temperature in the previous time step exceeds the [Inside air temperature setpoint](#).
- **8-Cooling** (*OnIfHighZoneCooling*) - Shading is on if zone cooling rate in the previous time step is non-zero.

The following two control types can be used to reduce zone cooling load. They are applicable to any Shading type but are most appropriate for interior or exterior blinds, interior or exterior shades with low insulating value, or switchable glazing.

- **9-Day cooling and solar + night** (*OnNight/OnDayIfCoolingAndHighSolarOnWindow*) - Shading is on at night. Shading is on during the day when the solar radiation incident on the window exceeds the [Solar setpoint](#) and the zone cooling rate in the previous time step is non-zero.
- **10-Day cooling and solar** (*OffNight/OnDayIfCoolingAndHighSolarOnWindow*) - Shading is off at night. Shading is on during the day when the solar radiation incident on the window exceeds the [Solar setpoint](#) and if the zone cooling rate in the previous time step is non-zero.

The following three control types can be used to reduce zone heating load during the winter by reducing window conductive heat loss at night and leaving the window unshaded during the day to maximize solar gain. They are applicable to any Shading type but are most appropriate for interior or exterior shades with high insulating value ("movable insulation"). "Night" means the sun is down and "day" means the sun is up.

- **11-Night outside low air temp** (*OnNightIfLowOutsideTemp/OffDay*) - Shading is on at night if the outside air temperature is less than [Outside air temperature setpoint](#) and schedule, if specified, allows shading. Shading is off during the day.
- **12-Night inside low air temp** (*OnNightIfLowInsideTemp/OffDay*) - Shading is on at night if the zone air temperature in the previous time step is less than the [Inside air temperature setpoint](#) and schedule, if specified, allows shading. Shading is off during the day.
- **13-Night heating** (*OnNightIfHeating/OffDay*) - Shading is on at night if the zone heating rate in the previous time step is greater than zero and schedule, if specified, allows shading. Shading is off during the day.

The following two control types can be used to reduce zone heating and cooling load. They are applicable to any Shading type but are most appropriate for translucent interior or exterior shades with high insulating value ("translucent movable insulation").

- **14-Night outside low air temp + day cooling** (*OnNightIfLowOutsideTemp/OnDayIfCooling*) - Shading is on at night if the outside air temperature is less than the [Outside air temperature setpoint](#). Shading is on during the day if the zone cooling rate in the previous time step is non-zero.
- **15-Night heating + day cooling** (*OnNightIfHeating/OnDayIfCooling*) - Shading is on at night if the zone heating rate in the previous time step is non-zero. Shading is on during the day if the zone cooling rate in the previous time step is non-zero.
- **16-Horizontal solar** (*OnIfHighHorizontalSolar*) - Shading is on if total (beam plus diffuse) horizontal solar irradiance exceeds the [Solar setpoint](#) and schedule, if specified, allows shading.
- **17-Outdoor air temp + Solar on window** (*OnIfHighOutdoorAirTempAndHighSolarOnWindow*) - Shading is on if the outside air temperature exceeds the [Outside air temperature setpoint](#) and if the solar radiation incident on the window exceeds the [Solar setpoint](#).
- **18-Outdoor air temp + Horizontal solar** (*OnIfHighOutdoorAirTempAndHighHorizontalSolar*) - Shading is on if the outside air temperature exceeds the [Outside air temperature setpoint](#) and if the horizontal solar radiation on the window exceeds the [Solar setpoint](#).

Solar setpoint

The solar setpoint can be defined when one of these shading control types has been selected:

- **4-Solar**
- **9-Day cooling and solar + night**
- **10-Day cooling and solar**
- **16-Horizontal solar**
- **17-Outdoor air temp + Solar on window**
- **18-Outdoor air temp + Horizontal solar**

When the solar radiation (in W/m² or BTU/h-ft²) is higher than the setpoint entered here and any other required condition are met (control types 9, 10, 17 and 18) and the schedule allows shading operation then the window shading is activated.

Inside air temperature setpoint

The Inside air temperature setpoint can be defined when one of these shading control types has been selected:

- **7-Inside air temp**
- **12-Night inside low air temp**

For cooling control types, when the inside air temperature (in °C or °F) is *higher* than the setpoint entered here and the schedule allows shading operation, the window shading is activated. For heating control types the logic is reversed and shading is activated when the temperature is *below* this setpoint.

Outside air temperature setpoint

The Outside air temperature setpoint can be defined when one of these shading control types has been selected:

- **6-Outside air temp**
- **11-Night outside low air temp**
- **14-Night outside low air temp + day cooling**
- **17-Outdoor air temp + Solar on window**
- **18-Outdoor air temp + Horizontal solar**

When the outside air temperature (in °C or °F) is higher than the setpoint entered here and the schedule allows shading operation, the window shading is activated.

Slat Angle Control

Where a [Window shading](#) system from the **Slatted blinds** category is selected, the angle of the slats must be defined using the settings below.

Slat angle control type

The angle of the blind slats can be controlled using 3 options:

- **1-Fixed slat angle**, the default option where the angle of the slats is fixed to the [Slat angle](#) set on the Window shading dialog for the whole simulation.
- **2-Scheduled slat angle** where the angle of the slats is defined by a schedule selected on the **Slat angle schedule** below.
- **3-Block beam solar** where the slats are automatically orientated to be perpendicular to the direction of the sun's rays.

Slat angle schedule

The schedule defines the angle of the slats for every timestep in the simulation. This setting is only available when the **2-Scheduled slat angle** [Slat angle control type](#) is set. See Window shading dialog for the definition of [Slat angle](#).

Note: the operation of the blinds is controlled independently of the slat angle.

Operation

Operation schedule

For all shading control options apart from **Glare** and **Always on**, the shading can only be active when this schedule value is 1. Select the schedule **On** if the blind is always activated or if the only control is one of the others listed above.

Limitations

- If two daylight zones share an interior window, neither zone can have an exterior window whose shading device does glare control (i.e. [Control type](#) is **5-Glare**).
- If two daylight zones share an interior window, neither zone can have an exterior window with daylight control (i.e. [Control type](#) is **2-Daylight**).

Window Operation And Free Aperture

[Openings tab](#) in model data under **External Glazing**, **Internal Glazing** and **Roof Glazing** headers

Operation

The operation of windows can be defined by entering data under the **Operation** header. This data is only shown:

- When [Natural ventilation](#) is activated on the HVAC tab at the current level and Calculated natural ventilation is selected, or,
- For Internal glazing or both Calculated and Scheduled natural ventilation.

Schedule definition

Select the method to be used to define the timing of the window operation. Options are:

- **1-Follow occupancy** - the window can be opened when the zone is occupied and not when it is not occupied.
- **2-Custom schedule** - the operation of the window is controlled by a specific schedule which is displayed when this option is selected.

Operation schedule

When the **Schedule definition** method (above) has been set to **2-Custom schedule** the schedule to be used to define window opening times can be selected. It defaults to the occupancy schedule based on the most recently loaded [Activity template](#).

Control option (internal windows only)

The control option allows you to define how *internal windows* behave in natural ventilation systems and whether they operate purely based on a schedule or, when using Calculated natural ventilation, whether they are linked to the main natural ventilation controllers defined on the HVAC tab. This data is not available for external windows. The options are:

For **Calculated** natural ventilation:

- **1-Constant** - the size of the internal window airflow opening (when open) is constant and the opening of internal windows is controlled entirely by schedule, i.e. it is not affected by any of the natural ventilation controls on the HVAC tab. In particular the [Internal control mode](#) setting on the HVAC tab is not used.
- **2-Use Internal control mode** - the internal windows are controlled using natural ventilation [Internal control mode](#) settings on the HVAC tab in combination with the operation schedule described below.

Note: When using Calculated natural ventilation and the **1-Constant** control option is selected, airflow through internal windows will occur (when enabled by the operation schedule) regardless of whether natural ventilation is [activated](#) on the HVAC tab. The **2-Use Internal control mode** option should only be selected when natural ventilation has been [activated](#) on the HVAC tab.

For **Scheduled** natural ventilation:

- **1-Control by schedule** - the internal windows are controlled entirely by schedule and are not affected by any of the natural ventilation controls on the HVAC tab.
- **2-Closed** - the internal windows are always closed and do not allow any airflow to be exchanged.

Note: When using Scheduled natural ventilation and the **1-Control by schedule** control option is selected, airflow through internal windows will occur (when enabled by the operation schedule) regardless of whether natural ventilation is [activated](#) on the HVAC tab.

Operation schedule

The schedule that specifies when window venting is available. A zero or negative schedule value means window venting is not allowed. A value greater than zero means venting can occur if other venting control conditions (specified by natural ventilation [Control Mode](#)) are satisfied.

Note: Fractional availability schedules do not restrict the venting in any way. When the schedule value > 0 the window can open, when it is 0 the window is closed.

An exception to this rule is for [CIBSE TM59:2026](#) simulations when using the [Restrict venting factor by schedule](#) option which, when set, does allow time-varying window aperture restriction to be defined through this schedule.

This schedule allows you to turn off venting at certain times of the day (at night, for example), of the week (on weekends, for example), or of the year (during the winter, for example).

If used with natural ventilation [Control Mode](#) = **4-Constant**, the ventilation rate is constant only when this schedule allows venting; otherwise the ventilation rate is set to zero. If Ventilation Control Mode = **1-NoVent**, this schedule has no effect.

The windows are 'open' when:

$T_{\text{zone_air}} > T_{\text{setpoint}}$ AND $T_{\text{zone_air}} > T_{\text{outside_air}}$ AND the schedule value = 1

The ventilation rate is calculated based on the pressure difference across the opening calculated from wind and stack pressure effects using equations described in [Natural Ventilation Modelling](#).

Free Aperture

The size and position of window opening airflow paths can be defined by entering data under the **Free Aperture** header. This data is used in both:

- a. EnergyPlus Calculated natural ventilation Airflow network simulations and
- b. CFD simulations

Note: While free aperture data is shown when the Scheduled natural ventilation model option is selected, this data is not used in EnergyPlus simulations. In this case the data is only made available to allow it to be used for CFD boundary conditions.

Opening position

The position of the opening in the window can be selected from

- **1-Top** - the aperture is in the top of the window. Air flow in and out will flow through the top of the window.
- **2-Bottom** - the aperture is in the bottom of the window. Air flow in and out will flow through the bottom of the window.
- **3-Right** - the aperture is in the right of the window (looking from outside). Air flow in and out will flow through the right of the window.
- **4-Left** - the aperture is in the left of the window (looking from outside). Air flow in and out will flow through the left of the window.

Windows should be rectangular to have an airflow opening in them and the position of the opening on the window can be defined by selecting from the above list. Options **3-Right** and **4-Left** will give identical simulation results as the EnergyPlus Airflow Network only accounts for the z-position of openings. The difference is useful for CFD simulations though.

Note: Non-rectangular windows (or rotated rectangular windows) can still be used as airflow openings. In this case for Airflow Network opening size purposes, EnergyPlus treats triangular windows as if they were rectangular with height and width based on the maximum dimensions in each direction.

% Glazing area opens

The % Glazing area opens data allows you to define the openable window "free area" as a percentage of the window area. For example enter 20 if the window free area is 20% of the total glazed window area. In simple terms this is the fraction of the window that can be opened, but it should also take into account the effect of any obstructions and in practice a more correct definition for the free area is the "cross-sectional (measurable, geometric) area".

The "free area" is multiplied by the [discharge coefficient](#) to give the "effective area" for the Airflow Network calculations within EnergyPlus.

Note: When calculating the **% Glazing area opens** you should divide the openable window free area by the total glazed area of the window, excluding the frame. The expandable section below under [Opening Areas For Fully Opening Windows with Frames](#) provides advice on how to deal with fully opening windows with large frames to avoid under-representing airflow.

Tip: A good rule of thumb to approximate of the free area for a top hung window is the bottom rectangular "opening area" plus 2 x the triangular side opening area. For closely spaced windows the side triangles should be reduced to account for the reduced effective opening area. A similar calculation can be made for side and bottom hung windows.

A couple of good reference for how to measure or otherwise obtain this data are:

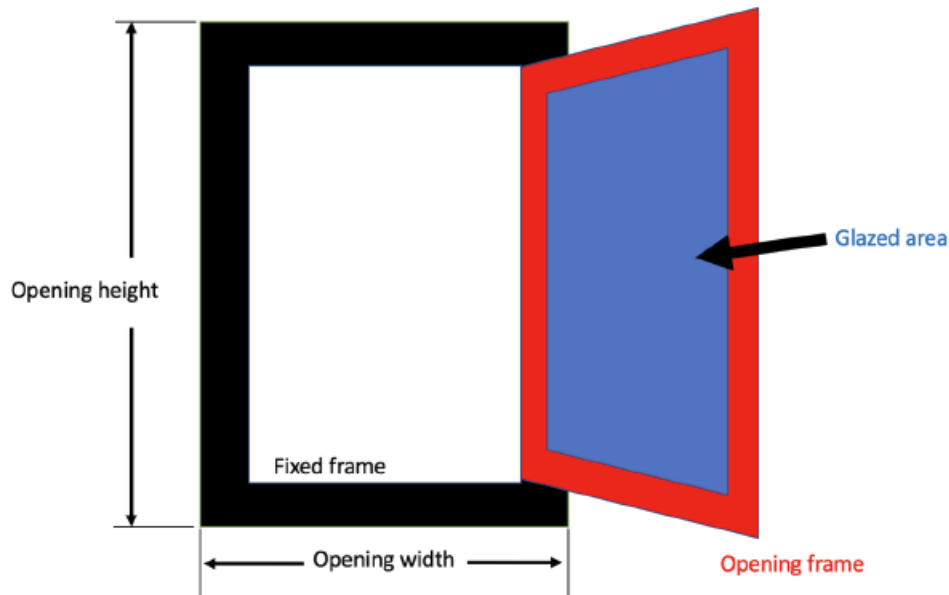
- [A review of ventilation opening area terminology](#) by, Benjamin M. Jones, Malcolm J. Cook, Shaun D. Fitzgerald, Christopher R. Iddon., 2015.
- "Air of Credibility" in the CIBSE Journal May 2016 (same authors as above).

Note: if [modulation](#) is switched on (it isn't by default) the glazing opening areas defined here are multiplied by a factor to account for external conditions in EnergyPlus Airflow Network simulations when using Calculated natural ventilation.

Tip: In some cases it is best to switch off the [Lump similar windows on surface](#) model option when using Calculated natural ventilation. Particular cases where this is important are a) when there are windows of the same size and type but different **% Glazing area opens** values on the same surface and b) where the air flow rate through individual windows is required, e.g. when preparing boundary conditions for a linked EnergyPlus CFD study.

Opening Areas For Fully Opening Windows with Frames

Because EnergyPlus only uses the glazed area for window airflow calculations, for windows with significant frames that fully open, a correction needs to be applied to avoid under-representing the airflow in the simulation.



Considering the example in the image above, when drawing a glazed opening such as a window or patio door, the opening height and width are entered including the frame. The width of the frame, which includes any fixed parts (black) and opening parts (red) is entered as a single frame width figure and DesignBuilder subtracts this to give the glazed area.

However, because only the glazed area is used for airflow calculations, the airflow through the opening would be under-represented if we simply entered **% Glazing area opens** to be 100% without further correction. The wider the frame, the bigger the issue. For example, a typical uPVC glazed door might have a frame width of 0.14m, of which 0.04m is the fixed frame (black) and 0.1m is part of the opening area (red) along with the glazing. For a 2.1m high by 0.9m wide glazed door, the free area opening size when fully open (subtracting the black fixed frame area) would be 1.66 m², which is 1.47 times the glazed only area in blue (1.13 m²) used in the airflow network ventilation calculations.

This issue can have significant implications for airflow modelling in some overheating assessments. It is not possible to correct for it by increasing the glazed area (either by increasing the opening size or reducing the frame size) because that would affect solar gains.

In our example case of the large glazed door, with 100% openable area there is no scope to increase the **% Glazing area opens** beyond 100%. **The recommended workaround is to modify the discharge coefficient to account for the missing frame area in the airflow model.** In our example, multiplying the default discharge coefficient for the window (0.65) by 1.47 (actual ventilation free area divided by glazed only area) gives a discharge coefficient of 0.95 which can be entered to compensate for the under representation of opening size.

Credit: Stephen Porritt for this workaround and the image.

Internal Window operation in CFD

Like other internal openings, internal windows cannot have airflow specified to define air movement in CFD simulations. Because CFD is for a snapshot in time and so does not use schedules, if you have loaded simulation results for a time where an internal window was closed *due to the schedule or some control setting*, to ensure that you see the same behaviour in CFD you can set the **% Glazing area opens** to zero in a copy of the simulation model.

Discharge coefficient

The constant discharge coefficient for the window (C_d) is used for Calculated natural ventilation only. The value must be between 0.0 and 1.0. Separate values can be entered for internal, external and roof windows.

For open doors and holes, a discharge coefficient can be entered on the Advanced tab of the model options dialog under the [Natural Ventilation](#) header.

Theory explaining the discharge coefficient

From the IEA Annexe 20 document [Air flow Through Large Openings in Buildings](#).

The EnergyPlus Airflow Network model is based on the following empirical powerlaw relationship between the flow and the pressure difference across a crack or opening in the building envelope:

$$Q = C(\Delta P)^n$$

The volumetric flow rate, Q [m³/s], is a simple function of the pressure drop, ΔP [Pa], across the opening. A common variation of the powerlaw equation is:

$$F = C(\Delta P)^n$$

where the mass flow rate, F [kg/s], is a simple function of the pressure drop. A third variation is related to the orifice equation:

$$Q = C_d A \sqrt{\frac{2\Delta P}{\rho}}$$

where:

C_d = discharge coefficient, and
 A = orifice opening area.

Theoretically, the value of the flow exponent, n , should lie between 0.5 and 1.0. Large openings are characterized by values very close to 0.5, while values near 0.65 have been found for small crack-like openings.

Suggested values for discharge coefficient

The literature suggests a large range of different values for discharge coefficient (symbol C_d often used in orifice equations). For example IEA Annexe 20 Air flow Through Large Openings in Buildings indicates that discharge coefficients reported in the literature vary between 0.3 and 0.8 and that it is not understood what causes the differences.

Other sources recommend a value close to 0.6. For example the [CONTAMW 2.0 User Manual](#) states that the *discharge coefficient, is related to the dynamic effects and is typically close to 0.6 for an orifice and slightly higher for other openings in buildings.*

ASHRAE propose the following correlation based on interzone temperature differences:

$$C_d = 0.4 + 0.0045 \delta T$$

for the range δT 0.5 to 40.

Given other uncertainties in natural ventilation calculations (wind pressure coefficients, effective areas of real-world openings and crack flows etc), using a discharge coefficient between 0.60 and 0.65 should provide sufficient accuracy.

Scheduled Natural Ventilation

Airflow through individual *external* windows, vents, doors and holes are not modelled explicitly with [Scheduled natural ventilation](#) because the natural ventilation airflow from outside is defined using the total [natural ventilation flow rate](#) and [operation](#) on the [HVAC](#) tab. *No operation data or free aperture needs to be entered on the Openings tab for external windows in this case.*

Note: Natural Ventilation can be [overridden to be off](#) on the HVAC tab for both Calculated and Scheduled model options.

Doors

[Openings tab](#) in model data

External Doors

External doors are used for modelling heat conduction. They can also provide airflow between the exterior and interior when the [Calculated natural ventilation](#) model option is selected.

Doors can be added to the model by navigating to the appropriate surface and either:

- Setting the **Auto generate** option and settings the Preferred width and height options. In this case, for surfaces that do not contain any custom openings, a single door will be positioned at the left side of the surface looking from outside.
- Alternatively, for more control over the placement position of the doors, you can draw the doors using the [Draw Door](#) command. Note that on surfaces where custom openings have been defined, any changes to door layout model data on the [Opening tab](#) is ignored.

Note: if you have drawn custom doors at the surface level then these will be used even if the **Auto generate** checkbox is not checked. The **Auto generate** checkbox only controls the automatic generation of doors, not their use in the calculations.

The additional door model data that can be added is described under:

- [Door Placement](#) - defines the size and position of door when using the Auto generate option.
- [Door Operation](#) - defines the times when doors can open.

Important Note: The external door operation settings are only made available for editing when the [Natural Ventilation > On](#) checkbox is checked on the HVAC tab.

Glazed doors

Doors use opaque constructions and if the door you wish to model is transparent then it should be modelled using a window. If only a portion of the door is glazed then the door could be modelled as part door and part window.

Internal Doors

Regardless of the Natural ventilation model option selected, data on internal doors can always be accessed. In this case the properties of the selected door component(s) define the resistance to heat conduction and airflow (see details below under **Door Modelling**).

Note: Internal door data is not available at zone level to avoid ambiguity when different settings are made in adjacent zones. Surface level internal door settings are generally inherited from block level, however settings for *inter-block* doors are inherited from the building level. For details on these rules see: "Zone Level Data for Internal Surfaces" on page 698.

Otherwise the same comments made above for External doors also apply to Internal doors.

Door Modelling

Calculated Natural Ventilation (Simulations only)

Each door is modelled using:

- An EnergyPlus **Door**. The heat conduction through the **Door** component is combined with the Wall and Partition results.
- An EnergyPlus **AirflowNetwork:MultiZone:Component:DetailedOpening** and other associated AirflowNetwork components, providing a flow path. You can define Door operation.

Scheduled Natural Ventilation (and Heating and Cooling Design Calculations)

Each door is modelled using:

- An EnergyPlus **Door**. The heat conduction through the **Door** component is combined with the Wall and Partition results.
- Outdoor air ventilation airflows are defined explicitly for zones as a whole and external doors do not contribute to this.
- Internal doors also have an EnergyPlus **ZoneMixing** object using [parameters](#) set in the [Model options](#) dialog. External doors do not exchange air with Scheduled natural ventilation.

More details on how door operation is modelled in the [Door Operation](#) help topic.

Door Placement

[Openings tab](#) in model data under Doors

On surfaces where no custom openings have been defined, you can create doors simply by checking the 'Doors fitted?' options under the External and Internal Doors headers. In this case a single door is positioned on each surface in place of one of the windows in the automatically generated facade layout.

Door Operation

[Openings tab](#) in model data under Doors

The operation of doors can be defined by entering data under Operation headers:

- **% Area door opens** - defines the area of the door which opens relative to the total area to give an approximation of the actual opening size and,
- **% Time door is open** - defines the average time the door is in the open position. Accounts for the fact that during occupied periods the door may open and close repeatedly.
- **Opening position** - defines where in the door the airflow opening is located. Select from:
 - 1-Top,
 - 2-Bottom,
 - 3-Right and
 - 4-Left

The way this data is used depends on the Natural ventilation model option set, but in both cases the concept of doors opening and closing can be approximated by reducing the size of the door opening area to account for the fact that the door is closed for much of the time.

Note: the same **Opening position** and **% Area door opens** data is used for both EnergyPlus Calculated natural ventilation and CFD calculations. **% Time door is open** is not used in CFD.

Internal Door operation in CFD

Like other internal openings, internal doors cannot have airflow specified to define air movement in CFD simulations. Because CFD is for a snapshot in time and so does not use schedules, if you have loaded simulation results for a time where an internal door was closed *due to the schedule setting*, to ensure that you see the same behaviour in CFD you can set the **% Area door opens** to zero in a copy of the simulation model.

Calculated Natural Ventilation

The door opening size used in the simulation is calculated by multiplying the **% Area door opens** by the **% Time the door is open** by the gross door area. The position of the opening within the door is defined by the opening position. The ventilation rate through the door opening is calculated at each time step based on the wind + stack pressure difference across the opening at the time.

Operation schedule

The Operation schedule is used to control timing of the door opening. Note that the schedule should have values of 0 and 1 only and intermediate fractional values do not indicate partial opening.

Scheduled Natural Ventilation

External door opening is not modelled explicitly with the Scheduled natural ventilation model data setting. Instead, the total natural ventilation flow rate and operation is defined for each zone on the [HVAC tab](#). No operation data is entered on the Openings tab for external doors.

Internal doors - when [parameters](#) set in the [Model options](#) enable it, airflow through doors is modelling using the EnergyPlus **ZoneMixing** object which mixes air between the two adjacent zones. DesignBuilder uses an approximate empirical relationship to calculate the mixing flow:

$$\text{Flow (m3/s)} = \text{Door area (m2)} * \% \text{ area door opens} * \% \text{ time door is open} * \\ \text{FAirflowRatePerOpeningArea}$$

where *FAirflowRatePerOpeningArea* is a factor used to obtain the mixing flow from the opening area. The [value of this factor](#) can be defined on the Model options dialog on the Advanced tab under Natural ventilation. The default value is 0.1 m3/s-m2.

A schedule can be applied to control timing of the mixing flow.

Note: The [Airflow through internal openings](#) model option must be checked to obtain airflow through internal doors when using Scheduled natural ventilation.

Operation schedule

The Operation schedule is used to control timing of the mixing flow. Note that the schedule should have values of 0 and 1 only and intermediate fractional values do not indicate partial opening.

Note: For both Scheduled and Calculated natural ventilation options, door opening is controlled entirely by schedule and is not affected by natural ventilation controls.

Vents And Holes

[Openings tab in model data](#)

External Vents

External vents are typically used for modelling airflow through external vent openings for the purposes of providing fresh air and/or zone cooling. They are only used when the [Calculated natural ventilation](#) model option is selected. When using [Scheduled natural ventilation](#) external vents should, in general, not be added to the model since, in this case, outdoor airflow is defined explicitly at zone level and the vent settings cannot be accessed to define the conductance and texture.

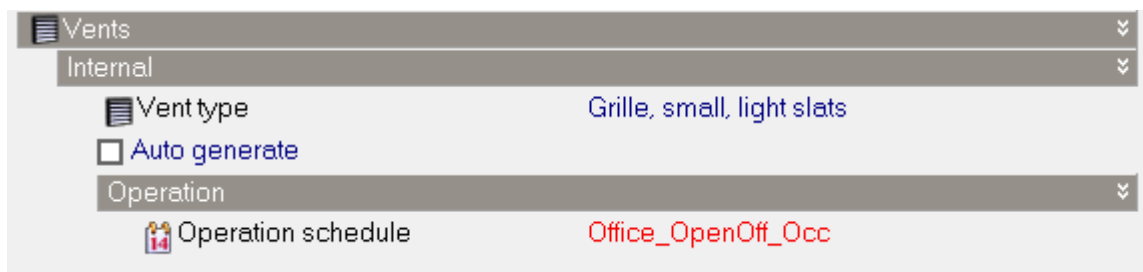
Vents can be added to the model by navigating to the appropriate surface and either:

- Setting the **Auto generate** option and settings the Preferred width and height options. In this case a vent will be positioned under each of the windows (if the vent spacing is the same as the window spacing), or,
- Alternatively, for more control over the placement position of the vents, you can draw the vents using the [Draw vent](#) command. Note that on surfaces where custom openings have been defined, any changes to vent layout model data on the [Opening tab](#) is ignored.

Note: if you have drawn custom vents at the surface level then these will be used even if the **Auto generate** checkbox is not checked. The **Auto generate** checkbox only controls the automatic generation of vents, not their use in the calculations.

Vent type

The [properties of the vent](#) are defined by the selection of the vent type:



Select the [vent component](#) that holds additional property data on the vent type such as the construction, texture and discharge coefficient to be applied.

The additional vent model data that can be added is described under:

- [Vent Placement](#) - defines the size and position of vents when using the Auto generate option.
- [Vent Operation](#) - defines the times when vents can operate.

Important Note: The external vent operation settings are only made available for editing when the [Natural Ventilation > On](#) checkbox is checked on the HVAC tab.

Roof Vents

Roof vents are defined as any vents placed on an external pitched or flat roof surface. The vent type for roofs is defined separately under the **Roof** header.

Although roof vents have a separate vent type selection you should use the same operation schedule as for External vents to define simulated vent opening availability. This data can be accessed under the Vents > External > Operation header.

Internal Vents

Regardless of the Natural ventilation model option selected, data on internal vents can always be accessed. In this case the properties of the selected vent component(s) define the resistance to heat conduction and airflow (see details below under [Vent Modelling](#)).

Note: Internal vent data is not available at zone level to avoid ambiguity when different settings are made in adjacent zones. Surface level internal vent settings are generally inherited from block level, however settings for *inter-block* vents are inherited from the building level. For details on these rules see: "Zone Level Data for Internal Surfaces" on page 698.

Otherwise the same comments made above for External vents also apply to Internal vents.

Vent Modelling

Calculated Natural Ventilation (Simulations only)

Each vent is modelled using:

- An EnergyPlus [Door](#). The heat conduction through the vent [Door](#) component is combined with the Wall and Partition results.
- An EnergyPlus [AirflowNetwork:MultiZone:Component:DetailedOpening](#) and other associated AirflowNetwork components, providing a controllable flow path. You can define [Vent operation](#).

Scheduled Natural Ventilation (and Heating and Cooling Design Calculations)

Each vent is modelled using:

- An EnergyPlus [Door](#). The heat conduction through the vent [Door](#) component is combined with the Wall and Partition results.
- Outdoor air ventilation airflows are defined explicitly for zones as a whole and external vents do not contribute to this.
- Internal vents also have an EnergyPlus [ZoneMixing](#) object using [parameters](#) set in the [Model options](#) dialog. External vents do not exchange air explicitly with Scheduled natural ventilation.

Holes

[Holes](#) are modelled in a similar way to vents with some exceptions:

- Holes must be drawn at surface level.
- Airflow through holes is uncontrolled, i.e. the airflow path is active all the time.
- Drawing holes can be used as a way to [merge](#) two zones when the [Merge zones connected by holes](#) Model option is set.
- Holes are used internally with DesignBuilder to model the heat transfer and airflow through [Virtual partitions](#).
- The [discharge coefficient to be used for all holes](#) (and doors) in the model is defined on the Advanced tab of the Model options dialog under the Natural Ventilation header. The default discharge coefficient is 0.65 is used. For vents the [vent component](#) defines the discharge coefficient.

Hole Modelling

Calculated Natural Ventilation (Simulations only)

Each hole is modelled using an EnergyPlus sub-surface and an airflow connection:

- Sub-surface:
 - When using the [1-Minimum shadowing](#) or [2-Full exterior](#) solar distribution Model options external holes are modelled using 'perfectly clear' windows which transmit all solar radiation and visible light, while internal holes are modelled using EnergyPlus Doors and an [Infrared Transparent](#) construction.

- When using [3-Full interior and exterior](#) solar distribution, all holes are represented using 'perfectly clear' windows which transmit all solar radiation and visible light. In this case heat conduction through window components are displayed in results as **Internal Natural vent** and **External Natural vent** as appropriate.
- EnergyPlus **AirflowNetwork:MultiZone:Component:DetailedOpening** and other associated AirflowNetwork components, providing a non-controllable flow path.

Scheduled Natural Ventilation (and Heating and Cooling Design Calculations)

Each hole is modelled using an EnergyPlus sub-surface as follows:

- External holes are represented using 'perfectly clear' windows which transmit all solar radiation and visible light. In this case the heat conduction through windows is displayed in results as **External Natural vent**. External holes do not exchange air explicitly with Scheduled natural ventilation.
- Internal holes are modelled using an internal "door" between the 2 zones and, if the [Airflow through internal openings](#) Model option is selected, an EnergyPlus **ZoneMixing** object is also included using [parameters](#) set in the Model options dialog.

Vent Placement

[Openings tab in model data under Vents](#)

On surfaces where no custom openings have been defined, you can create vents simply by checking the **Auto generate** options under the External and Internal Vents headers. In this case vents are positioned on each surface using **Vent spacing** and **Vent height above floor** in the automatically generated facade layout. Typically, a single vent is positioned under each of the windows (if the vent spacing is entered to be the same as the window spacing).

Note: When using the **Auto generate** option, please be aware that, as for windows, vents can only be automatically generated for walls and sloping roofs. Auto generation isn't possible on horizontal surfaces such as flat roofs.

Note: It is often more convenient to place vents manually by drawing them at surface level or by copying previously drawn vents at building level.

Vent area

Enter the area of the area of each vent (in m2 or ft2).

Vent spacing

The spacing between the each vent on the facade (in m or ft). In many cases this will be the same as the [window spacing](#) if auto generated windows are also being used. The **vent spacing** is the centre-to-centre spacing between vents, not the gap between vents.

Vent height above floor

The height of the vent centre above the zone floor level (in m or ft).

Vent Operation

[Openings tab in model data under Vents](#)

The operation of vents can be defined by entering timing data under **Operation** headers.

Control option (internal vents only)

The control option allows you to define how *internal vents* behave in natural ventilation systems and whether they operate purely based on a schedule or, when using Calculated natural ventilation, whether they are linked to the main natural ventilation controllers defined on the HVAC tab. This settings is not available for external vents. The options are:

For **Calculated** natural ventilation:

- **1-Constant** - the size of the internal vent airflow opening (when open) is constant and the opening of internal vents is controlled entirely by schedule, i.e. it is not affected by any of the natural ventilation controls on the HVAC tab. In particular the [Internal control mode](#) setting on the HVAC tab is not used.
- **2-Use Internal control mode** - the internal vents are controlled using natural ventilation [Internal control mode](#) settings on the HVAC tab in combination with the operation schedule described below.

For **Scheduled** natural ventilation:

- **1-Control by schedule** - the internal vents are controlled entirely by schedule and are not affected by any of the natural ventilation controls on the HVAC tab.
- **2-Closed** - the internal vents are always closed and do not allow any airflow to be exchanged.

The way this data is used depends on the Natural ventilation model option set:

Calculated Natural Ventilation

The ventilation rate through the vent is calculated based on the wind and stack pressure calculated at each time step in the simulation.

Operation schedule

The Operation schedule is used to control timing of the vent opening availability. Note that the schedule should have values of 0 and 1 only and intermediate fractional values do not indicate partial opening.

Note: Vents on a roof surface use the operation schedule under the Vents > External > Operation header to define simulated vent opening availability.

Scheduled Natural Ventilation

External vent opening is not modelled explicitly with the Scheduled natural ventilation model data setting. Instead, the total natural ventilation flow rate and operation is defined for each zone on the [HVAC tab](#). *No operation data is entered on the Openings tab for external vents.*

Internal vents are modelled using the EnergyPlus **ZoneMixing** object which mixes air between the two adjacent zones. DesignBuilder uses an approximate empirical relationship to calculate the mixing flow:

$$\text{Flow (m3/s)} = \text{Door area (m2)} * \% \text{ area door opens} * \% \text{ time door is open} * \text{FAirflowRatePerOpeningArea}$$

where *FAirflowRatePerOpeningArea* is a factor used to obtain the mixing flow from the opening area. The [value of this factor](#) can be defined on the Model options dialog on the Advanced tab under Natural ventilation. The default value is 0.1 m3/s-m2.

A schedule can be applied to control timing of the mixing flow.

Note: The [Airflow through internal openings](#) model option must be checked to obtain airflow through internal vents when using Scheduled natural ventilation.

Operation schedule

The Operation schedule is used to control timing of the mixing flow. Note that the schedule should have values of 0 and 1 only and intermediate fractional values do not indicate partial opening.

Wind Pressure Coefficients

[Openings tab in model data](#)

You can enter [Wind pressure coefficients](#) (WPC) for wind incidence angles in 45° increments for each surface in the model. DesignBuilder automatically sets up default WPC data as surfaces are created using published [Wind Pressure Coefficient template data](#).

Wind pressure coefficients are only available when using the Calculated natural ventilation model option.

Note: the wind incidence angles are relative to the surface normal for each surface.

Lighting Model Data

[Lighting tab in model data](#)

Templates

You can load generic Lighting data by clicking on the Template option under Lighting Template and then by clicking on the '...' at the right of the line.

When you make a selection from the list of [Lighting templates](#), data from selected template will be loaded to the model.

With [Early gains](#), the lighting gains in the space are separated into Task and General lighting.

You can also open these headers to enter lighting details where they are different from those loaded from the template:

- [General Lighting](#)
- [Task Lighting](#)
- [Lighting Control](#)
- [Exterior Lighting](#)
- "Lighting Cost" on page 467

When you load lighting data from template, the actual general lighting internal gain is calculated by dividing the W/m²/100 lux data in the template by the required [illuminance level](#) as set on the [Activity tab](#). Consequently you should make sure the correct activity data has been loaded in each zone before loading lighting templates. Also, you may need to explicitly load data in each zone by checking each zone in the Target tab of the 'Load data from template' dialog.

Lighting data generates simulation data at the **Zone level**.

See also the [Lighting data and daylight controls](#) tutorial

General Lighting

[Lighting tab in model data](#)

General lighting accounts for the main lighting in the room. It is assumed to be able to be replaced by daylight through lighting control.

Units

When the [Gain definition level](#) model option is set to **2-Zone by zone in model data** you can select the units to be used for lighting here on the Lighting tab. Choose from the options:

- **1-Power density** - where the General lighting gains are defined per floor area. This method has the advantage that the level of gains can clearly be understood by looking at W/m² or W/ft² values and floor area. However it can cause problems when loading data from lighting templates in that if the activity is changed subsequently with a different illuminance requirement, the lighting power density is not updated. This issue has led to the introduction of the second option.
- **2-Normalised power density** - Although the units for Normalised power density (W/m²-100 lux or W/ft²-fc) may seem quite obscure, this method of defining lighting gains has advantages over using simple Power density. Namely that the actual lighting level is associated with the lighting system type without the need to refer to the zone activity and hence illuminance requirement. So if a particular lighting system type is installed throughout a building, but the building has a range of activities each with different illuminance requirements, it will usually be possible to enter the lighting **Normalised power density** value once only at building level, set the activity for each zone and obtain realistic lighting gains.
- **3-Absolute zone power** - where the lighting gains are defined as W without reference to the zone floor area. This option is used when the actual absolute lighting gains levels are known in W. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Note: When using the **3-Absolute zone power** option, the lighting gains data is normally entered individually for each zone because values inherited from building and block levels will typically not be correct.

Gain

Set the design level of energy consumption due to general lighting. Depending on the [Lighting gain units](#) model option setting, the gains data is entered as **Power density** (in W/m² or W/ft²) or as **Normalised power density** (W/m²-100lux or W/ft²-fc) or as **Absolute zone power** (in W).

The fuel is assumed to be *electricity*.

Power density

When the **1-Power density** [Lighting gain units](#) model option is selected gains are entered in W/m² or W/ft².

Note: When loading lighting data from template and using the **1-Power density** option, the actual lighting gain per area is calculated by dividing the Normalised power density data in the template by the required [illuminance level](#) as currently set on the [Activity tab](#). If you select a new Activity with different illuminance level requirements, the Lighting tab does not automatically update the Power density.

Normalised power density

When the **2-Normalised power density** [Lighting gain units](#) model option is selected the maximum lighting gains are defined as W/m²-100lux or W/ft²-fc and the actual lighting energy used for the zone in the simulation is based on this value plus floor area and illuminance requirements as follows:

Max Lighting power (W) = Lighting energy (W/m2-100lux) x Zone floor area (m2) x [Zone Illuminance requirement](#) / 100

Absolute zone power

When the **3-Absolute zone power** [Lighting gain units](#) model option is selected the lighting gains are defined as W without reference to the zone floor area. This option used when the actual absolute lighting gains levels are known in W. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Calculations

For all of the above 3 options, the lighting energy should be the maximum electrical power input to lighting in a zone, including ballasts, if present. This value is multiplied by a schedule fraction (below) during the simulation to get the lighting power in a particular time step.

The electrical input to lighting ultimately appears as heat that contributes to zone loads or to return air heat gains. In EnergyPlus this heat is divided into four different fractions. Three of these are given by the input fields Return Air Fraction, Fraction Radiant and Fraction Visible. A fourth, defined as the fraction of the heat from lights convected to the zone air, is calculated by EnergyPlus as:

Convected Fraction = 1.0 – (Return Air Fraction + Radiant Fraction + Visible Fraction)

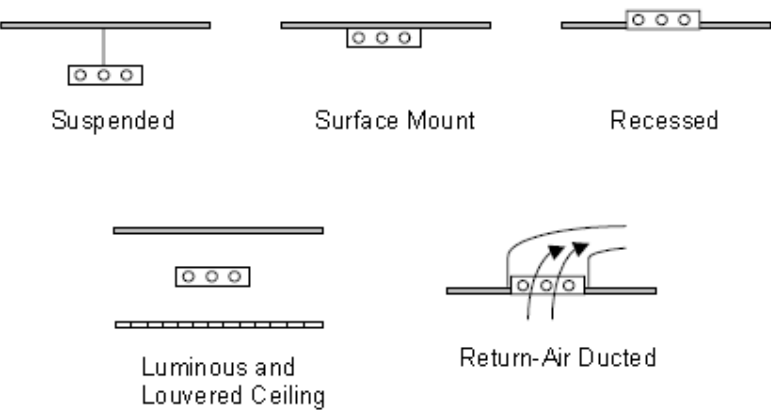
Note: An error message is displayed if Return Air Fraction + Radiant Fraction + Visible Fraction exceeds 1.0. These fractions depend on the type of lamp and luminaire, whether the luminaire is vented to the return air, etc.

Schedule

If the [Internal gains operate with occupancy](#) model option is not set (the default state) then General lighting operation is governed by the operation schedule selected here. The level of lighting gains are multiplied by the time-varying factor in the schedule to give the gain for each timestep in the simulation. Otherwise, if the [Internal gains operate with occupancy](#) model option is set then these gains are synchronised with the [Occupancy schedule](#).

Luminaire type

Select from:



When you select using the drop-down list an appropriate set of defaults for the luminaire type is loaded according to the table below, assuming fluorescent lighting.

Data	1-Suspended	2-Surface Mount	3-Recessed	4-Luminous and Recessed Ceiling	5-Return-air Ducted
Return Air Fraction	0.0	0.0	0.0	0.0	0.54
Radiant Fraction	0.42	0.72	0.37	0.37	0.18
Visible Fraction	0.18	0.18	0.18	0.18	0.18
Convected Fraction	0.40	0.10	0.45	0.45	0.10

Approximate values of Return Air Fraction, Radiant Fraction and Visible Fraction for overhead fluorescent lighting for different luminaire configurations. These values assume that no light heat goes into an adjacent zone. Source: *Lighting Handbook: Reference & Application, 8th Edition, Illuminating Engineering Society of North America, New York, 1993, p. 355.*

See below for more [Luminaire Performance Data](#).

Return air fraction

For Return Air Ducted luminaires this is the fraction of the heat from lights that is convected out of the room and into the zone return air (normally into a return plenum). If the return air flow is zero or the zone has no return air system, the program will put this fraction into the zone air. Return Air Fraction should normally be greater than zero for **5-Return-air ducted** luminaires. This value is not used for non-return air ducted luminaires, i.e. for **Luminaire types** 1-4.

Radiant fraction

The fraction of heat from lights that goes into the zone as long-wave (thermal) radiation. The program calculates how much of this radiation is absorbed by the inside surfaces of the zone according the area times thermal absorptance product of these surfaces.

Fraction visible

The fraction of heat from lights that goes into the zone as visible (short-wave) radiation. The program calculates how much of this radiation is absorbed by the inside surfaces of the zone according the area times solar absorptance product of these surfaces.

Note: selecting an option from the **Luminaire type** drop-down list loads default Return Air Fraction, Radiant Fraction and Visible Fraction data from the table above.

Fraction convected

The fraction of the heat from lights convected to the zone air is calculated from:

Fraction convected = 1.0 – (Return Air Fraction + Fraction Radiant + Fraction Visible)

You will get an error message if Return Air Fraction + Fraction Radiant + Fraction Visible exceeds 1.0.

This data is provided on screen for information purposes.

Luminaire Performance Data

Data on luminaire performance can also be obtained from ASHRAE Fundamentals 2013, 18.6 Table 3:

Table 3 Lighting Heat Gain Parameters for Typical Operating Conditions

Luminaire Category	Space Fraction	Radiative Fraction	Notes
Recessed fluorescent luminaire without lens	0.64 to 0.74	0.48 to 0.68	• Use middle values in most situations • May use higher space fraction, and lower radiative fraction for luminaire with side-slot returns • May use lower values of both fractions for direct/indirect luminaire • May use higher values of both fractions for ducted returns
Recessed fluorescent luminaire with lens	0.40 to 0.50	0.61 to 0.73	• May adjust values in the same way as for recessed fluorescent luminaire without lens
Downlight compact fluorescent luminaire	0.12 to 0.24	0.95 to 1.0	• Use middle or high values if detailed features are unknown • Use low value for space fraction and high value for radiative fraction if there are large holes in luminaire's reflector
Downlight incandescent luminaire	0.70 to 0.80	0.95 to 1.0	• Use middle values if lamp type is unknown • Use low value for space fraction if standard lamp (i.e. A-lamp) is used • Use high value for space fraction if reflector lamp (i.e. BR-lamp) is used
Non-in-ceiling fluorescent luminaire	1.0	0.5 to 0.57	• Use lower value for radiative fraction for surface-mounted luminaire • Use higher value for radiative fraction for pendant luminaire

Source: Fisher and Chantrasrisalai (2006).

The Space fraction in the above table from ASHRAE is the fraction of the gain entering the room and corresponds to 1 - Return air fraction.

Also, approximate values of Return Air Fraction, Fraction Radiant and Fraction Visible for overhead fluorescent lighting for a variety of luminaire configurations are provided below. The data is based on ASHRAE 1282-RP "Lighting Heat Gain Distribution in Buildings" by Daniel E. Fisher and Chanvit Chantrasrisalai.

Fixture No.	Luminaire Feature	Return Air Fraction	Fraction Radiant	Fraction Visible	fconvected
1	Recessed, Parabolic Louver, Non-Vented, T8	0.31	0.22	0.20	0.27
2	Recessed, Acrylic Lens, Non-Vented, T8	0.56	0.12	0.20	0.12
3	Recessed, Parabolic Louver, Vented, T8	0.28	0.19	0.20	0.33
4	Recessed, Acrylic Lens, Vented, T8	0.54	0.10	0.18	0.18
5	Recessed, Direct/Indirect, T8	0.34	0.17	0.16	0.33
6	Recessed, Volumetric, T5	0.54	0.13	0.20	0.13
7	Downlights, Compact Fluorescent, DTT	0.86	0.04	0.10	0.00
8	Downlights, Compact Fluorescent, TRT	0.78	0.09	0.13	0.00
9a	Downlights, Incandescent, A21	0.29	0.10	0.6	0.01
9b	Downlights, Incandescent, BR40	0.21	0.08	0.71	0.00
10	Surface Mounted, T5HO	0.00	0.27	0.23	0.50
11	Pendant, Direct/Indirect, T8	0.00	0.32	0.23	0.45
12	Pendant, Indirect, T5HO	0.00	0.32	0.25	0.43

Task And Display Lighting

[Lighting tab in model data](#)

Task and Display Lighting accounts for lighting used for specific tasks (e.g. desk lamps) or for displaying items (e.g. display lighting for goods in shops).

Note: Task and Display Lighting is not offset by lighting control.

The following other data is assumed:

Data	Value
------	-------

Radiant fraction	0.42
Visible fraction	0.18
Return air fraction	0

Gain

Enter the gain due to task and display lighting (in W/m² or W/ft²).

Note that when you load lighting data from template, the [Default lighting display density lighting gain](#) value on the Activity tab is copied here overriding any previously set value.

Operation

If 'Internal gains operate with occupancy' under model options is not set Task lighting has a specific operation schedule set under Task Lighting, otherwise these gains are synchronised with occupancy.

Lighting Control

[Lighting tab in model data](#)

You can control the electric lights according to the availability of natural daylight. When Lighting control is switched on, illuminance levels are calculated at every time step during Simulations and Cooling design calculations and then used to determine how much the electric lighting can be reduced. The daylight illuminance level in a zone depends on many factors, including sky condition, sun position, photocell sensor positions, location, size, and glass transmittance of windows, window shades and reflectance of interior surfaces. Reduction of electric lighting depends on daylight illuminance level, illuminance set point, fraction of zone controlled and type of lighting control.

Tip: When running Cooling design calculations you may prefer to size without lighting control, in which case you should switch the option off at building level and clear to default to ensure it is off in all zones.

Note: You should not use daylight control in zones without any windows - this will generate errors preventing calculations from running. You should therefore use the building and block level lighting control setting with care, checking for any zones without windows and switching off lighting control at the zone level.

Lighting control is available for [General lighting](#) but not for [Task and display lighting](#). If no general lighting is specified then lighting control won't be applied.

See also the [Lighting data and daylight controls](#) tutorial

Lighting Areas

When lighting control is switched on, by default all of the lights in the zone are controlled by the first (main) lighting sensor and **% Zone Controlled by Lighting Area 1** has value 100%. Some larger spaces may have **2 Lighting Areas**, each area having its own lighting sensor and covering its own area of the zone. The area is not defined geometrically in DesignBuilder - it simply reflects the fraction of the total General overhead electric lighting that can be dimmed by its lighting sensor.

You may wish to use 2 Lighting Areas for these situations:

1. A large zone may need more than one sensor if the output from main sensor does not represent the daylight available in other parts of the zone. For example a zone having facades in both South and East directions would need 2 sensors (and 2 Lighting Areas), one on each facade.
2. The zone has 2 different activity areas which have been lumped into a single zone for convenience. The 2 activities have different visual illuminance requirements and the Lighting Areas for both can benefit from daylighting through daylighting control. An example would be a zone having both office and recreational space. The office space may have a lighting requirement of 500 lux and the recreational space may need 300 lux.

Note 1: if the zone has a different sensor for the back half (for example) you should check the **Second light sensor** option.

Note 2: if the zone has **perimeter and core areas**, perimeter lights having daylight control and core lights not, then you only need to use a single lighting sensor. The fact that the core lights are not controlled can be modelled using **% Zone Controlled by Lighting Area 1**. If 60% of the lights in the zone are located in the core zone and do not have daylight control then set **% Zone Controlled by Lighting Area 1** to 40.

Perimeter Area

A typical perimeter area would be the areas of the zone that are close to a window. By convention perimeter areas are defined as the area within a fixed distance from a normal to the perimeter. In the UK NCM the fixed distance is 6m.

If you wish to create a separate thermal zone to model the different lighting and solar conditions in the perimeter area, you can use the [Offset snap](#) option to draw [Virtual partitions](#).

Daylight illuminance on the 'working plane' is calculated for up to two locations in each zone during the simulation and [illuminance setpoints](#) are maintained where necessary by [electric lighting](#). The sensor is always located in the [working plane](#) which, by default, is 0.8m above the floor. By default, when lighting control is switched on, a single photo sensor monitors the daylight illuminance at the first sensor location and controls the lights for the whole of the zone. You can however set up 2 **Lighting Areas** in each zone, by checking the **Second lighting area** check box. In this case you can set a target illuminance for the second lighting area separately and also define the percentage of the zone covered by Lighting Area 2.

Note 1: the floor area covered by the 2 sensors does not need to add to 100%. The lights in the remaining floor area operate without daylight control.

Note 2: the target illuminance for Lighting Area 1 is set on the Activity tab under [Environmental Control](#).

If there is only one lighting area then a percentage equal to

$$100 - (\% \text{Zone covered by Lighting Area 1})$$

is assumed to have no lighting control..

% Zone Controlled by Lighting Area 2

The fraction of the zone floor-area whose electric lighting is controlled by the daylight illuminance on the second sensor. This data is required if Lighting Area 2 is being used. A percentage equal to

$$100 - [\% \text{Zone Controlled by Lighting Area 1}] - [\% \text{Zone Controlled by Lighting Area 2}]$$

is assumed to have no lighting control (i.e. in this uncontrolled area of the space, the lights operate purely according to the schedule without any dimming/switching).

Procedure

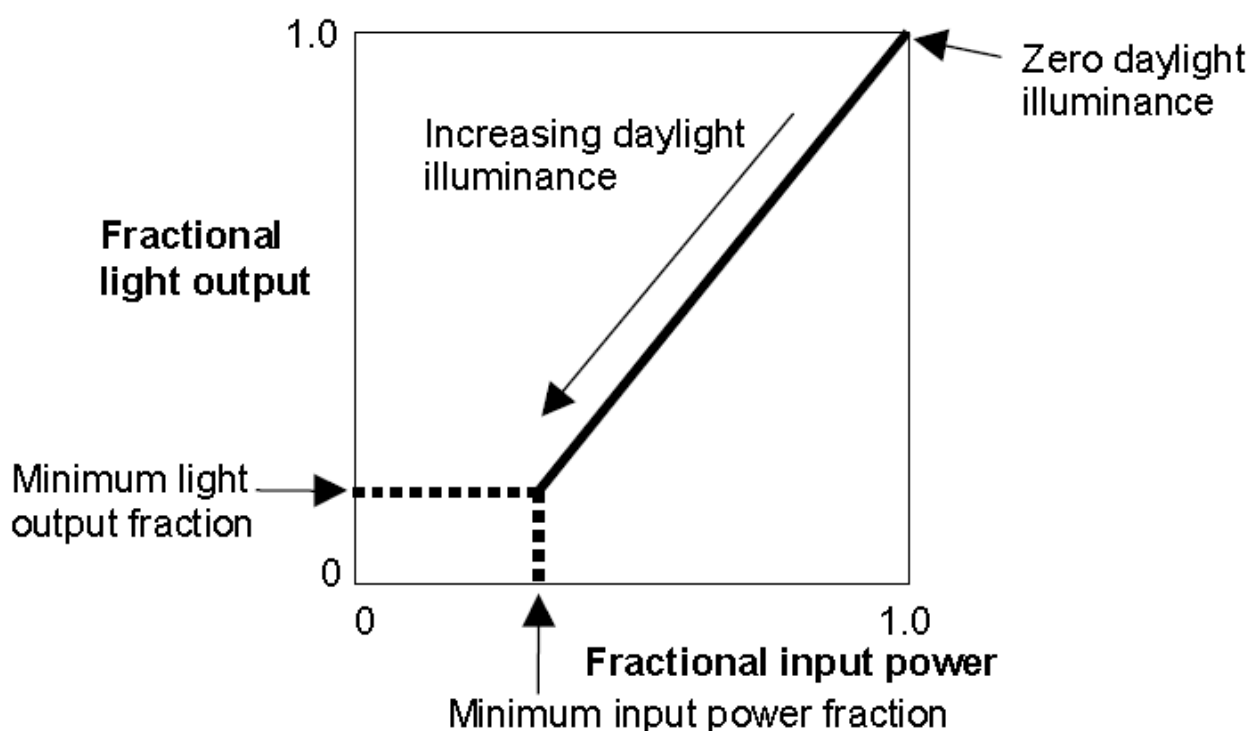
The procedure for introducing the lighting control is:

1. Check the **Lighting Control On** checkbox (at building level if all zones are to have lighting control).
2. Enter the working plane height which should normally be the height of the lighting control sensor above the top of the floor surface of the zone.
3. Select the type of control as described below.
4. [Position the first sensor](#) in each zone having lighting control. The z-position is set by the [Working plane height](#) on the Advanced tab of the Model options dialog.
5. Set the **% Zone covered by Lighting Area 1**. If all the lights in the zone are in Lighting Area 1 (i.e. they are controlled by the 1st (main) lighting sensor) then enter 100%.
6. If there are two lighting areas then [position the second sensor](#).
7. Set the **% Zone covered by Lighting Area 2**. If all the lights in the zone are in Lighting Area 2 (i.e. they are controlled by the 2nd lighting sensor) then enter $100 - \% \text{Zone covered by Lighting Area 1}$.

Note: by default the main lighting sensor is positioned in the geometric centre of the zone. It is important that you check the position of all lighting sensors to obtain accurate results. For example, if the lighting sensor is positioned right next to a window receiving large amounts of daylight, the lighting sensor will signal 'enough daylight' even if the rest of the zone is only dimly lit and electric lighting energy will be underestimated (daylight savings over-estimated).

Linear Control

With **Linear** control, the overhead lights dim continuously and linearly from maximum electric power, maximum light output to minimum electric power, minimum light output as the daylight illuminance increases. The lights stay at the minimum point with further increase in the daylight illuminance.



The **Minimum input power fraction** for **Linear** control type is the lowest power the lighting system can dim down to, expressed as a fraction of maximum input power. For **Linear/off** lighting control, this is the power fraction reached just before the lights switch off completely.

The **Minimum output fraction** for **Linear** control type, is the lowest lighting output the lighting system can dim down to, expressed as a fraction of maximum light output. This is the fractional light output that the system produces at minimum input power. For **Linear/off** lighting control, this is the light output fraction reached just before the lights switch off completely.

Note: Linear control provides an idealised lighting control mechanism which can be useful for calculating upper limits on the potential for savings when using natural daylight to offset electric lighting.

Linear/Off Control

Linear/off control is the same as Linear control except that the lights switch off completely when the minimum dimming point is reached.

Stepped Control

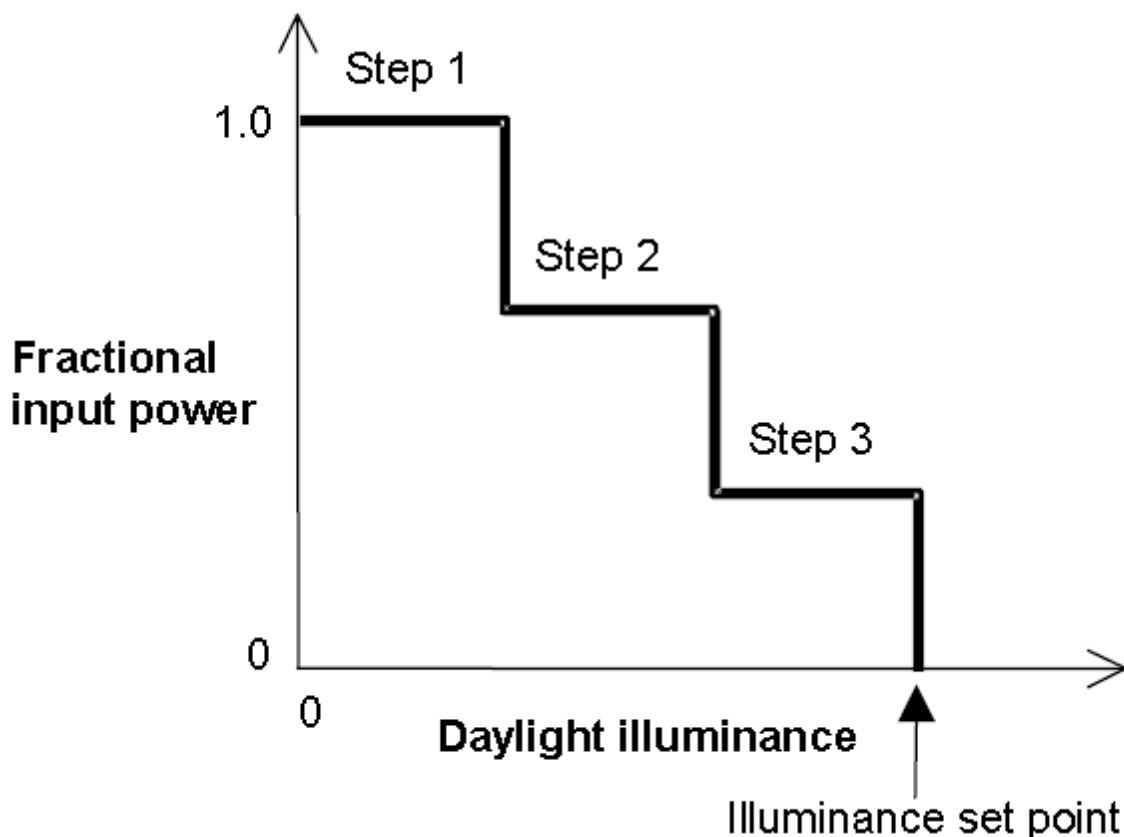
Stepped control allows you to switch lighting on/off according to the availability of natural daylight in discrete steps. Whereas the Linear control described above provides precisely controlled illuminance by dimming the lights, the stepped control models blocks of lights switching on/off according to the electric lighting requirement.

The electric power input and light output vary in discrete, equally spaced steps. The number of steps can be set.

Example of a Stepped Lighting Control System with 3 Steps:

Daylight illuminance	Fraction of lights that are on
0-200	1.0
200-400	$\frac{2}{3}$
400-600	$\frac{1}{3}$
600 and above	0.0

Graphical representation of above example:



See also [EnergyPlus Daylight Calculations](#).

Glare

Maximum allowable glare index

If a daylight zone has windows with [Window shading devices](#), the shades will be deployed if the daylight glare at sensor 1 exceeds the value of Maximum allowable glare index. To get this type of glare control you have to specify [Shading control type](#) as **5-Glare** for one or more windows in the zone (see [Window Shading Control](#)).

Recommended Values of Maximum Allowable Discomfort Glare Index:

Activity or zone type	Maximum Allowable Discomfort Glare Index*
Art Galleries	16
Factories	
Rough work	28
Engine assembly	26
Fine assembly	24
Instrument assembly	22
Hospital wards	18
Laboratories	22
Museums	20
Offices	22
School classrooms	20

*Source *EnergyPlus documentation*

View angle relative to y-axis

Daylight glare from a window depends on occupant view direction. It is highest when you look directly at a window and decreases as you look away from a window. This field specifies the view direction for calculating glare. It is the angle, measured clockwise in the horizontal plane, between the positive y-axis and the occupant view direction, i.e. it is the bearing of the persons line of sight. For example, with [Site orientation](#) = 0, in the Northern Hemisphere, 90 = Facing East, 180 = Facing South etc).

Algorithms

By default, DesignBuilder uses the EnergyPlus Daylighting:Detailed object to model the control of electric lighting according to daylight illumination levels. You can find more information on the algorithms used in the [EnergyPlus Daylight Calculations](#) section.

Output

For any zone simulated with daylighting controls, an illuminance map can be output with a 10 x 10 grid of additional daylighting reference points. The resulting map is output as a comma delimited text file called eplusmap.csv in the [EnergyPlus folder](#). This file can be imported into a spreadsheet program for rapid visualisation of the daylighting illuminance patterns in a zone. To generate this file, at least one zone must have daylight control and the Daylight map output selection on the Output tab of the Simulation output options dialog must be made.

Setting The Position Of Daylight Sensors

Working plane height

You can set the height of the working plane (otherwise known as desk height) to define the z-position of the lighting sensor. The x-y coordinate is set by [moving the lighting sensor](#) marker(s) at zone level.

The **Working plane height** is used for EnergyPlus daylighting simulations only and is not used in Radiance daylighting where a separate value is defined on the [Daylighting calculation options dialog](#).

Horizontal Position

To calculate the default position of the first and second daylight sensors DesignBuilder follows these steps:

1. For zones that are non-convex (zones where some surfaces cannot see all of the other surfaces in the zone, e.g. L-shaped zone) the floor plan is broken down into the smallest possible number of convex areas.
2. The geometric centre of the largest of the convex areas derived in step 1 is used as the basis for the position of the lighting sensors.
3. The specific position of the first and second sensors are 0.5m to the "left" and "right" of the geometric centre position derived in step 2.



You can move the sensor by following these steps:

1. Move to the zone level on the layout tab.
2. Select the sensor you wish to move. This enables the Move toolbar icon. You will only see the daylight sensor if lighting control is active in the current zone.
3. Click on the [Move](#) toolbar icon to start the move process.
4. Click once to define the reference or 'handle' position and click again to define the new position for the sensor.

If the second sensor is enabled in the zone then you will be able to position that in the same way.

See also: [Placing Light Sensors](#)

Exterior Lighting

[Lighting tab](#) in model data

Exterior lighting allows you to account for the energy consumption of lights used to illuminate the exterior of the building. Options are available for scheduled operation and nighttime switching.

Operation schedule

The schedule allows the exterior lights consumption to be operationally different, hour to hour as well as seasonally. Fractional values in the basic schedule are applied to the design level data below.

Design level

This data is normally used to represent the maximum electrical input to exterior lighting fixtures that is then multiplied by a schedule fraction (see previous field). Alternatively, the lighting design level could be a "diversity factor" applied to a schedule of real numbers.

Control Option

This data is used to determine how the exterior lights are controlled. There are currently two options:

- **Schedule only** where the exterior lights always follow the operation schedule.
- **Schedule + override off on daytime** where the exterior lights will not run when the sun is up, no matter what the schedule indicates. Using this option makes it simple to model exterior lights that are controlled by a photocell or other controller that ensures that outdoor lights will not run during the daytime. However, this control works off of the position of the sun and therefore does not operate exactly like a photocell. During the night, the schedule values are still applied in the usual way.

Lighting Cost

[Lighting tab in model data](#)

The electric lighting cost is calculated zone by zone by multiplying the zone floor area by the cost per floor area (GIFA) for the zone on the Lighting tab. This data will normally be loaded from the Lighting template to ensure that lighting performance and cost data are consistent.

Tip: The [Cost data](#) help topic provides an overview of the places in DesignBuilder where costs are entered.

Cost per area

Enter the cost per floor area (GIFA) for the lighting system.

HVAC Model Data

[HVAC tab in model data](#)

Templates

You can load generic HVAC data by clicking on the Template option under HVAC Template and then by clicking on the '...' at the right of the line.

When you make a selection from the list of [HVAC templates](#), data from selected template will be loaded to the model.

HVAC Template	
Template	Fan Coil Unit (4-Pipe), Air cooled Chiller
System	Default

System

The **System** setting data is used to group zones together that are served by the same AHU in Heating and Cooling design calculations. This allows the heating and cooling capacity of the AHU to be calculated and displayed on the Cooling design and Heating design screens.

By default DesignBuilder provides a single System called **Default** and all zones are allocated to it. You can create new Systems and select them in the same way that any other component would be created and selected. Systems have no data other than a name.

This item is shown with a green background as it is used in Heating and Cooling design calculations only.

The rest of the HVAC help section is split into 2 main sections covering the 2 HVAC model options:

- [Simple HVAC](#)
- [Detailed HVAC](#)

Also, independent of HVAC model options:

- [Natural Ventilation](#)
- [Earth Tubes](#)
- [Domestic Hot Water](#)
- [Air Temperature Distribution](#)

See also the [Simple HVAC Data](#) tutorial

Simple HVAC

[HVAC tab in model data - Simple HVAC](#)

Simple HVAC is suitable for use at early design stages or for other analyses where detailed treatment of heating, cooling and mechanical ventilation systems is required.

In summary, with Simple HVAC:

- Energy use is calculated based on space loads and system CoP value.
- Operational control provided by setpoints and schedules.
- Simple checkbox adds or removes the systems.
- Templates can be used to set performance characteristics of different systems.
- Auxiliaries and DHW are modelled based on schedules.
- Mech-vent, HR, nat-vent, mixed mode ventilation etc can also be modelled.

You can open headers to enter HVAC details where they are different from those loaded from the template:

- [Auxiliary Energy](#)
- [Mechanical Ventilation](#)
- [Fans](#)
- [Outside Air](#)
- [Heating](#)
- [Cooling](#)

See also the [Simple HVAC Data](#) tutorial

Auxiliary Energy

[HVAC tab in model data - Simple HVAC](#)

When using Simple HVAC, this data is used to calculate the electric energy consumption due to fans and pumps, control gear and any other HVAC-related equipment not accounted for elsewhere. The data used for this calculation depends on the [Simple HVAC Auxiliary](#) model options setting.

NCM Setting

When the [1-NCM Simple HVAC auxiliary energy calculations](#) option is set, only one data item is required to define the auxiliary energy consumption. This data is used at the zone level and the auxiliary energy consumption for the building is displayed at building level in the Simulation results as **Auxiliary Energy**.

Auxiliary energy

The total annual Auxiliary energy consumption can be entered as kWh/m² (SI units) or Btu/ft² (IP units). Auxiliary energy accounts for any additional energy consumption due to fans, pumps, control gear etc. It is specified per floor area and is modelled as being consumed 24/7 during the simulation. Note that when using the NCM option, auxiliary energy consumption will not vary according to mechanical ventilation flow rates or any other settings - it is a very basic way to account for auxiliary energy consumption.

The default auxiliary energy method is based on that published in the UK National Calculation Methodology (NCM).

The energy consumption related to Auxiliary energy is shown at building level on the Simulation results screen.

Separate Fans And Pumps Setting

When the [2-Separate fans and pumps Simple HVAC auxiliary energy calculations](#) option is set, the auxiliary energy calculations are carried out separately for fans and pumps and their respective energy consumption values are displayed separately in the simulation results.

- **Fan energy** is calculated based on fan pressure rise and efficiency values as defined under [Mechanical ventilation](#) > [Fans](#). The fan energy consumption data is displayed in simulation results as **Fan energy** and will depend on the actual mechanical ventilation flow rates applied in the simulation.
- **Pump**, control and any other electrical energy associated with the HVAC system is calculated using a power density value (W/m² or W/ft²) plus a schedule to define time/seasonal variations.

Pump etc energy

The pump energy can be defined as a power density (W/m² or W/ft²) and this is multiplied by the value of the associated schedule (below) to calculate the pump energy for each zone at any time in the simulation. The data is used at the zone level and the total energy consumption for the building is displayed at building level in the Simulation results as **Pump energy**.

When the [Mechanical ventilation method](#) model option is set to **1-Room ventilation**, fans are modelled in EnergyPlus based on the [Room ventilation](#) pressure and efficiency settings and the energy consumption of the fans and pumps are reported separately in results. When the [Mechanical ventilation method](#) option is set to **2-Ideal loads** EnergyPlus does not model fan energy consumption separately and in this case the total consumption of fans and pumps should be entered under **Pumps etc** and the results are reported under "Pumps".

Schedule

Select the schedule defining the time variation of the pump energy consumption.

Mechanical Ventilation

[HVAC tab in model data - Simple HVAC](#)

Check the mechanical ventilation checkbox to indicate that outside air and/or re-circulated air is delivered to the zone. You can use this setting to define mechanical ventilation and air delivered through centrally ducted air conditioning systems or local fresh air systems.

The way that mechanical ventilation is modelled in Simple HVAC depends on the [Mechanical ventilation method](#) model option:

- [1-Room ventilation](#) where mechanical ventilation is modelled using the EnergyPlus *ZoneVentilation:DesignFlowRate* data separate from the main HVAC system. This allows fan energy and heat pick up to be modelled and also more sophisticated zone temperature controlled mechanical ventilation. Heat recovery, economiser and humidity controls are not available with this option though.
- [2-Ideal loads](#) where mechanical ventilation is modelled using the EnergyPlus *ZoneHVAC:IdealLoadsAirSystem* data, allowing the effects of heat recovery, economiser and humidity control to be modelled but not fan energy and heat pickup.

Note: The specification of mechanical ventilation outside air delivery rates is the same for both options as described in the [Outside air](#) section.

1-Room Ventilation

With the [Mechanical ventilation method](#) model option the introduction of outside air through fans is achieved using the EnergyPlus *ZoneVentilation:DesignFlowRate* data separate from the main HVAC system. Energy consumption and heat pick up from fans is included as described below.

Fan Type

When using Simple HVAC, enter the type of fan. Select from:

- **1-Supply** - fan blows outside air into the zone and the zone receives heat from the fan if **Fan in air** (below) > 0
- **2-Extract** - fan exhausts air from the zone. In this case there is no fan heat pickup and it is not possible to enter Fan pressure rise and Fan total efficiency. It is assumed that makeup air comes from outside.

Note: In Simple HVAC the [Auxiliary energy](#) data accounts for all electric fan and pump distribution EnergyPlus controls and any other electrical energy use associated with HVAC that is not already accounted for elsewhere.

Fan pressure rise

When the Fan type is **1-Supply**, enter the pressure rise at full flow and standard conditions. Standard conditions are considered 20°C at sea level, 101325 Pa.

See also [Calculating Fan Pressure Rise](#).

Fan total efficiency (%)

When the Fan type is **1-Supply**, enter the product of the fan motor and impeller efficiency of the supply fan. This is the ratio of the power delivered to the air to the electrical input power at maximum flow expressed as a percentage. The motor efficiency is the power delivered to the shaft divided by the electrical power input to the motor. The fan efficiency is power delivered to the air divided by the shaft power. The power delivered to the fluid is the mass flow rate of the air multiplied by the pressure rise divided by the air density. Must be greater than 0 and less than or equal to 100.

Fan power and temperature pick up calculations

The calculations for fan power and airflow temperature pick up are detailed in the [EnergyPlus Engineering Document](#). A summary is shown below:

Total Fan Power = Mass flow rate. DeltaP / (Total fan efficiency . Air density)

Shaft Fan Power = Motor efficiency . Total Fan Power

Heat to air = Shaft Fan Power + (Total Fan Power - Shaft Fan Power) . Motor in air fraction

2-Ideal Loads

With the **Ideal loads** option, Mechanical ventilation is modelled with heating and cooling using the EnergyPlus *ZoneHVAC:IdealLoadsAirSystem* data. In this case there are options to include the effects of heat recovery, economiser, humidification and dehumidification.

Note: With the **Ideal loads** option, fan energy and heat pickup are not included in the simulation. In this case you can approximate fan energy using [auxiliary energy](#) data but if you need more detailed fan modelling you should use either the [Room ventilation](#) option or [Detailed HVAC](#) model options.

Economiser

Economisers are used to provide cooling when the outdoor temperature is lower than the indoor temperature. An economiser is a damper opening that draws up to 100% outside air when the outside air is cooler than the temperature inside the building, thereby providing **free cooling**. An outdoor air economy cycle can reduce cooling energy requirements by some 20% to 30%, or around 5% of the air conditioning energy use and are often required by energy codes for larger air conditioning units.

Choose from 3 options:

- **1-None** where no economiser operation will be simulated and outside fresh air is based purely on the Mechanical ventilation Outside air rate and associated operation schedule.
- **2-Differential dry bulb** the economiser increases the outdoor air flow rate above the minimum outdoor air flow when there is a cooling load and the outdoor air temperature or enthalpy is below the zone exhaust air temperature.
- **3-Differential enthalpy** the economiser increases the outdoor air flow rate above the minimum outdoor air flow when there is a cooling load and the outdoor air enthalpy is below that of the zone exhaust air.

Note: The economiser works together with the cooling system and if cooling is not selected the economiser will not operate.

Max outdoor air rate when economiser operates

The upper limit of outside airflow when the economiser is working in ac/h. The default is 15 ac/h.

Note: This value is not used a) when the [Cooling limit type](#) is set to **2-Limit capacity** and b) when the [HVAC Autosize](#) model option is **1-Adequate**.

Heat recovery

When heat recovery is active you can choose the type of heat recovery. Select from:

- **1-Sensible** which provides sensible heat recovery whenever the zone exhaust air temperature is more favourable than the outdoor air temperature.
- **2-Enthalpy** where latent and sensible heat recovery are provided whenever the zone exhaust air enthalpy is more favourable than the outdoor air enthalpy.

With the EnergyPlus Ideal loads system used in Simple HVAC heat recovery heating is only available when a zone is heated, i.e. when the **Heated** checkbox is checked on the HVAC tab. Likewise cooling heat recovery will only be available if the **Cooled** checkbox is checked. That is not to say that heat recovery isn't available for zones with heating at times when the heating system is not operating - heat recovery heating is available at any time in this case when the outside air is cooler than zone exhaust air, regardless of whether heating is operating or not. If you need to model heat recovery for systems without heating then you should use Detailed HVAC which is more flexible.

Sensible heat recovery effectiveness

The sensible heat recovery effectiveness, where effectiveness is defined as the change in supply temperature divided by the difference in entering supply and relief air temperatures. The default is 0.70.

Latent heat recovery effectiveness

The latent heat recovery effectiveness, where effectiveness is defined as the change in supply humidity ratio divided by the difference in entering supply and relief air humidity ratios. The default is 0.65.

Outside Air

[HVAC tab](#) in model data under [Mechanical Ventilation](#) header - Simple HVAC

You can set the amount, timing and control of outside air delivery through the HVAC air distribution system on the HVAC tab.

Outside air requirement data is used at the **zone** level for all HVAC configurations and options, though defaults settings can be made at building and block levels.

The [Mech Vent + Nat Vent + Infiltration](#) output data can be displayed on the [Heating design](#), [Cooling design](#) and [Simulation](#) screens when you select **1-All** or **3-Comfort** data. This is the sum of fresh air delivered through the HVAC system, infiltration and natural ventilation outside airflow in air changes per hour (ac/h) and when used together with the other environmental output it can be very useful in checking for occupant discomfort.

Outside air definition method

Defines the method used to set the *maximum outside air* delivery rate. The actual outside air flow rate during simulation is modified by the [Operation schedule](#) described below. The options are:

- **1-By zone** - enter the zone **outside air** delivery rate in air changes per hour in the control directly below. The air flow rate in m3/s is calculated from the ac/h data using:

$$\text{m3/s} = \text{ac/h} \times \text{ZoneVolume} / 3600$$

where:

`ZoneVolume` is the actual air volume of the space calculated according to Model options for [excluding floor/ceiling constructions](#).

Note: this zone volume may be slightly different from the value used for the purposes of calculating air flow rates for [infiltration](#) which uses the *total* zone volume.

- **2-Min fresh air (Per person)** - maximum mechanical ventilation rate is defined using [Minimum fresh air requirements](#) per person as set on the [Activity](#) tab. The air flow rate in m3/s is calculated from this as:

$$\text{m3/s} = \text{MinFreshAir}_{\text{per person}} \times \text{NumberPeople} / 1000$$

where:

`NumberPeople` = Occupancy density (people / m2) * Zone floor area (m2).
`MinFreshAirper person` is the minimum fresh air rate per person (l/s/person or CFM).

- **3-Min fresh air (Per area)** - maximum mechanical ventilation rate is defined using [Minimum fresh air requirements](#) as set on the [Activity](#) tab. The air flow rate in m3/s is calculated as:

$$\text{m3/s} = \text{MinFreshAir}_{\text{per area}} \times \text{ZoneFloorArea} / 1000$$

where:

ZoneFloorArea is the actual floor area of the space.

MinFreshAir_{per area} is the minimum fresh air rate per floor area (l/s-m2 or CFM/ft2).

- **4-Min fresh air (Sum per person + per area)** - maximum mechanical ventilation rate is defined using [Minimum fresh air requirements](#) for both as set on the [Activity](#) tab. The air flow rate in m3/s is calculated as:

$$\text{m3/s} = (\text{MinFreshAir}_{\text{per area}} \times \text{ZoneFloorArea} + \text{MinFreshAir}_{\text{per person}} \times \text{NumberPeople}) / 1000$$

where:

ZoneFloorArea is the actual floor area of the space.

MinFreshAir_{per area} is the minimum fresh air rate per floor area (l/s-m2 or CFM/ft2).

NumberPeople = Occupancy density (people / m2) * Zone floor area (m2).

MinFreshAir_{per person} is the minimum fresh air rate per person (l/s/person or CFM).

- **5-Min fresh air (Max per person and per area)** - maximum mechanical ventilation rate is defined using [Minimum fresh air requirements](#) for both as set on the [Activity](#) tab. The air flow rate in m3/s is calculated as:

$$\text{m3/s} = \text{Maximum of } (\text{MinFreshAir}_{\text{per area}} \times \text{ZoneFloorArea} / 1000) \text{ and } (\text{MinFreshAir}_{\text{per person}} \times \text{NumberPeople} / 1000)$$

where:

ZoneFloorArea is the actual floor area of the space.

MinFreshAir_{per area} is the minimum fresh air rate per floor area (l/s-m2 or CFM/ft2).

NumberPeople = Occupancy density (people / m2) * Zone floor area (m2).

MinFreshAir_{per person} is the minimum fresh air rate per person (l/s/person or CFM).

- **6-Design flow rate** - where the maximum mechanical ventilation rate is defined directly in m3/s or ft3/min.
- **7-Indoor air quality procedure** - choose this option when using Detailed HVAC to simulate a DCV system where CO2 is controlled in the zone. See "DCV Modelling" on page 2143 for a worked example.

Outside air flow rate

Enter the minimum zone **outside air** delivery rate (in air changes per hour) when **Outside air definition method** (above) is **1-By zone**.

The actual minimum amount of outside air entering the AHU can be modified by the **Operation schedule** (below).

Note: For [Heating design calculations](#), the **Outside air flow rate** value is used [unmodified](#) by a schedule when mechanical ventilation is switched on and [included](#) in the analysis.

Operation

Schedule

Select the schedule that defines the time-varying operation of the mechanical ventilation system. The fractional values in the schedule are multiplied by the **Outside air flow rate** (above).

Heating

[HVAC tab in model data - Simple HVAC](#)

The data required to define the heating system will depend on whether the [HVAC model option](#) is set to [Simple](#) or [Detailed](#) and if using Detailed the type of Detailed HVAC system selected.

Note: for all systems, [heating setpoint temperatures](#) are defined on the [Activity tab](#).

Heated

If the zone is to be heated then check this checkbox. When it is checked the rest of the items described on this page become available.

Heating capacity

The Heating capacity can be entered manually when using Simple HVAC and when the [HVAC sizing](#) model option is set to either **2-Manual** or **4-Autosize when not set**.

This setting is not available when either of the **1-Adequate** or **3- Autosize** HVAC sizing model options are selected.

When **4-Autosize when not set** is selected, if immediately prior to a [Simulation](#) the Heating capacity is not set or it has a value of zero then a DesignBuilder Heating design autosizing simulation is automatically run prior to the simulation to calculate it.

Tip: You can change heating capacities by hand if you wish, but you should bear in mind that, when **4-Autosize when not set** is selected, if the Model options or model geometry are changed, all heating and cooling capacities are reset to zero. You can set the [HVAC sizing](#) model option to **2-Manual** to avoid losing your capacity settings due to model edits.

For more information about how this works, see [HVAC sizing](#).

Heating Plant

Heating fuel

Select the type of fuel used to generate the heating energy - choose from:

Interface Fuel Name

- 1-Electricity from grid
- 2-Natural Gas
- 3-Oil
- 4-Coal
- 5-LPG
- 6-Biogas
- 7-Anthracite
- 8-Smokeless Fuel (inc Coke)
- 9-Dual Fuel Appliances (Mineral + Wood)
- 10-Biomass
- 11-Waste Heat

Fuel in results

- Electricity
- Gas
- Oil
- Solid fuel
- Bottled gas
- Bottled gas
- Solid fuel
- Solid fuel
- Other
- Other
- Other

You should select **1-Electricity from grid** you wish the results displayed in DesignBuilder to agree with those calculated by EnergyPlus directly for District heating and displayed on the Summary tab of the Simulation screen.

Heating system CoP

The heating system Coefficient of Performance is used to calculate the fuel consumption required to meet heating demand. The value represents the total seasonal efficiency of the entire heating system and should include the effect of all energy consumption associated with building heating such as any boiler fan energy, boiler inefficiency, control equipment etc. It does not include energy involved in air or water distribution (fans or pumps) or control gear, which is accounted for in the [Auxiliary energy](#).

Heating system CoP data is specified by zone so you can model different heating efficiencies in each zone. But generally you should access this data from the building level and allow zones to take on building default.

Note: When comparing the heating fuel consumption results with the data displayed on the [Summary](#) tab, it is important to realise that DesignBuilder applies a fuel type and a CoP to the idealised heating systems used in EnergyPlus when using Simple HVAC. On the other hand EnergyPlus displays the unprocessed loads as **District heating** in the tables on the Summary tab. This is because the idealised systems in EnergyPlus are considered to be met directly without use of HVAC equipment through a district heating type system.

Local Heating Units

Heating type

There are two heating types available:

- **1-Convective** - the space is heated by an air system and controlled to the air temperature set point. The system is modelled using the EnergyPlus "Ideal Loads" system.
- **2-Radiative/convective units** - used for modelling general systems where a radiant heat is a factor. Allows modelling of convective systems, hot water radiator systems, underfloor heating systems, baseboard heating etc. using EnergyPlus 'High Temp Radiant System'.

1-Convective

When using [Simple HVAC](#) you can specify the heating supply air conditions of the air.

Maximum supply air delivery temperature

The maximum dry-bulb air temperature of the air supplied for heating the zone (in °C or °F).

Maximum supply air humidity ratio

The constant humidity ratio (mass of water per mass of dry air) of the warm supply air to be delivered when heating is required. The default humidity ratio is 0.0156. You should be careful to enter a value that is a valid condition, i.e. it is under the 100% saturation line on the [Psychrometric Chart](#).

Heating limit type

There are 3 ways that EnergyPlus ideal loads systems can limit zone heating capacity:

- **1-Limit flow rate** - means that the heating supply air flow rate will be limited to a flow rate value calculated based on the autosizing model option settings. When using the **3-Autosize HVAC sizing method** the flow rate limit is calculated by EnergyPlus sizing calculations. Otherwise if the DesignBuilder sizing method is used the flow rate is calculated based on the required heating capacity calculated by DesignBuilder and the **Maximum supply air delivery temperature** (above).
- **2-Limit capacity** - means that the sensible heating capacity will be limited to a capacity value calculated based on the autosizing model option settings. When using the **3-Autosize HVAC sizing method** the capacity is calculated by EnergyPlus sizing calculations. Otherwise if the DesignBuilder sizing method is used the flow rate is calculated based on the required heating

capacity calculated by DesignBuilder and the **Maximum supply air delivery temperature** (above).

- **3-Limit flow rate and capacity** - means that both flow rate and capacity will be limited to the above values.

Technical description

EnergyPlus uses the following 4 settings to define the maximum heating input for ideal loads systems:

- Maximum Heating Supply Air Temperature
- Maximum Heating Air Flow Rate,
- Maximum Sensible Heating Capacity
- Heating Limit Type

When using the default **3-Autosize HVAC sizing** Model option the "Maximum Heating Supply Air Temperature" and "Heating Limit Type" IDF fields are set using corresponding data from the HVAC tab, and the "Maximum Heating Air Flow Rate" and "Maximum Sensible Heating Capacity" fields are calculated by EnergyPlus before the simulation in the design sizing calculations.

Alternatively, when using either of the **2-Manual** or **4-Autosize when not set HVAC sizing** Model options, the "Maximum Heating Supply Air Temperature", "Maximum Sensible Heating Capacity" and "Heating Limit Type" IDF fields are set using corresponding data from the HVAC tab, and DesignBuilder calculates the corresponding "Maximum Heating Air Flow Rate" based on the user-defined [Heating capacity](#) and [Maximum supply air delivery temperature](#) data. In this case all of the 4 related IDF fields are specified in the IDF by DesignBuilder.

Or when the **1-Adequate HVAC sizing** Model option is selected, DesignBuilder sets the EnergyPlus "Heating Limit Type" IDF field to "NoLimit" which means that the heating system can meet any load. In this case the other 3 fields are not needed and so are not provided in the IDF.

In all of the above cases, apart from when the **1-Adequate HVAC sizing** Model option is selected, EnergyPlus holds values for each of the 4 settings used to define the maximum heating input for ideal loads systems, and the way these are used in the simulation is defined by the **Heating limit type** per the above descriptions.

Note: In practice this setting should not make a noticeable difference to the thermal heat balance of the zone during heating since the maximum heat delivery to the zone is already defined before the simulation and this setting simply tells EnergyPlus exactly how to apply the limit.

2-Radiant/Convective Units

When the [Simple HVAC](#), **1-Room ventilation** [Mechanical ventilation](#) and **1-DesignBuilder Simple HVAC autosizing method** model options are set you can model radiant heating systems. Set the heating system type as **2-Radiant/convective units** (above) and enter the heating radiant fraction and the way radiant heat is distributed through the space.

Important: Recent versions of EnergyPlus no longer support the radiant/convective option combined with Simple HVAC cooling systems, so if you need radiant heating and cooling in the same zone then you should switch to Detailed HVAC.

Heating radiant fraction

The fraction of the power input to the radiant heater that is actually radiant heat transfer. The fraction should be between 0 and 1. In conjunction with the Radiant distribution option, it defines the breakdown of how the power input to the heater is distributed to the rest of the zone.

Radiant distribution

The Radiant distribution allows you to control how the radiant heat from the Radiant unit is distributed around the room. Options are:

- **1-Uniform** - radiant heat is distributed uniformly around the zone.
- **2-Floor** - heat is distributed uniformly across all floors in the zone. Use this for modelling floor heating systems.

Technical Note

DesignBuilder uses the EnergyPlus 'HIGH TEMP RADIANT SYSTEM' to model radiant heating systems when in Simple HVAC mode. This system type is intended for modelling 'high-temperature' radiant sources but it can equally well be used for low temperature radiant sources.

Note: for all Simple HVAC systems, [heating setpoint temperatures](#) are defined on the [Activity tab](#).

Sizing Zone Equipment

By default EnergyPlus does not account for mechanical and natural ventilation (outdoor air) loads in Heating design calculations so the options below are provided to add an extra load for Winter design day only.

Mechanical ventilation load

Define how mechanical ventilation loads should be dealt with in Heating design calculations. Options are:

- **1-Met by zone equipment** where mechanical ventilation loads are met entirely by zone equipment and so the capacity of the zone heating systems must account for mechanical ventilation loads.
- **2-Exclude** where the mechanical ventilation loads are not met by zone heating equipment so zone heating sizing calculations can safely ignore them.
- **3-Partially met by AHU** where the mechanical ventilation loads are partially met by heating coils in the AHU but there is still some residual load to be met by zone heating equipment. In this case the supply air temperature must also be entered. See below.

Note: The default **Mechanical ventilation load** setting is **1-Met by zone equipment**. For DOAS systems which introduce air that is partially or fully pre-heated, this setting should be changed to **2-Exclude** or to **3-Partially met by AHU** to ensure that zone equipment is not oversized.

Note: This setting applies only to Heating design calculations. For Detailed HVAC simulations the [DOAS settings](#) on the HVAC zone group dialog are used instead to account for any additional loads due to mechanically supplied fresh air. For Simple HVAC the full outdoor air load is assumed when sizing heating and cooling systems, no preheat/precool is assumed when sizing zone equipment.

System supply temperature

When Mechanical ventilation load is set to **3-Partially met by AHU** then you must enter the supply air temperature to enable the residual additional zone loads required to be accounted for.

Exclude heat recovery from zone heating sizing

If the savings provided by heat recovery can be safely included in the heating sizing calculations then leave this checkbox unchecked. Otherwise for a safer calculation where zone heating systems are sized based on no heat recovery being available then check this option.

Natural ventilation load

If natural ventilation is activated then you can define how natural ventilation loads should be dealt with in Heating design and simulation autosizing simulations. Options are:

- **1-Met by zone equipment** the default option where natural ventilation loads are met entirely by zone equipment and so the capacity of the zone heating systems must account for natural ventilation loads. Note that in Heating design calculations, when this option is selected, natural ventilation will be applied without regard for schedule or [mechanical ventilation setpoint](#) temperature settings.
- **2-Exclude** where the natural ventilation loads are not met by zone heating equipment so zone heating sizing calculations can safely ignore them.

Operation

Heating operation schedule

This schedule data is used in conjunction with the [heating setpoint temperatures](#) on the [Activity tab](#) to define the time-varying heating setpoint schedule in the zone. The operation schedule defines the times when full and setback setpoints should be met and the setpoint data on the Activity tab define the actual setpoint values. See [Defining setpoint temperature schedules](#) for more on this.

Note: the zone heating operation schedule is also used in Detailed HVAC with the default **1-Simple HVAC Detailed HVAC Activity data** setting.

Cooling

[HVAC tab](#) in model data - Simple HVAC

You can enter details on the cooling system including the maximum capacity (at zone level).

Note: for all systems, [cooling setpoint temperatures](#) are defined on the [Activity tab](#).

Cooled

If the zone is to be actively cooled then check this checkbox. When it is checked the rest of the items described on this page become available.

Cooling capacity

The Cooling capacity can be entered manually when using Simple HVAC and when the [HVAC sizing](#) model option is set to either **2-Manual** or **4-Autosize when not set**.

This setting is not available when either of the **1-Adequate** or **3- Autosize** HVAC sizing model options are selected.

When **4-Autosize when not set** is selected, if immediately prior to a [Simulation](#) the Cooling capacity is not set or it has a value of zero then a DesignBuilder Cooling design autosizing simulation is automatically run prior to the simulation to calculate it.

Tip: You can change cooling capacities by hand if you wish, but you should bear in mind that, when **4-Autosize when not set** is selected, if the Model options or model geometry are changed, all heating and cooling capacities are reset to zero. You can set the [HVAC sizing](#) model option to **2-Manual** to avoid losing your capacity settings due to model edits.

For more information about how this works, see [HVAC sizing](#).

Supply Air Conditions

When using [Simple HVAC](#) you can specify the cooling supply air conditions of the air.

Minimum supply air delivery temperature

The minimum dry-bulb air temperature of the air supplied for cooling the zone (in °C or °F).

Minimum supply air humidity ratio

The minimum humidity ratio (kg of water per kg of dry air) of the cool supply air. The default is 0.0077 kg-H₂O/kg-air which corresponds to a 10C (50F) dew point. You should be careful to enter a value that is a valid condition, i.e. it is under the 100% saturation line on the [Psychrometric Chart](#).

Cooling limit type

There are 3 ways that EnergyPlus ideal loads systems can limit zone cooling capacity (for simulations only):

- **1-Limit flow rate** - means that the cooling supply air flow rate will be limited to a flow rate value calculated based on the autosizing model option settings. When using the **3-Autosize HVAC sizing method**, the flow rate limit is calculated by EnergyPlus sizing calculations. Otherwise if the DesignBuilder sizing method is used, the flow rate is calculated based on the required cooling capacity calculated by DesignBuilder and the **Minimum supply air delivery temperature** (above).
- **2-Limit capacity** - means that the sensible cooling capacity will be limited to a capacity value calculated based on the autosizing model option settings. When using the **3-Autosize HVAC sizing method** the capacity is calculated by EnergyPlus sizing calculations. Otherwise if the DesignBuilder sizing method is used the flow rate is calculated based on the required cooling capacity calculated by DesignBuilder and the **Minimum supply air delivery temperature** (above).
- **3-Limit flow rate and capacity** - means that both flow rate and capacity will be limited to the above values.

Technical description

EnergyPlus uses the following 4 settings to define the maximum cooling input for ideal loads systems:

- Minimum Cooling Supply Air Temperature
- Maximum Cooling Air Flow Rate,
- Maximum Total Cooling Capacity
- Cooling Limit Type

When using the default **3-Autosize HVAC sizing** Model option the "Minimum Cooling Supply Air Temperature" and "Cooling Limit Type" IDF fields are set using corresponding data from the HVAC tab, and the "Maximum Cooling Air Flow Rate" and "Maximum Total Cooling Capacity" fields are calculated by EnergyPlus before the simulation in the design sizing calculations.

Alternatively, when using either of the **2-Manual** or **4-Autosize when not set HVAC sizing** Model options, the "Maximum Cooling Supply Air Temperature", "Maximum Sensible Cooling Capacity" and "Cooling Limit Type" IDF fields are set using corresponding data from the HVAC tab, and DesignBuilder calculates the corresponding "Maximum Cooling Air Flow Rate" based on the user-

defined [Cooling capacity](#) and [Minimum supply air delivery temperature](#) data. In this case all of the 4 related IDF fields are specified in the IDF by DesignBuilder.

Or when the **1-Adequate HVAC sizing** Model option is selected, DesignBuilder sets the EnergyPlus "Cooling Limit Type" IDF field to "NoLimit" which means that the cooling system can meet any load. In this case the other 3 fields are not needed and so are not provided in the IDF.

In all of the above cases, apart from when the **1-Adequate HVAC sizing** Model option is selected, EnergyPlus holds values for each of the 4 settings used to define the maximum cooling input for ideal loads systems, and the way these are used in the simulation is defined by the **Cooling limit type** per the above descriptions.

Note: In practice this setting should not make a noticeable difference to the thermal heat balance of the zone during cooling since the maximum cooling delivery to the zone is already defined before the simulation and this setting simply tells EnergyPlus exactly how to apply the limit.

Cooling Plant

Cooling fuel

Select the type of fuel used to generate the cooling energy - choose from:

Interface Fuel Name

1-Electricity from grid
2-Natural Gas
3-Oil
4-Coal
5-LPG
6-Biogas
7-Anthracite
8-Smokeless Fuel (inc Coke)
9-Dual Fuel Appliances (Mineral + Wood)
10-Biomass
11-Waste Heat

Fuel in results

Electricity
Gas
Oil
Solid fuel
Bottled gas
Bottled gas
Solid fuel
Solid fuel
Other
Other
Other

You should select **1-Electricity from grid** you wish the results displayed in DesignBuilder to agree with those calculated by EnergyPlus directly for District cooling and displayed on the Summary tab of the Simulation screen.

Cooling system CoP

The cooling system coefficient of performance is used to calculate the fuel consumption required to meet cooling demand. The value represents the total seasonal efficiency of the entire cooling system and should include the effect of all energy consumption associated with building cooling such as fan and pump energy, chiller inefficiency, control equipment etc.

Cooling system CoP data is specified by zone so you can model different cooling efficiencies in each zone. But generally you should access this data from the building level and allow zones to take on building default.

Note: When comparing the cooling fuel consumption results with the data displayed on the [Summary](#) tab, it is important to realise that DesignBuilder applies a fuel type and a CoP to the idealised cooling systems used in EnergyPlus when using Simple HVAC. On the other hand EnergyPlus displays the unprocessed loads as **District cooling** in the tables on the Summary tab. This is because the idealised systems in EnergyPlus are considered to be met directly without use of HVAC equipment through a district cooling type system.

Operation

Cooling operation schedule

This schedule data is used in conjunction with the [cooling setpoint temperatures](#) on the [Activity tab](#) to define the time-varying cooling setpoint schedule in the zone. The operation schedule defines the times when full and setback setpoints should be met and the setpoint data on the Activity tab define the actual setpoint values. See [Defining setpoint temperature schedules](#) for more on this.

Note: the zone cooling operation schedule is also used in Detailed HVAC with the default **1-Simple HVAC Detailed HVAC Activity data** setting.

Humidity Control

[HVAC tab](#) in model data - Simple HVAC

When using the Ideal loads [Mechanical ventilation method](#) you have the option to control humidity by checking the Humidification and/or Dehumidification checkboxes on the HVAC tab.

Note: In real HVAC systems humidity control can require considerable energy consumption to achieve. If you require an accurate assessment of this energy then you should consider using [Detailed HVAC humidity control](#) which allows you to track energy flows in more detail.

Humidification

Humidification control type

When humidification is selected there are 2 further options:

- **1-Constant supply humidity ratio** means that during heating the supply air will always be at the Maximum heating supply humidity ratio.
- **2-Humidistat** means that the humidity in the zone is controlled using the [Humidification setpoint](#) defined on the Activity tab. The ideal loads system will attempt to meet the humidistat request, i.e. it will humidify according to the setpoint.

Dehumidification

Dehumidification control type

When dehumidification is selected there are 3 further options:

- **1-Constant supply humidity ratio** means that during cooling the supply air will always be at the Minimum cooling supply humidity ratio.
- **2-Humidistat** means that the humidity in the zone is controlled using the [Dehumidification setpoint](#) defined on the Activity tab. The ideal loads system will attempt to meet the humidistat request, i.e. it will dehumidify according to the setpoint.
- **3-Constant sensible heat ratio** means that the ideal loads system will be controlled to meet the sensible cooling load, and the latent cooling rate will be computed using the constant sensible heat ratio (SHR) defined below.

For **3-Constant sensible heat ratio** and **2-Humidistat** options, if the mixed air humidity ratio is less than the target humidity ratio, then the mixed air humidity ratio will be used. For all options, the supply air humidity ratio will never be allowed to exceed saturation at the supply dry bulb temperature. The selected dehumidification

control type is always applied when the unit is in cooling mode. If the unit is in deadband mode (not actively heating the supply air) control type Humidistat will be active. If the unit is in heating mode, control type Humidistat will be active if the Humidification Control Type field below is set to Humidistat or None. This allows the ideal loads system to heat and dehumidify at the same time.

Cooling sensible heat ratio

When the **Dehumidification control type** above is set to **3-Constant sensible heat ratio** the ideal loads system will be controlled to meet the sensible cooling load, and the latent cooling rate will be computed using the value of Cooling Sensible Heat Ratio (SHR), where $SHR = \text{Sensible Cooling} / \text{Total Cooling (sensible plus latent)}$. The default is 0.7.

Humidity Control Linkage With Heating And Cooling Schedules

In Simple HVAC, the timing of humidification control is the same as that for heating. So at times when heating is available (in setpoint or setback mode) humidification control is also available. Likewise the timing of dehumidification control is the same as that for cooling. So at times when cooling is available (in setpoint or setback mode) dehumidification control is also available.

Detailed HVAC

HVAC tab in model data - Detailed HVAC

When the [Detailed HVAC model option](#) is selected and the [Detailed HVAC Activity](#) data model option is set to **1-Simple HVAC Data**, the HVAC tab is used to define:

- Operation/setpoint schedules (under the [Heating](#) and [Cooling](#) headers),

Note: With Detailed HVAC simulations the **Heating on** and **Cooling on** settings have no effect on simulations, other than to show/hide the schedule data. They are always used in Heating and Cooling design calculations though.

- [Outdoor air ventilation requirement](#) (under the Mechanical Ventilation header)
- "Environmental Impact Factors" on page 520
- [Natural ventilation](#) and
- [Air temperature distribution](#) settings.

Note: For heating and cooling design calculations the humidity and temperature setpoints and ventilation requirement data comes from the Activity and HVAC tabs as described under the **1-Simple HVAC** column in the [table under Detailed HVAC Activity data](#), regardless of the **Detailed HVAC Activity Data** setting.

The main HVAC data is entered by clicking on the <HVAC System> navigator node.

Further details of Detailed HVAC are provided in a [separate section](#).

Natural Ventilation

HVAC tab in model data

You can control the operation of natural ventilation using the data under the Natural Ventilation header on the HVAC tab. When using [Calculated](#) natural ventilation model option there is additional data on the [Openings](#) tab to control the timing and extent of operation of openings.

Note: the Natural ventilation checkbox must be ticked here on the HVAC tab to allow window, vent and door [operation](#) to be defined on the Openings tab.

The data shown under Natural ventilation on the HVAC tab depends on whether you are using the Scheduled or Calculated [Natural ventilation](#) setting in Model options.

- [Scheduled Natural Ventilation Data](#)
- [Calculated Natural Ventilation Data](#)

Both Scheduled and Calculated Natural ventilation options are available with both [Simple](#) and [Detailed](#) HVAC options.

See also:

- "Mixed Mode Zone Equipment" on page 492
- "Mixed mode Cooling" on page 496
- "Mixed Mode Tutorial" on page 499

Scheduled Natural Ventilation Data

[HVAC tab](#) in model data

See the [Scheduled natural ventilation and infiltration](#) tutorial

Summary of Scheduled Natural Ventilation data definition

When the [Natural ventilation model option](#) is set to **Scheduled**, and for heating and cooling design calculations, natural ventilation is defined as follows:

- Check the **On** checkbox under the Natural Ventilation header.
- Select the Outside air definition method
- Set the maximum zone natural ventilation rate in air changes per hour (if flow type is **1-By zone**), or in m³/s or ft³/min (if the flow type is **3-Design flow rate**).
- Select the operation schedule.
- Enter the relevant control parameters on the HVAC tab and on the Activity tab.

Find out more about [Natural Ventilation Modelling](#) under [Modelling Advice](#).

Natural Ventilation

Natural ventilation On

Check the **On** checkbox under the Natural Ventilation header to activate natural ventilation. This causes all of the natural ventilation data on the HVAC tab to be accessible.

Outside air definition method

The Outside air definition method allows you to select the method to be used to set the maximum outside air natural ventilation rate. The options are:

- **1-By zone** - enter the zone natural ventilation rate in air changes per hour in the control directly below. The design air flow rate is calculated from the ac/h data using:

$$V_{\text{design}} = ac/h \times \text{ZoneVolume} / 3600$$

where `ZoneVolume` is the actual air volume of the space calculated according to options for optionally [excluding floor/ceiling constructions](#).

Note: this zone volume may be different from the value used for the purposes of calculating air flow rates for [infiltration](#) which uses the *total* zone volume.

- **2-Min fresh air per person** - maximum natural ventilation rate is defined using [Minimum fresh air requirements](#) as set on the [Activity](#) tab. The design air flow rate in m3/s is calculated as:

$$V_{\text{design}} = \text{MinFreshAir} \times \text{NumberPeople} / 1000$$

`NumberPeople` = Occupancy density (people / m2) * Zone floor area (m2)

`MinFreshAir` is the minimum fresh air rate per person (l/s/person)

- **3-Design flow rate** - where the maximum natural ventilation rate is entered directly in m3/s or ft3/min.

The actual ventilation rate in m3/s is calculated at each time step of the simulation using the equation below:

$$\text{Ventilation} = V_{\text{design}} F_{\text{schedule}} [A + B (T_{\text{in}} - T_{\text{out}}) + C \cdot \text{Wind Speed} + D (\text{Wind Speed}^2)]$$

Each of the parameters A-D in the equation above are explained below under [Delta T and Wind Speed Coefficients](#).

F_{schedule} is described below under Operation > Schedule.

Operation

Schedule

Select the schedule used to modify the maximum design natural volume flow rate (V_{design}) (see Design Flow Rate Calculation Method field and related subsequent fields). This fraction between 0.0 and 1.0 is included as F_{schedule} in the above equation.

Outdoor Temperature Limits

Outdoor min temperature control

Check this checkbox if you wish to define a restriction on natural ventilation operation when outdoor temperature drops below a setpoint temperature. Use this option to help avoid over-cooling a space, which could result in a heating load.

Min temperature definition

There are 2 ways to define the outdoor min temperature:

- **1-By value** - use this option when the setpoint does not change with time in which case you can enter a fixed setpoint temperature.
- **2-By schedule** - use this option when the setpoint changes with time in which case you can select a schedule of time-varying temperature setpoints.

Outdoor min temperature

This is the fixed outdoor temperature (in °C or °F) below which ventilation is shut off. This control is visible when the **1-By value** option is selected for [Min temperature definition](#).

Outdoor min temperature schedule

This is the schedule of outdoor temperatures (in °C) below which ventilation is shut off. This control is visible when the **1-By schedule** option is selected for **Min temperature definition**.

See also [Defining setpoint temperature schedules](#).

Outdoor max temperature control

Check this checkbox if you wish to define a restriction on natural ventilation operation when outdoor temperature rises above a setpoint temperature. Use this option to help avoid over-heating a space, which could result in a cooling load.

Max temperature definition

There are 2 ways to define the outdoor max temperature:

- **1-By value** - use this option when the setpoint does not change with time in which case you can enter a fixed setpoint temperature.
- **2-By schedule** - use this option when the setpoint changes with time in which case you can select a schedule of time-varying temperature setpoints.

Outdoor max temperature

This is the fixed outdoor temperature (in °C or °F) above which ventilation is shut off. This control is visible when the **1-By value** option is selected for **Max temperature definition**.

Outdoor max temperature schedule

This is the schedule of outdoor temperatures (in °C) above which ventilation is shut off. This control is visible when the **1-By schedule** option is selected for **Max temperature definition**.

See also [Defining setpoint temperature schedules](#).

Delta T Limits

Delta T limit control

Check this option to limit natural ventilation operation based on the temperature difference between the indoor and outdoor air dry-bulb temperature. It allows ventilation to be stopped if the temperature outside is too warm and could potentially heat the space.

Uncheck this option to keep the ventilation on even if the outdoor temperature is greater than the indoor temperature. This is useful for uncontrolled natural ventilation (open windows) or as a way to estimate the effect of required ventilation air for load calculations.

Delta T definition

There are 2 ways to define the Delta T to be used in temperature difference control:

- **1-By value** - use this option when the Delta T does not change with time in which case you can enter a fixed value.
- **2-By schedule** - use this option when the Delta T changes with time in which case you can select a schedule of time-varying Delta T values.

Delta T

This is the temperature difference (in °C or °F) between the indoor and outdoor air dry-bulb temperatures below which ventilation is shut off. This field allows ventilation to be stopped if the temperature outside is too warm and could potentially heat the space. For example, if the user specifies a delta temperature of 2°C, ventilation is assumed to be available if the outside air temperature is at least 2°C cooler than the zone air temperature. If the outside air dry-bulb temperature is less than 2°C

cooler than the indoor dry-bulb temperature, then ventilation is automatically turned off. The values for this field can include negative numbers. This allows ventilation to occur even if the outdoor temperature is above the indoor temperature.

The Delta Temperature is used in the code in the following way:

IF ((IndoorTemp - OutdoorTemp) < DeltaTemperature) Then ventilation is not allowed.

For example:

Value of Natural ventilation delta T	Effect
0	Natural ventilation only operates when <i>OutdoorTemp < IndoorTemp</i>
-100	Natural ventilation unrestricted by Inside-Outside delta T
2	Natural ventilation only operates when <i>OutdoorTemp + 2 < IndoorTemp</i>

Delta T schedule

This is the schedule of temperature difference values (in °C) between the indoor and outdoor air dry-bulb temperatures below which ventilation is shut off. See above **Delta T** description for more details.

Delta T And Wind Speed Coefficients

The coefficients below can be used to set up inside to outside Delta T and wind speed dependency of the Scheduled natural ventilation rates using the equation at the end of the [Outside air definition method](#) section.

Technical

The question of typical values for these coefficients is subject to debate. Ideally, one should do a detailed analysis of the ventilation situation and then determine a custom set of coefficients using methods such as those laid out in Chapter 26 of the ASHRAE Handbook of Fundamentals. The defaults are 1,0,0,0 which give a constant volume flow of ventilation under all conditions.

The equation must always yield a non-negative results; negative values are set to 0.0. BLAST (one of the EnergyPlus predecessors) used the following values as defaults: 0.606, 0.03636, 0.1177, 0. These coefficients produce a value of 1.0 at 0C deltaT and 3.35 m/s (7.5mph) wind speed, which corresponds to a typical summer condition. At a winter condition of 40C deltaT and 6 m/s (13.4 mph) wind speed, these coefficients would increase the ventilation rate by a factor of 2.75.

In DOE-2 (the other EnergyPlus predecessor), the air change method defaults are (adjusted to SI units) 0, 0, 0.224 (wind speed), 0. With these coefficients, the summer conditions above would give a factor of 0.75, and the winter conditions would give 1.34. A wind speed of 4.47 m/s (10 mph) gives a factor of 1.0.

The source of the BLAST defaults is noted in the BLAST documentation as: "Empirical equation and the coefficient default were determined from ASHRAE journal articles and other data on the effects of outdoor weather conditions." The source of the DOE-2 defaults is based on examining the infiltration relationships described in the ASHRAE Handbook of Fundamentals.

The local outdoor dry-bulb temperature used in the above basic equation (T_{out}) is typically a function of the height of the zone centroid above ground. The local outdoor dry-bulb temperature calculation procedure is described in the "Local Outdoor Air Temperature Calculation" section of the Engineering Reference.

The local outdoor wind speed used in the above basic equation (Wind Speed) is also a function of the height of the zone centroid above ground. The local outdoor wind speed calculation procedure is described in the "Local Wind Speed Calculation" section of the Engineering Reference.

Constant

This constant term coefficient is the "A" parameter in the above ventilation equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter, however, is a constant under all conditions and is not modified by any environmental effect. As a result, it is dimensionless.

Temperature

The temperature term coefficient is the “B” parameter in the above ventilation equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter is modified by the temperature difference between the outdoor and indoor air dry-bulb temperatures. The units for this parameter are inverse Celsius.

Velocity

The velocity term coefficient is the “C” parameter in the above ventilation equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter is modified by the speed of wind being experienced outside the building. The units for this parameter are s/m.

Velocity squared

The velocity squared term coefficient is the “D” parameter in the above ventilation equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter is modified by square of the speed of wind being experienced outside the building. The units for this parameter are s²/m².

Summary Of Scheduled Natural Ventilation Operation Rules

Scheduled natural ventilation is active at any time in the simulation based on the rules in the table below.

Condition	Rule	Data Location
If Natural ventilation indoor min temperature control checked	Operation schedule has value > 0 AND $T_{in} > \text{Natural ventilation min temperature}$ AND	HVAC tab Activity tab
If Natural ventilation indoor max temperature control checked	$T_{in} < \text{Natural ventilation max temperature}$ AND	Activity tab
If Outdoor min temperature control checked	$T_{out} > \text{Outdoor min temperature}$ AND	HVAC tab
If Outdoor max temperature control checked	$T_{out} < \text{Outdoor min temperature}$ AND	HVAC tab
If Delta T control checked	$T_{in} - T_{out} > \text{Delta T}$	HVAC tab

Where:

T_{in} is the inside dry bulb air temperature
 T_{out} is the outside dry bulb air temperature

Note: The Outdoor temperature and Delta T control rules described on this page can be used in combination with any [indoor temperature min/max controls](#) that may also have been selected on the activity tab to set up multiple restrictions on natural ventilation operation.

Tip: To ensure that natural ventilation operates regardless of temperature inside and outside you should uncheck all of the checkbox options listed on the "Condition" column in the above table.

Internal Windows, Doors And Vents

Airflow through *external* windows, vents, doors and holes are not modelled with **Scheduled** natural ventilation because the natural ventilation airflow from outside is explicitly defined (above). Internal windows, vents, doors and holes cause air to be exchanged between the two adjacent zones using the EnergyPlus **ZoneMixing** object. The way this works can be defined on the Advanced tab of the Model options dialog under the [Natural Ventilation](#) > Scheduled headers.

In addition, you can override operation of *internal* windows, vents and doors to be **off** as follows:

Window, Vent, Door position	Override Off Action
Internal windows and vents	Use the Internal Window and Vent Operation Control option data, selecting 2-Closed .
Doors	Set the door Operation schedule to Off

Calculated Natural Ventilation Data

[HVAC tab](#) in model data

See the [Calculated natural ventilation](#) tutorial

When the [Natural ventilation model option](#) is set to **Calculated**, natural ventilation data is set up using the following procedure:

1. Check the **On** checkbox under the Natural Ventilation header.
2. On the [Openings tab](#) set window, door and vent opening sizes and operation times.

You can override the operation of windows, vents and door to be **off** as follows:

Window or Vent position	Override Off Action
External	Switch off the Natural ventilation On checkbox
Internal adjacent to unconditioned semi-exterior zone	Switch off the Natural ventilation On checkbox in the occupied zone
Other internal windows and vents	Use the Internal Window and Vent Operation Control option data, selecting 1-Control by schedule and set the schedule to Off .
	When using the 1-Constant Control option on the Openings tab, airflow through internal windows/vents will occur (when enabled by the operation schedule) regardless of whether natural ventilation is activated here on the HVAC tab.
Doors	Set the door Operation schedule to Off

Note that even when operation is switched off as described above, windows, vents and doors still provide a small flow path through the crack between the opening and the surrounding surface if the [Model infiltration](#) option is checked.

Note 1: For **Cooling design** calculations, natural ventilation is always handled as [Scheduled](#) even when the Natural ventilation model option is set to [Calculated](#). In this case you can still enter natural ventilation air change rates and operation schedule data on the HVAC tab but *it is used only for design simulations*. The captions for data used for design calculations only is displayed with a green background in the interface.

Note 2: With both the Scheduled and Calculated options, natural ventilation can be [overridden](#) to be off in Heating and Cooling design calculations. In Heating design calculations natural ventilation is switched off by default because it is considered to be a cooling mechanism - windows and vents will normally be closed during the coldest winter periods.

Wind Factor

When carrying out conservative design simulations you may prefer to exclude (or reduce) the effects of wind pressure from Calculated natural ventilation. To exclude wind-driven airflow from the analysis altogether set the Wind factor to 0. For full treatment of wind effects set it to 1, and for intermediate treatment of wind set it to a number between 0 and 1. You can also check the impact of increasing wind pressure by entering values greater than 1.

The wind factor entered here is used to multiply the [Wind Pressure Coefficients](#) entered at surface level when writing out the modified values for EnergyPlus.

Note: Although this setting is called "Wind factor" for convenience, it is actually used as a wind pressure coefficient multiplier. The wind pressure of a surface as calculated by EnergyPlus during simulations as:

$$\text{Wind Pressure} = C_p * 0.5 * \rho * \text{Wind Speed} * \text{Wind Speed}$$

So to obtain the effect of multiplying wind speed, you should enter a squared value here. For example, if you would like to check the impact of halving the wind speeds in the weather file you should enter $0.5^2 = 0.25$ as the Wind factor.

Control mode

For calculated natural ventilation the control mode defines the way that the exterior openings are opened and closed based on air temperatures or enthalpy. The data can be entered at building, block, zone, surface or opening level.

In the descriptions that follow:

T_{out} is the outdoor air temperature,
 T_{zone} is the previous time step's zone air temperature,
 $T_{adjacent\ zone}$ is the previous time step's air temperature for the adjacent zone for interior surfaces,
 T_{set} is the [zone natural ventilation setpoint temperature](#),
 H_{zone} is the specific enthalpy of zone air from the previous time step, and
 H_{out} is the specific enthalpy of outdoor air.

The allowed choices for natural ventilation **Control mode** are:

- **1-NoVent** - all of the zone's exterior openable windows and doors are closed at all times independent of indoor or outdoor conditions. The [operation schedule](#) is ignored in this case.
- **2-Temperature** - all of the zone's openable exterior windows and doors are opened if $T_{zone} > T_{out}$ and $T_{zone} > T_{set}$ and [operation schedule](#) allows venting.
- **3-Enthalpy** - All of the zone's openable exterior windows and doors are opened if $H_{zone} > H_{out}$ and $T_{zone} > T_{set}$ and [operation schedule](#) allows venting.
- **4-Constant** - Whenever an opening's [operation schedule](#) allows venting, all of the zone's openable exterior windows and doors are open, independent of indoor or outdoor conditions. This option is useful for modelling windows that are opened for fresh air regardless of inside/outside temperature/enthalpy. Note that "Constant" here means that the size of each opening is fixed while venting; the air flow through each opening can, of course, vary from time step to time step. As with the other control mode options, the [Operation schedule](#) defined on the Openings tab controls opening availability.
- **7-CEN 15251 Adaptive** - Openable exterior windows and door are opened if the operative temperature is greater than the comfort temperature calculated from the CEN15251 adaptive comfort model and the [Venting operation schedule](#) allows venting.
- **8-ASHRAE 55 Adaptive** - Openable exterior window and doors are opened if the operative temperature is greater than the comfort temperature calculated from the ASHRAE Standard 55-2010 adaptive comfort model and the [Venting operation schedule](#) allows venting.

Both of the Adjacent temperature control options (5 and 6) revert to **2-Temperature** when applied to exterior surfaces. This allows these options to be set at building level.

Internal control mode

The allowed choices for natural ventilation **Internal control mode** are:

- **1-NoVent** - all of the zone's interior openable windows and doors are closed at all times independent of indoor or outdoor conditions. The [operation schedule](#) is ignored in this case.
- **2-Temperature** - all of the zone's openable interior windows and doors are opened if $T_{zone} > T_{out}$ and $T_{zone} > T_{set}$ and [operation schedule](#) allows venting.
- **3-Enthalpy** - All of the zone's openable interior windows and doors are opened if $H_{zone} > H_{out}$ and $T_{zone} > T_{set}$ and [operation schedule](#) allows venting.
- **4-Constant** - Whenever an internal opening's [operation schedule](#) allows venting, all of the zone's openable exterior windows and doors are open, independent of indoor or outdoor conditions. This option is useful for modelling windows that are opened for fresh air regardless of inside/outside temperature/enthalpy. Note that "Constant" here means that the size of each opening is fixed while venting; the air flow through each opening can, of course, vary from time step to time step. As with the other control mode options, the [Operation schedule](#) defined on the Openings tab controls opening availability.
- **5-Adjacent temperature cooling** - interior windows and doors are opened if $T_{zone} > T_{adjacent\ zone}$ and $T_{zone} > T_{set}$ and [operation schedule](#) allows venting.
- **6-Adjacent temperature heating** - interior windows and doors are opened if $T_{zone} < T_{adjacent\ zone}$ and $T_{zone} > T_{set}$ and [operation schedule](#) allows venting. This option is a basic attempt to get round the EnergyPlus assumption that natural ventilation is always used for cooling and allow it to heat a zone as well. Note however that the zone setpoint is still used to switch off ventilation if the temperature goes too low (rather than too high as might be expected for a heating control).
- **7-CEN 15251 Adaptive** - Openable interior windows and door are opened if the operative temperature is greater than the comfort temperature calculated from the CEN15251 adaptive comfort model and the [Venting operation schedule](#) allows venting.
- **8-ASHRAE 55 Adaptive** - Openable interior window and doors are opened if the operative temperature is greater than the comfort temperature calculated from the ASHRAE Standard 55-2010 adaptive comfort model and the [Venting operation schedule](#) allows venting.

Note: **Internal control mode** is used in the simulation only when the Internal window or vent [Control option](#) is set to **2-Use Internal control mode**, otherwise it is ignored.

Identifying which zone is "more occupied"

For interior openings the zone temperature and setpoint used to control operation is that of the "more occupied" of the 2 adjacent zones. The rules for establishing which zone is "more occupied" are as follows:

Zone is "more occupied" if:

- If zone is occupied and adjacent zone is unconditioned, or, if the 2 zones have the same zone type then the assessment is made based on:

Zone is "more occupied" if:

- It is heated and adjacent zone is not, or otherwise,
- if both zones are heated then if it has a higher setpoint temperature, or otherwise,
- It is cooled and adjacent zone is not, or otherwise,
- if both zones are cooled then if it has a lower setpoint temperature.

If both zones have identical zone type and heating and cooling setpoints are the same then an arbitrary assignment is made.

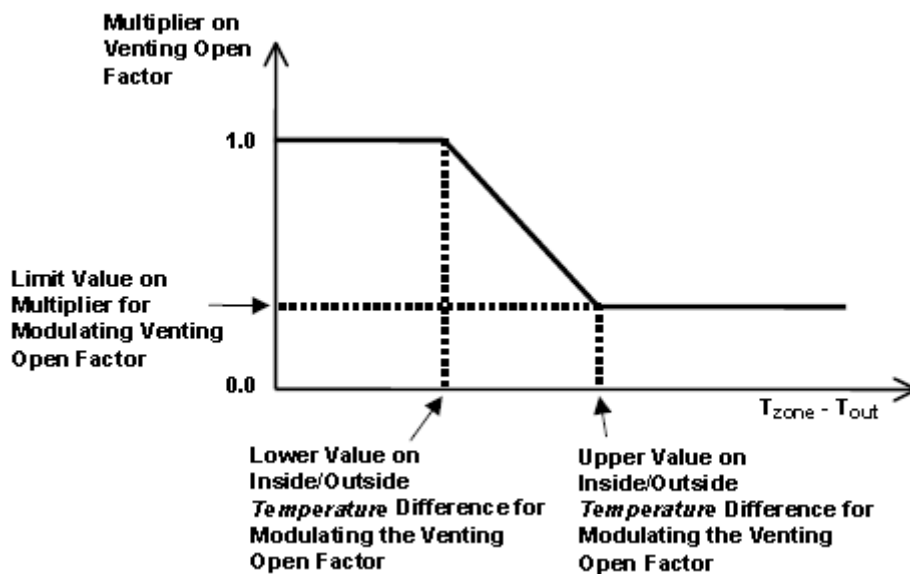
Tip: **Control mode** data is available at surface and opening level allowing you to control different windows in different ways.

Modulation of Openings

You can modulate the window/door openings when using Calculated natural ventilation by making appropriate settings to:

- **Modulate opening areas** - switch on modulation by checking this option.
- **Limit value of opening modulation factor** - gives the reduction on opening area for low outside air temperatures.
- **Lower value of Tin-Tout** - lower value on inside/outside temperature difference for modulating the venting open factor.
- **Upper value of Tin-Tout** - upper value on inside/outside temperature difference for modulating the venting open factor (defines 'low outside air temperatures').

This data is used within EnergyPlus to calculate a factor between 0 and 1 that multiplies the opening factor of each window and door in the building. Modulation of the openings can reduce the large temperature swings that can occur if the windows/doors are too far open when they are venting, especially when there is a large inside-outside temperature difference. This can avoid discomfort due to introduction of cold air and/or prevent heating system operation in summer when not really required.



Temperature difference ($T_{in} - T_{out}$)	Opening Multiplier
$T_{in} - T_{out} \leq [\text{Lower value of } T_{in} - T_{out}]$	1.0
$[\text{Lower value of } T_{in} - T_{out}] < T_{in} - T_{out} < [\text{Upper value of } T_{in} - T_{out}]$	Factor varies linearly from 1.0 to Limit value of Opening modulation factor.
$T_{in} - T_{out} \geq [\text{Upper Value of } T_{in} - T_{out}]$	Factor = Limit value of opening modulation factor.

For example if the Lower value of $T_{in} - T_{out}$ is 0, the Upper value is 15 and the Limit value of opening modulation factor is 0.05, the windows and doors are only opened 5% of their full opening area when the outside air is 15K colder than inside. When the inside and outside temperatures are the same the windows are open to their maximum opening area and when the temperature difference is 7.5K the windows are opened halfway between these extremes.

Important Notes

1. The purpose of the Modulate option is to prevent discomfort due to very cold air being introduced into the zone. You should keep in mind though that because this control will also prevent ventilation when the temperature in the space is very hot and the outside air is not particularly cold because of the large temperature difference between inside and outside.

For example if the upper modulation temperature difference is 15K and the limiting modulation factor is 0.05 then the maximum window and vent opening area will be reduced to 5% of their normal size if the temperature in the zone is 35°C and the temperature outside is 20°C. In this case the natural ventilation through the 'almost closed' openings would probably not be adequate to cool the zone back down again. So you should not use this option if you wish natural ventilation to occur when the temperature inside is very high.

2. With Modulation active, openings will be completely closed if the difference between inside and outside air temperatures is greater than **Upper value of Tin-Tout**.
3. While the modulation graph above shows the opening factor to be 1 when the inside temperature is greater than outside temperature, by default the natural ventilation openings will all be closed in this case based on the [2-Temperature control mode](#).

This modulation data can also be accessed from the HVAC tab under the Natural Ventilation > Options headers.

Tip: Although basic DesignBuilder natural ventilation options don't allow controls based on absolute outdoor air temperature, it is possible to apply upper and/or lower outdoor air temperature limits for natural ventilation through use of the mixed mode Temperature control schedule and the [Mixed mode minimum and maximum temperatures](#).

Tip: Modulation settings are available at surface and opening level allowing you to control different windows in different ways.

Mixed Mode Zone Equipment

HVAC tab in model data

See the [Mixed mode ventilation](#) tutorial

Mixed mode on

You can include mixed mode natural ventilation (called hybrid ventilation in EnergyPlus) and HVAC [change-over](#) control for zone equipment by checking the **Mixed mode on** checkbox on the HVAC tab under the Natural ventilation header.

The mixed mode control serves two purposes:

1. Prevents simultaneous natural ventilation and HVAC system cooling operation, and,
2. Allows you to test various ventilation strategies to maximize natural ventilation in order to reduce heating/cooling loads.

The mixed mode change-over controller links the associated HVAC system with the operable windows and vents in the zones served by the HVAC system. The controller overrides the natural ventilation controls for windows and vent openings, closing these openings under when conditions are unfavourable for natural ventilation and allowing the HVAC system to operate. If the controller determines that conditions are favourable for natural ventilation, then the HVAC system is disabled and the openings are able to operate based on the standard natural ventilation control mechanism (natural ventilation setpoint, maximum opening area, modulation etc).

Mixed mode controls can be used when using either Calculated or Scheduled natural ventilation, Simple or Detailed HVAC.

Note: Mixed model data on the HVAC tab is used for a) Simple HVAC and b) zone equipment such as FCU, PTAC etc when using Detailed HVAC. This data is not used to control Detailed HVAC AHUs. The AHU mixed mode controls are on the [Air Handling Unit](#) dialog and any mixed mode settings made on that dialog will override any mixed mode settings made on the HVAC model data tab for zones connected to that AHU.

See also the [Summary of Mixed mode modelling](#) for background information.

Mixed mode control zones

When using Detailed HVAC air systems a control zone is required to represent the zone conditions for all zones served by the AHU. The control zone conditions are fed into the HVAC mixed mode controller. The control zone is defined on the AHU dialog for Detailed HVAC.

Wind And Rain

Opening factor function of wind speed curve

Select a [Linear](#) or [Quadratic](#) performance curve that parameterizes the variation of opening factor as a function of outdoor wind speed. The output of this curve is multiplied by the opening factor of opening objects to give the final openness. This field only works with the AirflowNetwork (Calculated natural ventilation) opening objects.

Close windows and vents when raining

Indicates whether the rain indicator in the hourly weather data is used to shut off natural ventilation. This can help simulate conditions where one would normally close windows to avoid rain penetration in a space. Any possible rain getting into a space will not be counted as a zone load.

Maximum wind speed

The wind speed (in m/s or ft/min) above which natural ventilation is shut off. This can help simulate conditions where one would normally close windows to avoid wind problems in a space (papers blowing around, etc).

Temperature Control

Minimum outdoor temperature

The outdoor temperature (in °C or °F) below which the entire mixed mode ventilation system is shut off when the control mode schedule value = 1 (Temperature). This lower temperature limit is intended to avoid overcooling a space, which could result in a heating load.

Maximum outdoor temperature

The outdoor temperature (in °C or °F) above which the entire mixed mode ventilation system is shut off when the control mode schedule value = 1 (Temperature). This upper temperature limit is intended to avoid overheating a space, which could result in a cooling load.

Note: When using Scheduled natural ventilation, the **Minimum/Maximum outdoor temperature** Mixed mode shut off values described above will override the settings made under the **Outdoor Temperature Limits** header.

Natural Ventilation

☒ **On**

Outside air definition method: 1-By zone

Outside air (ac/h): 5.000

Operation

Schedule: Office_OpenOff_Occ

Outdoor Temperature Limits

☒ **Outdoor min temperature control**

Min temperature definition: 1-By value

Min temperature (°C): -100.0

☒ **Outdoor max temperature control**

Max temperature definition: 1-By value

Max temperature (°C): 100.0

Delta T Limits: >>

Delta T and Wind Speed Coefficients: >>

Mixed Mode Zone Equipment

☒ **Mixed mode on**

Wind and Rain: >>

Temperature Control

Min outdoor temperature (°C): -100.0

Max outdoor temperature (°C): 100.0

These settings override the Outdoor Temperature Limits highlighted above

Enthalpy Control

Minimum outdoor enthalpy

The outdoor enthalpy (in J/kg or Btu/lb) below which the entire mixed mode ventilation system is shut off when the control mode schedule value = 2 (Enthalpy).

Maximum outdoor enthalpy

The outdoor enthalpy (in J/kg or Btu/lb) above which the entire mixed mode ventilation system is shut off when the control mode schedule value = 2 (Enthalpy).

Dewpoint Control

Minimum outdoor dewpoint

The outdoor dew point temperature (in °C or °F) below which the entire mixed mode ventilation system is shut off when the control mode schedule value = 3 (Dew point). This lower dew point temperature limit is intended to avoid dehumidifying a space.

Maximum outdoor dewpoint

The outdoor dew point temperature (in °C or °F) above which the entire mixed mode ventilation system is shut off when the control mode schedule value = 3 (Dew point). This upper dew point temperature limit is intended to avoid humidifying a space.

Advanced

Control mode schedule

This schedule determines whether or not for a given time the mixed mode ventilation control is to be applied. Schedule values equal to zero indicate no ventilation control, resulting in natural ventilation and HVAC system operation being performed based on their own controls. Schedule values equal to one denote temperature control for either cooling or heating, which is determined internally based on thermostat set point. The temperature control is restricted between the minimum and maximum outdoor temperatures provided in two additional input fields (below). Schedule values equal to two denote enthalpy control, which is restricted between minimum and maximum outdoor enthalpy values given in two additional input fields (below). Schedule values equal to three denote dew point control for either dehumidification or humidification. Schedule values equal to four represent outdoor ventilation air control.

Tip: The standard Calculated DesignBuilder natural ventilation options don't allow controls based on absolute outdoor air temperature, but it is possible to apply upper and/or lower outdoor air temperature limits for calculated natural ventilation through use of the Temperature control schedule mixed mode and the Mixed mode minimum and maximum temperatures.

Schedule value

0

1

2

3

4

(Calculated natural ventilation only)

Operation mode

No mixed mode control - HVAC and natural ventilation operate separately according to their own controls.

Temperature control for either cooling or heating - temperature control is restricted between minimum and maximum outdoor temperatures. When the outside air temperature falls below the minimum value or goes above the maximum value the openings are closed and the HVAC system operates.

Enthalpy control, outside enthalpy is restricted between minimum and maximum outdoor enthalpy values.

Dew point control, outside dew point temperature is restricted between minimum and maximum outdoor dew point values.

The minimum outdoor ventilation air for a given time in ac/h. The program calculates the natural (outdoor) ventilation in the controlled zone first and compares the amount of outdoor air brought by opening windows or doors and other small openings to this minimum value. If the amount of outdoor air from natural ventilation is less than the minimum value, the natural ventilation is shut off (i.e., the window or door openings are closed) and the HVAC system may operate if needed. Otherwise, the natural ventilation is on and the HVAC system is shut off. The amount of outdoor ventilation air entering the controlled zone is determined as air from outdoors and not from other adjacent zones. Therefore, this option is only applied to a zone having a window or door exposed to outdoors.

Minimum outdoor ventilation air schedule

The schedule is only used if [Ventilation Control Mode](#) = 4 (Outdoor Ventilation Air Control). This schedule determines the minimum outdoor ventilation air for a given time in the units of air change per hour (ac/h). The program calculates the natural (outdoor) ventilation in the controlled zone first and compares the amount of outdoor air introduced by opening windows or doors and other small openings to this minimum value. If the amount of outdoor air from natural ventilation is less than the minimum value, the natural ventilation is shut off (i.e., the window or door openings are closed) and the HVAC system may operate if needed. Otherwise, the natural ventilation is on and the HVAC system is shut off. The amount of outdoor ventilation air entering the controlled zone is determined as air from outdoors and not from other adjacent zones. Therefore, this option is only applied to a zone having a window or door exposed to outdoors.

AirflowNetwork control type schedule

This schedule is available when Calculated natural ventilation is set. It determines for a given simulation timestep how the openings respond to the mixed mode ventilation control. Schedule values equal to 0 indicate individual ventilation control based on the control requirements specified for each individual AirflowNetwork opening object. Schedule values equal to 1 denote "group" control. The opening objects exposed to outdoors in the controlled zone served by the primary [air loop](#) will be considered as a master to provide group control.

Simple airflow control type schedule

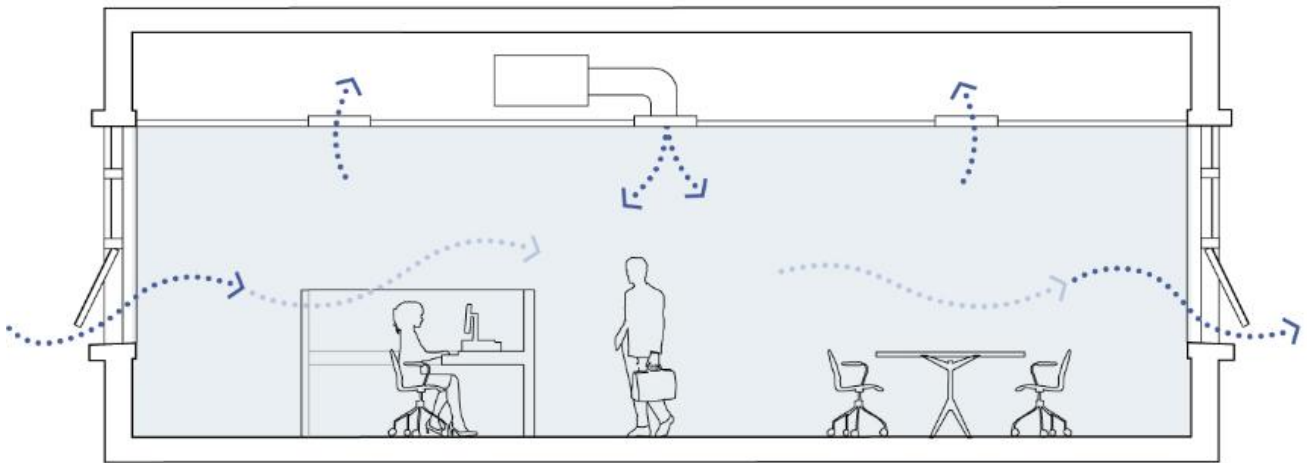
This schedule is available when the Scheduled natural ventilation model option is set. It works with ZoneVentilation:* and ZoneMixing objects only. This schedule determines for a given simulation timestep how the ZoneVentilation:* and ZoneMixing objects respond to the hybrid ventilation control when the hybrid ventilation control allows the objects to operate.

Schedule values equal to zero indicate the individual ventilation control based on the control requirements from their own objects. Schedule values equal to one denote group control. The ZoneVentilation:* and ZoneMixing objects in the zones served by the primary air loop are controlled by a single object, whose name is provided in the following input field.

Mixed Mode Cooling

In mixed-mode buildings, natural ventilation is the primary means of cooling, but when this is inadequate to provide comfort conditions, active cooling is introduced. The [Center for the Built Environment \(CBE\)](#) describes mixed mode buildings as follows.

Mixed-mode buildings employ a hybrid approach to space conditioning that combines operable windows and mechanical cooling. By taking advantage of the strengths of both systems, well-designed mixed-mode buildings can be more comfortable and use less energy. People who have more control over their indoor environments are typically more satisfied with temperature, air movement, and air quality. Mixed-mode buildings give occupants control while still allowing central control to keep the building near the comfort zone. The three basic categories of mixed-mode buildings describe how natural ventilation and mechanical cooling operate at different times and places. **Concurrent** systems allow open windows and air conditioning at the same time in the same place; **Changeover** systems allow them at different times in the same places; **Zoned** systems allow them at the same time in different places.



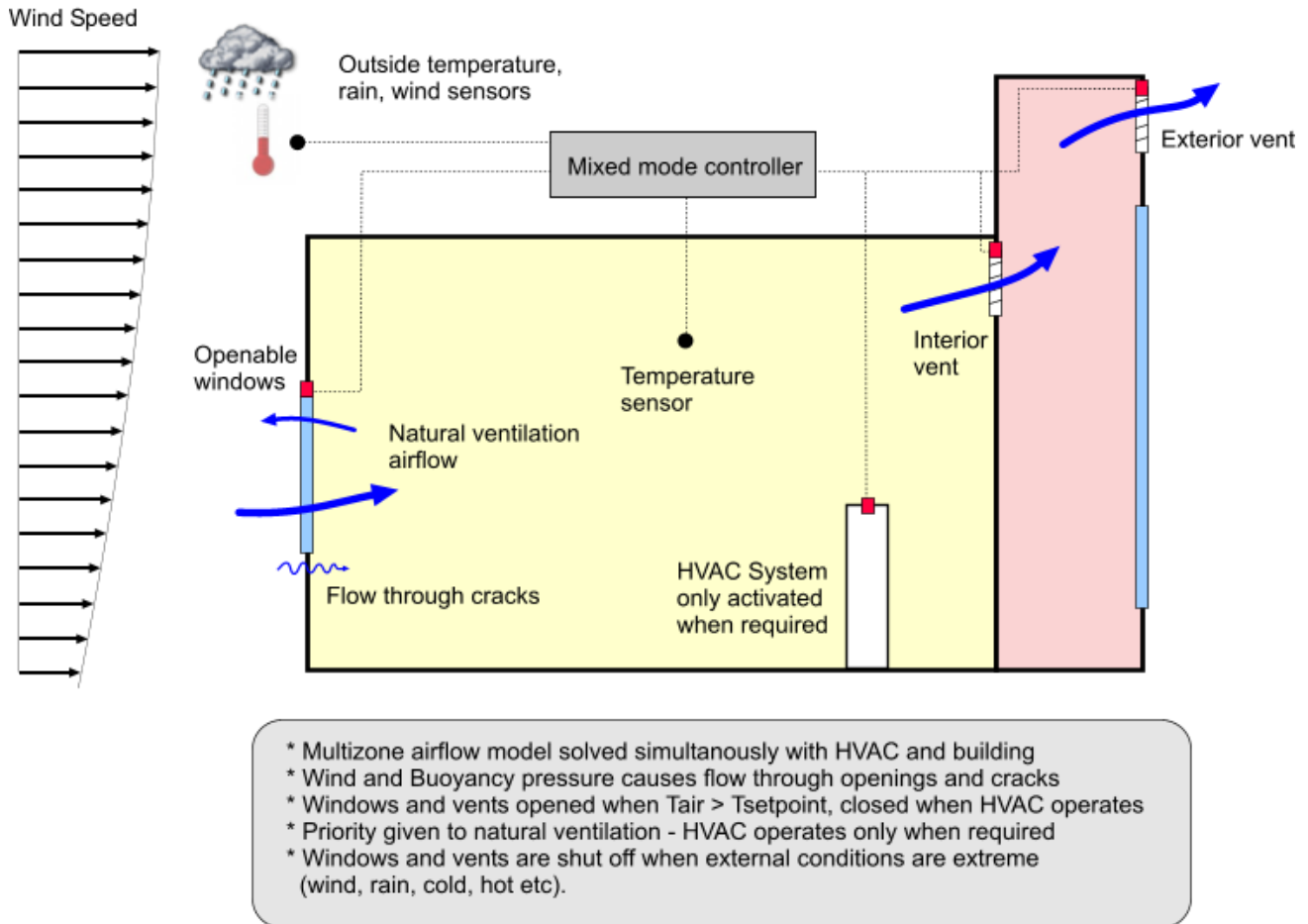
Mixed Mode Schematic from the [CBE website](#)

DesignBuilder can model all three types of mixed mode system.

1. **Concurrent** (*same space, same time*)
 Concurrent mixed-mode operation is the most prevalent design strategy in practice today, in which the air-conditioning system and operable windows operate in the same space and at the same time. The HVAC system may serve as supplemental or “background” ventilation and cooling while occupants are free to open windows based on individual preference. Typical examples include open-plan office space with standard VAV air-conditioning systems and operable windows, where perhaps perimeter VAV zones may go to minimum air when sensor indicates that a window has been opened.
2. **Changeover** (*same space, different times*)
 Changeover designs are becoming increasingly common, where the building switches between natural ventilation and air-conditioning modes on a seasonal or even daily basis. The building automation system may determine the mode of operating based on outdoor temperature, an occupancy sensor, a window (open or closed) sensor, or based on operator commands. Typical examples include individual offices with operable windows and personal air conditioning units that shut down for a given office any time a sensor indicates that a window has been opened; or a building envelope where automatic louvers open to provide natural ventilation when the HVAC system is in economizer mode, and then close when the system is in cooling or heating mode.
3. **Zoned** (*different spaces, same time*)
 Zoned systems are also common, where different zones within the building have different conditioning strategies. Typical examples include naturally ventilated office buildings with operable windows and a ducted heating/ventilation system, or supplemental mechanical cooling provided only to conference rooms. For many mixed-mode buildings, operating conditions sometimes deviate somewhat from their original design intent (e.g., a building originally designed for seasonal changeover between air-conditioning and natural ventilation may, in practice, operate both systems concurrently).

Note: **Concurrent** and **Zoned** systems can be modelled simply by including natural ventilation with HVAC systems without using DesignBuilder mixed mode control. In DesignBuilder, the term “mixed mode” refers specifically to **Changeover** systems where the HVAC and the natural ventilation do not take place at the same time.

Mixed Mode Natural Ventilation Flow and Control



DesignBuilder supports mixed mode in all combinations of natural ventilation and HVAC model options.

The steps to setting up a change-over mixed mode model in DesignBuilder are:

1. Include active cooling and natural ventilation by switching on the [Natural ventilation](#) on and [Cooled](#) options on the HVAC tab.
2. Also, include mechanical ventilation if there is outside air introduced through the HVAC system. Note that Detailed HVAC AHUs allow [Mixed mode controls](#) to be defined to model optimal interaction between the natural ventilation system (Scheduled or Calculated) and the AHU.
3. Ensure that the [setpoints](#) for natural ventilation and cooling operation are correct. The natural ventilation (window/vent opening) setpoint should ideally be 2°C lower than the cooling setpoint to ensure maximum use of natural ventilation and to avoid the active cooling system being used unnecessarily and also 2°C higher than the heating .temperature. Typical values might be heating setpoint temperature = 21°C, natural ventilation setpoint temperature = 23°C, cooling setpoint temperature = 25°C. Heating should ideally be switched off in the summer to avoid unnecessary heating caused by natural ventilation overcooling the space.
4. See the [mixed mode](#) section on the HVAC help for detailed information on the data that can be set, including conditions that require windows and vents to be closed such as rain, wind and low outside air temperature.

Note: "Mixed mode" means the same as the term "Hybrid ventilation" used in the EnergyPlus documentation.

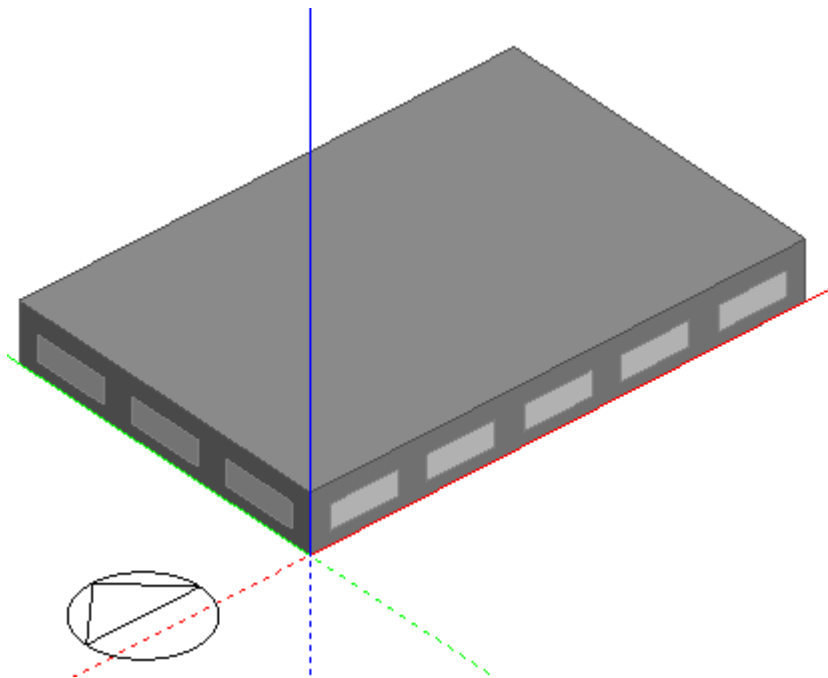
Mixed Mode Tutorial

This tutorial will help you get up to speed with running a changeover mixed mode natural ventilation study in a few easy steps.

In general, mixed mode controls can be used with either Calculated or Scheduled natural ventilation, Simple or Detailed HVAC. This tutorial demonstrates the process for the [Simple HVAC](#) and [Scheduled natural ventilation](#) model options as the easiest way to learn how to apply the mixed mode settings.

Step 1 - Create a base model

Start by creating a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m as shown below. Use default Model options.



In Simple HVAC mode, HVAC components are controlled using the Model data inputs on the Activity and HVAC tabs. The following steps describe the settings required on these tabs.

Step 2 – Make setpoint temperature settings on the Activity tab

To prevent simultaneous operation of the cooling and natural ventilation systems, the setpoint temperatures must be defined carefully. The recommended deadband between natural ventilation and heating/cooling is 2°C.

In this example, these setpoint temperatures should be entered as follows:

- [Heating setpoint](#): 21°C
- [Natural ventilation setpoint](#): 23°C
- [Cooling setpoint](#): 25°C.

The Activity tab, Environmental Control settings should now look like this:

Environmental Control		⌵
Heating Setpoint Temperatures		⌵
🔥 Heating (°C)	21	
🔥 Heating set back (°C)	12.0	
Cooling Setpoint Temperatures		⌵
❄️ Cooling (°C)	25	
❄️ Cooling set back (°C)	28.0	
Humidity Control		⌵
Ventilation Setpoint Temperatures		⌵
Natural Ventilation		⌵
☒ Indoor min temperature control		
Min temperature definition	1-By value	⌵
🔥 Min temperature (°C)	23	
☐ Indoor max temperature control		

Step 3 – Load the “Radiator heating, Boiler HW, Mixed mode Nat Vent, Local comfort cooling” HVAC Template

1. By default, DesignBuilder will load the “Fan Coil Unit” HVAC template in new models. For this example, you should change that by navigating to building level, clicking on the HVAC model data tab and loading the “Radiator heating, Boiler HW, Mixed mode Nat Vent, Local comfort cooling” HVAC Template.
2. Set the [maximum natural ventilation rate](#) to 3 ac/h.
3. Select the “Winter heating (Northern Hemisphere)” [Heating Operation Schedule](#) to ensure that heating is switched off in summer.

The HVAC tab should now look something like the screenshot below:

HVAC Template		▼
Template		Radiator heating, Boiler HW, Mixed mode Nat
Mechanical Ventilation		>>
Auxiliary Energy		▼
Pump etc energy (W/m ²)	0.0000	
Schedule	Office_OpenOff_Occ	
Heating		▼
☒ Heated		
Fuel	2-Natural Gas	▼
Heating system seasonal CoP	0.850	
Sizing Zone Equipment		>>
Type		>>
Operation		▼
Schedule		Winter heating (Northern Hemisphere)
Cooling		▼
☒ Cooled		
Cooling system	Default	
Fuel	1-Electricity from grid	▼
Cooling system seasonal CoP	1.800	
Supply Air Condition		>>
Operation		▼
Schedule		Office_OpenOff_Cool
Humidity Control		>>
DHW		>>
Natural Ventilation		▼
☒ On		
Outside air definition method	1-By zone	▼
Outside air (ac/h)	3.000	
Operation		▼
Schedule		Office_OpenOff_Occ
Outdoor Temperature Limits		>>
Delta T Limits		>>
Delta T and Wind Speed Coefficients		>>
Mixed Mode Zone Equipment		▼
☒ Mixed mode on		
Wind and Rain		>>
Temperature Control		>>
Enthalpy Control		>>
Dew Point Control		>>
Advanced		>>

Note that the heating, cooling, natural ventilation and mixed mode controls are all switched on.

Step 4 – Review the Mixed Mode Zone Equipment Settings

DesignBuilder provides a set of [Mixed Mode Zone Equipment](#) settings to allow various detailed aspects of the system to be controlled, as shown below.

Mixed Mode Zone Equipment	
☒ Mixed mode on	
Wind and Rain	
☐ Opening factor function wind speed c...	Opening Factor Function of Wind Speed Curve
☒ Close windows and vents when raining	
Max wind speed (m/s)	40.0
Temperature Control	
Min outdoor temperature (°C)	-100.0
Max outdoor temperature (°C)	100.0
Enthalpy Control	
Min outdoor enthalpy (J/kg)	20000
Max outdoor enthalpy (J/kg)	30000
Dew Point Control	
Min outdoor dew point temperature (°C)	15.0
Max outdoor dew point temperature (°C)	30.0
Advanced	
Control mode schedule	Mixed mode temperature control
Min outdoor ventilation air schedule	Minimum outdoor ventilation air schedule: Always 1 ac/
Scheduled airflow control type sched...	Simple Airflow Control Type Schedule: Always 0

For this tutorial you can use the defaults.

Step 5 – Reduce Lighting gains

To ensure that lighting internal gains are consistent with a modern naturally ventilated building, a more efficient system than the default “Reference” template should be loaded.

At building level, go to the Lighting tab and select the “LED with linear control” Lighting template.

The data on the Lighting tab should now be as below.

Lighting Template	
Template	LED with linear control
General Lighting	
☒ On	
Normalised power density (W/m2-100 lux)	2.5000
Schedule	Office_OpenOff_Light
Luminaire type	1-Suspended
Radiant fraction	0.420
Visible fraction	0.180
Convective fraction	0.400
Lighting Control	
☒ On	
Working plane height (m)	0.80
Control type	1-Linear
Min output fraction	0.100
Min input power fraction	0.100
Glare	>>
Lighting Area 1	>>
Lighting Area 2	>>

Step 6 – Reduce Solar gains

Similarly, to ensure that solar gains are kept in check and consistent with a modern naturally ventilated building, some window shading should be introduced.

At building level, go to the Openings tab and check the [Window shading](#) checkbox. By default, this will provide high reflectivity internal blinds for all windows.

The data on the Openings tab should now be as below.

Glazing Template	
Template	Project glazing template
External Windows	
Glazing type	Project external glazing
Layout	Preferred height 1.5m, 30% glazed
Dimensions	
Type	3-Preferred height
Window to wall %	30.00
Window height (m)	1.50
Window spacing (m)	5.00
Sill height (m)	0.80
Outside reveal depth (m)	0.000
Frame and Dividers	
Shading	
☒ Window shading	
Type	Blind with high reflectivity slats
Position	1-Inside
Control type	3-Schedule
Slat Angle Control	
Operation	
Schedule definition	1-Follow occupancy
☐ Local shading	

Step 7 - Run a simulation

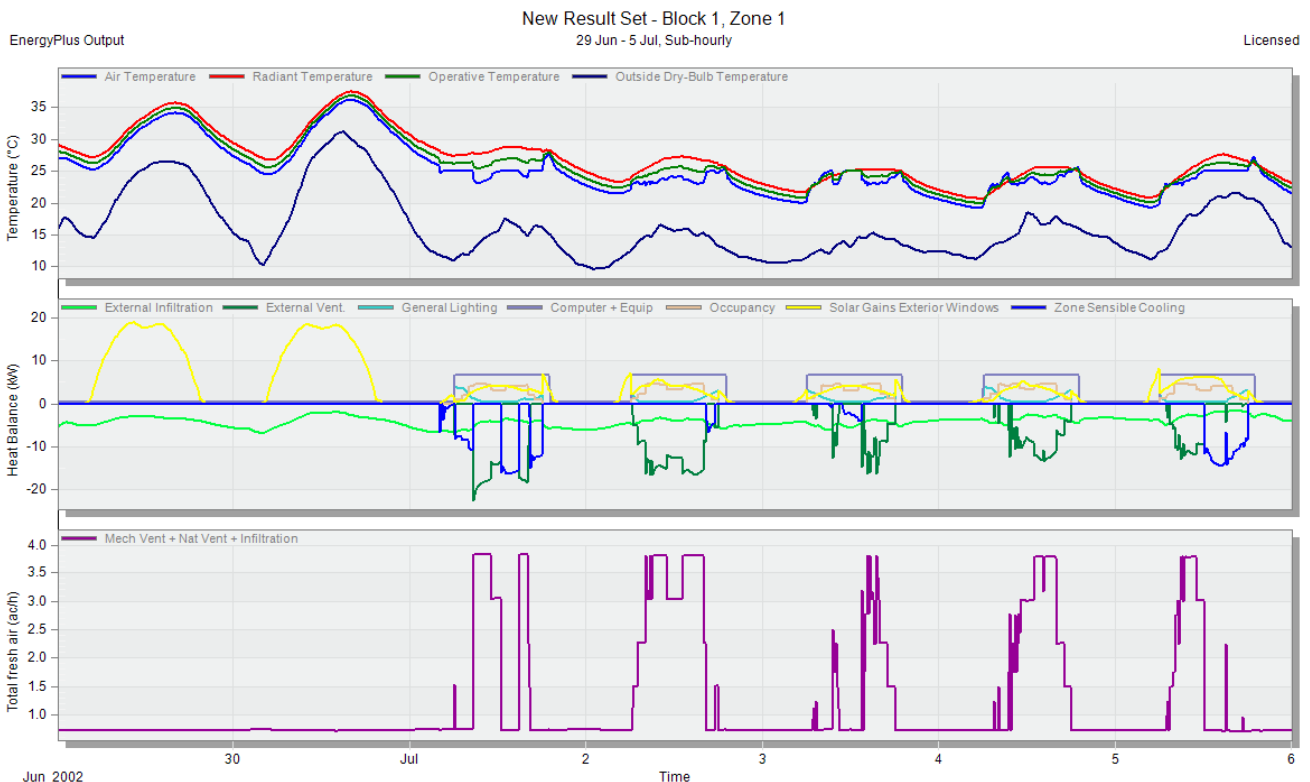
We are now ready to run a test simulation.

1. Go to the Simulation screen and press the **Update** toolbar icon to open the [Simulation Calculation options](#) dialog.
2. Click on the "Summer typical week" Info panel link to load simulation dates for the typical summer week for this location.
3. Check the **Sub-hourly Output Interval for Reporting** to help you check the correct operation of the system.
4. On the [Options](#) tab, select the 6 [Timesteps per hour](#) option.
5. Press OK to run the Simulation.

Step 8 - Review Results

Select the **5-Sub-hourly** Interval in the Display options bottom left panel and review the results.

You should see graphs similar to those below.



The temperature graph shows that, during the working week, the inside air temperatures are controlled to be between 23 and 25°C during occupied periods. Temperatures are controlled to 23°C when natural ventilation is active, but when natural ventilation cannot meet the cooling demand, mechanical cooling is activated to maintain temperatures below 25°C.

The heat balance graph shows that cooling (blue line) and natural ventilation (green line) occur independently (never simultaneously) to meet the cooling load as required.

Earth Tubes

HVAC tab in model data

An earth tube is a long, underground metal or plastic pipe through which air is drawn. During the cooling season, as air travels through the pipe, it gives up some of its heat to the surrounding soil and enters the room as cooler air. Similarly, during heating season, as air travels through the pipe, it receives some of its heat from the soil and enters the room as warmer air. Simple earth tubes in EnergyPlus can be controlled by a schedule and through the specification of minimum, maximum, and delta temperatures as described below. As with infiltration and ventilation, the actual flow rate of air through the earth tube can be modified by the temperature difference between the inside and outside environment and the wind speed. See [equation below](#).

Note 1: Earth tubes are only available when using the [Scheduled natural ventilation](#) model option.

Note 2: Earth tube settings are used for heating and cooling design as well as in simulations.

CalcSoilSurfTemp – Auxiliary Programs Document

The **CalcSoilSurfTemp** program is simple and requires only two input fields : soil condition and soil surface condition in addition to a valid weather file. For soil condition, the user should select the number corresponding to the actual condition of the soil surrounding the earth tube from the four typical options to determine the thermal diffusivity and thermal conductivity of the surrounding soil. For soil surface conditions, the user should select the number corresponding to the actual condition of the ground surface above the earth tube from the eight following options:

- 1. BARE AND WET,
- 2. BARE AND MOIST,
- 3. BARE AND ARID,
- 4. BARE AND DRY,
- 5. COVERED AND WET,
- 6. COVERED AND MOIST,
- 7. COVERED AND ARID and
- 8. COVERED AND DRY.

This determines the absorption coefficient and the fraction of evaporation rate of the ground surface.

From this information and an analysis of the weather for the location selected, the **CalcSoilSurfTemp** program (ref. [Auxiliary Programs](#) document) calculates the three parameters described below- [Average soil surface temperature](#), [Amplitude of soil surface temperature](#) and [Phase constant of soil surface temperature](#). The full input description of an earth tube is given below.

Include earthtube

Check this option to include zone air delivery through earth tubes.

Outside air definition method

You can set the amount, timing and control of outside air delivery through the earth tubes in the same way as for Mechanical ventilation. Refer to the [Outside air](#) section for details of how this works.

The outside air definition data is used to calculate the the maximum amount of air mass flow rate of the earth tube expected at design conditions (E_{design} in the above equation in units of m³/s. The design value is modified by the schedule fraction (see next field) and user specified coefficients (last four fields).

Operation

Schedule

Select the [schedule](#) that modifies the maximum design volume flow rate (see previous field). The fraction in the schedules should vary between 0.0 and 1.0 and is referred to as F_{schedule} in the above equation.

Control

Minimum zone temperature when cooling

This is the indoor temperature (in °C or °F) below which the earth tube is shut off. This lower temperature limit is intended to avoid overcooling a space and thus result in a heating load. For example, if the user specifies a minimum temperature of 20°C, earth tube is assumed to be available if the zone air temperature is above 20°C. If the zone air temperature drops below 20°C, then earth tube is automatically turned off.

Maximum zone temperature when heating

This is the indoor temperature (in °C or °F) above which the earth tube is shut off. This higher temperature limit is intended to avoid overheating a space and thus result in a cooling load. For example, if the user specifies a maximum temperature of 20°C, earth tube is assumed to be available if the zone air temperature is below 20°C. If the zone air temperature rises above 20°C, then earth tube is automatically turned off.

Delta temperature

This is the temperature difference (in °C or °F) between the indoor and outdoor air dry-bulb temperatures below which the earth tube is shut off. This is to allow the earth tube to be stopped either if the temperature outside is too warm and could potentially heat the space or if the temperature outside is too cold and could potentially cool the space. For example, if the user specifies a delta temperature of 2°C, earth tube is assumed to be available if the temperature difference between indoor and outdoor temperature is at least 2°C. If the outside air dry-bulb temperature is less than 2°C cooler or warmer than the indoor dry-bulb temperature, then the earth tube is automatically turned off.

Fan Electric Consumption Configuration

Earth tube type

This alpha character string defines the type of earth tube as one of the following options:

- **1-Natural** which is assumed to be air movement/exchange that will not consume any fan energy or is the result of natural air flow through the tube and into the building. Values for fan pressure and efficiency for a natural flow earth tube are ignored. The conditions of the air entering the space are assumed to be equivalent to the air which is cooled or heated by passing along the pipe.
- **2-Intake** where values for fan pressure and efficiency define the fan electric consumption and an appropriate amount of fan heat is added to the air stream.
- **3-Exhaust** where values for fan pressure and efficiency define the fan electric consumption.

Fan pressure rise

This is the pressure rise experienced across the fan (in Pascals (N/m²) or psi). This is a function of the fan and plays a role in determining the amount of energy consumed by the fan. This value is only entered for fan-driven earth tubes.

Fan total efficiency

This is the total fan efficiency (a decimal number between 0.0 and 1.0). This is a function of the fan and plays a role in determining the amount of energy consumed by the fan. This value is only entered for fan-driven earth tubes.

Pipe Specifications

Pipe radius

This is the radius of the earth tube/pipe (in m or ft). This plays a role in determining the amount of heat transferred from the surrounding soil to the air passing along the pipe. If the pipe has non-circular cross section, user can use the concept of hydraulic diameter as follows.

$$D = 4 \cdot A / \text{Perimeter}$$

However, since this field requires the pipe radius, hydraulic diameter should be divided by two.

Pipe thickness

This is the thickness of the pipe wall (in m or ft). This plays a role in determining the amount of heat transferred from the surrounding soil to the air passing along the pipe.

Pipe length

This is the total length of the pipe (in m or ft). This plays a role in determining the amount of heat transferred from the surrounding soil to the air passing along the pipe. As the length of the pipe becomes longer, the amount of the heat transfer becomes larger.

Pipe thermal conductivity

This is the thermal conductivity of the pipe (in W/mC or Btu-in/h-ft²-F). This plays a role in determining the amount of heat transferred from the surrounding soil to the air passing along the pipe.

Pipe depth under ground surface

This is the depth of the pipe under the ground surface (in m or ft). This plays a role in determining the temperature of the soil surrounding the pipe.

Soil Heat Transfer Calculations

Soil condition

This alpha character string defines the actual condition of the soil surrounding the earth tube and can be one of any of the following options:

- **1-Heavy and Saturated,**
- **2-Heavy and Damp,**
- **3-Heavy and dry** or
- **4-Light and Dry.**

This determines the thermal diffusivity and thermal conductivity of the surrounding soil, which play a role in determining the amount of heat transferred from the surrounding soil to the air passing along the pipe.

Average soil surface temperature

This is the annual average soil surface temperature straight above the earth tube, which plays a role in determining the temperature of the soil surrounding the pipe. This field should be calculated in advance using the separate **CalcSoilSurfTemp** program.

Amplitude of soil surface temperature

This is the amplitude of soil surface temperature above the earth tube, which plays a role in determining the temperature of the soil surrounding the pipe. This is the difference between the maximum and minimum soil surface temperature for the whole year divided by two. This field should be calculated in advance using the separate **CalcSoilSurfTemp** program. It is a Delta temperature value.

Phase constant of soil surface temperature

This is the phase constant of the soil surface temperature straight above the earth tube, which play a role in determining the temperature of the soil surrounding the pipe at particular time. This is the time elapsed from the beginning of the year until the soil surface temperature reaches the minimum value of the year. This field should be calculated in advance using the separate **CalcSoilSurfTemp** program.

Flow Term Coefficients

The basic equation used to calculate air flow rate of earth tube in EnergyPlus is:

$$\text{EarthTubeFlowRate} = E_{\text{design}} F_{\text{schedule}} (A + B \cdot \text{abs}(T_{\text{zoneair}} - T_{\text{odb}}) + C \cdot \text{WindSpeed} + D \cdot \text{WindSpeed}^2)$$

Constant term flow coefficient

This number is the “A” parameter in the above earth tube equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter, however, is a constant under all conditions and is not modified by any environmental effect. As a result, it is dimensionless.

Temperature term flow coefficient

This number is the “B” parameter in the above earth tube equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter is modified by the temperature difference between the outdoor and indoor air dry-bulb temperatures. The units for this parameter are inverse Celsius.

Velocity term flow coefficient

This number is the “C” parameter in the above earth tube equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter is modified by the speed of wind being experienced outside the building. The units for this parameter are s/m.

Velocity squared term flow coefficient

This number is the “D” parameter in the above earth tube equation. It is part of the user specified modifying parameters that are a function of environmental factors. This parameter is modified by square of the speed of wind being experienced outside the building. The units for this parameter are s²/m².

Outputs

Note: Airflow introduced into zones by Earth tubes is included in the **Mech Vent + Nat Vent + Infiltration** ac/h output. Also the heating/cooling effect of Earth tubes on the zone is included in the **Zone Vent** heat balance output. If you would like to see further details on the impact of the Earth tubes in your results then you should include one or more of the output variables below to an [external idf file](#), "include" it in the main IDF file generated by DesignBuilder and view results in the [Results Viewer](#).

Current Earth Tube output variables:

- HVAC,Sum,Earth Tube Zone Sensible Cooling Energy [J]
- HVAC,Average,Earth Tube Zone Sensible Cooling Rate [W]
- HVAC,Sum,Earth Tube Zone Sensible Heating Energy [J]
- HVAC,Average,Earth Tube Zone Sensible Heating Rate [W]
- HVAC,Sum,Earth Tube Air Flow Volume [m³]
- HVAC,Average,Earth Tube Air Volume Flow Rate [m³/s]
- HVAC,Sum,Earth Tube Air Flow Mass [kg]
- HVAC,Average,Earth Tube Air Mass Flow Rate [kg/s]
- HVAC,Sum,Earth Tube Fan Electric Energy [J]
- HVAC,Average,Earth Tube Fan Electric Power [W]
- HVAC,Average,Earth Tube Zone Inlet Air Temperature [C]
- HVAC,Average,Earth Tube Ground Interface Temperature [C]
- HVAC,Average,Earth Tube Outdoor Air Heat Transfer Rate [W]

Earth Tube Zone Sensible Cooling Energy [J]

Earth Tube Zone Sensible Cooling Rate [W]

These are the energy and rate associated with the zone cooling provided by the air from the earth tube. This occurs when the earth tube outlet air temperature is less than zone air temperature.

Earth Tube Zone Sensible Heating Energy [J]

Earth Tube Zone Sensible Heating Rate [W]

These are the energy and rate associated with the zone heating provided by the air from the earth tube. This occurs when the earth tube outlet air temperature is greater than the zone air temperature.

Earth Tube Air Flow Volume [m³]

The volume flow of air through the earth tube.

Earth Tube Air Volume Flow Rate [m³/s]

The volume flow rate of air through the earth tube.

Earth Tube Air Flow Mass [kg]

The mass flow of air through the earth tube.

Earth Tube Air Mass Flow Rate [kg/s]

The mass flow rate of air through the earth tube.

Earth Tube Fan Electric Energy [J]

Earth Tube Fan Electric Power [W]

These are the fan electricity consumption and power for intake or exhaust earth tube types.

Earth Tube Zone Inlet Air Temperature [C]

This is the temperature of the air entering the zone after passing through the earth tube [C]. This temperature includes the cooling or heating of outdoor air as it passes along the pipe. When intake fan assist is used, then the additional heat due to the fan is included in the inlet air temperature.

Earth Tube Ground Interface Temperature [C]

This is the average temperature of the ground along the outer surface of the earth tube [C].

Earth Tube Outdoor Air Heat Transfer Rate [W]

This is the rate of heat transfer from the earth tube to the outdoor air [W]. Positive values indicate the rate at which outdoor air is preheated; negative values indicate the rate of precooling.

Domestic Hot Water

[HVAC tab](#) in model data Simple HVAC

The effect of hot water consumption (hot water taps, showers etc) on building energy use can be included in your thermal model by switching on [DHW](#) on the HVAC tab.

When using Simple HVAC, Domestic Hot Water (DHW) is modelled in DesignBuilder by using hot water consumption rates based on the [activity](#) in each zone. DHW hot water consumption is defined on the [Activity tab](#) and the definition of the hot water generating equipment is described below) is defined on the [HVAC tab](#).

Type

Three types of DHW are available:

- **1-Boiler** - heat is supplied by the same boiler used for heating the building
- **2-Gas** - gas-fired hot water system
- **3-Electric** - electric immersion heater

Delivery hot water temperature

The temperature of the water supplied to the taps.

Mains supply temperature

The average temperature of the water delivered to the building over the year. This can be approximated to the annual average outside air temperature.

Operation

The DHW Operation schedule defines the timing of the hot water draw off.

The [DHW consumption rate](#) is defined on the Activity tab in l/m2-day (gal/ft2-day in IP units) but this needs processing before being passed into E+ which requires max DHW flow in m3/s. To do this DesignBuilder needs to use the concept of number of operation hours per day (or 'Typical hours per day') based on this operation schedule.

When a [Compact schedule](#) is used to define DHW operation, DesignBuilder calculates the 'Typical hours per day' based on the day in the year which has the most operation hours. Fractional hours operation are included as fractions. A 30 minute timestep through the year is used to perform this calculation.

Example 'Typical hours per day' calculation for Compact schedules

Schedule: Compact,
Example Occupancy,
Fraction,
Through: 31 Dec,
For: Weekdays SummerDesignDay,
Until: 08:00, 1,
Until: 16:00, 0,
Until: 18:00, 0.5,
Until: 24:00, 1,
For: Weekends,
Until: 08:00, 1,
Until: 20:00, 0,
Until: 24:00, 1,
For: AllOtherDays,
Until: 08:00, 1,
Until: 16:00, 0,
Until: 18:00, 0.5,
Until: 24:00, 1;

For this schedule, the Weekdays give the most operation hours:

Until: 08:00, 1,
Until: 16:00, 0,
Until: 18:00, 0.5,
Until: 24:00, 1,

In this case 'Typical hours per day' would be calculated as:

$8 * 1 + 8 * 0 + 2 * 0.5 + 6 * 1 = 15 \text{ hours.}$

When 7/12 schedules are used the typical occupancy hours per day is determined from **January/Monday** data.

When typical workday schedules are used, this is simply the number of hours occupied during the working day.

The DHW volumetric consumption rate supplied to EnergyPlus is calculated as:

$$\text{DHW (m3/s)} = \text{DHW (l/m2-day)} \times 0.001 \times \text{m2} / (\text{Typical hours per day} \times 3600)$$

Note: the zone DHW operation schedule is also used in Detailed HVAC when the default **1-Simple HVAC Detailed HVAC Activity data** setting is made.

DHW Type

A range of DHW types can be selected but note that different types of DHW system do not affect results in EnergyPlus simulations, apart from **1- Same as HVAC** which causes heating CoP and fuel to be taken from the heating system. Other options all give the same results.

- 1- Same as HVAC
- 2- Dedicated DHW boiler
- 3- Stand alone water heater
- 4- Instantaneous DHW only
- 5- Instantaneous Combi
- 6- Direct (or off-peak) electricity
- 7- Heat pump

Air Temperature Distribution

[HVAC tab](#) in model data

By default EnergyPlus assumes that air temperature within a zone is completely uniform (i.e. the air is fully mixed). However a number of other modelling options are provided to account for non-uniform room air temperatures that may occur within the interior air volume of a zone.

Although there are many types of analyses (comfort, indoor air quality, etc.) that might benefit from localized modelling of how room air varies across space, only the temperature distribution of room air within the zone is currently addressed in EnergyPlus. This allows surface heat transfer and air system heat balance calculations to be made taking into account natural thermal stratification of air and different types of intentional air distribution designs such as under-floor and side-wall displacement ventilation that purport to extract room air at higher-than-mean temperatures. Note that EnergyPlus does not have completely general methods of modelling room air that are applicable to every conceivable type of airflow that might occur in a zone. Such models (e.g. RANS-CFD) are too computationally expensive to use with EnergyPlus for the foreseeable future. The models that are available in EnergyPlus offer only limited modelling capabilities for select room airflow configurations. Also note that because the complete mixing model for room air has long been the standard in building energy simulation, there is not currently a consensus on how to best model non-uniform air temperatures in buildings. Therefore, it is up to the user to have a good understanding of when, where, and how to apply the room air models available in EnergyPlus. The rest of this section provides some guidance in the way of examples and further discussion of the models available in EnergyPlus.

The following room air temperature distribution models are provided:

- **1-Mixed** the default option where the air temperature is modelled as being uniform through the zone.
- **2-Dynamic gradient** which allows air temperature stratification within a zone to be modelled by setting up a temperature gradient which varies dynamically depending on any of: outside temperature, inside temperature, inside-outside temperature difference, heating load or cooling load.
- **3-Three node displacement ventilation** which uses 3 nodes to model spaces that are served by a low velocity floor-level displacement ventilation air distribution system and the dominant sources of heat gain are from people and other localized sources located in the occupied part of the room.
- **4-Underfloor air distribution interior** which uses upper and a lower sub-zone nodes to model interior spaces that are served by an underfloor air distribution system and the dominant sources of heat gain are from people, equipment, and other localized sources located in the occupied part of the room.
- **5-Underfloor air distribution exterior** which uses upper and a lower sub-zone nodes to model exterior spaces that are served by an underfloor air distribution system. The dominant sources of heat gain should be from people, equipment, and other localized sources located in the occupied part of the room, as well as convective gain coming from a warm window.

Air Temperature Distribution - Dynamic Gradient

[HVAC tab in model data](#)

The **Dynamic Gradient** Air Distribution model data allows you to model stratification within a zone by setting up a temperature gradient which varies dynamically depending on:

- Outside temperature.
- Inside temperature.
- Inside-outside temperature difference.
- Heating load.
- Cooling load.

Gradient interpolation mode

This field specifies how the program will vary between the two gradients. Select one of the following keywords to choose the simulation data used to scale: 'Outdoor Environment Drybulb Temperature', 'Zone Drybulb Temperature', 'Delta Outdoor and Zone Temperature', 'Sensible Cooling Load', and 'Sensible Heating Load'. These are explained in detail below. All of these options have several things in common. They are essentially hard-coded. There is no support for a general method. The interpolation scheme is based on some variable that might reasonably be expected to correlate with gradient changes. This variable's current value is used to continually adjust the value of the vertical gradient for room air temperature.

- **1-Outside temperature.** This option directs the program to interpolate between upper and lower values of the vertical gradient based on the outdoor air temperature. If the outdoor temperature exceeds the upper limit set under the **Upper Conditions** header, then the gradient entered in the **Upper Conditions > Temperature gradient** field is used. Similarly if the outdoor air temperature is below the value set in the **Lower Conditions > Temperature** field, then the gradient entered in the **Lower Conditions > Temperature gradient** is used. For outdoor temperatures that lie between the upper and lower bounds, the gradient is determined by linear interpolation between the two.
- **2-Inside temperature.** This option directs the program to interpolate between upper and lower values of the vertical gradient based on the mean zone air temperature. If the mean zone air temperature exceeds the upper limit set under the **Upper Conditions** header, then the gradient entered in the **Upper Conditions > Temperature gradient** field is used. Similarly if the mean zone air temperature is below the value set in the **Lower Conditions > Temperature** field, then the gradient entered in the **Lower Conditions > Temperature gradient** is used. For mean zone air temperatures that lie between the upper and lower bounds, the gradient is determined by linear interpolation between the two.
- **3-Inside outside Delta T.** This option directs the program to interpolate between upper and lower values of the vertical gradient based on the difference between the outdoor environment and the mean zone air temperature. If the temperature difference exceeds the upper limit set under the **Upper Conditions** header, then the gradient entered in the **Upper Conditions > Temperature gradient** field is used. Similarly if the temperature difference is below the **Lower Conditions > Temperature** field, then the gradient entered in the **Lower Conditions > Temperature gradient** is used. For temperature differences that lie between the upper and lower bounds, the gradient is determined by linear interpolation between the two.
- **4-Sensible cooling load.** This option directs the program to interpolate between upper and lower values of the vertical gradient based on the sensible cooling load. If the cooling load exceeds the upper limit set under the **Upper Conditions** header, then the gradient entered in the **Upper Conditions > Temperature gradient** field is used. Similarly if the cooling load is below the value set in the **Lower Conditions > Heat rate** field, then the gradient entered in the **Lower Conditions > Temperature gradient** is used. For cooling loads that lie between the upper and lower bounds, the gradient is determined by linear interpolation between the two.
- **5-Sensible heating load.** This option directs the program to interpolate between upper and lower values of the vertical gradient based on the sensible heating load. If the heating load exceeds the upper limit set under the **Upper Conditions** header, then the gradient entered in the **Upper Conditions > Temperature gradient** field is used. Similarly if the heating load is below the value set in the **Lower Conditions > Heat rate** field, then the gradient entered in the **Lower Conditions > Temperature gradient** is used. For heating loads that lie between the upper and lower bounds, the gradient is determined by linear interpolation between the two.

Upper Conditions

Temperature

This field is used to enter the upper bound on temperature values (in °C or °F). It is required for the interpolation modes based on temperature.

Heat rate

This field is used to enter the upper bound on heat rate values. It is required for the interpolation modes based on load. Both heating and cooling loads are entered as positive numbers (in W).

Temperature gradient

This field specifies the gradient, or slope, of temperature changes in the vertical direction (in °C/m or °F/ft) when the selected temperature or heat load is at or above the upper condition.

Lower Conditions

Temperature

This field is used to enter the lower bound on temperature values (in °C or °F). It is required for the interpolation modes based on temperature.

Heat rate

This field is used to enter the lower bound on heat rate values. It is required for the interpolation modes based on load. Both heating and cooling loads are entered as positive numbers (in W).

Temperature gradient

This field specifies the gradient, or slope, of temperature changes in the vertical direction (in °C/m or °F/ft) when the selected temperature or heat load is at or below the lower condition.

Operation.

Operation schedule

The operation schedule defines the times when air temperature distribution is to be calculated. When the value of the schedule is 1, the distribution calculations take place, when the value is 0, the zone air temperature is fully mixed.

Heights

Thermostat height

Thermostat height specifies the distance above the floor where the thermostat for HVAC control is situated. This height is used by the model to determine the thermostat temperature relative to the mean air temperature by applying the gradient.

Return air height

Return air height specifies the distance above the floor where the air leaves the zone and returns to the air system. This height is used by the model to determine the return air temperature relative to the mean air temperature by applying the gradient.

Range of applicability

The air temperature distribution within the zone is used as follows:

1. To calculate boundary conditions for conduction through surfaces (walls, roofs, glazing etc).
2. To include the effects of extracting relatively warmer air at the ceiling level when using Detailed HVAC.
3. To include the vertical position of the thermostat sensor for HVAC control.

How It Works

Changing the Distribution mode to **2-Dynamic gradient** reveals the air temperature distribution data. By default the Interpolation mode is **3-Inside-outside DeltaT**. This means that the temperature gradient within the space is modelled as varying according to the inside-outside temperature difference. The Upper conditions define the most extreme temperature gradient, typically under design conditions. By default, the upper temperature difference is 10 °C and the corresponding temperature gradient is 4 °C/m. This means that when the temperature difference between inside and outside is 10 °C or more the internal temperature gradient is 4 °C/m. The Lower conditions data shows that when the temperature difference between inside and outside is zero there is no temperature gradient, i.e. the air is fully mixed. EnergyPlus calculates the temperature gradient for other inside-outside temperature differences by linear interpolation. The calculation is similar for the other interpolation modes but is instead based on outside temperature, inside temperature, heating or cooling load.

Limitations

The air temperature distribution mechanism does not include:

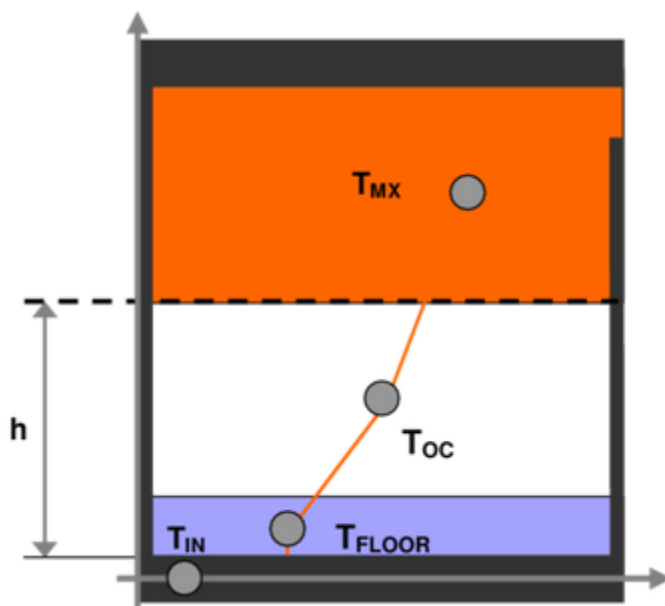
1. The effects of extracting relatively warmer air at the ceiling level when using Scheduled and Calculated natural ventilation.
2. The vertical position of the sensor for ventilation control.

Air Temperature Distribution - Three Node Displacement Ventilation

[HVAC tab in model data](#)

This model is applicable to spaces that are served by a low velocity floor-level displacement ventilation air distribution system. Furthermore, the dominant sources of heat gain should be from people and other localized sources located in the occupied part of the room. The model should be used with caution in zones which have large heat gains or losses through exterior walls or windows or which have considerable direct solar gain. The model predicts three temperatures in the room:

- A foot level temperature (T_{FLOOR}). The floor region is 0.2 meters deep and T_{FLOOR} represents the temperature at the mid-point of the region.
- An occupied sub-zone temperature (T_{OC}), representing the temperature in the region between the floor layer and the upper, mixed layer.
- An upper node representing the mixed-layer/outflow temperature (T_{MX}) essential for overall energy budget calculations and for modelling comfort effects of the upper layer temperature.



The following fields are used to define the **Three node displacement ventilation** room air temperature distribution.

Gain distribution schedule

Select the schedule defining the fractions of the convective portion of the internal gains in the occupied subzone that remain in the occupied subzone. The remainder of the convective portion of the internal gains in the occupied subzone enters the plumes and is carried to the upper subzone. The types of internal gains that are assumed to be located in the occupied subzone are:

- people
- task lights
- electric equipment
- gas equipment
- hot water equipment
- steam equipment
- other equipment

- baseboard heat

Types of internal gains that are assumed to be in the upper subzone are:

- general lights
- tubular daylighting devices
- high temperature radiant heaters

The schedule values should be between 0 and 1. A value of 1 means that all the convection gains from equipment, task lights and people are dispersed in the lower occupied subzone. Conversely a value of 0 puts all the lower subzone convective gains into the plumes rising into the upper well-mixed subzone.

Number of plumes per occupant

This field specifies number of plumes per occupant. Plumes are associated with localized sources of convective heat gain from people and equipment. For example, a value of 2.0 would be used if each occupant has a computer that generates a separate plume that does not merge with the plume from the occupant in the lower, occupied, subzone.

Thermostat height

This field is the height (in m or ft) of the thermostat/temperature control sensor above the floor.

Comfort height

The height (in m or ft) above the floor at which air temperature is calculated for comfort purposes. The air temperature at this height is used in calculating the available measures of comfort: Fanger, Pierce or KSU. The default is 1.1m.

Advanced

Temperature difference threshold for reporting

This field specifies a minimum temperature difference between the upper mixed subzone and the occupied subzone that will be used to trigger whether or not the displacement ventilation auxiliary outputs will be calculated. These outputs are Room Air Zone Transition Height, Room Air Zone Recommended Minimum Flow Fraction, Room Air Zone Average Temperature Gradient and Room Air Zone Maximum Temperature Gradient. They are set to negative values when the temperature difference is less than the threshold and the output Room Air Zone Is Mixed Status is set to 1.

The value should be greater than or equal to zero and is in units of Delta °C or Delta °F. The default value is 0.4°C.

Air Temperature Distribution - Under Floor Interior

[HVAC tab in model data](#)

This model is applicable to interior spaces that are served by an underfloor air distribution (UFAD) system. The dominant sources of heat gain should be from people, equipment, and other localized sources located in the occupied part of the room. The model should be used with caution in zones which have large heat gains or losses through exterior walls or windows or which have considerable direct solar gain.

The model predicts two temperatures in the room:

- An occupied subzone temperature (T_{oc}), representing the temperature in the region between the floor and the boundary of the upper subzone.
- An upper subzone temperature (T_{mx}) essential for overall energy budget calculations and for modelling comfort effects of the upper layer temperature.

The following fields are used to define the [Under Floor Air Distribution \(UFAD\) Interior](#) data.

Number of diffusers

The total number of diffusers in this zone. This field can be allowed to **Autocalculate** (in which case it is set to the design occupancy level; i.e., number of people). If the design occupancy is low or zero but there are still heat sources that could generate buoyancy driven plumes, the user should input a value based on the design supply air flow rate of the zone and the design flow rate of an individual diffuser. In the absence of any other information, divide the zone area by 100 ft². The default for this field is **Autocalculate**.

Power per plume

The power in watts incorporated in a buoyancy driven thermal plume. Normally we assume all the loads of a workstation create a single plume so that this represents the convective heat gain from a workstation – 1 person, 1 computer terminal, plus any task lighting. A typical value would be 220 W. However, the model assumes an “equivalent” plume derived from the zone extraction rate normalized to the number of workstations/occupants. This field should normally be allowed to default – the program will calculate a value based upon the occupancy and the extraction rate. The default is **Autocalculate**.

Design effective area of diffuser

This is the design air flow opening area in square meters of a single diffuser. The default value depends on the diffuser type. For swirl diffusers it is 0.075 m², for variable area diffusers 0.035 m², for horizontal swirl diffusers 0.006 m², and for linear bar grilles 0.03 m². The default is **Autocalculate**.

Diffuser slot angle from vertical

This input describes the angle at which air emerges from the diffusers. It should be the angle in degrees between the vertical and the diffuser slots. The default is **Autocalculate** in which case the setting depends on the diffuser type: for swirl diffusers it is 28°, for variable area diffusers 45°, for DV diffusers 73°, and for linear bar grilles 15°.

Diffuser type

The choices for this alpha field are:

- **1-Swirl** - a fixed area diffuser
- **2-Variable area** maintain an approximately constant exit velocity
- **3-Horizontal swirl** - fixed area diffuser
- **4-Linear bar grille** - fixed area diffusers normally used in exterior zones.
- **5-Custom** - the coefficients A to E should be entered to define the performance of the diffuser (below) rather than let the program set the coefficients based on diffuser type.

The swirl and displacement diffusers are fixed area diffusers. The variable area diffusers maintain an approximately constant exit velocity. Linear bar grilles are normally used in exterior zones and are fixed area diffusers. Custom is used to signify that the user intends to input the coefficients A – E (see below) rather than let the program set the coefficients based on diffuser type. The default is **1-Swirl**.

Custom Diffuser Coefficients

When the diffuser type is set to **5-Custom** the coefficients A to E should be entered to define the performance of the diffuser. The coefficients are used in the equation:

$$K_c = A * \text{Gamma}^{**B} + C + D * \text{Gamma} + E * \text{Gamma}^{**2}.$$

Gamma is a variable that characterizes the amount of stratification in a UFAD zone. K_c is the fraction of the total internal convective heat gain that is assigned to the lower (occupied) subzone. The coefficients in the formula are defaulted based upon diffuser type.

Coefficient A

The coefficient A in the above formula.

Coefficient B

The coefficient B in the above formula.

Coefficient C

The coefficient C in the above formula.

Coefficient D

The coefficient D in the above formula.

Coefficient E

The coefficient E in the above formula.

Height Settings**Thermostat height**

This field is the height (in m or ft) of the thermostat/temperature control sensor above the floor. The default is 1.2m.

Comfort height

The height (in m or ft) above the floor at which air temperature is calculated for comfort purposes. The air temperature at this height is used in calculating the available measures of comfort: Fanger, Pierce or KSU. The default is 1.1m.

Set transition height

If you would like to define the Transition height (below) then check this check box.

Transition height

An optional field to allow the transition height (in m or ft above floor) to be defined rather than have the program calculate it. The default is 1.7m.

Advanced**Temperature difference threshold for reporting**

This field specifies a minimum temperature difference between the upper subzone and the occupied subzone that will be used to trigger whether or not the UFAD auxiliary outputs will be calculated. These outputs are Room Air Zone Transition Height and Room Air Zone Average Temperature Gradient. They are set to zero when the temperature difference is less than the threshold and the output Room Air Zone Is Mixed Status is set to 1.

The value should be greater than or equal to zero and is in units of Delta °C or Delta °F. The default value is 0.4°C.

Air Temperature Distribution - Under Floor Exterior**[HVAC tab in model data](#)**

This model is applicable to exterior spaces that are served by an underfloor air distribution system. The dominant sources of heat gain should be from people, equipment, and other localized sources located in the occupied part of the room, as well as convective gain coming from a warm window. The model predicts two temperatures in the room:

- An occupied subzone temperature (T_{oc}), representing the temperature in the region between the floor and the boundary of the upper subzone..
- An upper subzone temperature (T_{mx}) essential for overall energy budget calculations and for modeling comfort effects of the upper layer temperature.

The following fields are used to define the **Under Floor Air Distribution (UFAD) Exterior** data.

Number of diffusers

The total number of diffusers in this zone. This field can allowed to **Autocalculate** (in which case it is set to the design occupancy level; i.e., number of people). If the design occupancy is low or zero but there are still heat sources that could generate buoyancy driven plumes, the user should input a value based on the design supply air flow rate of the zone and the design flow rate of an individual diffuser. In the absence of any other information, divide the zone area by 100 ft². The default for this field is **Autocalculate**.

Power per plume

The power in watts incorporated in a buoyancy driven thermal plume. Normally we assume all the loads of a workstation create a single plume so that this represents the convective heat gain from a workstation – 1 person, 1 computer terminal, plus any task lighting. A typical value would be 220 W. However, the model assumes an “equivalent” plume derived from the zone extraction rate normalized to the number of workstations/occupants. This field should normally be allowed to default – the program will calculate a value based upon the occupancy and the extraction rate. The default is **Autocalculate**.

Design effective area of diffuser

This is the design air flow opening area in square meters of a single diffuser. The default value depends on the diffuser type. For swirl diffusers it is 0.075 m², for variable area diffusers 0.035 m², for horizontal swirl diffusers 0.006 m², and for linear bar grilles 0.03 m². The default is **Autocalculate**..

Diffuser slot angle from vertical

This input describes the angle at which air emerges from the diffusers. It should be the angle in degrees between the vertical and the diffuser slots. The default is **Autocalculate** in which case the setting depends on the diffuser type: for swirl diffusers it is 28°, for variable area diffusers 45°, for DV diffusers 73°, and for linear bar grilles 15°.

Diffuser type

The choices for this alpha field are:

- **1-Swirl** - a fixed area diffuser
- **2-Variable area** maintain an approximately constant exit velocity
- **3-Horizontal swirl** - fixed area diffuser
- **4-Linear bar grille** - fixed area diffusers normally used in exterior zones.
- **5-Custom** - the coefficients A to E should be entered to define the performance of the diffuser (below) rather than let the program set the coefficients based on diffuser type.

The swirl and displacement diffusers are fixed area diffusers. The variable area diffusers maintain an approximately constant exit velocity. Linear bar grilles are normally used in exterior zones and are fixed area diffusers. Custom is used to signify that the user intends to input the coefficients A – E (see below) rather than let the program set the coefficients based on diffuser type. The default is **1-Swirl**.

Custom Diffuser Coefficients

When the diffuser type is set to **5-Custom** the coefficients A to E should be entered to define the performance of the diffuser. The coefficients are used in the equation:

$$K_c = A * \text{Gamma}^{**}B + C + D * \text{Gamma} + E * \text{Gamma}^{**}2.$$

Gamma is a variable that characterizes the amount of stratification in a UFAD zone. K_c is the fraction of the total internal convective heat gain that is assigned to the lower (occupied) subzone. The coefficients in the formula are defaulted based upon diffuser type.

Coefficient A

The coefficient A in the above formula.

Coefficient B

The coefficient B in the above formula.

Coefficient C

The coefficient C in the above formula.

Coefficient D

The coefficient D in the above formula.

Coefficient E

The coefficient E in the above formula.

Height Settings

Thermostat height

This field is the height (in m or ft) of the thermostat/temperature control sensor above the floor. The default is 1.2m.

Comfort height

The height (in m or ft) above the floor at which air temperature is calculated for comfort purposes. The air temperature at this height is used in calculating the available measures of comfort: Fanger, Pierce or KSU. The default is 1.1m.

Set transition height

If you would like to define the Transition height (below) then check this check box.

Transition height

An optional field to allow the transition height (in m or ft above floor) to be defined rather than have the program calculate it. The default is 1.7m.

Advanced

Temperature difference threshold for reporting

This field specifies a minimum temperature difference between the upper subzone and the occupied subzone that will be used to trigger whether or not the UFAD auxiliary outputs will be calculated. These outputs are Room Air Zone Transition Height and Room Air Zone Average Temperature Gradient. They are set to zero when the temperature difference is less than the threshold and the output Room Air Zone Is Mixed Status is set to 1.

The value should be greater than or equal to zero and is in units of Delta °C or Delta °F. The default value is 0.4°C.

HVAC Cost

Air temperature distribution tab on [HVAC Templates](#) Dialog.

The HVAC cost is calculated zone by zone by multiplying the zone floor area by the cost per floor area (GIFA) for the zone on the HVAC tab. This data will normally be loaded from the HVAC template to ensure that HVAC performance and cost data are consistent.

This cost should include any contributions to the total cost of the building HVAC system from plant components and not only the components within the particular zone.

Note: This data is used for both Simple and Detailed HVAC for costing purposes.

Tip: The [Cost data](#) help topic provides an overview of the places in DesignBuilder where costs are entered.

Cost per area

Enter the total cost per floor area (GIFA) for the HVAC system. This data is used when the [HVAC calculation method](#) is set to **1-Cost per area**.

HVAC cost per cooling load

Enter the indicative cost of the cooling element of this HVAC system type (in GBP/kW). This data is used when the [HVAC calculation method](#) is set to **2-Cost per capacity** option and the cost for each zone is calculated based on its HVAC system capacity.

HVAC cost per heating load

Enter the indicative cost of the heating element of this HVAC system type (in GBP/kW). This data is used when the [HVAC calculation method](#) is set to **2-Cost per capacity** option and the cost for each zone is calculated based on its HVAC system capacity.

Environmental Impact Factors

[HVAC tab in model data](#)

The Environmental Impact Factors data contains additional information that is not individually specified in the [Fuel emission factors](#) data. Source to Site conversion factors for District Heating, District Cooling and Steam are entered in this object.

This data can only be accessed when using the [Detailed HVAC](#) model option. It is available at building level only.

District heating efficiency

The District heating efficiency value is used to convert the district (purchased) heating energy to Natural Gas for the environmental impact calculation. The efficiency is a number between 0 and 1 and divided into the district heating energy and the default is 0.3.

District cooling COP

The District cooling COP value is used to convert the district (purchased) cooling energy to electricity for the environmental impact calculation. The Coefficient of Performance (COP) is a number greater than 0 and divided into the purchased cooling energy with a default of 3.0.

Steam conversion efficiency

The Steam conversion efficiency is used to convert the Steam usage to Natural Gas for the environmental impact calculation. The efficiency is a number between 0 and 1 and divided into the purchased heating energy and the default is 0.25.

Total carbon equivalent emission factor from N2O

The Intergovernmental Panel on Climate Change has studied the effects on the relative radiative forcing effects of various greenhouse gases. This effect, called Global Warming Potential (GWP), is described in terms of the Carbon Equivalent of a particular greenhouse gas. N2O (nitrous oxide) can be produced by some Fuel Types and has a default of carbon equivalent emission factor of 80.7272 kg C/kg N2O.

Total carbon equivalent emission factor from CH4

The Intergovernmental Panel on Climate Change has studied the effects on the relative radiative forcing effects of various greenhouse gases. This effect, called Global Warming Potential (GWP), is described in terms of the Carbon Equivalent of a particular greenhouse gas. CH4 (methane) can be produced by all Fuel Types and has a default carbon equivalent emission factor of 6.2727 kg C/kg CH4.

Total carbon equivalent emission factor from CO2

The Intergovernmental Panel on Climate Change has studied the effects on the relative radiative forcing effects of various greenhouse gases. This effect, called Global Warming Potential (GWP), is described in terms of the Carbon Equivalent of a particular greenhouse gas. CO2 (carbon dioxide) can be produced by all Fuel Types and has a default carbon equivalent emission factor of 0.2727 kg C/kg CO2.

On Site Electricity Generation

Generation tab in model data

EnergyPlus provides a range of different on site electricity generation options and these are coordinated during the simulation through use of one or more [Electric load centres](#). These define how the various generation components are operated and coordinated together.

DesignBuilder currently supports various types of electric generators PV, BIPV, Wind, Internal combustion engine and micro turbine.

Settings for electricity generation are made at Building level on the **Generation** tab under the **On Site Electricity Generation** header.

Include electric load centres

Check the **Include electric load centres** checkbox to model electric generation.

Number of electric load centres

Set the number of electric load centres to be modelled. Choose a value from 1 to 30.

Load centre 1, 2, 3 etc

Create and select a load centre for each of the load centres from 1 to the **Number of electric load centres** defined.

Note: The order of the Electric load centre objects selected on the Generation tab is significant and is used to structure how generators are dispatched, the first load centres and generators being managed before the later ones.

Note: Each Load centre can only be selected once. If you have multiple similar Load centres, each must be defined separately. For example, this configuration is allowed:

On Site Electricity Generation

☒ Include electric load centres

Number of electric load centres

Load centre 1
 PV DC with inverter 01

Load centre 2
 PV DC with inverter 02

Load centre 3
 PV DC with inverter 03

Load centre 4
 PV DC with inverter 04

Whereas this configuration with the same Load centre selected 4 times is not allowed:

On Site Electricity Generation

☒ Include electric load centres

Number of electric load centres

Load centre 1
 PV DC with inverter 01

Load centre 2
 PV DC with inverter 01

Load centre 3
 PV DC with inverter 01

Load centre 4
 PV DC with inverter 01

See also:

- "How to Model Photovoltaic (PV) Systems" on page 523
- "How to Model Wind Turbine Systems" on page 532
- "Electric Load Centre" on page 526

How To Model Photovoltaic (PV) Systems

How To Model A Standalone PV Solar System - Summary



You can include PV panels in your model by following the instructions below.

1. Position and size PV panels by following instructions in the [Adding Solar Collectors](#) topic.
2. To access the properties of the PV panel navigate to the solar collector object by double-clicking on the graphical object from building level or single-click on the solar collector item in the Navigator.
3. Once at the Solar collector object, define the performance and other panel properties on the **Constructions** tab under the [Solar Collector](#) header.
4. Define the electric side of the PV system at building level on the **Generation** tab under the [On Site Electricity Generation](#) header:
 - a. Check the **Include electric load centres** check box.
 - b. Add an [Electric load centre](#) and edit the details of the system.
 - c. Select one of [the DC buss types](#).
 - d. Select the [Inverter](#) type.
 - e. If using electrical storage define the [Storage](#) type.
 - f. Select the [building solar collector](#) on the [Generator list](#) tab.
 - g. Make sure that, having defined a new [Electric load centre](#), the correct one is selected on the **Generation** tab.
5. When the simulation is finished you can view solar PV contributions on the **Analysis** and **Summary** tabs:

Electric Loads Satisfied

	Electricity [kWh]	Percent Electricity [%]
Fuel-Fired Power Generation	0.00	0.00
High Temperature Geothermal*	0.00	0.00
Photovoltaic Power	88.59	0.06
Wind Power	0.00	0.00
Net Decrease in On-Site Storage	0.00	0.00
Total On-Site Electric Sources	88.59	0.06
Electricity Coming From Utility	156457.01	99.94
Surplus Electricity Going To Utility	0.00	0.00
Net Electricity From Utility	156457.01	99.94
Total On-Site and Utility Electric Sources	156545.60	100.00
Total Electricity End Uses	156545.60	100.00

How To Model A Building Integrated PV (BIPV) Solar System - Summary



You can include BIPV systems in your model by following the instructions below.

1. Define one or more constructions and/or glazing systems using the **Building Integrated Photovoltaics** category (for constructions) or the **Glazing Integrated Photovoltaics** category for glazing.
2. Define the properties of the PV system on the [Construction/glazing](#) dialog under the definition of the outermost layer.
3. Apply the BIPV systems by selecting the corresponding BIPV construction and/or glazing components on the relevant surfaces. Note that in larger models, the "Model Data Grid View (MDGV)" on page 656 is an efficient way to apply BIPV constructions to particular façades based on surface slope, orientation, tag etc.
4. Define the electric side of the PV system at building level on the **Generation** tab under the [On Site Electricity Generation](#) header:
 - a. Check the **Include electric load centres** check box.
 - b. Add an [Electric load centre](#) and edit the details of the system.
 - c. Select one of [the DC buss types](#).
 - d. Select the [Inverter](#) type.
 - e. If using electrical storage define the [Storage](#) type.
 - f. On the [Generator list](#) tab, select the DC generator type to be one of **2-Construction with BIPV** or **3-Glazing with BIPV**
 - g. Select under that the BIPV construction or glazing covered by that generator. Create one generator per BIPV construction and glazing used in the model. Any BIPV constructions not selected here will not generate electricity.
 - h. Make sure that, having defined a new [Electric load centre](#), the correct one is selected on the Generation tab.
5. When the simulation is finished you can view solar PV contributions on the Analysis and Summary tabs:

Electric Loads Satisfied

	Electricity [kWh]	Percent Electricity [%]
Fuel-Fired Power Generation	0.00	0.00
High Temperature Geothermal*	0.00	0.00
Photovoltaic Power	88.59	0.06
Wind Power	0.00	0.00
Net Decrease in On-Site Storage	0.00	0.00
Total On-Site Electric Sources	88.59	0.06
Electricity Coming From Utility	156457.01	99.94
Surplus Electricity Going To Utility	0.00	0.00
Net Electricity From Utility	156457.01	99.94
Total On-Site and Utility Electric Sources	156545.60	100.00
Total Electricity End Uses	156545.60	100.00

Electric Load Centre

Electric load centres are components used to include on-site electricity generators in a simulation. The electric load centre dispatches generators according to operation schemes and tracks and reports the amount of electricity generated and purchased. When using on-site generators, the program provides a “net” report where the total electricity used is reduced by the amount generated on site. Electrical demand tracking is done by the internal or custom meters used by EnergyPlus for reporting. The dispatching of different generators is based on expectations based on their nominal/rated electric power output. If the current conditions are such that the generator model determines that generation was higher or lower, then the results of the dispatch may differ from expectations.

Up to 5 different Electric load centre components can be included in simulations by making selections on the [On Site Electricity Generation](#) tab. A great deal of flexibility is available by mixing different load centres and operating schemes. If multiple load centres are used, the supervisory control will dispatch generators sequentially across the load centres.

Electric load centres effectively serve as a “load” to the generators and “supply” power to the rest of the building. The internal meters used by EnergyPlus for reporting do all of the demand tracking. The electric load centre manager sums all of the building and system electric loads and provides operating schemes for the generators. What is not provided by the on-site generation equipment, and electric storage units if specified, is met by (purchasing) off-site electricity.

The electricity produced from generators such as photovoltaic arrays, wind turbines and cogeneration is reported in the electricity produced output variable and will reduce the demand that the generators will try to meet for that timestep.

Load centres can include both power conditioning and [storage](#). Separate [inverter](#) models are used to condition DC power from [photovoltaic panels](#) and [BIPV systems](#) into AC power for the building and utility. Load centres serving PV need to be specified with a direct current (d.c.) buss. The other generators may have inverters inside the devices but these are already embedded in the generator models.

General

Name

This alpha field contains the identifying name for the electric load centre.

Generator operation scheme type

Select the type of operating scheme for the generator set. The various operating schemes affect how loads are dispatched to the generators, in effect telling the generators whether or not to run and requesting power levels. The available operating schemes are:

- **1-Base load** operates the generators at their rated (requested) electric power output when the generator is scheduled on. This scheme requests all generators scheduled on (available) to operate, even if the amount of electric power generated exceeds the total facility electric power demand.
- **2-Demand limit** - limits the amount of purchased electrical from the utility to an amount specified. The Demand limit scheme tries to have the generators meet all of the demand above the purchased electric limit.
- **3-Track electrical** tries to have the generators meet all of the electrical demand for the building.
- **4-Track schedule** tries to have the generators meet all of the electrical demand determined in a user-defined schedule.
- **5-Follow thermal** is used for co-generation systems where the generators are to be run mainly to meet the thermal demand and any excess electricity can be exported to the grid.
- **6-Follow thermal limit electrical** is as above for **5-Follow thermal** but in this case generator output is restricted to a maximum of the building’s current electrical demand so that no electricity is exported.

The **2-Demand limit**, **3-Track electrical** and **4-Track schedule** schemes will sequentially load the available generators. All demand not met by available generator capacity will be met by purchased electrical. Therefore, if **2-Demand limit**, **3-Track electrical**, or **4-Track schedule** is used and the available generators are not able

to meet demand, then purchased electricity will be used to offset the difference. If a generator is needed in the simulation for a small load and it is less than the minimum part load ratio the generator will operate at the minimum part load ratio and the excess will either reduce demand or the excess energy will be available for returning to the electric grid.

The **5-Follow thermal** and **6-Follow thermal limit electrical** schemes are for heat due to co-generation and run the generators to meet the thermal demand. The **5-Follow thermal** scheme allows excess electrical generation to be exported to the grid, while the **6-Follow thermal limit electrical** scheme restricts generator output to a maximum of the building's current electrical demand so that no electricity is exported. The thermal demand is determined from the plant modelling and depends on the flow requested by other components on the demand side of the plant loop, the loop temperatures, and the loop temperature set point. The electric load centre converts the thermal load to an electrical load using a nominal ratio of the thermal to electrical power production for each generator. For these schemes, the generator needs to be connected to the supply side of a plant loop and components that request hot water need to be on the demand side of the plant loop. This is different than the usual configuration for electrical following schemes where the generator is put on the demand side and request flow for the purposes of cooling the generator. Therefore a switch from one of the electrical-based operating schemes to one of the thermal-based operating schemes requires a substantial change in plant topology.

If the load centre includes electrical storage, then the choice of operating schemes will also affect how storage is managed.

Note: The **5-Follow thermal** and **6-Follow thermal limit electrical** schemes will only work for generators that are placed on the supply side of a plant loop and are being run to satisfy thermal load. When generators are run to satisfy electrical load (**1-Base load**, **2-Demand limit**, **3-Track electrical** and **4-Track schedule** scheme types), they're located on the demand side and request flow rate for cooling purposes only.

Technical

For all operating schemes except **1-Base load**, a total electric load reduction target (or thermal load converted to electrical equivalent) is established for the load centre based on the specific operating scheme. The load centre then requests that its generators operate, one-by-one in the order specified, until the target is met or exceeded. Generators that are not scheduled as 'available' for the simulation time step are not called to operate. The requested power demand to be met by each generator is the smaller of the nominal 'rated' electric power output (as specified in the Generators object) or the remaining total electric load reduction target for the load centre. After each electric generator is requested to operate, the actual electric power delivered by the generator, which may be greater than or less than the requested amount due to inputs specified in the generator performance model, is used to update the remaining total electric power target for the other generators associated with this load centre.

Most of the operating schemes will sequentially load the available electric load centres and generators. EnergyPlus can accept multiple Electric load centres with different operating schemes. Because of this, there are two levels of reporting, one for the whole building and a second for each load centre. The whole-building results are managed with the internal meters for the entire model. The individual load-centre results are summed for those generators connected to a particular load centre. The total electricity purchased is reported both in power and energy units. This value is positive when the amount of energy is purchased from the utility. This value can be negative when the total electricity produced is greater than the facility electrical needs. The excess will either be available for storage or to sell back to the electric utility company.

Demand Limit Scheme Purchased Electric Demand Limit

The demand limit above which the generators will try and meet the entire electrical load on the building minus the photovoltaic array if available (in W).

Track schedule

When the **4-Track schedule** option is used then you should select a schedule containing values for the "demand" loads placed on the generator(s). The schedule values should be in Watts.

Electrical buss type

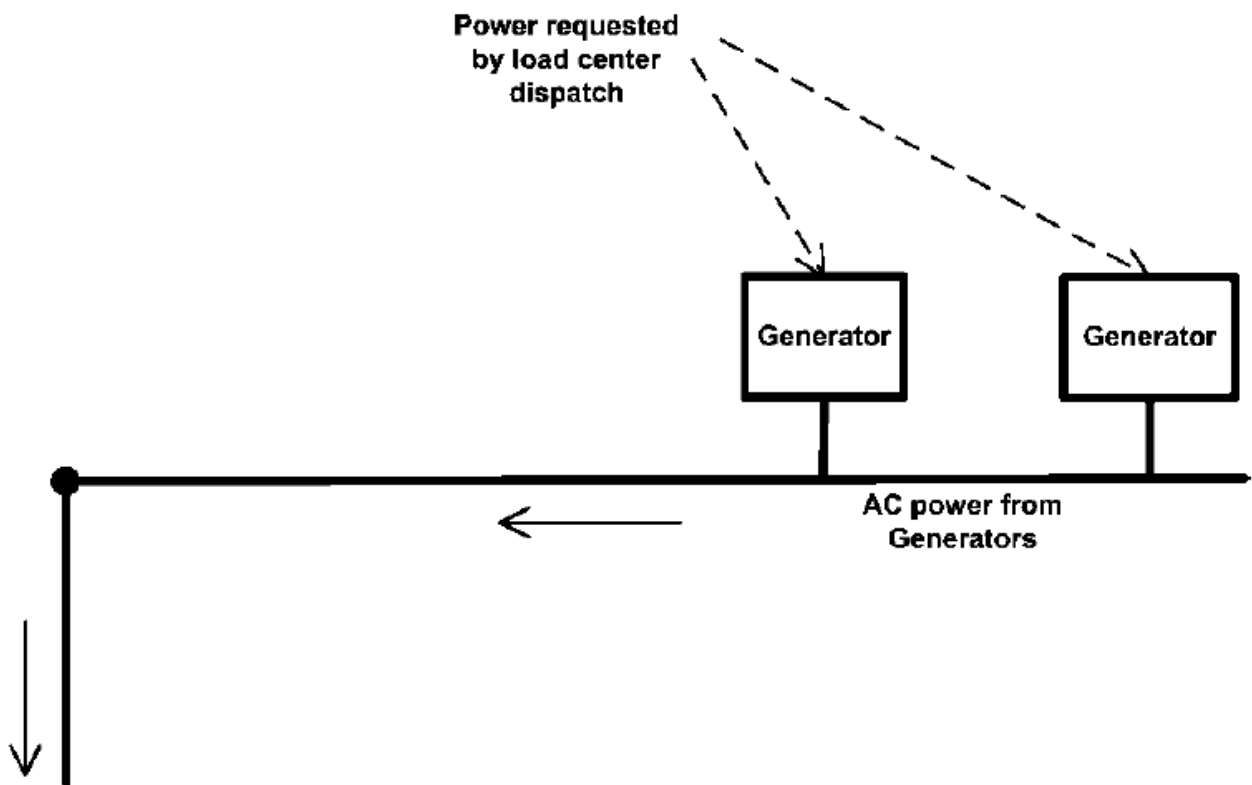
This alpha field is used to describe how the electric load centre is configured with respect to any power conditioning and/or storage equipment. There are five configurations for load centres available by using one of these keywords:

- [1-Alternating current](#)
- [2-Alternating current with storage](#)
- [3-Direct current with inverter](#)
- [4-Direct current with inverter DC storage](#)
- [5-Direct current with inverter AC storage](#)

All the generators connected to a specific load centre need to be of the same type (all AC or all DC). If the generators are DC, then an inverter is needed to convert the DC to AC.

Alternating Current

The most basic configuration is selected with the keyword “Alternating Current” for the Electrical Buss Type, shown in the following diagram.

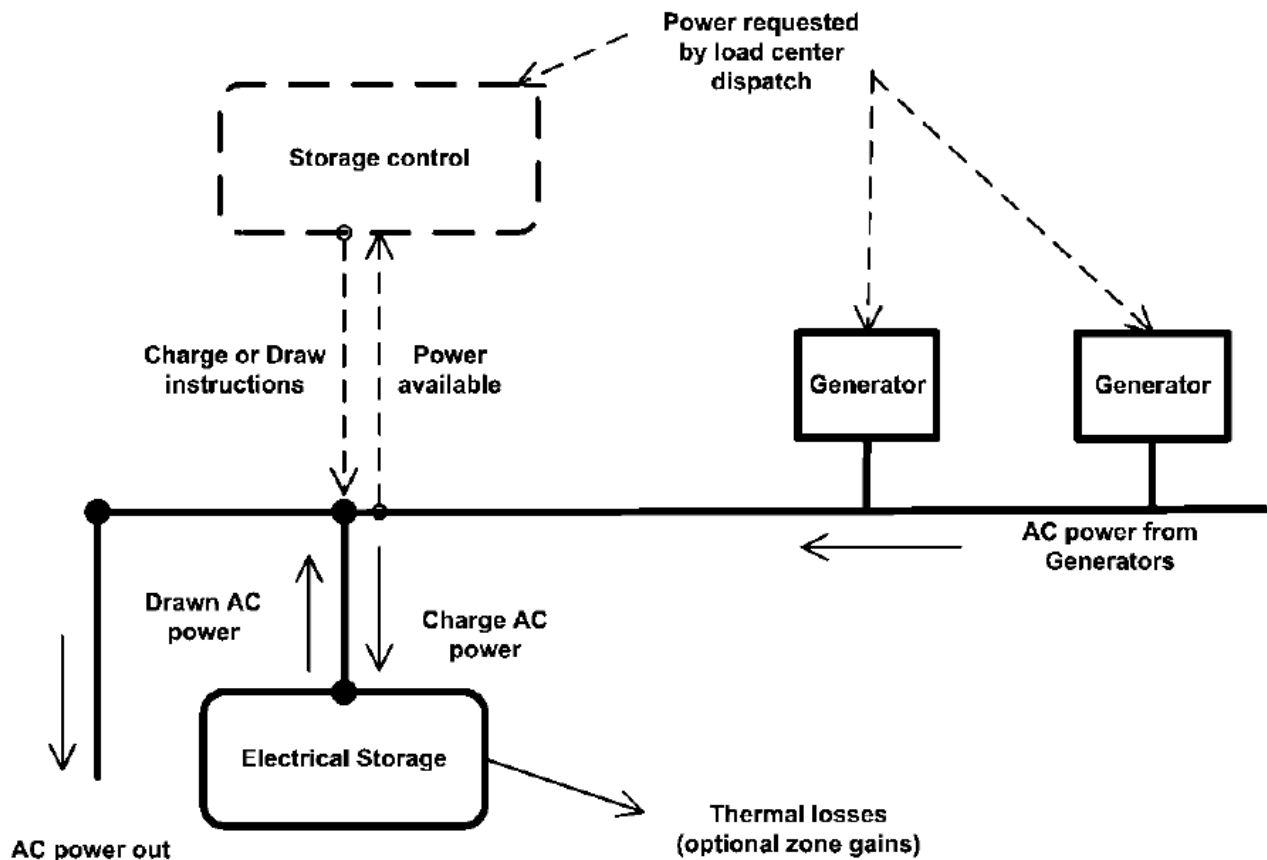


AC power out

The Alternating current load centres have AC generators with no storage and behave in the following way. All electric demand not met by the sum of the electrical power produced by the available generators will be met by purchased electricity. If a generator is needed in the simulation for a small load and the load is less than the generator's minimum part load ratio, the generator will operate at the minimum part load ratio and the excess will either reduce demand or the excess energy will be exported back to the electric utility company. The purchased electrical demand limit is the user input for the demand limit above which the generators will try and meet the entire electrical load on the building. It is possible to set up load centres with inconsistent or conflicting operating schemes, so users need to be careful.

Alternating current with storage

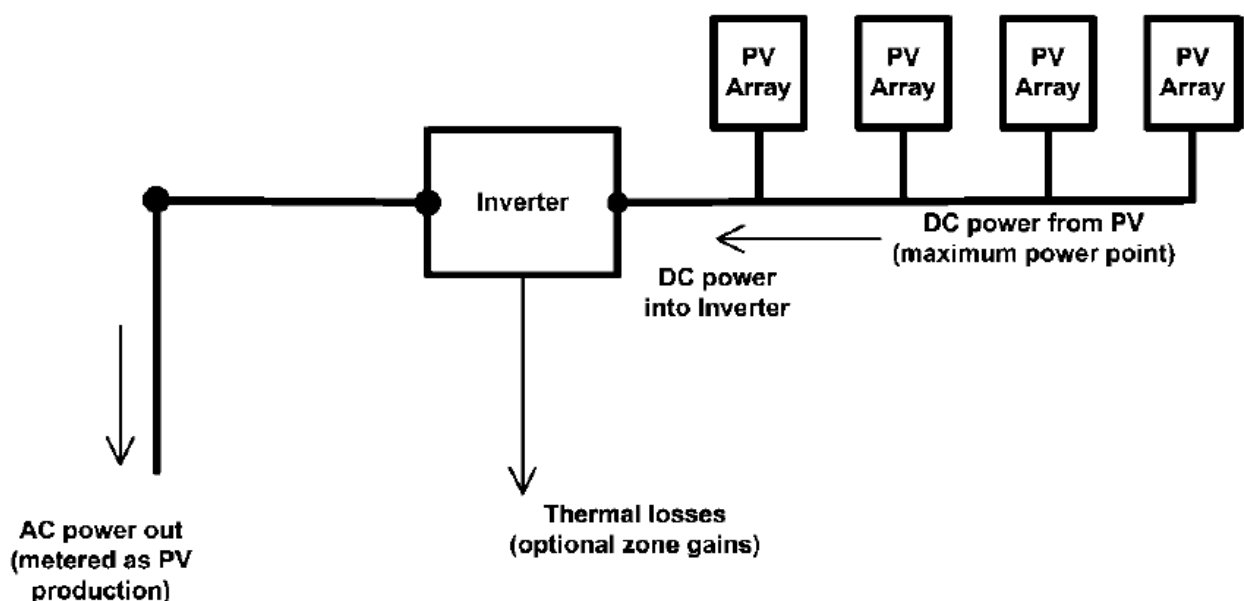
A configuration with AC generators with on-site electrical storage is selected using the **2-Alternating current with storage** Electrical buss type and is shown in the following diagram.



The **Alternating current with storage** load centres attempt to augment the generator electricity production so that the power requests are met. Storage control logic is discussed below under Electrical Storage.

Direct current with inverter

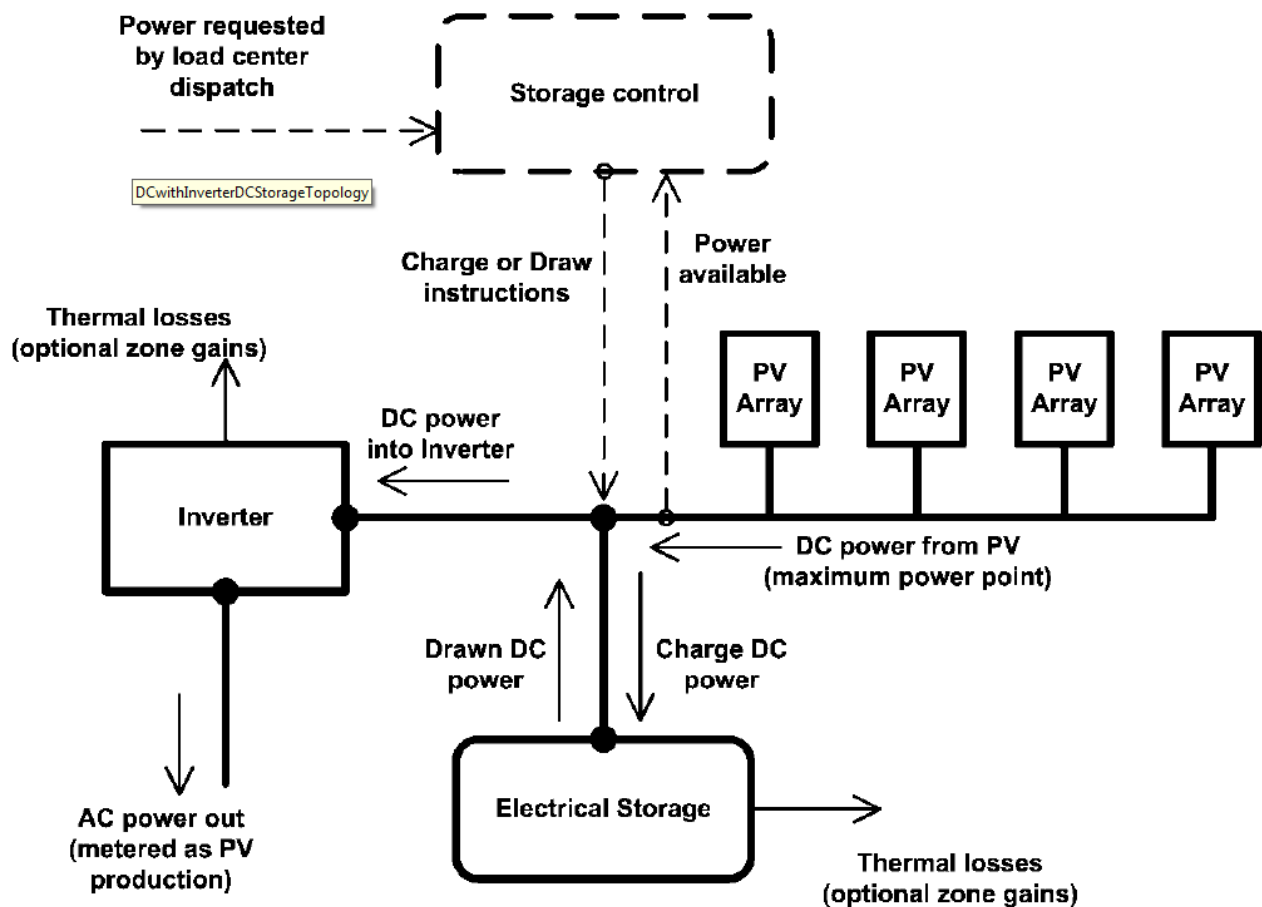
The basic configuration for photovoltaic generators is selected using the **3-Direct current with inverter** electrical buss type and is shown in the following diagram.



The **Direct current with inverter** load centres collect DC power from various PV arrays, run the DC power through an inverter and produce AC power. The PV arrays produce DC power based on the availability of sunshine and do not respond to load requests made by the electric load centre. The AC output from the inverter is what is recorded as electricity production.

Direct current with inverter DC storage

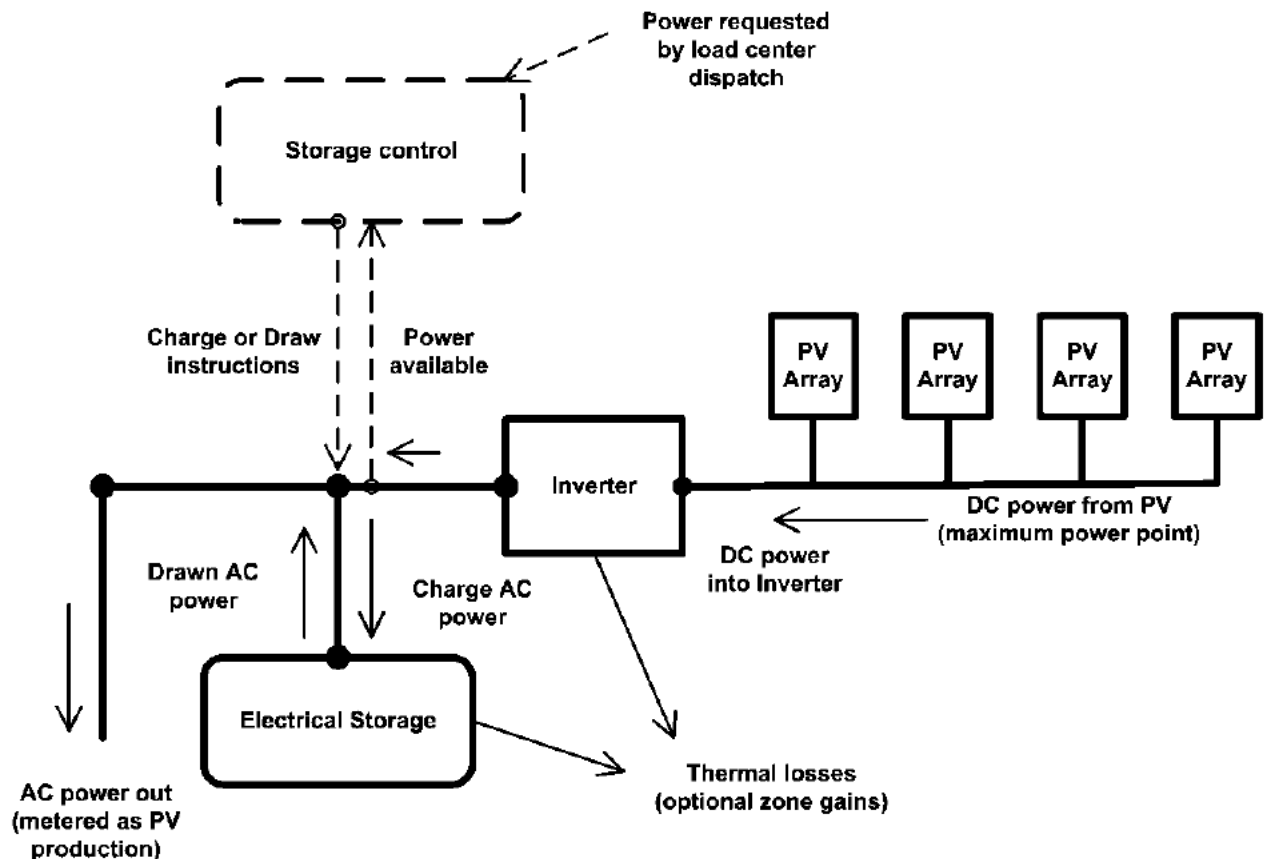
If the PV-based load centre is equipped with DC electrical storage that is connected before the inverter, then the buss type should be **4-Direct current with inverter DC Storage** and is shown in the following diagram.



The **Direct current with inverter DC storage** load centres charge or draw DC power to meet the requested electrical load.

Direct current with inverter AC storage

If the PV-based load centre is equipped with AC electrical storage that is connected after the inverter, then the buss type should be **5-Direct current with inverter AC storage** and is shown in the following diagram.



Inverter

The [inverter](#) selection option is only used if the [Electrical buss type](#) is set to **3-Direct current with inverter** and it allows you to select the inverter connected to this load centre (if any). There are three types of inverter models available:

- Simple,
- Look up table, or
- Function of Power.

Storage

The [storage](#) selection option is used to identify the electrical storage connected to this load centre (if any). It is only used if the Electrical Buss Type is set to **4-Direct current with inverter DC storage** or **5-Direct current with inverter AC storage**.

Cost

Distribution and electrical cost

Enter the cost of the electrical power distribution and any other related costs here.

The cost of the panels themselves is entered on the Constructions tab under the [Solar Collectors](#) header.

The cost entered here on the load centre will be included with the other renewable equipment costs under:

- Basic:
- UNIFORMAT II: F10 Special construction
- NRM1:

Note: In order for the cost of renewable systems such as solar PV and hot water panels and wind turbines to be included in the Construction cost report (and any cost-related KPIs used in an optimisation) those generators must be specified in an "Electric Load Centre" on page 526.

Generator List

On the Generator List tab you can define up to 30 generators per Electric load centre. Set the number of generators and select the generator to be used for each.

For DC Electric load centres the generators will be [PV solar collectors](#), [Construction BIPV](#) or [Glazing BIPV](#) systems.

For AC centres the generators will be [Wind turbines](#), or [Generators](#).

Tip: To model multiple PV panels without the need to draw all of the building panels you can reference the same panel multiple times on the Generator list tab. You should only do this if all of the panels have the same shading and if the shading cast by the panels does not impact on the building.

How To Model Wind Turbine Systems

How To Model A Wind Turbine System - Summary

You can include one or more [wind turbines](#) in your model by following the instructions below.

1. First define the electric side of the wind turbine system at building level on the **Generation** model data tab under the [On Site Electricity Generation](#) header:
 - a. Check the **Include electric load centres** check box.
 - b. Add an [Electric load centre](#) and edit the details of the system.
 - c. Use one of [the AC buss types](#).
 - d. If using storage define the [Storage](#) type.
 - e. Select the [wind turbine](#) on the [Generator list](#) tab.
 - f. Make sure that having defined a new [Electric load centre](#) that the correct one is selected on the **Generation** model data tab.
2. When the simulation is finished you can view wind turbine contributions in absolute and % terms on the Summary tab:

Electric Loads Satisfied

	Electricity [kBtu]	Percent Electricity [%]
Fuel-Fired Power Generation	0.00	0.00
High Temperature Geothermal*	0.00	0.00
Photovoltaic Power	0.00	0.00
Wind Power	13690.55	3.27
Net Decrease in On-Site Storage	0.00	0.00
Total On-Site Electric Sources	13690.55	3.27
Electricity Coming From Utility	408600.61	97.72
Surplus Electricity Going To Utility	4149.79	0.99
Net Electricity From Utility	404450.82	96.73
Total On-Site and Utility Electric Sources	418141.37	100.00
Total Electricity End Uses	418141.37	100.00

Wind Turbines

Wind turbines convert the kinetic energy of the surrounding air-stream into electricity. The EnergyPlus wind turbine model calculates the electrical power produced based on the characteristics of the turbine itself as well local environmental conditions such as wind speed and density of air at the height of the systems. The model obtains the weather information from the simulation weather data file and then determines the wind speed and air density at the specific height of the system. It also requires the user to input the annual average wind speed measured at the local site and the height of the measurement so that it factors in differences between the weather file wind data and the local wind data.

Tip: Wind turbines are included in the model by specifying them on the [Generator list](#) tab of the Electric load centre dialog when one of the a.c. [Buss types](#) is selected.

Technical

The model employs the general kinetic energy equation to calculate the performance characteristics of the horizontal axis wind turbine (HAWT) systems. It provides a simple approximation algorithm when the power coefficient, C_p , is available which represents the efficiency of the wind turbine in the wind power extraction from the ambient air stream. It also allows the user to input experimental constants so that the power coefficient can precisely be determined according to the characteristic of the airfoil of the system. As for the vertical axis wind turbine (VAWT) systems, it employs the general mathematical equations for straight-bladed Darrieus-type VAWT systems, which are common to VAWT systems. Various types of VAWT systems such as the Savonius-type and the curved-blade (or egg-beater) type may be simulated with this same model. It includes two different types of dynamic power control: Fixed Speed Fixed Pitch (FSFP) and Variable Speed Fixed Pitch (VSFP). Currently, it does not include an algorithm for modelling pitch control such as Fixed Speed Variable Pitch (FSVP) and Variable Speed Variable Pitch (VSVP). The model also has

the ability to account for transient losses associated with the power produced during dynamic control by a user-specified fraction.

The model does not include detailed algorithms for generators and inverters due to concerns for computational convergence, time, and usability. Instead, all conversion losses of these subsystems are included by applying a user-supplied total system efficiency to the maximum power extraction of the wind turbine. The field of the total system efficiency must be specified by the user.

Name

A unique user assigned name for a particular wind turbine system. Any reference to this unit by another object will use this name.

Availability schedule

The schedule selected here defines the times when the wind turbine system can run. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off and will not operate for the time period.

Rotor type

This field is the type of axis of the wind turbine. Select from 2 types

- **1-Horizontal axis** or
- **2-Vertical axis**.

A different algorithm is used in the calculation of the electrical power output of the wind turbine depending on the Rotor type.

Power control

This field is the type of rotor control for the wind turbine. This protects the system against the overloading for a system with no speed or pitch control and also to maximize the energy yield for the system. Four different control types are classified in the literature:

- **1-Fixed Speed Fixed Pitch** (FSFP),
- **2-Fixed Speed Variable Pitch** (FSVP),
- **3-Variable Speed Fixed Pitch** (VSFP), and
- **4-Variable Speed Variable Pitch** (VSVP).

Currently, FSFP and VSFP types can be modelled in EnergyPlus. The other two types will be modelled as VSFP. If the first FSFP control type is chosen, the model assumes the maximum power at a fixed rotor speed when the power output predicted is greater than the maximum until the rotor speed reaches the maximum wind speed (see next field). If one of the last three control options is chosen, the model assumes that the system produces a constant power at the rated wind speed when the wind speed is between the rated wind speed and cut-out wind speed. The default value is **3-Variable Speed Fixed Pitch** (VSFP).

Rated rotor speed

This field is the maximum rotational speed of the rotor at the rated power of the wind turbine in rev/min (revolution per minute). It is used to determine the tip speed ratio of the rotor and relative flow velocity incident on a single blade of the VAWT systems.

Rotor diameter

This field is the diameter of the rotor (in m or ft). Note that this field is not the height of the blade, but the diameter of the perpendicular circle from the vertical pole in the VAWT systems. It determines the swept area of the rotor of the HAWT systems and the chordal velocity of the VAWT systems.

Overall height

This field is the height of the hub of the HAWT system, or of the pole of the VAWT system (in m or ft). It is necessary to estimate local air density and the wind speed at this particular height where the wind turbine system is installed.

Number of blades

This field is the number of blades of the wind turbine. The azimuth angle of the rotor of the VAWT system is determined by dividing 360 degree by this field so that the model determines the chordal velocity component and the normal velocity component of the system. The default value is 3.

Rated power

This field is the nominal power output of the wind turbine system at the rated wind speed (in W or Btu/hr). Note that the maximum power of the system should be entered with no control, i.e. FSFP control type, can physically produce. Manufacturer data sometimes describes this as “peak power” or “rated capacity”. If the local wind speed is greater than the rated wind speed, the model assumes constant power output of this field.

Rated wind speed

This field is the wind speed that the wind turbine system indicates the peak in the power curve (in m/s or ft/min). The system produces the maximum power at this speed and the speed of the rotor is managed based on this wind speed.

Cut in wind speed

This field is the lowest wind speed where the wind turbine system can be operated (in m/s or ft/min). No power generation is achieved as long as the ambient wind speed is lower than this speed.

Cut out wind speed

This field is the greatest wind speed (in m/s or ft/min). When the wind speed exceeds this value, the wind turbine system needs to be stopped because of inefficiencies in the system. All systems that have either pitch or speed control must be stopped when the ambient wind speed exceeds this speed. Note that the user should input a wind speed above which physical damage to the system might be caused in the case of a FSFP system. It appears as “extreme/survival/design wind speed” in the literature. The system will be turned off when the ambient wind speed is over this speed.

Fraction system efficiency

This field is the overall system efficiency of the wind turbine system. It includes all the conversion losses as well as transient losses during the dynamic control when the ambient wind speed is between the rated wind speed and cut-out wind speed (see previous fields).

The user also has the ability to specify delivery losses from the system to the local area. If the user does not enter a fraction, the model assumes the default value of 0.835. Note that the fraction must be between zero and one.

Maximum tip speed ratio

This field is the maximum tip speed ratio between the rotor velocity and ambient wind velocity. The rotor speed varies with this ratio to maximize the power output when the rotor control types are variable speed ones. This field allows the user to adjust the power output from the particular system or to find the optimal tip speed ratio of the system. Optimal tip speed ratio is dependent on the number of blades. It is typically about 6, 5, and 3 for two-bladed, three-bladed, and four-bladed rotor, respectively. For the vertical axis wind turbine, it is smaller than horizontal axis wind turbine, and varies with the chord area. The default and maximum values are 5.0 and 12.0.

Power calculation method

There are 2 models to choose from:

- **1-Simple** - simple approximation model employs the general kinetic energy equation and a [Maximum power coefficient](#) to calculate power extraction from the wind.
- **2-Analytical** - The power output of the turbine is calculated from the [Power coefficient equation](#) using 6 parameters C_{1-6} .

Maximum power coefficient

This is the maximum fraction of power extraction possible from the ambient wind and is required only when the **1-Simple** Simulation method is selected. This value can be calculated from the power curve published in most manufacturers' specifications by using the kinetic energy equation as:

$$C_p = \frac{P}{0.5\rho AV^3}$$

where:

P = power production at the rated wind speed [W]

ρ = density of air [kg/m³]

A = swept area of rotor [m²]

V = rated wind speed [m/s]

C_p = power coefficient

The maximum and default coefficient values are 0.59 and 0.35 respectively.

Annual local average wind speed

This is the local annual average wind speed that represents a representative wind profile at the location of the system (in m/s or ft/min). It is used to factor the difference in wind speed between the weather file wind data and the locally measured wind data so that the model minimizes uncertainties caused by improper wind data at the particular location. Considerable differences between the weather file wind data and the local wind data typically appear so it is important to consider this carefully in order to use accurate local wind data in the simulation. The model internally determines a multiplier and it is multiplied by the weather file wind data adjusted at the height of the system.

Height for local average wind speed

This is the height that the local wind speed is measured (in m or ft). The annual average wind speed (see previous field) input by the user is internally recalculated by existing EnergyPlus functions at the height of the local station. This modified wind speed is then factored and applied to the weather file wind data. The minimum and default values are zero and 50m.

Vertical Axis

For vertical axis turbines you can enter details on the Blade chord area, drag coefficient and lift coefficient on the Vertical Axis tab.

Blade chord area

This is the blade chord area of a single blade of VAWT system in (m² or ft²). It is necessary to determine the net tangential and normal forces of a single blade.

Blade drag coefficient

This is the blade drag coefficient for a specific blade. It is for determining the tangential and normal force coefficients with the blade lift coefficient (below) so that the model can calculate the power output from the system. The user should be able to obtain this parameter for a specific blade from the manufacturers' data. This field is only required for VAWT systems.

Blade lift coefficient

This field is the blade lift coefficient for a specific blade. It is for determining the tangential and normal force coefficients with the blade drag coefficient (above) so that the model can calculate the power output from the system. The user should also be able to obtain it for a specific blade from the manufacturers' data. This field is only required for VAWT systems.

Power Coefficients

For horizontal axis turbines using the [2-Analytical Power calculation method](#) the Power coefficients can be entered on the Power Coefficients tab.

Power coefficient parameters C1-C6

These six fields are the parameters for the power coefficient equation shown below. These fields are used to determine the power coefficient of the system. The analytical approximation model of the power coefficient in EnergyPlus is:

$$C_p = C_1 \left(\frac{C_2}{\lambda_i} - C_3 \theta - C_4 \theta^x - C_5 \right) e^{-\frac{C_6(\lambda, \theta)}{\lambda_i}}$$

with

$$\frac{1}{\lambda_i} = \frac{1}{\lambda + 0.08\theta} - \frac{0.035}{\theta^3 + 1}$$

where:

C_p = power coefficient

C_{1-6} = empirical power coefficient parameters

λ = tip speed ratio (often known as TSR)

λ_i = tip speed ratio at ith time step

θ = azimuth angle of the pitch, 0 [degree]

The default values are given in the table below.

C_1	C_2	C_3	C_4	C_5	C_6
0.5176	116	0.4	0.0	5.0	21

Electrical Storage

Electrical energy provided by generators such as PV panels and wind turbines can be stored on site for later use using Storage components.

Tip: Electric storage devices are included in the model by specifying them on the [Electrical load centre](#) dialog when one of the storage [Electrical buss types](#) has been selected.

There are 2 ways to model the define the performance of the electrical storage device:

- [1-Simple storage](#) - simple definition of the performance of the storage device that is not intended to represent any specific type of storage technology.
- [2-Battery](#) - more detailed performance definition of a battery using the kinetic battery model.

The type of storage device is defined by selecting the appropriate **Category** option on the Storage dialog.

Simple Storage

The Simple storage option is accessed by selecting the **Simple storage** category on the [Storage](#) dialog. The Simple type of Electrical storage component is used to model storage of electricity in an [Electric load centre](#). It is a simple model that does not attempt to represent any of the detailed performance characteristics of a real storage device such as a battery. The type of power, AC or DC, depends on the configuration chosen as the [Electrical buss type](#) in the [Electric load centre](#).

Name

This field contains a unique name for the electric storage device.

Category

There are 2 categories to choose from:

- **Simple storage** for a simple definition of the performance of batteries or other storage device.
- **Battery** for a more detailed performance definition of a battery

The Category selected affects the type of model used and the data to be entered on the Storage dialog.

Availability schedule

This schedule describes the times when the storage device is available. If storage is not available, by scheduling a value of 0, then no electrical energy can be stored or drawn from the device. Any non-zero schedule value means the devices is available. If this field is blank, the schedule has values of 1 for all time periods.

Heat Gain To Zone

Attach to a zone

If the losses from the storage device should be represented as heat gains to a zone then check this box and select the zone in the next field.

Zone

This field contains the name of the thermal zone where the storage device is located. The selected zone will receive the storage losses as heat gains.

Radiative fraction for zone heat gains

This field contains the fraction of storage losses that enter the zone as long-wave thermal radiation. This should be a factor between 0.0 and 1.0. The balance of the losses are convective. This field is only used when a zone is attached.

Simple Options

Nominal energetic efficiency for charging

This field contains the charging efficiency. This is the energetic efficiency of storing electrical energy in the storage device. A value of 1.0 means the device does not lose any energy when charging.

Nominal discharging energetic efficiency

This field contains the discharging efficiency. This is the energetic efficiency of drawing electrical energy from the storage device. A value of 1.0 means the device does not lose any energy when drawing power.

Maximum storage capacity

This field describes the maximum amount of energy that can be stored in the device in Joules. Once the storage device is full, no additional energy can be stored in it.

Maximum power for discharging

This field describes the maximum rate at which electrical power can be discharged from the storage device (in W).

Maximum power for charging

This field describes the maximum rate at which electrical power can be stored in the device (in W).

Initial state of charge

This field describes the value for the initial state of charge in Joules. This is used to model the storage device as having some amount of stored energy at the beginning of the simulation period.

Battery

The Battery option is accessed by selecting the Battery category on the [Storage](#) dialog. Batteries can be used to store charge in an [Electric load centre](#). The battery bank is a collection of one or more individual battery modules. Given the surplus or deficit power from the electrical system and the state of charge from the previous time step, this object can model the voltage, current, and energy losses with charging and discharging during each time step. The cumulative battery damage can be also modelled and reported at the end of each simulation run.

The battery component allows both Lead-Acid and Nickel Cadmium batteries to be simulated. With input parameters derived from specific battery tests, the object is expected to support other battery types such as Lithium-ion batteries.

The kinetic battery model assumes that part of the battery's energy storage capacity is immediately available in discharging or charging while the rest is chemically bound. As a function of constant current, the battery capacity is related to three parameters: the maximum capacity at infinitesimal current, the capacity ratio of available charges, and the conversion ratio between available charges and bound charges. These parameters are usually obtained via curve fitting based on battery data sheets or test data.

Each individual battery module is modelled as a voltage source in series with an electrical resistance. KiBaM assumes that the internal resistance is constant and the open circuit voltage varies with the electric current, the state of charge and the operation mode (charging or discharging). For an individual battery module, the open circuit voltage at any time is correlated to the voltage at fully charged/discharged state and three other regression coefficients. These regression coefficients are usually obtained via curve fitting based on battery test data.

The object offers user the option to perform battery life calculation. If battery life is modelled, the user needs to provide a group of coefficients for the correlation between the number of cycles for battery failure and the corresponding cycle range. More detailed information can be found in the [Engineering Reference](#).

Name

This alpha field contains the identifying name for the battery bank.

Category

There are 2 categories of electrical storage device to choose from:

- [Simple storage](#) for a simple definition of the performance of batteries or other storage device.
- **Battery** for a more detailed performance definition of a battery

The Category selected affects the type of model used and the data to be entered on the Storage dialog.

Availability schedule

This alpha field contains the schedule name (ref. Schedule) that describes when the battery is available. A schedule value greater than 0 (usually 1 is used) indicates that electrical energy can be stored or drawn from the battery. A value less than or equal to 0 (usually 0 is used) denotes that the battery is not available. If this field is blank, the schedule has values of 1 for all time periods.

Heat Gain To Zone

Attach to a zone

If the losses from the storage device should be represented as heat gains to a zone then check this box and select the zone in the next field.

Zone

This field contains the name of the thermal zone where the battery is located. Entering a valid name of zone here will direct EnergyPlus to include the energy storage losses as heat gains to the named thermal zone. If the battery is not within a thermal zone, this field can be left blank and the thermal energy associated with storage losses is removed from the building model.

Radiative fraction for zone heat gains

This field contains the fraction of storage losses that enter the zone as long-wave thermal radiation. This numeric field should have a value between 0.0 and 1.0. The balance of the losses is convective. This data is only required when the [Attach to a zone](#) option is checked.

Battery Storage

Number of battery modules in parallel

This field defines the number of modules connected in parallel in the battery bank.

Number of battery modules in series

This field defines the number of modules connected in series in the battery bank. The total number of modules in a battery bank is equal to the number of modules in parallel times the number of modules in series.

Maximum module capacity

This field indicates the maximum capacity of one battery module. It is evaluated as the total Amp-hours available when a full battery is discharged at infinitesimal current. This capacity is for an individual battery module, not for the whole battery bank. The maximum capacity can be found from manufacture's data or derived from test data by curve fitting.

Initial fractional state of charge

This field describes the initial state of charge in terms of the fraction of maximum capacity as defined in the previous field.

Fraction of available charge capacity

The kinetic battery model assumes that battery is a two-tank electrical energy storage device: an available tank and a bound tank. The available tank can be immediately charged or discharged, while the bound tank can only be charged or discharged via the available tank.

This field specifies the fraction of total charge in the battery that is part of the available tank. The ratio of available charge capacity to total capacity is a parameter usually derived from test data by curve fitting.

Change rate from bound charge to available charge

This field specifies the rate at which the charge flows between the available tank and the bound tank. It is a parameter used to calculate the conversion between available charge and chemically bound charge when charging and discharging the battery. This parameter is usually derived from test data by curve fitting.

Fully charged module open circuit voltage

This field indicates the open circuit voltage for a fully charged battery module. It can be found from manufacture's data or battery tests.

Fully discharged module open circuit voltage

This field indicates the open circuit voltage for a fully discharged battery module. It can be found from manufacture's data or battery tests.

Voltage change curve for charging

This field specifies the name of a [Rectangular Hyperbola 2 performance curve](#) that determines the change of open circuit voltage (ΔE) as a function of the battery state of charge in charging. The change of open circuit voltage is relative to a fully discharged battery. The curve has the following form:

$$\Delta E = (C_1 * X) / (C_2 + X) + C_3 * X$$

where X is the battery charge divided by the maximum capacity at a given current. More details can be found from the [Engineering Reference](#).

Voltage change curve for discharging

This field specifies the name of a [Rectangular Hyperbola 2 performance curve](#) that determines the change of open circuit voltage (ΔE) as a function of the battery state of charge in discharging. The change of open circuit voltage is relative to a fully charged battery. The curve has the following form:

$$\Delta E = (C_1 * X) / (C_2 + X) + C_3 * X$$

where X is the removed charge divided by the maximum capacity at a given current. More details can be found from the [Engineering Reference](#).

Module internal electrical resistance

This field specifies the battery internal resistance in ohms. In theory, the electrical resistance within the battery varies with the state of charge and depends on whether it is in charging or discharging. In the kinetic battery model, the internal resistance is assumed as constant and the terminal voltage varies with current and state of charge. The internal resistance may be obtained from the battery manufacture or battery test data. Note that the field is for an individual module, not for the whole battery bank.

Maximum module discharging current

This field indicates the maximum current at which the battery can be discharged continuously. The limit on discharge current is usually defined by the battery manufacture to avoid battery damage. The limit is for an individual battery module.

Module cut-off voltage

This field specifies the minimum allowable voltage, below which the battery is generally regarded as empty. The cut-off voltage is usually defined by the battery manufacture and it is for an individual battery module.

Module charge rate limit

This field specifies the limit on charging current relative to the remaining charge until the battery is full. This limit reflects the common practice that the charge rate is reduced as the battery gets more charged.

Battery Life

Battery life calculation

This checkbox indicates whether the battery life model is activated in the simulation. If the battery life model is activated, the following 2 inputs are required.

Number of cycle bins

This field specifies the number of equally ranged cycle bins in battery life simulation. If 10 bins are used, the cycle ranges will include 10%, 20%, ..., 100%.

Battery cycle life curve

This field specifies the [Double Exponential Decay Curve](#) that correlates the cycles of battery failure (C_F) and fractional depth of discharge (R). The curve is:

$$C_F = C_1 + C_2 e^{C_3 R} + C_4 e^{C_5 R}$$

Inverters

Inverters are used to model conversion from Direct Current (DC) to Alternating Current (AC) in an electric load centre that contains photovoltaic modules. Such a load centre has an array of photovoltaic modules that feed an inverter with DC power and the inverter produces AC power.

Tip: Electric inverters can be included in the model by selecting them on the [Generator List](#) tab of the Electrical load centre dialog when one of the d.c. [Electrical buss types](#) has been selected.

Three types of inverter can be defined:

- [Simple](#) - fixed conversion efficiency.
- [Function of Power](#) - conversion efficiency depends on power and is defined using a performance curve.
- [Lookup table](#) - conversion efficiency depends on power and is defined using a look up table.

The type of inverter can be selected using the **Category** option on the Inverter dialog.

Note: Each inverter defined in the model can only be used within a single Load centre. In other words, each DC load centre must have its own a unique inverter selected.

Inverter - Simple

Simple [inverters](#) use a single fixed conversion efficiency value.

General

Name

The name of the inverter entered here must also be unique across all other inverters in the model.

Category

Select the type of inverter from the options below:

- **Simple** - fixed conversion efficiency.
- **Function of Power** - conversion efficiency depends on power and is defined using a performance curve.
- **Lookup table** - conversion efficiency depends on power and is defined using a look up table.

Availability schedule

The schedule that describes when the inverter is available. If the inverter is scheduled to not be available, by scheduling a value of 0, then it cannot produce AC power and will not consume ancillary power during standby. Any non-zero schedule value means the inverter is available to produce AC power and consume ancillary power during standby. If the Inverter is scheduled to be unavailable but it is being supplied with DC power anyway, then the energy is dissipated as heat.

Simple

Inverter efficiency

This field contains the value for inverter efficiency. In the simple model, efficiency is a constant. This is the so-called 1-term model. Input data for different types of inverters is available at:

https://www.gosolarcalifornia.org/equipment/inverter_tests/summaries/

The “CEC Efficiency” would be the value to enter here.

Heat Gains To Zone

Attach to a zone

If the inverter’s losses are to be modelled as heat gains to a zone in the building thermal model then check this option. Otherwise inverter losses are considered to be lost to outside.

Zone

When including inverter losses and heat gains to a zone select the building zone where the inverter is located.

Radiative fraction

This field contains the fraction of inverter thermal losses that enter the zone as long-wave thermal radiation. This should be a factor between 0.0 and 1.0. The balance of the losses is convective. This data is only required when including inverter losses and gains to a zone.

Inverter - Function Of Power

Function of Power [inverters](#) use a function of normalized power to calculate the efficiency of conversion from DC to AC.

General

Name

The name of the inverter entered here must also be unique across all other inverters in the model.

Category

Select the type of inverter from the options below:

- [Simple](#) - fixed conversion efficiency.
- [Function of Power](#) - conversion efficiency depends on power and is defined using a performance curve.
- [Lookup table](#) - conversion efficiency depends on power and is defined using a look up table.

Availability schedule

The schedule that describes when the inverter is available. If the inverter is scheduled to not be available, by scheduling a value of 0, then it cannot produce AC power and will not consume ancillary power during standby. Any non-zero schedule value means the inverter is available to produce AC power and consume ancillary power during standby. If the Inverter is scheduled to be unavailable but it is being supplied with DC power anyway, then the energy is dissipated as heat.

Function Of Power

Efficiency function of power curve

This is the curve representing the relationship between DC power input into the inverter and the efficiency with which that power is converted to AC. The curve can be of type [Linear](#), [Quadratic](#), or [Cubic](#). The curve's "x" value is DC power input normalized by the [Maximum continuous input power](#) (below). The result of the curve should be power conversion efficiency expressed as a fraction between 0.0 and 1.0.

Rated maximum continuous input power

This field contains the rated power input (in W). This is value used to normalize input power for use with the curve named in the previous field. This is DC power from the PV arrays going into the inverter.

Minimum efficiency

This field contains a minimum bound on the inverter efficiency. This value will be used as a limit on the curve's result.

Maximum efficiency

This field contains a maximum bound on the inverter efficiency. This value will be used as a limit on the curve's result.

Minimum power output

This field contains a lower limit on the AC power produced by the inverter (in W). If the resulting power output would be below this level, then the inverter is assumed to not produce any power and is in stand by mode. All DC power input is lost (unless it is going into storage).

Maximum power

This field contains an upper limit on the AC power produced by the inverter (in W). If the resulting power output would be above this level, then the power produced is capped at this level with the rest of input power converted to losses (unless it is going into storage).

Ancillary power consumed In standby

This field contains the ancillary power (in W) used by the inverter when not producing AC power. This is AC electricity consumed in standby mode. Standby mode occurs when the inverter is scheduled to be available but the incoming DC power is too low.

Heat Gains To Zone

Attach to a zone

If the inverter's losses are to be modelled as heat gains to a zone in the building thermal model then check this option. Otherwise inverter losses are considered to be lost to outside.

Zone

When including inverter losses and heat gains to a zone select the building zone where the inverter is located.

Radiative fraction

This field contains the fraction of inverter thermal losses that enter the zone as long-wave thermal radiation. This should be a factor between 0.0 and 1.0. The balance of the losses is convective. This data is only required when including inverter losses and gains to a zone.

Inverters - Lookup Table

Lookup Table inverters model DC to AC power conversion efficiency by interpolating values from a look up table. The data structure has been designed to be used with data contained in the Inverter Performance Test Summaries available at:

https://www.gosolarcalifornia.org/equipment/inverter_tests/summaries/

All of the data needed for this object can be found there by make and model of inverter. The core of the data set is a 3x6 matrix of efficiency at different voltage and power levels. These form the look up table used in the model. Note that EnergyPlus PV models currently model only power and not the voltage and current situation. Therefore, only the nominal voltage data are used. This is the middle row of data in the Go Solar summaries.

General

Name

The name of the inverter entered here must also be unique across all other inverters in the model.

Category

Select the type of inverter from the options below:

- [Simple](#) - fixed conversion efficiency.
- [Function of Power](#) - conversion efficiency depends on power and is defined using a performance curve.
- [Lookup table](#) - conversion efficiency depends on power and is defined using a look up table.

Availability schedule

The schedule that describes when the inverter is available. If the inverter is scheduled to not be available, by scheduling a value of 0, then it cannot produce AC power and will not consume ancillary power during standby. Any non-zero schedule value means the inverter is available to produce AC power and consume ancillary power during standby. If the Inverter is scheduled to be unavailable but it is being supplied with DC power anyway, then the energy is dissipated as heat.

Lookup Table

Rated maximum continuous output power

This field contains the rated maximum continuous output power (in W).

Night tare loss power

This field contains the “night tare loss” (in W). This is modelled as ancillary power consumed during standby.

Nominal voltage input

This field contains the nominal DC input voltage (in V). This is “Vnom” in the [Go Solar test summaries](#).

Efficiency At Nominal Voltage

At 10% Power

The fractional efficiency at nominal voltage and 10% power.

At 20% Power

The fractional efficiency at nominal voltage and 20% power.

At 30% Power

The fractional efficiency at nominal voltage and 30% power.

At 50% Power

The fractional efficiency at nominal voltage and 50% power.

At 75% Power

The fractional efficiency at nominal voltage and 75% power.

At 100% Power

The fractional efficiency at nominal voltage and 100% power.

Heat Gains To Zone

Attach to a zone

If the inverter’s losses are to be modelled as heat gains to a zone in the building thermal model then check this option. Otherwise inverter losses are considered to be lost to outside.

Zone

When including inverter losses and heat gains to a zone select the building zone where the inverter is located.

Radiative fraction

This field contains the fraction of inverter thermal losses that enter the zone as long-wave thermal radiation. This should be a factor between 0.0 and 1.0. The balance of the losses is convective. This data is only required when including inverter losses and gains to a zone.

Economics

Economics tab in model data

When using [Detailed HVAC](#) you can define 3 aspects of building economics on the **Economics** tab at building level:

- [Tariff analysis](#) - to calculate the costs of the building utility usage.

- [Construction costs](#) - include construction costs calculated on the [Cost and Carbon](#) tab in simulation inputs.
- [Life-cycle cost](#) - (LCC) to calculate the whole building life-cycle cost.

The data entered on the Economics tab will be passed to EnergyPlus to be included in simulations.

Note: You must have Economics and HVAC licence modules to work with the data on this tab. Detailed HVAC is required to provide access to components with the full set of EnergyPlus fuel types for tariff analysis and LCC. Simple HVAC cannot access tariff analysis (and hence LCC) as it uses ideal loads HVAC systems rather than EnergyPlus fuels.

Tariff Analysis

To calculate the costs of the utilities, fuels etc use the tariff analysis section of the Economics tab.

Include tariff analysis

To include a tariff analysis check the Include tariff analysis checkbox.

Number of tariffs

Enter the number of tariffs to be included in the simulation. You will be able to select a Tariff analysis component for each.

Tariff 1, 2 etc

Select the [Tariff analysis](#) component describing the characteristics of each tariff to be included. Up to 5 different tariffs can be selected depending on the **Number of tariffs** selected above.

Construction Costs

Include construction cost in simulation inputs

To include the construction capital cost of the building in the simulation you should check this checkbox. This must be done to include construction cost in a Life cycle cost calculation.

Summary results from the most recently run [Cost calculation](#) on the Cost and Carbon tab will be included in the "Component cost" component which is selected by default.

Construction cost

Select the [Construction cost](#) component.

Life Cycle Cost

Include life cycle cost analysis

To include a life cycle cost analysis check this check box.

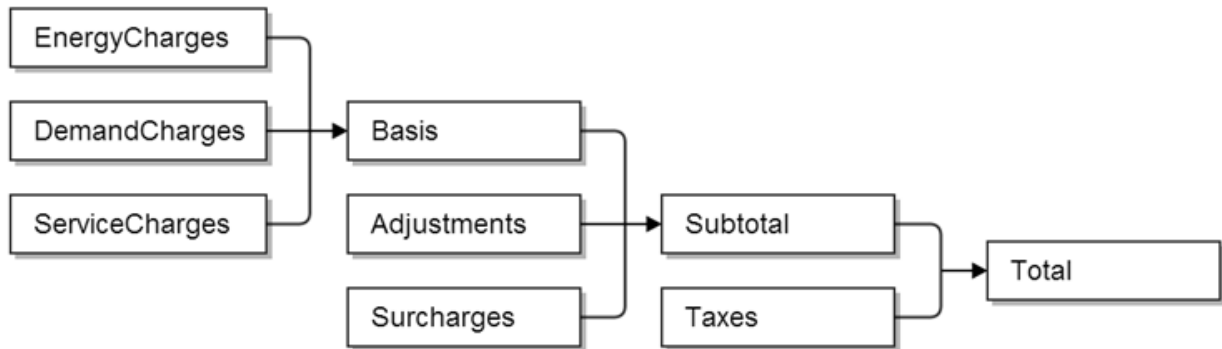
Life cycle cost

Select the [Life cycle cost](#) component defining the analysis to be included in the simulation.

Tariff Analysis

Background

Monthly utility bills are often directly related to monthly energy consumption and monthly peak demand. DesignBuilder EnergyPlus' approach to tariff analysis is to allow you to model the individual component charges that make up the utility bill and report the results on a monthly basis. The calculation is organised in hierarchy for economics charges illustrated below.

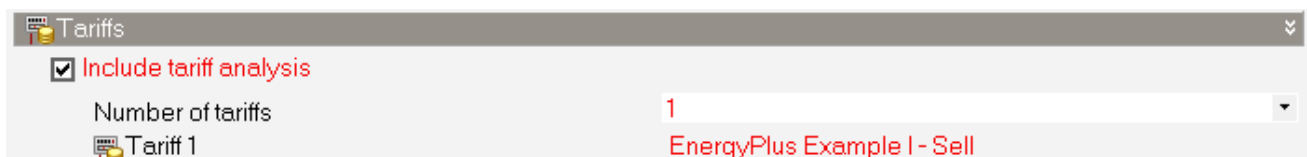


Any charges included in the EnergyCharges category are added together at the first step. The EnergyCharges, DemandCharges and ServiceCharges are added together to form the Basis. The Basis, Adjustments and Surcharges are added together to form the Subtotal. And finally the Total is obtained from the summation of Subtotal and Taxes, where the Total represents the total monthly charges on that tariff for the energy source used. In addition, each category name is usually used as a source variable when setting charges and doing tariff calculation.

Multiple Tariffs can be modelled, and it is common that electric and gas utilities have more than one rate that may be used. However, only one Tariff for each Output Meter object and Group Name is shown in the Economics Summary Report. The one that is shown in the report is chosen by first eliminating Tariffs that are not qualified due to their Qualify objects. If more than one Tariff object remains, the one that has the lowest cost for a combination of an Output Meter and Group Name will be selected. If the Group Name field is not checked, then the lowest cost Tariff object is chosen for each Meter.

Working With Tariff Analysis

DesignBuilder provides access to EnergyPlus' tariff analysis capabilities by allowing you to define one or more tariffs using the Tariffs dialog. To include a tariff analysis in your simulation, start by checking the **Include tariff analysis** checkbox. This allows you to set the number of tariffs to be defined and to assign a Tariff component to each one.



To edit a tariff, for example Tariff 1, click on the associated browse control and use the **Edit/view selected item** Info panel toolbar icon. This opens the Tariffs dialog.

There are 5 tabs on the Tariffs dialog:

- [Tariff](#)
- [Charge](#)
- [Qualify](#)
- [Ratchet](#)
- [User Variables](#)

See also:

- "Tariff Examples" on page 565

Tariff Analysis - Tariff Tab

Tariff tab on [Tariff Analysis](#) dialog

Tariff

Tariff name

The field gives the name of the tariff to be modelled. Tariffs are sometimes called rates. The name is used in identifying the output results and in associating all of the charges and other objects that make up a tariff.

Description

Enter a description for the tariff.

Category

Select a category for the tariff from:

- Electricity tariffs
- Gas tariffs
- Other tariffs

Output meter name

The name of any relevant meter defined by EnergyPlus but normally set to one of:

- 1-Electricity:Facility,
- 2-Gas:Facility,
- 3-ElectricityPurchased:Facility,
- 4-ElectricitySurplusSold:Facility,
- 5-ElectricityNet:Facility,
- 6-Other Defined Meter

This output meter name must point to a valid meter, otherwise it will lead to unexpected results.

Usually:

- the **Electricity:Facility** meter or the **ElectricityPurchased:Facility** is selected when the [buyFromUtility Buy or sell](#) option is used,
- the **ElectricitySurplusSold:Facility** meter is selected when the [sellToUtility Buy or sell](#) option is used and
- the **ElectricityNet:Facility** meter is selected when the [netMetering Buy or sell](#) option is used.

To calculate energy cost for specific energy use, the corresponding output meter name must be used (if input manually, it should be typed exactly as EnergyPlus defined). For example, if your utility cost calculation is intended for lighting electricity only, the InteriorLights:Electricity output meter name should be used here.

Note: The default **1-Electricity:Facility** option may not be suitable for models with on-site generation. In these cases Electricity:Purchased or ElectricityNet:Facility with appropriate Buy or Sell option should be chosen.

Conversion factor choice

A choice that allows several different predefined conversion factors to be used; otherwise, user defined conversion factors are used as defined in the next two fields.

The choices are:

- **1-UserDefined**
- **2-kWh**
- **3-Therm**
- **4-MMBtu**
- **5-MJ**
- **6-KBtu**
- **7-MCF**
- **8-CCF**

The following table shows the conversion factors when specifying one of the predefined choices:

Choice	Energy Conversion Factor	Demand Conversion Factor
kWh	0.0000002778	0.001
Therm	9.4781712E-09	0.00003412
MMBtu	9.4781712E-10	0.000003412
MJ	0.000001	0.0036
KBtu	9.4781712E-07	0.003412
MCF	9.4781712E-10	0.000003412
CCF	9.4781712E-09	0.00003412

Energy conversion factor

This field will be used only if Conversion factor choice is set to **1-UserDefined**. It defines a multiplier used to convert energy into the units specified by the utility in their tariff. If no conversion is required, input 1 here. This is also used for all objects (such as Charge, Qualify) used in the same tariff.

Demand conversion factor

This field will be used only if Conversion Factor Choice is set to **1-UserDefined**. It defines a multiplier used to convert demand into the units specified by the utility in their tariff. If no conversion is required, input 1 here. This is also used for all objects (such as Charge, Qualify) used in the same tariff.

Schedules

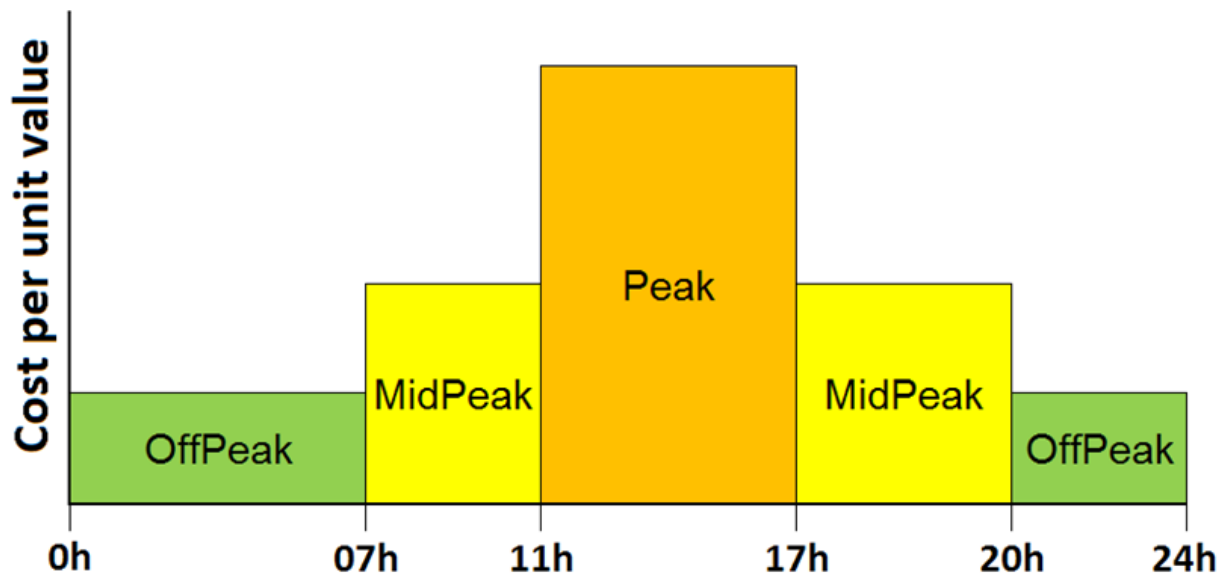
Use 'Time of use period schedule'?

Check this check box if you would like to include a time of use period schedule.

Time of use period schedule

Select the schedule that defines the time-of-use periods that occur each day. The period schedule is used to determine which variables are defined. The values for the different variables are:

- 1 is Peak
- 2 is Shoulder
- 3 is OffPeak
- 4 is MidPeak



Note: Different utilities use different terms to describe their rates. Sometimes the "shoulder" and the term "midpeak" both mean basically the same thing. To define a time of use period schedule, you do not need always use all of these terms. For terms "Shoulder" and "MidPeak", in DesignBuilder/EnergyPlus you can use either term as long as you use them consistently throughout the description of the utility tariff. Of course, you can use them both if the utility has four levels though it is very unusual.

The following variables are created automatically if these different periods are used in the schedule and include:

- peakEnergy
- peakDemand
- shoulderEnergy
- shoulderDemand
- offPeakEnergy
- offPeakDemand
- midPeakEnergy
- midPeakDemand

Some special variables are created that include only demand from one period that exceeds the demand from another period. For other months that do not exceed, the values are zero. These variables are seldom used but are available for the occasional rate that includes a clause looking at demands during one period that exceed another period.

- PeakExceedsOffPeak
- OffPeakExceedsPeak
- PeakExceedsMidPeak
- MidPeakExceedsPeak
- PeakExceedsShoulder
- ShoulderExceedsPeak

If no Period Schedule is specified then no variables mentioned above will be created.

Use 'Season schedule'?

Check this check box if you would like to include a season schedule.

Season schedule

Select the schedule that defines the seasons. The seasons are:

- 1 is Winter
- 2 is Spring
- 3 is Summer
- 4 is Autumn

The change in the season schedule must occur at the same time as the change in the Monthly Schedule. In other words, a season must end at the same time as a billing month. Variables are automatically created if a season schedule is used. These variables are set to 1 within the season and 0 for the months that are not in the season. The variables are:

- isWinter
- isNotWinter
- isSpring
- isNotSpring
- isSummer
- isNotSummer
- isAutumn
- isNotAutumn

Use 'month schedule'?

Check this check box if you would like to include a month schedule.

Month schedule

Select the schedule that defines the billing periods of the year. Normally this entry is allowed to default and a schedule will be internally used that has the breaks between billing periods occurring at the same time as the breaks between months, i.e., at midnight prior to the first day of the month. If other billing periods are used such as two month cycles or a single bill for an entire season, such as some natural gas companies do in the summer, then the month schedule may be used to redefine it. Make sure that the month schedule and season schedule are consistent, otherwise an error will be issued.

- 1 is the first month, usually January
- 2 is the second month, usually February
- 3 is the third month, usually March
- 4 is the fourth month, usually April
- 5 is the fifth month, usually May
- 6 is the sixth month, usually June
- 7 is the seventh month, usually July
- 8 is the eighth month, usually August
- 9 is the ninth month, usually September
- 10 is the tenth month, usually October
- 11 is the eleventh month, usually November
- 12 is the twelfth month, usually December

Thus, the schedule that represents this would look like the following. If no schedule is specified the following Month Schedule is used which defines the months of the year as the normal calendar months.

The use of the Month Schedule Name field is not usually required. Most utility tariffs have monthly bills and so this field would not need to be used. Specifically, if you have bills every two months, you would define a schedule to use in this field that had the value of 1 for January and February, the value of 2 for March and April, the value of 3 for May and June, etc. See below:

```
Schedule:Compact, TwoMonthBilling, number,
Through: 2/28, For: AllDays, Until: 24:00, 1,
Through: 4/30, For: AllDays, Until: 24:00, 2,
Through: 6/30, For: AllDays, Until: 24:00, 3,
Through: 8/31, For: AllDays, Until: 24:00, 4,
Through: 10/31, For: AllDays, Until: 24:00, 5,
Through: 12/31, For: AllDays, Until: 24:00, 6;
```

For other cases, some natural gas utilities define a single billing period for the summer months. In this case you would define a schedule that was 1 for January, 2 for February, 3 for March, 4 for April, 5 for May June July and August, 6 for September, 7 for October, 8 for November, 9 for December.

```
Schedule:Compact, SingleSummerBilling, number,
Through: 1/31, For: AllDays, Until: 24:00, 1,
Through: 2/28, For: AllDays, Until: 24:00, 2,
Through: 3/31, For: AllDays, Until: 24:00, 3,
Through: 4/30, For: AllDays, Until: 24:00, 4,
Through: 8/31, For: AllDays, Until: 24:00, 5,
Through: 9/30, For: AllDays, Until: 24:00, 6,
Through: 10/31, For: AllDays, Until: 24:00, 7,
Through: 11/30, For: AllDays, Until: 24:00, 8,
Through: 12/31, For: AllDays, Until: 24:00, 9;
```

Other Settings

Demand window length

The determination of demand can vary by utility. Some utilities use the peak instantaneous demand measured but most use a fifteen minute average demand or a one hour average demand. Some gas utilities measure demand as the use during the peak day or peak week.

The choices for demand window are:

- QuarterHour
- HalfHour
- FullHour
- Day
- Week

This field is defaulted to the shortest QuarterHour.

Note: The choice may be overridden based on the value of simulation time step so that they are consistent, i.e. demand window length should be exact multiples of simulation time step length in minute. For example, if the time step is 6, i.e. 10 minutes time step, then QuarterHour is not used and instead HalfHour is used. Or if time step is 3, i.e. 20 minutes time step, then QuarterHour and HalfHour are not used and instead FullHour is used.

Use 'Real time pricing charge schedule'?

Check this checkbox if you would like to use a real time pricing schedule (below).

Real time pricing charge schedule

This field is used with real time pricing rates. The schedule contains the cost of energy for that particular time period of the year. Real time rates can be modelled using a charge schedule with the actual real time prices entered. The charges should be consistent with the conversion factor specified in the tariff.

When this option is enabled, the Time of Use Period, Season and Month schedules are not needed unless Charge objects are used in defining other charges which involve these schedules.

Use 'Customer baseline load schedule'?

Check this checkbox if you would like to use a Customer baseline load schedule (below).

Customer baseline load schedule

Customer baseline load (CBL) schedule is used with real time pricing rates. This field provides the name of a schedule that contains the baseline energy use for the customer. Many real time rates apply the charges as a

credit or debit only to the difference between the baseline use and the actual use. The baseline use is established between the customer and the utility using a contract.

If this field is used the baseline totalEnergy and totalDemand is calculated, then compared to the metered value and therefore the real time rate charge based on the difference is calculated using the values defined in real time pricing charge schedule, which is either added or subtracted to the utility cost.

If this field is not used, the real time charge schedule applies to all energy from the meter and totalEnergy and totalDemand is based on metered value as usual.

Use 'Group name'?

Check this checkbox if you would like to use a Group name (below).

Group name

This field defines the group name of the tariff such as distribution, transmission, supplier, etc. If more than one tariff with the same group name is present and qualifies, only the lowest cost tariff is used. Usually the group name field is left blank which results in all tariffs using the same meter variable being compared and the lowest cost one being selected.

Monthly service charge

This gives a fixed monthly service charge that many utilities have. The entry may be numeric and gets added to the ServiceCharges variable. The entry can also be a variable name, where the values for each month will be used (this variable must be defined on 'User Variables' tab).

Minimum monthly charge

The minimum total charge for the tariff or if a variable name is entered here, its values for each month are used (the variable must be defined on 'User Variables' tab). The sum of the Subtotal and Taxes usually equals the total unless you have specified the minimum monthly charge and it is greater than the sum of the Subtotal and Taxes.

Buy or sell

It sets whether the tariff is used for buying, selling or both to the utility. This should be allowed to default to BuyFromUtility unless a power generation system is included in the building that may generate more power than the building needs during the year.

The choices are:

- BuyFromUtility – The values from the metered variable are used and are shown as being purchases from the utility.
- SellToUtility – The values from the metered variable are used for a "sell back" rate to the utility. The charges in the rate should be expressed as negative values.
- NetMetering – Negative values are used to reduce any positive values during the specific period on the tariff when negative values occur (sell back to utility).

A warning will be issued if the selection of this field does not match the type of meter (please refer to Output Meter Name field for more details).

Tariff Analysis - Charge Tab

Charge tab on [Tariff Analysis](#) dialog

Charge Selection

Number of charges

The number of charges applied in this tariff can be selected from this dropdown list.

Simple or Block?

Select the charge type according to utility rate settings from:

- **1-Simple** is used to compute energy and demand charges that are very simple. It may also be used for taxes, surcharges and any other charges that occur on a utility bill. As many Simple Charge objects as needed may be defined for a single tariff and they will be added together.
- **2-Block** is used to compute energy and demand charges that are structured in blocks of charges. It may also be used for Taxes, Surcharges and any other charges that occur on a utility bill, although those are more commonly simple flat charges, so Simple Charge is more commonly used for those functions. As many Block Charge objects as needed may be defined for a single tariff and they will be added together. Blocks are a structure used by most utilities for calculating energy and demand charges and they allow the utility to charge more or less per unit of energy or demand if more units are used.

Charge : Simple

Charge variable name

This is the name associated with the Simple charge object and will appear in the report, which is also used to store the results of the Simple charge calculation. Spaces are not significant in Charge variable names. They are removed during the utility bill calculation process.

Source variable

This is the name of the source used by the Simple charge. This is usually the name of the variable holding the energy or demand but may also be the name of any variable including the Subtotal or Basis if other charges are based on those. This should be typed exact as defined.

the Source variable is usually the name of the variable holding the energy or demand but may also be the name of any variable including the subtotal or basis if other charges are based on those.

The list of EnergyPlus pre-defined variable names is:

- totalEnergy
- totalDemand
- EnergyCharges
- DemandCharges
- ServiceCharges
- Basis
- Adjustments
- Surcharges
- Subtotal
- Taxes
- Total

If it is a time-of-use rate then:

- peakEnergy
- peakDemand
- shoulderEnergy
- shoulderDemand
- offPeakEnergy

- offPeakDemand
- midPeakEnergy
- midPeakDemand

In addition see the Tariff Report in the [Summary](#) output to see other native variables that may be available.

Also you can create additional user defined variables to model complex tariffs.

Season

This is the name (or number) of a season for which the Simple charge is calculated. If this is set to Annual, the calculations are performed for the entire year (all months), otherwise they are calculated only for the months in the season defined. The season is defined by the Season Schedule on the Tariff tab.

The choices for seasons are the following:

- **1-Annual**
- **2-Winter**
- **3-Spring**
- **4-Summer**
- **5-Fall/Autumn**

Category variable name

Charges are added in the hierarchy as described on [Tariff Analysis](#) page. This field shows where the charge should be added. The reason for entering this field appropriately is to get the charge reported in a reasonable category. The charge automatically gets added to the variable named after this category. The choices for this field are limited to:

- EnergyCharges
- DemandCharges
- ServiceCharges
- Basis
- Adjustment
- Surcharge
- Subtotal
- Taxes
- Total
- NotIncluded

Cost per unit value or variable name

This field contains either a single number or the name of a variable. The number is multiplied with all of the energy or demand or other source that is specified in the source field. If a variable is used, then the monthly values of the variable are multiplied against the variable specified in the source field (the variable should be defined on the [User variables](#) tab). If no value or variable is entered, a zero is assumed. For SellToUtility tariffs, the values in this field are usually expressed as negative numbers.

Charge : Block

Block charges are used extensively for tariff calculation. They are used to compute energy and demand charges that are structured in blocks of charges. They may also be used for taxes, surcharges and any other charges that occur on a utility bill but those are more commonly simple flat charges so Charge:Simple is more commonly used for those functions. As many Charge Blocks as needed may be defined for a single tariff and they will be added together. Blocks are a structure used by most utilities for calculating energy and demand charges and they allow the utility to charge more or less per unit of energy or demand if more units are used.

Charge variable name

This is the name associated with the Block charge object and will appear in the report, which is also used to store the results of the Block charge calculation. Spaces are not significant in Charge variable names. They are removed during the utility bill calculation process.

Source variable

This is the name of the source used by the Block charge. This is usually the name of the variable holding the energy or demand but may also be the name of any variable including the Subtotal or Basis if other charges are based on those. This should be typed exact as defined.

Season

This is the name (or number) of a season for which the Block charge is calculated. If this is set to Annual, the calculations are performed for the entire year (all months), otherwise they are calculated only for the months in the season defined. The season is defined by the [Season schedule](#) on the Tariff tab.

The choices for seasons are the following:

- **1-Annual**
- **2-Winter**
- **3-Spring**
- **4-Summer**
- **5-Fall/Autumn**

Category variable name

Charges are added in the hierarchy as described on [Tariff Analysis](#) page. This field shows where the charge should be added. The reason for entering this field appropriately is to get the charge reported in a reasonable category. The charge automatically gets added to the variable named after this category. The choices for this field are limited to:

- EnergyCharges
- DemandCharges
- ServiceCharges
- Basis
- Adjustment
- Surcharge
- Subtotal
- Taxes
- Total
- NotIncluded

Remaining into variable

If the blocks do not use all of the energy or demand from the source, some energy and demand remains, then the remaining amount should be assigned to a variable. If no variable is assigned and some amount of energy or demand is not used in the block structure a warning will be issued.

Block size multiplier value or variable name

The sizes of the blocks are usually used directly but if a value or a variable is entered below, the block sizes entered in the rest of the charge are first multiplied by the entered value prior to being used. This is common for rates that are kWh/kW rates and in that case the variable that holds the monthly total electric demand would be entered.

Block size multiplier or variable name > Use value or variable name

Select whether you wish to use a value or variable to define a multiplier to be applied to all energy or demand or other source that is specified in the source field. Select from:

- **1-Value** - where all of the energy or demand or other source that is specified in the source field are multiplied by this value.
- **2-Variable name** where the monthly values of the variable are multiplied by the variable specified in the source field (the variable should be defined on “User Variables” tab).

If a variable is used, then the monthly values of the variable are multiplied against the variable specified in the source field (the variable should be defined on “User Variables” tab). If no value or variable is entered, a zero is assumed. For SellToUtility tariffs, the values in this field are usually expressed as negative numbers.

Block size multiplier or variable name > Input multiplier value

When [Use value or variable name](#) is set to **1-Value**, enter the multiplier value to be used to multiply the source data (e.g. energy demand).

Block size multiplier or variable name > Input multiplier variable name

When [Use value or variable name](#) is set to **2-Variable name**, enter the name of the variable that defines the multiplier, e.g. **totalDemand**.

See: [EnergyPlus Example E – kWh/kW Energy Charges](#) for an example of how to use this setting.

Number of blocks

Select the number of blocks needed for this Block charge setting.

Block size 1 value

The size of the first block of the charges is entered here. For most rates that use multiple blocks, this will be the value for the first block size. Additional block sizes are entered with every other field.

Block 1 cost per unit value

The cost per unit value of the first block is entered here. This value is also repeated for every other field in the charge when required. As many blocks may be entered as are needed. For SellToUtility tariffs, the values in this field are usually expressed as negative numbers.

Note: The above two fields run as a group can be repeated many times up to the number defined in [Number of blocks](#) field. For the last pair of the two, a default variable named **remaining** represents a very large value which may be used when the remaining amount should be included in that block. This **remaining** is EnergyPlus reserved keyword.

Tariff Analysis - Qualify Tab

Qualify tab on [Tariff Analysis](#) dialog

Many utilities provide a large number of tariffs and separate their customers by their energy use or by the peak demand. The Qualify object allows only certain tariffs that are modelled to be used. If the results of the simulation fall outside of the range of qualifications, that tariff is still calculated but in the results the “Qualified” entry will say “No” and the Qualify object that caused its exclusion is shown. Multiple tariffs from the same utility can be modelled and only one of them should be used, make sure that the qualifiers are consistent especially within the same [Group name](#) (defined on the [Tariff](#) tab). Multiple Qualify objects can appear for a specific rate and they can be based on any variable.

The energy used and demands for each month vary. Since it is possible that they vary across the minimum and maximum values defined by this object, a choice in how to treat the thresholds is necessary. Utilities

typically have specific requirements about which months or consecutive months can be above or below a threshold to qualify for the rate.

Number of qualify checks

The number of qualify checks for this tariff is defined here.

Name

The name used for the Qualify object and displayed if the tariff qualify or does not qualify. It is also treated as a variable and contains:

- 1 for each month that does qualify,
- 0 for months that do not qualify, and
- -1 for months that are not part of the season selected.

Variable name

The name of the variable used. For energy and demand, the automatically created variables, totalEnergy and totalDemand should be used, respectively.

Qualify type

This defines criteria for checking tariff's qualification, the options can be either "Minimum" or "Maximum".

Threshold value or variable name

This is the minimum or maximum value set for the qualify check. If the variable has values that are less than this value when the qualify type is minimum then the tariff may be disqualified. If the variable has values that are greater than this value when the qualify type is maximum then the tariff may be disqualified. Depending on the threshold test entry (see below) this threshold value applies to all months, some of the months or just a single month. If a variable that contains this threshold values is used, it should be defined on [User Variables](#) tab.

Season

If the UtilityCost:Qualify only applies to a season, enter the season name from the following list:

- Annual
- Winter
- Spring
- Summer
- Fall (Autumn)

It defaults to Annual.

Threshold test

The **Number of months** (below) is used in one of two different ways depending on this setting. Select from:

- **1-Count** where the qualification is based on the count of the total number of months per year.
- **2-Consecutive** where the qualification is based on a consecutive number of months.

Input Number of months

Check this checkbox if you would like to enter the number of months.

Number of months

A number is used here from 1 to 12. If **Input Number of months** is not checked, 12 is assumed when the qualify type is minimum and 1 when the qualify type is maximum. This is the number of months that the threshold test applies to determine if the rate qualifies or not. If the season is less than 12 months (if it is not annual) then the value is automatically reduced to the number of months of the season.

Tariff Analysis - Ratchet Tab

Ratchet tab on [Tariff Analysis](#) dialog

Ratchets are commonly used by US utility tariffs for large commercial and industrial electricity customers as a mechanism to calculate the demand charges on monthly electricity bills. They may be used to ensure that a high demand spike in one month, or season, results in a minimum billing charge for several subsequent months, even if the customer's actual usage is lower.

Utilities charge customers based on two main components:

- **Energy Charge** (kWh): the total amount of electricity consumed over the billing period.
- **Demand Charge** (kW): The maximum rate at which electricity was used during a short interval within the billing period.

A ratchet is a clause in the tariff that links the demand charge for the current month to a high demand peak from a previous month or season. A typical demand ratchet determines the Billed Demand (kW amount the customer is charged for) as the greater of:

1. The actual maximum demand in the current billing month.
2. A percentage (often 50% to 90%) of the highest maximum demand in a specified "look-back" period (e.g., the previous 11 months, or the peak summer season).

Why Utilities Use Ratchets

A utility must design its infrastructure (generators, transformers, transmission lines) to handle the maximum demand of all its customers. Even if a customer only hits a high peak once a year, the utility still has to build and maintain the capacity to serve that spike year-round. Utilities use ratchets to ensure they can recover the costs of building and maintaining a reliable electricity grid, especially the capacity required to serve peak loads. Ratchets create a strong financial incentive for large customers to avoid or manage short, excessive spikes in power usage, thereby smoothing out the load on the grid and deferring the need for costly infrastructure upgrades.

Ratchet Example 1

Consider a tariff has a ratchet set at 80% of the highest peak demand in the previous 11 months.

- **January:** The customer uses its equipment heavily and hits a peak demand of 1,000 kW. This sets a new high mark.
- **February - December (Next 11 Months):**
 - The ratchet "locks in" a minimum billed demand of 80% of that \$1,000 kW peak, which is 800 kW.
 - If the actual maximum demand in any of these months is 500 kW, you will still be charged for the minimum ratchet amount of 800 kW.
 - If your actual maximum demand in a later month is 900 kW, which is higher than the ratchet minimum, you will be charged for the actual demand of 900 kW (and this new high may reset the ratchet minimum for the remaining months).

The effect is that one brief surge in energy use can determine the minimum demand charge you pay for nearly a full year.

Ratchet Example 2

A utility has a single ratchet that says that the billing demand for each month of the year is the higher value of the actual demand or 60% of the peak summer demand.

DesignBuilder input:

Ratchet	
How many ratchets applied?	1
Ratchet 1 settings	
Ratchet variable name	BillingDemand1
Baseline source variable	TotalDemand
Adjustment source variable	TotalDemand
Season from	4-Summer
Season to	1-Annual
Multiplier value or variable name	
Use value or variable name?	1-Value
Input multiplier value	0.60
Offset value or variable name	
Use value or variable name?	1-Value
Input offset value	0.00

EnergyPlus idf:

```
UtilityCost:Ratchet,
    BillingDemand1,      ! Ratchet Variable Name
    ExampleTariff1,     ! Tariff Name
    TotalDemand,        ! Baseline Source Variable
    TotalDemand,        ! Adjustment Source Variable
    Summer,            ! Season From
    Annual,             ! Season To
    0.60,               ! Multiplier Value or Variable Name
    0;                  ! Offset Value or Variable Name
```

Multiple Ratchets

Multiple ratchets may be used to manage different peak concerns - like summer heat vs. year-round capacity - and they are applied sequentially to ensure the billed demand reflects the highest possible charge across all rules.

When a utility tariff contains multiple demand ratchets, they are applied in a chain or sequence. The goal is to compare the maximum demand from different look-back periods and different rules, and ultimately select the highest result as the customer's final Billed Demand. The mechanics are as follows:

1. **Ratchet 1 (e.g., Annual Ratchet):** The calculation for the first ratchet is performed, comparing the current month's peak to the highest peak in the previous 11 months (or the year).¹ The higher of these two values becomes the interim Billed Demand, let's call it D_1 .
2. **Ratchet 2 (e.g., Seasonal Ratchet):** The calculation for the second ratchet is performed. This ratchet often compares a seasonal peak (like the highest summer peak) to the current month's demand. The Billed Demand from Ratchet 1 (D_1) is used as the starting value or baseline for this comparison.

Final Billed Demand = Greater of (D_1 OR a percentage of the highest kW peak during the last peak season).

Essentially, the output of the first ratchet is fed into the second, and the output of the second is fed into the third (if one exists). This layering guarantees the customer pays a demand charge based on the worst-case scenario (highest demand spike) as defined by any of the tariff's ratchet rules.

In DesignBuilder, if multiple ratchets occur in the same tariff, they can be "chained" together using the [Baseline source variable](#) replaced with the [Ratchet variable name](#) of the previous Ratchet.

Chained Example

A large commercial customer might have two ratchets:

- **Rule A (Annual):** Billed Demand is the greater of (Current Month kW OR 80% of the last 11 months' highest kW)
- **Rule B (Seasonal):** Billed Demand is the greater of (Current Month kW OR 90% of the highest kW peak during the previous Summer season).

The final demand is the highest result derived from both comparisons.

When Utilities Use Multiple Ratchets

Utilities employ multiple, chained ratchets when they need to recover costs that are driven by different factors at different times of the year.

1. Separating Seasonal vs. Annual Capacity Costs

The most common reason for using multiple ratchets is to distinguish between costs related to annual operations and costs related to seasonal stress on the grid.

- **Seasonal Peak Ratchet (90% Summer Rule):** This rule is designed to recover the cost of peaking generation and transmission/distribution upgrades needed specifically to handle the extreme load of the hottest summer days (high air conditioning use). The high summer demand sets a high minimum for several non-summer months.
- **Annual Ratchet (80% Rolling Rule):** This rule ensures recovery of costs for the base, fixed capacity that must be available year-round, regardless of the season.

2. Targeting Coincident vs. Non-Coincident Peaks

Some utilities use multiple ratchets to separate charges for:

- **Non-Coincident Peak (NCP):** The customer's absolute highest kW peak for the month, regardless of when it occurs.
- **Coincident Peak (CP):** The customer's kW demand measured at the exact same time as the utility's entire system hit its highest overall peak (the "system peak").

A complex tariff might use a ratchet for the NCP and a separate, chained ratchet for the CP, ensuring the customer is incentivised to lower their usage during both their individual peak and the grid's most stressed hours.

Ratchet

Ratchet variable name

This entry defines the name of the ratchet and the variable that stores the result of this ratchet. The ratchet variable name should be used wherever the ratchet should be applied and is often the source variable for Simple or Block charge objects.

Baseline source variable

The name of the variable used as the baseline value. When the ratcheted value exceeds the baseline value for a month the ratcheted value is used but when the baseline value is greater than the ratcheted value the baseline value is used. Usually the electric demand charge is used. The baseline source variable can be the results of another ratchet object. This allows utility tariffs that have multiple ratchets to be modelled.

Adjustment source variable

This field defines the variable that the ratchet is calculated from. It is often but not always the same as the baseline source variable. The ratcheting calculations using offset and multiplier are using the values from the adjustment source variable. See the bottom of this section for how ratchet works.

Season from

This is the name of the season that is being examined. The maximum value for all of the months in the named season is what is used with the multiplier and offset. This is most commonly Summer or Annual. When Monthly is used, the adjustment source variable is used directly for all months.

The choices are:

- Annual
- Winter
- Spring
- Summer
- Fall (Autumn)
- Monthly

Season to

This is the name of the season when the ratchet would apply. This is most commonly Winter. The ratchet only is applied to the months in the named season. The resulting variable for months not in the Season To selection will contain the values as appear in the baseline source variable. The choices are:

- Annual
- Winter
- Spring
- Summer
- Fall (Autumn)

Multiplier value or variable name

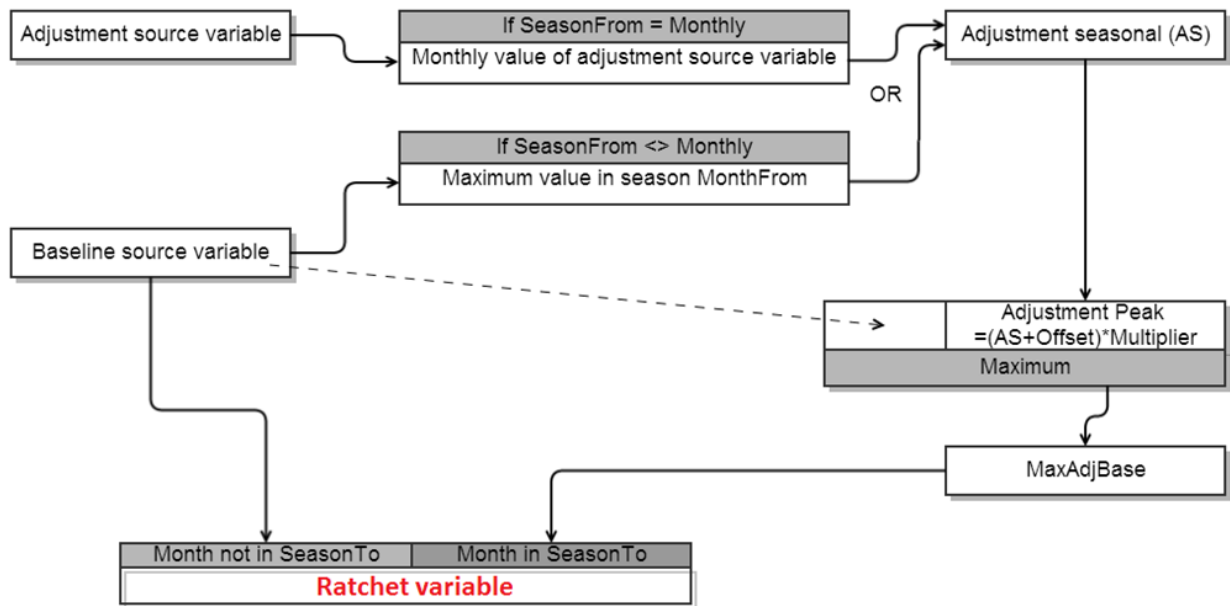
Often the ratchet has a clause such as “the current month demand or 90% of the summer month demand”. For this case a value of 0.9 would be entered here as the multiplier. This value may be given as one if no multiplier is needed. This can also be a variable name, which should be defined on [User variables](#) tab.

Offset value or variable name

A less common strategy is to say that the ratchet must be all demand greater than a value, in this case an offset that is added to the demand may be entered here. If entered, it is common for the offset value to be negative representing that the demand be reduced. The value can be zero if not offset is needed in the ratchet. This can also be a variable name, which should be defined on [User variables](#) tab.

Detailed step taken for ratchet calculation

- AdjSeasonal contains either:
 - o When SeasonFrom is not set to Monthly, the maximum for all months in Season From in the Adjustment Source Variable. This is a single value.
 - o When SeasonFrom is set to Monthly, the monthly values of the Adjustment Source Variable.
- $\text{AdjPeak} = (\text{AdjSeasonal} + \text{Offset}) * \text{Multiplier}$
- MaxAdjBase = maximum value of either AdjPeak or Baseline Source Variable
- The resulting Ratchet Variable contains:
 - o For months not in SeasonTo, the values of Baseline Source Variable
 - o For months in SeasonTo, the values of MaxAdjBase



For a simple typical case using totalDemand for both “Baseline source variable” and “Adjustment source variable”, please see the example at the top of this page.

Tariff Analysis - User Variables Tab

User variables tab on [Tariff Analysis](#) dialog

This tab defines and stores any user defined variables required for selection on other tabs of this dialog. Monthly values can be entered for each variable.

Note: Variables are defined using a text name which must be entered exactly on the previous tabs to match with the variable defined here.

The places on the other Tariff analysis dialog tabs where a variable can be entered are:

Tariff tab

- [Monthly service charge](#) variable name
- [Minimum monthly charge](#) variable name

Charge tab

- [Charge](#) variable name (Simple Charge setting)
- [Charge](#) variable name (Block Charge setting)

Qualify tab

- [Threshold](#) variable name

Ratchet tab

- [Multiplier](#) variable name
- [Offset](#) variable name

Variables Used

Number of variables used

Enter the number of variables to be defined.

Variable N Settings

For each variable enter the following data.

Name

This field defines the name of the variable.

Variable type

This field is used to indicate the kind of units that may be associated with the variable. It is used by IDF Editor to display the appropriate SI and IP units for the January through December values. The available options are shown below. If none of these options are appropriate, select Dimensionless which will have no unit conversion.

- Energy
- Demand
- Currency
- Dimensionless

January Through December Values

January value

February value

...

December value

Each field contains an entry of the monthly values of the variable. Normally 12 months are used for most utility rates and so 12 values should be entered.

Tariff Examples

One of the best ways to learn the details of EnergyPlus Economics and how to calculate operational energy costs is to study some example tariffs and how they are entered on the Tariff dialog. The examples below illustrate the process of setting up various tariffs for a range of cases of increasing complexity. Each example includes a description of the tariff to be modelled, step-by-step instructions on how to define it and sample summary output showing the corresponding EnergyPlus economics reports.

Note: The examples provided below are based on entering costs in US Dollars (USD/\$), but the principles are the same for all currencies. The [Currency](#) can be selected on the [International](#) tab of the Program options dialog.

List Of Examples

- [Create Base Model - Optional Preliminary Steps for Examples A-I](#)
- [Example A – Flat Energy Charge](#)
- [Example B – Block Energy Charge](#)
- [Example C – Block Energy and Demand Charges](#)

- [Example D - Seasonal Energy Charges with Minimum Charge](#)
- [Example E – kWh/kW Energy Charges](#)
- [Example F – Seasonal Time of Use Energy](#)
- [Example G – Blocks within Blocks](#)
- [Example H – Real Time Pricing](#)
- [Example I – Selling and Net-Metering](#)
- [Example J - Dual Ratchet Demand Tariff](#)

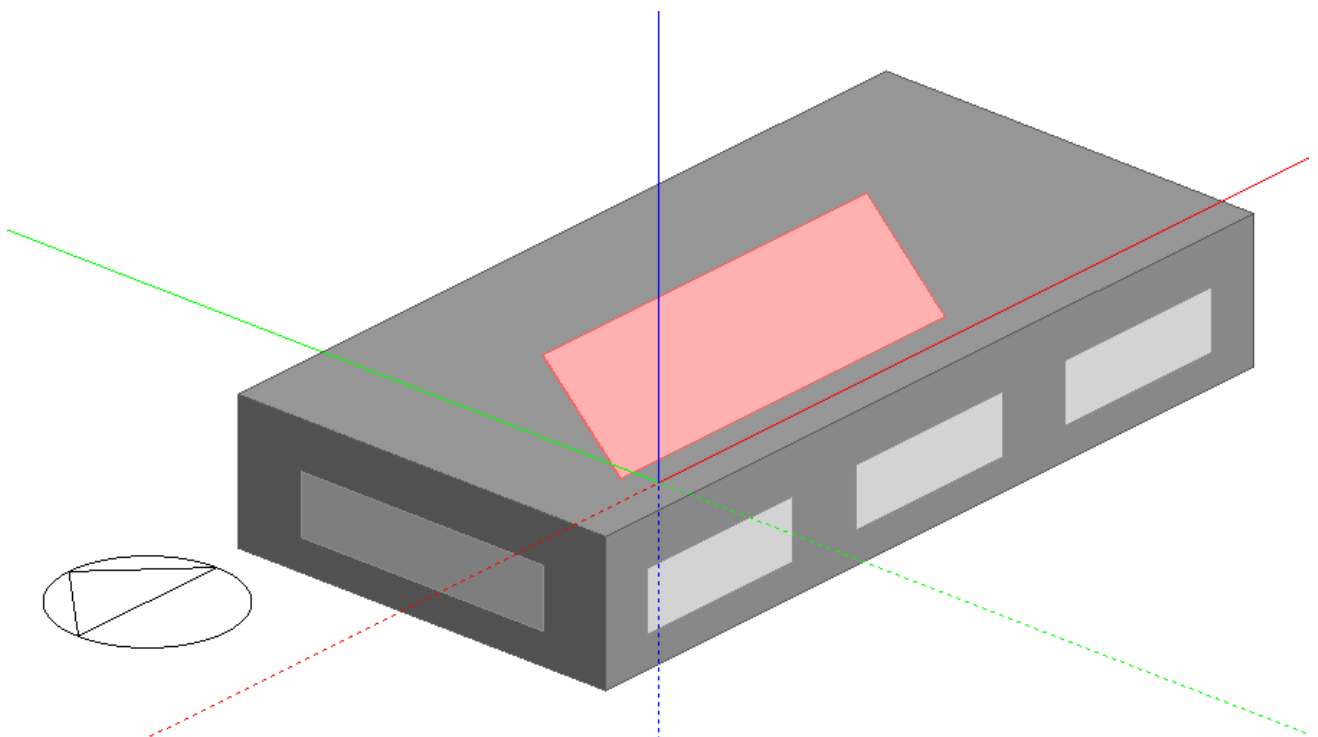
Create Base Model - Optional Preliminary Steps For Examples A-J

The tariffs described below under Examples A-J headings can be set up for any model. However, if you would like to reproduce the summary results provided to aid your learning process, you should apply the tariffs to the same base model used to create the results below. You can create the base model by following the steps below.

1. Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 20m x 10m as shown below. Start by using default Model options and template settings.
2. Set the [Detailed HVAC](#) model option. This is a required step as EnergyPlus Economics only works with full HVAC system definitions.
3. Load the default "FCU 4-pipe, Air-cooled Chiller" Detailed HVAC template.
4. Add a [PV panel](#) on the roof with dimensions 10m x 3m and rotate it so it faces south with an inclination of 45°.
5. Connect the PV panel to a distribution centre. To do this, go to the [Generation tab](#), create a new [Load centre](#) by copying the existing default "DC with Inverter" library load centre. On the **Generator List** tab, select your Solar collector.

The PV panel will be used in [Example I](#) to illustrate "net metering" and cases where electricity is sold back to the utility.

The model should now look similar to the screenshot below.



Example A – Flat Energy Charge

This example shows how to define a simple tariff which charges for electricity at a constant rate with a fixed monthly service fee. This simple approach is suitable for utilities or contracts that charge a flat rate without tiered pricing or demand charges.

- Customer charge: \$2.51 per month
- Energy charge: 5.5342 cents/kWh

This simple tariff can be set up by making the following settings on the Economics tab at building level.

To create this tariff structure, follow the steps below.

1. Create a new Tariff component:

The screenshot shows the 'Tariffs' dialog box. The 'Include tariff analysis' checkbox is checked. The 'Number of tariffs' is set to 1. Below this, 'Tariff 1' is listed with the name 'EnergyPlus Example A - Flat Energy Charge'.

2. On the **Tariff** tab of the Tariff dialog, set the fixed monthly fee to \$2.51 per month using the [Input monthly service charge value](#) setting as shown below:

The screenshot shows the 'Tariffs' dialog box with the 'Tariff' tab selected. The 'Tariff name' is 'EnergyPlus Example A - Flat Energy Charge'. The 'Description' is 'Example data'. The 'Source' is empty. The 'Category' is 'Electricity tariffs'. The 'Output meter name' is '1-Electricity:Facility'. The 'Conversion factor choice' is '2-kWh'. The 'Schedules' section has three unchecked checkboxes: 'Use 'Time of use period schedule'', 'Use 'Season schedule'', and 'Use 'Month schedule''. The 'Other Settings' section has four unchecked checkboxes: 'Use 'Real time pricing charge schedule'', 'Use 'Customer baseline load schedule'', and 'Use 'Group name''. The 'Monthly service charge' section has 'Monthly service charge or variable name' set to '1-Charge value' and 'Input monthly service charge value (USD)' set to '2.51'. The 'Minimum monthly charge' section has 'Minimum monthly charge or variable name' set to '1-Charge value' and 'Input minimum monthly charge value (USD)' set to '0.0000'. The 'Buy or Sell' section has 'Buy or Sell' set to '1-buyFromUtility'.

3. On the **Charge** tab:

- Set the Number of charges to 1.

- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **1-Simple**.
 - Set the [Input cost per unit value](#) to 0.055342 USD:

Tariffs

Tariff

Charge

Qualify

Ratchet

User variables

Charge Selection

Number of charges

1

Charge 1 settings

Simple or Block?

1-Simple

Charge variable name

FlatElectricityCharge

Source variable

totalEnergy

Season

1-Annual

Category variable name

1-EnergyCharges

Cost per unit value or variable name

Use value or variable name?

1-Value

Input cost per unit value (USD)

0.055342

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**
 Timestamp: **2025-09-11 11:48:03**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	979.83	0.00	0.00	979.83
Cost per Total Building Area [\$/m2]	5.36	0.00	0.00	5.36
Cost per Net Conditioned Building Area [\$/m2]	5.36	0.00	0.00	5.36

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE A - FLAT ENERGY CHARGE	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	979.83

Report: **Tariff Report**
 For: **ENERGYPLUS EXAMPLE A - FLAT ENERGY CHARGE**
 Timestamp: **2025-09-11 11:48:03**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	69.77	61.62	67.12	79.32	93.67	84.19	102.31	95.69	83.57	79.81	65.45	67.20	949.71	102.31
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	30.12	2.51
Basis (\$)	72.28	64.13	69.63	81.83	96.18	86.70	104.82	98.20	86.08	82.32	67.96	69.71	979.83	104.82
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Subtotal (\$) 72.28 64.13 69.63 81.83 96.18 86.70 104.82 98.20 86.08 82.32 67.96 69.71 979.83 104.82
 Taxes (\$) 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00
 Total (\$) 72.28 64.13 69.63 81.83 96.18 86.70 104.82 98.20 86.08 82.32 67.96 69.71 979.83 104.82

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
FLATELECTRICITYCHARGE (\$)	69.77	61.62	67.12	79.32	93.67	84.19	102.31	95.69	83.57	79.81	65.45	67.20	949.71	102.31	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65

Ratchets

Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----

Qualifies

Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndShoulderDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakAndMidPeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndMidPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example B – Block Energy Charge

This example adds tiered energy pricing blocks to a flat fee, where cost decreases as consumption increases. This is common for residential or small commercial tariffs with step-down pricing to make higher usage rates more affordable.

- Customer charge: \$5.75 per month
- Energy charges:
 - o 7.231 cents/kWh for first 200 kWh
 - o 6.656 cents/kWh for next 1000 kWh
 - o 5.876 cents/kWh for over 1200 kWh

To create this tariff structure, follow the steps below.

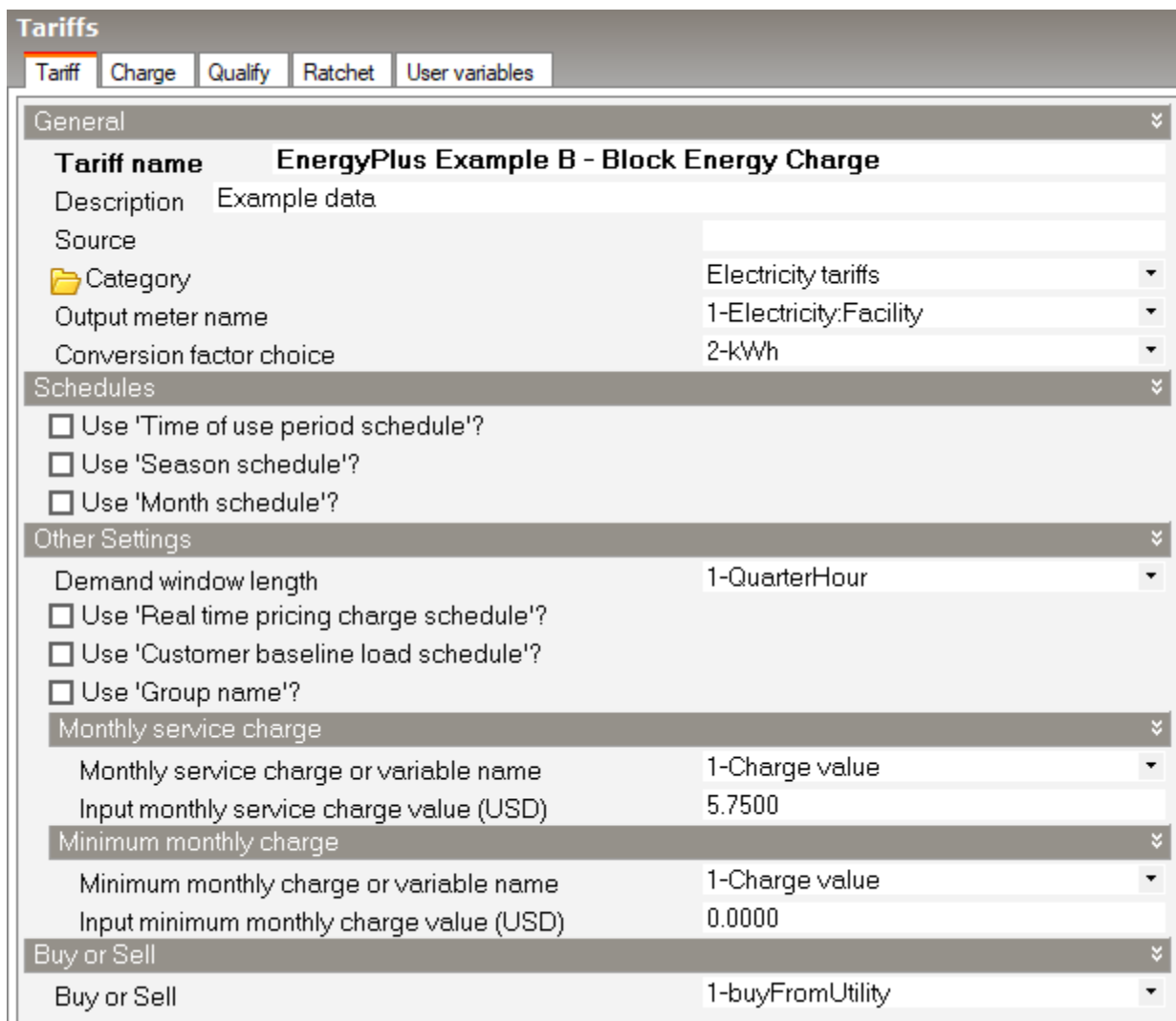
1. Create a new Tariff component:



The screenshot shows the 'Tariffs' dialog box. The 'Include tariff analysis' checkbox is checked. The 'Number of tariffs' is set to 1. The 'Tariff 1' dropdown shows 'EnergyPlus Example B - Block Energy Charge'.

2. On the **Tariff** tab of the Tariff dialog:

- Set the fixed monthly fee to \$5.75 per month using the [Input monthly service charge value](#) setting as shown below:



The screenshot shows the 'Tariffs' dialog box, 'Tariff' tab. The 'General' section is expanded, showing 'Tariff name' as 'EnergyPlus Example B - Block Energy Charge'. The 'Schedules' section has three unchecked checkboxes. The 'Other Settings' section is expanded, showing 'Monthly service charge' with 'Monthly service charge or variable name' set to '1-Charge value' and 'Input monthly service charge value (USD)' set to '5.7500'. The 'Buy or Sell' section is expanded, showing 'Buy or Sell' set to '1-buyFromUtility'.

3. On the **Charge** tab:

- Set the Number of charges to 1.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **2-Block**
 - Set [Source variable](#) to **totalEnergy**
 - Set the [Number of blocks](#) to 3
 - Set [Block size 1 value](#) to 200.
 - Set [Block 1 cost per unit value](#) to 0.07231 USD.
 - Set [Block size 2 value](#) to 1000.
 - Set the [Block 2 cost per unit value](#) to 0.06656 USD.
 - Set the [Block size 3 variable name](#) to **remaining**.
 - Set the [Block 3 cost per unit value](#) to 0.05876 USD.

The screenshot below show how the charge tab should look when you have finished.

Tariffs

Tariff
Charge
Qualify
Ratchet
User variables

Charge Selection

Number of charges
1

Charge 1 settings

Simple or Block?
2-Block

Charge variable name
BlockEnergyCharge

Source variable
totalEnergy

Season
1-Annual

Category variable name
1-EnergyCharges

Remaining into variable

Block size multiplier value or variable name

Use value or variable name?
1-Value

Input multiplier value
1.000

Number of blocks
3

Block size 1 value
200.00

Block 1 cost per unit value (USD)
0.072310

Block size 2 value
1000.00

Block 2 cost per unit value (USD)
0.066560

Block size 3 value or variable name

Use block size value or variable name?
2-Block size variable name

Block size 3 variable name
remaining

Block 3 cost per unit value
0.058760

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**
 Timestamp: **2025-09-11 11:44:35**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	1202.68	0.00	0.00	1202.68
Cost per Total Building Area [\$/m2]	6.58	0.00	0.00	6.58
Cost per Net Conditioned Building Area [\$/m2]	6.58	0.00	0.00	6.58

Tariff Summary

Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
----------	-----------	-------	-------------	-------	------------------

- 571 -

ENERGYPLUS EXAMPLE B - BLOCK ENERGY CHARGE Yes Yes ELECTRICITY:FACILITY Buy (none) 1202.68

Report: **Tariff Report**
 For: **ENERGYPLUS EXAMPLE B - BLOCK ENERGY CHARGE**
 Timestamp: **2025-09-11 11:44:35**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	84.59	75.26	81.77	94.73	109.97	99.90	119.14	112.11	99.24	95.25	79.86	81.87	1133.68	119.14
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	5.75	5.75	5.75	5.75	5.75	5.75	5.75	5.75	5.75	5.75	5.75	5.75	69.00	5.75
Basis (\$)	90.34	81.01	87.52	100.48	115.72	105.65	124.89	117.86	104.99	101.00	85.61	87.62	1202.68	124.89
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	90.34	81.01	87.52	100.48	115.72	105.65	124.89	117.86	104.99	101.00	85.61	87.62	1202.68	124.89
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	90.34	81.01	87.52	100.48	115.72	105.65	124.89	117.86	104.99	101.00	85.61	87.62	1202.68	124.89

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
BLOCKENERGYCHARGE (\$)	84.59	75.26	81.77	94.73	109.97	99.90	119.14	112.11	99.24	95.25	79.86	81.87	1133.68	119.14	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
PeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

PeakExceedsShoulder	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndShoulderDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakAndMidPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndMidPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example C – Block Energy And Demand Charges

This example uses separate block structures for energy and demand charges. This approach is suitable for modelling commercial tariffs that include both consumption and peak demand billing components. Demand charges encourage customers to manage their peak usage patterns, not just total consumption. It is similar to the previous example except it includes demand charges as well. The energy and demand charges vary by the amount of energy and demand consumed each month.

- Energy charges:
 - o 4.74 cents/kWh for first 20,000 kWh
 - o 4.24 cents/kWh for next 180,000 kWh
 - o 3.83 cents/kWh for additional kWh
- Demand charges:
 - o 5.38 dollars per kW for first 20 kW
 - o 4.23 dollars per kW for next 80 kW
 - o 3.60 dollars per kW for additional kW

To create this tariff structure, follow the steps below.

1. Create a new Tariff component:

Tariffs

☒ Include tariff analysis

Number of tariffs: 1

Tariff 1: EnergyPlus Example C - Block Energy and Demand Char

2. On the **Tariff** tab of the Tariff dialog, clear any fixed tariff data as shown below:

Tariffs				
Tariff	Charge	Qualify	Ratchet	User variables
General				
Tariff name	EnergyPlus Example C - Block Energy and Demand Charges			
Description	Example data			
Source				
Category	Electricity tariffs			
Output meter name	1-Electricity:Facility			
Conversion factor choice	2-kWh			
Schedules				
☐ Use 'Time of use period schedule'? ☐ Use 'Season schedule'? ☐ Use 'Month schedule'?				
Other Settings				
Demand window length	1-QuarterHour			
☐ Use 'Real time pricing charge schedule'? ☐ Use 'Customer baseline load schedule'? ☐ Use 'Group name'?				
Monthly service charge				
Monthly service charge or variable name	1-Charge value			
Input monthly service charge value (USD)	0.0000			
Minimum monthly charge				
Minimum monthly charge or variable name	1-Charge value			
Input minimum monthly charge value (USD)	0.0000			
Buy or Sell				
Buy or Sell	1-buyFromUtility			

3. On the **Charge** tab:

- Set the [Number of charges](#) to 2, the first covering energy, the other for demand.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **2-Block**.
 - Set [Charge variable name](#) to **BlockEnergyCharge**
 - Set [Source variable](#) to **totalEnergy**
 - Set the [Number of blocks](#) to 3
 - Set [Block size 1 value](#) to 20,000.
 - Set [Block 1 cost per unit value](#) to 0.0474 USD.
 - Set [Block size 2 value](#) to 180,000.
 - Set the [Block 2 cost per unit value](#) to 0.0424 USD.
 - Set the [Block size 3 variable name](#) to **remaining**.
 - Set the [Block 3 cost per unit value](#) to 0.0383 USD.
- Under the **Charge 2 settings** header:
 - Set [Simple or Block?](#) to **2-Block**.
 - Set [Charge variable name](#) to **BlockDemandCharge**
 - Set [Source variable](#) to **totalDemand**
 - Set [Category variable name](#) to **2-DemandCharges**
 - Set the [Number of blocks](#) to 3
 - Set [Block size 1 value](#) to 20.
 - Set [Block 1 cost per unit value](#) to 5.38 USD.
 - Set [Block size 2 value](#) to 80.
 - Set the [Block 2 cost per unit value](#) to 4.23 USD.
 - Set the [Block size 3 variable name](#) to **remaining**.
 - Set the [Block 3 cost per unit value](#) to 3.60 USD.

The screenshot below show how the charge tab should look when you have finished.

Tariffs	
Tariff	Charge
Qualify Ratchet User variables	
Charge Selection	
Number of charges	2
Charge 1 settings	
Simple or Block?	2-Block
Charge variable name	BlockEnergyCharge
Source variable	totalEnergy
Season	1-Annual
Category variable name	1-EnergyCharges
Remaining into variable	
Block size multiplier value or variable name	
Use value or variable name?	1-Value
Input multiplier value	1.000
Number of blocks	3
Block size 1 value	20000.00
Block 1 cost per unit value (USD)	0.047400
Block size 2 value	180000.00
Block 2 cost per unit value (USD)	0.042400
Block size 3 value or variable name	
Use block size value or variable name?	2-Block size variable name
Block size 3 variable name	remaining
Block 3 cost per unit value (USD)	0.038300
Charge 2 settings	
Simple or Block?	2-Block
Charge variable name	BlockDemandCharge
Source variable	totalDemand
Season	1-Annual
Category variable name	2-DemandCharges
Remaining into variable	
Block size multiplier value or variable name	
Use value or variable name?	1-Value
Input multiplier value	1.000
Number of blocks	3
Block size 1 value	20.00
Block 1 cost per unit value (USD)	5.380000
Block size 2 value	80.00
Block 2 cost per unit value (USD)	4.230000
Block size 3 value or variable name	
Use block size value or variable name?	2-Block size variable name
Block size 3 variable name	remaining
Block 3 cost per unit value (USD)	3.600000

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**

Timestamp: 2025-09-15 11:11:40

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	1211.76	0.00	0.00	1211.76
Cost per Total Building Area [\$/m2]	6.63	0.00	0.00	6.63
Cost per Net Conditioned Building Area [\$/m2]	6.63	0.00	0.00	6.63

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE C - BLOCK ENERGY AND DEMAND CHARGES	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	1211.76

Report: Tariff Report
For: ENERGYPLUS EXAMPLE C - BLOCK ENERGY AND DEMAND CHARGES
Timestamp: 2025-09-15 11:11:40
General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	59.79	52.76	57.53	68.04	80.41	72.27	87.84	82.10	71.67	68.37	56.05	57.59	814.42	87.84
DemandCharges (\$)	21.89	28.98	31.34	33.17	36.40	40.61	41.04	42.82	35.87	32.95	30.37	21.89	397.33	42.82
ServiceCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Basis (\$)	81.68	81.73	88.87	101.21	116.81	112.88	128.88	124.93	107.54	101.32	86.41	79.49	1211.76	128.88
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	81.68	81.73	88.87	101.21	116.81	112.88	128.88	124.93	107.54	101.32	86.41	79.49	1211.76	128.88
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	81.68	81.73	88.87	101.21	116.81	112.88	128.88	124.93	107.54	101.32	86.41	79.49	1211.76	128.88

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
BLOCKENERGYCHARGE (\$)	59.79	52.76	57.53	68.04	80.41	72.27	87.84	82.10	71.67	68.37	56.05	57.59	814.42	87.84	EnergyCharges
BLOCKDEMANDCHARGE (\$)	21.89	28.98	31.34	33.17	36.40	40.61	41.04	42.82	35.87	32.95	30.37	21.89	397.33	42.82	DemandCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1261.35	1113.04	1213.71	1435.50	1696.34	1524.70	1853.15	1732.16	1511.94	1442.50	1182.46	1215.03	17181.89	1853.15
TotalDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

TotalEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
TotalDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndShoulderDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakAndMidPeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndMidPeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndOffPeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

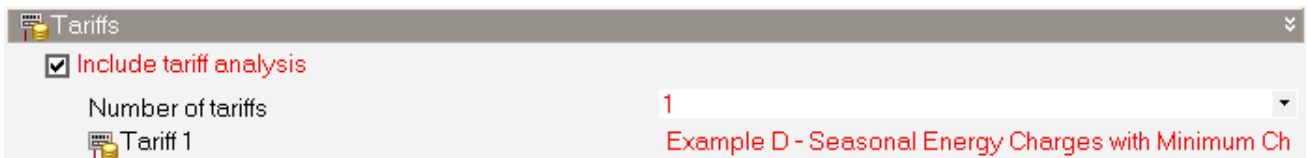
Example D - Seasonal Energy Charges With Minimum Charge

Utilities often have higher rates during peak seasons and minimum monthly charges to cover fixed costs. This example applies different block rates by season plus a minimum monthly fee. It is suitable for utilities with seasonal block structures (winter vs summer) and a guaranteed minimum bill.

- Minimum charge: \$7.00 per month
- Winter – October through May, energy charges:
 - 4.385 cents/kWh for first 650 kWh
 - 3.763 cents/kWh for next 350 kWh
 - 3.704 cents/kWh for over 1000 kWh
- Summer – June through September, energy charges:
 - 8.385 cents/kWh for first 650 kWh
 - 7.588 cents/kWh for next 350 kWh
 - 7.103 cents/kWh for over 1000 kWh

To create this tariff structure, follow the steps below.

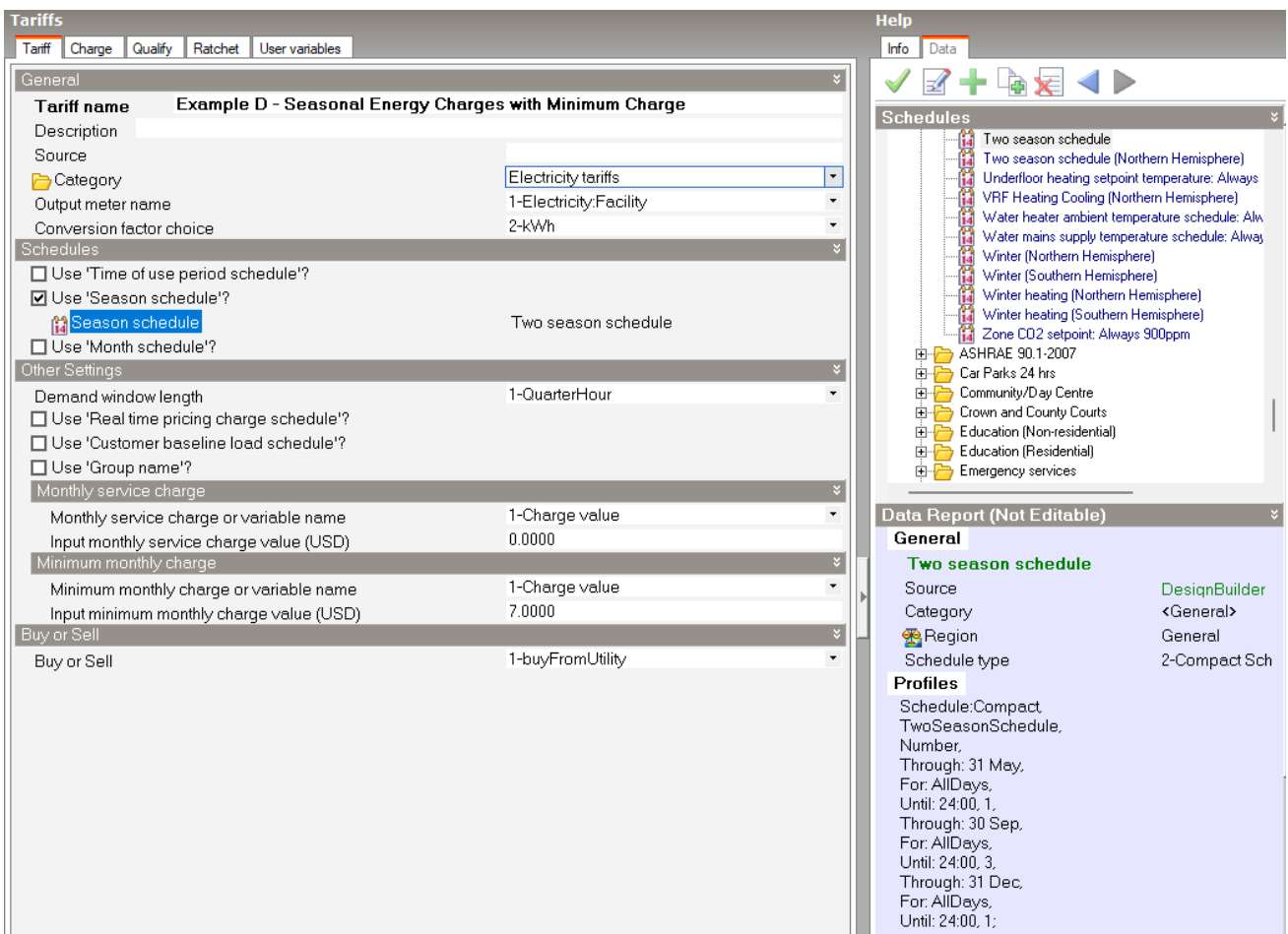
1. Create a new Tariff component:



2. On the **Tariff** tab of the Tariff dialog:

- Set the fixed monthly fee to \$7.00 per month using the [Input monthly service charge value](#) setting.
- Check the [Use 'Season schedule'](#) checkbox to reveal the Season schedule setting.
- Create a new [Seasonal schedule](#) by copying the "Two season schedule (Northern Hemisphere)" library schedule. Call the new schedule "Two season schedule". The library schedule already contains the right settings, so for this example, no further changes are needed to the copy. But for other definitions of summer (e.g. in the Southern hemisphere) you should make the necessary edits to the copied schedule.
- Select the newly created Seasonal schedule.

The Tariff tab should now look similar to the screenshot below.



3. On the **Charge** tab:

- Set the [Number of charges](#) to 2, the first covering winter, the other for summer.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **2-Block**.
 - Set [Charge variable name](#) to **WinterEnergyCost**
 - Set [Source variable](#) to **totalEnergy**
 - Set [Season](#) to **2-Winter**.
 - Set the [Number of blocks](#) to 3
 - Set [Block size 1 value](#) to 650.
 - Set [Block 1 cost per unit value](#) to 0.04385 USD.
 - Set [Block size 2 value](#) to 350.

- o Set the [Block 2 cost per unit value](#) to 0.03763 USD.
- o Set the [Block size 3 variable name](#) to **remaining**.
- o Set the [Block 3 cost per unit value](#) to 0.03704 USD.
- Under the **Charge 2 settings** header:
 - o Set [Simple or Block?](#) to **2-Block**.
 - o Set [Charge variable name](#) to **SummerEnergyCost**
 - o Set [Source variable](#) to **totalEnergy**
 - o Set [Season](#) to **4-Summer**.
 - o Set the [Number of blocks](#) to 3
 - o Set [Block size 1 value](#) to 650.
 - o Set [Block 1 cost per unit value](#) to 0.08385 USD.
 - o Set [Block size 2 value](#) to 350.
 - o Set the [Block 2 cost per unit value](#) to 0.07588 USD.
 - o Set the [Block size 3 variable name](#) to **remaining**.
 - o Set the [Block 3 cost per unit value](#) to 0.07103 USD.

The screenshot below show how the charge tab should look when you have finished.

Tariffs

Tariff

Charge

Qualify

Ratchet

User variables

Charge Selection

Number of charges

2

Charge 1 settings

Simple or Block?

2-Block

Charge variable name

WinterEnergyCost

Source variable

totalEnergy

Season

2-Winter

Category variable name

1-EnergyCharges

Remaining into variable

Block size multiplier value or variable name

Use value or variable name?

1-Value

Input multiplier value

1.000

Number of blocks

3

Block size 1 value

650.00

Block 1 cost per unit value (USD)

0.043850

Block size 2 value

350.00

Block 2 cost per unit value (USD)

0.037630

Block size 3 value or variable name

Use block size value or variable name?

2-Block size variable name

Block size 3 variable name

remaining

Block 3 cost per unit value

0.037040

Charge 2 settings

Simple or Block?

2-Block

Charge variable name

SummerEnergyCost

Source variable

totalEnergy

Season

4-Summer

Category variable name

1-EnergyCharges

Remaining into variable

Block size multiplier value or variable name

Use value or variable name?

1-Value

Input multiplier value

1.000

Number of blocks

3

Block size 1 value

650.00

Block 1 cost per unit value (USD)

0.083850

Block size 2 value

350.00

Block 2 cost per unit value (USD)

0.075880

Block size 3 value or variable name

Use block size value or variable name?

2-Block size variable name

Block size 3 variable name

remaining

Block 3 cost per unit value

0.071030

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**
 Timestamp: **2025-09-11 11:38:13**

Annual Cost

- 580 -

	Electricity	Natural Gas	Other	Total
Cost [\$]	937.46	0.00	0.00	937.46
Cost per Total Building Area [\$/m2]	5.13	0.00	0.00	5.13
Cost per Net Conditioned Building Area [\$/m2]	5.13	0.00	0.00	5.13

Tariff Summary

	Selected	Qualified	Meter	Buy or Group Sell	Annual Cost (\$)
ENERGYPLUS EXAMPLE D - SEASONAL ENERGY CHARGES WITH MINIMUM CHARGE	Yes	Yes	ELECTRICITY:FACILITY	Buy (none)	937.46

Report: **Tariff Report**For: **ENERGYPLUS EXAMPLE D - SEASONAL ENERGY CHARGES WITH MINIMUM CHARGE**Timestamp: **2025-09-11 11:38:13****General**

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	51.33	45.87	49.55	57.72	67.33	118.08	141.34	132.85	117.29	58.05	48.44	49.61	937.46	141.34
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Basis (\$)	51.33	45.87	49.55	57.72	67.33	118.08	141.34	132.85	117.29	58.05	48.44	49.61	937.46	141.34
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	51.33	45.87	49.55	57.72	67.33	118.08	141.34	132.85	117.29	58.05	48.44	49.61	937.46	141.34
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	51.33	45.87	49.55	57.72	67.33	118.08	141.34	132.85	117.29	58.05	48.44	49.61	937.46	141.34

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
WINTERENERGYCOST (\$)	51.33	45.87	49.55	57.72	67.33	0.00	0.00	0.00	0.00	58.05	48.44	49.61	427.90	67.33	EnergyCharges
SUMMERENERGYCOST (\$)	0.00	0.00	0.00	0.00	0.00	118.08	141.34	132.85	117.29	0.00	0.00	0.00	509.56	141.34	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94

PeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	8.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	4.00	1.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	4.00	1.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	8.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndShoulderDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakAndMidPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndMidPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example E – KWh/kW Energy Charges

This example shows how to model a more sophisticated rate structure which encourages efficient energy use by linking energy pricing to monthly demand levels. Higher load factors (more consistent energy use) receive better rates. Energy blocks are multiplied by monthly peak demand to reflect load factor incentives. Such tariffs may be offered when customers with consistent energy use are more cost-effective to serve.

- Customer charge: \$14.00 per month
- Energy charges:
 - o 8.756 cents/kWh for all consumption not greater than 200 hours times the demand
 - o 6.812 cents/kWh for all consumption in excess of 200 hours and not greater than 400 hours times the demand
 - o 5.029 cents/kWh for all consumption in excess of 400 hours times the demand.

This tariff uses a single Charge object. The monthly charge is defined on the Tariff tab. In this case the Block size multiplier is set to the **totalDemand** variable.

To create this tariff structure, follow the steps below.

1. Create a new Tariff component:

Tariffs

☒ Include tariff analysis

Number of tariffs

1

Tariff 1

EnergyPlus Example E - kWh per kW Energy Charges

2. On the **Tariff** tab of the Tariff dialog:

- Set the fixed monthly fee to \$14.00 per month using the [Input monthly service charge value](#) setting.

The Tariff tab should now look similar to the screenshot below:

Tariffs	
Tariff Charge Qualify Ratchet User variables	
General	
Tariff name	EnergyPlus Example E - kWh per kW Energy Charges
Description	Example data
Source	
Category	Electricity tariffs
Output meter name	1-Electricity:Facility
Conversion factor choice	2-kWh
Schedules	
☐ Use 'Time of use period schedule'?	
☐ Use 'Season schedule'?	
☐ Use 'Month schedule'?	
Other Settings	
Demand window length	1-QuarterHour
☐ Use 'Real time pricing charge schedule'?	
☐ Use 'Customer baseline load schedule'?	
☐ Use 'Group name'?	
Monthly service charge	
Monthly service charge or variable name	1-Charge value
Input monthly service charge value (USD)	14.0000
Minimum monthly charge	
Minimum monthly charge or variable name	1-Charge value
Input minimum monthly charge value (USD)	0.0000
Buy or Sell	
Buy or Sell	1-buyFromUtility

3. On the **Charge** tab:

- Set the [Number of charges](#) to 1 to cover energy use.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **2-Block**.
 - Set [Charge variable name](#) to **BlockEnergyCharge**
 - Set [Source variable](#) to **totalEnergy**
 - Under the [Block size multiplier or variable name](#) header, set [Use value or variable name](#) to **totalDemand** to indicate that block size values entered below are to be multiplied by the demand in kW.
 - Set the [Number of blocks](#) to 3
 - Set [Block size 1 value](#) to 200.
 - Set [Block 1 cost per unit value](#) to 0.08756 USD.
 - Set [Block size 2 value](#) to 200.
 - Set the [Block 2 cost per unit value](#) to 0.06812 USD.
 - Set the [Block size 3 variable name](#) to **remaining**.
 - Set the [Block 3 cost per unit value](#) to 0.05029 USD.

The Charge tab should now look similar to the screenshot below.

Tariffs

Tariff

Charge

Qualify

Ratchet

User variables

Charge Selection

Number of charges

1

Charge 1 settings

Simple or Block?

2-Block

Charge variable name

BlockEnergyCharge

Source variable

totalEnergy

Season

1-Annual

Category variable name

1-EnergyCharges

Remaining into variable

Block size multiplier value or variable name

Use value or variable name?

2-Variable name

Input multiplier variable name

totalDemand

Number of blocks

3

Block size 1 value

200.00

Block 1 cost per unit value (USD)

0.087560

Block size 2 value

200.00

Block 2 cost per unit value (USD)

0.068120

Block size 3 value or variable name

Use block size value or variable name?

2-Block size variable name

Block size 3 variable name

remaining

Block 3 cost per unit value

0.050290

Summary Output

Report: **Economics Results Summary Report**

For: **Entire Facility**

Timestamp: **2025-09-11 11:30:42**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	1623.87	0.00	0.00	1623.87
Cost per Total Building Area [\$/m2]	8.88	0.00	0.00	8.88
Cost per Net Conditioned Building Area [\$/m2]	8.88	0.00	0.00	8.88

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE E - KWH PER KW ENERGY CHARGES	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	1623.87

Report: **Tariff Report**

For: **ENERGYPLUS EXAMPLE E - KWH PER KW ENERGY CHARGES**

Timestamp: **2025-09-11 11:30:42**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

- 584 -

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	101.70	97.00	105.23	121.56	141.53	132.89	155.48	148.67	128.76	122.03	102.48	98.54	1455.87	155.48
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	14.00	14.00	14.00	14.00	14.00	14.00	14.00	14.00	14.00	14.00	14.00	14.00	168.00	14.00
Basis (\$)	115.70	111.00	119.23	135.56	155.53	146.89	169.48	162.67	142.76	136.03	116.48	112.54	1623.87	169.48
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	115.70	111.00	119.23	135.56	155.53	146.89	169.48	162.67	142.76	136.03	116.48	112.54	1623.87	169.48
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	115.70	111.00	119.23	135.56	155.53	146.89	169.48	162.67	142.76	136.03	116.48	112.54	1623.87	169.48

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
BLOCKENERGYCHARGE (\$)	101.7	97.0	105.2	121.5	141.5	132.8	155.4	148.6	128.7	122.0	102.4	98.5	1455.8	155.4	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndShoulderDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakAndMidPeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndMidPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

PeakAndOffPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example F – Seasonal Time Of Use Energy

Utilities face different costs throughout the day and year. Time-of-use rates pass these cost variations on to customers, encouraging usage during low-cost periods and discouraging use during high-cost periods. For example, tariffs sometimes have higher costs for energy consumed during the daytime than at night. In this example, four simple rates combine to create on-peak/off-peak and summer/winter distinctions. This approach is suitable for time-of-use tariffs where rates vary by both season and time of day.

- Monthly charge: \$37.75 per month
- Winter – October through May
 - o On Peak: 8.315 cents/kWh
 - o Off-Peak: 2.420 cents/kWh
- Summer – June through September
 - o On Peak: 14.009 cents/kWh
 - o Off-Peak: 6.312 cents/kWh

The on-peak period is defined as the hours starting at 10am and ending at 7pm, Monday through Friday for June through September and 3pm to 10pm Monday through Friday for October through May. All other hours are considered off-peak.

The tariff is only applicable for customers that use 50KW for at least one month of the year. This restriction is implemented by entering details on the Qualify tab.

This tariff uses four different Charge:Simple objects to capture the variation of energy cost with time.

To create this tariff structure, follow the steps below.

1. Create a new Tariff component:

Tariffs

☒ Include tariff analysis

Number of tariffs: 1

Tariff 1: EnergyPlus Example F - Seasonal Time of Use Energy

2. On the **Tariff** tab of the Tariff dialog:

- Set the fixed monthly fee to \$37.75 per month using the [Input monthly service charge value](#) setting.
- Check the [Use 'Time of use period schedule'?](#) checkbox.
- Use the [Time of use period schedule](#), to define the on peak and off peak times. You can base your schedule on the library "Time of day schedule" by copying it, renaming the new schedule "TimeOfDaySchedule" and making any changes needed. As it happens the schedule shouldn't need any changes. It should look like the screenshot below.

Edit schedule - TimeOfDaySchedule

Schedules

General

General

Name TimeOfDaySchedule

Description

Source DesignBuilder

Category <General>

Region General

Schedule type 2-Compact Schedule

Profiles

Schedule: Compact,
TimeOfDaySchedule,
Number,
Through: 31 May,
For: AllDays,
Until: 15:00, 3,
Until: 22:00, 1,
Until: 24:00, 3,
Through: 30 Sep,
For: AllDays,
Until: 10:00, 3,
Until: 19:00, 1,
Until: 24:00, 3,
Through: 31 Dec,
For: AllDays,
Until: 15:00, 3,
Until: 22:00, 1,
Until: 24:00, 3;

- Check the [Use 'Season schedule'](#) checkbox to reveal the Season schedule setting.
- Create a new [Seasonal schedule](#) by copying the "Two season schedule (Northern Hemisphere)" library schedule. Call the new schedule "Two season schedule". The library schedule already contains the right settings, so for this example, no further changes are needed to the copy. But for other definitions of summer (e.g. in the Southern hemisphere) you should make the necessary edits to the copied schedule.
- Select the newly created Seasonal schedule.

The Tariff tab should now look similar to the screenshot below:

Tariffs				
Tariff	Charge	Qualify	Ratchet	User variables
General				
Tariff name	EnergyPlus Example F - Seasonal Time of Use Energy			
Description	Example data			
Source				
Category	Electricity tariffs			
Output meter name	1-Electricity:Facility			
Conversion factor choice	2-kWh			
Schedules				
☒ Use 'Time of use period schedule'?				
Time of use period schedule	TimeOfDaySchedule			
☒ Use 'Season schedule'?				
Season schedule	Two season schedule			
☐ Use 'Month schedule'?				
Other Settings				
Demand window length	1-QuarterHour			
☐ Use 'Real time pricing charge schedule'?				
☐ Use 'Customer baseline load schedule'?				
☐ Use 'Group name'?				
Monthly service charge				
Monthly service charge or variable name	1-Charge value			
Input monthly service charge value (USD)	37.7500			
Minimum monthly charge				
Minimum monthly charge or variable name	1-Charge value			
Input minimum monthly charge value (USD)	0.0000			
Buy or Sell				
Buy or Sell	1-buyFromUtility			

3. On the **Charge** tab:

- Set the [Number of charges](#) to 4 to cover the summer/winter on-peak/off-peak cases.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **1-Simple**.
 - Set [Charge variable name](#) to **SummerOnPeak**
 - Set [Source variable](#) to **peakEnergy**
 - Set [Season](#) to **4-Summer**
 - Set [Category variable name](#) to **1-EnergyCharges**
 - Enter the [Input cost per unit value](#) as 0.14009
- Under the **Charge 2 settings** header:
 - Set [Simple or Block?](#) to **1-Simple**.
 - Set [Charge variable name](#) to **SummerOffPeak**
 - Set [Source variable](#) to **offPeakEnergy**
 - Set [Season](#) to **4-Summer**
 - Set [Category variable name](#) to **1-EnergyCharges**
 - Enter the [Input cost per unit value](#) as 0.06312
- Under the **Charge 3 settings** header:
 - Set [Simple or Block?](#) to **1-Simple**.
 - Set [Charge variable name](#) to **WinterOnPeak**
 - Set [Source variable](#) to **peakEnergy**
 - Set [Season](#) to **2-Winter**
 - Set [Category variable name](#) to **1-EnergyCharges**
 - Enter the [Input cost per unit value](#) as 0.08315
- Under the **Charge 4 settings** header:

- o Set [Simple or Block?](#) to **1-Simple**.
- o Set [Charge variable name](#) to **WinterOffPeak**
- o Set [Source variable](#) to **offPeakEnergy**
- o Set [Season](#) to **2-Winter**
- o Set [Category variable name](#) to **1-EnergyCharges**
- o Enter the [Input cost per unit value](#) as 0.02420

The Charge tab should now look similar to the screenshot below.

Tariffs	
Tariff	Charge
Qualify Ratchet User variables	
Charge Selection	
Number of charges	4
Charge 1 settings	
Simple or Block?	1-Simple
Charge variable name	SummerOnPeak
Source variable	peakEnergy
Season	4-Summer
Category variable name	1-EnergyCharges
Cost per unit value or variable name	
Use value or variable name?	1-Value
Input cost per unit value (USD)	0.140090
Charge 2 settings	
Simple or Block?	1-Simple
Charge variable name	SummerOffPeak
Source variable	offPeakEnergy
Season	4-Summer
Category variable name	1-EnergyCharges
Cost per unit value or variable name	
Use value or variable name?	1-Value
Input cost per unit value (USD)	0.063120
Charge 3 settings	
Simple or Block?	1-Simple
Charge variable name	WinterOnPeak
Source variable	peakEnergy
Season	2-Winter
Category variable name	1-EnergyCharges
Cost per unit value or variable name	
Use value or variable name?	1-Value
Input cost per unit value (USD)	0.083150
Charge 4 settings	
Simple or Block?	1-Simple
Charge variable name	WinterOffPeak
Source variable	offPeakEnergy
Season	2-Winter
Category variable name	1-EnergyCharges
Cost per unit value or variable name	
Use value or variable name?	1-Value
Input cost per unit value (USD)	0.024200

On the **Qualify** tab:

- Set the [Number of qualify checks](#) to 1 to allow the minimum demand requirement.
- Under the **Qualify check 1 settings** header:
 - Set the [Name](#) to **MinDemand**
 - Set the [Variable name](#) to **2-totalDemand**
 - Set [Qualify type](#) to **1-Minimum**
 - Enter the [Input threshold value](#) as 50
 - Set [Season](#) to **1-Annual**
 - Set [Threshold test](#) to **1-Count**
 - Check the [Input 'Number of months'?](#) checkbox
 - Set the [Number of months](#) to 1.

The Qualify tab should now look similar to the screenshot below.

Tariffs

Tariff

Charge

Qualify

Ratchet

User variables

Qualify check

Number of qualify checks1

Qualify check 1 settings

Name

MinDemand

Variable name

2-totalDemand

Qualify type

1-Minimum

Threshold value or variable name

Use value or variable name?1-Value

Input threshold value

50.00

Season

1-Annual

Threshold Test

1-Count

☒ Input 'Number of months'?

Number of months

1

Summary Output

Report: **Economics Results Summary Report**
For: **Entire Facility**
Timestamp: **2025-09-11 11:27:02**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	0.00	0.00	0.00	0.00
Cost per Total Building Area [\$/m2]	0.00	0.00	0.00	0.00
Cost per Net Conditioned Building Area [\$/m2]	0.00	0.00	0.00	0.00

Tariff Summary

	Selected Qualified		Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE F - SEASONAL TIME OF USE ENERGY	No	No	ELECTRICITY:FACILITY	Buy	(none)	1707.55

Report: **Tariff Report**
For: **ENERGYPLUS EXAMPLE F - SEASONAL TIME OF USE ENERGY**
Timestamp: **2025-09-11 11:27:02**

General

Parameter
Meter ELECTRICITY:FACILITY
Selected No

Group (none)
 Qualified No
 Disqualifier MINDEMAND
 Computation automatic
 Units kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	55.72	49.12	53.82	65.19	76.57	180.73	218.27	204.99	180.22	64.30	52.00	53.64	1254.55	218.27
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	453.00	37.75
Basis (\$)	93.47	86.87	91.57	102.94	114.32	218.48	256.02	242.74	217.97	102.05	89.75	91.39	1707.55	256.02
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	93.47	86.87	91.57	102.94	114.32	218.48	256.02	242.74	217.97	102.05	89.75	91.39	1707.55	256.02
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	93.47	86.87	91.57	102.94	114.32	218.48	256.02	242.74	217.97	102.05	89.75	91.39	1707.55	256.02

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
SUMMERONPEAK (\$)	0.00	0.00	0.00	0.00	0.00	154.19	184.88	174.44	154.53	0.00	0.00	0.00	668.04	184.88	EnergyCharges
SUMMEROFFPEAK (\$)	0.00	0.00	0.00	0.00	0.00	26.55	33.38	30.54	25.69	0.00	0.00	0.00	116.17	33.38	EnergyCharges
WINTERONPEAK (\$)	35.55	31.27	34.52	43.02	50.22	0.00	0.00	0.00	0.00	41.47	32.98	34.21	303.24	50.22	EnergyCharges
WINTEROFFPEAK (\$)	20.16	17.84	19.30	22.16	26.34	0.00	0.00	0.00	0.00	22.83	19.02	19.43	167.10	26.34	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
PeakEnergy	427.60	376.11	415.12	517.41	604.00	1100.63	1319.75	1245.23	1103.07	498.69	396.58	411.45	8415.64	1319.75
OffPeakEnergy	833.07	737.28	797.62	915.83	1088.60	420.56	528.90	483.88	407.05	943.47	786.01	802.90	8745.18	1088.60

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
MINDEMAND	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakEnergy	427.60	376.11	415.12	517.41	604.00	1100.63	1319.75	1245.23	1103.07	498.69	396.58	411.45	8415.64	1319.75
PeakDemand	4.05	5.17	5.79	6.15	6.75	7.53	7.60	7.94	6.66	6.11	5.10	4.05	72.90	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	833.07	737.28	797.62	915.83	1088.60	420.56	528.90	483.88	407.05	943.47	786.01	802.90	8745.18	1088.60
OffPeakDemand	4.07	5.44	5.82	6.15	6.73	6.25	6.54	6.62	5.87	6.12	5.64	4.07	69.32	6.73
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	0.00	0.00	0.00	0.0047	0.0173	1.28	1.06	1.33	0.7863	0.00	0.00	0.00	4.47	1.33
OffPeakExceedsPeak	0.0214	0.2718	0.0247	0.00	0.00	0.00	0.00	0.00	0.00	0.0091	0.5367	0.0220	0.89	0.54
PeakExceedsMidPeak	4.05	5.17	5.79	6.15	6.75	7.53	7.60	7.94	6.66	6.11	5.10	4.05	72.90	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.05	5.17	5.79	6.15	6.75	7.53	7.60	7.94	6.66	6.11	5.10	4.05	72.90	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	8.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	4.00	1.00

IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	4.00	1.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	0.00	0.00	0.00	0.00	1.00	1.00	1.00	8.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	427.60	376.11	415.12	517.41	604.00	1100.63	1319.75	1245.23	1103.07	498.69	396.58	411.45	8415.64	1319.75
PeakAndShoulderDemand	4.05	5.17	5.79	6.15	6.75	7.53	7.60	7.94	6.66	6.11	5.10	4.05	72.90	7.94
PeakAndMidPeakEnergy	427.60	376.11	415.12	517.41	604.00	1100.63	1319.75	1245.23	1103.07	498.69	396.58	411.45	8415.64	1319.75
PeakAndMidPeakDemand	4.05	5.17	5.79	6.15	6.75	7.53	7.60	7.94	6.66	6.11	5.10	4.05	72.90	7.94
ShoulderAndOffPeakEnergy	833.07	737.28	797.62	915.83	1088.60	420.56	528.90	483.88	407.05	943.47	786.01	802.90	8745.18	1088.60
ShoulderAndOffPeakDemand	4.07	5.44	5.82	6.15	6.73	6.25	6.54	6.62	5.87	6.12	5.64	4.07	69.32	6.73
PeakAndOffPeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example G – Blocks Within Blocks

Some utilities create complex rate structures that combine load factor incentives with traditional tiered pricing. This requires nested block calculations. Example G shows how to implement nested block structures by first allocating a demand-scaled block, then applying standard blocks to the resulting subsets. It is suitable for modelling complex tariffs combining scaled demand blocks and traditional kWh tiers for advanced load factor management.

- Monthly charge: \$35 per month
- Energy charges:
 - For all consumption not greater than 200 hours times the demand use:
 - o 10.32 cents/kWh for the first 1000 kWh
 - o 7.43 cents/kWh for the next 4000 kWh
 - o 6.23 cents/kWh for the next 5000 kWh
 - o 4.27 cents/kWh for the remaining kWh less than 200 hours times the demand
 - For all consumption in excess of 200 hours and not greater than 400 hours times the demand, use:
 - o 6.82 cents/kWh
 - For all consumption in excess of 400 hours times the demand, use:
 - o 5.03 cents/kWh

To set up “block within a block”, Charge:Block settings are made to separate out the first 200 kWh/kW. The “EnergyFirst200kWhPerkW” charge performs this. It uses demand to multiply the first block size by 200 and the cost for the block is simply 1 since it passes through the energy to the variable “EnergyFirst200kWhPerkW”. The remaining energy goes into the **restOfEnergy** variable as specified in the [Remaining into variable](#). After this has been evaluated we have two new variables that hold energy. Each of these variables are then separately used in Charge:Block objects to evaluate the different parts of the example tariff. By using the **Remaining into variable** along with the concept of variables, many very complex tariffs may be modelled.

To create this tariff structure, follow the steps below.

1. Create a new Tariff component:

Tariffs

☒ Include tariff analysis

Number of tariffs

1

Tariff 1

EnergyPlus Example G - Blocks within Blocks

2. On the **Tariff** tab of the Tariff dialog:

- Set the fixed monthly fee to \$35 per month using the [Input monthly service charge value](#) setting.

The Tariff tab should now look similar to the screenshot below:

Tariffs				
Tariff	Charge	Qualify	Ratchet	User variables
General				
Tariff name	EnergyPlus Example G - Blocks within Blocks			
Description	Example data			
Source				
Category	Electricity tariffs			
Output meter name	1-Electricity:Facility			
Conversion factor choice	2-kWh			
Schedules				
☐ Use 'Time of use period schedule'?				
☐ Use 'Season schedule'?				
☐ Use 'Month schedule'?				
Other Settings				
Demand window length	1-QuarterHour			
☐ Use 'Real time pricing charge schedule'?				
☐ Use 'Customer baseline load schedule'?				
☐ Use 'Group name'?				
Monthly service charge				
Monthly service charge or variable name	1-Charge value			
Input monthly service charge value (USD)	35.0000			
Minimum monthly charge				
Minimum monthly charge or variable name	1-Charge value			
Input minimum monthly charge value (USD)	0.0000			
Buy or Sell				
Buy or Sell	1-buyFromUtility			

3. On the **Charge** tab:

- Set the [Number of charges](#) to 3.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **2-Block**.
 - Set [Charge variable name](#) to **EnergyFirst200kWhPerkW**
 - Set [Source variable](#) to **totalEnergy**
 - Set [Season](#) to **1-Annual**
 - Set [Category variable name](#) to **10-NotIncluded**
 - Set [Remaining into variable](#) to **restOfEnergy**
 - Set [Use value or variable name](#) to **2-Variable name**
 - Enter the [Input multiplier variable name](#) as **totalDemand**
 - Set [Number of blocks](#) to 1
 - Set [Use block size value or variable name](#) to **1-Block size value**
 - Set [Block size 1 value](#) to 200
 - Set [Block 1 cost per unit value](#) to 1
- Under the **Charge 2 settings** header:
 - Set [Simple or Block?](#) to **2-Block**.
 - Set [Charge variable name](#) to **CostOfFirst200kWhPerkW**
 - Set [Source variable](#) to **EnergyFirst200kWhPerkW**

- o Set [Season](#) to **1-Annual**
- o Set [Category variable name](#) to **1-EnergyCharges**
- o Leave [Remaining into variable](#) blank
- o Set [Use value or variable name](#) to **1-Value**
- o Set the [Input multiplier value](#) to 1
- o Set [Number of blocks](#) to 4
- o Set [Use block size value or variable name](#) to **1-Block size value**
- o Set [Block size 1 value](#) to 1000
- o Set [Block 1 cost per unit value](#) to 0.103200
- o Set [Block size 2 value](#) to 4000
- o Set [Block 2 cost per unit value](#) to 0.074300
- o Set [Block size 3 value](#) to 5000
- o Set [Block 3 cost per unit value](#) to 0.062300
- o For Block 4, set [Use block size value or variable name](#) to **2-Block size variable name**
- o Set [Block size 4 variable name](#) to **remaining**
- o Set [Block 4 cost per unit value](#) to 0.042700
- Under the **Charge 3 settings** header:
 - o Set [Simple or Block?](#) to **2-Block**.
 - o Set [Charge variable name](#) to **CostOfRestOfEnergy**
 - o Set [Source variable](#) to **restOfEnergy**
 - o Set [Season](#) to **1-Annual**
 - o Set [Category variable name](#) to **1-EnergyCharges**
 - o Leave [Remaining into variable](#) blank
 - o Set [Use value or variable name](#) to **2-Variable name**
 - o Set the [Input multiplier variable name](#) to **totalDemand**
 - o Set [Number of blocks](#) to 2
 - o Set [Use block size value or variable name](#) to **1-Block size value**
 - o Set [Block size 1 value](#) to 200
 - o Set [Block 1 cost per unit value](#) to 0.068200
 - o For Block 2, set [Use block size value or variable name](#) to **2-Block size variable name**
 - o Set [Block size 2 variable name](#) to **remaining**
 - o Set [Block 2 cost per unit value](#) to 0.050300

The Charge tab should now look similar to the screenshot below.

Tariffs	
Tariff	Charge
Qualify Ratchet User variables	
Charge Selection	
Number of charges	3
Charge 1 settings	
Simple or Block?	2-Block
Charge variable name	EnergyFirst200kWhPerkW
Source variable	totalEnergy
Season	1-Annual
Category variable name	10-NotIncluded
Remaining into variable	restOfEnergy
Block size multiplier value or variable name	
Use value or variable name?	2-Variable name
Input multiplier variable name	totalDemand
Number of blocks	1
Block size 1 value or variable name	
Use block size value or variable name?	1-Block size value
Block size 1 value	200.00000
Block 1 cost per unit value	1.000000
Charge 2 settings	
Simple or Block?	2-Block
Charge variable name	CostOfFirst200kWhPerkW
Source variable	EnergyFirst200kWhPerkW
Season	1-Annual
Category variable name	1-EnergyCharges
Remaining into variable	
Block size multiplier value or variable name	
Use value or variable name?	1-Value
Input multiplier value	1.000
Number of blocks	4
Block size 1 value	1000.00
Block 1 cost per unit value (USD)	0.103200
Block size 2 value	4000.00
Block 2 cost per unit value (USD)	0.074300
Block size 3 value	5000.00
Block 3 cost per unit value (USD)	0.062300
Block size 4 value or variable name	
Use block size value or variable name?	2-Block size variable name
Block size 4 variable name	remaining
Block 4 cost per unit value	0.042700
Charge 3 settings	
Simple or Block?	2-Block
Charge variable name	CostOfRestOfEnergy
Source variable	restOfEnergy
Season	1-Annual
Category variable name	1-EnergyCharges
Remaining into variable	
Block size multiplier value or variable name	
Use value or variable name?	2-Variable name
Input multiplier variable name	totalDemand
Number of blocks	2
Block size 1 value	200.00
Block 1 cost per unit value (USD)	0.068200
Block size 2 value or variable name	
Use block size value or variable name?	2-Block size variable name
Block size 2 variable name	remaining
Block 2 cost per unit value	0.050300

Summary Output

Report: **Economics Results Summary Report**
For: **Entire Facility**
Timestamp: **2025-09-11 11:18:57**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	2016.42	0.00	0.00	2016.42
Cost per Total Building Area [\$/m2]	11.03	0.00	0.00	11.03
Cost per Net Conditioned Building Area [\$/m2]	11.03	0.00	0.00	11.03

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE G - BLOCKS WITHIN BLOCKS	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	2016.42

Report: **Tariff Report**
For: **ENERGYPLUS EXAMPLE G - BLOCKS WITHIN BLOCKS**
Timestamp: **2025-09-11 11:18:57**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	114.46	111.47	118.70	134.15	152.57	141.83	164.25	156.52	140.01	134.72	116.43	111.30	1596.42	164.25
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	35.00	35.00	35.00	35.00	35.00	35.00	35.00	35.00	35.00	35.00	35.00	35.00	420.00	35.00
Basis (\$)	149.46	146.47	153.70	169.15	187.57	176.83	199.25	191.52	175.01	169.72	151.43	146.30	2016.42	199.25
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	149.46	146.47	153.70	169.15	187.57	176.83	199.25	191.52	175.01	169.72	151.43	146.30	2016.42	199.25
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	149.46	146.47	153.70	169.15	187.57	176.83	199.25	191.52	175.01	169.72	151.43	146.30	2016.42	199.25

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct
ENERGYFIRST200KWHPERKW (\$)	813.74	1088.49	1163.30	1230.68	1349.17	1505.30	1520.17	1588.64	1331.93	1223.81
COSTOFFIRST200KWHPERKW (\$)	83.98	109.77	115.33	120.34	129.14	140.74	141.85	146.94	127.86	119.83
COSTOFRESTOFENERGY (\$)	30.48	1.70	3.37	13.82	23.42	1.08	22.40	9.58	12.15	14.89

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
ENERGYFIRST200KWHPERKW	813.74	1088.4	1163.3	1230.6	1349.1	1505.3	1520.1	1588.6	1331.9	1223.8	1127.8	813.75	14756.8	1588.6
RESTOFENERGY	446.93	24.90	49.44	202.57	343.43	15.90	328.48	140.47	178.18	218.35	54.73	400.60	2403.98	446.93

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndShoulderDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakAndMidPeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndMidPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example H – Real Time Pricing

Some utilities offer hourly pricing that varies based on real-time market conditions which are typically announced day-ahead. Hourly energy prices are imported from a schedule, with no predefined charges. This example shows how to model real-time pricing tariffs where the utility provides hourly prices in advance.

To model this type of utility rate a schedule is used that contains the prices on an hourly basis. The [Real time pricing charge schedule](#) contains the prices and the [Customer baseline load schedule](#) setting is used to set the schedule for the customer baseline load. Not all utilities use a customer baseline load in which case this schedule does not need to be entered. The Period, Season and Month Schedules are not needed for these tariffs unless Charges are used.

The example real time pricing schedule results in the same energy cost as [Example F – Seasonal Time of Use Energy](#). Note that this is only an example and usually for real time pricing the schedule values would vary throughout the year.

To create this tariff structure, follow the steps below.

1. Create a new Tariff component:



Tariffs

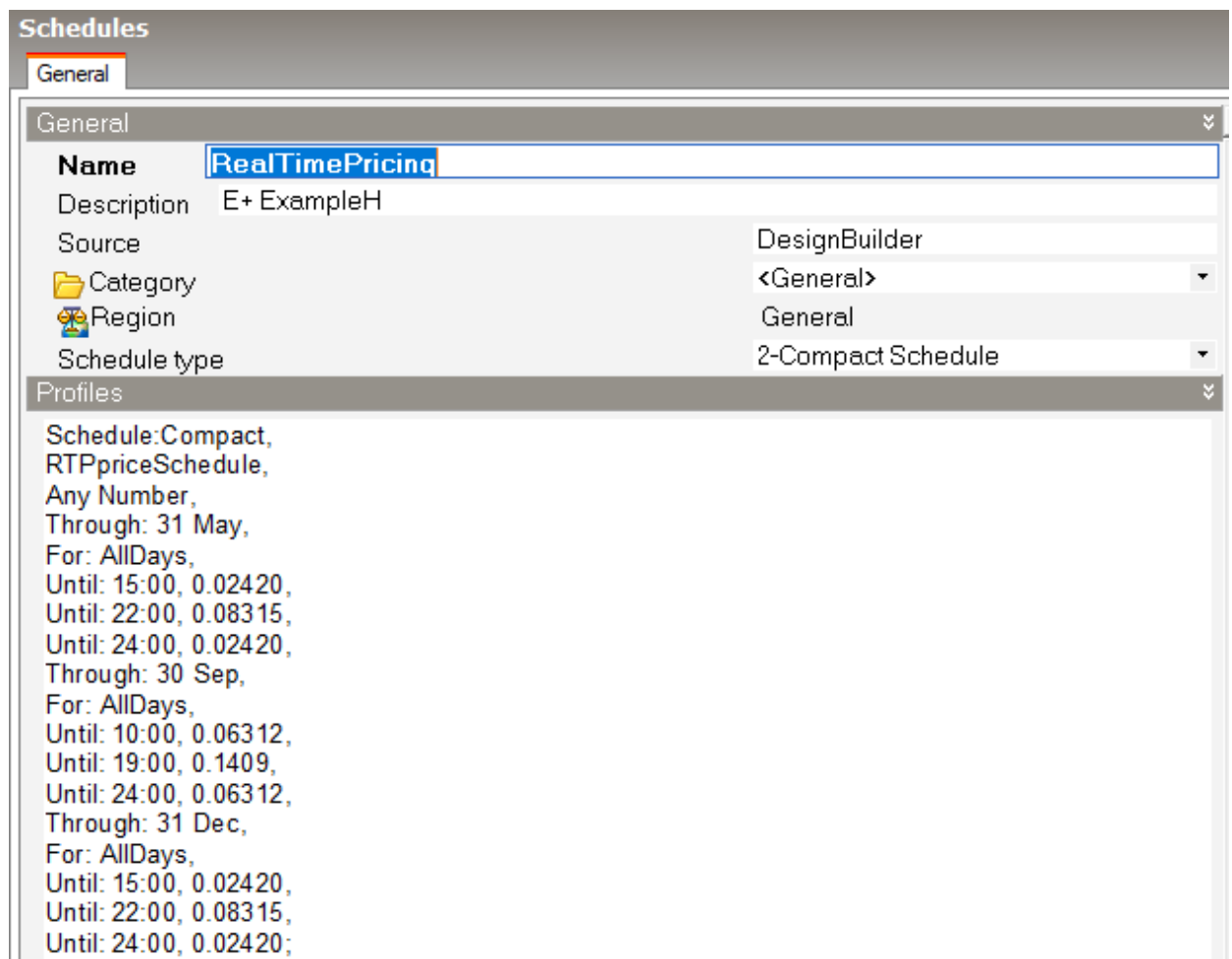
☒ Include tariff analysis

Number of tariffs: 1

Tariff 1: EnergyPlus Example H - Real Time Pricing (RTP)

2. On the **Tariff** tab of the Tariff dialog:

- Set the fixed monthly fee to \$37.75 per month using the [Input monthly service charge value](#) setting.
- Check the [Use 'Real time pricing charge schedule'?](#) checkbox.
- Click on the [Real time pricing charge schedule](#) control and make a copy of the library schedule: "Real time pricing schedule". Name the copy of the schedule "RealTimePricing" and select it. The copied library schedule should already contain the right settings for this example, so for our purposes, no further changes are needed to the copy. The schedule should look like the screenshot below. Of course, for your project work, you would make the necessary edits to the copied schedule.



Schedules

General

General

Name: RealTimePricing

Description: E+ ExampleH

Source: DesignBuilder

Category: <General>

Region: General

Schedule type: 2-Compact Schedule

Profiles

Schedule:Compact,
RTPpriceSchedule,
Any Number,
Through: 31 May,
For: AllDays,
Until: 15:00, 0.02420,
Until: 22:00, 0.08315,
Until: 24:00, 0.02420,
Through: 30 Sep,
For: AllDays,
Until: 10:00, 0.06312,
Until: 19:00, 0.1409,
Until: 24:00, 0.06312,
Through: 31 Dec,
For: AllDays,
Until: 15:00, 0.02420,
Until: 22:00, 0.08315,
Until: 24:00, 0.02420;

- Select the newly created Real time schedule.

The Tariff tab should now look similar to the screenshot below:

Edit Tariff - EnergyPlus Example H - Real Time Pricing (RTP)

Tariffs

Tariff Charge Qualify Ratchet User variables

General

Tariff name **EnergyPlus Example H - Real Time Pricing (RTP)**

Description Real time pricing

Source

Category Electricity tariffs

Output meter name 1-Electricity:Facility

Conversion factor choice 2-kWh

Schedules

☐ Use 'Time of use period schedule'?

☐ Use 'Season schedule'?

☐ Use 'Month schedule'?

Other Settings

Demand window length 1-QuarterHour

☒ Use 'Real time pricing charge schedule'?

Real time pricing charge schedule RealTimePricing

☐ Use 'Customer baseline load schedule'?

☐ Use 'Group name'?

Monthly service charge

Monthly service charge or variable name 1-Charge value

Input monthly service charge value (USD) 37.7500

Minimum monthly charge

Minimum monthly charge or variable name 1-Charge value

Input minimum monthly charge value (USD) 0.0000

Buy or Sell

Buy or Sell 1-buyFromUtility

Help

Info Data

Schedules

RealTimePricing

Refrigeration case credit fraction schedule

Refrigeration case defrost drip down schedule

Refrigeration case defrost schedule

Refrigeration case lighting schedule

Refrigeration case restocking schedule

Refrigeration condenser water outlet temperature schedule

Refrigeration evaporative condenser availability schedule

Refrigeration walkin defrost schedule

Refrigeration walkin drip-down schedule

Refrigeration walkin glass reach-in door opening schedule

Refrigeration walkin heating power schedule

Data Report (Not Editable)

General

RealTimePricing

E+ ExampleH

Source DesignBuild

Category <General>

Region General

Schedule type 2-Compact S

Profiles

Schedule: Compact, RTPpriceSchedule, Any Number, Through: 31 May, For: AllDays, Until: 15:00, 0.02420, Until: 22:00, 0.08315, Until: 24:00, 0.02420, Through: 30 Sep, For: AllDays, Until: 10:00, 0.06312

3. On the **Charge** tab of the Tariff dialog, there should be no charges, i.e. [Number of charges](#) should be set to 0. See below.

Tariffs

Tariff Charge Qualify Ratchet User variables

Charge Selection

Number of charges 0

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**
 Timestamp: **2025-09-11 11:24:51**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	2016.42	0.00	0.00	2016.42
Cost per Total Building Area [\$ /m2]	11.03	0.00	0.00	11.03
Cost per Net Conditioned Building Area [\$ /m2]	11.03	0.00	0.00	11.03

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE G - BLOCKS WITHIN BLOCKS	Yes	Yes	ELECTRICITY: FACILITY	Buy (none)		2016.42

Report: **Tariff Report**
 For: **ENERGYPLUS EXAMPLE H - REAL TIME PRICING (RTP)**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	55.72	49.12	53.82	65.19	76.57	181.62	219.34	205.99	181.12	64.30	52.00	53.64	1258.41	219.34
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	37.75	453.00	37.75
Basis (\$)	93.47	86.87	91.57	102.94	114.32	219.37	257.09	243.74	218.87	102.05	89.75	91.39	1711.41	257.09
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	93.47	86.87	91.57	102.94	114.32	219.37	257.09	243.74	218.87	102.05	89.75	91.39	1711.41	257.09
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	93.47	86.87	91.57	102.94	114.32	219.37	257.09	243.74	218.87	102.05	89.75	91.39	1711.41	257.09

Charges

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max Category

Corresponding Sources for Charges

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
TotalDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakEnergy	1260.67	1113.39	1212.74	1433.24	1692.60	1521.19	1848.65	1729.11	1510.12	1442.16	1182.59	1214.35	17160.81	1848.65
PeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndShoulderDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
PeakAndMidPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndMidPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
PeakAndOffPeakDemand	4.07	5.44	5.82	6.15	6.75	7.53	7.60	7.94	6.66	6.12	5.64	4.07	73.78	7.94
RealTimePriceCosts	55.72	49.12	53.82	65.19	76.57	181.62	219.34	205.99	181.12	64.30	52.00	53.64	1258.41	219.34
AboveCustomerBaseCosts	55.72	49.12	53.82	65.19	76.57	181.62	219.34	205.99	181.12	64.30	52.00	53.64	1258.41	219.34
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	1260.6	1113.3	1212.7	1433.2	1692.6	1521.1	1848.6	1729.1	1510.1	1442.1	1182.5	1214.3	17160.8	1848.6
	7	9	4	4	0	9	5	1	2	6	9	5	1	5
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example I – Selling And Net-Metering

When the building contains a generator, photovoltaics, or other sources of electrical power, there is an opportunity for the building generator to produce more power than the building uses. The excess power can either be sold to the utility or used to offset the energy purchased.

This example demonstrates two separate approaches:

- Selling to Utility:** Sell surplus electricity back to utility at a fixed credit rate. The customer is generating electricity but maintains separate accounting for purchases and sales, with different rates. Use this approach if your utility contract pays a sell-back credit at a specified rate.
- Net-Metering:** Net-metering offsets consumption using a separate tariff. A single meter runs forward for consumption and backward for generation, with net monthly billing. Use this approach if your contract allows net-metering where exported energy directly reduces consumption charges.

I.a. Selling to Utility

When the excess power is to be sold, two utility rates need to be defined, one for buying electricity from the utility and the other for times where credit is received for excess energy sent back to the grid.

In this example, tariffs and charges are set up as follows:

- Customer charge: \$2.51 per month
- Energy charge: 5.5342 cents/kWh
- Sell excess energy to utility at: 2.0 cents/kWh.

This approach uses two separate tariff objects: one for purchasing electricity and a separate one for selling excess electricity back to the utility, usually at a different (usually lower) rate. This requires defining distinct tariff components with "buyFromUtility" and "sellToUtility" buy/sell fields and meters representing purchased and sold electricity separately.

Note: A negative "Cost per value" number should be entered when credit for selling electricity to the utility is required.

To create this tariff, follow the steps below.

1. The first step is to define a rate for *selling* electricity. To do this, create a new Tariff component and name it: "EnergyPlus Example Ia - Sell to Utility"
2. On the **Tariff** tab of the Tariff dialog:

- Clear all cost settings
- Set the [Output meter name](#) to **4-ElectricitySurplusSold:Facility**
- Set [Buy or Sell](#) to **2-sellToUtility**

The Tariff tab should now look like the screenshot below.

Tariffs

Tariff Charge Qualify Ratchet User variables

General

Tariff name EnergyPlus Example Ia - Sell to Utility

Description Example data

Source

Category Electricity tariffs

Output meter name 4-ElectricitySurplusSold:Facility

Conversion factor choice 2-kWh

Schedules

☐ Use 'Time of use period schedule'?

☐ Use 'Season schedule'?

☐ Use 'Month schedule'?

Other Settings

Demand window length 1-QuarterHour

☐ Use 'Real time pricing charge schedule'?

☐ Use 'Customer baseline load schedule'?

☐ Use 'Group name'?

Monthly service charge

Monthly service charge or variable name 1-Charge value

Input monthly service charge value (USD) 0.0000

Minimum monthly charge

Minimum monthly charge or variable name 1-Charge value

Input minimum monthly charge value (USD) 0.0000

Buy or Sell

Buy or Sell 2-sellToUtility

3. On the **Charge** tab:

- Set the [Number of charges](#) to 1.
- Under the **Charge 1 settings** header:
 - o Set [Simple or Block?](#) to **1-Simple**.
 - o Set [Charge variable name](#) to **GeneratedElectricSold**
 - o Set [Source variable](#) to **totalEnergy**
 - o Set [Season](#) to **1-Annual**
 - o Set [Category variable name](#) to **1-EnergyCharges**
 - o Set [Use value or variable name](#) to **1-Value**
 - o Set [Input cost per unit value](#) to -0.020

The Charge tab should now look like the screenshot below.

Tariffs	
Tariff	Charge
Charge Selection	
Number of charges	1
Charge 1 settings	
Simple or Block?	1-Simple
Charge variable name	GeneratedElectricSold
Source variable	totalEnergy
Season	1-Annual
Category variable name	1-EnergyCharges
Cost per unit value or variable name	
Use value or variable name?	1-Value
Input cost per unit value (USD)	-0.020000

Press OK to save the Tariff data and close the dialog.

The next step is to define a rate for *buying* electricity and a new Tariff component needs to be created for this.

- On the [Economics](#) model data tab, set the **Number of tariffs** to 2.
- Create a second Tariff component and name it: "EnergyPlus Example 1a - Buy from Utility".
- Open the second Tariff in the Tariff dialog and on the **Tariff** tab of the Tariff dialog, set the fixed monthly fee to \$2.51 per month using the [Input monthly service charge value](#) setting as shown below:

Tariffs

Tariff Charge Qualify Ratchet User variables

General

Tariff name EnergyPlus Example Ia - Buy from Utility

Description Example data

Source

Category Electricity tariffs

Output meter name 1-Electricity:Facility

Conversion factor choice 2-kWh

Schedules

☐ Use 'Time of use period schedule'?

☐ Use 'Season schedule'?

☐ Use 'Month schedule'?

Other Settings

Demand window length 1-QuarterHour

☐ Use 'Real time pricing charge schedule'?

☐ Use 'Customer baseline load schedule'?

☐ Use 'Group name'?

Monthly service charge

Monthly service charge or variable name 1-Charge value

Input monthly service charge value (USD) 2.5100

Minimum monthly charge

Minimum monthly charge or variable name 1-Charge value

Input minimum monthly charge value (USD) 0.0000

Buy or Sell

Buy or Sell 1-buyFromUtility

7. On the **Charge** tab:

- Set the Number of charges to 1.
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **1-Simple**.
 - Set the [Input cost per unit value](#) to 0.055342 USD:

The Charge tab should now look like the screenshot below.

Tariffs

Tariff Charge Qualify Ratchet User variables

Charge Selection

Number of charges 1

Charge 1 settings

Simple or Block? 1-Simple

Charge variable name FlatElectricityCharge

Source variable totalEnergy

Season 1-Annual

Category variable name 1-EnergyCharges

Cost per unit value or variable name

Use value or variable name? 1-Value

Input cost per unit value (USD) 0.055342

The screenshot below shows how the Economics tab should look with both Tariffs set up:

Tariffs

☒ Include tariff analysis

Number of tariffs

2

Tariff 1

EnergyPlus Example Ia - Sell to Utility

Tariff 2

EnergyPlus Example Ia - Buy from Utility

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**
 Timestamp: **2025-09-11 13:02:57**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	958.97	0.00	0.00	958.97
Cost per Total Building Area [\$/m2]	5.25	0.00	0.00	5.25
Cost per Net Conditioned Building Area [\$/m2]	5.25	0.00	0.00	5.25

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE IA - SELL TO UTILITY	Yes	Yes	ELECTRICITYSURPLUSSOLD:FACILITY	Sell	(none)	-22.0
ENERGYPLUS EXAMPLE IA - BUY FROM UTILITY	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	981.00

Report: **Tariff Report**
 For: **ENERGYPLUS EXAMPLE IA - SELL TO UTILITY**
 Timestamp: **2025-09-11 13:02:57**

General

	Parameter
Meter	ELECTRICITYSURPLUSSOLD:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	-0.9	-0.6	-2.1	-2.2	-2.4	-3.6	-3.1	-3.2	-1.9	-0.9	-0.7	-0.3	-22.0	-0.3
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Basis (\$)	-0.9	-0.6	-2.1	-2.2	-2.4	-3.6	-3.1	-3.2	-1.9	-0.9	-0.7	-0.3	-22.0	-0.3
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	-0.9	-0.6	-2.1	-2.2	-2.4	-3.6	-3.1	-3.2	-1.9	-0.9	-0.7	-0.3	-22.0	-0.3
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	-0.9	-0.6	-2.1	-2.2	-2.4	-3.6	-3.1	-3.2	-1.9	-0.9	-0.7	-0.3	-22.0	-0.3

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
GENERATEELECTRICSSOLD (\$)	-0.9	-0.6	-2.1	-2.2	-2.4	-3.6	-3.1	-3.2	-1.9	-0.9	-0.7	-0.3	-22.0	-0.3	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	46.07	27.70	103.93	109.71	121.09	181.01	156.95	161.28	96.68	45.64	35.82	15.70	1101.58	181.01

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	46.07	27.70	103.93	109.71	121.09	181.01	156.95	161.28	96.68	45.64	35.82	15.70	1101.58	181.01
TotalDemand	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
PeakEnergy	46.07	27.70	103.93	109.71	121.09	181.01	156.95	161.28	96.68	45.64	35.82	15.70	1101.58	181.01
PeakDemand	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	46.07	27.70	103.93	109.71	121.09	181.01	156.95	161.28	96.68	45.64	35.82	15.70	1101.58	181.01
PeakAndShoulderDemand	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
PeakAndMidPeakEnergy	46.07	27.70	103.93	109.71	121.09	181.01	156.95	161.28	96.68	45.64	35.82	15.70	1101.58	181.01
PeakAndMidPeakDemand	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	46.07	27.70	103.93	109.71	121.09	181.01	156.95	161.28	96.68	45.64	35.82	15.70	1101.58	181.01
PeakAndOffPeakDemand	2.51	2.91	3.50	3.76	3.73	3.68	3.68	3.68	3.56	3.17	2.29	1.75	38.21	3.76
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Other Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
NotIncluded	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Report: **Tariff Report**

For: **ENERGYPLUS EXAMPLE IA - BUY FROM UTILITY**

Timestamp: **2025-09-11 13:02:57**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

EnergyCharges (\$)	69.81	61.60	67.17	79.44	93.88	84.38	102.56	95.86	83.67	79.83	65.44	67.24	950.88	102.56
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	2.51	30.12	2.51
Basis (\$)	72.32	64.11	69.68	81.95	96.39	86.89	105.07	98.37	86.18	82.34	67.95	69.75	981.00	105.07
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	72.32	64.11	69.68	81.95	96.39	86.89	105.07	98.37	86.18	82.34	67.95	69.75	981.00	105.07
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (\$)	72.32	64.11	69.68	81.95	96.39	86.89	105.07	98.37	86.18	82.34	67.95	69.75	981.00	105.07

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
FLATELECTRICITYCHARGE (\$)	69.81	61.60	67.17	79.44	93.88	84.38	102.56	95.86	83.67	79.83	65.44	67.24	950.88	102.56	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1261.35	1113.04	1213.71	1435.50	1696.34	1524.70	1853.15	1732.16	1511.94	1442.50	1182.46	1215.03	17181.89	1853.15

Ratchets

Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----

Qualifies

Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
TotalDemand	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakEnergy	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakDemand	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
ShoulderEnergy	5	4	1	0	4	0	5	6	4	0	6	3	9	5
ShoulderDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakExceedsPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakExceedsMidPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakExceedsPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakExceedsShoulder	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
PeakAndShoulderDemand	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndMidPeakEnergy	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakAndMidPeakDemand	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
ShoulderAndOffPeakEnergy	5	4	1	0	4	0	5	6	4	0	6	3	9	5
ShoulderAndOffPeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
RealTimePriceCosts	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1

AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

I.b. Net-Metering

In this example, a "net-metering" tariff is defined with the [Buy or Sell](#) field set to **3-netMetering**, and the [Output meter](#) set to **5-ElectricityNet:Facility**. The Tariff is set up as follows:

- Customer charge: \$30 per month
- Net metering charge: 6.5 cents/kWh

To create this tariff, follow the steps below.

1. On the [Economics](#) model data tab, set the **Number of tariffs** to 1.
2. Create a new Tariff component and name it: "EnergyPlus Example Ib - Net-Metering"

The screenshot below shows how the Economics tab should now look with the Tariff set up:



5. Open the Tariff component in the Tariff dialog and on the **Tariff** tab:

- Set the [Output meter name](#) to **5-ElectricityNet:Facility**
- Set the fixed monthly fee to \$30 per month using the [Input monthly service charge value](#) setting.
- Set [Buy or Sell](#) to **3-netMetering**

Tariffs				
Tariff	Charge	Qualify	Ratchet	User variables
General				
Tariff name	EnergyPlus Example 1b - Net-Metering			
Description	Example data			
Source				
Category	Electricity tariffs			
Output meter name	5-ElectricityNet.Facility			
Conversion factor choice	2-kWh			
Schedules				
☐ Use 'Time of use period schedule'? ☐ Use 'Season schedule'? ☐ Use 'Month schedule'?				
Other Settings				
Demand window length	1-QuarterHour			
☐ Use 'Real time pricing charge schedule'? ☐ Use 'Customer baseline load schedule'? ☐ Use 'Group name'?				
Monthly service charge				
Monthly service charge or variable name	1-Charge value			
Input monthly service charge value (USD)	30.0000			
Minimum monthly charge				
Minimum monthly charge or variable name	1-Charge value			
Input minimum monthly charge value (USD)	0.0000			
Buy or Sell				
Buy or Sell	3-netMetering			

6. On the **Charge** tab:

- Set the [Number of charges](#) to 1.
- Under the **Charge 1 settings** header:
 - o Set [Simple or Block?](#) to **1-Simple**.
 - o Set [Charge variable name](#) to **NetMeteredCharge**
 - o Set [Source variable](#) to **totalEnergy**
 - o Set [Season](#) to **1-Annual**
 - o Set [Category variable name](#) to **1-EnergyCharges**
 - o Set [Use value or variable name](#) to **1-Value**
 - o Set [Input cost per unit value](#) to 0.0650

Tariffs

Tariff

Charge

Qualify

Ratchet

User variables

Charge Selection

Number of charges

1

Charge 1 settings

Simple or Block?

1-Simple

Charge variable name

NetMeteredCharge

Source variable

totalEnergy

Season

1-Annual

Category variable name

1-EnergyCharges

Cost per unit value or variable name

Use value or variable name?

1-Value

Input cost per unit value (USD)

0.065000

Summary Output

Report: **Economics Results Summary Report**

For: **Entire Facility**

Timestamp: **2025-09-11 13:09:35**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [\$]	1182.21	0.00	0.00	1182.21
Cost per Total Building Area [\$ /m2]	6.47	0.00	0.00	6.47
Cost per Net Conditioned Building Area [\$ /m2]	6.47	0.00	0.00	6.47

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (\$)
ENERGYPLUS EXAMPLE IB - NET-METERING	Yes	Yes	ELECTRICITYNET:FACILITY	Net	(none)	1182.21

Report: **Tariff Report**

For: **ENERGYPLUS EXAMPLE IB - NET-METERING**

Timestamp: **2025-09-11 13:09:35**

General

	Parameter
Meter	ELECTRICITYNET:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (\$)	71.19	59.18	58.26	61.66	71.65	63.27	81.46	75.34	70.58	73.19	64.89	71.55	822.21	81.46
DemandCharges (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ServiceCharges (\$)	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	360.00	30.00
Basis (\$)	101.19	89.18	88.26	91.66	101.65	93.27	111.46	105.34	100.58	103.19	94.89	101.55	1182.21	111.46
Adjustment (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (\$)	101.19	89.18	88.26	91.66	101.65	93.27	111.46	105.34	100.58	103.19	94.89	101.55	1182.21	111.46
Taxes (\$)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Total (\$) 101.19 89.18 88.26 91.66 101.65 93.27 111.46 105.34 100.58 103.19 94.89 101.55 1182.21 111.46

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
NETMETEREDCHARGE (\$)	71.19	59.18	58.26	61.66	71.65	63.27	81.46	75.34	70.58	73.19	64.89	71.55	822.21	81.46	EnergyCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1095.29	910.42	896.28	948.56	1102.25	973.35	1253.20	1159.11	1085.83	1125.98	998.29	1100.84	12649.40	1253.20

Ratchets

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1095.29	910.42	896.28	948.56	1102.25	973.35	1253.20	1159.11	1085.83	1125.98	998.29	1100.84	12649.40	1253.20
TotalDemand	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
PeakEnergy	1095.29	910.42	896.28	948.56	1102.25	973.35	1253.20	1159.11	1085.83	1125.98	998.29	1100.84	12649.40	1253.20
PeakDemand	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1095.29	910.42	896.28	948.56	1102.25	973.35	1253.20	1159.11	1085.83	1125.98	998.29	1100.84	12649.40	1253.20
PeakAndShoulderDemand	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
PeakAndMidPeakEnergy	1095.29	910.42	896.28	948.56	1102.25	973.35	1253.20	1159.11	1085.83	1125.98	998.29	1100.84	12649.40	1253.20
PeakAndMidPeakDemand	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1095.29	910.42	896.28	948.56	1102.25	973.35	1253.20	1159.11	1085.83	1125.98	998.29	1100.84	12649.40	1253.20
PeakAndOffPeakDemand	4.07	4.06	5.21	5.55	5.83	6.10	6.14	6.39	5.85	5.65	5.08	4.07	64.01	6.39
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Example J - Dual Ratchet Demand Tariff

This example shows how to model a dual ratchet demand structure where the billed demand is the greater of a rolling 12-month peak or a fixed minimum threshold. This approach is suitable for commercial tariffs that combine historical peak demand tracking with guaranteed minimum billing levels to ensure utility cost recovery.

Note: This example uses the GBP currency instead of USD. You can change [Currency](#) on the International tab of the Program options dialog.

Determination of Distribution Demand will be the highest of:

- The highest average kW measured during any 30-minute interval of the current and previous 11 billing months (rolling 12-month peak).
- A fixed minimum demand of 500 kW.

The example demonstrates two separate approaches to modelling the minimum demand:

- Ratchet Multiplier and Offset:** This approach uses a very small but non-zero multiplier in the UtilityCost:Ratchet object to simulate the minimum billing threshold.
- Block Charge based Minimum Billing:** The same minimum billing condition can be modelled directly through a UtilityCost:Charge:Block object.

J.a. Ratchet Multiplier and Offset

Use a very small but non-zero ratchet multiplier. In this case, the offset should be set to $500 \times (1 / \text{multiplier})$. For example, you can use a multiplier of 0.001 and an offset of 500,000.

To create this tariff structure, starting by ensuring you have the GBP currency set in Program options and follow the steps below.

1. Create a new Tariff component:



2. On the **Tariff** tab of the Tariff dialog:

- Set the [Demand window length](#) to **2-HalfHour** to ensure correct 30-minute average interval.

The Tariff tab should now look similar to the screenshot below:

Tariffs				
Tariff	Charge	Qualify	Ratchet	User variables
General				
Tariff name	Example Ja - Ratchet Multiplier and Offset			
Description				
Source				
Category	Electricity tariffs			
Output meter name	1-Electricity:Facility			
Conversion factor choice	2-kWh			
Schedules				
☐ Use 'Time of use period schedule'? ☐ Use 'Season schedule'? ☐ Use 'Month schedule'?				
Other Settings				
Demand window length	2-HalfHour			
☐ Use 'Real time pricing charge schedule'? ☐ Use 'Customer baseline load schedule'? ☐ Use 'Group name'?				
Monthly service charge				
Monthly service charge or variable name	1-Charge value			
Input monthly service charge value (GBP)	0.0000			
Minimum monthly charge				
Minimum monthly charge or variable name	1-Charge value			
Input minimum monthly charge value (GBP)	0.0000			
Buy or Sell				
Buy or Sell	1-buyFromUtility			

Note: The [Demand window length](#) choice may be overridden based on the value of [Simulation time step](#) so that they are consistent, i.e. demand window length should be exact multiples of the simulation time step length in minutes. For example, if the time step is 6, i.e. a 10 minutes time step, then **QuarterHour** is not used and instead **HalfHour** is used. Or if time step is 3, i.e. 20 minutes time step, then **QuarterHour** and **HalfHour** are not used and instead **FullHour** is used.

3. On the **Ratchet** tab:

- Set the Number of ratchets to 2.
- Under the **Ratchet 1** settings header:
 - Set [Ratchet variable name](#) to **BillingDemandA**
 - Set [Baseline source variable](#) to **TotalDemand**
 - Set [Adjustment source variable](#) to **TotalDemand**
 - Set [Season from](#) to **1-Annual**
 - Set [Season to](#) to **1-Annual**
 - Under the [Multiplier value or variable name](#) header, set [Use value or variable name?](#) to **1-Value**
 - Under the [Multiplier value or variable name](#) header, set [Input multiplier value](#) to 1
 - Under the [Offset value or variable name](#) header, set [Use value or variable name?](#) to **1-Value**
 - Under the [Offset value or variable name](#) header, set [Input offset value](#) to 0
- Under the **Ratchet 2** settings header:
 - Set [Ratchet variable name](#) to **BillingDemandB**
 - Set [Baseline source variable](#) to **BillingDemandA**
 - Set [Adjustment source variable](#) to **TotalDemand**
 - Set [Season from](#) to **6-Monthly**

- Set [Season to](#) to **1-Annual**
- Under the [Multiplier value or variable name](#) header, set [Use value or variable name?](#) to **1-Value**
- Under the [Multiplier value or variable name](#) header, set [Input multiplier value](#) to 0.001
- Under the [Offset value or variable name](#) header, set [Use value or variable name?](#) to **1-Value**
- Under the [Offset value or variable name](#) header, set [Input offset value](#) to 500,000

The **Ratchet** tab should now look similar to the screenshot below.

Tariffs

Tariff
Charge
Quality
Ratchet
User variables

Ratchet

How many ratchets applied?
2

Ratchet 1 settings

Ratchet variable name
BillingDemandA

Baseline source variable
TotalDemand

Adjustment source variable
TotalDemand

Season from
1-Annual

Season to
1-Annual

Multiplier value or variable name

Use value or variable name?
1-Value

Input multiplier value
1.000

Offset value or variable name

Use value or variable name?
1-Value

Input offset value
0.000

Ratchet 2 settings

Ratchet variable name
BillingDemandB

Baseline source variable
BillingDemandA

Adjustment source variable
TotalDemand

Season from
6-Monthly

Season to
1-Annual

Multiplier value or variable name

Use value or variable name?
1-Value

Input multiplier value
0.001

Offset value or variable name

Use value or variable name?
1-Value

Input offset value
500000.000

Summary Output

Report: **Economics Results Summary Report**
 For: **Entire Facility**
 Timestamp: **2025-10-17 13:27:12**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [£]	300.00	0.00	0.00	300.00
Cost per Total Building Area [£/m2]	1.64	0.00	0.00	1.64
Cost per Net Conditioned Building Area [£/m2]	1.64	0.00	0.00	1.64

Tariff Summary

	Selected Qualified		Meter	Buy or Sell	Group	Annual Cost (£)
EXAMPLE JA - RATCHET MULTIPLIER AND OFFSET	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	300.00

- 614 -

Report: **Tariff Report**
 For: **EXAMPLE JA - RATCHET MULTIPLIER AND OFFSET**
 Timestamp: **2025-10-17 13:27:12**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
DemandCharges (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00
ServiceCharges (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Basis (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00
Adjustment (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Surcharge (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00
Taxes (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
ENERGYCOST (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00	DemandCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
BILLINGDEMANDB	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	6000.10	500.01

Ratchets

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
BILLINGDEMANDB	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	500.01	6000.10	500.01
BILLINGDEMANDA	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	95.52	7.96

Qualifies

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec Sum Max

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
TotalDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96

ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
PeakAndShoulderDemand	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndMidPeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
PeakAndMidPeakDemand	5	4	1	0	4	0	5	6	4	0	6	3	9	5
ShoulderAndOffPeakEnergy	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
PeakAndOffPeakDemand	5	4	1	0	4	0	5	6	4	0	6	3	9	5
RealTimePriceCosts	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

J.b. Block Charge based Minimum Billing

This same case can also be modelled in a different way. The idea is that the 500 kW ratchet essentially means the customer always pays, for example, 25 pounds every month on the demand charge. This can be implemented using a UtilityCost:Charge:Block by charging 25 GBP for the first kW, 0 GBP for the next 499 kW, and, for instance, 0.05 GBP thereafter.

1. Create a new Tariff component:

Tariffs

☒ Include tariff analysis

Number of tariffs: 1

Tariff 1: Example Jb - Block Charge based Minimum Billing

2. The **Tariff** tab should be the same as for the previous example, [J.a. Ratchet Multiplier and Offset](#)

3. On the **Charge** tab:

- Set the Number of charges to 1
- Under the **Charge 1 settings** header:
 - Set [Simple or Block?](#) to **2-Block**
 - Set [Charge variable name](#) to **DemandCost**
 - Set [Source variable](#) to **BillingDemand**
 - Set [Season](#) to **1-Annual**
 - Set [Category variable name](#) to **2-DemandCharges**
 - Leave [Remaining into variable](#) blank
 - Under the [Block size multiplier or variable name](#) header, set [Use value or variable name?](#) to **1-Value** and the [Input multiplier value](#) to 1
 - Set [Number of blocks](#) to 3
 - Set [Block size 1 value](#) to 1
 - Set [Block 1 cost per unit value](#) to 25
 - Set [Block size 2 value](#) to 499
 - Set [Block 2 cost per unit value](#) to 0
 - Set [Block size 3 variable name](#) to **remaining**
 - Set [Block 3 cost per unit value](#) to 0.05

The screenshot below shows how the charge tab should look when you have finished.

The screenshot shows the 'Tariffs' application interface with the 'Charge' tab selected. The interface is divided into several sections for configuring a charge.

Charge Selection	
Number of charges	1
Charge 1 settings	
Simple or Block?	2-Block
Charge variable name	DemandCost
Source variable	BillingDemand
Season	1-Annual
Category variable name	2-DemandCharges
Remaining into variable	
Block size multiplier value or variable name	
Use value or variable name?	1-Value
Input multiplier value	1.000
Number of blocks	3
Block size 1 value	1.00
Block 1 cost per unit value (GBP)	25.000000
Block size 2 value	499.00
Block 2 cost per unit value (GBP)	0.000000
Block size 3 value or variable name	
Use block size value or variable name?	2-Block size variable name
Block size 3 variable name	remaining
Block 3 cost per unit value (GBP)	0.050000

4. On the **Ratchet** tab:

- Set the Number of ratchets to 1.
- Under the Ratchet 1 settings header:
 - Set [Ratchet variable name](#) to **BillingDemandA**
 - Set [Baseline source variable](#) to **TotalDemand**
 - Set [Adjustment source variable](#) to **TotalDemand**
 - Set [Season from](#) to **1-Annual**
 - Set [Season to](#) to **1-Annual**
 - Under the [Multiplier value or variable name](#) header, set [Use value or variable name?](#) to **1-Value**
 - Under the [Multiplier value or variable name](#) header, set [Input multiplier value](#) to 1
 - Under the [Offset value or variable name](#) header, set [Use value or variable name?](#) to **1-Value**
 - Under the [Offset value or variable name](#) header, set [Input offset value](#) to 0

The Ratchet tab should now look similar to the screenshot below.

Tariffs

Tariff
Charge
Qualify
Ratchet
User variables

Ratchet

How many ratchets applied?1

Ratchet 1 settings

Ratchet variable nameBillingDemand
Baseline source variableTotalDemand
Adjustment source variableTotalDemand
Season from1-Annual
Season to1-Annual
Multiplier value or variable name
Use value or variable name?1-Value
Input multiplier value1.000
Offset value or variable name
Use value or variable name?1-Value
Input offset value0.000

Summary Output

Report: **Economics Results Summary Report**
For: **Entire Facility**
Timestamp: **2025-10-17 13:35:53**

Annual Cost

	Electricity	Natural Gas	Other	Total
Cost [£]	300.00	0.00	0.00	300.00
Cost per Total Building Area [£/m2]	1.64	0.00	0.00	1.64
Cost per Net Conditioned Building Area [£/m2]	1.64	0.00	0.00	1.64

Tariff Summary

	Selected	Qualified	Meter	Buy or Sell	Group	Annual Cost (£)
EXAMPLE JB - BLOCK CHARGE BASED MINIMUM BILLING	Yes	Yes	ELECTRICITY:FACILITY	Buy	(none)	300.00

Report: **Tariff Report**
For: **EXAMPLE JB - BLOCK CHARGE BASED MINIMUM BILLING**
Timestamp: **2025-10-17 13:35:53**

General

	Parameter
Meter	ELECTRICITY:FACILITY
Selected	Yes
Group	(none)
Qualified	Yes
Disqualifier	n/a
Computation	automatic
Units	kWh

Categories

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
EnergyCharges (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
DemandCharges (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00
ServiceCharges (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Basis (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00
Adjustment (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Surcharge (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Subtotal (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00	
Taxes (£)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
Total (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00	

Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max	Category
DEMANDCOST (£)	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	25.00	300.00	25.00	DemandCharges

Corresponding Sources for Charges

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
BILLINGDEMAND	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	95.52	7.96

Ratchets

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
BILLINGDEMAND	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	7.96	95.52	7.96

Qualifies

Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----	-----

Native Variables

	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Sum	Max
TotalEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
TotalDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
OffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
MidPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsOffPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
OffPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsMidPeak	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
MidPeakExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakExceedsShoulder	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderExceedsPeak	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsWinter	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsNotWinter	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsSpring	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSpring	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsSummer	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotSummer	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
IsAutumn	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
IsNotAutumn	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	12.00	1.00
PeakAndShoulderEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndShoulderDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
PeakAndMidPeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndMidPeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
ShoulderAndOffPeakEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
ShoulderAndOffPeakDemand	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
PeakAndOffPeakEnergy	1261.3	1113.0	1213.7	1435.5	1696.3	1524.7	1853.1	1732.1	1511.9	1442.5	1182.4	1215.0	17181.8	1853.1
	5	4	1	0	4	0	5	6	4	0	6	3	9	5
PeakAndOffPeakDemand	4.07	5.39	5.82	6.17	6.77	7.55	7.63	7.96	6.67	6.12	5.64	4.07	73.85	7.96
RealTimePriceCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
BelowCustomerBaseCosts	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
AboveCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

BelowCustomerBaseEnergy	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
-------------------------	------	------	------	------	------	------	------	------	------	------	------	------	------	------	------

Further Examples

Another source of examples is the large set of tariff input objects provided with EnergyPlus in the macro data set file called "UtilityTariffObjects.imf" located in the "MacroDataSets" folder where EnergyPlus is installed. This file contains sets of objects in EnergyPlus' IDF file format that define tariffs for commercial customers from a collection of U.S. utility companies in August 2005.

Construction Costs

Component cost modelling provides an early design stage estimate of the initial construction capital costs associated with the building and services being modelled. The construction costs input to the EnergyPlus costing calculations are derived from the cost outputs from the [Carbon](#) tab of the [Cost and Carbon](#) tab. Initially, the idea of EnergyPlus construction costs is to ensure that cost estimates are consistent with the EnergyPlus model and allow using results calculated during a simulation such as equipment sizes and surface areas.

The DesignBuilder EnergyPlus cost estimating capabilities are intended for early stage estimation and do not cover all the types of analyses included in more detailed cost estimating software.

The usual reason for including component costs in an EnergyPlus simulation will be as part of a larger investment analysis such as life-cycle costing.

Note: The term "component cost" is used almost synonymously with "construction cost" in the DesignBuilder documentation. The "component cost" terminology comes from the EnergyPlus ComponentCost:LineItem, ComponentCost:Adjustments and ComponentCost:Reference data used to include construction costs in the simulation.

To understand how to include cost estimates in EnergyPlus inputs, it may help to first define some terminology and provide an overview of the process used in EnergyPlus. There are three broad steps involved in the EnergyPlus component cost calculations.

1. Construction costs are calculated by summing individual "line items" through calculations carried out on the Cost and Carbon screen.
2. Project costs are calculated by adjusting construction costs to account for other factors such as profit or design fees.
3. Finally the current simulation results are compared with equivalent results for a reference case so that marginal alterations can be calculated.

Each of above steps involves using one of the three Component cost data sets described below. These 3 steps are organized by corresponding tabs on the Construction costs dialog:

- [Line Item](#)
- [Adjustments](#)
- [Reference](#)

The results can be included in the **Construction Cost Estimate Summary** table by checking the **Component Cost Economics Summary** output option under the **Summary Annual Reports** header.

Example

The construction cost calculation example below shows the **Component Cost Economics Summary** output and how this is used as the basis for a comparison with a reference case with adjustment factors applied. The process is as follows:

1. Ensure that the Detailed HVAC model option is selected which enables the **Economics** model data tab at building level.
2. Run a [Cost calculation](#) on the Cost and Carbon tab.
3. Return to the Edit screen, navigate to the Economics tab and switch on the **Include construction costs in simulation inputs** checkbox to reveal the Construction cost component selection. The default construction cost option is called "Component cost".
4. Make a copy of the default "Component cost" Construction cost data and select it. It will be called "Copy of Component cost" by default. Working with your own component will allow you to make edits if necessary.
5. Edit the new selected "Copy of Component cost" construction cost component. On the first tab you will find the latest costs calculated on the Cost and Carbon tab, summed for each high level category.
6. Make any changes required on the [Adjustments](#) and [Reference](#) tabs and **press OK to save the costs**.
7. Go the Simulation screen and run a simulation.
8. After the simulation is complete, click on the Summary tab, click on the "Table of Contents" link and then on the "Component Cost Economics Summary" link to view the component construction costs.

	Line No.	Item Name	Quantity.	Units	£ per Qty.	SubTotal £
--	1	STRUCTURE COSTS	1	Ea.	50110.62	50110.62
--	2	HVAC COSTS	1	Ea.	35793.30	35793.30
--	3	LIGHTING COSTS	1	Ea.	14317.32	14317.32
--	4	SUB-STRUCTURE COSTS	1	Ea.	28326.65	28326.65
--	5	SUPER STRUCTURE COSTS	1	Ea.	99669.80	99669.80
--	6	GLAZING COSTS	1	Ea.	10654.88	10654.88
--	7	RENEWABLES COSTS	1	Ea.	23175.25	23175.25
--	8	SURFACE FINISH COSTS	1	Ea.	25856.59	25856.59
Line Item SubTotal		--	--	--	--	287904.41

		Reference case info input by user (typical from previous model)	Current info calculated for model	
		Reference Bldg.	Current Bldg. Model	Difference
Construction cost = sum of three terms	Line Item SubTotal (£)	290000.00	287904.41	-2095.60
	Misc. Costs (£)	7158.66	5965.55	-1193.10
	Regional Adjustment (£)	40413.58	39966.31	-447.30
Adjustment fractions applied to construction cost	Design Fee (£)	21942.20	23368.54	1426.34
	Contractor Fee (£)	25317.92	23368.54	-1949.40
	Contingency (£)	37132.95	33383.63	-3749.30
	Permits, Bonds, Insurance (£)	13502.89	13353.45	-149.40
Project cost = sum of eight terms	Commissioning (£)	4050.87	5007.54	956.68
	Cost Estimate Total (£)	439519.05	432317.97	-7201.10
Cost Per Conditioned Building Area (£/m2)		1841.91	1811.73	-30.20

Normalized project cost =
project cost divided by
conditional floor area

Difference = Current value - Reference value

Corresponding Adjustment and Reference Inputs:

Construction cost Data	
Line item	Adjustments
Adjustment Settings	
Miscellaneous cost per conditioned area (GBP/m2 GIFA)	25.000
Regional adjustment factor	1.136
Design and engineering fees (additional fraction)	0.070
Contractor fee (additional fraction)	0.070
Contingency (additional fraction)	0.100
Permits, bonding and insurance (additional fraction)	0.040
Commissioning fee (additional fraction)	0.015

Construction cost Data	
Line item	Reference
Reference Settings	
Reference line item costs (GBP)	2900000.000
Reference miscellaneous cost per conditioned area (GBP/m...	30.000
Reference regional adjustment factor	1.136
Reference design and engineering fees (additional fraction)	0.065
Reference contractor fee (additional fraction)	0.075
Reference contingency (additional fraction)	0.110
Reference permits, bonding and insurance (additional fraction)	0.040
Reference commissioning fee (additional fraction)	0.012

Construction Costs - Line Item

Line item tab on [Construction cost](#) Dialog

Name

Enter the name of this Component cost component.

Description

Enter the name of this Component cost component.

Category

The category cannot be changed on the dialog.

The way line item data is entered depends on which of these 2 categories is selected:

- **DesignBuilder** - Line item data is read automatically from the current results on the [Cost and Carbon](#) tab when opening the dialog, If the [cost calculations](#) are updated then data on this dialog must also be updated by re-opening the dialog and saving it again.
- **User defined** - Up to 10 line item data items can be entered manually.

Number of line items

Enter the number of line items to be inputs up to a maximum of 10. The number is fixed at 8 for cost data from the [DesignBuilder](#) category.

Line Item N Inputs

Each line item entered creates a cost line item in the EnergyPlus input and contributes to the total building cost estimate.

When using the DesignBuilder cost estimate, all Line items are obtained by first running the [Cost and Carbon](#) calculation. The line items are obtained from the sub totals in this output:

- Structure costs
- HVAC costs
- Lighting costs
- Sub-structure costs
- Super structure costs
- Glazing costs
- Renewables costs
- Surface finish costs

For details on how these costs are calculated, refer to the [Cost and Carbon](#) calculation section.

Alternatively, you can use your own line item data (as opposed to those calculated by DesignBuilder) by selecting a Component cost component from the [User defined](#) folder. Data from this folder has all fields editable.

Line items are reported by EnergyPlus and displayed in tabular form on the [Summary](#) tab of the Simulation screen:

	Line No.	Item Name	Quantity.	Units	£ per Qty.	SubTotal £
--	1	STRUCTURE COSTS	1	Ea.	50110.62	50110.62
--	2	HVAC COSTS	1	Ea.	35793.30	35793.30
--	3	LIGHTING COSTS	1	Ea.	14317.32	14317.32
--	4	SUB-STRUCTURE COSTS	1	Ea.	28326.65	28326.65
--	5	SUPER STRUCTURE COSTS	1	Ea.	99669.80	99669.80
--	6	GLAZING COSTS	1	Ea.	10654.88	10654.88
--	7	RENEWABLES COSTS	1	Ea.	23175.25	23175.25
--	8	SURFACE FINISH COSTS	1	Ea.	25856.59	25856.59
Line Item SubTotal		--	--	--	--	287904.41

Line item type

The only option in the version is **1-General**.

Item name

This field is used to refer to a specific instance of an object.

Cost per each

This field is used to enter cost information on per-each basis. The unit is local currency.

Quantity

This field is used to directly enter the line item quantity. The units should correspond to what is used in the Per Each field.

Construction Costs - Adjustments

Adjustments tab on [Construction cost](#) Dialog

Data on the Adjustments tab can be used to allow EnergyPlus to perform various modifications to the construction costs to arrive at a more realistic estimate for total project costs than is possible through summing the construction cost line items alone. It allows the line item model to be extended so that the overall costs of the project will reflect various profit and fees.

Note: Adjustments cost data is used only in EnergyPlus simulations and is not used in DesignBuilder's [Cost calculations](#) on the [Cost and Carbon](#) tab.

Miscellaneous cost per conditioned area

This optional field can be used to enter a cost model for miscellaneous project costs that are not included in the line item modelling. Miscellaneous costs are entered in cost (local currency) per conditioned floor area. This field allows including the costs of project elements that go beyond what can be modelled using Line item objects. The value input in this field will be multiplied by the conditioned floor area and will be listed in the [Construction Cost Estimate Summary](#) table.

Regional adjustment factor

This field can be used to enter an adjustment factor to account for regional differences in construction costs. The factor will be applied to the modelled and miscellaneous construction costs to determine an amount that should be added, or subtracted, to account for local variations in construction costs. The default factor is 1.0, which means no regional adjustment. The additive adjustment is calculated by subtracting 1.0 from the value entered in this field. This field is useful for using national average data in the line item and miscellaneous construction cost models and then altering the results based on regional differences.

The total base costs calculated by applying the regional adjustment factor to be modelled and miscellaneous construction costs will be used for calculating other additional costs in terms of multiplying the fractions in the following five fields.

Design and engineering fees

This field can be used to enter a fraction of total base costs that should be added to account for design and engineering fees. Note it is not the same as a fraction of the total cost estimate.

Contractor fee

This field can be used to enter a fraction of total base costs that should be added to account for Contractor fees. Note it is not the same as a fraction of the total cost estimate.

Contingency

This field can be used to enter a fraction of total base costs that should be added for contingency. Note that is not the same as a fraction of the total cost estimate.

Permits, bonding and insurance

This field can be used to enter a fraction of total base costs that should be added to account for permitting, bonding, and insurance costs to the project. Note that is not the same as a fraction of the total cost estimate.

Commissioning fee

This field can be used to enter a fraction of total based and miscellaneous construction costs that should be added to account for commissioning services. Note that is not the same as a fraction of the total cost estimate.

Construction Costs - Reference

Reference tab on [Construction cost](#) Dialog

Data on this tab can be used to enable EnergyPlus to compare the current cost estimate with the results of a previously calculated reference building. It parallels the data on the [Adjustments](#) tab but adds a field for entering the cost line item model result for the reference building.

All the various adjustments to the project cost estimate are available here as factors that differ between the current and the reference building models to be applied. The factors entered here are applied to the reference building while the factors listed in the Adjustment object are applied to the current building model cost estimate. The results are included in the **Construction Cost Estimate Summary** table by checking the **Component Cost Economics Summary** output option.

Note: Reference cost data is used only in EnergyPlus simulations and is not used in DesignBuilder's [Cost calculations](#) on the [Cost and Carbon](#) tab.

Reference line item costs

This field is used to enter a cost representing the total for the reference building to be compared to the line item cost model for the current building model. A typical source of data for this would be the "Line Item Subtotal" from a previous simulation.

Reference miscellaneous cost per conditioned area

This field can be used to enter a cost model for miscellaneous project costs that are not included in the line item modelling. Miscellaneous costs are entered in cost (local currency) per conditioned floor area. This field allows including the costs of project elements that go beyond what can be modelled using Line item objects. The value input in this field will be multiplied by the conditioned floor area and listed in the **Construction Cost Estimate Summary** table.

Reference regional adjustment factor

This field can be used to enter an adjustment factor to account for regional differences in construction costs. The factor will be applied to the modelled and miscellaneous construction costs to determine an amount that should be added, or subtracted, to account for local variations in construction costs. The default factor is 1.0, which means no regional adjustment. The additive adjustment is calculated by subtracting 1.0 from the value entered in this field (e.g. 1.05 means a 5% increase).

This field is useful for using national average data in the line item and miscellaneous construction cost models and then altering the results based on regional differences.

The total base costs calculated by applying regional adjustment factor to modelled and miscellaneous construction costs will be used for calculating other additional costs in terms of multiplying the fractions in the following five fields.

Reference design and engineering fees

This field can be used to enter a fraction of total base costs that should be added to account for design and engineering fees. Note that it is not the same as a fraction of the total cost estimate.

Reference contractor fee

This field can be used to enter a fraction of total base costs that should be added to account for Contractor fees. Note that it is not the same as a fraction of the total cost estimate.

Reference contingency

This field can be used to enter a fraction of total base costs that should be added for contingency. Note that it is not the same as a fraction of the total cost estimate.

Reference permits, bonding and insurance

This field can be used to enter a fraction of total base costs that should be added to account for permitting, bonding, and insurance costs to the project. Note that it is not the same as a fraction of the total cost estimate.

Reference commissioning fee

This field can be used to enter a fraction of total base costs that should be added to account for commissioning services. Note that it is not the same as a fraction of the total cost estimate.

Life Cycle Cost

Life-cycle costing (LCC) is used with building energy simulation in order to justify energy efficiency upgrades. Many alternative building technologies that result in energy savings cost more initially, or may cost more to maintain, than traditional solutions. In order to justify selecting these energy savings technologies, it is essential to combine both initial and future costs in the decision process. Using life-cycle costs provides a framework to combine initial costs and future costs into a single combined measure, called the “present value.” Present value is a metric that combines all costs and reduces (or discounts) those costs that occur in the future. Discounting future costs is based on the principal of the time value of money.

The calculations are based on discounting the future values according to normal life-cycle costing techniques as described in NIST Handbook 135 [Life-Cycle Costing Manual for the Federal Energy Management Program](#).

The following is a list of life-cycle costing related data that provide a way to describe the parameters and costs associated with the life-cycle of a building or building system. These objects are organized by tabs on the Life Cycle Cost dialog:

- [Parameters](#)
- [Recurring costs](#)
- [Non-recurring costs](#)
- [Use price escalation](#)
- [Use adjustment](#)

The [utility costs](#) and [first costs](#) are used along with other current and future costs input using the Recurring costs and Non-recurring cost data and these are combined into the present value life-cycle cost metric. The Parameters object establishes the set of assumptions for the analysis along with Use price escalation (often from databases) and Use adjustment objects.

Note 1: It is important to understand that the comparison of different simulation results and their present values is not performed by EnergyPlus. Instead, EnergyPlus provides the present value calculations for a specific simulation combining the energy costs, first costs, and future costs and you need to make the comparison between the results of multiple simulations when needed.

Note 2: Life-Cycle Costing (LCC) should not be confused with [Life Cycle Analysis](#) (LCA). With LCC the result is an economic evaluation of current and future expenditures in order to make a decision on alternative investments. In LCA, the environmental impact such as equivalent CO2 production involved in the materials, delivery, manufacturing, and construction are combined with environmental impacts of the building in operation and the eventual removal of the building and is used to understand the overall environmental impact or embodied energy.

Tip: You can carry out powerful analyses to simultaneously optimise whole life cycle cost and whole life cycle environmental impact performance using DesignBuilder's optimisation tools. Choose the [LCC](#) and [LCA](#) KPIs for this sort of analysis.

Output

The results of using the LCC analysis are included automatically in the Summary report under “Life Cycle Cost Report.” This report shows the costs and the timing of costs, often called “cash flows,” along with the present value in several different tables. The tabular results show the present value of all current and future costs.

Life Cycle Cost - Parameters Tab

Parameters tab on [Life Cycle Cost Dialog](#)

Provides inputs related to the overall life-cycle cost analysis. It establishes many of the assumptions used in computing the present value. It is important that when comparing the results of multiple simulations that the fields in the Parameters objects are the same for all the simulations. To help check this is the case, the first table in the Life-Cycle Cost Report shows the inputs to this object.

Name

An identifying name this LCC analysis.

Discount Settings

Discounting convention

The field specifies if the discounting of future costs should be computed as occurring at the end, the middle or the beginning of each year. The most common discounting convention uses the end of each year, therefore without a specific reason, the end of year should be used. The year being used starts with the base year and month (see below) and repeats every full year.

All costs assumed to occur during that duration are accumulated and shown as an expense either at the beginning, middle or end of the year. The options are:

- **1-BeginningOfYear**
- **2-MidYear**
- **3-EndOfYear**

Note: Specifically, some military projects may require using the middle of each year.

Inflation approach

This field is used to determine if the analysis should use constant dollars or current dollars which is related to how inflation is treated. The two options are:

- **1-ConstantDollar** (Default option)
- **2-CurrentDollar**

If **1-ConstantDollar** is selected, then the Real discount rate input (see below) is used and it excludes the rate of inflation. If **2-CurrentDollar** is selected, then the Nominal discount rate (see below) input is used and it includes the rate of inflation. For most analyses, using the **1-ConstantDollar** option will be easier since the effect of inflation may be ignored.

Reference: From NIST Handbook 135: "The constant dollar approach has the advantage of avoiding the need to project future rates of inflation or deflation. The price of a good or service stated in constant dollars is not affected by the rate of general inflation. For example, if the price of a piece of equipment is \$1,000 today and \$1,050 at the end of a year in which prices in general have risen at an annual rate of 5 percent, the price stated in constant dollars is still \$1,000; no inflation adjustment is necessary. In contrast, if cash flows are stated in current dollars, future amounts include general inflation, and an adjustment is necessary to convert the current dollar estimate to its constant-dollar equivalent. This adjustment is important because constant- and current-dollar amounts must not be combined in an LCCA."

Real discount rate

Enter the real discount rate as a decimal, e.g. for a 3% rate, enter the value 0.03. This input is used when the [Inflation approach](#) is **1-ConstantDollar**. The real discount rate reflects the interest rates needed to make current and future expenditures have comparable equivalent values when general inflation is ignored. When the [Inflation approach](#) is set to **2-CurrentDollar** this input is ignored.

Nominal discount rate

Enter the nominal discount rate as a decimal, e.g. for a 5% rate, enter the value 0.05. This input is used when the [Inflation approach](#) is **2-CurrentDollar**. The nominal discount rate reflects the interest rates needed to make current and future expenditures have comparable equivalent values when general inflation is included. When [Inflation approach](#) is set to **1-ConstantDollar** this input is ignored.

Inflation rate

Enter the rate of inflation for general goods and services as a decimal, e.g. for a 2% rate, enter the value 0.02. When [Inflation approach](#) is set to **1-ConstantDollar** this input is ignored.

Example: Below is an example from the NIST Handbook 135 1995 edition.

The following example shows that both approaches result in the same present value and thus support the same decision.

Example: Suppose you want to know the present value of an AC compressor that you expect to replace in 15 years. If it were replaced today, the price would be \$5,000. Due to advanced manufacturing processes, you expect that the price of compressors will increase at a rate of 2 percent lower than general price inflation. You estimate the rate of general price inflation to be 5 percent per year. You know that your real discount rate is 3 percent. To sum

$I = 0.05$	$t = 15 \text{ years}$
$d = 0.03$	$D = (1 + 0.03)(1 + 0.05) - 1 = 0.0815$
$e = -0.02$	$E = (1 - 0.02)(1 + 0.05) - 1 = 0.029$

real discount rate, d

nominal discount rate, D

**Constant dollars and
real discount rate**

**Current dollars and
nominal discount rate**

$$\begin{aligned} PV &= F_t \times \left[\frac{1 + e}{1 + d} \right]^t \\ &= 5000 \times \left[\frac{1 - 0.02}{1 + 0.03} \right]^{15} \\ &= 5000 \times 0.4741 \\ &= \$2,370.30 \end{aligned}$$

$$\begin{aligned} PV &= F_t \times \left[\frac{1 + E}{1 + D} \right]^t \\ &= 5000 \times \left[\frac{1 + 0.029}{1 + 0.0815} \right]^{15} \\ &= 5000 \times 0.4741 \\ &= \$2,370.30 \end{aligned}$$

Dates For Study

Base date month

Enter the month that is the beginning of study period, also known as the beginning of the base period. The options are:

- **1-January** (default)

- 2-February
- 3-March
- 4-April
- 5-May
- 6-June
- 7-July
- 8-August
- 9-September
- 10-October
- 11-November
- 12-December

Reference: According to NIST 135 “the base date is the point in time to which all project related costs are discounted in an LCCA [life cycle cost analysis]. The base date is usually the first day of the study period for the project, which in turn is usually the date that the LCCA is performed. In a constant dollar analysis, the base date usually defines the time reference for the constant dollars (e.g. 1995 constant dollars). It is essential that you use the same base date and constant-dollar year for all of the project alternatives to be compared. If you set the base date to the date that the LCCA is performed, then the constant dollar basis for the analysis will be the current date, and you can use actual costs as of that date without adjusting for general inflation.”

Base date year

Enter the four digit year that is the beginning of study period, such as a year expressed in four digits like “2011”. The study period is also known as the base period. See more details in the previous field.

Service date month

Enter the month that is the beginning of building occupancy. Energy costs computed by EnergyPlus are assumed to occur during the year following the service date. The service date must not be earlier than the Base date. The options are:

- 1-January (default)
- 2-February
- 3-March
- 4-April
- 5-May
- 6-June
- 7-July
- 8-August
- 9-September
- 10-October
- 11-November
- 12-December

Service date year

Enter the four digit year that is the beginning of occupancy, such as two years after the previously entered year expressed in four digits so example is like “2013”.

Reference: According to NIST Handbook 135: “The service date is the date on which the project is expected to be implemented; operating and maintenance costs (including energy- and water-related costs) are generally incurred after this date, not before.”

Other Settings

Length of study period in years

Enter the number of years of the study period. It is the number of years that the study continues based on the start at the base date.

The default value is 25 years, and only integers are allowed that indicate whole years.

Reference: According to NIST Handbook 135, “the study period for an LCCA is the time over which the costs and benefits related to a capital investment decision are of interest to the decision maker. Thus, the study period begins with the base date and includes both the planning/construction period (if any) and the relevant service period for the project. The service period begins with the service date and extends to the end of the study period.”

Tax rate

Enter the overall marginal tax rate for the project costs. This does not include energy or water taxes. The single tax rate entered here is not intended to be a replacement of the complex calculations necessary to compute personal or corporate taxes; instead it is an approximate that may be used for a simple analysis assuming a constant tax rate is applied on all costs. The tax rate entered should be based on the marginal tax rate for the entity and not the average tax rate. Enter the tax rate results in present value calculations after taxes. Most analyses do not factor in the impact of taxes and assume that all options under consideration have roughly the same tax impact. Due to this, many times the tax rate can be left to default to zero and the present value results before taxes are used to make decisions. The value should be entered as a decimal value, e.g. for 15% enter 0.15. For an analysis that does not include tax impacts, enter 0.0. The default is 0.

Depreciation method

For an analysis that includes income tax impacts, this entry describes how capital costs are depreciated. Only one depreciation method may be used for an analysis and is applied to all capital expenditures. Only analyses that include tax impacts need to select a depreciation method. The options are:

- **1-ModifiedAcceleratedCostRecoverySystem-3year**
- **2-ModifiedAcceleratedCostRecoverySystem-5year**
- **3-ModifiedAcceleratedCostRecoverySystem-7year**
- **4-ModifiedAcceleratedCostRecoverySystem-10year**
- **5-ModifiedAcceleratedCostRecoverySystem-15year**
- **6-ModifiedAcceleratedCostRecoverySystem-20year**
- **7-StraightLine-27year**
- **8-StraightLine-31year**
- **9-StraightLine-39year**
- **10-StraightLine-40year**
- **11-None** (default)

Depreciation allowances reduce the actual/nominal tax dollars paid by the owner. Thus, analyses using depreciation should be conducted in nominal dollars. For an analysis that does not include tax effects, **11-None** should be selected.

Note: For more details about depreciation methods and which one to choose, refer to IRS Publication 946 – How to Depreciate Property. It states that “Depreciation is an annual income tax deduction that allows you to recover the cost or other basis of certain property over the time you use the property. It is an allowance for Fair market value the wear and tear, deterioration, or obsolescence of the Intangible property.” Details on which depreciation method to choose depends on the property being depreciated, IRS Publication 946 and your accountant will be the best sources of information in determining which depreciation method to choose.

Life Cycle Cost - Recurring Costs Tab

Recurring costs tab on [Life Cycle Cost Dialog](#)

Recurring costs are costs that repeat over time on a regular schedule during the study period. If costs associated with equipment do repeat but not on a regular schedule, use the [Non-recurring costs](#) tab instead.

Number of recurring costs

The number of recurring costs applied in this life-cycle cost can be selected from the dropdown list. For each recurring cost the following data must be entered.

Recurring Cost N Settings

Name

The identifier used for this recurring cost. The name is used in identifying the cash flow equivalent in the output results in the Life-Cycle Cost Report.

Category

Enter the category of the recurring costs. Choose the closest category. The options include:

- **1-EnergyCosts**
- **2-WaterCosts**
- **3-Maintenance**
- **4-Repair**
- **5-Operation**
- **6-Replacement**
- **7-MinorOverhaul**
- **8-MajorOverhaul**
- **9-OtherOperational**

The default value is **3-Maintenance**.

Note: For recommendations on estimating other operational costs, see NIST 135 Section 4.6.3.

Cost

Enter the cost (local currency) for the recurring costs. Enter the cost for each time it occurs. For example, if the annual maintenance cost is \$500, enter 500 here.

Start of costs

Enter when the costs start. The First year of cost is based on the number of years past the **Start of costs**. For most maintenance costs, the **Start of costs** should be **1-ServicePeriod**. The options are:

- **1-ServicePeriod**
- **2-BasePeriod**

The default value is **1-ServicePeriod**.

Years from start

This field and **Months from start** (below) together represent the time from either the start of the Service period, on the service month and year, or start of the base period, on the base month and year (depending on selection from the **Start of costs** field) that the costs start to occur. Normally, for most maintenance costs that

begin in the first year that the equipment is in service the **Start of costs** is the service period and the **Years from start** will be 0. Only integers should be entered representing whole years. The default value is 0.

Months from start

This field and **Years from start** (above) together represent the time from either the start of the Service period, on the service month and year, or start of the base period, on the base month and year (depending on the **Start of costs** field) that the costs start to occur. Normally, for most maintenance costs the **Start of costs** is the Service period and the **Months from start** will be 0. Only integers should be entered representing whole months. The **Years from start** (times 12) and **Months from start** are added together. The default value is 0.

Repeat period years

This field and **Repeat period months** (below) indicate how much time elapses between re-occurrences of the cost. For costs that occur every year, such as annual maintenance costs, the **Repeat period years** should be 1 and **Repeat period months** should be 0. Only integers should be entered representing whole years. The default value is 1.

Repeat period months

This field and **Repeat period years** (above) indicate how much time elapses between re occurrences of the cost. For costs that occur every year the **Repeat period years** should be 1 and **Repeat period months** should be 0. For, costs that occur every eighteen months, the **Repeat period years** should be 1 and the **Repeat period months** should be 6. Only integers should be entered representing whole years. The **Repeat period years** (times 12) and **Repeat period months** are added together. The default value is 0.

Annual escalation rate

Enter the annual escalation rate as a decimal. For a 1% rate, enter the value 0.01. This input is used only when the [Inflation approach](#) is **2-CurrentDollar**. The default value is 0.

Life Cycle Cost - Non-Recurring Costs Tab

Non-Recurring costs tab on [Life Cycle Cost](#) Dialog

A non-recurring cost happens only once during the study period. For costs that occur more than once during the study period on a regular schedule, use the [Recurring costs](#) tab.

Number of non-recurring costs

The number of non-recurring costs applied in this life-cycle cost can be selected from this dropdown list. For each non-recurring cost the following data must be entered.

Non-Recurring Cost N Settings

Name

The identifier used for the object. The name is used in identifying the cash flow equivalent in the output results in the Life-Cycle Cost Report.

Category

Enter the category of the non-recurring costs. Choose the closest category. The options include:

- **1-Construction**
- **2-Salvage**
- **3-OtherCapital**

The default is **1-Construction**.

Reference: For recommendations on estimating capital and salvage (residual) costs, see NIST 135 Section 4.5.

Cost

Enter the non-recurring cost value. For construction and other capital costs the value entered is typically a positive value. For salvage costs, the value entered is typically a negative value which represents the money paid to the investor for the equipment at the end of the study period.

Start of costs

Enter when the costs start. The first year of cost is based on the number of years past the **Start of costs**. For most non-recurring costs the **Start of costs** should be base period which begins at the base month and year. The options are:

- **1-ServicePeriod**
- **2-BasePeriod**

The default value is **2-BasePeriod**.

Years from start

This field and **Months from start** (below) together represent the time from either the start of the service period, on the service month and year, or start of the base period, on the base month and year (depending on the **Start of cost** field) that the costs start to occur. Normally, for most capital costs the **Start of costs** is the base period and the **Years from Start** will be 0. These would be first costs for the building equipment or system. Salvage costs are usually shown as happening in the last year of the analysis. Only integers should be entered representing whole years. The default value is 0.

Months from start

This field and **Years From start** (above) together represent the time from either the start of the service period, on the service month and year, or start of the base period, on the base month and year (depending on the **Start of cost** field) that the costs start to occur. Normally, for most capital costs the **Start of costs** is the base period and the **Months from start** will be 0. Only integers should be entered representing whole months. The **Years From start** (times 12) and **Months from start** are added together. The default value is 0.

Life Cycle Cost - Use Price Escalation Tab

Use price escalation tab on [Life Cycle Cost Dialog](#)

The Use price escalation data will commonly be part of a dataset on life cycle costs.

Reference: The values for this object may be found in the annual supplement to NIST Handbook 135 in Tables Ca-1 to Ca-5. A dataset file comes with EnergyPlus in the DataSet directory and includes the LCCUsePriceEscalationDataSetXXXX.idf file which includes the value for the supplement to NIST 135 from the year indicated. The object provides inputs for the escalations of energy and water costs assuming that they change differently than inflation. According to the NIST 135 supplement the values are “present projected fuel price indices for the four Census regions and for the United States. These indices, when multiplied by annual energy costs computed at base-date prices provide estimates of future-year costs in constant base-date dollars. Constant-dollar cost estimates are needed when discounting is performed with a real discount rate (i.e., a rate that does not include general price inflation).”

The **LCCUsePriceEscalationDataSetXXXX.idf** is periodically updated by the EnergyPlus development team using data available from NIST along with a spreadsheet to convert that data into an IDF dataset file. The NIST 135 annual supplement is available from NIST or through US DOE FEMP program.

<https://nvlpubs.nist.gov/nistpubs/ir/2019/NIST.IR.85-3273-34.pdf>

If requested, NIST personnel will provide a file called ENCOSTxx.TXT which contains the data used to create Table Ca-1 to Table Ca-5 in the supplement. To convert the data in the ENCOSTxx.TXT file to an IDF file, open the ENCOSTxx.TXT file in a text editor and copy the entire contents into the first tab called “Step 1” of the conversion spreadsheet. The conversion spreadsheet file named

ConvertENCOSTtoEnergyPlusLifeCycleCost.xls is available upon request from the EnergyPlus development team. The “Step 2” tab separates the data into columns. The “Step 3” tab computes ratios that are the same as Table Ca-1 to Table C1-6 in the supplement. Checking these values against what appears in the printed supplement is recommended. The “Step 4” and “Step 5” tabs rearrange the data and the “Step 6” tab shows the data in a format that can be easily copied and pasted into a new idf dataset file. The final step requires using your text editor to replace the fuel names with those used by EnergyPlus. The replacement names are listed on the “FuelNameConversion” tab.

Number of escalations used

The number of price escalations applied in this life-cycle cost can be selected from this dropdown list. The following fields can be repeated up to the number selected in this field.

Use Price Escalation N Settings

Name

The identifier used for this escalation. The name usually identifies the location (such as the state, region, country or census area) that the escalations apply to. In addition the name should identify the building class such as residential, commercial, or industrial and the use type such as electricity, natural gas, or water.

Resource

Select the resource from:

- **1-Electricity**
- **2-FuelOil1**
- **3-NaturalGas**
- **4-PropaneGas**
- **5-FuelOil2**
- **6-Coal**
- **7-Steam**
- **8-Gasoline**
- **9-Diesel**
- **10-Water**

Escalation start year

This field and the **Escalation start month** define the time that corresponds to “Year 1 Escalation” such as a year expressed in four digits like “2013”, when the escalation rates are applied. This field and the **Escalation start month** define the time that escalation begins.

Escalation start month

This field and the **Escalation start year** define the time that corresponds to “Year 1 Escalation” such as a year expressed in four digits such as “2013”, when the escalation rates are applied. This field and the **Escalation start year** define the time that escalation begins. The options are:

- **1-January** (default)
- **2-February**
- **3-March**
- **4-April**
- **5-May**
- **6-June**

- 7-July
- 8-August
- 9-September
- 10-October
- 11-November
- 12-December

Number of years of escalation

This number can be selected by the dropdown list. Often 25 to 50 years of projected values are required.

Escalation Data

Year 1 escalation

The escalation in price of the energy or water use for the first year expressed as a decimal.

Year n escalation

The escalation in price of the energy or water use for the n-th year expressed as a decimal. As mentioned above, normally 25 to 50 years data is used.

Note: If the number of years of escalation is less than the number of years in the analysis period, the remaining years will assume no escalation, therefore normal inflation will be the only affect for these years.

Life Cycle Cost - Use Adjustment Tab

Used by advanced users to adjust the energy or water use costs for future years. This should not be used for compensating for inflation but should only be used to increase the costs of energy or water based on assumed changes to the actual use of water or energy, such as due to changes in the future function of the building or the impact of future adjacent buildings on energy use. This object is not commonly used and should be used with caution. The adjustments begin at the start of the service period.

Number of adjustments used

The number of adjustments applied in this life-cycle cost can be selected from this dropdown list. The following fields can be repeated up to the number selected in this field.

Adjustment N Settings

Name

The identifier used for this use adjustment data.

Resource

Select the resource from:

- 1-Electricity
- 2-FuelOil1
- 3-NaturalGas
- 4-PropaneGas
- 5-FuelOil2
- 6-Coal
- 7-Steam
- 8-Gasoline

- [9-Diesel](#)
- [10-Water](#)

Number of years of adjustment

This number can be selected by the dropdown list. Often 25 to 50 years of values are required.

Adjustment Data

Year 1 multiplier

The multiplier to be applied to the end use cost for the first year in the service period. The total utility costs for the selected end use is multiplied by this value. For no change enter 1.0.

Year n multiplier

The multiplier to be applied to the end use cost for each following year. The total utility costs for the selected end use is multiplied by this value. For no change enter 1.0.

Note: If the number of years of Use price escalation is less than the number of years in the analysis period, the remaining years will assume no escalation, therefore normal inflation will be the only affect for these years.

Miscellaneous

Miscellaneous tab in model data

You can control the output data that is generated in the design calculations and simulations as well as various other miscellaneous calculation settings by making selections on the **Miscellaneous** model data tab.

- "Outputs - General" on page 636
- [Heating Design Output Options](#)
- [Cooling Design Output Options](#)
- [Simulation Output Options](#)
- "Daylighting Outputs" on page 640

Outputs - General

[Miscellaneous](#) model data tab

Name in last EnergyPlus calculation

This is the IDF name of the current building, zone, surface or opening. The IDF name is set when the model inputs for an EnergyPlus simulation are being generated.

For surfaces and openings it is possible for more than one EnergyPlus objects to be generated per corresponding DesignBuilder object, and in this case multiple IDF names are separated by commas.

Tag

You can enter your own tag text to define the current object for use in [EMS optional arguments](#) and [target selection tools](#).

Output Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog, General tab on [Heating Design Calculation Options](#) dialog and [Miscellaneous](#) model data tab

Heating design output options apply to the whole building and are only visible at building level when accessed from the [Outputs](#) model data tab.

Include unoccupied zones in block and building totals and averages

By default data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Store surface output

Causes the following output to be stored for all walls, floors, roofs, ceilings, partitions etc:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Loss
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Store opening output

Only visible when the [Store surface output](#) option is selected, this option causes the following data to be stored for windows vents, doors, holes, sub-surfaces:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Loss
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Warning: Use the surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be stored.

Output Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog, General tab on [Cooling Design Calculation Options](#) dialog and [Miscellaneous](#) model data tab

Cooling design output options apply to the whole building and are only visible at building level when accessed from the [Outputs](#) model data tab.

Include unoccupied zones in block and building averages and totals

By default, data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Store surface output

Causes the following output to be stored for all walls, floors, roofs, ceilings, partitions etc:

- Inside Surface Temperature
- Outside Surface Temperature
- Surface Heat Gain
- Solar Incident
- External Sunlit Fraction
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Store opening output

Only visible when the [Store surface output](#) option is selected, this option causes the following data to be stored for windows vents, doors, holes, sub-surfaces:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Gain
- Solar Incident
- Solar Transmitted
- External Sunlit Fraction
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Warning: Use the surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be stored.

Simulation Output Options

[Miscellaneous](#) model data tab

General

Output options

Simulation output options can be accessed at building, block, zone, surface and opening levels giving you control of the elements in the model for which output data will be generated.

The way that output options are accessed on the Miscellaneous model data tab depends on the setting of the [Allow custom outputs](#) option which is always available at building level here on the Miscellaneous tab and also on the Simulation tab of the Model options dialog and on the Output tab of the Simulation Calculation Options dialog. When **Allow custom outputs** is unselected (the default setting) all simulation output options can be accessed from the building level only of the Miscellaneous model data tab. There is no flexibility to select different options for different blocks, zone, surfaces and opening. However when **Allow custom outputs** is selected you can select most of the outputs on a zone-by-zone, surface-by-surface and opening-by-opening basis.

Note: All of the output options listed below are explained in more detail on the main [Output options](#) page.

Firstly these output options are always only available at building level, since they apply to all results:

- [Building and block output of zone data](#)
- [Include unoccupied zones in block and building total and averages](#)
- [Allow custom outputs](#)

These outputs are available at building level only when **Allow custom outputs** is unselected, or at building, block and zone levels when it is selected:

Energy:

- [Surface heat transfer](#)
- [Internal gains including solar](#)
- [Energy, HVAC etc](#)
- [Latent loads](#)

Comfort and Environmental:

- [Environmental](#)
- [Fresh air supply](#)
- [Simple ASHRAE Standard 55](#)
- [Adaptive ASHRAE Standard 55](#)
- [CIBSE TM52](#)
- [CIBSE TM59](#)
- [Fanger](#)
- [Pierce two-node](#)
- [Kansas State University two-node](#)
- [Temperature distribution](#)

[Detailed Daylight outputs](#)

[Summary Annual reports](#)

[Summary Monthly reports](#)

[Miscellaneous outputs](#)

Store surface output

When the [Allow custom outputs](#) option is *unselected*:

The Store surface output and Store opening output options are available at building level only. You can therefore choose whether or not to generate *all* data for *all* surfaces, and likewise for *all* openings. It is not possible to be selective in this case, i.e. it is not possible to select outputs surface-by-surface or opening-by-opening.

Otherwise when **Allow custom outputs** is *selected*:

The surface and opening outputs option options are available at block, zone, surface and opening levels. To access the output options for a particular surface or opening, you should first navigate to that

surface or opening and switch on the **Store surface output** option. This causes results for that surface or opening to be generated and stored but it does not yet select any results for display.

While these settings are used by the program at surface and opening levels, the normal rules of inheritance apply, so you can, for example, navigate to a zone and select the default output options for all surfaces and openings in that zone.

To request surface or opening results select from these options:

- **Store opening/sub-surface output** - *at surface level and above in the model hierarchy, this is a default value for openings and sub-surfaces.* It must be selected for a particular opening if any data for that opening or sub-surface is to be generated.
- **Inside Surface Temperature** - for surfaces and openings.
- **Outside Surface Temperature** - for surfaces and openings.
- **Surface Heat Gain** - for surfaces.
- **Glazing Heat Gain** - for openings.
- **Solar Incident** - for surfaces and openings.
- **Solar Transmitted** - for window openings.
- **External Sunlit Fraction** - for surfaces and openings.
- **Internal Convective Heat Transfer Coefficient** - for surfaces and openings.
- **External Convective Heat Transfer Coefficient** - for surfaces and openings.
- **Airflow** - infiltration and ventilation data is available at both surface and opening level when using the [Calculated natural ventilation](#) model option.

See: "Surface and Opening Levels" on page 1627 for information on the data generated when these surface and opening level options are selected.

Warning: Use these surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be generated and stored. You should select only the data you need for the building elements of interest.

Daylighting Outputs

[Miscellaneous](#) model data tab

Working plane height

Enter the working plane height above the top of the floor (in m or ft). This can be entered at building, block or zone level and is used by DesignBuilder at zone level. It allows you to enter different working plane heights for different zones.

Note: The same attribute can be accessed on the [Daylighting calculation options](#) dialog but for building level only .

CFD

CFD tab in model data

This tab allows you set various types of boundary condition for [CFD calculations](#).

If you have previously loaded simulation results as boundary conditions using the [CFD Boundary Conditions Edit](#) dialog then this data will be displayed at surface and opening levels.

CFD Boundary Defaults

The attributes that can be set on this tab are shown with the boundary condition type to which they apply. Note that all attributes are displayed at zone level and above in the hierarchy because these act as defaults, but at surface and opening level only the data applicable to the current object is displayed.

Note: Data specified on this tab does not affect EnergyPlus simulations.

Inside surface temperature (internal surfaces)

Applies to: *All Internal surfaces and all internal openings*

This setting provides a default inside surface temperature (in °C or °F) for internal surfaces (partitions, internal floors and ceilings) as well as all internal openings. It can be set for all levels in the model hierarchy from building level down.

This value is also used for virtual partitions to represent the average temperature of the surfaces in the adjacent room.

Important Note: Zero thickness internal partitions and floors/ceilings separating two zones in a block or building level CFD simulation are ignored, and in this case the domain will be seen as a single contiguous space. This boundary condition setting therefore has no effect on the simulation in this case.

Inside surface temperature (external surfaces)

Applies to: *All external surfaces and external doors and sub-surfaces*

This setting provides a default inside surface temperature (in °C or °F) for all external opaque surfaces and openings, including walls, roofs, external floors etc as well as external doors and sub-surfaces. It can be set for all levels in the model hierarchy from building level down.

It does not include external windows though, which are included in the item below.

Inside surface window temperature (external windows)

Applies to: *External windows*

This setting defines the inside surface temperature for external windows (in °C or °F). It can be set for all levels in the model hierarchy from building level down.

Incoming air temperature

Applies to: *Windows, Vents, Doors and Holes*

Temperature of air entering the space through windows, vents, doors and holes (in °C or °F).

Incoming air temperature

Applies to: *Supply diffusers*

Temperature of air entering the space through [Supply diffusers](#) (in °C or °F).

Flow in

Applies to *Windows, Vents, Doors and Holes*

Defines the flow rate through the opening into the zone (in l/s or ft³/min).

Flow out

Applies to *Windows, Vents, Doors and Holes*

Defines the flow rate through the opening out of the zone (in l/s or ft³/min).

Component Blocks, Assemblies and CFD Boundary Condition Patches

Data on this tab is also used for entering boundary conditions for component blocks, assemblies and for boundary condition patches drawn onto zone or component block surfaces. For details see:

- [Component Blocks and Assemblies as CFD Boundaries](#)
- [Boundary Conditions for Internal CFD Analyses](#)

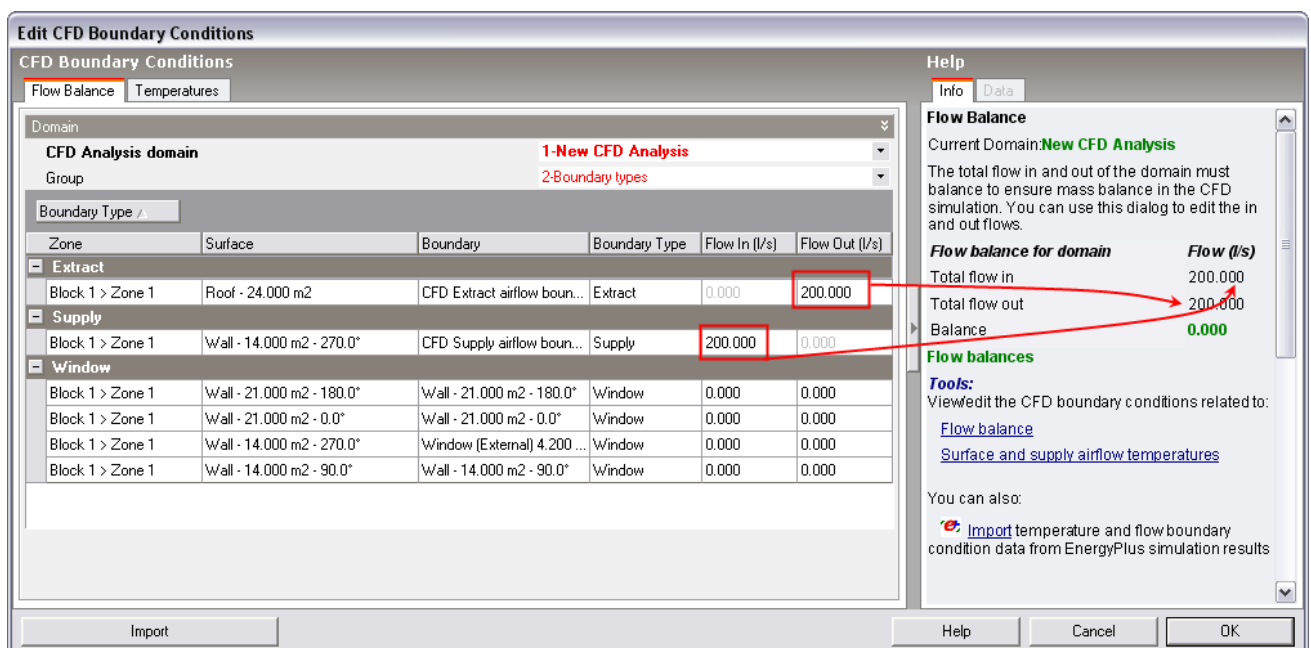
See also:

- [CFD Boundary Conditions Editor](#)
- [Preparing Simulations to Generate CFD Boundary Conditions](#)

CFD Boundary Conditions Editor

The CFD Boundary Conditions Editor allows you to view and edit all of the boundary conditions in the selected domain in one place. In particular you can:

- Check and set the airflow rate through each supply and extract boundary condition and also through each window, vent, door and hole in the selected domain.
- Check that total flow in balances with total flow out (a key requirement for the CFD calculations). If flows do not balance there are options to automatically apply a correction flow to one of the existing openings to ensure balance.
- Check and set the temperatures of each boundary condition where temperature is defined (surfaces, supply flow, windows, temperature boundary conditions).
- Import surface and opening boundary conditions for a specific time/date from a previous EnergyPlus simulation.



CFD Analysis Domain

You can select the CFD Analysis domain from previously defined CFD Analyses - this defines the extent of the model that is displayed and checked for flow balance. If you have more than one analysis then you must make sure to select the correct one here before starting. Alternatively you can select the whole building or the

current block or zone. This selection does not limit the EnergyPlus simulation data that is imported, it only defines the boundary condition data that is displayed on the dialog and the domain for airflow balance checks.

Group

The Group options allows you to group boundary condition data by one of:

- **1-None** - each boundary condition is displayed in a simple list without any grouping.
- **2-Boundary types** - boundary conditions are displayed groups according to type (extract, supply, temperature, window etc). See screenshot above.
- **3-Zones and surfaces** - the boundary conditions are grouped by zone and then by surface.

Under some circumstances the display can be erratic when using the **2-Boundary type** grouping and in this case you may find it better to use one of the other grouping options.

You can edit the **flow in** and **flow out** airflow rates and temperatures as appropriate on the dialog and the changes are only saved to the model when you press the OK button. No changes are made if you press Cancel.

Edit Boundary Conditions

Airflow

The Zone, Surface, Boundary and Boundary Type columns indicate the local of the boundary condition and it's type. You can edit the flow rates in the Flow In and Flow Out columns for windows, vents, doors and holes and the Flow In for supply diffusers and Flow Out for extracts.

If flows do not already balance this will be indicated in red on the Info panel to the right and you can select one of the openings in the dialog and click on the **Correct balance using selected opening** link. A flow is added or subtracted as appropriate to get flows to balance.

Temperature

You can edit the temperature for all temperature-related boundary conditions. There are 2 temperature columns, one for surface temperatures and the other for air temperature. The air temperature column is only relevant for openings and supply diffusers and is the temperature of the air entering the space. Openings such as windows, vents and doors can have different surface and air temperatures defined. If the airflow rate through the opening is zero then you will not be able to edit the data in the air temperature column.

Import EnergyPlus Boundary Conditions

You can import data that have been calculated in a previous EnergyPlus simulation as boundary conditions for the CFD simulation. The data that can be imported is:

- Surface inside temperatures of walls, floors, roofs, partitions, ceilings, windows, doors and sub-surfaces.
- Flow in and flow out of windows, vents, doors and holes (only when using the Calculated natural ventilation model option).

To import simulation data click on the **Import** button in the Boundary Conditions Editor (above).

Import Dialog

The Import data dialog allows you to select boundary conditions for a particular snapshot in time from the simulation results.

Interval

You can read CFD boundary conditions from either sub-hourly or hourly results. Select from:

- **1-Sub-hourly** where boundary condition data is loaded from sub-hourly EnergyPlus results.

- **2-Hourly** where boundary condition data is loaded from sub-hourly EnergyPlus results.

Time, Day, Month

Select the time and date to use for the import. The date must lie between the simulation start and end periods for the last simulation carried out.

Press OK to import the results to the CFD Boundary Conditions Editor.

When the **Interval** is set to **1-Sub-hourly** the times listed in the drop list will be in the format Hour:Minutes based on the currently set [Number of timesteps per hour](#). When the **Interval** is set to **2-Hourly** the time is simply the hour.

Note: all available opening airflow and inside surface temperature results from the most recent EnergyPlus simulation will be imported. That is the imported results are not restricted to the current CFD domain or zone (if this is a subset of the whole building).

Preparing Simulations To Generate CFD Boundary Conditions

You can use DesignBuilder simulations to generate boundary conditions for the CFD calculations. EnergyPlus simulations calculate the overall thermal behaviour of the building in terms of inside surface temperatures and natural ventilation flow rates then CFD allows you to 'zoom in' on the details of the temperature and flow distribution within the space for a particular snapshot in the simulation.

The simulation data that can be used as boundary conditions in CFD calculations are:

- Surface inside temperatures
- Natural ventilation airflow rates through external windows, doors, vents and holes.

To obtain access to the above data the simulation must generate inside surface temperatures and any natural ventilation airflow rates to be included in the CFD calculations. Before starting you should decide on the extent of the CFD domain and the time period you wish to analyse. Then only generate simulation data for these areas and time periods. Otherwise, for large models you might generate an impractical amount of data and overload the software.

The steps required to generate boundary condition data from simulation are:

1. If the CFD simulation is to use natural ventilation airflows through windows calculated by EnergyPlus then you must use the [Calculated natural ventilation](#) model option. [Scheduled](#) natural ventilation does not calculate airflows through individual windows.

2. Because CFD calculations do not use infiltration flows you may want to switch off [Infiltration](#) treatment on the [Constructions](#) model data tab. This will help ensure that the flow balance with little or no need for extra correction flows to be introduced on the [CFD Boundary Conditions](#) dialog. It also speeds up simulations by reducing the complexity of the Airflow network in EnergyPlus.
3. Switch off the [Lump similar windows on surface](#) model option to ensure that all windows are modelled individually.
4. Prepare the simulation model data and carry out test simulations to check that the model is behaving as expected during the period of interest (e.g. summer design conditions). Look at hourly results.
5. Navigate to the level in the model covering the CFD domain. If you are modelling a single zone in the CFD simulation, then navigate to that zone, if you are modelling a block then go that block or if you are modelling the whole building then go to building level. Then on the [Options](#) model data tab select the following output options:
 - **Store opening output** - ensure that data for windows, vents, doors, holes and sub-surfaces is included in the output.
 - **Inside Surface Temperature** - to store inside surface temperatures for all surfaces and openings in the domain.
 - **Airflow in** - to record the flow in through windows, vents, doors and hole (natural ventilation calculations only).
 - **Airflow out** - to record the flow out through windows, vents, doors and hole (natural ventilation calculations only).
6. Once the model is prepared, click on the Simulation tab and request **hourly** or **sub-hourly** results and run the simulation.
7. Check the simulation results by examining hourly results before feeding them into the CFD calculations. Check that airflow through windows is in the expected direction based on wind direction and expected wind pressures etc.
8. Start the [CFD Boundary Conditions Editor](#) and follow instructions for importing the simulation hourly or sub-hourly results
9. Add any assemblies or component blocks with any further boundary conditions needed to represent heat, temperature and flow boundaries imposed by HVAC, lighting and other equipment and occupants.
10. Make sure you have the same total convective heat from your internal gains in the CFD model as you had in the EnergyPlus simulation.
11. Check that CFD results correspond with the boundary conditions provided by the simulation.

Note: It is not currently possible to set flow boundary conditions on *internal* openings. So although EnergyPlus calculates flow rates through interior openings, DesignBuilder CFD will ignore these flows and re-calculate them using a more detailed method taking into account distributed air velocities and pressures. This means that the flows through internal openings calculated by the CFD won't match exactly with the flows calculated by the EnergyPlus Airflow network. EnergyPlus flow rates through *exterior* openings are always respected by DesignBuilder CFD when used as boundary conditions.

Note: [Infiltration](#) airflow is not modelled in CFD and if you are looking at summer time natural ventilation you may wish to switch Infiltration off in simulations. Winter time simulation may need infiltration included if it is significant.

Infiltration In CFD

DesignBuilder CFD doesn't include the concept of infiltration as cracks cannot be graphically defined and there can be no porous airflow through CFD surfaces as there can in EnergyPlus. Because airflow through cracks in simulations using [Calculated natural ventilation](#) can be significant, [importing](#) EnergyPlus opening airflows from simulations that include crack leakage can sometimes cause flow balance problems, especially in zones with only a single opening. It is therefore usually advisable to exclude the effects of [infiltration](#) from simulations that are to be used for setting boundary conditions for CFD. A warning message to this effect is given if you attempt to import simulation data generated using Calculated natural ventilation and infiltration.

Another reason for excluding infiltration from simulations to be used for setting up CFD boundary conditions is that the flow is likely to [balance](#) much more easily without the need to add large balancing flows to one or more openings.

Assembly Library

DesignBuilder is provided with an assembly library which contains a small database of typical components that may be useful in defining a CFD model. Some assemblies in the library include thermal gains while others simply provide an obstruction to airflow.

Add Assembly To Library

You can add your own assemblies to the model assembly library by selecting one or more component blocks when at building or block level and using the **Add assembly to library** toolbar command (or **Edit > Add assembly to library** menu option). This causes the Drawing options panel in the bottom left of the screen to be opened. Here you can enter a name for the assembly and define any default boundary conditions.

Note: The boundary conditions selected here will act as default settings at assembly block level and will not override any hard-set attributes made within the constituent component blocks themselves.

The full process is as follows:

1. Select the component blocks and/or CFD fans to be included in the assembly at block or building level. It does not matter which of these levels the blocks are originally created at. It will be possible to import the resultant assembly at building or block level regardless of where it was created.
2. Click on the **Add assembly to library** toolbar command.
3. In the Drawing options panel enter a unique name for the assembly and the default assembly boundary conditions.
4. Select the [Assembly creation mode](#).
5. Move the mouse to highlight the plane of the assembly to be used as the attachment plane and click. The **attachment plane** will be used later when placing the assembly.
6. Then click again to define the **attachment point** on the attachment plane. This will be the point that will be used to define the actual position of the assembly when it is placed later.
7. Click again to add the assembly to the library.

Assembly creation mode

Select from 3 options:

- **1-Replace with assembly instance** - the selected component blocks will be replaced by the new assembly.
- **2-Leave component blocks** - the selected component blocks are left unaffected by the creation of the new assembly.
- **3-Delete** - the selected component blocks are deleted once the new assembly has been created.

Export/Import

The assembly library data can be exported for use in other projects using the **File > Export > Export assembly library** menu command.

The exported assembly library .asl file can be loaded into other projects using the **File > Import > Import assembly library** menu command.

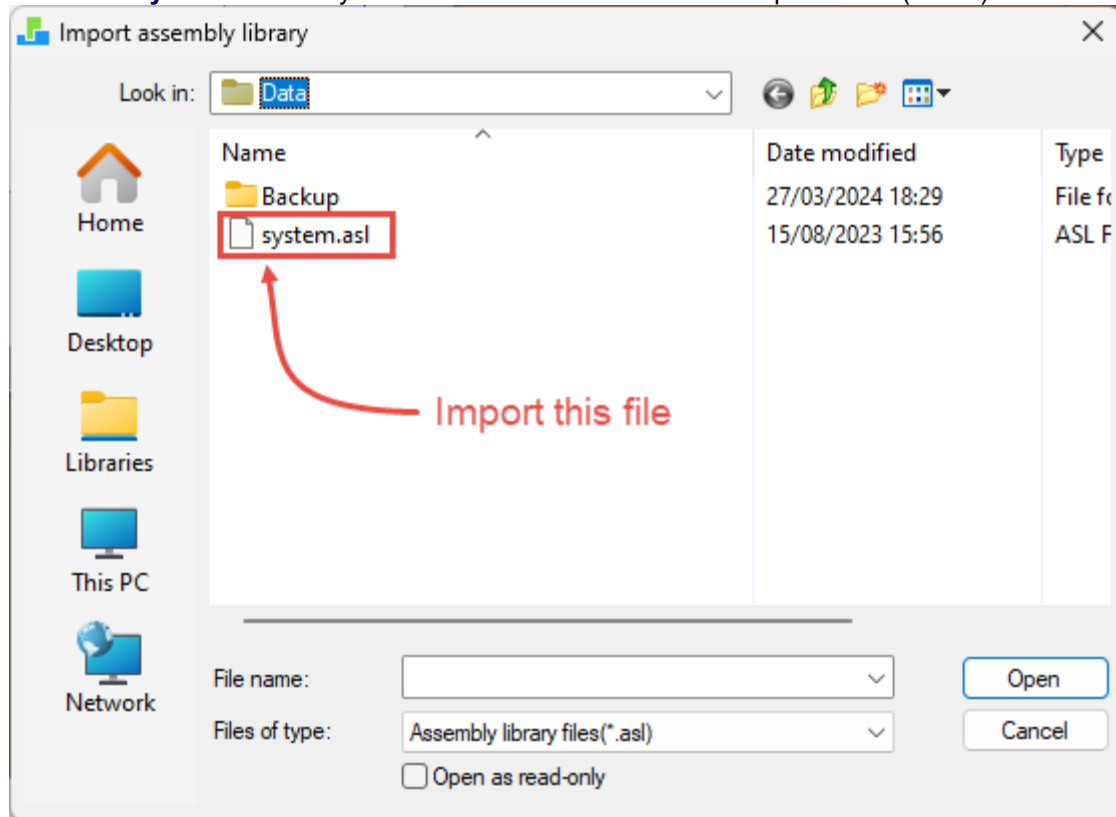
See also:

- "Assembly Block" on page 205
- "Component Blocks & Assemblies as CFD Boundaries" on page 1922

Updating The Assembly Library

From time to time new assemblies are added to the DesignBuilder assembly library and these new assemblies will be available when a new model is created with the new version of the software. However, previous models do not include newly added assemblies by default. To update the assembly library in an older model, follow the steps below.

1. Open the model requiring the updated library.
2. Select the **Import > Import assembly library** menu command.
3. Select the **system.asl** library file that is included in the default import folder (below).



The assembly library in your loaded project should now be up to date with the latest library provided with the software.

Detailed Equipment

Equipment tab in model data (*zone level only*)

Warning: This option is intended for modelling existing buildings where individual items of equipment have been recorded as part of a detailed survey and is not intended for general design or assessment simulation work.

When the [Gains model option](#) is set to **Detailed** you can enter data on individual items of equipment in each zone.

[Details on editing Equipment data](#)

Note: For other modelling applications you should use the [Early gains model option](#) because data can be set at the building level and/or loaded from template with much less effort.

Number	Rated power ...	Schedule	Radiant fraction	Latent fraction	Fraction heat lost
Computer accessories					
1	0,0000	On	0,10	0,00	0,00
Computers					
10	150,0000	8:00 - 18:00 ...	0,10	0,00	0,00
Fluorescent lights					
36	32,0000	On	0,10	0,00	0,05

Edit Equipment

[Detailed gains](#)

Use this screen to edit the equip for the current zone

With 'Detailed' gains detail, info for each item of energy using equip its' operation is listed in detail.

This mode must be used with 7/ Operation Schedules.

[Add equipment](#)

[Edit selected equipment](#)

[Delete selected equipment](#)

Edit Detailed Equipment

You can model time-varying internal gains and energy consumption of individual items of equipment in each zone by adding equipment data at the zone level on the [Equipment tab](#). To add an item of equipment click on the Add toolbar icon, or if you are running DesignBuilder in Learning mode you can use the link in the Info panel to the right of the screen. This opens the **Edit Equipment** dialog.

You can define the category of internal gain (below), the number of items of equipment, the rated power per item of equipment as well as the operational schedule.

The **Radiant fraction** is the fraction of heat emitted by the device as long-wave radiation. The rest of the heat is considered to be convective i.e. it is modelled as heating up the zone air directly.

The **Latent fraction** is the fraction of the rated power which is converted to latent energy and affects the moisture balance in the zone instead of the sensible heat balance.

A number of categories of gains are available. These are listed below together with the output data with which they are associated:

- **Fluorescent lights** - General lighting
- **Tungsten lights** - Task lighting
- **Office equipment** - Computer + equipment
- **Computers** - Computer + equipment
- **Computer accessories** - Computer + equipment
- **Telecoms** - Computer + equipment
- **Lifts** - Miscellaneous
- **Catering** - Catering
- **Small power** - Miscellaneous
- **Heating** - Miscellaneous
- **Cooling** - Miscellaneous
- **Pumps** - Miscellaneous
- **Fans** - Miscellaneous
- **DHW** - Miscellaneous.
- **Process** - Miscellaneous
- **External lighting** - Task lighting

- **Other external** - Miscellaneous
- **Miscellaneous** - Miscellaneous

Note that all of the above gains are modelled as scheduled gains in the zone and the data is reported as such in the program output. So for example categorising an item of equipment as DHW does not affect in any way [DHW modelling](#) as defined on the [HVAC tab](#). Likewise data for Heating, Cooling, Pumps and Fans is simply modelled as a scheduled gain in the zone, completely unconnected with HVAC modelling.

Cost Data Overview

This page provides an overview of the various sources of data used in a building construction cost summary calculation.

The capital cost of constructing the building is defined in a number of places in the model:

1. Within the building model on component dialogs:
 - a. [Construction](#) - cost of constructions
 - b. [Material](#) - cost of material layers (cost per area or volume),
 - c. [Glazing](#) - cost of glazing systems (cost per window area)
 - d. [Window shading devices](#) such as blinds and shades (cost per window area)
 - e. [Local shading](#) such as overhangs and louvres (cost per window area)
 - f. [Wind turbines](#) connected to Load centres.
 - g. [Renewable generation distribution and electrical costs](#) from Load centres.

and on model data tabs:

- a. [Constructions](#) - sub-structure, structure and surface finishes.
 - b. [Lighting](#) - cost of lighting systems (cost per GIFA)
 - c. [HVAC](#) - cost of HVAC systems (cost per GIFA or capacity)
2. Building cost summary data on the [Building Cost Summary](#) dialog (accessed from [Tools](#) menu).

This data together with the building geometry allows the construction cost (aka capital expenditure or Capex) of the building to be calculated.

A [report of the cost](#) can be viewed on the Cost and Carbon screen.

Tip: The Capex totals can also be used as objective or constraints within optimisation or parametric studies.

Note: The costs provided in DesignBuilder's library databases were derived from a range of sources, including consultations with a professional cost modeller, published materials and extrapolation. They are intended to be indicative only and a starting point for users to build their own project and region-specific cost data sets.

1. Super-Structure And Sub-Structure Costs

In general the sub-structure cost is the cost of all works below ground level and super-structure cost is the cost of all works above ground level.

Opaque wall, roof, floor and partition fabric costs are calculated using the model geometry, and construction selected for each surface. The cost of each surface is calculated from its area and the type of construction

used. For an external wall the cost for a surface is calculated from its surface area (minus the area of any windows and other openings) and the cost of the construction per area. The cost of the construction per area is calculated in the way defined on the [Cost tab](#) for the construction.

Note: If a construction has more below ground surface area than above ground surface area, then this construction is defined as a below ground construction.

Glazing costs are calculated using the model geometry, the window areas and the glazing selected for each window. The cost of each window is calculated from its area and the type of glazing used. The cost of the glazing per area is defined on the [Cost tab](#) for the glazing component.

Note: For costing purposes the Roof/wall transition angle is assumed to be a fixed 70°, which mean if the slope is greater than 70°, then it is defined as a window, otherwise it is defined as a skylight.

Sub-structure costs are calculated as the sum of the [Sub-structure settings](#) on the Constructions tab and the construction costs of below-grade surfaces.

2. Building Cost Summary Data

You can access the extra building cost summary data on the [Building Cost Summary](#) dialog which is accessed from the Tools menu.

Building Cost Summary Data

The Building Cost Summary data dialog is accessed from the [Tools menu](#).

The dialog includes controls to allow you to select the cost calculation method. When either the NRM1 or UNIFORMAT II formats are selected, additional summary cost data for building can be entered to support the calculation.

Tip: The [Cost data](#) help topic provides an overview of the places in DesignBuilder where costs are entered.

Cost model

Select the model to use from:

- **1-Basic** for a simple report including only building construction costs.
- **2-UNIFORMAT II** for a summary cost report based on the standard ASTM UNIFORMAT II format.
- **3-NRM1** for a summary cost report based on the standard RICS New Rules of Measurement 1 format.

Calculation method

HVAC calculation method options are:

- **1-Cost per area** where the cost of the HVAC system is defined per floor area where typical only data is available.
- **2-Cost per capacity** where the cost of the HVAC system is defined per heating and/or cooling capacity of the zone HVAC system. This is normally a more realistic method given the right cost data. It requires the heating and cooling design calculations to have been previously carried out before cost reports can be provided. In the case of optimisation studies HVAC heating and cooling capacities are determined from the main simulation design sizing outputs.

UNIFORMAT II

UNIFORMAT II is one of the most widely used element-based classification standards in North America. Developed through an industry and government consensus, the standard is intended to reduce the effort in construction planning, scheduling and cost estimation in early design phases. Using it helps to ensure consistency in the economic evaluation of building projects over time and from project to project. It also enhances project management and reporting at all stages of the building life cycle—planning, programming, design, construction, operations, and disposal.

Elements are classified in three hierarchical levels of definition. Level 1 identifies seven Major Group Elements, such as the Substructure, Shell, and Interiors. Level 2 further subdivides Level 1 into Group Elements. The Shell, for example, includes the Superstructure, Exterior Closure, and Roofing. Level 3 breaks the Group Elements further into Individual Elements. Exterior enclosure, for example, includes Exterior Walls, Windows, and Exterior Doors.

Note: While DesignBuilder requires most of its data to be input using **Level 2** elements, it reports costs using both Level 1 and 2 formats.

Building and sitework elemental costs are presented with cost analysis parameters providing an efficient tool for communicating economic information to decision makers in a quickly understood, concise format that helps them make project choices. Owners, developers, programmers, cost planners, project managers, schedulers, architects and engineers, operating and maintenance staff, manufacturers, specification writers, and educators use the classification.

A major benefit of performing an economic analysis based on an elemental framework rather than on a product-based classification is the reduction in time and costs for evaluating alternatives at the early design stage. This encourages more economic analyses and more economically efficient choices among buildings and building elements. Other UNIFORMAT II benefits include providing a standardized format for collecting and analyzing historical data for use in estimating and budgeting future projects; providing a checklist for the cost estimation process as well as the creativity phase of the value engineering job plan; providing a basis for training in cost estimation; facilitating communications among members of a project team regarding the scope of work and costs in each discipline; and establishing a database for automated cost estimating.

The latest version of the UNIFORMAT II standard can be purchased at: <https://www.astm.org/Standards/E1557.htm> and a useful report from NIST - [NISTIR 6389](#).

The data that is not already included in the model should be entered on the UNIFORMAT II tab of this dialog. Much of the data is structured as a cost per area or per each followed by the area or the number of items in that category.

A Substructure

The [substructure costs](#) are defined in the model through use of zone cost per floor area data (A10) and the constructions themselves within the below grade constructions (A20):

- **A10 Foundation:** includes standard foundation, special foundations and slab on grade. The area used in the calculation is the ground floor area.
- **A20 Basement construction:** includes basement excavation and basement walls.

B Shell

The cost of the superstructure is defined within the model through the area of the various exterior surfaces making up the superstructure and the cost per area of their constituent opaque and glazed constructions.

- **B10 Superstructure:** includes floor and roof construction
- **B20 Exterior enclosure:** includes exterior walls, [structure](#), windows and doors
- **B30 Roofing:** includes roof coverings and roof openings

C Interiors

The cost of interior construction (C10) is defined within the model through the area of the various surfaces making up the interior construction and the cost per area of their constituent opaque and glazed constructions.

C10 Interior construction: includes partitions, interior doors and fittings

C20 Stairs: includes stair construction and stair finishes

C30 Interior finishes: includes wall finishes, floor finishes and ceiling finishes

D Services

Note that [HVAC costs](#) (D30) are included in the model.

D10 Conveying: includes elevators, lifts, escalators, moving walks and other conveying systems

D20 Plumbing: includes plumbing fixtures, domestic water distribution, sanitary waste, rain water drainage, and other plumbing systems

D30 HVAC: includes energy supply, heat generating systems, cooling generating systems, distribution systems, terminal & package units, controls and instrumentation, systems testing & balancing and other HVAC systems and equipment. This data is included zone by zone in the model on the [HVAC tab](#) and so does not need to be entered again here.

D40 Fire protection: includes sprinklers, standpipes, fire protection specialities and other fire protection systems.

D50 Electrical: includes electrical service & distribution, lighting & branch wiring, communication & security and other electrical systems.

E Equipment And Furnishings

E10 Equipment: includes commercial equipment, institutional equipment, vehicular equipment and other equipment

E20 Furnishing: includes fixed furnishings and movable furnishings

F Special Construction And Demolition

F10 Special construction: includes special structures, integrated construction, special construction systems, special facilities, special controls and instrumentation. Renewables costs, ([PV panels](#), [distribution and electrical](#), solar hot water systems and wind turbines) are included here.

F20 Selective building construction: includes building elements demolitions and hazardous components abatement.

G Sitework And Utilities

G10 Site preparation: includes site clearing, site demolition and relocations, site earthwork and hazardous waste removal/remediation.

G20 Site improvements: includes roadways, parkings lots, pedestrian paving, site development and landscaping.

G30 Site mechanical utilities: includes water supply, sanitary sewer, storm sewer, heating distribution, cooling distribution, fuel distribution and other site mechanical utilities.

G40 site electrical utilities: includes electrical distribution, site lighting, site communications & security and other site electrical utilities

G50 Other site construction: includes service and pedestrian tunnels and other site systems.

Z General Requirements And OH And P

Z10 Design allowance: cost of design work.

Z20 Field Requirements, office overhead & profit: includes field requirements and office overhead & profits

Z30 Inflation allowance.

NRM1

Published by the Royal Institute of Chartered Surveyors (RICS), NRM1 provides guidance on the quantification of building works for the purpose of preparing cost estimates and cost plans. It is described by RICS as the cornerstone of good cost management of construction projects, enabling more effective and accurate cost advice to be given to clients and other project team members, as well as facilitating better cost control. It helps by providing a standard format and sequence for priced works for the purpose of providing best practice guidance on the preparation of order of cost estimates, elemental cost models, cost plans, cost analyses and benchmark analyses.

The latest version of this standard can be downloaded from the [RICS website](#).

The data that is not already included in the model should be entered on the NRM1 tab of this dialog. Much of the data is structured as a cost per area or per each followed by the area or the number of items in that category.

Facilitating Works

- **Toxic / hazardous / contaminated material treatment:** Includes toxic or hazardous material removal, contaminated land and eradication of plant growth
- **Major demolition works:** Includes demolition works and soft strip works, temporary support to adjacent structures
- **Specialist groundwork's:** Includes site dewatering and pumping, soil stabilisation measures and ground gas venting measures
- **Temporary diversion works**
- **Extraordinary site investigation works:** Includes archaeological investigation, reptile / wildlife mitigation measures and other extraordinary site investigation works.

Substructure

The [substructure costs](#) are defined in the model through use of zone cost per floor area data and the constructions themselves within the below grade constructions.

- **Substructure:** Includes standard foundations, specialist foundations, lowest floor construction, basement excavation and basement retaining walls

Superstructure

The cost of the superstructure is defined within the model through the area of the various exterior surfaces making up the superstructure and the cost per area of their constituent opaque and glazed constructions. The exception is the stairs and ramps cost which is entered on this dialog.

- **Frame:** Includes steel frames, space frame / decks, concrete casings to steel frames, concrete frames, timber frames or specialist frames
- **Upper floors:** Includes floors, balconies and drainage to balconies.
- **Roof:** Includes roof structure, roof coverings, specialist roof systems, roof drainage, rooflights, skylights and openings.
- **Stairs and ramps:** Includes stair/ramp structures, stair/ramp finishes and stair/ramp balustrades and handrails.
- **External walls:** Includes external enclosing walls above ground level, external enclosing walls below ground level, solar/rain screening, external soffits, subsidiary walls, balustrades, proprietary balconies and facade access/cleaning systems
- **Windows and external doors:** Includes external windows and external doors.
- **Internal walls and partitions:** Includes internal walls, partitions, balustrades and handrails, moveable room dividers and cubicles
- **Internal doors**

Internal Finishes

[Internal finishes data](#) is entered for each surface using inheriting defaults on the Constructions tab.

- **Wall finishes**
- **Floor finishes:** includes finishes to floors and raised access floors
- **Ceiling finishes:** includes finishes to ceilings, false ceilings and demountable suspended ceilings (**be careful not to double count this in the construction costs**).

Fittings, Furnishings And Equipment

- **Fittings, furnishings and equipment:** includes general, domestic kitchen and special purpose fittings, furnishings and equipment. In addition, it also includes signs or notices, works of art, non-mechanical and non-electrical equipment, internal planting and bird and vermin control.

Services

- **Sanitary installation:** includes sanitary appliances, sanitary ancillaries
- **Service equipment**
- **Disposal installations:** includes foul drainage above ground, chemical, toxic and industrial liquid waste drainage and refuse disposal
- **Water installations:** includes main water supply, cold and hot water distribution, local hot water distribution and steam and condensate distribution
- **Heat source**
- **Space heating, air conditioning and ventilation (HVAC):** includes central heating, local heating, central cooling, local cooling, central and local air conditioning, central and local ventilation, special ventilation and smoke extract or control.
- **Electrical installations:** includes electrical mains and sub-mains distribution, power installations, lighting installations, specialist lighting installations, local electricity generation systems and earthing and bonding systems
- **Fuel installations:** includes fuel storage and fuel distribution systems
- **Life and conveyor installations:** includes lifts and enclosed hoists, escalators, moving pavements, powered stairlifts, conveyors, doc levellers and scissor lifts, cranes and unenclosed hoists, carlifts, car stacking systems, tumtables and the like, document handling systems and life and conveyor systems.
- **Fire and lightning protection:** includes fire fighting systems, fire suppression systems and lightning protection
- **Communication, security and control systems:** includes communication systems, security systems and central control/building management systems
- **Specialist installations:** includes specialist piped supply installations, specialist refrigeration systems, specialist mechanical installations and specialist electrical/electronic installations.
- **Builder's work in connection with service**

Complete Buildings And Building Units

- **Prefabricated buildings and building units:** includes complete building, building units and pods

Work To Existing Buildings

- **Minor demolition works and alteration works**
- **Repairs to existing services**
- **Damp-proof courses/fungus and beetle eradication**
- **Facade retention**
- **Cleaning existing surfaces:** includes cleaning existing surfaces and protective coatings to existing surfaces.
- **Renovation works:** includes masonry repairs, concrete repairs, metal repairs, timber repairs and plastic repairs.

External Work

- **Site preparation works:** includes site clearance and preparatory groundworks
- **Roads, paths, pavings and surfacing:** includes roads, paths and pavings, special surfaces and pavings.
- **Soft landscaping, planting and irrigation systems:** includes seeding and turfing, external planting and irrigation systems
- **Fencing, railings and walls:** includes fencing and railings, walls and screens, retaining walls, barriers and guardrails
- **External fixtures:** includes site/street furniture and equipment and ornamental features
- **External drainage:** includes surface water and foul water drainage, ancillary drainage systems, external chemical, toxic, industrial liquid waste drainage and land drainage
- **External services:** includes water main supply, electricity mains supply, external transformation devices, electricity distribution to external plant and equipment, gas main supply, telecommunications and other communication systems, external security systems, external/street lighting systems, local/district heating installations, builder's work in connection with external services.
- **Minor building works and ancillary buildings**

Main Contractors Preliminaries

- **Employer's requirements:** includes site accommodation, site records and completion and post-completion requirements
- **Main contractor's cost items:** includes management and staff, site establishment, temporary services, security, safety and environmental protection, control and protection, mechanical plant, temporary works, site records, completion and post-completion requirements, cleaning, fees and charges, site services, insurance, bonds, guarantees and warranties.

Main Contractor's Overhead And Profits

- Main contractor's overheads
- Main contractor's profits

Project Design Team Fee

- **Consultant's fees:** includes project/design team consultants' fees or other consultants' fees, site investigation fees and specialist support consultants' fees
- **Main contractor's pre-construction fees:** includes management and staff, specialist support services fees, temporary accommodation, services and facilities charges, and main contractor's overheads and profits.
- **Main contractor's design fees:** main contractor's design consultant's fees
- **Other development/project costs:** includes land acquisition costs, employer finance costs, fees, charges, planning contributions, insurances, archaeological works, other specialist fieldwork, decanting and relocation costs, tenant's costs/contributions, marketing costs and other employer costs.

Risk Allowance Estimate

- **Design development risks**
- **Construction risks**
- **Employer change risks**
- **Employer other risks**

Inflation Allowance

- **Tender inflation**
- **Construction inflation**

Model Data Management

Tools are provided to help manage the building model:

- "Model Data Grid View (MDGV)" on page 656 - view and edit model data in a grid
- [Load Data from Template](#) - load predefined data sets to the model from a template
- "Save Data to Template" on page 689 - save model data to create a new template
- "Reload Templates" on page 691 - reload any "hard-selected" templates, i.e. those shown in red on the model data user interface
- [Clear Data to Default](#) - clear data back to default values inherited from level above in model hierarchy)
- [Merging Zones](#) - control merging of zones from one dialog
- [Load Whole Building Infiltration](#) - load measured results from pressurisation tests

Model Data Grid View (MDGV)

Accessed from [Tools](#) menu

The **Model Data Grid View** (MDGV) tools allow the model data for the current building to be viewed and/or edited using a grid. It is possible to select from a range of "layouts" which define which data is to be displayed.

Viewing the building model data in a grid provides a global view of the data across the whole model and makes comparisons between settings for different parts of the model much easier than when using the usual model data input screen.

See also the [Model Data Grid View & QA checking](#) tutorial

Management Dialog

The MDGV tools are all accessed from the main **Model Data Grid View Management** dialog which is launched from the  toolbar icon or from the [Tools](#) menu and is active for the current building in the model. The Model Data Grid View tools can be accessed from building level and below, but are not available from site level.

The title of the dialog identifies the currently selected analysis type. The dialog presents a list of layouts which are:

- Filtered to those that are compatible with the current model's currently selected [Analysis type](#).
- Grouped by the level for at which they display data e.g. Building, Zone, Surface, Opening.
- Indicated as "System" or "User" under the **Layout Type** column.

The management dialog provides tools to access 2 different areas of functionality:

- [Model Data](#) – with **View** / **Edit** / **Import** buttons which can be used to access the model data itself.
- [Layout Management](#) – with **New** / **Edit** / **Copy** / **Delete** buttons which can be used to manage and modify the layouts, i.e. the definitions of which model data is to be shown.

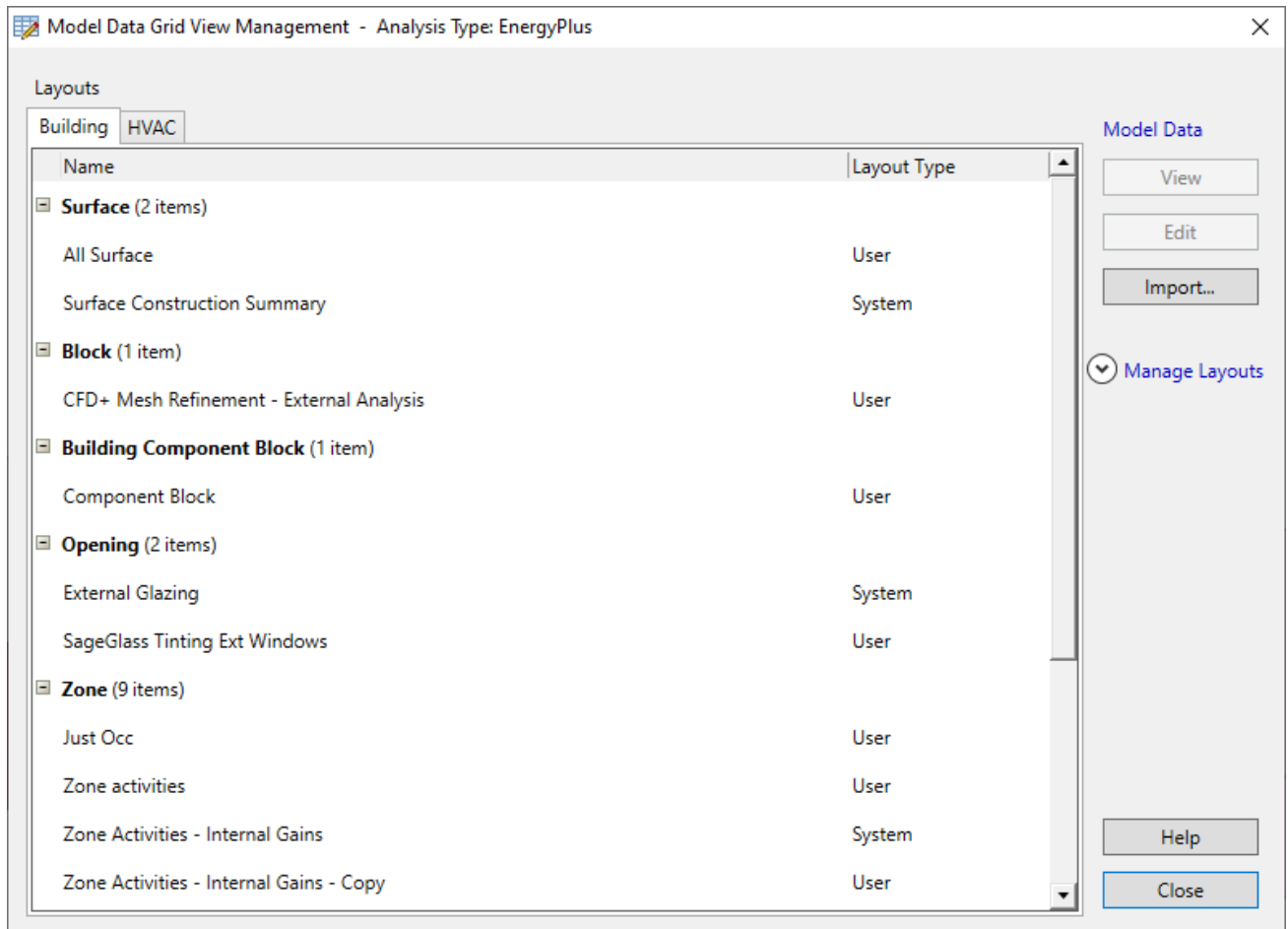
You can use the Model Data Grid View tools to view and edit model data for the following objects:

- Buildings
- Building blocks
- Component and assembly blocks (at building and block levels)
- Zones
- Surfaces
- Openings

- All HVAC loops and components

The "building" model data is accessed by clicking on the **Building** tab. This lists all of the above layouts apart from the HVAC components as shown in the screenshot below.

For example, to edit window model data in the grid, click on the "External Glazing" layout and then click on the **Edit** button under the blue Model Data header in the top right.

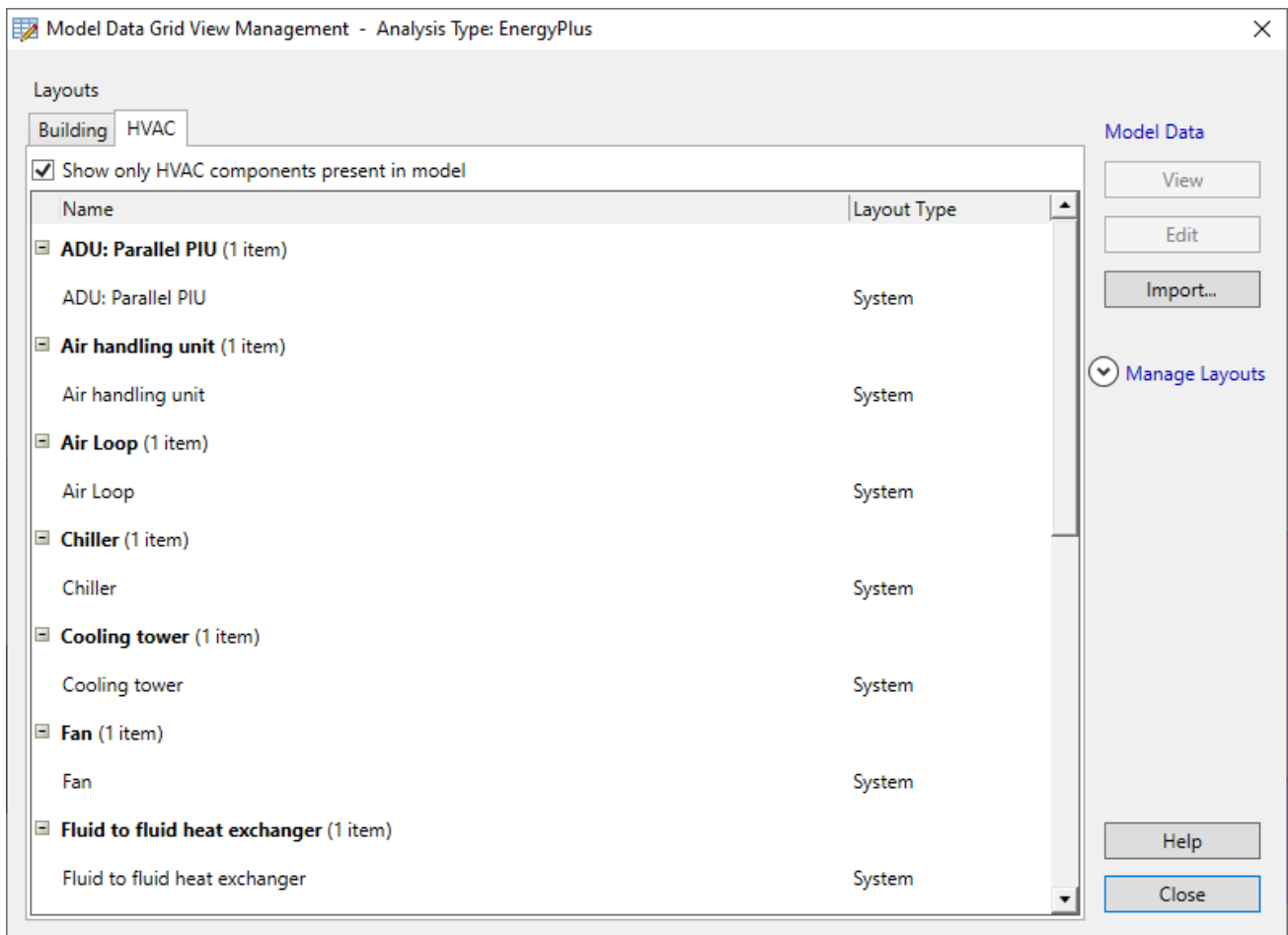


The text in the **Layout Type** column (see above screenshot) flags whether the layout was provided by DesignBuilder with the program ("System"), or whether it is a custom layout created by the user ("User").

The HVAC data is accessed by first clicking on the **HVAC** tab to list available layouts.

You can choose to list only the HVAC components present in the model by checking the checkbox above the list. It is checked by default. Otherwise all available layouts are listed even if the corresponding components are not in the model.

For example, to edit the air loops in the grid, click on the "Air Loop" layout and then click on the **Edit** button under the blue Model Data header in the top right.



For more details on Model Data Grid View see:

- "Model Data Grid View - Viewing and Editing Model Data" on page 658
- "Model Data Grid View - Importing Model Data" on page 669
- "Model Data Grid View - Layout Management" on page 671
- "Model Data Grid View - Layout Editing" on page 674

Model Data Grid View - Viewing And Editing Model Data

Layout Selection

One or more layouts can be selected from the Layouts list on the Management dialog. When multiple layouts are selected the sum of all attributes from each layout is used to determine which model data will be presented.

Multiple layout selection is achieved using the <Ctrl> key and selecting additional layouts under the same level grouping. To select multiple layouts for viewing / editing they must be under the same level grouping, e.g. **Zone**.

Launching The View/Edit Grid Dialog

Once selected, model data for the chosen layout(s) can be viewed or edited by clicking the **View** or **Edit** button under the **Model Data** header on the **Model Data Grid View Management** dialog. This causes the selected model data to be loaded into the Model Data Grid dialog.

The only difference between view and edit modes is that in edit mode the data can be edited, while in view mode it cannot. The section below explains how to switch between the two modes.

The dialog title identifies the chosen layout (the first chosen layout and "... " if more than one layout was chosen). Below is an example of the data grid view created from selecting 2 layouts at Zone level:

Zone HVAC... [Read Only]													
☐ Edit Clear Filters Export...													
Search													
Block													
Activity													
Floor Areas and Volumes													
Mechanical Ventilation													
Heating													
Cooling													
Operation													
HVAC													
Zone	Floor area (m2)	On	Outside air definitio...	Outside air (ac/h)	Heated	Fuel	Heating system seas...	Cooled	Cooling system	Fuel	Cooling system seas...	Schedule	Humidification
1F first floor East (1 item)													
Open office	239.6	☒	2-Min fresh air (Per...	3.000	☒	2-Natural Gas	0.830	☒	Default	1-Electricity from grid	1.670	Office_OpenOff_Cool	☐
1F first floor West (1 item)													
Open office	239.6	☒	2-Min fresh air (Per...	3.000	☒	2-Natural Gas	0.830	☒	Default	1-Electricity from grid	1.670	Office_OpenOff_Cool	☐
0F ground floor West (1 item)													
Open office	239.6	☒	2-Min fresh air (Per...	3.000	☒	2-Natural Gas	0.830	☒	Default	1-Electricity from grid	1.670	Office_OpenOff_Cool	☐
0F ground floor East (1 item)													
Open office	239.6	☒	2-Min fresh air (Per...	3.000	☒	2-Natural Gas	0.830	☒	Default	1-Electricity from grid	1.670	Office_OpenOff_Cool	☐
Core circulation (1 item)													
Circulation	101.6	☒	2-Min fresh air (Per...	3.000	☒	2-Natural Gas	0.830	☒	Default	1-Electricity from grid	1.670	Office_Circulation_C...	☐
Atrium Reception (1 item)													
Reception	44.8	☒	2-Min fresh air (Per...	3.000	☒	2-Natural Gas	0.830	☒	Default	1-Electricity from grid	1.670	Office_Reception_Cool	☐
Rooftight (2 items)													
Reception rooftoplight...	---	---	---	---	---	---	---	---	---	---	---	---	---
Core rooftoplight > Cor...	---	---	---	---	---	---	---	---	---	---	---	---	---

Switching Between Edit and View Modes

In the top left of the data grid dialog is the **Edit** checkbox. When the data grid is in edit mode this is checked. It can be unchecked and the data grid will be in view mode in which case data can no longer be edited. If any edits had been made prior to switching to View mode you will be prompted to establish whether the edits should be applied or discarded.

Columns left of the fixed splitter

Depending on the level grouping the layout belongs to there will be a set amount of viewable data on the left hand side of the data grid view and default grouping detailed as below:

Level	Set Data	Default Grouping
Site	Site Name	None
Building	Building Name	None
Block	Block Name	None
Zone	Zone Name	Block
Surface	Surface Name, Type, Area, Orientation, Slope	Block, Zone
Opening	Type, Adjacency, Area, Orientation, Slope	Block, Zone, Surface

A “splitter” separates the set data and model data attributes on the data grid view.

	Activity
	Floor Areas and Volumes
Zone	Floor area (m2)

The grid view header has 3 rows of header information:

- Attribute caption, e.g. "Outside air (ac/h)",
- Parent header caption for that attribute provides the context, e.g. "Mechanical Ventilation"
- Model data tab header, e.g. "HVAC".

For example:

Activity	HVAC		
Floor Areas and Volumes	Mechanical Ventilation		
Floor area (m2)	On	Outside air definitio...	Outside air (ac/h)

Showing / Hiding Data Grid Columns

For each of the 3 levels of header information described above it is possible to show and hide columns by right clicking on a column.

Group Level

At the “tab” level the top most grouped header will show / hide columns including parent headers and attribute headers beneath:

Activity	
Floor Areas and Volumes	✓ Activity ✓ HVAC ✓ Lighting
Floor area (m2)	

At the parent grouped header level the 2nd header will show / hide columns including attribute headers beneath:

Activity	
Floor Areas and Volumes	✓ Floor Areas and Volumes - HVAC Template ✓ Mechanical Ventilation - Operation - Economiser (Free Cooling) - Heat Recovery - Heating - Operation - Cooling - Operation - Humidity Control - DHW - Water Temperatures ✓ Operation ✓ Lighting Template ✓ General Lighting ✓ Lighting Control ✓ Lighting Area 1 ✓ Lighting Area 2 ✓ Task and Display Lighting
Floor area (m2)	
239.6	
239.6	
239.6	
239.6	
101.6	
44.8	

Individually

At the attribute header level the 3rd header will show / hide individual attribute columns:

Construction Template	Construction
Template	External walls
	Zone
	Block
	✓ Type
Project construction	✓ Area (m2)
Project construction	✓ Orientation (°)
Project construction	✓ Slope (°)
Project construction	✓ Template
Project construction	✓ External walls
Project construction	Below grade walls
Project construction	Flat roof
Project construction	Pitched roof (occupied)
	Pitched roof (unoccupied)
	✓ Internal partitions
Project construction	Semi-exposed walls
Project construction	Semi-exposed ceiling
Project construction	Semi-exposed floor
Project construction	✓ Ground floor
Project construction	✓ Basement ground floor
Project construction	✓ External floor
Project construction	✓ Internal floor

Grouping by Column

Grouping of attribute columns can be achieved by dragging the desired column into the grouping row:

Block			
Activity - Occupancy - Schedule			
Activity Template			
Zone	Template	Zone type	Zone multiplier
[-] 1F first floor East (1 item) ▶			
[-] 1F first floor East ▶ Office_OpenOff_Occ (1 item) ▶			
Open office	Generic Office Area	1-Standard	1
[-] 1F first floor West (1 item) ▶			
[-] 1F first floor West ▶ Office_OpenOff_Occ (1 item) ▶			
Open office	Generic Office Area	1-Standard	1
[-] 0F ground floor West (1 item) ▶			
[-] 0F ground floor West ▶ Office_OpenOff_Occ (1 item) ▶			
Open office	Generic Office Area	1-Standard	1
[-] 0F ground floor East (1 item) ▶			
[-] 0F ground floor East ▶ Office_OpenOff_Occ (1 item) ▶			
Open office	Generic Office Area	1-Standard	1
[-] Core circulation (1 item) ▶			
[-] Core circulation ▶ Office_Circulation_Occ (1 item) ▶			
Circulation	Office_Circulation	1-Standard	1
[-] Atrium Reception (1 item) ▶			
[-] Atrium Reception ▶ Office_Reception_Occ (1 item) ▶			
Reception	Office_Reception	1-Standard	1

Filtering

Rows in the data grid can be filtered. By hovering the mouse pointer at the right-hand side of the attribute column caption a drop down arrow will appear and clicking this will display the filter control:

Zone	Template	Zone type	Zone multiplier
1F first floor East (1 item)			
1F first floor East	Office_OpenOff_Occ (1 item)		
Open office	Generic Office Area		
1F first floor West (1 item)			
1F first floor West	Office_OpenOff_Occ (1 item)		
Open office	Generic Office Area	1-Standard	1
0F ground floor West (1 item)			
0F ground floor West	Office_OpenOff_Occ (1 item)		
Open office	Generic Office Area	1-Standard	1

More than one filter can be set up on different columns and can be cleared individually, or all at once, by clicking the **Clear Filters** button.

Tip: By setting appropriate filters, and using the bulk edit facilities, you can make changes to only the objects that meet your selection criteria, effectively mimicking a search and replace operation.

Searching

At the top of the data grid is a search box. Any text entered here and matched in the data grid will be highlighted in a yellow/brown colour. Search matches can be navigated via the search up and down arrows. The "x" symbol closes the search box (cancelling the search), which can be re-opened by entering Ctrl+F on the keyboard.

Office_C							
Block							
Activity - Occupancy - Schedule							
Zone	Template	Zone type	Zone multiplier	Include zone in ther...	Include zone in Radi...	Occupancy density (...)	Schedule
Core circulation (1 item)							
Core circulation	Office_Circulation_Occ (1 item)						
Circulation	Office_Circulation	1-Standard	1	☒	☒	0.1100	Office_Circulation_Occ

Exporting data

The visible data grid view may be exported to a comma separated file (.CSV) for subsequent editing in another program, e.g. in MS Excel. Clicking the **Export** button will launch a dialog to specify the file name and choose its location.

Data in the exported CSV file can be edited in a spreadsheet tool, saved and [imported](#) back into the model.

Tip: To edit a CSV file that was exported by the Model data grid tool, simply double-click on it from Windows Explorer. Make sure to save the file in CSV form as well if you intend to import the data back into DesignBuilder.

Editing Data in a Grid Cell

In edit mode all data cells can be edited except for:

- Light yellow shaded:



- Grey non viewable box:



Grey non viewable boxed cells appear when:

- o The data is not relevant to view / edit e.g. activity, lighting and HVAC data for merged child zones.
- o The data is not appropriate to view / edit e.g. blocking out wall construction data for a floor or roof surface.
- o An attribute visibility rule defined in a layout fails the criteria for that rule. E.g. to provide a list of roof windows, a rule might check that the parent surface custom attribute Surface Type has a particular value to decide whether an opening is to be displayed.

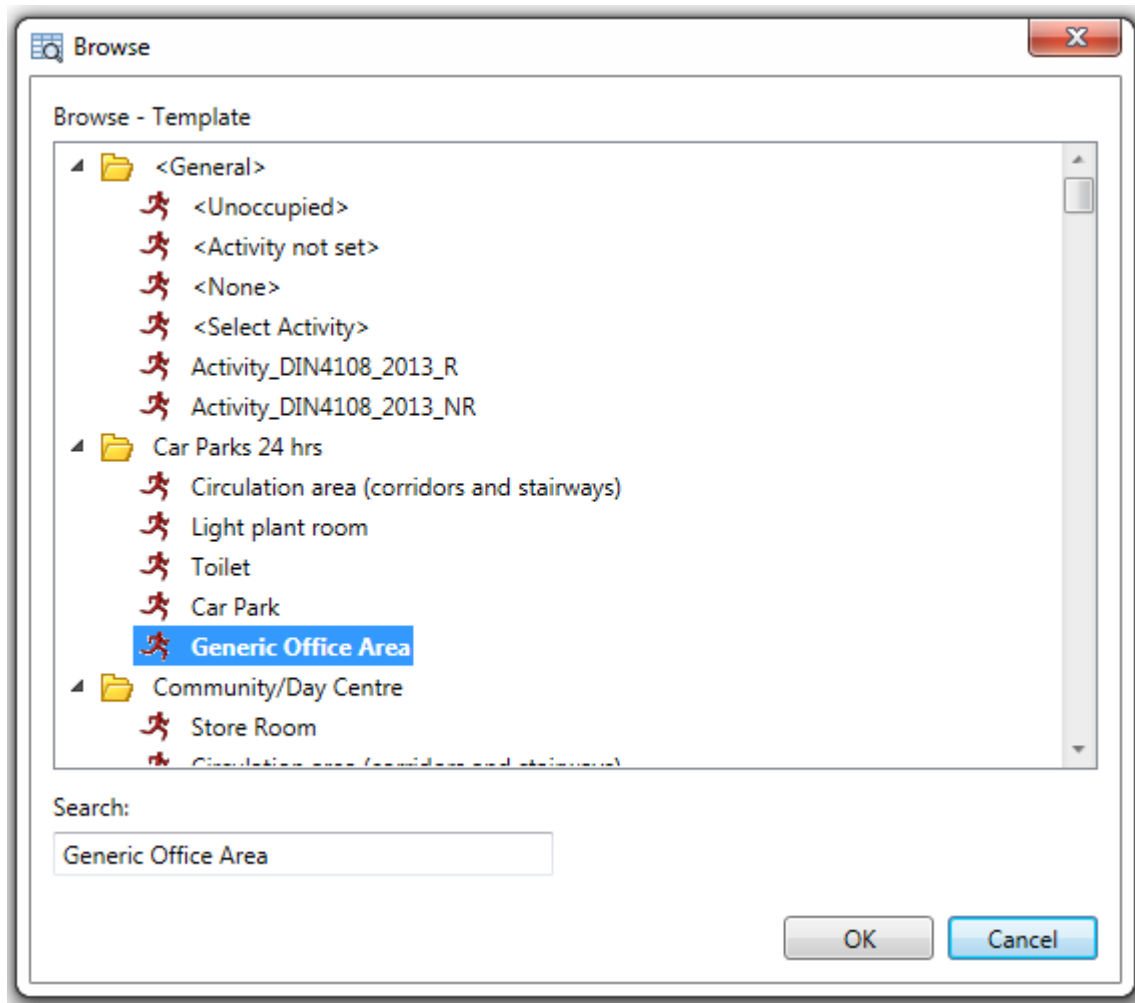
Clicking in a selected cell will set it into editing state.

The following type of edits are available depending on the type of model data:

- **Numeric / Text** - Entered content will be validated against the type of model data being edited to ensure it is appropriate i.e. whole number, decimal etc.
- **List** – a drop down list of valid values will be shown to make a choice from:

Activity			
Zone	Template	Zone type	Zone multi
1F first floor East (1 item) ▶			
Open office	Generic Office Area	1-Standard	1
1F first floor West (1 item) ▶			
Open office	Generic Office Area	1-Standard	1
0F ground floor West (1 item) ▶			
Open office	Generic Office Area	1-Standard 2-Semi-exterior unconditioned 3-Cavity 4-Plenum	

- **Browse** – a browse dialog will appear to allow selection from categorised values, with the initial select value being defaulted to the current model data:



Colour Codes

The Model Data Grid View uses colours to provide extra information about the data.

Text colour

On the main model data dialogs DesignBuilder uses text colours to indicate which values are inherited from levels above in the model hierarchy using blue. Values that are hard set (i.e. not inherited) are displayed in red. A similar colour coding system is used in the Model Data Grid View. In general values that are hard set are shown in **red** and values that are inherited are shown in **blue**. There are a couple of differences in the grid though:

- Values that have been edited in the current grid edit session are shown in **red** (even before they have been saved to the model). All other values are shown in **blue**.
- Detailed HVAC objects don't use inheritance and so are treated differently. Data for HVAC objects is always displayed in **blue**, unless it has been edited in the current session.

Cell background colour

The cell background colour is generally white with the following exceptions:

- Cells with data that can be viewed but not edited are displayed with a light yellow background.

Air Loop OG2	2.64722
--------------	---------

- Cells that are not available for display or edit are shown with a grey background (see also above). In the case below the **Floor area per person** data cannot be edited if the zone is not occupied:

Occupied?	Floor area per perso...
☒	10.00
☒	10.00
☐	—
☐	—

- Rows that are selected are shown with a blue/grey background colour:

▶	Component block 3	1-Standard	☒	1-Building	Component block m...
---	-------------------	------------	-------------------------------------	------------	----------------------

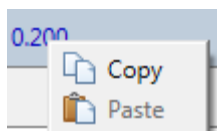
- Rows that are selected for **Bulk edit** are displayed with a lavender background colour:

Building Component...	Component block ty...	Shades and reflects	Level	Material
Component block 1	1-Standard	☒	1-Building	Component block m...
Component block 2	1-Standard	☒	1-Building	Component block m...
Component block 3	1-Standard	☒	1-Building	Component block m...

- Rows that contain a search term typed into the **Search** bar at the top of the grid are shown in a dark yellow colour.

Copy & Paste

A selected data cell's content can be copied to the clipboard via <Ctrl>+C or a right click context menu:



Where the type of model data is appropriate to paste into a data cell the <Ctrl>+V or the Paste menu item will be available to paste what is on the clipboard.

Bulk Edit

Multiple rows on the data grid can be selected by holding down the <Ctrl> key and left mouse clicking additional rows for selection or by holding down the <Ctrl> key + <Shift> key and clicking rows below for contiguous selection.

When 2 or more rows are selected, the **Bulk Edit** check box towards the top left of the data grid dialog can be checked and bulk edit mode will be enabled for the selected rows with the currently editable rows becoming highlighted with a lavender (blue/purple) background colour:

☒ Edit
 ☒ Bulk Edit

Clear Filters

Export...

Search (Ctrl+F)

^
 v
 x

Block

</					

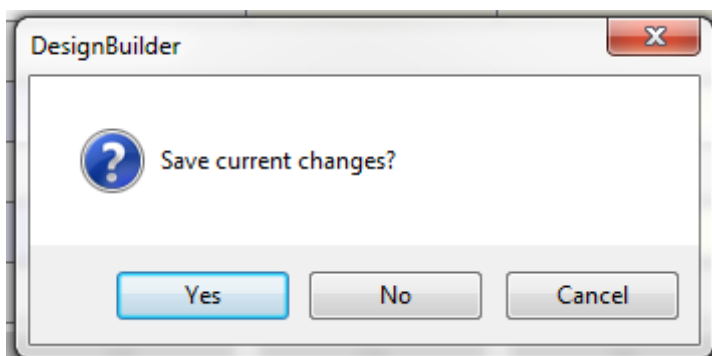
Editing a data cell for one of the selected rows will change the data cell for the corresponding data cells in the other bulk edit selected rows:

☒ Edit ☒ Bulk Edit Clear Filters Export...		
Search (Ctrl+F) ^ v x		
Block		
		Activi
Zone	Template	Zone type
1F first floor East (1 item) ▶		
Open office	Generic Office Area	1-Standard
1F first floor West (1 item) ▶		
Open office	Circulation area (corridors and stairways)	1-Standard
0F ground floor West (1 item) ▶		
Open office	Generic Office Area	1-Standard
0F ground floor East (1 item) ▶		
Open office	Generic Office Area	1-Standard
Core circulation (1 item) ▶		
Circulation	Circulation area (corridors and stairways)	1-Standard
Atrium Reception (1 item) ▶		
Reception	Circulation area (corridors and stairways)	1-Standard

Unchecking the **Bulk Edit** checkbox will return the data grid to its previous edit mode state.

Applying changes

Current changes may be applied to the model by clicking the “Apply” button. If there are changes yet to be applied and either the dialog is closed or Edit mode is switched back to view, there will be a confirmation dialog to determine what should happen next:



If switching from Edit mode to View mode and the “No” option is chosen the latest changes will be reverted to what is stored in the current model.

Note: Applying changes is not the same as saving the file. Any changes applied to the model still require the model file to be saved.

Grid Limitations

You should take care when making changes in the grid to ensure that you enter valid values or make valid selections. In the main model data user interface, irrelevant settings are hidden, drop lists and browse lists are configured to display only allowed values and numeric range checks are applied depending on the specific model settings and configuration. However the grid does not provide the same level of responsiveness and so, while the drop lists and browse lists will generally provide allowed settings, in some cases extra settings that do not apply will be displayed in the grid. If in doubt about which settings need to be edited please refer to the allowed values in the help and to the main model data user interface which is fully responsive to settings.

Using MDGV for Detailed HVAC Data

You should be especially careful with making changes to Detailed HVAC components some of which have complex rules to show/hide data based on other settings which have not yet implemented for the MDGV. More specifically, while the main HVAC component dialogs display only data that is relevant to selected options, the Grid edit HVAC views show *all* data settings regardless of whether they are relevant or not. The MDGV Layouts will be tidied in future to only show relevant settings, but in the meantime care is needed when using the MDGV for HVAC data to ensure that you are editing the right settings.

In summary, when using MDGV for adjusting Detailed HVAC settings please be aware that not all the HVAC data displayed in MDGV may be relevant to your model. Before editing you should:

1. Make sure you know which settings you wish to modify.
2. Modify the data for all similar components using MDGV.
3. Check at least 2 components using the popup dialogs from the main screen, to ensure that the new settings took effect as intended.

Model Data Grid View - Importing Model Data

You can import model data that was previously exported as a CSV file and subsequently modified in another application such as MS Excel or a text editor.

Selecting a file for import

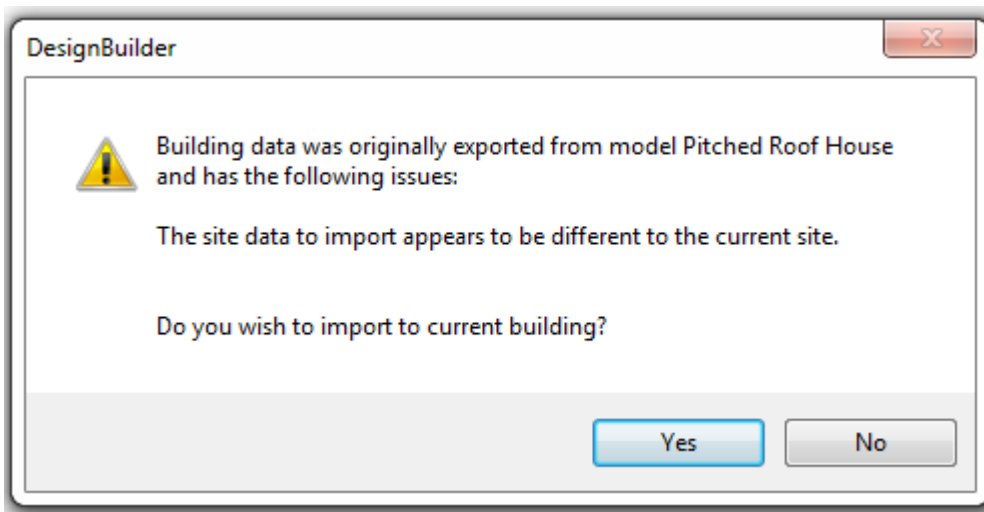
Clicking the **Import** button on the management dialog will open a standard file selection dialog filtered to files with “.CSV” extensions only. When the required CSV file is selected and **Open** clicked the import process will commence.

Checks performed prior to import

A series of checks are performed prior to importing a CSV file.

Establish whether Site / Building identifiers match

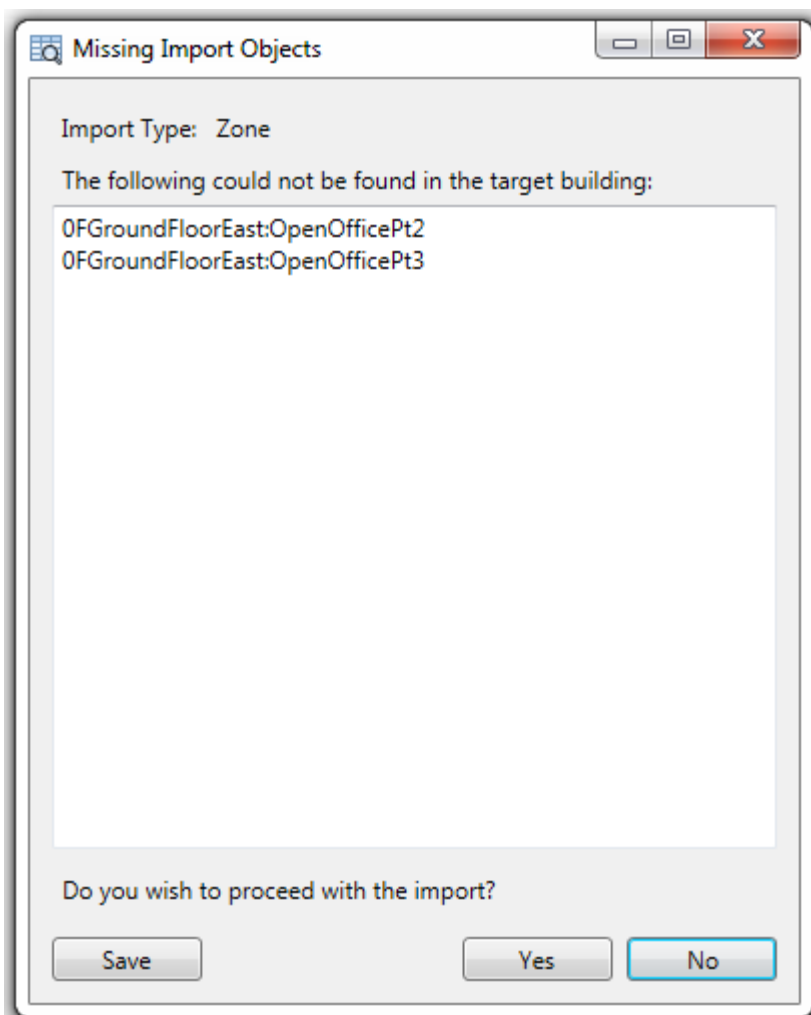
A check is made to check whether site / building identifiers match. If this is not the case the following warning message dialog is displayed:



Choosing **No** will exit the import process. Choosing **Yes** will progress the import process to the next check.

Establish whether Model objects match

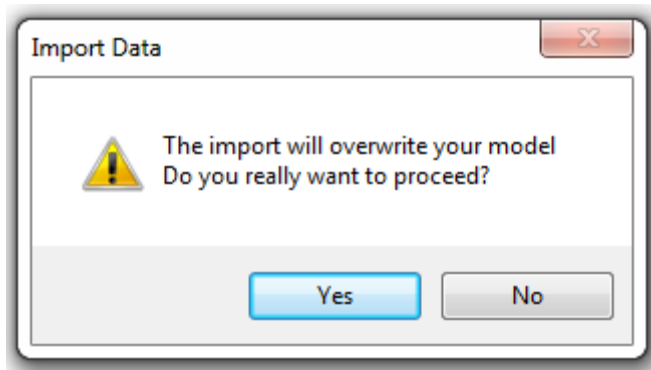
A check is made to establish whether model objects in the CSV file match with model objects in the current model. If model objects in the CSV are not found then the following warning message dialog is displayed:



Choosing **No** will exit the import process. Choosing **Yes** will progress the import process. The **Save** button saves the list of unresolved model objects to a text file for subsequent checking.

Final commit check prior to import

A final confirmation dialog is displayed offering a last chance to exit the import process:



Choosing **No** will exit the import process. Choosing **Yes** will import the model data.

Import completion

Upon completion of the import process the data grid dialog is displayed showing what was imported:

Model data grid view

☒ Edit

☐ Bulk Edit

Clear Filters

Export...

Search

^ v x

Zone

Block

Surface	Type	Area (m2)	Orientation (°)	Slope (°)	Construction Template	Constr		
					Template	Internal partitions	External walls	Below grade walls
Zone 2 security office (9 items)								
Zone 2 security office 00 ground rear N1 (9 items)								
Floor - 33.820 m2	Floor	33.820	95.0	180.0	Project construction...	—	—	—
Ceiling - 33.820 m2	Flat Roof	33.820	275.0	0.0	Project construction...	—	—	—
Partition - 6.781 m2...	Wall	6.781	264.4	90.0	Project construction...	0	2012 Wall	2012 Wall
Wall - 9.912 m2 - 26...	Wall	9.912	264.1	90.0	Project construction...	0	2012 Wall	2012 Wall
Partition - 32.573 m2...	Wall	32.573	170.1	90.0	Project construction...	Solid Partition 1990	2012 Wall	2012 Wall
Partition - 16.681 m2...	Wall	16.681	83.6	90.0	Project construction...	Solid Partition 2010	2012 Wall	2012 Wall
Partition - 0.267 m2...	Wall	0.267	350.1	90.0	Project construction...	Solid Partition 2010	2012 Wall	2012 Wall
Partition - 30.769 m2...	Wall	30.769	350.1	90.0	Project construction...	Solid Partition 2010	2012 Wall	2012 Wall
Partition - 1.363 m2...	Wall	1.363	350.1	90.0	Project construction...	Solid Partition 1990	2012 Wall	2012 Wall

Import Limitations

- Importing Detailed HVAC component data is not supported.

Model Data Grid View - Layout Management

Introduction to Layouts

Layouts define which model attributes will be shown for a specified level of model data. Layouts can apply to one or more analysis types.

It is possible to create your own custom layouts to allow you to set up display or edit sessions with exactly the model data that you need for your projects.

A simple example of a layout definition could be one where:

- The Level of model object type is Opening.

- The allowed [Analysis types](#) are EnergyPlus, SBEM, DSM.
- The model attributes to be displayed are Window height, Window width.

In the above example, the layout, when selected, would allow the Window height and width to be viewed for all openings in the model for any of the 3 main analysis types.

System layouts

System layouts are pre-defined layouts supplied with the software to cover aspects of building models considered to be of general purpose use for viewing / editing in a data grid.

User layouts

User layouts are those created by users to allow customised views of the model.

Management of Layouts

Management of layouts is performed on the Layout management dialog using the buttons under the **Manage Layouts** header.

The table below shows the actions that may or may not be performed on layout definitions:

Action / Layout type	System	User
New	No	Yes
Copy	Yes	Yes
Edit	No	Yes
Delete	No	Yes

Creating a Layout

Clicking **New** will launch the [Layout Editor](#) to allow a new User Layout definition to be created.

Editing a Layout

Clicking **Edit** will launch the [Layout Editor](#) to allow a User Layout definition to be modified. System layouts cannot be edited.

Copying a Layout

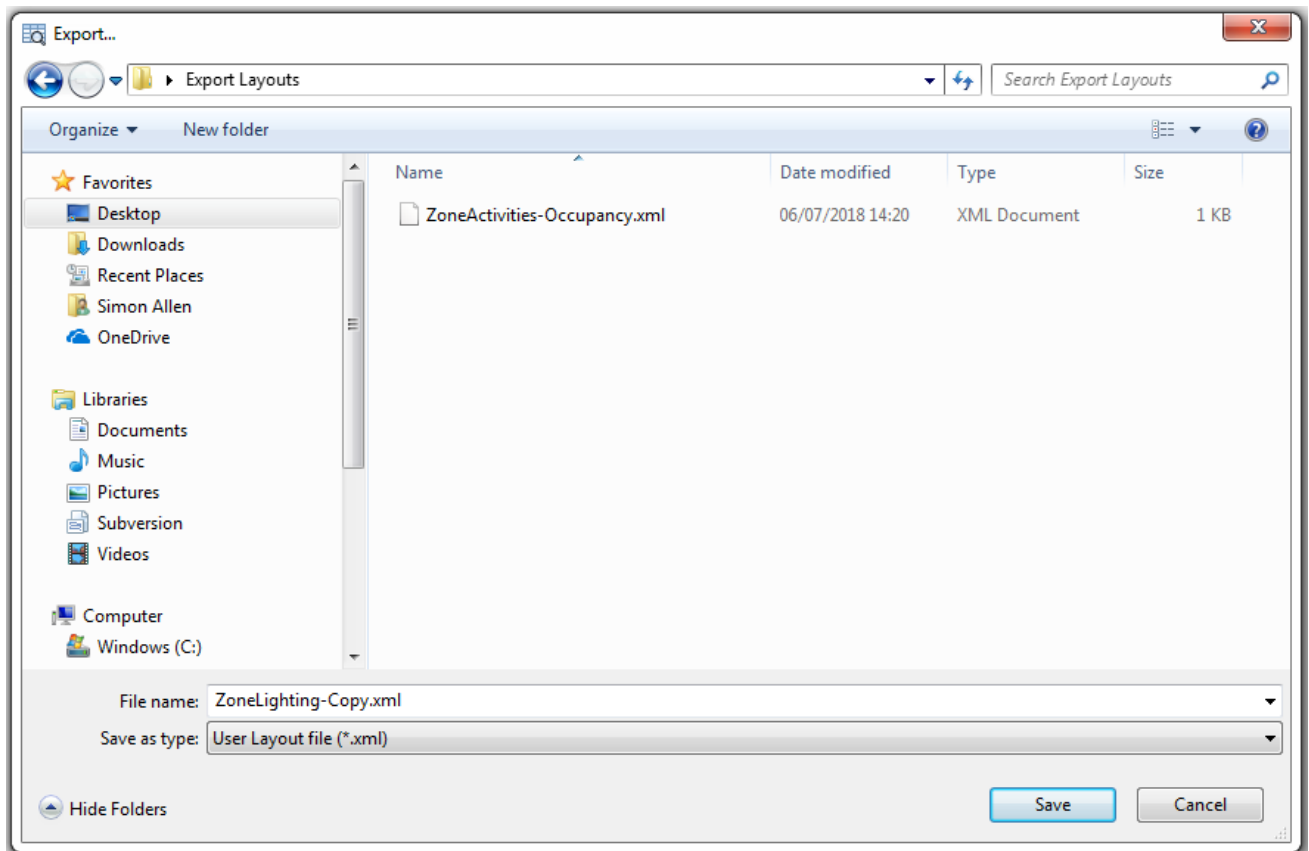
Clicking **Copy** will copy a selected System or User Layout . The new Layout is displayed under the appropriate model object type in the Layouts list.

Deleting a Layout

Clicking **Delete** will prompt for confirmation to delete the selected User Layout. System layouts cannot be deleted.

Exporting a Layout

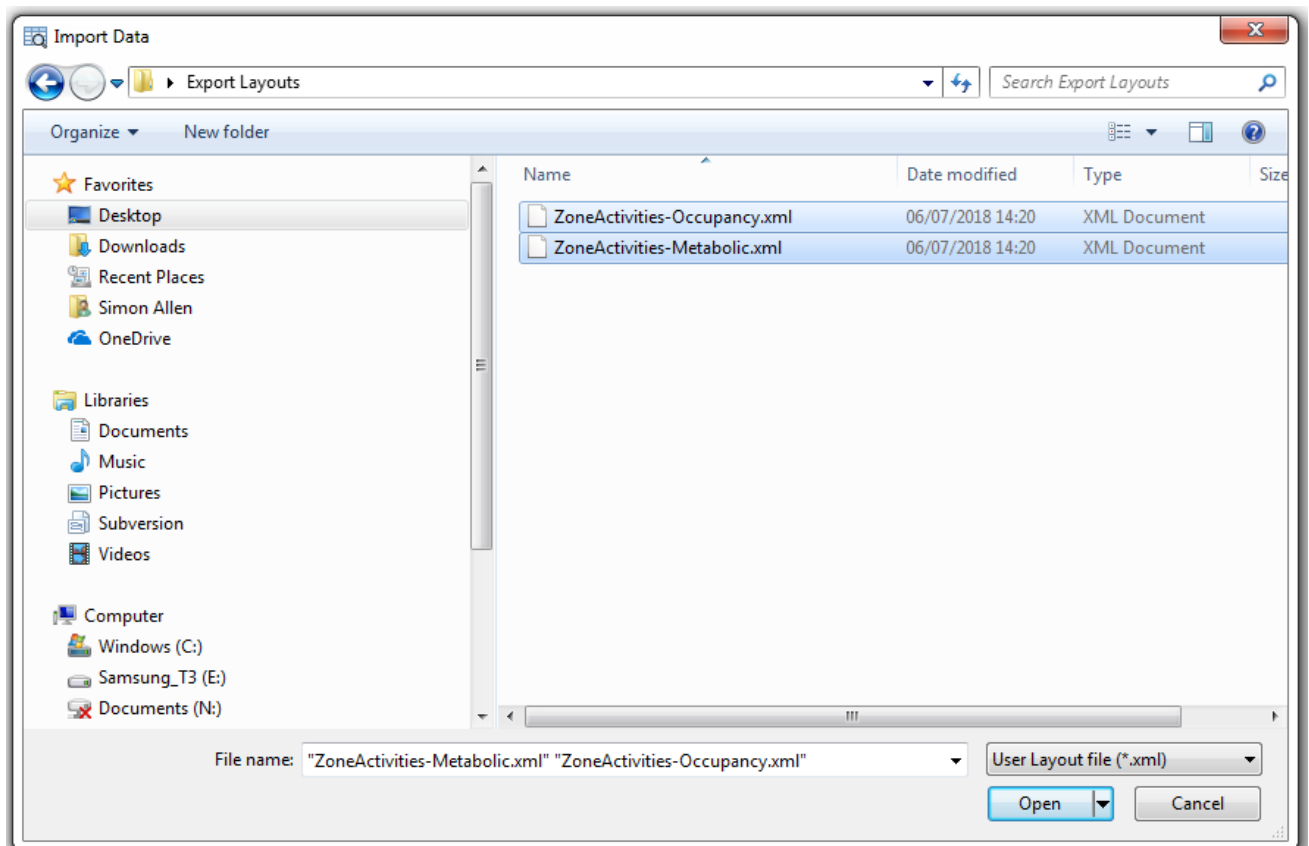
When a user layout is selected, clicking **Export Layout** will open a standard file save dialog allowing for a choice of location and file name to be specified for the layout:



This allows you to export layouts for subsequent import on another machine.

Importing Layouts

Clicking **Import Layout** will open a standard file open dialog allowing for the selection of one or more user layout files to be imported:



Where there is a clash in layout name with existing layouts a new name is given. For example, if a layout named “Internal Doors” exists and a layout of the same name is imported it will be given the name “Internal Doors (1)”.

Model Data Grid View - Layout Editing

The Layout Editor dialog allows user-defined custom Layouts to be set up for use alongside the pre-defined layouts provided with the software. The Layout editor can be launched from the management dialog via the Manage Layouts > **New** and **Edit** buttons. On first opening the Layout Editor dialog will look something like this:

The screenshot shows the 'Layout Editor' dialog box. It features a 'Name' field at the top left. To its right are 'Analysis Types' (with checkboxes for EnergyPlus, SBEM, and DSM) and a 'Level' dropdown menu (set to Building). Below these is the 'Attributes' section, which contains a tree view of various attributes. To the right of the tree is the 'Attribute Visibility Rules' section, which includes input fields for 'Object', 'Attribute', 'Comparison', and 'Value(s)'. At the bottom of the dialog are 'Help', 'OK', and 'Cancel' buttons.

There are 4 areas of data to be entered on the dialog:

- **Analysis types** which allows you to specify which Analysis types the layout applies to.
- **Level** which defines the data that will be displayed and editable (typically zone, surface or opening).
- The list of **Attributes** that can be displayed when the layout is accessed for view/edit.
- For each attribute any special **Visibility rules**.

These 4 areas are covered in more detail below.

When creating a new Layout, the **Analysis type** is set by default to match that of the current model's [Analysis type](#) (typically EnergyPlus, SBEM or DSM) and the **Level** is set to **Building**. The list of available attributes is based on the Analysis type and Level selections and will change when different combinations of Analysis types and level are selected.

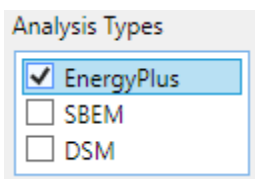
Layout Details

Name

Enter a descriptive identification for the layout. This will be displayed on the list of Layouts on the main Model Data Grid View Management dialog.

Analysis Types

Select one or more analysis types to define which are compatible for the layout being defined. For example if your layout should only be available for EnergyPlus models for then check that checkbox only. In this case the Layout will appear in the Layout list on the main Model data grid view [management dialog](#) for models where the Analysis type is set to EnergyPlus. However, if your model is located in the UK and the Analysis type for your model is set to SBEM or DSM then this layout won't appear in the Layout list.

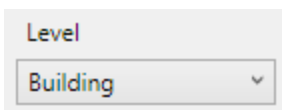


Level

Select the model object the layout will apply to. Options are:

- **Site,**
- **Building,**
- **Block,**
- **Zone,**
- **Surface,**
- **Opening.**

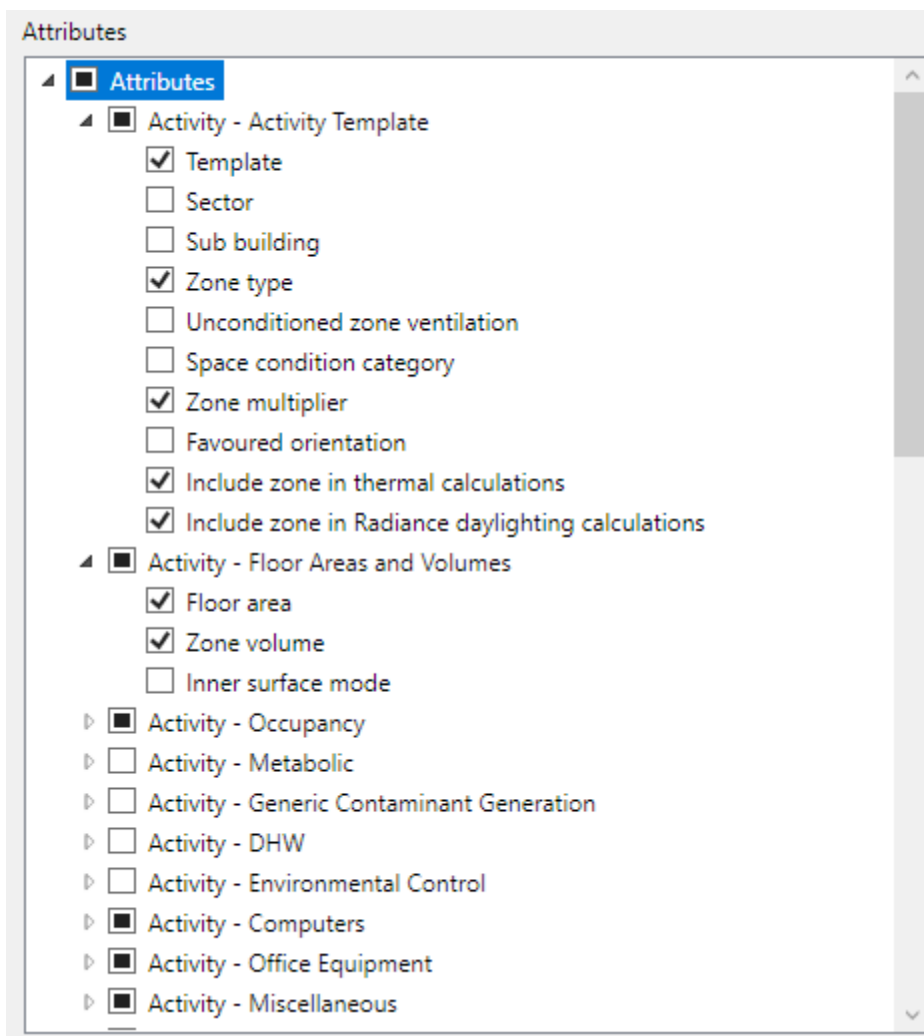
For example if you select the **Zone** level, the layout will display data for zones, i.e. each row in the model data grid view will be for a zone. In this case the attributes specified for each zone are defined as described below.



Attributes

The attributes to be displayed as columns in the model data grid view are defined by making appropriate selections in the list of attributes by checking the required attributes.

The attributes list is grouped into categories based on the model data tab in which it appears: Activity, Construction, Lighting, Outputs, CFD etc. The tree view nodes can be expanded to reach the required attributes which can have their respective checkboxes checked to select the attribute to be included with the layout. See below.



The attributes displayed in the list depends on the [Analysis types](#) and model object [Level](#) selected (see above).

If the Layout Editor was launched for an existing Layout via the **Edit** button from the Layout management group then the attributes to be displayed will have their respective check boxes checked.

Custom Attributes

To enhance the information presented on the data grid, some additional read-only attributes are included in the list depending on the data level. If custom attributes are available for the selected level they will appear at the bottom of the attributes list. Custom Attributes will always appear in the data grid, unless an attribute visibility rule has been defined to hide them in certain circumstances.

Committing / Cancelling Layout Edits

Clicking “OK” will create/update a new/existing Layout.

Clicking “Cancel” will return to Layout Management without saving changes.

Attribute Visibility Rules

When you select an attribute in the list on the left of the dialog, the settings in the [Attribute Visibility Rules](#) group on the right of the dialog are enabled and you can define a rule to customise the visibility of that attribute in the data grid. Visibility rules are set up to ensure that attributes are only displayed in the model data grid view where appropriate. Visibility rules work through a simple comparison using an attribute from a level higher up the hierarchy than the objects being displayed. There are 2 types of rules:

- **Column visibility** rules which check the value of a level attribute far enough up the hierarchy that the evaluation is the same for all objects (rows) to be displayed in the grid. This allows a whole column to be shown or hidden depending on the evaluation of the rule. Column visibility rules are based on building or site attributes. An example of a column visibility rule is where the visibility of the **Natural ventilation outside air flow rate** is to controlled based on the [Natural ventilation model option](#). Because this model option is a building level setting it applies equally to all zones (rows) in the grid and so the column can be hidden entirely if the Calculated natural ventilation model option is set.
- **Cell visibility** rules are checked separately for each row and control the visibility of individual cells. Individual cells are "hidden" by displaying a blank / grey data grid cell. An example of a cell visibility rule is one where the Glazing type is only shown for openings where the parent surface is of External wall surface type. Hidden cells are displayed as:



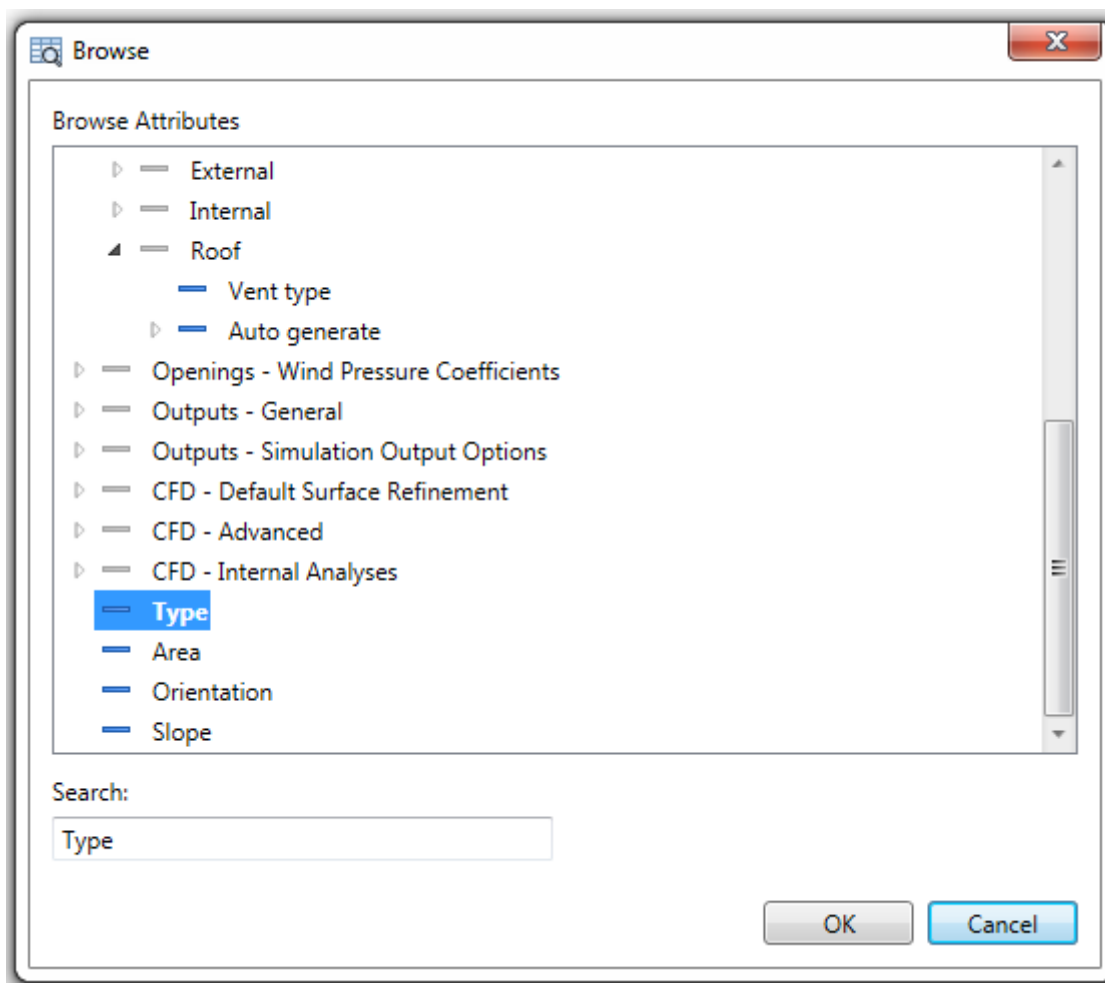
For both types of rule you may need to refer to some attribute option lists to help you define the visibility rules. Some commonly used attribute settings are listed below under [Attributes Commonly used in Visibility Rules](#).

Object

The **Object** selection determines which model object level will be used as the basis for the rule. Note that only levels "above" the level specified in the Layout level are available. For example if the layout [Level](#) is set to **Zone** then the only **Object** options allowed are Block, Building and Site.

Attribute

The **Attribute** to be used in the visibility rule can be chosen by clicking the browse button "... " to display the Attribute Browse dialog:



Attributes in this tree view are based on the selected model Object selected and the Analysis types as described above.

Custom attributes (if applicable for the selected model Object) are found at the bottom of the tree view.

Comparison

The comparison operator for the rule. Currently available options are:

- = equal to
- != not equal to
- < less than
- > greater than
- <= less than or equal to
- >= greater than or equal to

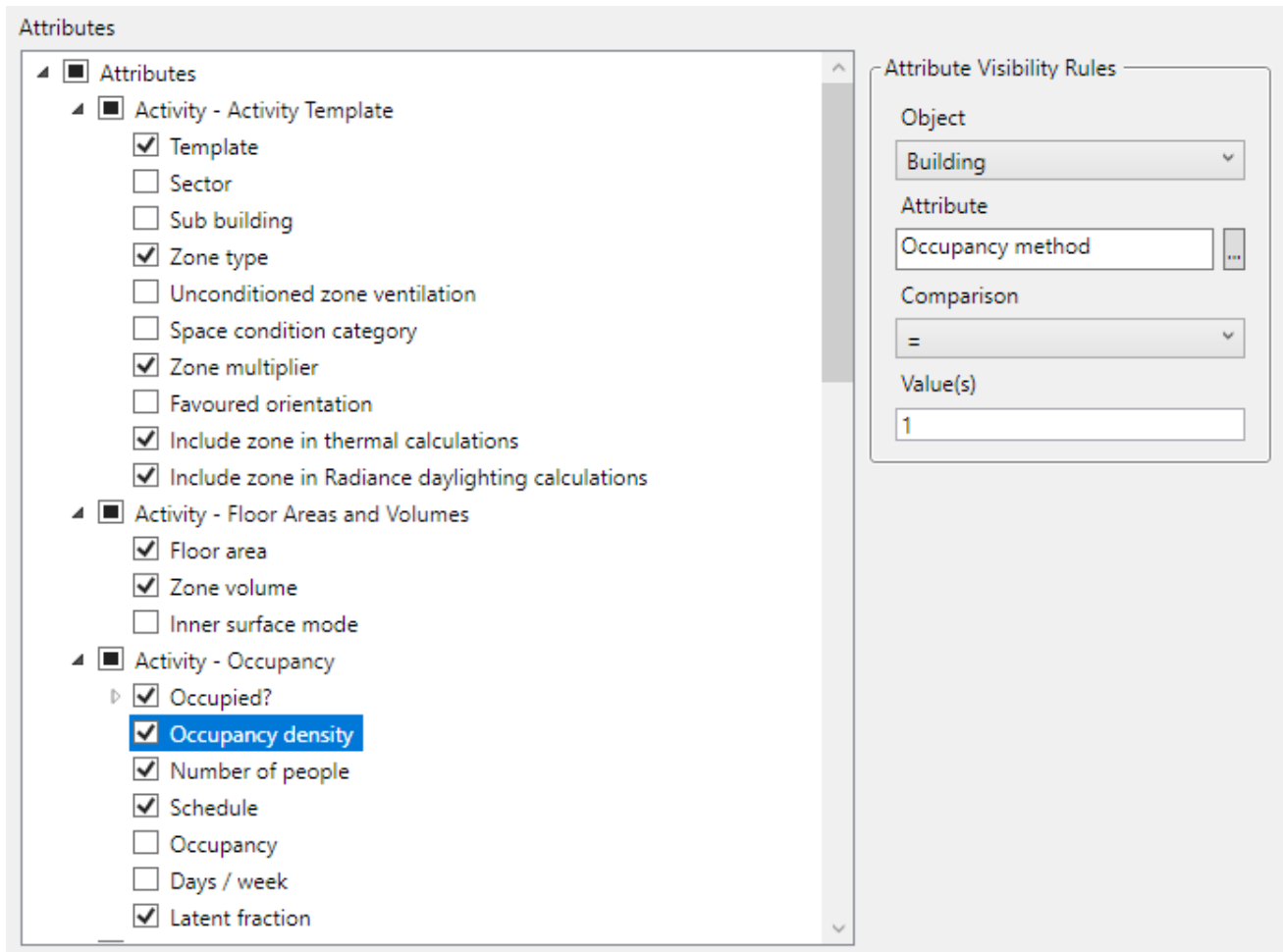
Value

The value to compare the chosen attribute value against. A range of values can be specified by separating with commas. E.g. 1,2,10. So, for example, for Model Object "Site" for attribute "Analysis Type" with a comparison on "=" and a value = "1,2,10" the rule will be passed if the current model's [Analysis type](#) is **1-EnergyPlus**, **2-SBem** or **10-DSM**.

Types of rule

Column visibility rules

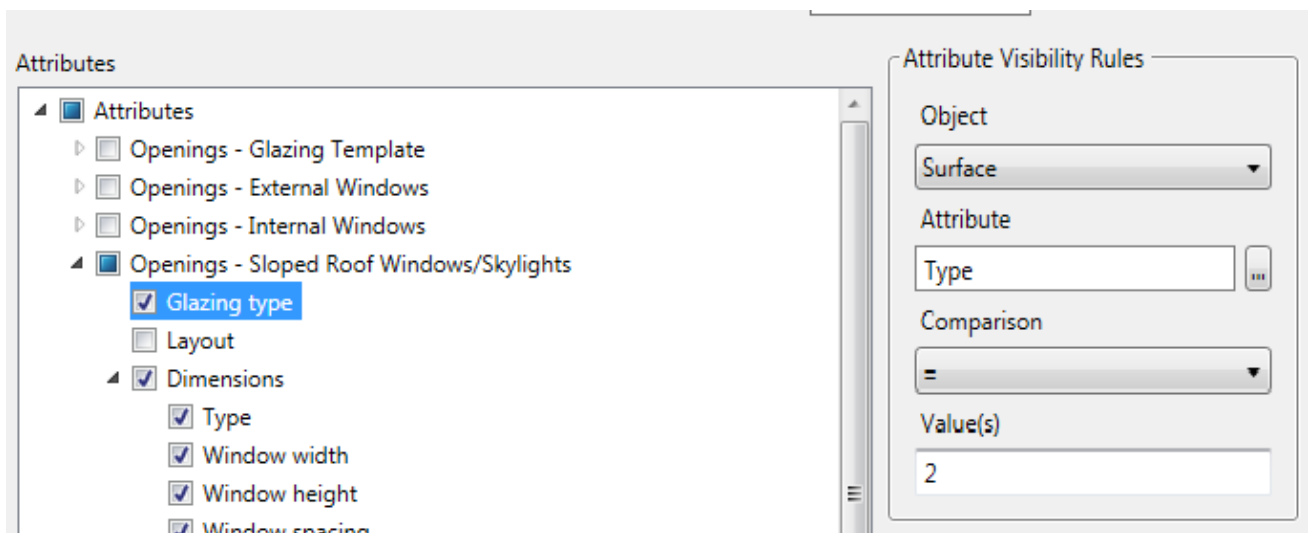
A rule applied using a high level attribute will impact the visibility of the attribute for all objects in the grid, i.e. the column won't appear in the data grid. The example below shows the how the rule for the [Occupancy density](#) attribute is set up, based on the value of the [Occupancy method](#).



Note: Column visibility rules use Site or Building level attributes such as [Analysis type](#) (a site level attribute) or building model options. An exception would be for a layout set up to list building level data for multiple buildings on the site in which case building attributes would not be an option and only site level attributes could be used.

Cell visibility rules

Rules applied at a lower level allow the visibility attributes to be controlled on a per object basis. For example the screenshot below shows a case where data grid cells are shown for the Roof Glazing type attribute for openings where the opening's parent [Surface type](#) has a value of 2, i.e. it is a pitched roof:



This rule results in a grid view similar to that shown below where the rule applied only allows the Roof glazing type to be shown for the 2 pitched roof windows:

Type	Adjacency	Slope (°)	Glazing type
Window	External	90.0	—
Window	External	90.0	—
Window	External	90.0	—
Window	External	90.0	—
Window	External	31.1	Project roof glazing
Window	External	31.1	Project roof glazing

Attributes Commonly Used In Visibility Rules

Analysis type

The [Analysis type](#) is a site level attribute set on the [Display tab](#) of the Model option options dialog defining the purpose of the analysis. It controls which user interface items, data, calculations and outputs are available.

Analysis type can be used for controlling the availability of entire layouts through use of the [Analysis types](#) checkboxes as described above or individual attributes through the [Attribute visibility rules](#).

The most common Analysis types are:

- **1-EnergyPlus** - the analysis type for general purpose simulation work.
- **2-SBEM** - for SBEM EPC and Building Regulations compliance calculations in the UK and Ireland at NOS levels 3 and 4. This option is only available when the site is located in the UK or Ireland.
- **7-RT2012** - for Building Regulations compliance calculations in France. This option is only available when the site is located in France.
- **10-DSM** - for DSM EPC and Building Regulations compliance calculations in the UK and Ireland at NOS level 5. This option is only available when the site is located in the UK or Ireland.

Surface type

It can be useful to show/hide opening level attributes based on the parent surface type. For example displaying **Glazing type** for surfaces of type 3 (walls). See above under [Cell visibility rules](#). The full list of surface types used internally by DesignBuilder is as follows:

- **0-Floor**
- **1-Flat roof**
- **2-Pitched roof**
- **3-Wall**
- **4-Hole**
- **8-Sloped floor**

Model Data Grid View Examples

General Steps

To use the MDGV tool follow these steps:

1. Click on the **Model data grid view** toolbar icon to open the main Management dialog.
2. If a Layout already exists that provides the data you need, select it from the list.
3. Otherwise, if you need to create a custom layout, expand the **Manage Layouts** header to view the options, then click on the **New** button to [create a layout](#).
4. Once you have selected a layout in the list click on either the **View** or the **Edit** button to load the requested model data to the grid.
5. This opens the main Model Data Grid View dialog, displaying the data requested in the Layout for viewing or editing. If you selected to view the data you can always enable editing later by checking the Edit checkbox at the top of the grid dialog.
6. To change data on multiple rows in the grid select the rows you would like to edit using the mouse together with the <Ctrl> or <Shift> keys and then check the **Bulk Edit** checkbox. This highlights the selected rows in blue and then any edits you make to one of the rows will be applied to all selected rows.
7. Once you have finished making changes click on the **Apply** button on the top right of the main grid to save your changes and close the dialog.

Detailed HVAC Example - Bulk Edit Fan Settings

This example shows how to edit the properties of all fans in a Detailed HVAC model.

1. Either load an existing Detailed HVAC model with multiple fans included, or create a new model with several zones, set the Detailed HVAC model option and load the "FCU 4-Pipe, Air Cooled Chiller" Detailed HVAC template for all zones.
2. Click on the **Model data grid view** toolbar icon to open the main Management dialog.
3. Select the **HVAC** tab at the top to view the HVAC Layouts.
4. Tick the **Show only HVAC components present in the model** checkbox to help you find the relevant HVAC Layouts.
5. Select the "Fan" Layout from the list.
6. Click on the **Edit** button to load the fan model data to the grid on the main Model Data Grid View dialog.
7. If you loaded the "FCU 4-Pipe, Air Cooled Chiller" Detailed HVAC template you will notice that all FCU fans have a Pressure rise setting of 100 Pa.
8. Select the fans you would like to edit using the mouse together with the <Ctrl> or <Shift> keys and then check the **Bulk Edit** checkbox. This causes the selected rows to turn blue/mauve as shown below.

Fan [Bulk Edit]

☒ Edit ☒ Bulk Edit

Search (Ctrl+F) ^ v x

Drag a column header here to group by that column.

General				
Name	Zone Group	Type	Fan total efficiency	Pressure rise (Pa)
BLOCK1:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	100.0
BLOCK2:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	100.0
BLOCK3:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	100.0
BLOCK4:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	100.0
BLOCK5:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	100.0
BLOCK6:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	100.0

9. Change the **Pressure rise** for one of the fans to 120 Pa and press <Enter>.

Note that all same change is automatically applied to all selected fans as shown below. Changed cells are shown with a turquoise blue background.

Fan [Bulk Edit]

☒ Edit ☒ Bulk Edit

Search (Ctrl+F) ^ v x

Drag a column header here to group by that column.

General				
Name	Zone Group	Type	Fan total efficiency	Pressure rise (Pa)
BLOCK1:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	120.0
BLOCK2:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	120.0
BLOCK3:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	120.0
BLOCK4:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	120.0
BLOCK5:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	120.0
BLOCK6:ZONE1 Fan...	Zone Group	3-On/Off	0.70000	120.0

10. Once you have finished making changes click on the **Apply** button on the top right of the main grid to save your changes and close the dialog.

Note: The above bulk edits could perhaps be made more easily using the **Target** tab on the Fan dialog for one of the fans, requesting the changes to be applied across the equivalent fans in the same HVAC zone group. However, for larger models with multiple zone groups or AHUs the MDGV method is generally be the most efficient way to make bulk edits.

Load Data From Template

Model Data Management



The **Load Data from Template** dialog allows bulk changes to be made by loading templates to selected locations in a model in a controlled way.

Access the Load Data from Template from:

1. The Toolbar
2. The Edit menu
3. [Mouse right-click in the Model data](#). In this case, by default, any template data loaded applies only to the selected model data attribute.

[Quick view](#) of the Load data from template tool in action

See also the [Glazing and openings](#) tutorial

You can select one or more data sources from the following template libraries:

- Construction template
- Glazing template
- Facade template
- Activity template
- Lighting template
- HVAC template

When at site level only one template can be loaded:

- Location template

When you select one of the above template libraries as a source of data you must also select the template from which to load:



In the above case, data will be loaded from the **Energy code - Mediumweight** Construction template to whichever objects are selected on the Target tab.

Target Objects

On the **Search** tab you can select how you want to set data and, where appropriate, which model objects to load to.

- **1-Selected objects** - choose this option if you want to load the same data to one or more building, block, zone or surface. Check the boxes next to the target objects to select them as targets.
- **2-Reload templates** - reloads template data for the current building and all its blocks, zones, surfaces and openings. Data is only loaded if the template selection is non-default. If the template inherits from the hierarchical level above, data is not loaded. See also [Reload Templates](#).
- **3-Search & replace** - allows you to search the model for objects having particular browse item selections and replace those settings with data from a template that you select.

Reload Templates

[Model Data Management](#)

The **Edit > Reload templates** menu command provides a quick way to reload template data for the current building and all its blocks, zones, surfaces and openings. Data is only reloaded if the template selection is non-default. If the template inherits from the hierarchical level above, data is not loaded. This ensures that any model data that was manually overridden reverts back to template settings. After running this command you should not see any model data shown in red bold.

The command opens the [Load data from template](#) dialog with the option **2-Reload templates** selected and all model data tabs selected for updating. You can modify these settings, for example if you only wish to reload data on certain model data tabs.

This option can be conveniently accessed from the [Edit](#) menu.

Tip: Reload templates can be run after overriding attribute data previously loaded from template to ensure that the latest template settings are used.

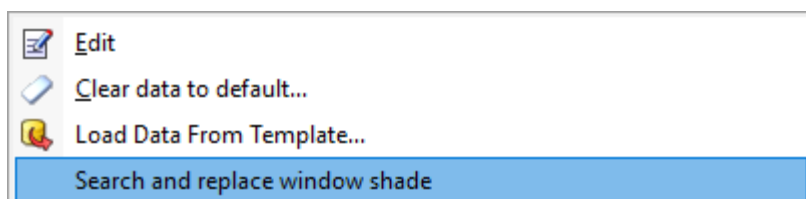
Search and Replace

The **Search and Replace** option allows you to search the model for objects having particular browse item selections and replace those settings with data from a template that you select. The tool works differently depending on whether the Load data from template dialog was opened from the toolbar or from the right-click context menu.

Tip: The [Model Data Grid View](#) includes [filter](#) and bulk edit tools that can be used to bulk edit models, and in most case, that is the preferred way to go about it.

Search and replace - accessed from the context menu

If you wish to search for particular browse option settings and replace them with alternative settings from a specific template, use the context menu by right-clicking on the browse item of interest and selecting the Search and replace ... option.



In the example above, the user right-clicked on the [Window shading type](#) browse item. Then, selecting the **Search and replace window shade** option opens the **Load Data from Template** dialog with the **3-Search &**

replace method selected. By default the search term is the currently selected object in the underlying browse item. In the above case, the currently selected Window shading type.

You can select the other objects that will be included in the search and replace operation using the checkboxes against the TreeView list under the **Search in** header below.

On the **Replace With** tab you can select the template(s) that will be used as a source for the replace data. Since the search and replace operation only applies to a single item when the dialog is accessed from the context menu, there is no point in selecting anything other than the Glazing template in this example as this is the source of the required Window shading type data.

Alternatively, you can select the **All** option on the **Replace With** tab and select any other templates that should be used as a source. In this case, any selected templates will be fully loaded to any model objects (blocks, zones, surfaces and windows) matching the search term. In our example, we will only select Glazing template.

Pressing the OK button causes the Window shading type setting from the Glazing template selected on the **Replace With** tab to be loaded to all model objects selected on the **Search** tab.

Search and replace - accessed from the toolbar/menu

In cases where you wish to load data from multiple templates for each selected object that meets your criteria, you can open the **Load Data from Template** dialog by clicking on the toolbar icon or from the Edit menu and set the Method on the Search tab to **3-Search and Replace**. When search and replace is accessed in this way, each template is handled individually. For example, if Activity template search is active (**Activity** is selected on the **Search For** tab) and, during the search processing a match is made with the selected Activity template on the **Search For** tab, then the Activity template selected on the **Replace With** tab will be used to replace it, regardless of any settings for Construction, Glazing, Facade templates etc.

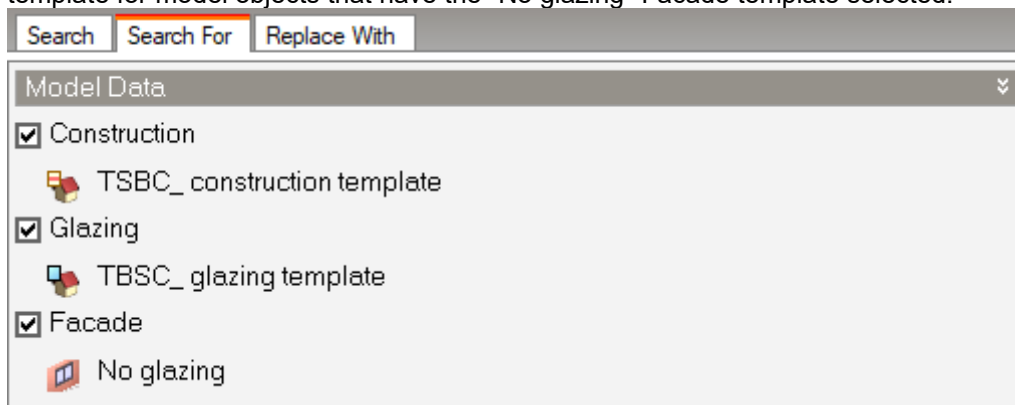
In other words, for each object selected on the **Search** tab, the tool checks whether the **Activity** checkbox on the **Search For** tab is checked. If it is and the selected Activity below that is selected in model data for this object, then the selected activity on the **Replace With** tab is loaded to that object. Then a similar check is made for Construction, Glazing, Facade, Lighting and HVAC templates. Then the operation moves on the next object selected on the **Search** tab until all objects have been processed.

On the **Search** tab:

1. Select the **3-Search and Replace** Method.
2. Check the checkboxes to define the model data objects (blocks, zones, surfaces and windows) you wish to search.

On the **Search For** tab:

1. Check the checkboxes to enable you to select the items you wish to search for.
2. For example, in the screenshot below, the user wishes to load a new construction template for model objects that have the "TSBC_construction_template" selected. They also wish to load a new Glazing template for model objects that have the "TBSC_glazing_template" selected. And load a new Facade template for model objects that have the "No glazing" Facade template selected.



On the **Replace With** tab:

1. Note that the **All** model data setting is selected at the top of the dialog and greyed out, indicating that the process will replace all of the selected templates, as opposed to the case where Search and replace is accessed from the context menu when it applies to only to the selected attribute.
2. Check the checkbox is associated with the various template types to be loaded, and,
3. Select the required templates to be loaded.

When the OK button is pressed, for each model object selected on the **Search** tab, DesignBuilder checks to see whether all of the search conditions are satisfied as defined by selections on the **Search For** tab, and if they are, it loads all of the settings requested on the **Replace With** tab.

Selection tools

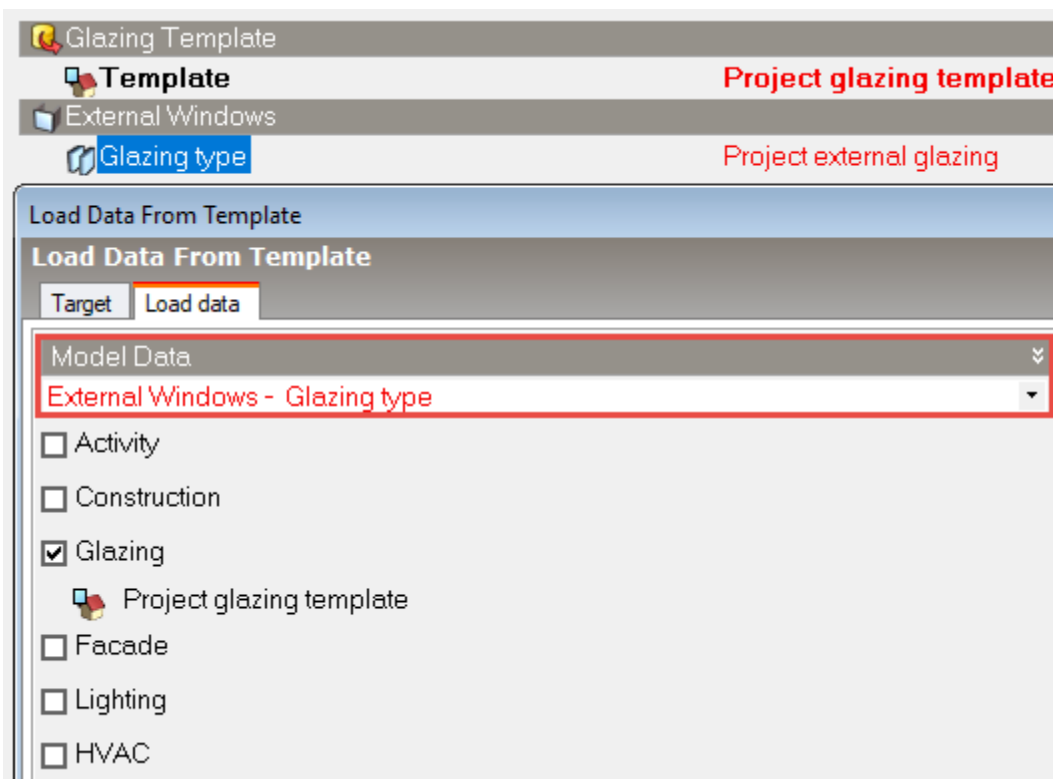
When on the **Search** tab, selection tools on the Info panel enable you to control the selection of building objects:

- **Select all** selects all objects in the navigator.
- **Select all down to zone level** selects all buildings, blocks and zones.
- **Select none** clears the selection.
- **Selection filter** opens the [Filter](#) selection dialog.

Tip: The [Filter](#) selection tool provides a convenient way to select multiple objects such as zones, surfaces and windows which have properties in common.

Load data from template - accessed from the context menu

When opening the dialog by right-clicking on a Model data item and selecting the **Load data from template** menu context menu option, by default any template data loaded applies only to the selected model data attribute, in the case below, the external glazing type. This behaviour can be changed by selecting the **All** Model data drop list option (screenshot below). In this case, when the **All** option is selected, data is loaded to all attributes on the tab in the same way as if the dialog had been opened from the Toolbar.



Making global lighting settings

If you are using the **2-Normalised power density** [Lighting gain units](#) model option

When you load lighting data from template you should bear in mind that DesignBuilder sets the lighting output from the lights based on:

- a. [Normalised power density](#) (W/m²-100 lux or W/ft²-fc) using data from the [Lighting template](#) and,
- b. [Illuminance levels](#) required in the zone as set on the [Activity tab](#).

If you are using the **3-Absolute zone power** [Lighting gain units](#) model option

If you have many types of activity in the building you can load lighting data for each zone taking into account the required illuminance level in the zone by selecting the appropriate zones in the Target tab.

Filter Selection

The Filter selection dialog provides tools to allow the selection of objects on the [Load data from template](#) dialog to be controlled. It is accessed by clicking on the Selection filter link in the Info panel of the Load data from template dialog.

Note: Selections made using this dialog are additive to any already made. To clear any previous selections first you should use the **Select none** option on the [Load data from templates](#) dialog before opening the Filter selection dialog.

See also the [Glazing and openings](#) tutorial

General

There are 3 general selection methods selected from the **Filter by** drop list:

- **1-Zones** - the filter tools will be used to select zones with particular text in their name or tag.
- **2-Surfaces** - the filter tools will be used to select particular surfaces.
- **3-Openings** - for selecting particular openings.

Adjacency selection

When either **2-Surfaces** or **3-Openings** are selected as the **Filter by** method you can define whether the selection should apply to internal, external or to both internal and external objects. Select from:

- **1-Internal** - select only internal surfaces/openings.
- **2-External** - select only external surfaces/openings.
- **3-Internal and External** - select both internal and external surfaces/openings.

Zones

When the **1-Zones** selection method is used you can filter either by zone name or tag.

Filter by zone name

If you wish to filter by the zone name then check this checkbox.

Selection type

The filter can be set up to either include or exclude text. Choose from:

- **1-Including text** where the filter is used to select any zones whose names include the text entered into the **Filter text** below.
- **2-Excluding text** where filter is used to select any zones whose names *do not* include the text entered into the **Filter text** below.

Filter text

Enter the filter text. For example if you wanted to select all zones in the model with the text "corridor" in their zone name then set the Selection type (above) to 1-Including text and enter "corridor" here and press OK. This can be useful as a quick way to load activity, lighting and HVAC templates to particular zone types.

Tip: This tool is much easier to use if zones are named consistently when setting up models.

Match whole zone name only

If the matching should be done on the whole name then check this checkbox. Otherwise the matching will be carried by searching for the filter text anywhere in the zone name.

Filter by tag

If you wish to filter by the zone [tag](#) then check this checkbox.

Selection type

The filter can be set up to either include or exclude text. Choose from:

- **1-Including text** where the filter is used to select any zones whose [tags](#) include the text entered into the **Filter text** below.
- **2-Excluding text** where filter is used to select any zones whose [tags](#) *do not* include the text entered into the **Filter text** below.

Filter text

Enter the filter text. For example if you wanted to select all zones in the model with the text "East" in their zone tag then set the Selection type (above) to **1-Including text** and enter "East" here and press OK. This can be useful as a quick way to load activity, lighting and HVAC templates to particular zone types.

Match whole tag only

If the matching should be done on the whole [tag](#) then check this checkbox. Otherwise the matching will be carried by searching for the filter text anywhere in the tag.

Surfaces

When the **2-Surfaces** selection method is used you can select surfaces based on their orientation and/or slope with an option to limit selection to zones that are already selected on the Load data from template dialog on the Target tab.

Apply orientation range

Check this checkbox if you would like to select surfaces having a particular range of orientations.

Lower orientation limit

Enter the lower end of the orientation range. For example to be sure of selecting all surfaces that are within 5° of south-facing you could enter 175 here and 185 for the upper orientation limit below.

Upper orientation limit

Enter the upper end of the orientation range.

Apply slope range

Check this checkbox if you would like to select surfaces having a particular range of slopes.

Lower slope limit

Enter the lower end of the slope range. For example to be sure of selecting all surfaces that are vertical you could enter 90 here and 90 for the upper slope limit below.

Upper slope limit

Enter the upper end of the slope range.

Filter by tag

If you wish to filter by the surface [tag](#) then check this checkbox.

Selection type

The filter can be set up to either include or exclude text. Choose from:

- **1-Including text** where the filter is used to select any surfaces whose [tags](#) include the text entered into the **Filter text** below.
- **2-Excluding text** where filter is used to select any surfaces whose [tags](#) *do not* include the text entered into the **Filter text** below.

Filter text

Enter the filter text. For example if you wanted to select all surfaces in the model with the text "Curved" in their zone tag then set the Selection type (above) to **1-Including text** and enter "Curved" here and press OK. This can be useful as a quick way to load construction, glazing and/or facade templates to particular surfaces.

Match whole tag only

If the matching should be done on the whole [tag](#) then check this checkbox. Otherwise the matching will be carried by searching for the filter text anywhere in the tag.

Limit selection to currently selected zones only

To ensure that only surfaces in zones that have already been selected on the Load data from templates Target tab are selected check this checkbox.

Openings

When the **3-Openings** selection method is used you can select openings based on their orientation, slope and/or tag with an option to limit selection to zones that are already selected on the Load data from template dialog on the Target tab. Settings work in the same way as for surfaces described above.

Save Data To Template

[Model Data Management](#)



The **Save data to template** tool can be useful to help create a template based on current model settings. For example, if you have set up a particular group of glazing and shading model data selections that you would like to apply to other parts of the model, you can use this tool to create a new [Glazing template](#) using all of the related glazing and shading settings for the current location on the current model data tab. You can then use the [Load data from template](#) tool to apply those settings as required to other parts of the model.

This tool is available from any of the following model data tabs:

- Activity - Activity template,
- Openings - Glazing template
- Lighting - Lighting template
- HVAC - HVAC template

The corresponding template type created is also listed above.

Tip: **Save data to template** can be thought of as the reverse of the [Load data from template](#) process in that data is flowing from the model to a template instead of from the template to the model.

Example

To create a template to quickly apply the settings below made on the Glazing model data tab follow these steps:

Glazing Template	
Template	Project glazing template
External Windows	
Glazing type	Project external glazing
Layout	Preferred height 1.5m, 30% glazed
Dimensions	
Type	3-Preferred height
Window to wall %	30.00
Window height (m)	1.50
Window spacing (m)	5.00
Sill height (m)	0.80
Reveal >>	
Frame and Dividers >>	
Shading	
☒ Window shading	
Type	SageGlass Green
Control type	1-Daylight only
☒ Glare override	
Glare solar beam incident threshold ...	200.0
Glare solar altitude lower threshold (°)	15.0
Glare solar altitude upper threshold (°)	70.0
Operation	
Operation schedule	Office_OpenOff_Occ
☒ Local shading	
Type	0.5m Overhang

1. When you have finished making the model data settings to be saved as a template, click on the **Save data to template** toolbar icon . You could be at building, block, zone, surface or window level for this and the settings of the current object will be saved.

- Open the Save Data to Template dialog, see screenshot below.

Create Template from Model Data - GlazingTemplates

Select the name, category and source for the new template.

Name: SageGlass green with glare control

Category: Project

Source: Project XYZ

Help Cancel OK

- Provide a name for the new template. Enter "SageGlass green with glare control"
- Select the category as "Project"
- Enter some text for the Source, e.g. "Project XYZ".
- When you have finished press OK to close the dialog and create the template.
- Once the template has been created, you are asked whether you would like to select the newly created template for your current object. If you reply yes then, in this example, the new "SageGlass green with glare control" Glazing template will be selected and the Openings model data tab will appear as shown below.

Glazing Template	
Template	SageGlass green with glare control
External Windows	
Glazing type	Project external glazing
Layout	Preferred height 1.5m, 30% glazed
Dimensions	
Type	3-Preferred height
Window to wall %	30.00
Window height (m)	1.50
Window spacing (m)	5.00
Sill height (m)	0.80
Reveal	
Frame and Dividers	
Shading	
☒ Window shading	
Type	SageGlass Green
Control type	1-Daylight only
☒ Glare override	
Glare solar beam incident threshold (W/m2)	200.0
Glare solar altitude lower threshold (°)	15.0
Glare solar altitude upper threshold (°)	70.0
Operation	
Operation schedule	Office_OpenOff_Occ
☒ Local shading	
Type	0.5m Overhang

Reload Templates

[Model Data Management](#)

The **Reload templates** option provides a quick way to reload template data for the current building and all its blocks, zones, surfaces and openings. Data is only reloaded if the template selection is non-default. If the template inherits from the hierarchical level above, data is not loaded. This ensures that any model data that was manually overridden reverts back to template settings. After running this command you should not see any model data shown in red bold.

The command opens the [Load data from template](#) dialog with the option **2-Reload templates** selected and all model data tabs selected for updating. You can modify these settings, for example if you only wish to reload data on certain model data tabs.

This option is accessed from the [Edit](#) menu.

Tip: This option can be run after changing template data to ensure that the latest template settings are loaded to the model.

Clear Data To Default

[Model Data Management](#)



The Clear Data to Default dialog allows you to:

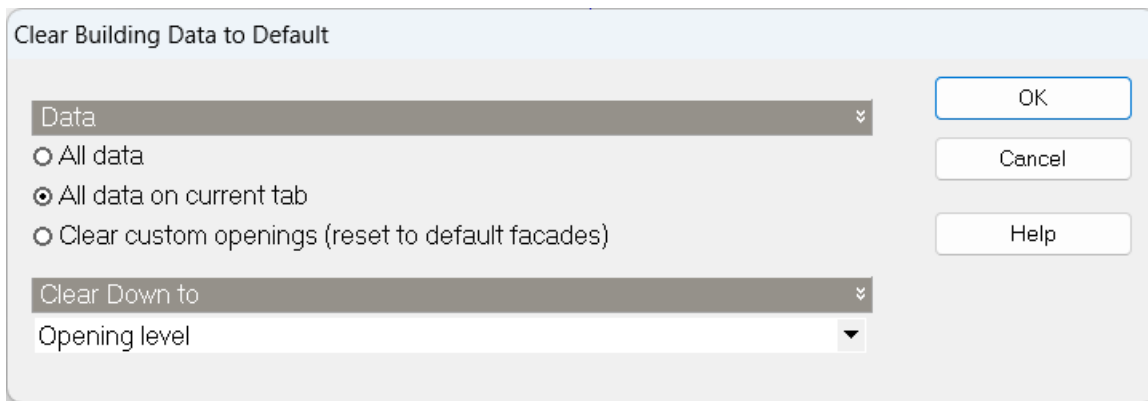
- Clear Model Data
- Clear custom openings and revert back to [default facades](#).

The options are:

- **All data** - choose this clear all attributes at the current level
- **All data on current tab** - choose this to clear only the attributes associated with the current model data tab. For example, clear all Constructions data.
- **Currently selected data** (name of currently selected attribute). This option only appears on the dialog when the dialog is opened by right-clicking on a specific attribute. In this case only the selected attributed is cleared.
- **Clear custom openings** - choose to clear any custom openings and revert back to using [default facades](#). This option is only available when the 3-D model is visible.

'Clearing to default' removes any hard-set 'user data' (shown in red) from the current object and instead default data is passed from the 'parent' object in the [hierarchy](#) (Model Data shown in blue).

You can control the extent of the 'clearance' by setting the **Clear Down to** data at the bottom. Leave this data as **Surface level** if you want all data in the currently selected model object (including child objects) to be returned to a default state.



Note: The Clear data to default command does not apply to [CFD boundary](#) object attributes such as window **Flow in** and **Flow out**, Supply diffuser **Flow rate** attribute and all other attributes associated with [CFD boundary objects](#).

See also the [Model data hierarchy and inheritance](#) tutorial

Clear Data to Default used at Building Level

Using **Clear data to default** at building level has a special function. It causes all data below building level in the model hierarchy to be cleared *but does it not clear the data at building level*. This makes sense when you consider that building level is effectively top level as buildings don't inherit any model data from Site level.

The way to understand use of this function at building level is that data at block level and below is cleared which ensures that building data is used throughout the model.

How to Access

The **Clear data to default** dialog can be accessed in 2 main ways:

- From the top toolbar or [Edit menu](#), in which case the clearing can be applied to all data on the current tab or to all data.
- From the context menu when right-clicking on a specific attribute in the model data, in which case only data for the selected attribute is cleared.

Merging Zones

Merging zones is the process whereby zones that are identified as being separate in the model can be merged together with other zones in simulations. This can help to simplify models and speed subsequent calculations.

The Merge Zones dialog can be accessed from the [Tools](#) menu when on the Edit screen. It provides convenient access to the 3 ways of merging zones in DesignBuilder:

1. **Merge zones of same activity** model option.
2. **Merge zones connected by holes** model option.
3. **Selective zone merging**

These zone merging mechanisms are described below. They can also be accessed from the [Model options](#) dialog, though this does not provide the controls for selective merging that are available from the Merge Zones dialog.

Tip: [Selective zone merging](#) is generally the preferred way to merge zones as it gives a greater degree of control over which zone is used as the parent - rather than leaving DesignBuilder to decide based on activity matches or hole connections. This is important because the parent zone's settings are applied to the entire merged group. If a zone with unrepresentative conditions is selected as the parent, the resulting model may no longer represent the building accurately.

More information on merging zones can be found in the [Drawing holes and merging zones](#) tutorial

Merge zones of same activity

This option is a quick and easy way to reduce the number of zones, by merging all zones having the same activity template selection. It should be used with care as it could lead to a very crude zone merging with many zones lumped together.

Merge zones connected by holes

Use this option to merge zones that are connected by [holes drawn at surface level](#) (*not* the holes used to represent the airflow path in [virtual partitions](#)). You can merge one or more zones within the same block or across multiple blocks.

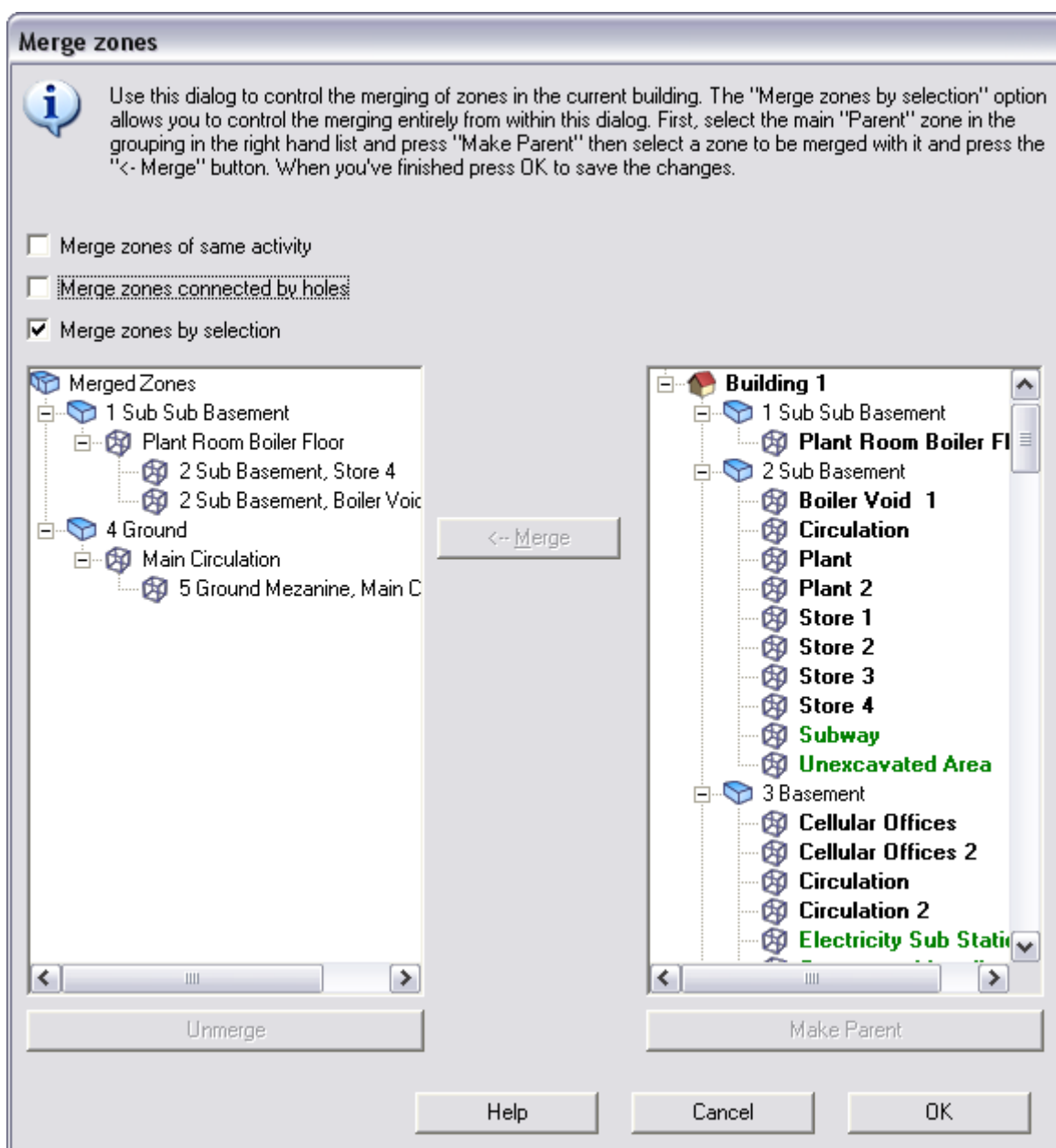
Where holes have been cut in a floor surface to remove it so upper and lower zones can be merged, any small remaining border in the cut floor surface should ideally be removed from the floor area calculation for the combined zone. To do this, you should navigate to the floor surface and tick the option to [Exclude this surface area from the total zone floor area](#) in the Adjacency section of the Construction tab.

Note: when merging 2 or more stacked zones using holes in the floor and the **Merge zones connected by holes option**, you should be aware that the *combined floor area used to calculate internal gains in the simulations will exclude the area of any holes in the floor*. In this case, if you require gains to be calculated using the total upper and lower floor area you should either ensure that the area of the hole is very small or else use the [selective zone merging](#) option instead and avoid using holes.

Selective Zone Merging

Selective merging controls can be used to merge zones irrespective of their activities or any connection by holes. This is done from Merge Zones dialog (accessed from the [Tools](#) menu) by checking the **Merge zones by selection** checkbox and assigning a parent zone for each zone group and then assigning one or more child zones to the parent to merge them with it. Note that once merged, zones can easily be un-merged again using the [Unmerge](#) button on this dialog.

Once the merge has been applied, the parent zone will control the activity, lighting, HVAC and all other zone-based selections for the entire zone group. Constructions and glazing can still be applied to the child zones.



Any changes made to zone merging settings are only applied when the dialog is closed using the OK button. Closing the dialog with the Cancel button discards any changes.

Creation of the Parent Zone

One zone must be selected to become the parent of the zone group. This zone must not already be either a parent zone or a member of an existing zone group. Selecting a suitable zone in the right hand pane will enable the **Make Parent** button located beneath the right hand pane of the dialog. Clicking this button will create a new entry for the parent zone in the left hand pane (The new zone in the left hand pane will be selected and highlighted).

Merging Zones

With a zone selected in the left hand pane it is now possible to select zones in the right hand pane to become members of the zone group. Zones to be merged cannot already be merged members of an existing group, however the parent of an existing group can be merged. Selecting a valid zone to be merged will enable the **Merge** button located between the two panes of the dialog. Clicking on this button carries out the merge and the newly merged zone will be shown on the tree in the left hand pane. If the parent of an existing zone group is selected and merged then the parent and all of its children become members of the merged zone group.

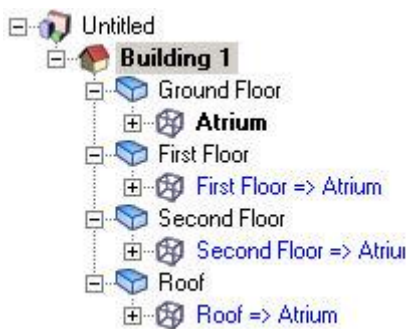
Unmerging Zones

A single child zone can be removed from a zone group by selecting it in the left hand pane, then using the **Unmerge** button located beneath the pane.

Selecting the parent of a zone group then using the **Unmerge** button will remove all the child zones from the group but will not remove the parent itself. If a parent zone having no children is selected, the text of the unmerge button changes to **Remove Parent**, and the parent zone can be removed from the tree in the left hand pane.

Merging Zones - Discussion

When zones are merged using one of the above methods, combined merged zones are treated as a single zone in thermal calculations. When zones are merged, one of the zones is marked as being the 'parent' zone, and the other zones are 'child' zones. The child zones are marked in blue in the Navigator together with the parent zone they have been merged into as shown below.



The above screenshot of the Navigation list shows the First floor zone, Second floor zone and Roof zones all merged into the Atrium zone.

The parent zone is the zone that is used for defining Activity, HVAC, Lighting (and all other zone related data) for itself and all its child zones. You cannot access this zone-related Model data for merged child zones. You must go to the parent zone to edit the Model data for the combined zone. The internal gains are calculated based on the data set on the Activity tab for the parent zone but using the combined total floor area of all of the merged zones.

Constructions, glazing and other surface level data can be defined separately in each of the parent and child zones in the normal way.

Tip: See also [Zone multiplier](#) for a manual way to reduce models having similar zones and [Boolean operations](#) for how to combine blocks geometrically.

Why merge zones?

You may want to merge zones for various reasons:

- Speed up simulations.
- A geometrically complex space in the actual building is to be modelled using more than one block in DesignBuilder (e.g. [dormer windows](#), atria).
- To reduce the number of zones and simplify HVAC system descriptions when exporting EnergyPlus IDF data for use outside DesignBuilder.
- If you have created a tall merged zone, you may want to use the Air distribution model data on the HVAC tab to create a dynamic vertical temperature gradient. This approach to modelling tall spaces is preferred over connecting a number of horizontal slices with air flow openings, because it does not rely on getting correct airflow conductances between the horizontal slices.

Surfaces between merged zones

When generating data for the calculations, surfaces between zones are defined twice, once from each side. Surfaces between 2 zones that have been merged are called **Surfaces within zone**. By default these are modelled in the same way as standard surfaces but with both sides adjacent to the same zone. If the [Surfaces within zone treated as adiabatic](#) option is set, the 'other side' in each of the 2 surfaces is modelled as adiabatic.

Disadvantages of merging

If done without consideration of the issues, merging zones can reduce cooling load peaks resulting in an underestimate of cooling equipment sizes required to condition the building (and associated energy consumption in a simulation). To understand this, consider a building in the Northern hemisphere with two zones, a North zone and a South zone connected to each other along the E-W axis. When solar gains are highest, the South zone may experience a cooling load and the North Zone not. Now imagine the same 2 spaces merged into a single zone. The combined single zone will now almost certainly have a lower cooling load than the load in the original South zone because the air and surfaces on the South side of the merged space is cooled by the shaded North side.

As simple guide, where the temperatures, internal/solar gains, ventilation, HVAC system and other operational parameters in the actual separate zones are about the same as each other then the merging process should not affect simulation accuracy unduly, however, when merging zones which have different conditions and/or load profiles, the merging will reduce accuracy.

In most cases, merging zones will speed up simulations but occasionally simulations can slow down, especially at the initialisation stage. This may be because any very large zones generated through merging can have many windows which can slow down solar and daylighting initialisation.

Merging rules

1. When merging zones using the [Merge zones of same activity](#) or [Merge zones connected by holes](#), zones connected by the merge option are lumped together into a single 'parent' zone. In a vertical stack of zones the parent will be the lowest zone in the stack.
2. The zone surfaces in merged child zones are generated in exactly the same way as they would have been without the merging but are associated with the parent zone rather than the original child zone.
3. In child merged zones, you can access Constructions and Openings surface-based Model data but not space related data such as Activity, Lighting, HVAC. This is accessed from the parent zone only.

Effect of merging zones on EnergyPlus simulations

EnergyPlus IDF input files are generated without entries for child merged zones (they are not included in the simulation), but their surfaces, windows, doors are included and are associated with the zone description of the parent zone. So in a merged zone model, simulation results are not available for the child zone separately but are available for the overall merged zone. Results for the merged zone can be obtained from the simulation, cooling design and heating design screens either by clicking on the parent zone in the navigator, or by clicking on one of the child zones - the same data for the merged zone is displayed in either case.

Note: For buildings with any zone merging set, heating and cooling design and simulation results are not shown at block level.

Potential problem when merging

One problem users can run into with merged models is related to the Calculated natural ventilation option. For windows placed in zones which are merged child zones where the child zone has the natural ventilation option on the HVAC tab switched off the Operation data on the Openings tab will be hidden. In order to make sure that the natural ventilation is activated for the zone you must temporarily switch off merging, switch on natural ventilation on the HVAC tab for the child zone then switch merging back on again. You should now find that the Operation data is shown.

Merging buildings

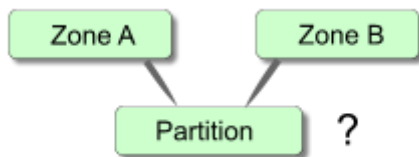
You can also 'merge' 2 existing buildings (including component blocks) using the [Transfer blocks](#) command from the Edit menu.

Effect of merging zones on Daylighting simulations

Daylighting simulations ignore the zone merge settings. If it is important for you to combine results for multiple zones for the purposes of daylighting credit calculations you should consider finding another way to combine zones, possibly through the block Union tool to combine blocks or by removing partitions.

Tip: For zones that are stacked one on top of the other with holes in top and bottom to represent slices through an atrium, you can uncheck the [Include zone in daylighting calculations](#) checkbox for the intermediate zones to avoid seeing results for them. Include the lowest zone though so its results are reported.

Zone Level Data For Internal Surfaces



Model data that applies to internal surfaces and openings is generally not available at zone level. This helps to avoid ambiguity in inheritance rules when different settings are made in adjacent zones.

To understand why this is the case, consider a partition between zones A and B. If "partition type 1" is selected as the [Internal partitions](#) model data in zone A, and "partition type 2" is selected in zone B, the inheritance rules indicate that the shared partition could either inherit from zone A ("partition type 1") or from zone B ("partition type 2").

DesignBuilder avoids such ambiguities by preventing Internal partitions from being set at zone level. It does this by hiding the setting when at zone level. Surface level Internal partitions model data settings are instead inherited from block level.

Note that settings for *inter-block* partition surfaces are inherited from the building level for similar reasons.

To summarise:

- Internal surface and opening model data settings are not available for editing at zone level. See below for a full list of the model data settings affected by these rules.
- For internal partitions adjacent to 2 zones in the *same block*, surface level settings are inherited from *block* level.
- For internal partition surfaces adjacent to 2 zones in *different blocks*, surface level settings are inherited from the *building* level.

These rules apply to the following model data settings related to internal surfaces:

Constructions tab

- [Internal partitions](#)
- [Semi-exposed walls](#)
- [Internal sub-surfaces](#)
- [Internal door](#)

Openings tab

- All data under [Internal windows](#) header
- All data under [Doors > Internal](#) header
- All data under [Vents > Internal](#) header

In general, if you need to apply different settings for internal partitions, glazing or any other data listed above in different parts of the model (i.e. you can't just make a building level setting to apply the same change across the whole building), then it may be best to make changes at surface level. In this case the change is automatically applied to the corresponding surface on the other side so there can be no ambiguity.

Tip: If you find that the above rules are slowing down your modelling process because you need to apply multiple different settings to internal surfaces or openings across your model, then you should consider using the [Model data grid tool](#) to speed up the process by making changes to multiple surfaces at once.

The screenshot displays a software interface with a table of surface data and a 'Browse' dialog box.

Table Headers:

- Block
- Zone
- Surface
- Type
- Area (m2)
- Orientation (°)
- Slope (°)
- Construction
- Semi-Exposed
- Walls
- Sub-Surfaces
- Internal Windows
- Internal

Table Data (01 Ground Floor North - 002 BBB Retail 3 (6 items)):

Surface	Type	Area (m2)	Orientation (°)	Slope (°)	Construction
Wall - 77,781 m2 - 1...	Wall	77,781	180,0	90,0	KT CLT Partition 180
Wall - 33,374 m2 - 2...	Wall	33,374	256,5	90,0	KT CLT Partition 180
Partition - 83,635 m2...	Wall	83,635	0,0	90,0	KT CLT Partition 180
Partition - 7,455 m2...	Wall	7,455	90,0	90,0	KT CLT Partition 180
Partition - 24,990 m2...	Wall	24,990	90,0	90,0	KT CLT Partition 180
Partition - 1,965 m2...	Wall	1,965	0,0	90,0	KT CLT Partition 180

Table Data (01 Ground Floor North - 002 BBB Retail 1 (12 items)):

Surface	Type	Area (m2)	Orientation (°)	Slope (°)	Construction
Wall - 89,977 m2 - 1...	Wall	89,977	180,0	90,0	KT CLT Partition 180
Wall - 84,258 m2 - 1...	Wall	84,258	17,4	90,0	KT CLT Partition 180
Wall - 79,678 m2 - 9...	Wall	79,678	90,0	90,0	KT CLT Partition 180
Wall - 17,509 m2 - 2...	Wall	17,509	270,0	90,0	KT CLT Partition 180
Partition - 7,690 m2...	Wall	7,690	287,4	90,0	KT CLT Partition 180
Partition - 7,275 m2...	Wall	7,275	0,0	90,0	KT CLT Partition 180
Partition - 6,589 m2...	Wall	6,589	270,0	90,0	KT CLT Partition 180
Partition - 24,990 m2...	Wall	24,990	270,0	90,0	KT CLT Partition 180
Partition - 24,395 m2...	Wall	24,395	270,0	90,0	KT CLT Partition 180

Browse Dialog:

- Search: KT CLT Partition 180
- OK
- Cancel

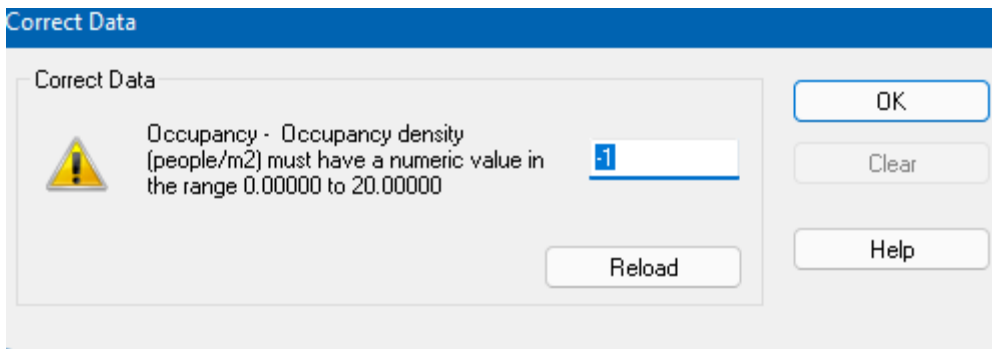
Load Whole Building Infiltration

The Load Infiltration dialog allows whole building infiltration data obtained from pressurisation tests also referred to as "blower door" tests to be entered and the appropriate infiltration rate is set in model data based on the [infiltration data entry options](#) selected. The whole building infiltration value must be entered in m3/s at 50 Pa pressure difference (even if you have selected [IP units](#)). This flow rate is applied over the whole building and depending on whether the option to "Apportion infiltration to zones based on surface area" is selected, the infiltration flow is either simply set at building level (if the option is unchecked) or is applied differently to each zone depending on the amount of external exposed surface area (if it is checked).

The infiltration flow rates are calculated from the entered whole building value using equations described in EN 12831 including consideration of the exposed area of surfaces in each zone, building height, [wind exposure index](#) and the [infiltration units](#) selected in model options.

Correct Data

When typing a numeric value into a model data text box, if the value is out of range, i.e. it is lower than the minimum value or greater than the maximum value allowed, then the **Correct Data** dialog is displayed. The dialog provides information on the limits as well as ways to fix the error by clearing the value to default or by reloading from the associated template.



Reload

If the data can be loaded from a template then pressing the **Reload** button will reload the data from template to the dialog. If you agree with the change then press the OK button to complete the correction.

Clear

Press the **Clear** button to clear the value to default and close the dialog. The **Clear** button is only enabled when there is a default value to use so for example it is not available when working at building level which is at the top of the model data [hierarchy](#).

Help

Loads this page.

OK

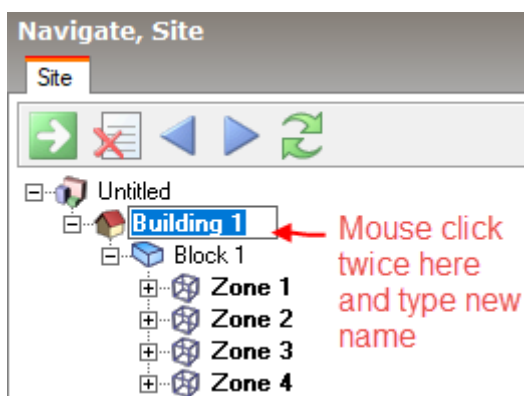
Press the OK button to apply the value in the dialog text box to model data.

Naming Model Objects

Taking an organised approach to naming your model objects can be well worth the effort when it comes to editing and checking models and analysing results. To help with this DesignBuilder allows you to provide a name for most objects in a model. This section provides advice on how to name objects in your model.

Building name

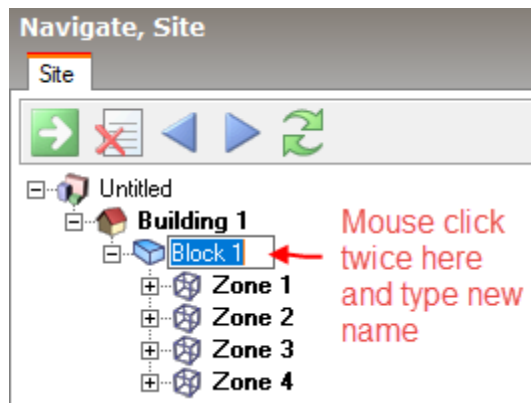
The building name can be edited in the Navigator by clicking on the existing building name to select it and then clicking again to enter edit mode, in the same way that you would rename a file in Windows File Explorer. See screenshot below.



The building name is used as a description alongside the filename on [Simulation Manager](#) lists.

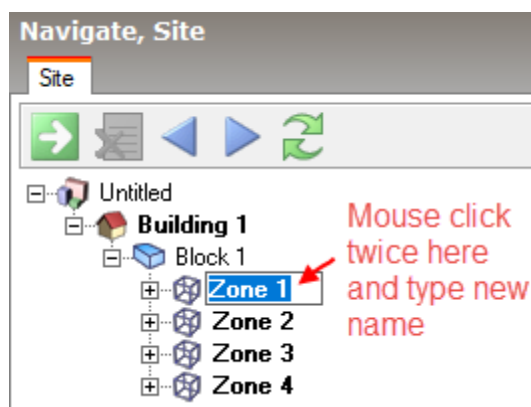
Block name

The block name can be edited in the navigator by clicking on the existing block name to select it and then clicking again to enter edit mode, in the same way that you would rename a file in Windows File Explorer. The block name is used as part of IDF zone names and described below under [EnergyPlus IDF names](#).



Zone name

The zone name can be edited in the navigator by clicking on the existing zone name to select it and then clicking again to enter edit mode, in the same way that you would rename a file in Windows File Explorer. The zone name is used as part of IDF zone names and described below under [EnergyPlus IDF names](#).

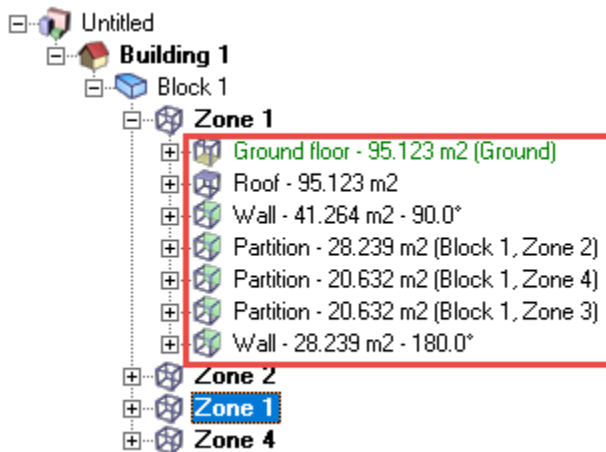


Note: The zone name must be unique within the Block to allow IDF names to be generated without extra characters being added by DesignBuilder to ensure uniqueness.

Tip: If you need to rename a lot of zones you can switch on the [Double-click in Navigator to edit model data](#) program option to speed up the process. This can be useful for identifying blocks, zones, surfaces by single-clicking in the Navigator and saves time when renaming many zones at once from block level as it becomes possible to click on the zone Navigator entries to rename them without navigating down to each zone and then having to navigate back up again to block level to rename the next zone.

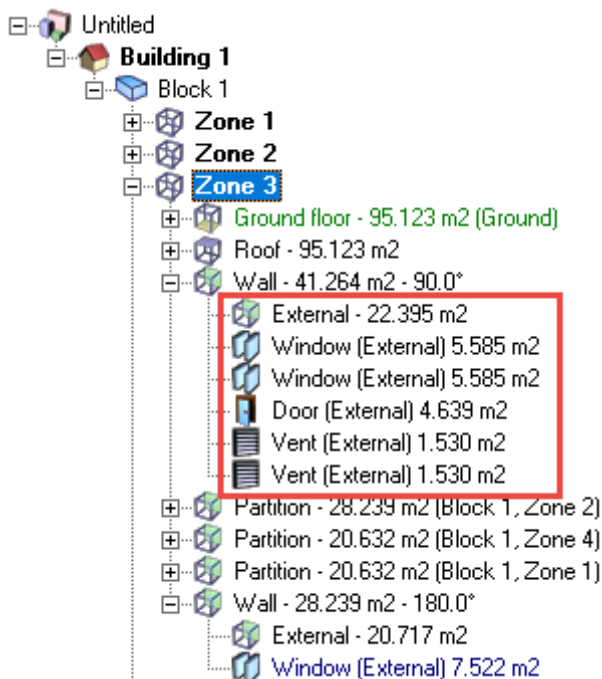
Surface name

Surfaces are given names automatically by DesignBuilder and these names cannot be changed by users.



Opening name

Openings are given names automatically by DesignBuilder and these names cannot be changed by users.



HVAC components

For Detailed HVAC systems, sub-loop components such as boilers, chillers, pumps, heat exchangers etc have editable names. On the other hand, sub-components such as AHU heating coils, water heater heating elements and all zone components and sub-components have hard-set, non-editable names derived from their parent components.

EnergyPlus IDF Names

DesignBuilder allows you to use a wider set of characters to name objects than are supported by EnergyPlus. This section explains how user-defined names are converted to EnergyPlus IDF names.

Zone names

Any characters in user-defined names not supported by EnergyPlus need to be replaced with characters that are allowed before being used as zone IDF object names. So if you work with intermediate EnergyPlus input or output files and need to be able to easily read IDF object names, or if you write your own [EMS scripts](#), or if

you use the Results Viewer, you should aim to ensure that the block and zone names you enter are such that the none of the steps below cause characters to be removed or replaced.

In summary, zone IDF names are created by adding the parent block name to the zone name itself, separated by the ":" character. For example, "Zone 5" in "Block 1" would be given an IDF zone name of "BLOCK1:ZONE5". The details of the block and zone name process are as described below.

1. The only characters allowed in DesignBuilder-generated IDF zone names are: 0-9, a-z, A-Z. All non-standard characters are replaced by the nearest English equivalent. E.g. "é" would be replaced by "e".
2. Space characters are removed.
3. If no corresponding English equivalent exists then the character is replaced by "X". E.g. "-" would be replaced by "X".
4. Words are capitalised.
5. If the name is greater than 20 characters, all vowels are removed.
6. If the name is still greater than 20 characters, the name is arbitrarily truncated to 20 characters.
7. The block and zones names processed as described above are added together to form a full IDF zone name as block name followed by ":" followed by zone name.
8. The full IDF name is checked for uniqueness and if not unique, an index number is added preceded by the "Q" character. For example if Zone 1 is defined twice in Block 1 then the second zone would be named "BLOCK1:ZONE1Q1" to ensure a difference from the first which would be named "BLOCK1:ZONE1".

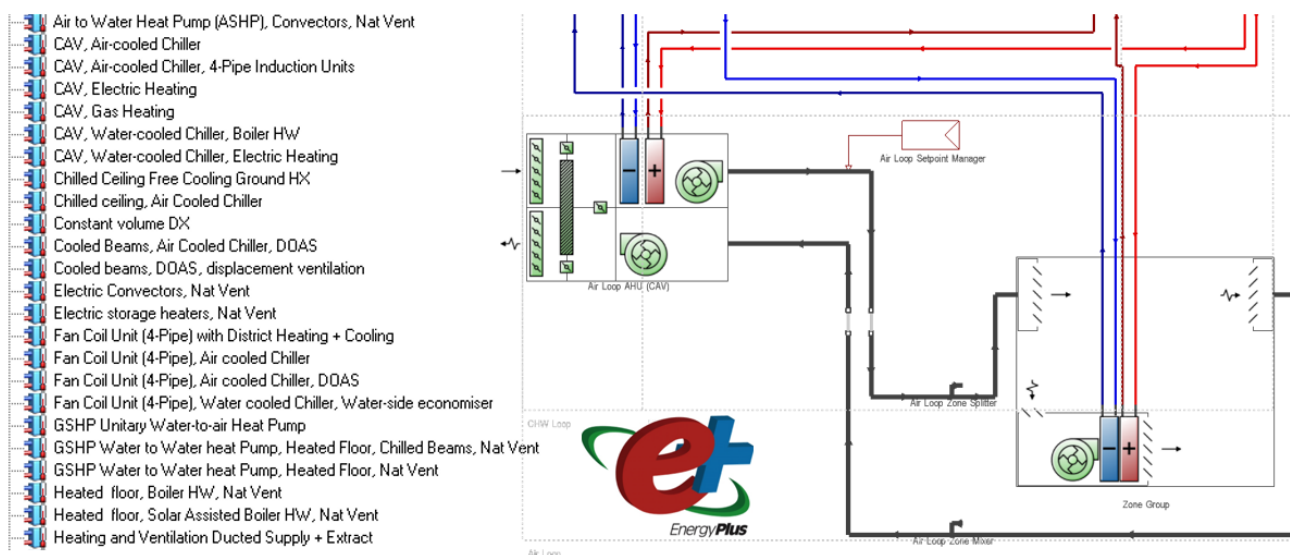
Note: Steps 1-6 are applied to both zone and block names individually.

HVAC component names

HVAC component names remain unaltered except in the case where a duplicate name is found in which case an index is added to the end of the name to produce a unique name.

Detailed HVAC

This section describes the graphical interface provided by DesignBuilder for the assembly of component based HVAC systems which may be combined with building models for energy simulation using EnergyPlus.



Overview

HVAC system models in DesignBuilder are assembled by placing a number of pre-defined air and water distribution loops together with groups of zones which are then connected to form complete systems.

See the [Introduction to Detailed HVAC](#) tutorial

In summary, with Detailed HVAC:

- Energy use is calculated based on detailed performance and control parameters such as performance curves, setpoint managers, sequential controls etc.
- Diagrammatic system is built by connecting components of most common type of E+ systems to form loops.
- Air and water distribution systems are modelled using loops for hot water, chilled water loop, condenser and air supply.
- Zone groups are used to specify secondary, zone level system components and zones can have multiple types of terminal units simultaneously.

The [air](#) and [water](#) loops incorporate a number of HVAC components connected together by pipes or ducts to form distribution systems. These loops can be modified to incorporate any additionally required components and then connected to associated components in other loops or to heating, cooling and ventilation equipment located in zones to form complete systems.

Zones are placed in the form of [zone groups](#) which enable a number of zones having the same HVAC equipment to be grouped together. Each zone in the group can have different equipment ratings. The zone group mechanism greatly simplifies the HVAC schematic assembly in that only one set of equipment and connections needs to be defined for the group while maintaining the flexibility of changing equipment properties for individual zones within the group.

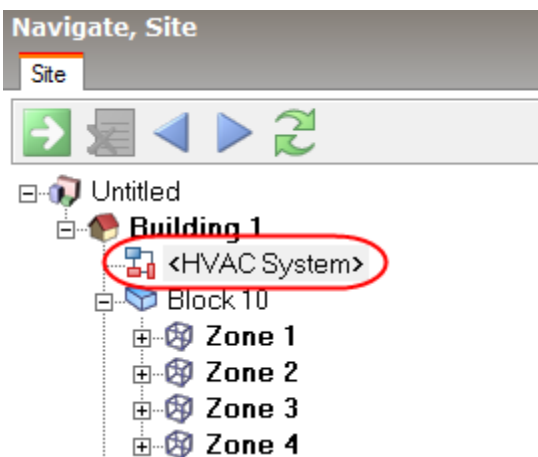
The pre-defined loops incorporate [setpoint managers](#) which enable the loop to run without any additional control. However, additional setpoint managers can be added to the various loops to customise the control of the system.

Data associated with all loops, zones and associated [components](#) can be edited via dialogs for specific system requirements.

After placing loops, zone groups and setpoint managers and completing the required loop connections to associated equipment, an EnergyPlus simulation can be conducted for the combined building and HVAC system model.

Accessing Detailed HVAC

To access detailed HVAC, you first need to select the [Detailed HVAC](#) option under the HVAC header on the [Model Options dialog](#). After closing the Model Options dialog, a new **<HVAC System>** node will appear in the Navigator tree list:



Click on this **<HVAC System>** node to access the HVAC edit screen.

Detailed HVAC Contents

- [HVAC Loops](#)
- [HVAC Zones and Zone Groups](#)
- [HVAC Component Hierarchy and Navigation](#)
- [HVAC System Control - Setpoint Managers](#)
- [Working at HVAC System Level \(Templates, Loops, Zone Groups\)](#)
- [Working at Sub Loop Level](#)
- [Working at HVAC Zone Group Level](#)
- [Working at HVAC Zone Level](#)
- [Working at Component Level](#)
- [Autosizing HVAC Components](#)
- [Humidity Control](#)
- "HVAC Components" on page 787

Detailed HVAC Terminology

The following points on HVAC terminology may help when referring to the documentation.

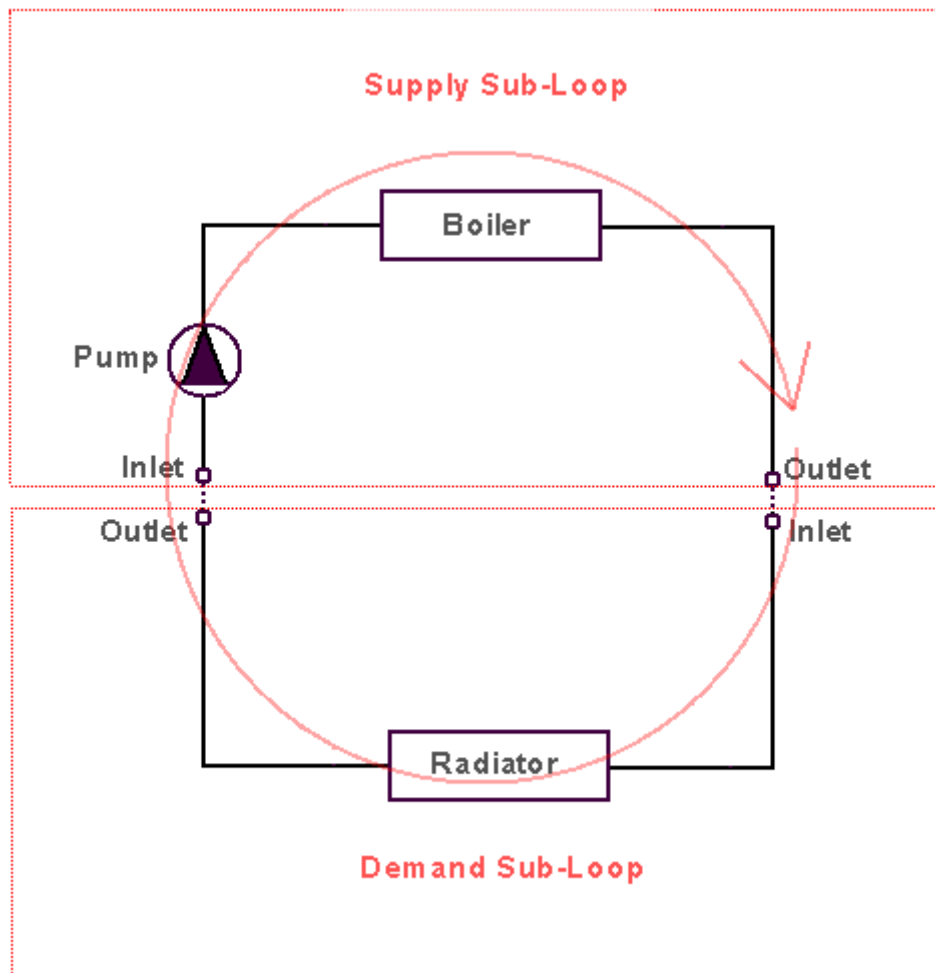
- The word "supply air" refers to the air being supplied to the zone. So for example the "supply air flow rate" is the rate of air delivered to the zone. It is not necessarily the *fresh air* delivery rate (though if there is no recirculation then it will be).

HVAC Loops

An HVAC loop is used to model air and water distribution systems such as mechanical ventilation systems, hot and chilled water piped distribution systems and condenser water distribution systems. A loop is divided into two sub-loops, a **demand** sub-loop and a **supply** sub-loop.

See the [Adding loops and connecting components](#) tutorial

At its simplest, a sub-loop comprises an inlet connection (pipe or duct) which is connected in series via a number of components to an outlet connection. The inlet connection of each loop sub-loop connects with the outlet connection of the other loop sub-loop:



The **demand** sub-loop is typically that section of a loop from which energy is **extracted** from a system and the **supply** sub-loop is that section of a loop to which energy is **supplied** by the system.

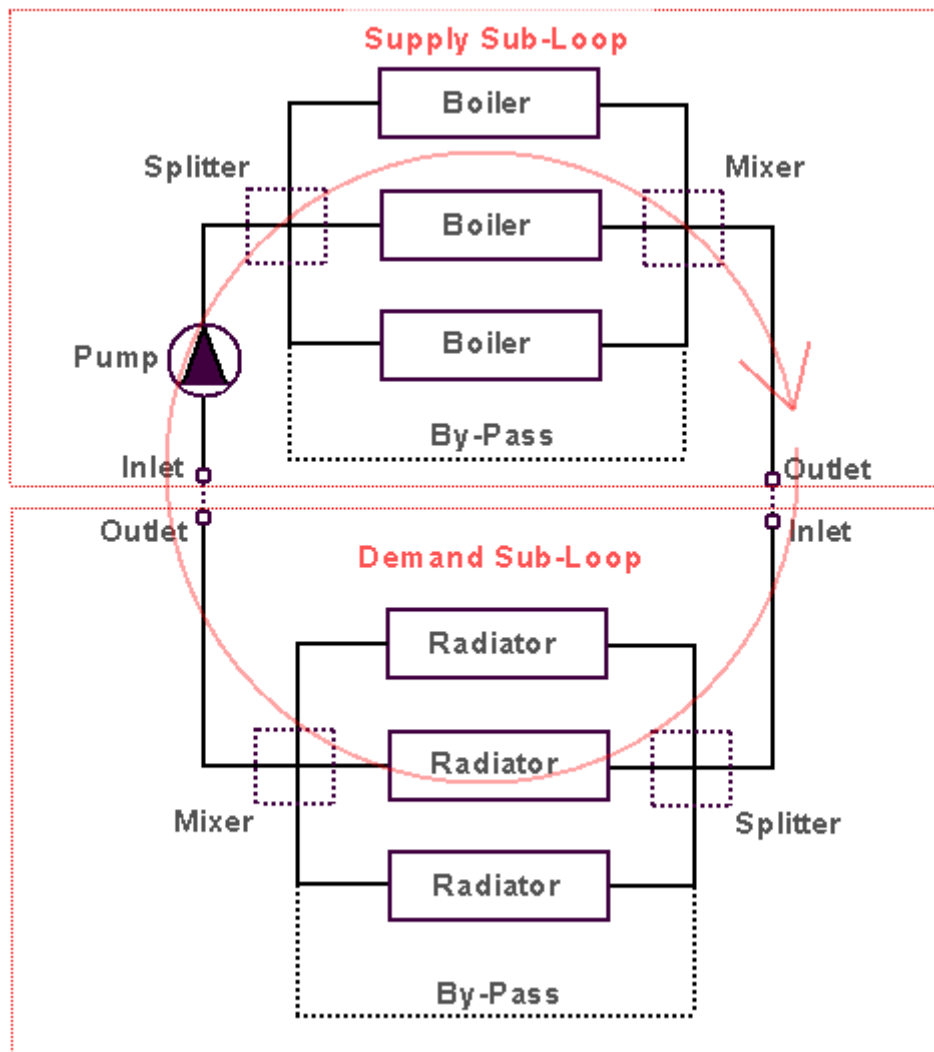
Consequently, **supply** sub-loops incorporate equipment such as boilers in the case of heating systems or chillers in the case of cooling systems, the supply sub-loop of an air system incorporates items such as heating and cooling coils. The supply sub-loop also contains circulating equipment such as pumps and fans.

Similarly, the **demand** sub-loop of a heating system includes items such as air conditioning heating coils or zone equipment such as hot water radiators, fan-coil unit heating coils, under-floor heating, etc. and similarly a chilled water demand sub-loop contains equipment such as air condition cooling coils, zone cooled beams, chilled ceilings, etc. The sub-loop of an air loop typically incorporates zone air distribution equipment such as VAV boxes, etc.

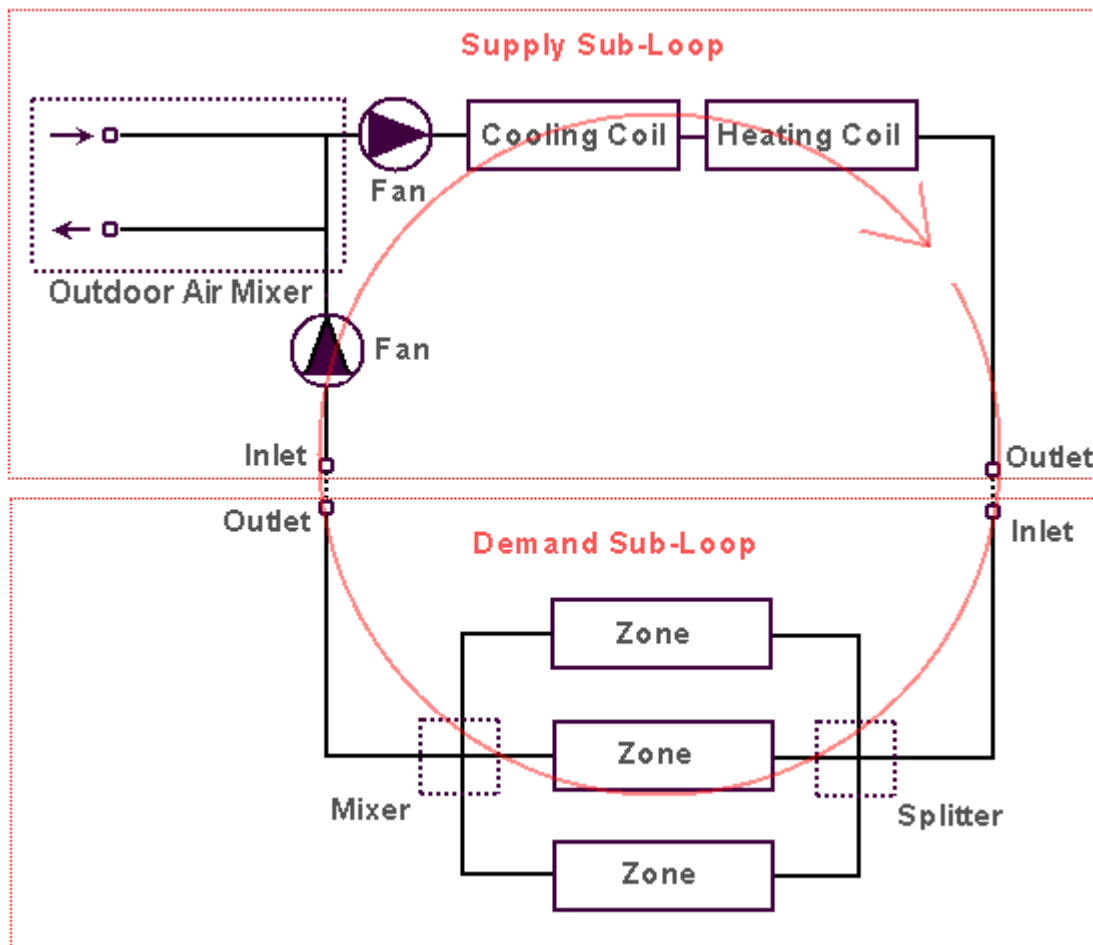
There are 4 main types of loop:

- **Plant loops** (hot and chilled water loops),
- **Condenser loops,**
- **VRF loops,**
- **Air loops.**

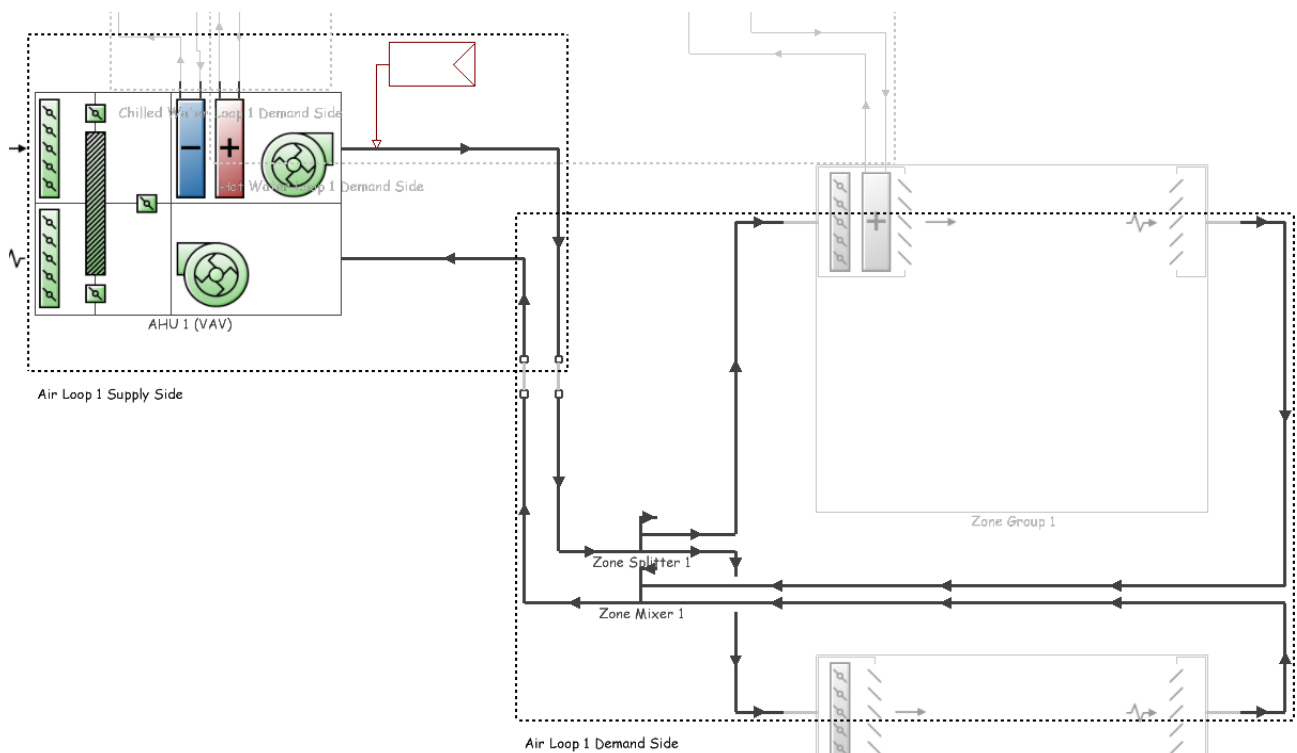
Plant and condenser sub-loops incorporate a flow splitter and a flow mixer which allow a number of parallel branches to be defined. Each parallel branch would normally comprise an inlet connection from the splitter, a component and an outlet connection to the mixer. A by-pass parallel branch (without a component) is also included to ensure flow continuity.



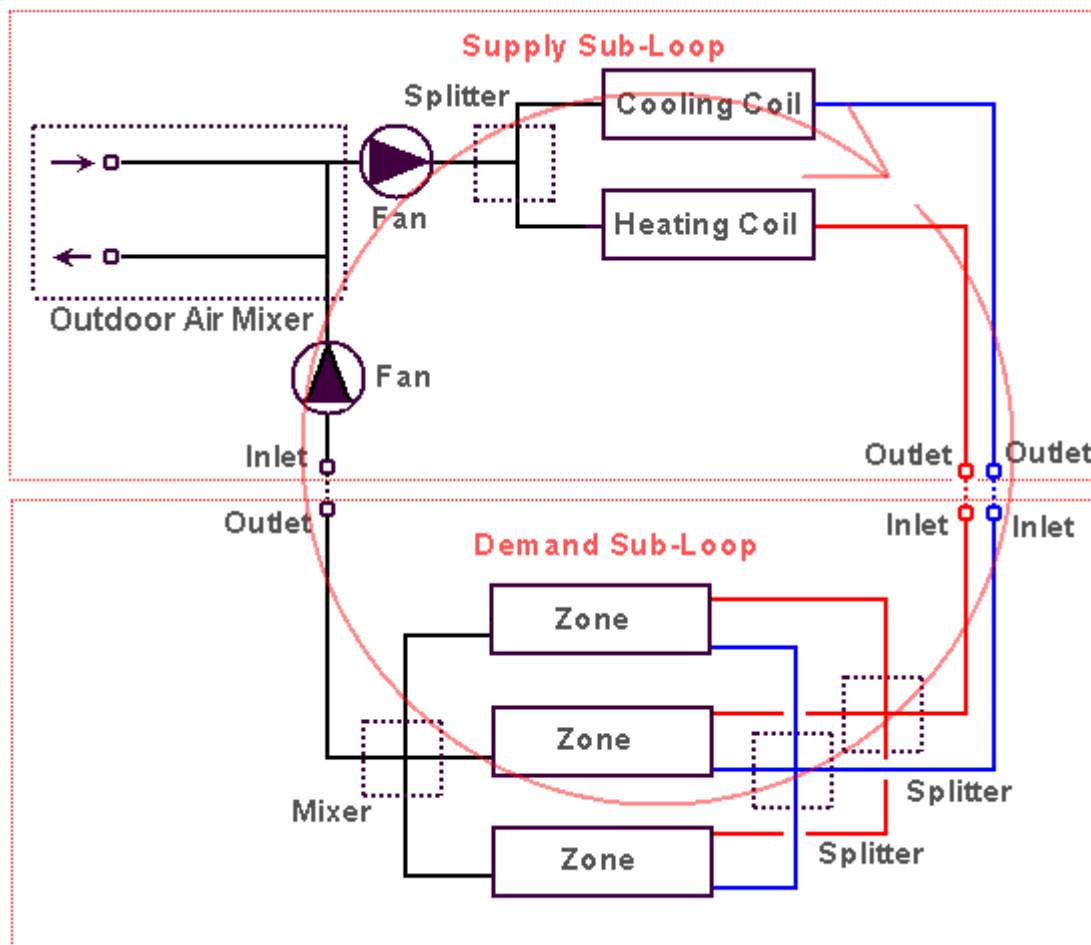
The **supply** sub-loops for **single duct air loops** on the other hand generally comprise a single branch which incorporates an inlet connection connected to various components in series including an outdoor air mixer, fans, cooling coils, heating coils, etc. which are in turn connected to an outlet connection. Air loop supply sub-loops do not contain flow splitters or mixers. **Demand** sub-loops for **single duct air loops** incorporate the supply path connections to zone air distribution equipment and zone return air path connections. Air loop demand sub-loops also incorporate a flow splitter and a flow mixer allowing air distribution equipment in a number of zones to be connected to the supply and return paths:



The outdoor air mixer and fan components in DesignBuilder HVAC are incorporated within an air handling unit component. Optionally, additional components such as heat recovery, heating coils, cooling coils, humidifiers and outdoor air pre-treatment coils can also be added to air handling units:



Air **supply** sub-loops for **dual duct systems** incorporate two outlet connections, one for the hot duct and one for the cold duct; dual duct supply sub-loops also incorporate a single air splitter to split the flow between the hot and cold ducts:



The **demand** sub-loops for **dual duct air loops** similarly contain two inlet connections for hot and cold ducts which can then be connected to zone dual duct air distribution terminals via air splitters and a mixer.

There are various pre-defined loops in DesignBuilder HVAC, 4 [air loops](#), 5 [plant loops](#) a [VRF loop](#) and a [condenser loop](#).

Air Loops

Air loops are used by EnergyPlus to distribute mechanically driven air to zones. In the Air loops provided by DesignBuilder, the air flow supplied into the zones is the same as the flow out. In other words air leakage is not accounted for.

There are 5 pre-defined air loops provided by DesignBuilder detailed HVAC:

- "Air Loop - Generic single duct" on page 710
- "Air Loop - Generic dual duct" on page 710
- "air loop - Unitary heat Cool" on page 711
- "air loop - Unitary heat pump Air to Air" on page 712
- "Air Loop - Unitary Heat Pump Water To Air" on page 712

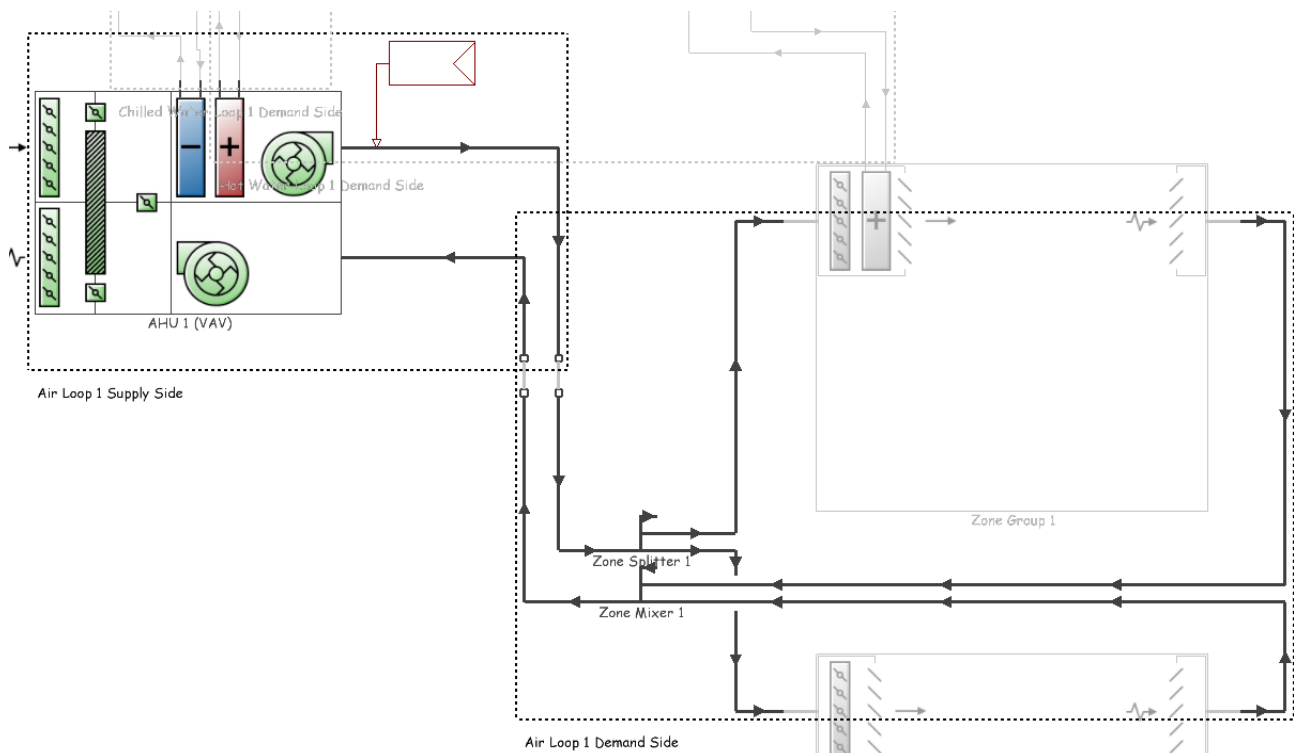
See the [Adding air loops and AHUs](#) tutorial

Air Loop - Generic Single Duct

This air loop incorporates a [Generic air handling unit \(AHU\)](#) component in the supply sub-loop to which may be added various sub-components including pre-treatment coils (heating and cooling), heat recovery, recirculation with economiser, heating and cooling coils and a humidifier. The air handling unit automatically includes supply and extract fans. The demand sub-loop incorporates a flow splitter and a flow mixer to enable the loop to be connected up to zone equipment. The loop incorporates a single scheduled [setpoint manager](#) which by default is pre-defined to maintain a supply temperature.

See the [Adding air loops and AHUs](#) tutorial

The image below shows a single duct air loop connected to 2 [Zone groups](#).



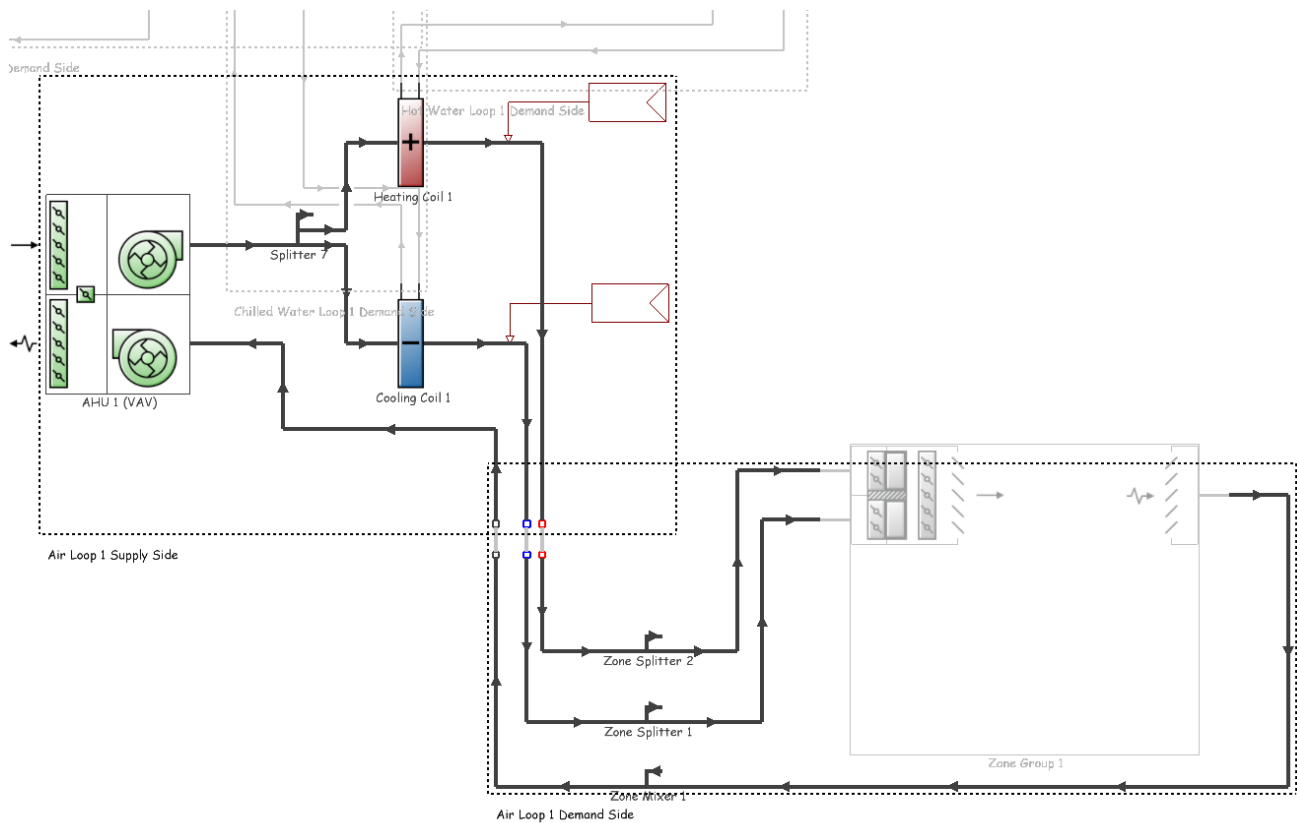
See also:

- [Air Loop Data](#)

Air Loop - Generic Dual Duct

This air loop is similar to the [Generic single duct air loop](#) but includes two outlet connections for the hot and cold ducts in the supply-sub loop together with associated heating and cooling coils. The loop also incorporates a [Warmest setpoint manager](#) to control the cold duct temperature and a [Coldest setpoint manager](#) to control the hot duct temperature.

The image below shows a dual duct air loop connected to a [Zone group](#).



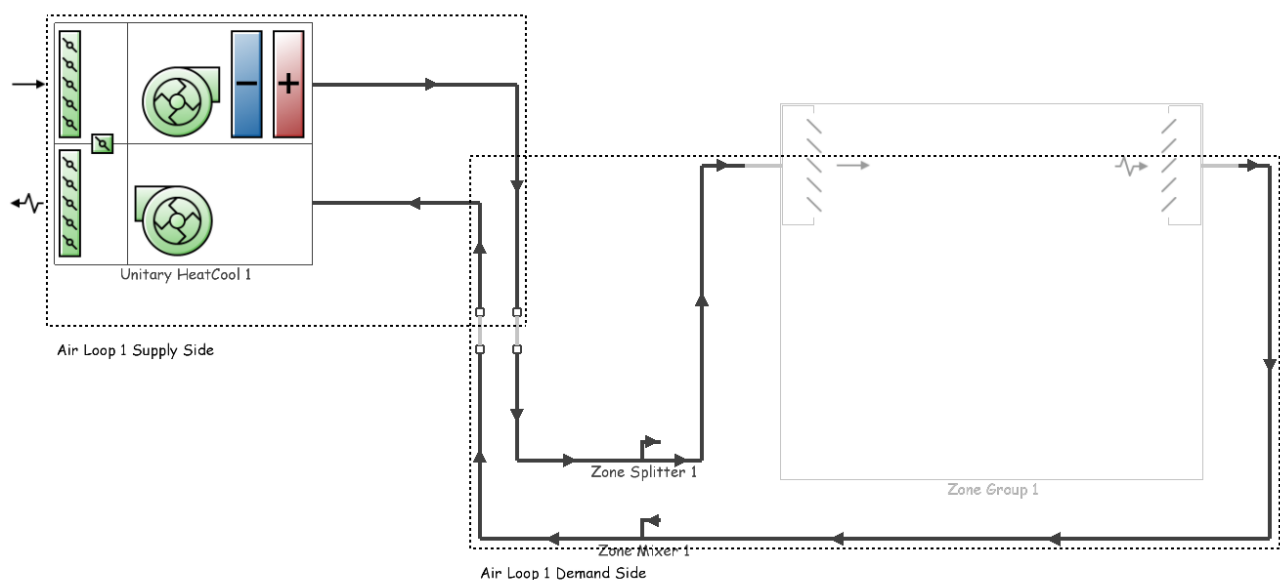
See also:

- [Air Loop Data](#)

Air Loop - Unitary Heat Cool

The unitary heat cool air loop is similar to the [Generic single duct air loop](#) but instead of a [Generic air handling unit](#), the supply sub-loop contains a [Packaged unitary heat cool AHU](#) component which incorporates [DX heating](#) and [DX cooling coils](#) consequently does not require any plant or condenser loops to be defined.

The image below shows a unitary heat cool air loop connected to a [Zone group](#).



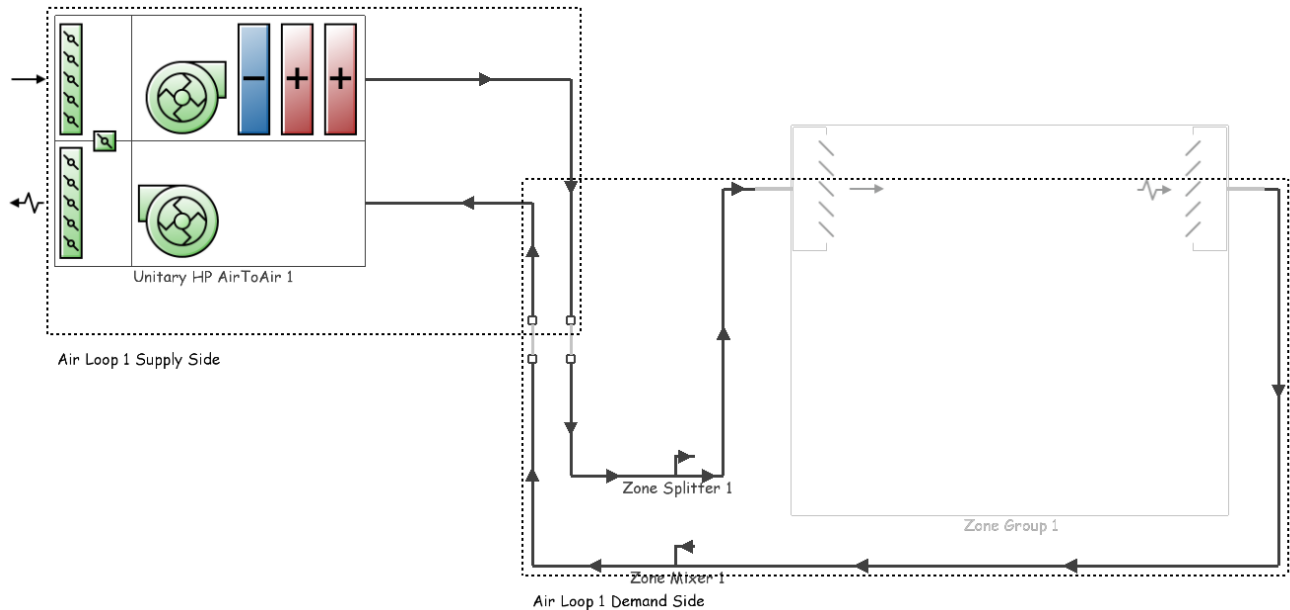
See also:

- [Air Loop Data](#)

Air Loop - Unitary Heat Pump Air To Air

This unitary air loop is similar to the [Generic single duct air loop](#) but instead of a [Generic air handling unit](#), the supply sub-loop contains a [Packaged unitary air to air heat pump](#) component which incorporates [DX heating](#) and [DX cooling coils](#) together with a supplementary electric, hot water or fuel heating coil.

The image below shows a unitary heat pump air to air connected to a [Zone group](#).



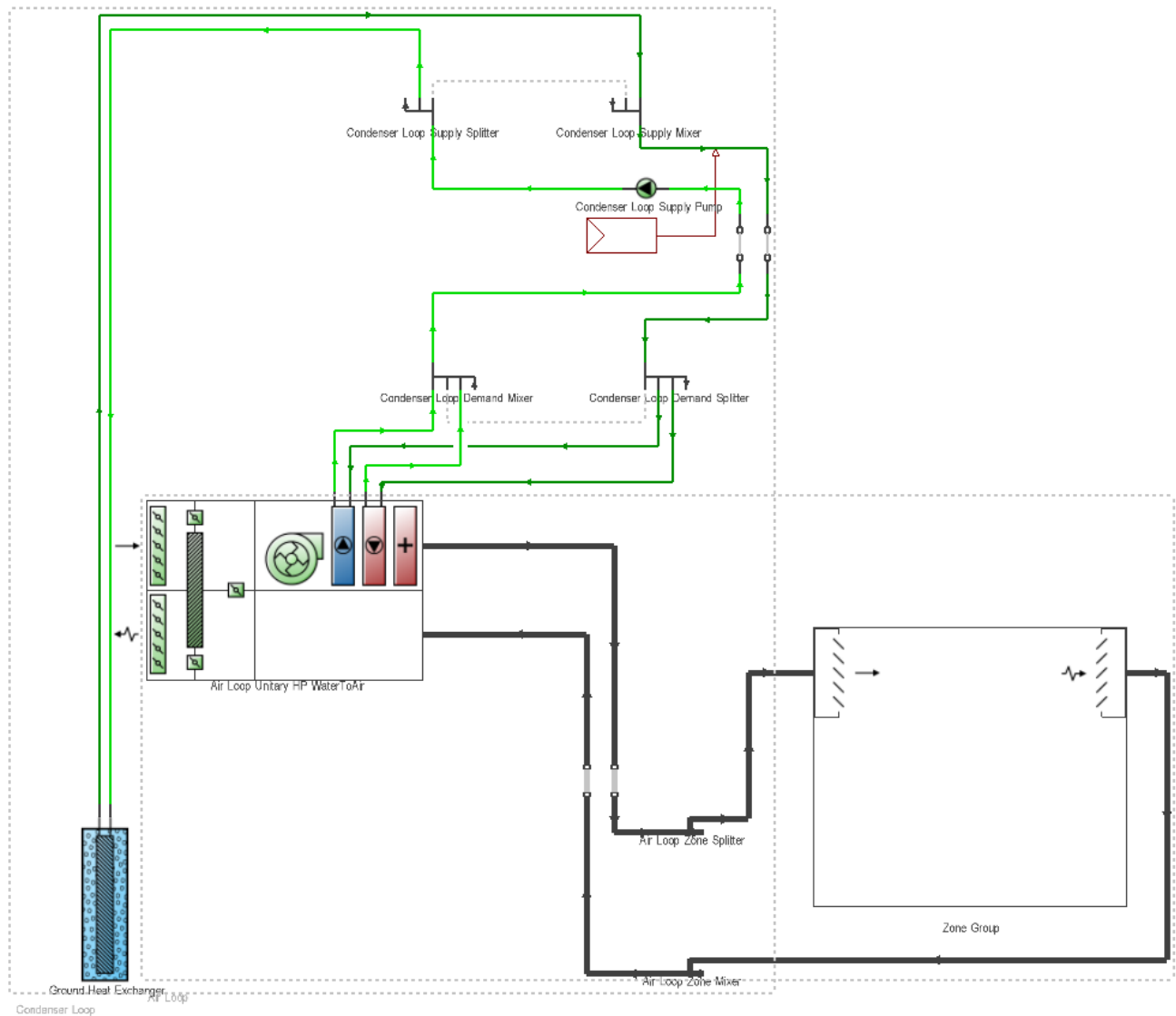
See also:

- [Air Loop Data](#)

Air Loop - Unitary Heat Pump Water To Air

This unitary air loop is similar to the [Generic single duct air loop](#) but instead of a [Generic air handling unit](#), the supply sub-loop contains a [Packaged unitary heat pump water to air](#) component which incorporates an On/Off fan component, a WaterToAirHeatPump cooling coil component, a WaterToAirHeatPump heating coil component. The source side (water) of the heat pump is connected to a condenser loop with a heat exchanger (ground heat exchanger or other type) or a plant loop with a heating source such as a boiler and a cooling source such as a chiller or cooling tower.

The image below shows a unitary heat pump water to air connected to a [Zone group](#).



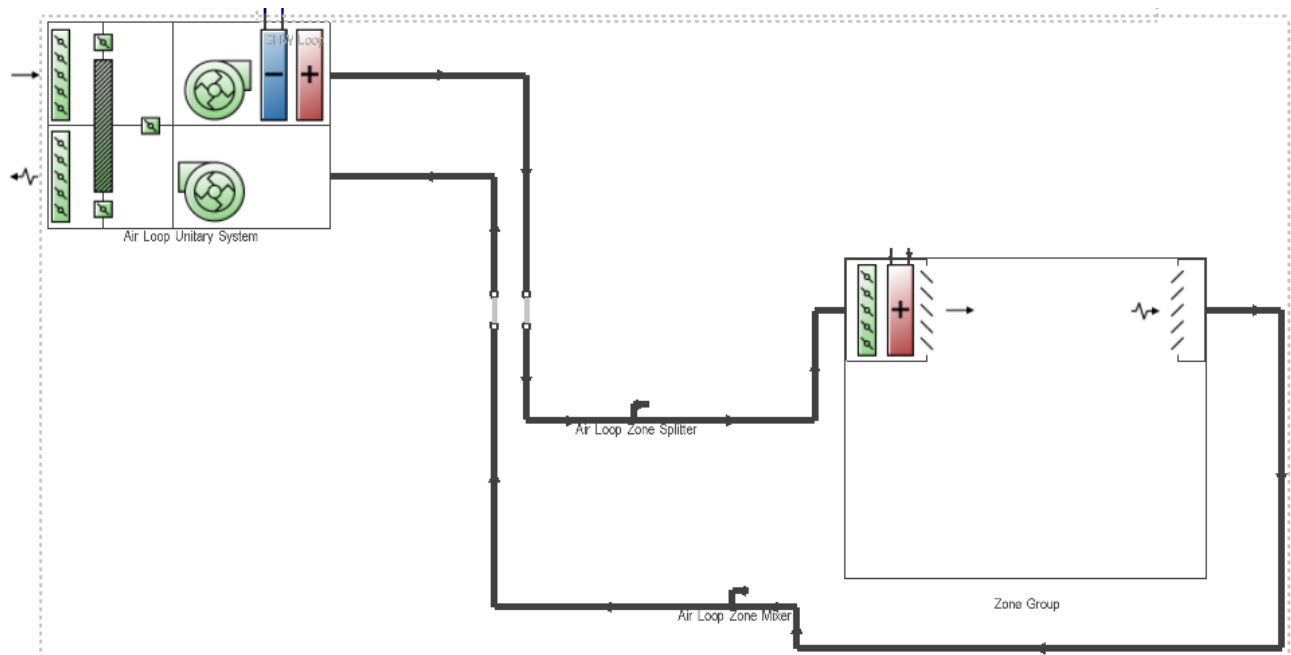
See also:

- [Air Loop Data](#)

Air Loop - Unitary Generic

The Generic Unitary Air Loop consists of a fan component (on/off, constant volume or variable volume), a cooling coil component, a heating coil component, and a reheat coil as shown below. When a draw through configuration is required, the fan is placed directly after the heating coil. If dehumidification control is selected, a reheat coil component is also required. If the reheat coil is present and the dehumidification control type input is not specified as CoolReheat, the reheat coil will not be active.

Note: All of the fan and coil components are optional which allows these loops to be configured for fan-only, heating-only, cooling-only, or both heating and cooling.



The image above shows a unitary generic air loop connected to a [Zone group](#) with heat recovery and recirculation.

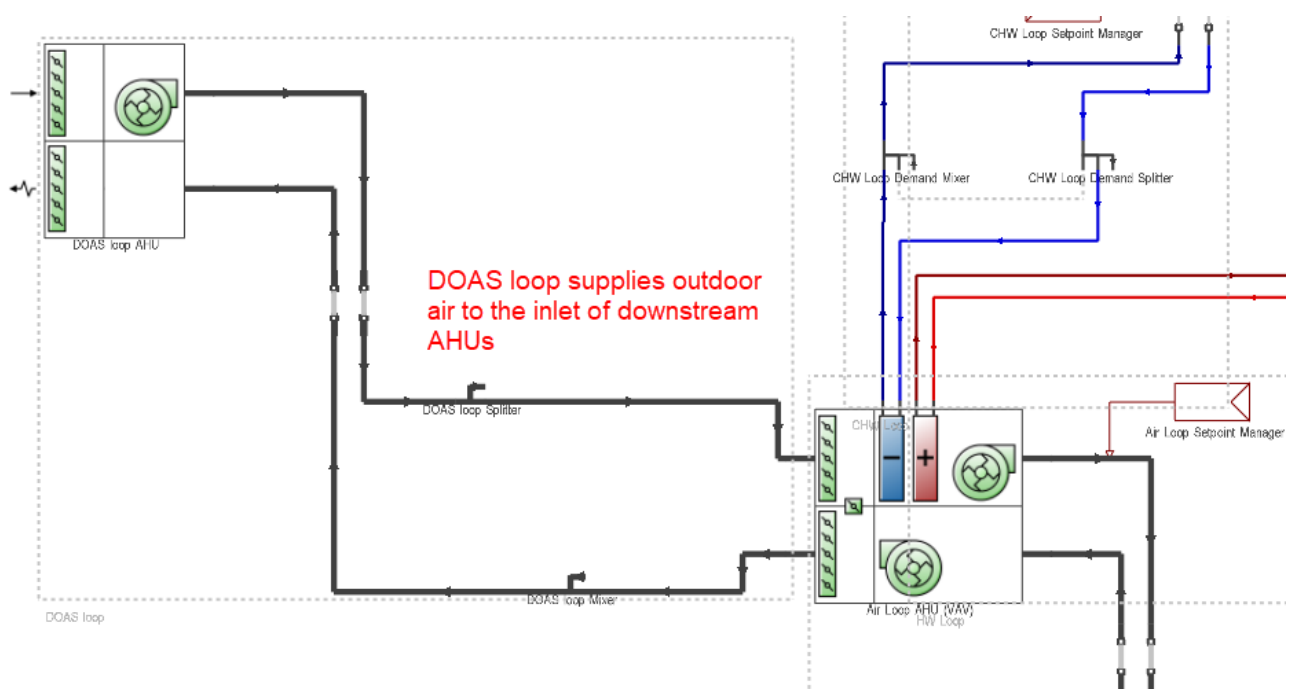
See also:

- [Air Loop Data](#)

Air Loop - Dedicated Outdoor Air System

The Dedicated Outdoor Air System Air loop (or DOAS loop) is a central air system that delivers outdoor air to one or more child air loop systems. The amount of delivered outdoor air is based on the sum of the outdoor air flow demand from the air loop systems that it serves.

The central DOAS system can pre-treat outdoor air with given precool and preheat air conditions before distributing it to the downstream air loops.



The image above shows a DOAS loop connected to a downstream VAV Air loop.

See also:

- [DOAS Loop Data](#)

Plant And Condenser Loops

A set of pre-defined plant loops are provided:

- [Hot water plant loop](#)
- [Chilled water plant loop](#)
- [Primary/secondary chilled water plant loop](#)
- [Domestic hot water/solar plant loop](#)
- "Solar and Auxiliary Heating Loops" on page 720

A single condenser loop is provided:

- [Condenser loop](#)

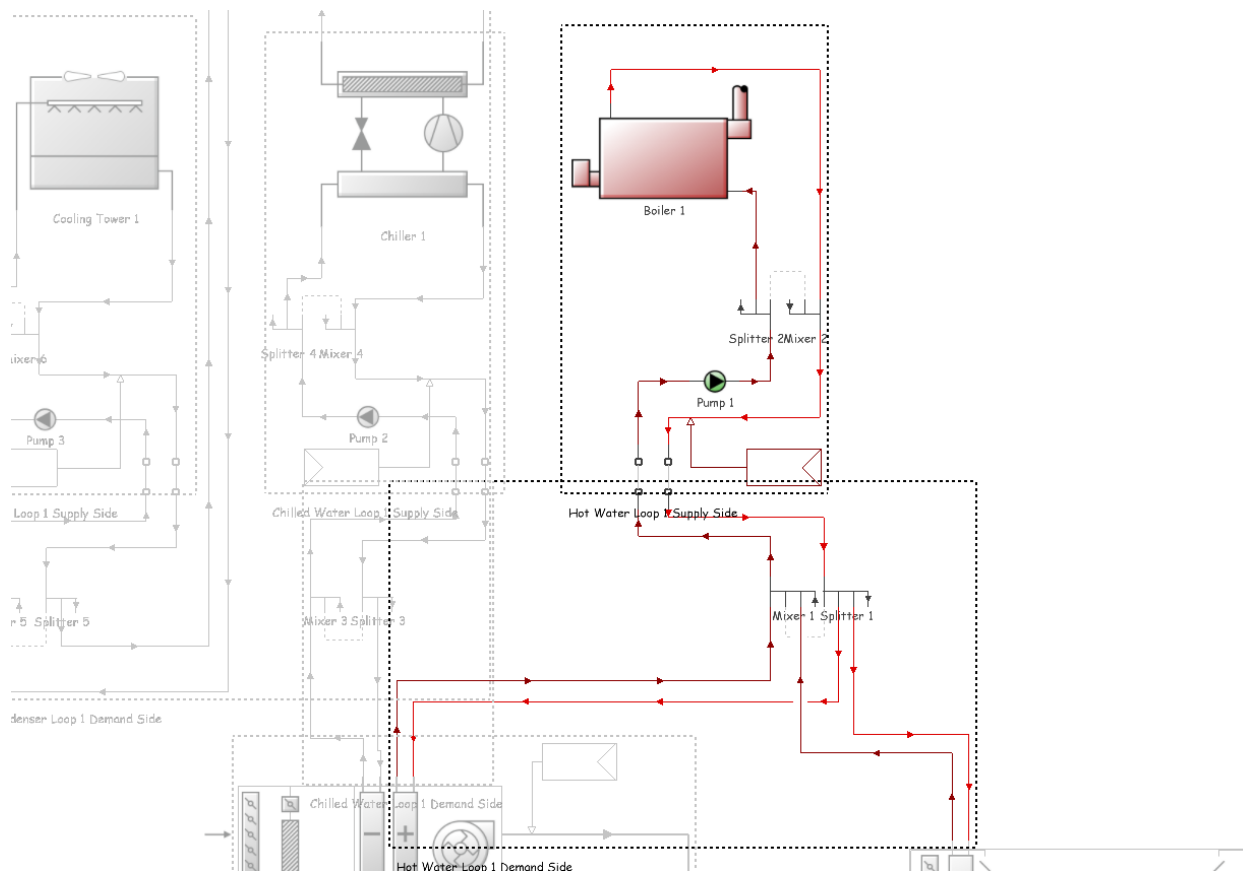
A **plant equipment operation** facility is included to provide plant and condenser loops with a mechanism for controlling the operation of the loop and to specify which equipment is available under what conditions. This allows you to model scheduled sequencing of plant equipment depending on time-varying load requirements. For example you may want to model a multiple chiller system and schedule low load chillers to handle loads up to a certain magnitude and then switch in higher load chillers as the cooling load increases. This feature is available through the plant/condenser loop [Plant Equipment Operation](#) tab.

Hot Water Plant Loop

The hot water plant supply sub-loop incorporates a one or more boilers connected in series or parallel, flow splitter and flow mixer together with a circulating pump.

The sub-loop also includes a scheduled temperature [setpoint manager](#) which is set up by default to control the off-plant temperature at a constant 80°C.

The demand sub-loop incorporates a flow splitter and flow mixer in order that the loop can be connected to heating plant components. The image below shows a hot water plant loop connected to 2 [water heating coils](#).



See the [Adding loops and connecting components](#) and [Editing HVAC loop and component properties](#) tutorials

See also:

- "Plant and Condenser Loop Data" on page 731

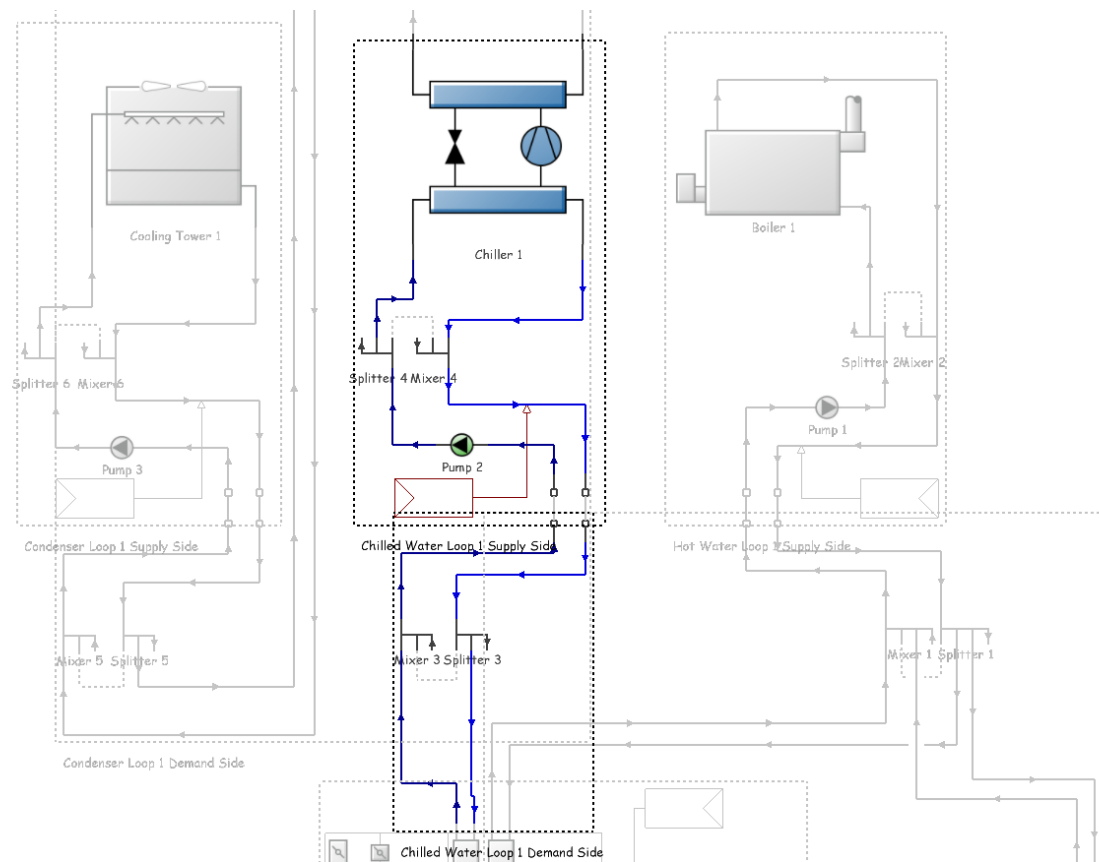
Chilled Water Plant Loop

The chilled water plant supply sub-loop incorporates one or more chillers connected in either series or parallel, flow splitter and flow mixer together with a circulating pump.

By default, the chiller has a water cooled condenser and connections are included to enable the condenser to be connected up to a condenser loop. Air cooled chillers can also be selected.

The sub-loop also includes a scheduled temperature [setpoint manager](#) which is set up by default to control the off-plant temperature at a constant 6.0°C.

The demand sub-loop incorporates a flow splitter and flow mixer in order that the loop can be connected to cooling plant components. The image below shows a chilled water loop connected to a [water cooling coil](#).



See also:

- "Plant and Condenser Loop Data" on page 731

Primary/Secondary Chilled Water Plant Loop

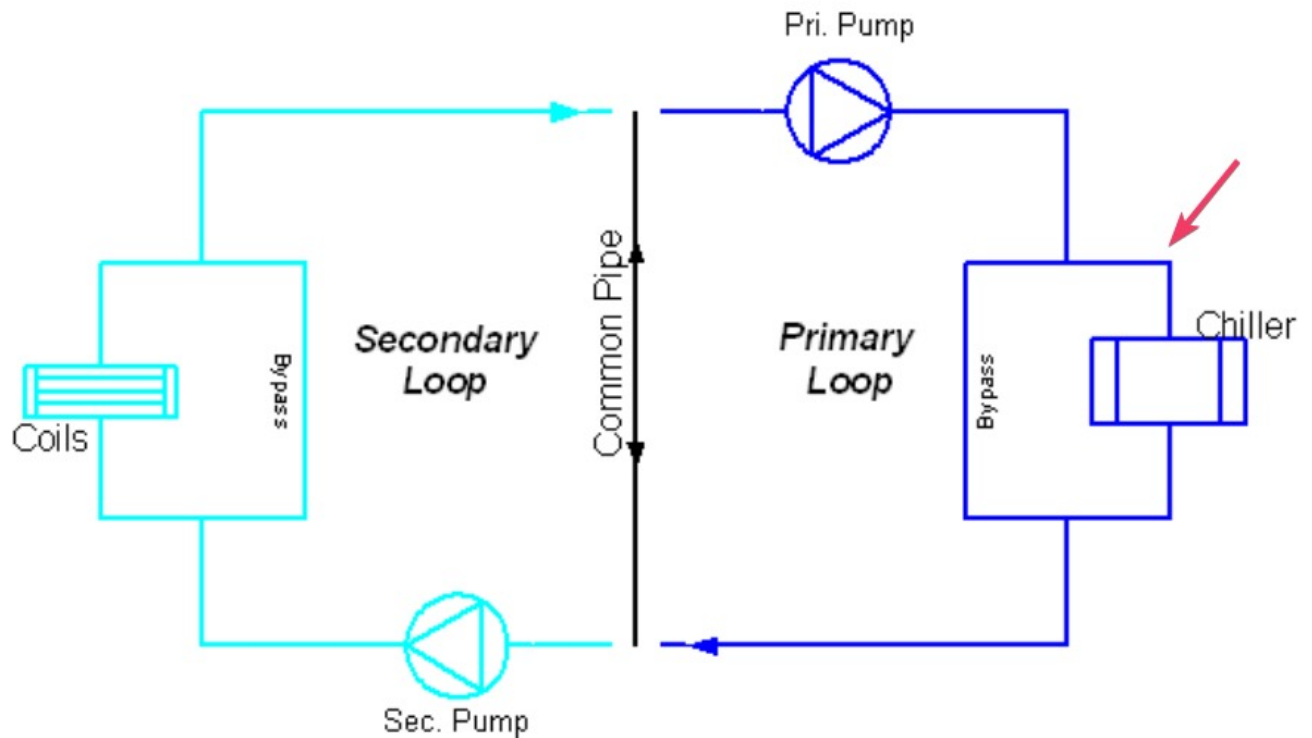
The primary/secondary chilled water plant loop uses the EnergyPlus Common Pipe Simulation mechanism to represent a chilled water circuit comprising both primary and secondary pumped sub-loops. The primary sub-loop incorporates one or more chillers connected in either series or parallel, flow splitter and flow mixer together with a circulating pump.

By default, the chiller has a water cooled condenser and connections are included to enable the condenser to be connected up to a condenser loop. Air cooled chillers can also be selected.

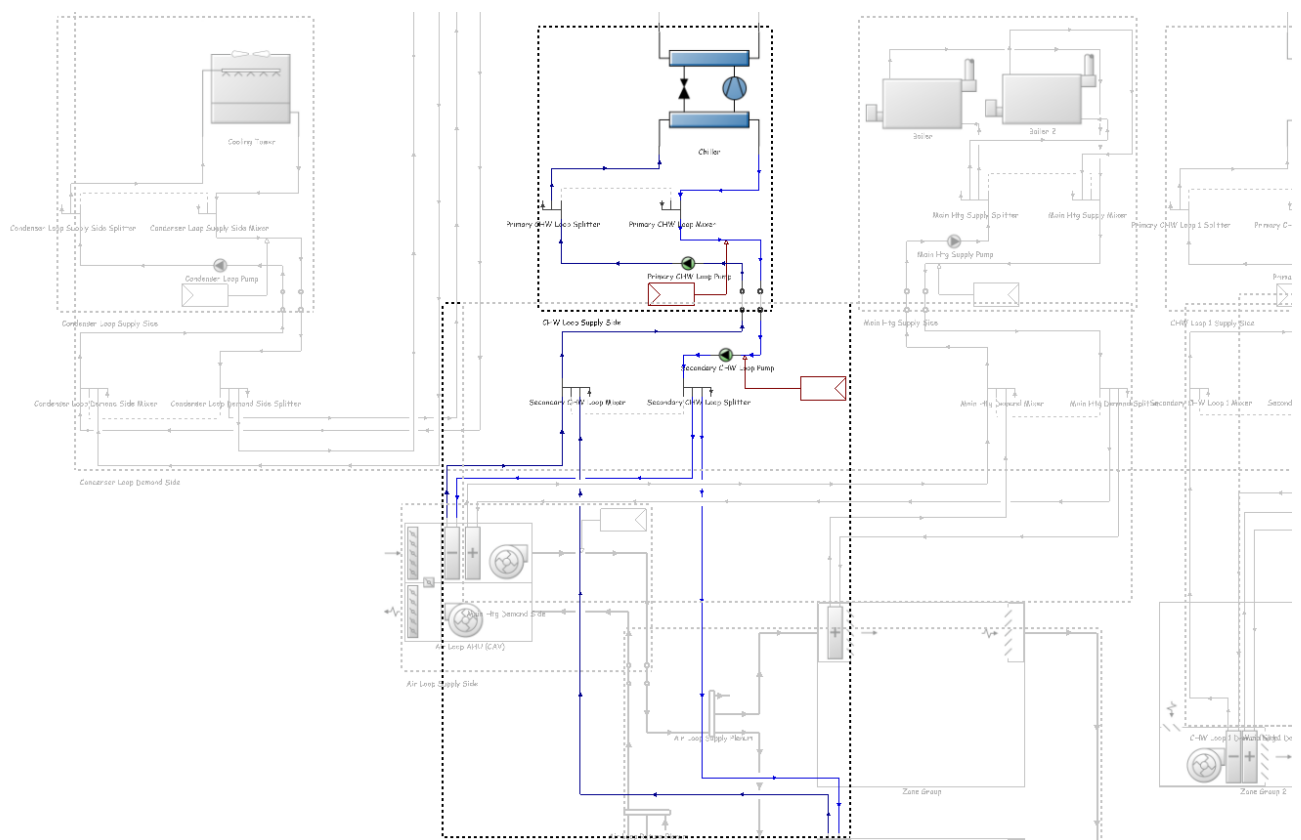
The primary sub-loop also includes a scheduled temperature [setpoint manager](#) which is set up by default to control the sub-loop outlet temperature at a constant 6°C.

The secondary sub-loop includes a scheduled temperature setpoint manager which is pre-defined to control the sub-loop inlet temperature at a constant 6°C but the schedule defining this temperature can be replaced with any required alternative. The secondary sub-loop also incorporates a flow splitter and flow mixer in order that the loop can be connected to cooling plant components.

Note: In an EnergyPlus common-pipe (primary-secondary) loop the pump must be located on the first (inlet) branch of the half-loop. Branch-level pumps located just ahead of a chiller, as shown by the red arrow in the layout below, are *not* supported.



Use the common-pipe option only when a single loop pump on the primary side is acceptable. If your project includes a pump mounted directly at each chiller, consider one of the alternative modelling approaches outlined in [this Knowledge Base article](#), rather than attempting to reposition the pump within a single common-pipe loop - something that EnergyPlus does not currently support.



See also:

- "Plant and Condenser Loop Data" on page 731

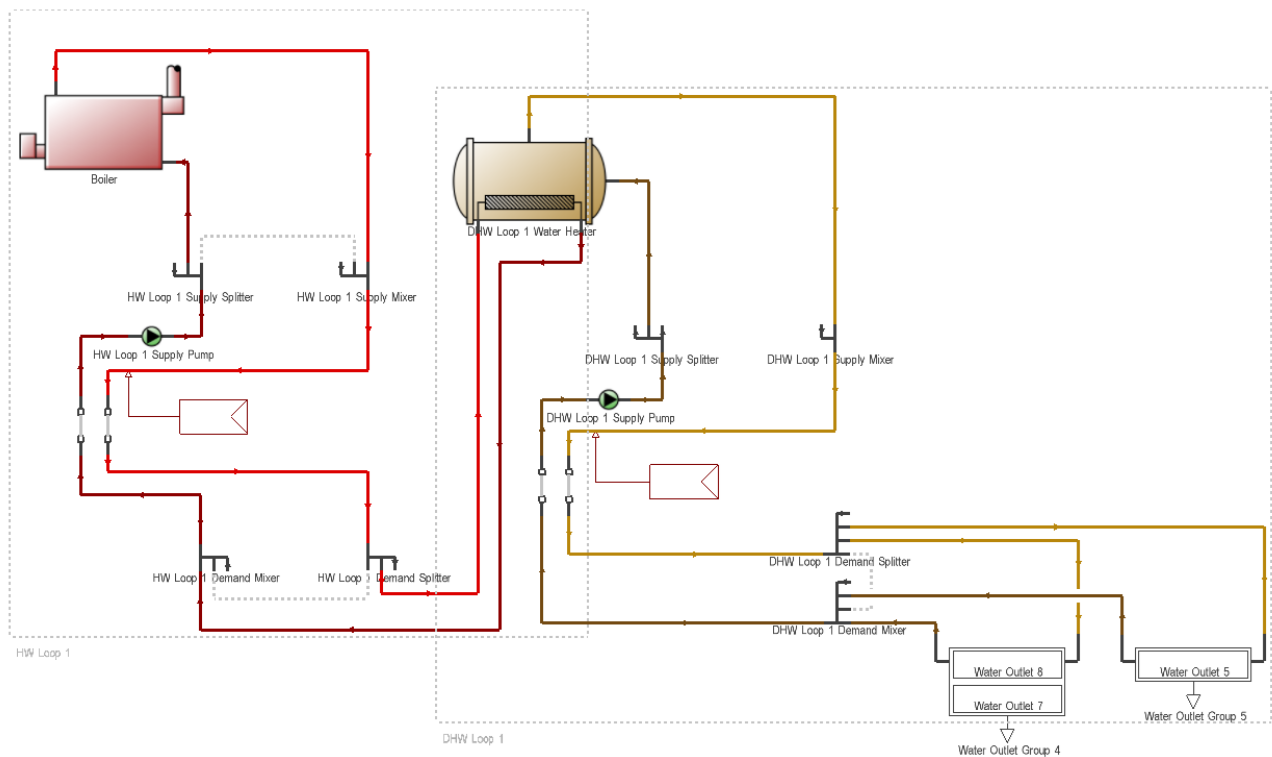
Domestic Hot Water Plant Loop

The domestic hot water (DHW) plant supply sub-loop incorporates one or more [water heaters](#) connected in series or parallel, flow splitter and flow mixer together with a circulating pump.

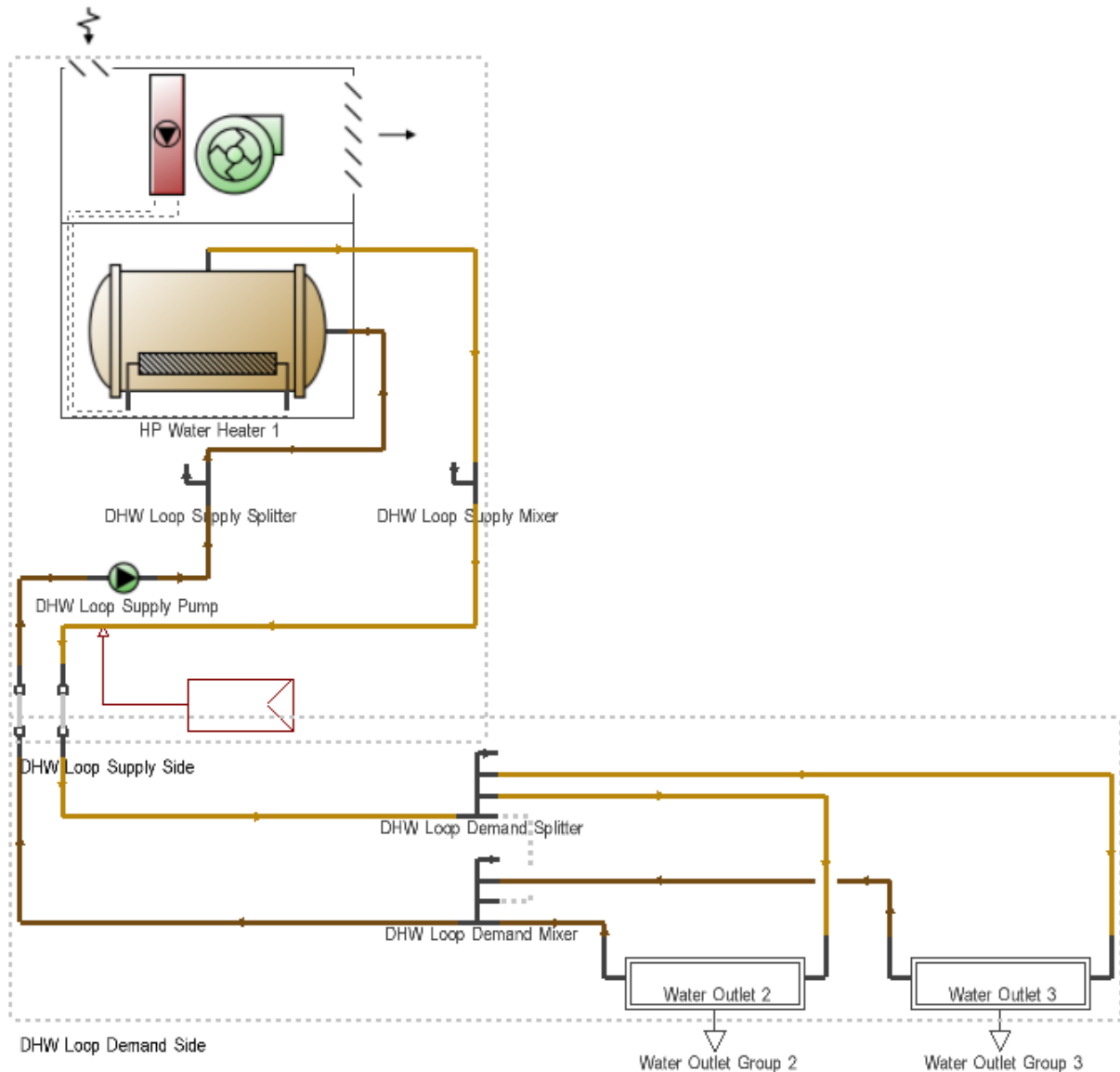
The sub-loop also includes a scheduled temperature [setpoint manager](#) which is set up by default to control the off-plant temperature at a constant 55°C.

The demand sub-loop incorporates a flow splitter and flow mixer in order that the loop can be connected to hot water outlets.

The screenshot below shows a typical DHW application with a hot water plant loop connected to DHW loop with a water outlet group.



Another common DHW loop configuration uses an [Air Source Heat Pump](#) on the supply side as shown below:



See also:

- "Plant and Condenser Loop Data" on page 731

Solar And Auxiliary Heating Loops

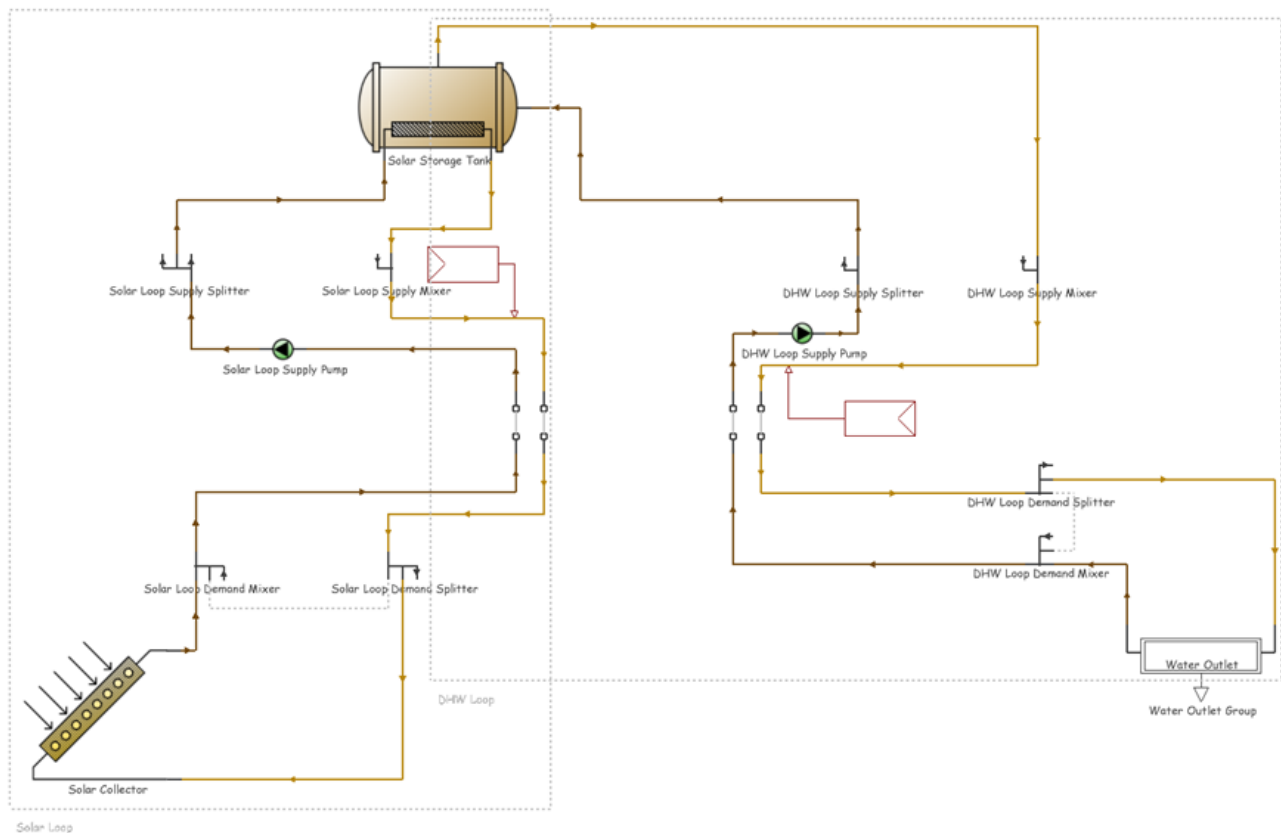
DesignBuilder enables you to model solar-heated water for domestic hot water (DHW) and/or space heating use. Various configurations of solar heating systems can be created with increasing levels of sophistication using a combination of solar collectors, pumps, water tanks, and water heaters. These include:

- Solar heating system: A simple system where the solar collector is the only heating source linked directly to a DHW loop to supply heated water to zones.
- Solar and Auxiliary heating system: A two-tank system is required to model a solar collector with downstream back-up auxiliary heating which can boost the water temperature if required.
- High-temperature Solar and Auxiliary heating system: Adding a three-way tempering valve enables much higher storage temperatures thereby increasing the capacity of the system to store free solar energy or auxiliary heat purchased during off-peak times.
- Boiler-assisted High-temperature Solar heating system: Boilers can also be connected to serve as the auxiliary heating sources.

These 4 system types are described in more detail below.

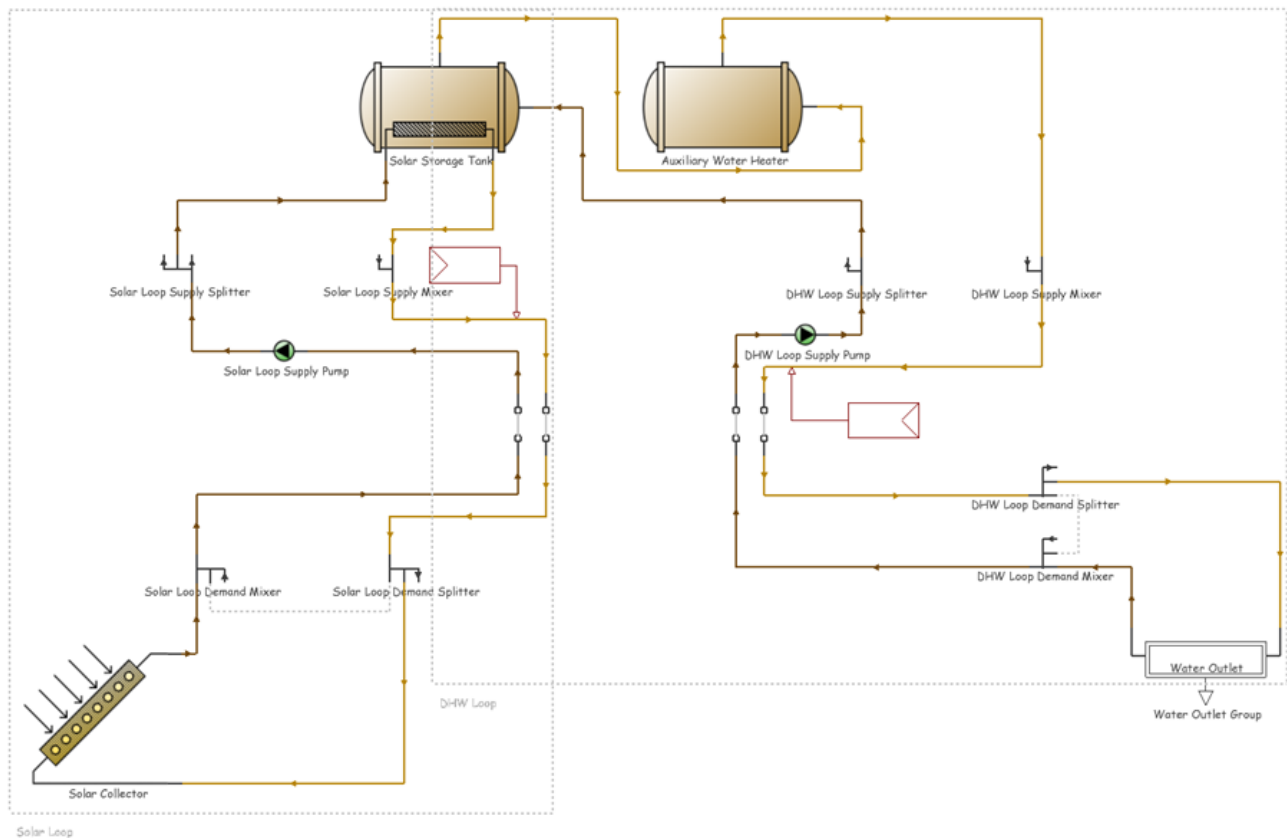
Solar Heating System (without auxiliary heating)

A solar heating system can be constructed using the solar loop which comprises a combination of solar collectors, pumps, water tanks and water heaters. [Solar collectors](#) are connected on the demand side of the plant loop. Multiple collector modules can be combined in series and parallel. The supply side of the solar plant loop contains a water heater, circulation pump and setpoint manager to control the loop setpoint. A solar loop can be connected directly to domestic hot water (DHW) loop outlets by first deleting the water heater from the supply side of the [DHW plant loop](#) and then connecting the solar water heater to the DHW supply side splitter and mixer:



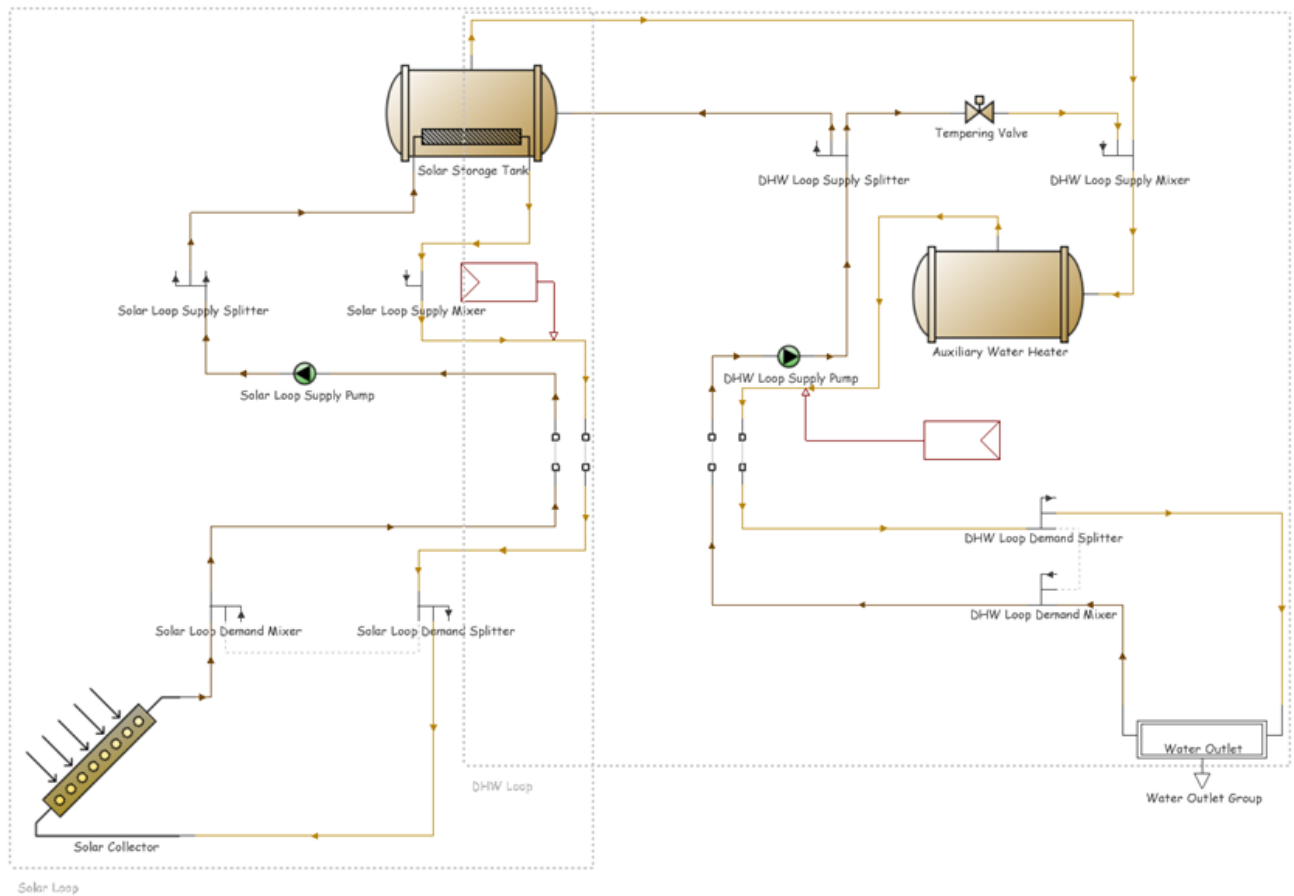
Solar and Auxiliary Heating System

To incorporate auxiliary heating it is necessary to model a two-tank system. The storage tank gathers heat directly from the solar collectors and stores it for later use. The auxiliary water heater is positioned downstream of the storage tank on the supply side of the DHW plant loop. The auxiliary water heater, or booster water heater, provides additional heat if the storage tank water is not hot enough. The auxiliary water heater can be modelled as an instantaneous/tankless water heater or as a standard tanked water heater with heating source (see [Water Heater component](#)):



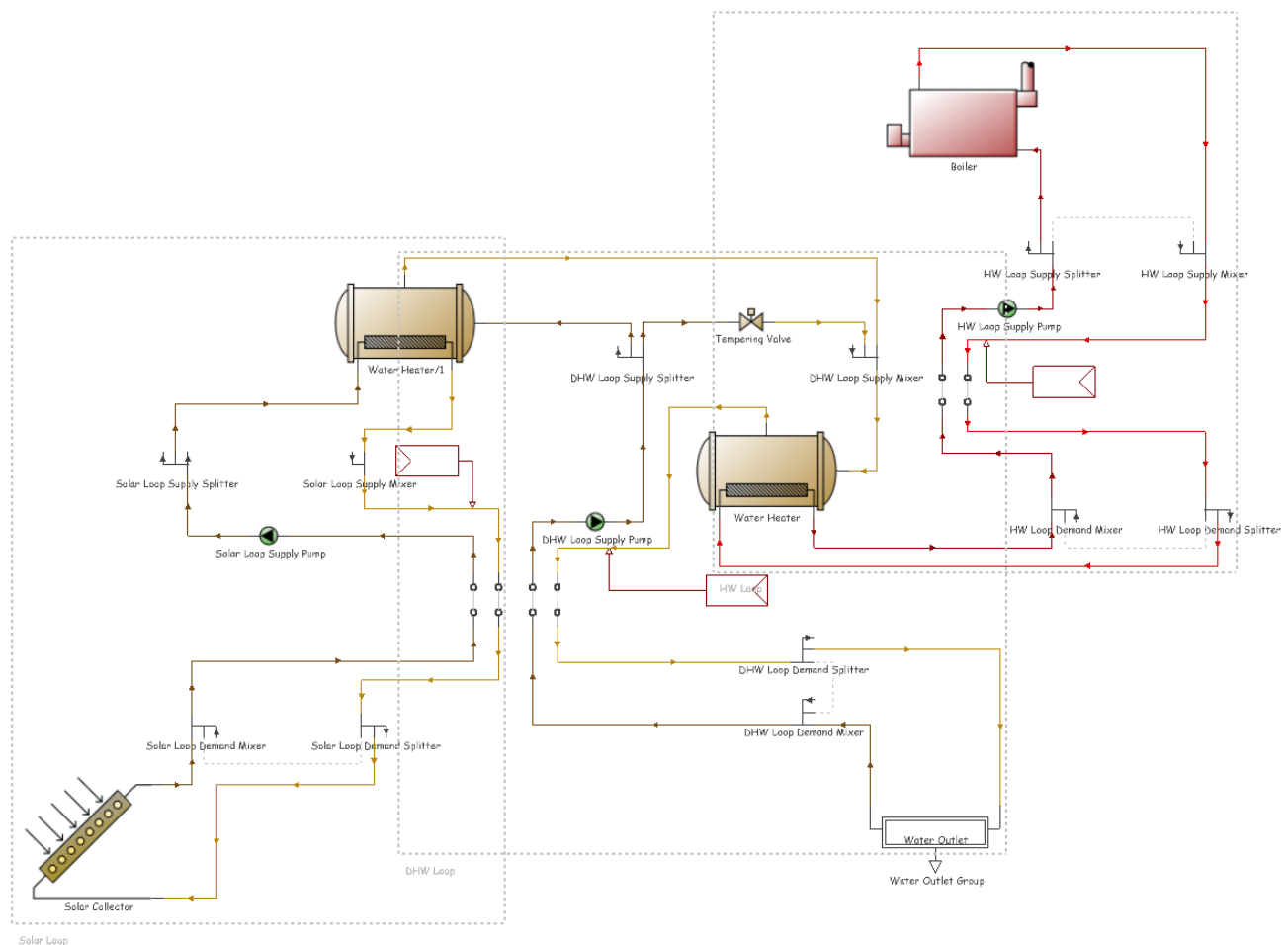
High-temperature Solar and Auxiliary Heating System

Another strategy to consider for solar heating systems is to allow the storage tank to reach a much higher temperature than necessary for the end use. This allows the tank to store more energy from the solar collectors, when it is available. However, for applications such as domestic hot water, it is undesirable and unsafe to supply excessive hot water temperatures at the point of demand. To take advantage of higher storage temperatures, yet still avoid scalding temperatures at the tap, the hot water leaving the storage tank can be tempered with cold water using a three-way valve ([tempering valve](#)) to achieve the target temperature. An additional auxiliary DHW heating loop is provided by DesignBuilder HVAC which incorporates this arrangement. A solar loop water heater can be directly connected to the supply side splitter and mixer of an auxiliary heating loop:



Boiler-assisted High-temperature Solar Heating System

Another common solar hot water loop arrangement similar to above but this time with auxiliary heating from a hot water loop with [boiler](#):



This arrangement of components and loops models a typical solar heating system with 2 tanks: a solar tank with water at potentially high temperatures and an auxiliary tank containing water at temperatures that can be used by heating and/or DHW purposes. The **Solar Assisted DHW** HVAC template provides this functionality.

More information on how to configure and control Solar and Auxiliary heating loops can be found on the [Solar Hot Water](#) help page.

Why is the solar collector on the *demand* side of the loop?

The demand is the loop demand rather than a component demand, i.e. the demand is created by the need to satisfy the setpoint on the solar loop supply sub-loop outlet. In other words, the more the outlet temperature from the water heater element drops below the setpoint, the higher the demand on the collectors.

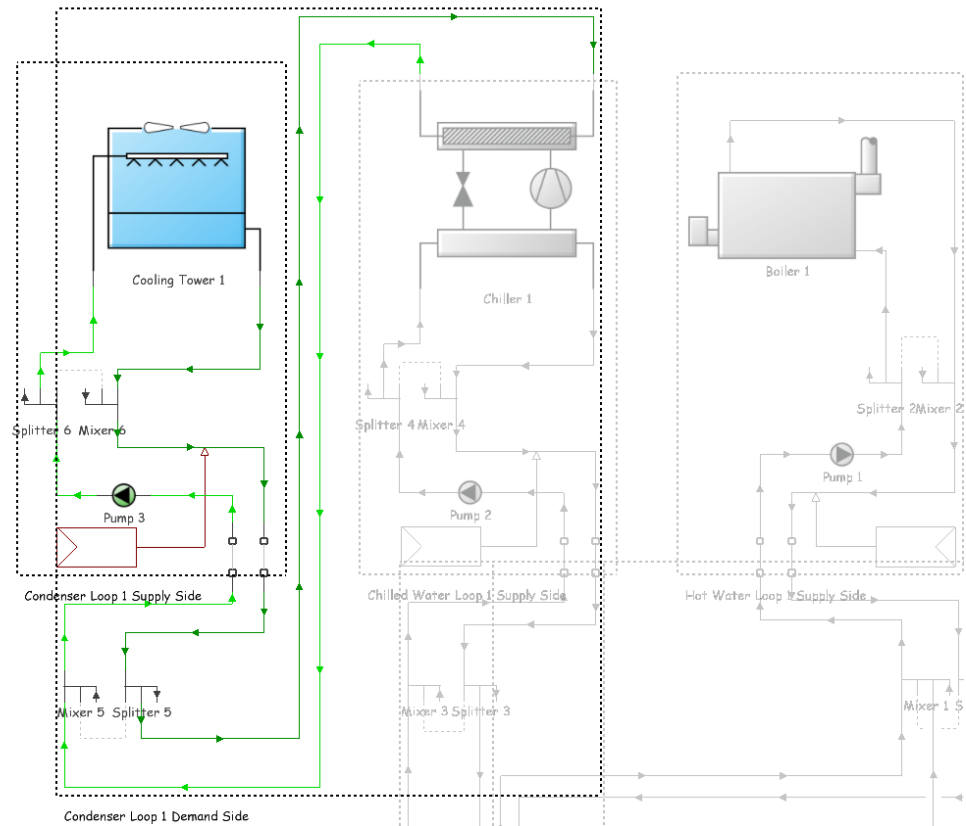
See also:

- "Solar Loop Data" on page 736

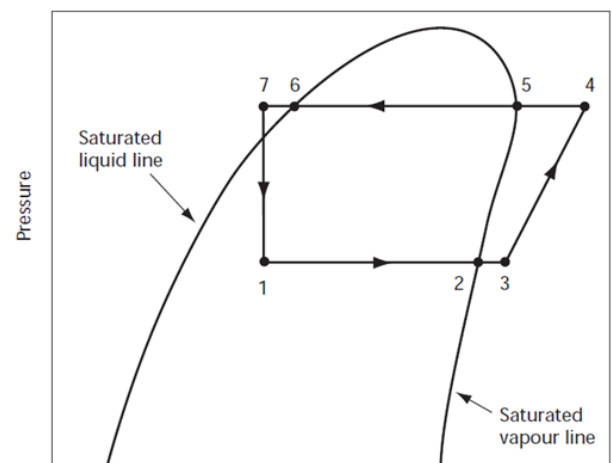
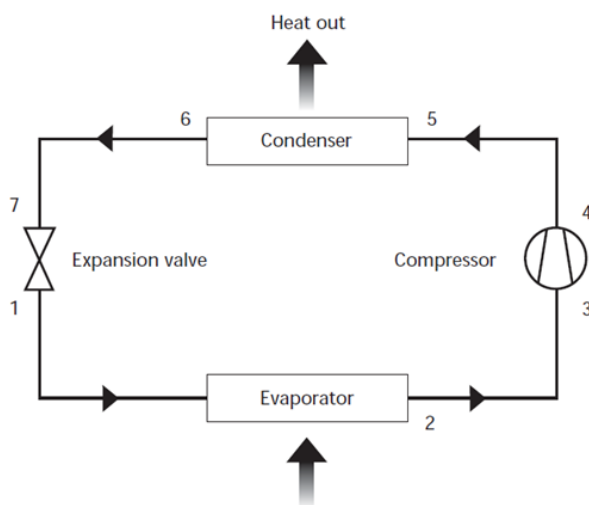
Condenser Loop

The condenser supply sub-loop incorporates one or more [cooling towers](#) connected in series or parallel, flow splitter and flow mixer together with a circulating pump. The default sub-loop also includes a scheduled temperature [setpoint manager](#) which is pre-defined to control the off-plant temperature at a constant 29°C.

The demand sub-loop incorporates a flow splitter and flow mixer in order that the loop can be connected to condenser components such as a [water cooled chiller](#).



The graph below (reproduced with permission from CIBSE) illustrates the vapour compression cycle. The condenser process takes the fluid from saturated vapour (point 5) to saturated liquid (point 6) on the graph below.



CIBSE Guide B Figures 4.8 and 4.9

See the [Condenser loops cooling tower](#) tutorial

See also:

- "Plant and Condenser Loop Data" on page 731

VRF Loops

A VRF system is an air-conditioning system that varies the refrigerant flow rate using variable speed compressor(s) in the outdoor unit, and the electronic expansion valves (EEVs) located in each indoor unit. The system meets the space cooling or heating load requirements by maintaining the zone air temperature at the setpoint. The ability to control the refrigerant mass flow rate according to the cooling and/or heating load enables the use of as many as 60 or more indoor units with differing capacities in conjunction with one single outdoor unit. This unlocks the possibility of having individualized comfort control, simultaneous heating and cooling in different zones, and heat recovery from one zone to another. It may also lead to more efficient operations during part-load conditions.

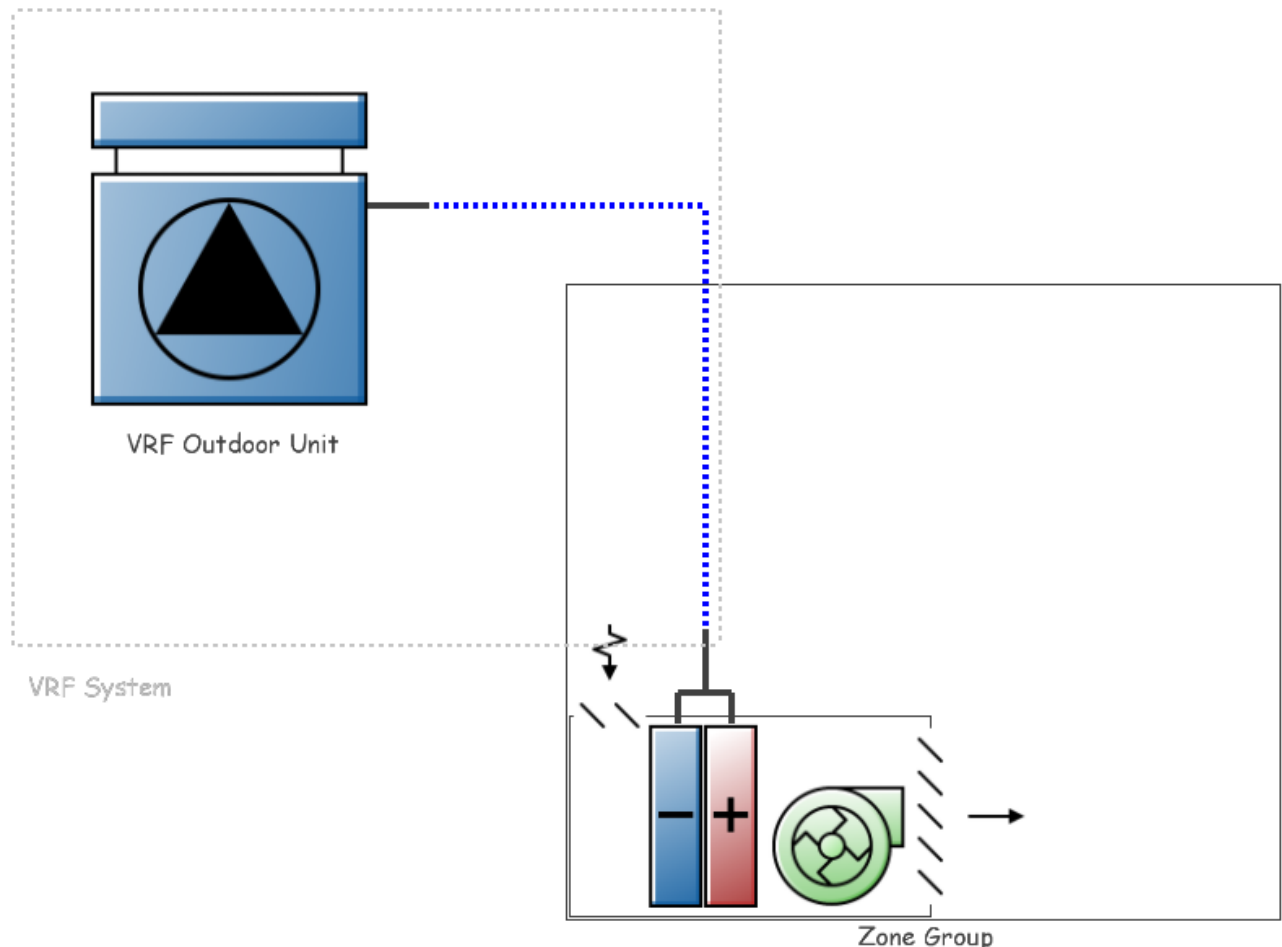
There are two alternative VRF models in EnergyPlus:

- The system provided in DesignBuilder is the system curve based model (VRF-SysCurve). In this model, a number of system level curves are used to describe the VRF system performance. This model corresponds to the AirConditioner:VariableRefrigerantFlow object.
- Physics based model (VRF-FluidTCtrl) has been recently introduced into EnergyPlus. It is able to consider the dynamics of more operational parameters and is applicable for fluid temperature control. This model corresponds to the AirConditioner:VariableRefrigerantFlow:FluidTemperatureControl object.

Refer to the [Engineering reference](#) for more technical details of the two models.

VRF indoor units are placed within HVAC zones like other [zone forced air units](#). They include pre-configured direct-expansion cooling and/or heating coils.

VRF loops are not like other loops in DesignBuilder HVAC as they consist simply of a single [VRF outdoor unit](#) connected to one or more [VRF indoor units](#) through a dotted line.



There is no data associated with the VRF loop itself - it mainly serves as a container for the VRF outdoor unit and the connections to the indoor units.

Loop Data

There are 3 plant and air loop dialogs:

- "Air Loop Data" on page 727
- Editing DOAS Air Loop
- "Plant and Condenser Loop Data" on page 731
- "Solar Loop Data" on page 736

Air Loop Data

The air loop edit dialog allows you to control the way that the air system is sized and in particular in how the design supply and outside air flow rates as well as heating and cooling capacities are calculated.

See the [Adding air loops and AHUs](#) and [Editing Generic air loop and system components](#) tutorials

General

Name

This is the name of the loop. If the supplied loop name is found to already exist, the software will automatically append an integer to create a unique name.

Design return air flow fraction of supply air flow

This setting specifies the design air loop return air flow rate as a fraction of the supply flow rate when there is no exhaust flow. It can be used to set zero return air flow for a DOAS system or to model a pressurized system where the return flow is a fraction of the supply flow. The return air flow rate will never be greater than the current supply air flow rate multiplied by this fraction. It may be less if there is unbalanced exhaust from any zones served by this air loop. The default is 1.0, the minimum allowed value is 0 and the maximum is 1.

Sizing

The rest of the Air loop dialog provides the sizing inputs needed to perform a central forced air system design air flow, heating capacity, and cooling capacity calculation for a system serving one or more zones. The information needed consists of the outside environmental conditions and the design supply air temperatures, outdoor air flow rate, and minimum system air flow ratio.

The outside conditions come from the design days in the input. A **system sizing** calculation is performed for the design day defined in the input file and the resulting maximum heating and cooling air flow rates and capacities are used in the **component sizing** calculations.

Supply air conditions are specified by inputting a supply air temperature for cooling, a supply air temperature for heating, and a preheat temperature. The system sizing calculation sums the zone design air flow rates to obtain a system supply air flow rate. The design conditions and the outdoor air flow rate are used to calculate a design mixed air temperature. The temperature plus the design supply air temperatures allows the calculation of system design heating and cooling capacities.

Note: Detailed HVAC [autosizing calculations](#) are not to be confused with [Heating](#) and [Cooling design](#) calculations, which use separate simpler data entry from the [HVAC tab](#). No Detailed HVAC data is used for Heating and Cooling design calculations, which are based on the EnergyPlus Ideal loads system.

Design outdoor air flow rate

The design outdoor air flow rate (in m3/s or ft3/min). Generally this should be the minimum outdoor air flow. It is used for both heating and cooling design calculations. The assumption for cooling is that any outdoor air economizer will be closed. If **Autosize** is input the Design outdoor air flow rate used in the simulations will be taken from the sum of the zone outdoor air flow rates based on the [Sizing option](#) below.

Note that the Design supply air flow rate is defined in the AHU dialog contained within the Air loop.

Central heating maximum system air flow ratio

This ratio is used in calculating the central system heating capacity and is only required when the [Fan type](#) of the attached AHU is **2-Variable volume**. It should be set so as to define the expected system flow rate when maximum heating demand occurs. Thus if the system is VAV with the zone VAV dampers held at minimum flow when there is a zone heating demand, this ratio should be set to the minimum flow ratio. If the zone VAV dampers are reverse action and can open to full flow to meet heating demand, this ratio should be set to 1.

The value entered must be between 0 and 1.

Sizing option

The Sizing option is set to as follows:

- **1-Non-coincident** where the central system air flow rate will be sized on the sum of the non-coincident zone air flow rates. This option is normally used for CAV systems.
- **2-Coincident** where the central system air flow rate will be sized on the sum of the coincident zone air flow rates. This option is normally used for VAV systems,

Type of load to size on

For CAV air loops you can define the way the central system supply air flow rate is sized. The choices are:

- **1-Sensible** means that the central system supply air flow rate will be determined by combining the zone design air flow rates, which have been calculated to meet the zone *sensible* loads from the design days. In addition this option tells the program to size the central cooling coil using entering air flow rate and air conditions at the sensible load peak.
- **2-Ventilation requirement** means that the central system supply air flow rate will be determined by the system ventilation requirement.
- **3-Total** means that the central system supply air flow rate will be determined by combining the zone design air flow rates, which have been calculated to meet the zone *total sensible + latent* loads from the design days. In addition this option indicates that the program should size the central cooling coil at the air flow rate and conditions at the total load peak.

The central heating coil is always sized at the conditions at the peak sensible heating load.

For VAV systems you should select either **1-Sensible** or **3-Total**.

Important Note: For CAV air loops used to model a dedicated outdoor air system (DOAS), you must set the **Type of load to size on** to **2-Ventilation requirement**. This is true even if the air is being pre-heated and/or pre-cooled to temper it before delivery.

System outdoor air method

The method used to calculate the system minimum outdoor air flow. The two choices are:

- **1-Zone sum** which sums the outdoor air flows across all zones served by the system.
- **2-Standard 62.1 ventilation rate procedure** (VRP) which uses the multi-zone equations defined in 62.1 to calculate the system outdoor air flow. VRP may also adjust autosized air terminal maximum and minimum supply flow rates if needed to ensure adequate outdoor air flow rate to each zone. VRP considers zone air distribution effectiveness and zone diversification of outdoor air fractions.
- **3-Standard 62.1 simplified procedure** (SP) is similar to VRP and was introduced in ASHRAE Standard 62.1-2019. The main difference compared to VRP is that it uses a simplified approach to calculate a system's ventilation efficiency which is in turn used to calculate the system outdoor air flow. Additionally, when set to autosize, the minimum (primary) air flow (or air flow fraction) of these types of air terminals will be calculated following SP: "Variable Air Volume with No Reheat ADU" on page 1121 (Constant minimum Air flow fraction or Fixed minimum air flow fraction), "Variable Air Volume with Reheat ADU" on page 1116 (Constant minimum air flow fraction or Fixed minimum air flow fraction), "Series Powered Induction Unit with Reheat ADU" on page 1123 (Minimum primary air flow fraction), "Parallel Powered Induction Unit with Reheat ADU" on page 1124 (Minimum primary air flow fraction).

Note: When the **Type of load to size on** is set to **2-Ventilation requirement** the **System outdoor air method** control is set to **1-Zone sum** and the control is greyed out.

Zone maximum outdoor air fraction

For an air loop, when a zone requires outdoor air higher than the user specified **Zone maximum outdoor air fraction**, the zone supply air flow will be capped at the maximum value. This allows the system level outdoor air flow to be reduced while the total supply air flow increases. Valid values are from 0 to 1.0. Default is 1.0 which indicates zones can have 100% outdoor air maintaining backward compatibility. This inputs work for constant volume air systems, single and dual duct VAV systems.

Heating

Preheat design temperature

This is the design air temperature exiting the preheat coil (°C or °F).

Preheat design humidity ratio

This is the design humidity ratio exiting the preheat coil (in kg/kg).

Central heating design supply air temperature

This is the design supply air temperature for heating (°C or °F). It can be either the reset temperature for a single duct system or the actual hot duct supply air temperature for dual duct systems. It should be the temperature at the exit of the main heating coil.

The default value for newly added air loops is 16°C corresponding to a reheat-type system where the zone temperature control is provided through local reheat terminal units.

Note: For a centralised system where the AHU is to provide zone heating, you should increase this value to say 35-50°C in order for the heating coil sizing to be performed correctly.

100% Outdoor air in heating

This data is set as follows:

- **1-No** for systems with recirculation set in the AHU, the system will be sized for heating using minimum outside air.
- **2-Yes** for systems without recirculation set in the AHU, the system will be sized for heating using 100% outdoor air.

Central heating design supply air humidity ratio

This is the design humidity ratio at the exit of the central heating coil (in kg/kg) . The default is 0.008.

Heating design air flow method

The input must be either:

- **1-Design day** which means the program will calculate the system design heating air flow rate using the System Sizing input data and a design day simulation.
- **2-Flow/System** which means that the program will use the input of the field [Heating Design Air Flow Rate](#) as the system design heating air flow rate.

The default method is **1-Design day**, i.e., the program uses the calculated design values.

Heating design air flow rate

This is the design system heating air flow rate (m³/s or ft³/min). This input is an alternative to using the program calculated value. This input is used if [Heating design air flow method](#) is **2-Flow/System**. This value will not be multiplied by any sizing factor or by zone multipliers. If using zone multipliers, this value must be large enough to serve the multiplied zones.

Cooling

Pre-cool design temperature

This is the design air temperature exiting the pre-cooling coil (°C or °F).

Pre-cool design humidity ratio

This is the design humidity ratio exiting the pre-cooling coil (kg/kg).

Central cooling design supply air temperature

This is the design supply air temperature for cooling (in °C or °F). This should be the temperature of the air exiting the central cooling coil.

This value should typically be lower than the [Central heating design supply air temperature](#).

100% Outdoor air in cooling

This data is set as follows:

- **1-No** for systems with recirculation set in the AHU, the system will be sized for cooling using minimum outside air
- **2-Yes** for systems without recirculation set in the AHU, the system will be sized for cooling using 100% outdoor air.

Central cooling design supply air humidity ratio

This is the design humidity ratio at the exit of the central cooling coil (in kg/kg). The default is 0.008.

Cooling design air flow method

The input must be either:

- **1-Design Day** which means that EnergyPlus will calculate the system design cooling air flow rate using the System Sizing input data and a design day simulation.
- **2-Flow/System** where the [Cooling design air flow rate](#) below is used as the system design cooling air flow rate.

The default method is **1-Design Day**, i.e. the program uses the calculated design values.

Cooling design air flow rate

This is the design system cooling air flow rate (m³/s or ft³/min). This input is an alternative to using the program calculated value. This input is used if [Cooling design air flow method](#) is **2-Flow/System**. This value will not be multiplied by any sizing factor or by zone multipliers. If using zone multipliers, this value must be large enough to serve the multiplied zones.

Availability schedule

Schedule values greater than zero (usually 1 is used) indicate that the system is on. Schedule values less than or equal to zero (usually 0 is used) denote that the system is off. This schedule overrides the air loop fan schedules for determining whether the fans are on.

DOAS Loop Data

The DOAS loop edit dialog allows you to configure the preconditioning settings for the DOAS system.

General

Name

This is the name of the DOAS loop. If the supplied loop name is found to already exist, the software will automatically append an integer to create a unique name.

Heating

Preheat design temperature

The design air temperature exiting the preheat coil (if any) (in °C or °F).

Preheat design humidity ratio

The design humidity ratio exiting the preheat coil (if any) (in kgWater/kgDryAir or lbWater/lbDryAir).

Cooling

Precool design temperature

The design air temperature exiting the precool coil (if any) (in °C or °F).

Precool design humidity ratio

The design humidity ratio exiting the precool coil (if any) (in kgWater/kgDryAir or lbWater/lbDryAir).

Plant And Condenser Loop Data

The plant and condenser loop edit dialog incorporates two tabs, the first tab is General and incorporates loop characteristic and sizing data, the second tab covers [Plant Equipment Operation](#) allowing you to set up various equipment operation schemes for the loop.

See the [Adding loops and connecting components](#), [Editing HVAC loop and component properties](#) and the [Condenser loops cooling tower](#) tutorials

General

Name

If the supplied loop name is found to already exist, the software will automatically append an integer to create a unique name.

Fluid type

You can define the type of fluid used in the loop as a mixture of water and glycol along with the particular concentration being used. EnergyPlus will accurately model the thermal properties of the fluid based on glycol concentration and fluid temperature at each time step of the simulation.

Select the fluid type from 3 options:

- **1-Water.**
- **2-EthyleneGlycol.**
- **3-PropyleneGlycol.**

Glycol concentration

When one of the glycol fluid types is selected above then you can enter the concentration of glycol in the loop fluid here. It should be entered as a decimal fraction between 0.0 and 1.0. A concentration of 0.0 refers to pure water. A concentration of 1.0 would denote 100% of the fluid referenced in the Fluid type input above.

Plant loop volume

This is the volume of the plant loop (in m³ or gal). This is the loop volume for the entire loop, i.e. both the demand side and the supply side. This is used for the loop capacitance calculation which updates the loop temperature from one time-step to the next and introduces some time delay between the supply and demand sides of the loop. The loop volume could be calculated from pipe size data but this is not usually known. If zero volume is specified the demand inlet temperature is set to the supply outlet temperature at the previous system time-step. If a very large capacitance is specified unrealistic time delay may result and there may be poor response to changes in loop setpoint temperature. If the loop volume is auto-calculated, the EnergyPlus provided calculation gives reasonable values for most system sizes. This calculation is described in the [EnergyPlus Engineering Reference](#) manual.

Flow Type

Plant loop flow type

Two options are available:

- **1-Constant flow**, for constant flow plant loops. In this case a [constant speed pump](#) will automatically be specified.
- **2-Variable flow** for variable flow plant loops. In this case a [variable speed pump](#) will automatically be specified.

The flow type is fixed as **1-Constant flow** for DHW and condenser loops

Temperatures

Maximum loop temperature

This is the maximum allowable temperature (in °C or °F) for this loop.

Minimum loop temperature

This is the minimum allowable temperature (in °C or °F) for this loop.

Flow Rates

Maximum loop flow rate

This is the maximum loop flow rate (in m³/sec or ft³/min). This parameter is also used when the user chooses to auto-calculate the loop volume. See the [Plant loop volume](#) above and the [EnergyPlus Engineering Manual](#) for more details.

Note: This value is generally autosizable but only when the condenser loop does not contain any Vertical ground heat exchanger components. In cases where a vertical ground heat exchanger is included the **Maximum loop flow rate** should normally take the same value as entered on the ground heat exchanger dialog for [Maximum flow rate](#) and [Design flow rate](#). The Condenser loop **Maximum loop flow rate** can be entered as higher than the ground heat exchanger flow rate but in this case it will be limited to that value during the simulation.

Minimum loop flow rate

This is the minimum loop flow rate (in m³/sec or ft³/min).

Load distribution scheme

The Load distribution scheme selects the algorithm used to sequence equipment operation in order to meet the plant loop demand. Currently, 3 schemes are functional:

- **1-Sequential** uses each piece of equipment to its maximum part load ratio and will operate the last required piece of equipment between its minimum and maximum part load ratio in order to meet the loop demand.
- **2-Optimal** operates each piece of equipment at its optimal part load ratio and will operate the last component between its minimum and maximum part load ratio in order to meet the loop demand.
- **3-Uniform** evenly distributes the loop demand amongst all available components on the equipment list for a given load range.

Plant loop demand calculation scheme

The **Plant loop demand calculation scheme** determines the amount of heating or cooling necessary to bring the temperature of the Plant loop to its setpoint. When this value is determined then the **Load distribution scheme** explained in the previous section takes this value and distributes the load to the appropriate equipment. The demand calculation scheme determines how the load is calculated. See more information in the EnergyPlus Engineering Document. There are 2 plant loop demand calculation schemes:

- **1-SingleSetPoint** scheme in which the Plant loop requires that a [Setpoint manager](#) sets a single setpoint value. Examples of this setpoint manager are [1-Scheduled](#), SetpointManager:OutsideAir etc.
- **2-DualSetPoint** scheme in which the Plant loop requires that a [17-Dual setpoint](#) SPM sets the high and low setpoint values for heating and cooling to implement a mixed water loop.

Sizing

Design loop exit temperature

The water temperature (in °C or °F) at the exit of the supply side of the plant loop, Thus this is the temperature of the water supplied to the inlet of chilled or hot water coils in the case of plant loops or chiller condensers in the case of condenser loops.

Loop design temperature difference

This is the design temperature rise (for cooling or condenser loops) or fall (for heating loops) (in °C or °F) across the demand side of a plant or condenser loop.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the Plant loop is available during the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the device can be on for the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the device must be off.

Outside temperature operation

If you select this option you can define the external temperatures below and above which, the plant or condenser loop will turn off as a deadband. The definition is different depending on the type of plant loop as shown below.

Loop type	External cut-in temperature	External cut-out temperature
Hot water plant loops	The temperature (in °C or °F) below which the plant is on.	The temperature (in °C or °F) above which the plant is forced off.
Chilled water and condenser plant loops	The temperature (in °C or °F) is the temperature above which the plant is on.	The temperature(in °C or °F) below which the plant is forced off.

Note: When either of the cut-in or cut-out conditions occurs they override the availability schedule. For example in the case of a hot water loop, when the **Outside temperature operation** check box is selected, if the outside air dry-bulb temperature is below the **External cut-in temperature** then the loop is cycled on (it is available) or if it is above the **External cut-out** temperature then the loop availability is forced off (not available), otherwise the loop availability is controlled by the **Availability schedule** alone.

Plant Equipment Operation

Plant and condenser loops have a mechanism for controlling the operation of the loop to allow control over which equipment is available under what conditions. Since there may be multiple control schemes that are assigned various priorities associated with each loop, an overall operation scheme must be defined. Each operation scheme must have the type of operation scheme together with the name of the scheme and the schedule that defines its availability.

Note: The order in which the individual operation schemes appear in this list defines its priority relative to the others with the same scheduled availability. The first scheme appearing in the list is given the highest priority; the second scheme has second highest priority, etc. In other words, if according to its schedule, the first operation scheme is available, then it is used by the simulation to define how the plant or condenser loop operates. If it is not available, the second operation scheme in the list is checked to see if it is available until the highest priority scheme that is also available is found. To allow for simultaneous heating and cooling, there is one configuration where multiple operation schemes are available at a given timestep. This

is allowed if a user inputs both a heating range based operation and a cooling range based operation with overlapping schedules. The simulation will look at the current loop demand to determine if the cooling scheme should be used or the heating scheme. In other cases, if two schemes overlap (are available at the same time), EnergyPlus will raise an error and the simulation will be aborted.

Number of schemes

This is the number of operational schemes that are to be associated with the plant or condenser loop. A maximum of ten schemes are allowed for each loop. After setting the required number of schemes, the dialog will be automatically populated with sections for each scheme, the settings for each operation scheme being described below.

Scheme

Scheme operation type

This is the type of control scheme used. The available options will depend on the current loop type. For condenser and chilled water plant loops, the options are:

- **1-Uncontrolled** and
- **2-Cooling load**

for heating and domestic hot water plant loops, the options are:

- **1-Uncontrolled** and
- **3-Heating load**.

for chilled water loops with [Ice thermal storage](#) components included, 2 further Scheme operation types are allowed:

- **4-Component Setpoint**, for use only with Ice thermal storage systems, this option requires various setpoint managers, temperature schedules, etc. to allow the system to operate. As complex definitions are possible for this scheme, it allows the possibility of more accurate and flexible modelling, for instance, setting hourly variation of setpoint temperatures at the outlet of each piece of equipment. The screenshot in the [Controls and Chilled Water Plant Loop Operation Scheme Settings](#) section of the Ice thermal storage help topic shows how to use Setpoint managers to control an upstream chiller with ice thermal storage series configuration with the scheme operation type set to **4-Component Setpoint**.
- **5-Energy Thermal Storage**, for use only with Ice thermal storage systems, this scheme operation type internally creates the setpoint managers required by the equipment and so allows a simple and convenient approach by eliminating the need for both setpoint manager and schedule for each piece of equipment. The only data required is a charging setpoint temperature and a discharging setpoint temperature. The screenshot in the [Controls and Chilled Water Plant Loop Operation Scheme Settings](#) section of the Ice thermal storage help topic shows how to control a downstream chiller with ice thermal storage series configuration with the scheme operation type set to **5-Energy Thermal Storage**.

Uncontrolled loop operation simply specifies a group of equipment that runs 'uncontrolled'. If the loop runs, this equipment will run also, unless turned off by the loop flow resolver to maintain continuity in the fluid loop.

The **2-Cooling load** and **3-Heating load** options allow different load ranges to be defined and which items of equipment will operate for each range. If either of these options is selected, at least one operational range must be defined. For each range, there is a lower limit for the load range, an upper limit for the load range, and a list of the equipment items that will operate in this range.

Scheme equipment

This setting is only displayed if the [Scheme operation type](#) is set to **1-Uncontrolled**. A list of valid items of plant or condenser equipment is displayed together with check boxes allowing you to define which items of equipment will be turned on for this scheme.

Scheme schedule

This is the schedule for the control scheme. This schedule consists of weeks and days, with the days containing "0 or 1" for each hour of the day. This binary schedule (0 for off, 1 for on) determines if the control scheme is operating for that hour of the day or not.

Number of ranges

This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load** and is used to define the number of ranges required for the heating or cooling load scheme.

Range

Range lower limit

This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load**. This is the lower demand range (W) for the equipment defined for the current range. If demand is below this value, then the equipment will not turn on to meet the demand.

Range upper limit

This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load**. This is the upper demand range (W) for the equipment defined for the current range. If demand is above this value, then the equipment will not turn on to meet the demand.

Range Equipment

A list of valid items of plant or condenser equipment is displayed together with check boxes allowing you to define which items of equipment will be turned on for this range. This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load**.

Note: A priority drop-list associated with each item of equipment allows you to define the order in which equipment will be turned on when the loop demand is within this range.

Solar Loop Data

The Solar loop edit dialog incorporates two tabs, the first tab is General and incorporates loop characteristic and sizing data, the second tab covers [Plant Equipment Operation](#) allowing you to set up various equipment operation schemes for the loop.

General

Name

If the supplied loop name is found to already exist, the software will automatically append an integer to create a unique name.

Fluid type

There is currently only one fluid type available which is **1-Water**.

Plant loop volume

This is the volume of the plant loop (in m³ or gal). This is the loop volume for the entire loop, i.e. both the demand side and the supply side. This is used for the loop capacitance calculation which updates the loop temperature from one time-step to the next and introduces some time delay between the supply and demand

sides of the loop. The loop volume could be calculated from pipe size data but this is not usually known. If zero volume is specified the demand inlet temperature is set to the supply outlet temperature at the previous system time-step. If a very large capacitance is specified unrealistic time delay may result and there may be poor response to changes in loop setpoint temperature. If the loop volume is auto-calculated, the EnergyPlus provided calculation gives reasonable values for most system sizes. This calculation is described in the [EnergyPlus Engineering Reference](#) manual.

Flow Type

Plant loop flow type

Two options are available:

- **1-Constant flow**, for constant flow plant loops. In this case a [constant speed pump](#) will automatically be specified.
- **2-Variable flow** for variable flow plant loops. In this case a [variable speed pump](#) will automatically be specified.

Temperatures

Maximum loop temperature

This is the maximum allowable temperature (in °C or °F) for this loop.

Minimum loop temperature

This is the minimum allowable temperature (in °C or °F) for this loop.

Flow Rates

Maximum loop flow rate

This is the maximum loop flow rate (in m³/sec or ft³/min). This parameter is also used when the user chooses to auto-calculate the loop volume. See the [Plant loop volume](#) above and the [EnergyPlus Engineering Manual](#) for more details.

This value is autosizable.

Minimum loop flow rate

This is the minimum loop flow rate (in m³/sec or ft³/min).

Load distribution scheme

The Load distribution scheme selects the algorithm used to sequence equipment operation in order to meet the plant loop demand. Currently, 3 schemes are functional:

- **1-Sequential** uses each piece of equipment to its maximum part load ratio and will operate the last required piece of equipment between its minimum and maximum part load ratio in order to meet the loop demand.
- **2-Optimal** operates each piece of equipment at its optimal part load ratio and will operate the last component between its minimum and maximum part load ratio in order to meet the loop demand.
- **3-Uniform** evenly distributes the loop demand amongst all available components on the equipment list for a given load range.

Plant loop demand calculation scheme

There is currently only one plant loop demand calculation scheme which is the **1-SingleSetPoint** scheme. In this scheme the Plant loop requires that a [Setpoint manager](#) set a single setpoint value. The **Plant loop demand calculation scheme** determines the amount of heating or cooling necessary to bring the

temperature of the Plant loop to its setpoint. When this value is determined then the [Load distribution scheme](#) explained in the previous section takes this value and distributes the load to the appropriate equipment. The demand calculation scheme determines how the load is calculated. See more information in the EnergyPlus Engineering Document.

Temperature Protection

Low temperature protection

Check this option to apply low temperature protection for the loop. The system will turn on if the temperature at the solar collector water outlet node is below the trigger temperature input defined below.

Temperature below which system is on

Set the temperature below which the system must switch on to protect the loop from damage caused by low temperatures such as freezing. Units are °C or °F. The default value is 0°C.

High temperature protection

Check this option to apply high temperature protection for the loop. The system will turn off if the temperature at the loop water heater DHW outlet node is above the temperature input defined below.

Temperature above which system is off

Set the temperature above which the system must switch off to protect the loop from damage caused by high temperatures. Units are °C or °F. The default value is 90°C.

Differential Thermostat

The aim of the differential thermostat settings is to prevent the water heater from being supplied with inadequately heated water by the solar collector. These settings allow an availability manager to be provided to switch the loop on or off depending on the temperature differential between a collector outlet and the outlet from the water heater. A [differential sensor check box](#) on the solar collector dialog allows you to define the collector to be used by the differential sensor (in cases where there is more than one solar collector on the loop).

Note: only one solar collector on the loop can have its [Differential thermostat hot node sensor](#) checkbox checked.

Differential thermostat

Check this option to use a differential thermostat on the [Solar HW loop](#).

Temperature difference to turn system on

The temperature (in Delta°C or Delta°F) above which the loop can operate.

Temperature difference to turn system off

The temperature (in Delta°C or Delta°F) below which the loop will be shut off.

Sizing

Design loop exit temperature

The water temperature (in °C or °F) at the exit of the supply side of the plant loop, Thus this is the temperature of the water supplied to the inlet of chilled or hot water coils in the case of plant loops or chiller condensers in the case of condenser loops.

Loop design temperature difference

This is the design temperature rise (for cooling or condenser loops) or fall (for heating loops) (in °C or °F) across the demand side of a plant or condenser loop.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the Plant loop is available during the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the device can be on for the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the device must be off.

Plant Equipment Operation

Plants and condenser loops have a mechanism for controlling the operation of the loop to allow control over which equipment is available under what conditions. Since there may be multiple control schemes that are assigned various priorities associated with each loop, an overall operation scheme must be defined. Each operation scheme must have the type of operation scheme together with the name of the scheme and the schedule that defines its availability.

Note: The order in which the individual operation schemes appear in this list defines its priority relative to the others with the same scheduled availability. The first scheme appearing in the list is given the highest priority; the second scheme has second highest priority, etc. In other words, if according to its schedule, the first operation scheme is available, then it is used by the simulation to define how the plant or condenser loop operates. If it is not available, the second operation scheme in the list is checked to see if it is available until the highest priority scheme that is also available is found. To allow for simultaneous heating and cooling, there is one configuration where multiple operation schemes are available at a given timestep. This is allowed if a user inputs both a heating range based operation and a cooling range based operation with overlapping schedules. The simulation will look at the current loop demand to determine if the cooling scheme should be used or the heating scheme. In other cases, if two schemes overlap (are available at the same time), EnergyPlus will raise an error and the simulation will be aborted.

Number of schemes

This is the number of operational schemes that are to be associated with the plant or condenser loop. A maximum of ten schemes are allowed for each loop. After setting the required number of schemes, the dialog will be automatically populated with sections for each scheme, the settings for each operation scheme being described below.

Scheme

Scheme operation type

This is the type of control scheme used. The available options will depend on the current loop type. For condenser and chilled water plant loops, the options are:

- **1-Uncontrolled** and
- **2-Cooling load**

for heating and domestic hot water plant loops, the options are:

- **1-Uncontrolled** and
- **3-Heating load**.

Uncontrolled loop operation simply specifies a group of equipment that runs 'uncontrolled'. If the loop runs, this equipment will run also, unless turned off by the loop flow resolver to maintain continuity in the fluid loop.

The **2-Cooling load** and **3-Heating load** options allow different load ranges to be defined and which items of equipment will operate for each range. If either of these options is selected, at least one operational range must be defined. For each range, there is a lower limit for the load range, an upper limit for the load range, and a list of the equipment items that will operate in this range.

Scheme equipment

This setting is only displayed if the [Scheme operation type](#) is set to **1-Uncontrolled**. A list of valid items of plant or condenser equipment is displayed together with check boxes allowing you to define which items of equipment will be turned on for this scheme.

Scheme schedule

This is the schedule for the control scheme. This schedule consists of weeks and days, with the days containing "0 or 1" for each hour of the day. This binary schedule (0 for off, 1 for on) determines if the control scheme is operating for that hour of the day or not.

Number of ranges

This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load** and is used to define the number of ranges required for the heating or cooling load scheme.

Range

Range lower limit

This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load**. This is the lower demand range (W) for the equipment defined for the current range. If demand is below this value, then the equipment will not turn on to meet the demand.

Range upper limit

This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load**. This is the upper demand range (W) for the equipment defined for the current range. If demand is above this value, then the equipment will not turn on to meet the demand.

Range Equipment

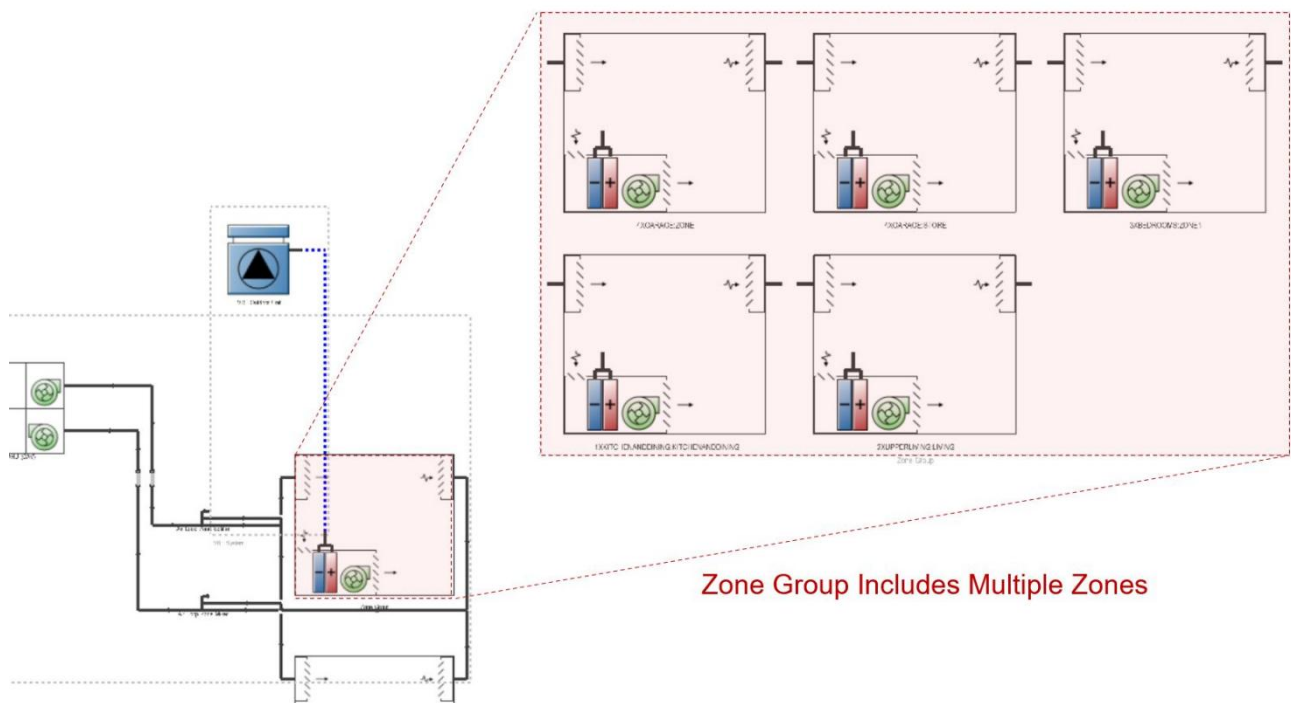
This setting is only displayed if the Scheme Operation Type is set to **2-Cooling load** or **3-Heating load**. A list of valid items of plant or condenser equipment is displayed together with check boxes allowing you to define which items of equipment will be turned on for this range.

Note: A priority drop-list associated with each item of equipment allows you to define the order in which equipment will be turned on when the loop demand is within this range.

HVAC Zones And Zone Groups

HVAC zone groups are used to simplify HVAC layouts by grouping together zones that have identical configurations of HVAC equipment but not necessarily the same equipment ratings. Consequently, only a

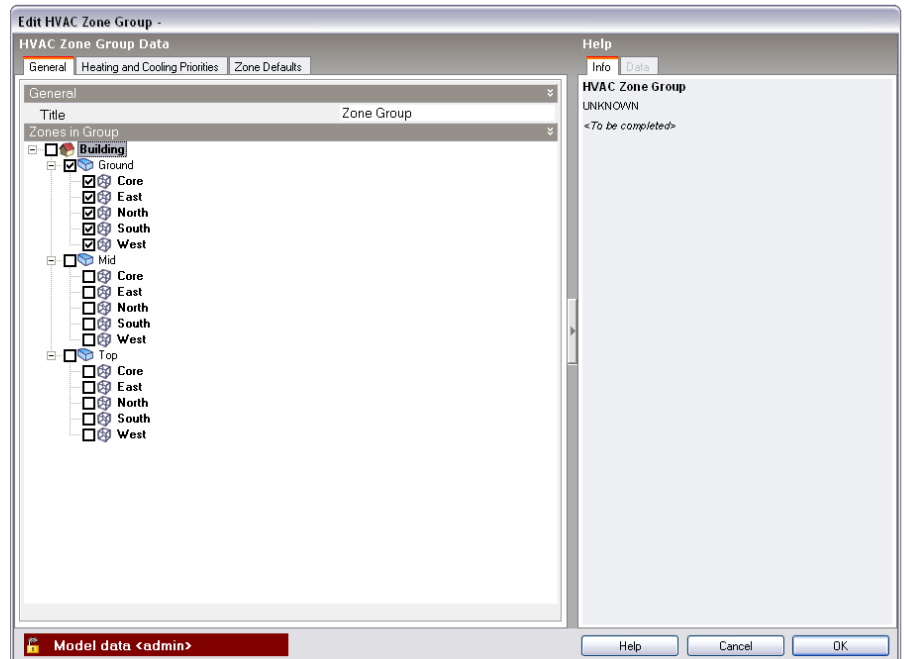
single set of equipment items needs to be added to the group and only one set of connections between air/plant loops and zone group equipment need to be established.



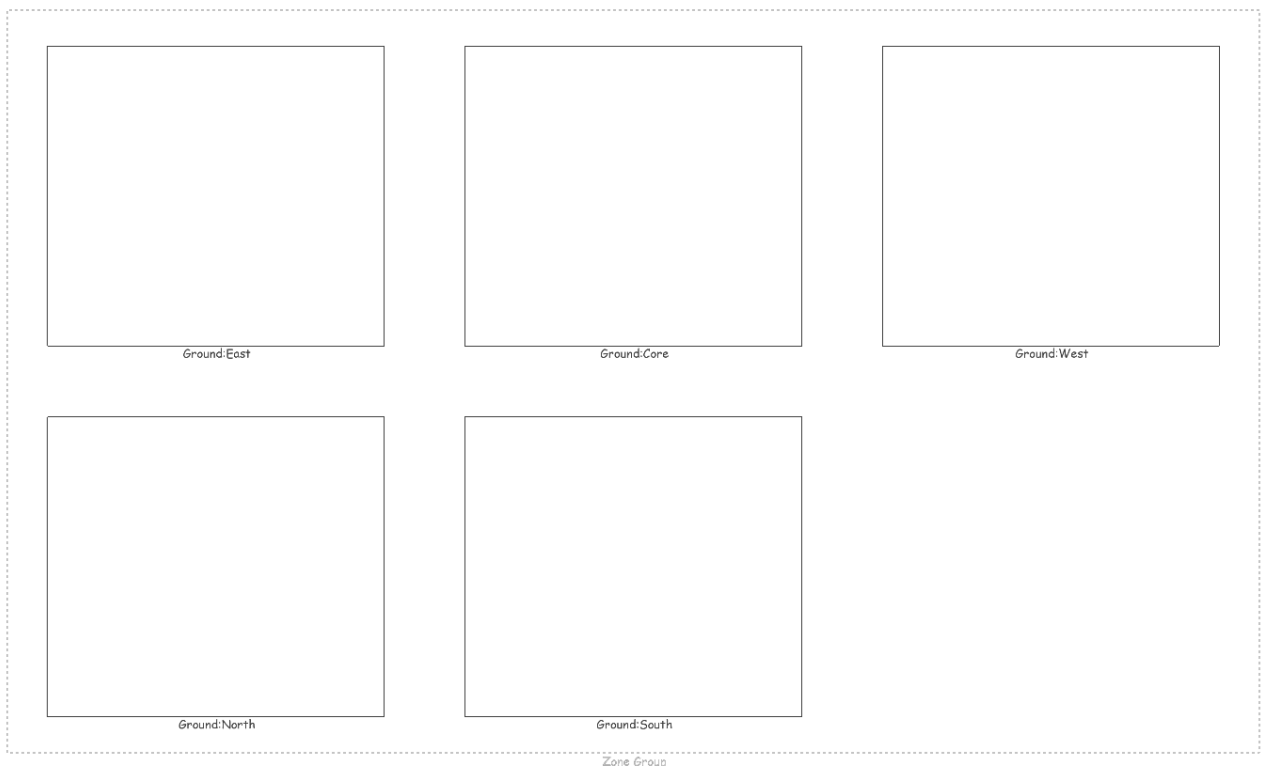
See the [HVAC zone groups and components](#) tutorial

As an example, consider a building model where five ground floor zones are to be cooled and ventilated using a VAV box without reheat supplied by a single AHU and the zones are to be heated using radiators supplied by the same heating circuit. A single zone group can be used to incorporate the equipment for the five zones within the HVAC layout.

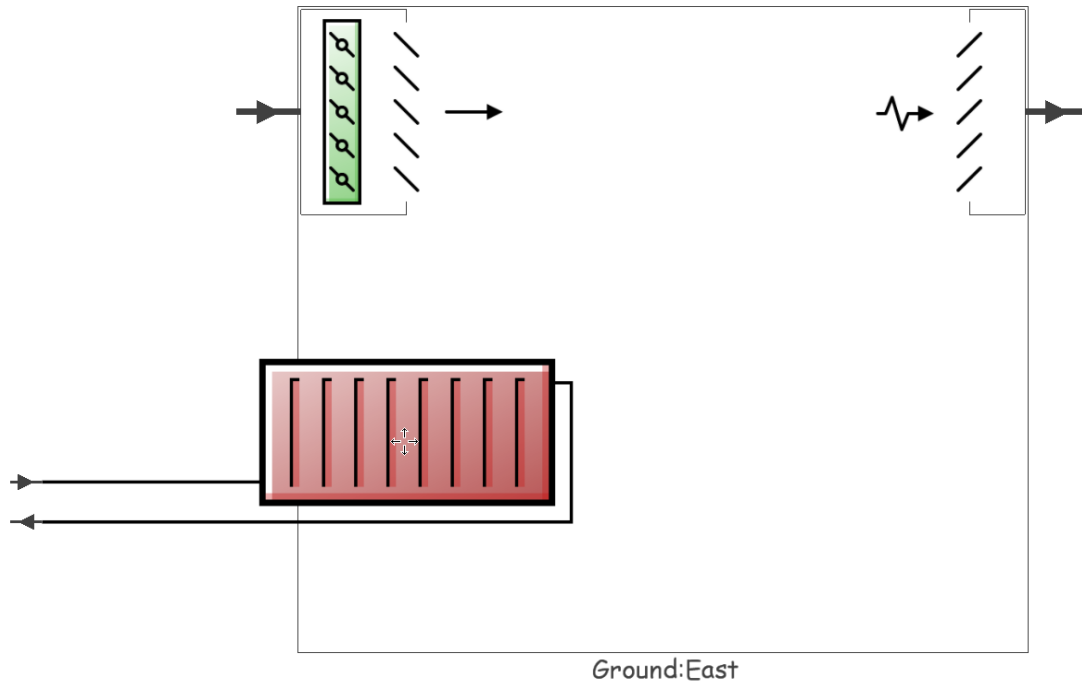
From HVAC System level, a zone group can be [placed](#) and then [edited](#) to include the ground floor zones:



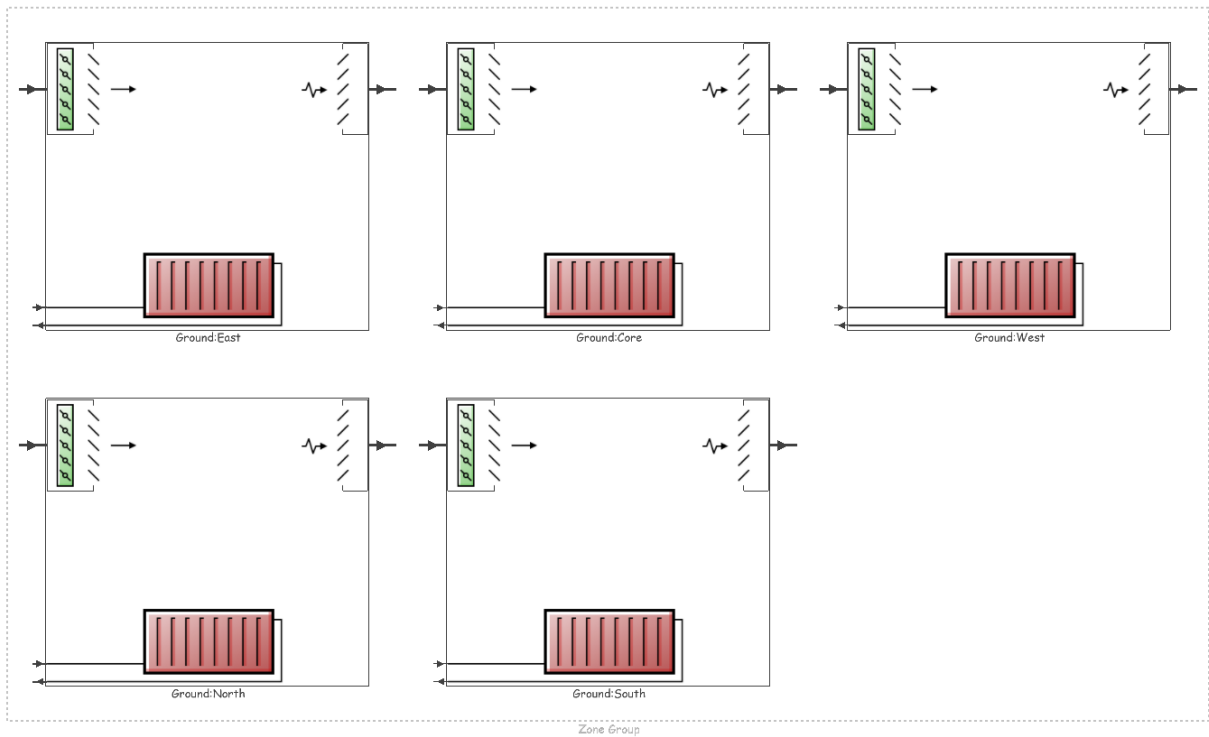
After adding the zones, the zone group becomes active and you can navigate down to it, either by double-clicking on the graphic in the Edit screen or by selecting it from the navigator. At the zone group level, you can see all of the zones added to the group:



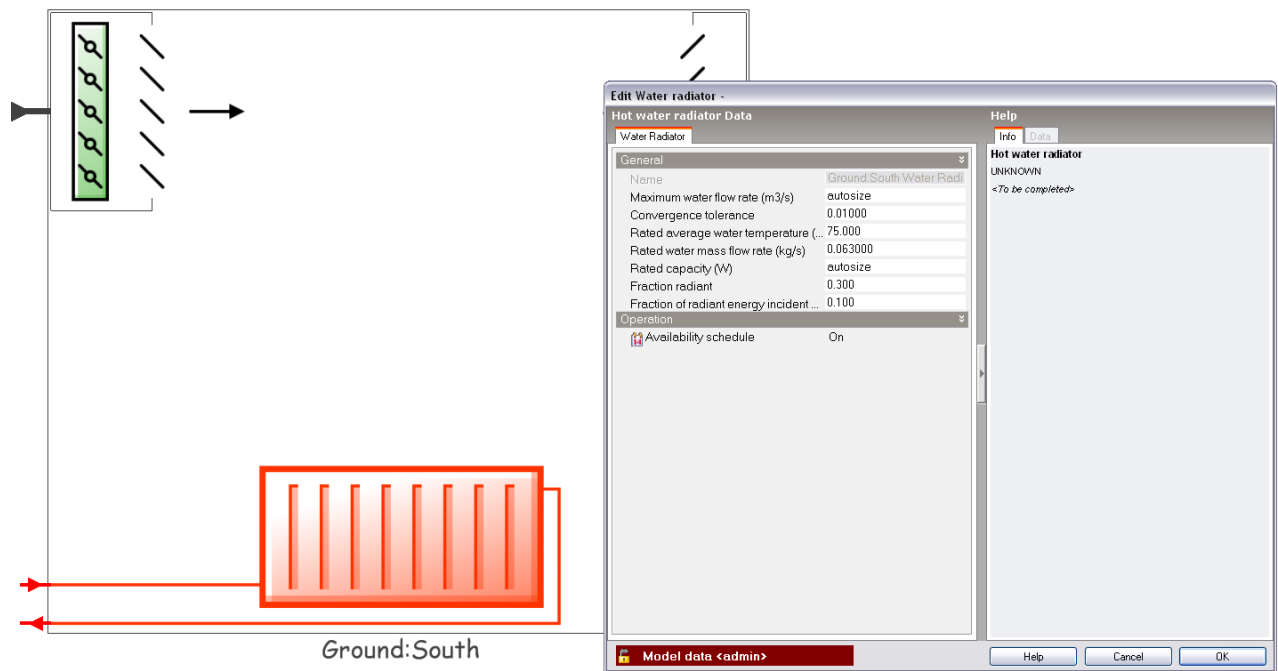
You can then navigate down to any one of the zones to add the required equipment:



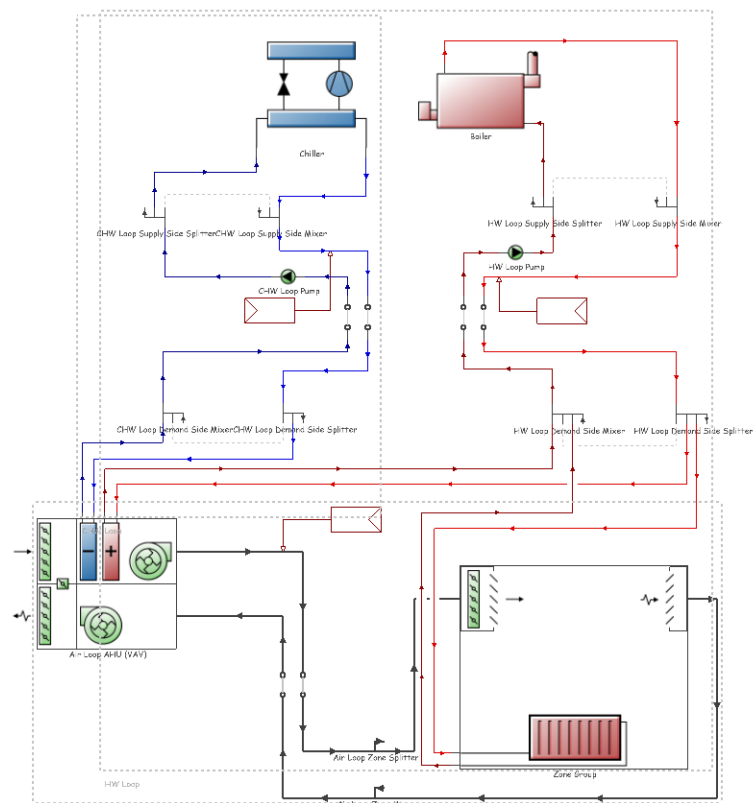
Now when you move back up to the zone group level, you will notice that the zone equipment has automatically been added to all zones in the group:



You can now navigate down to equipment in any zone to change the equipment default settings:



At the system level, only one set of connections is required to connect the equipment in all of the constituent zones up to the air and plant loops:



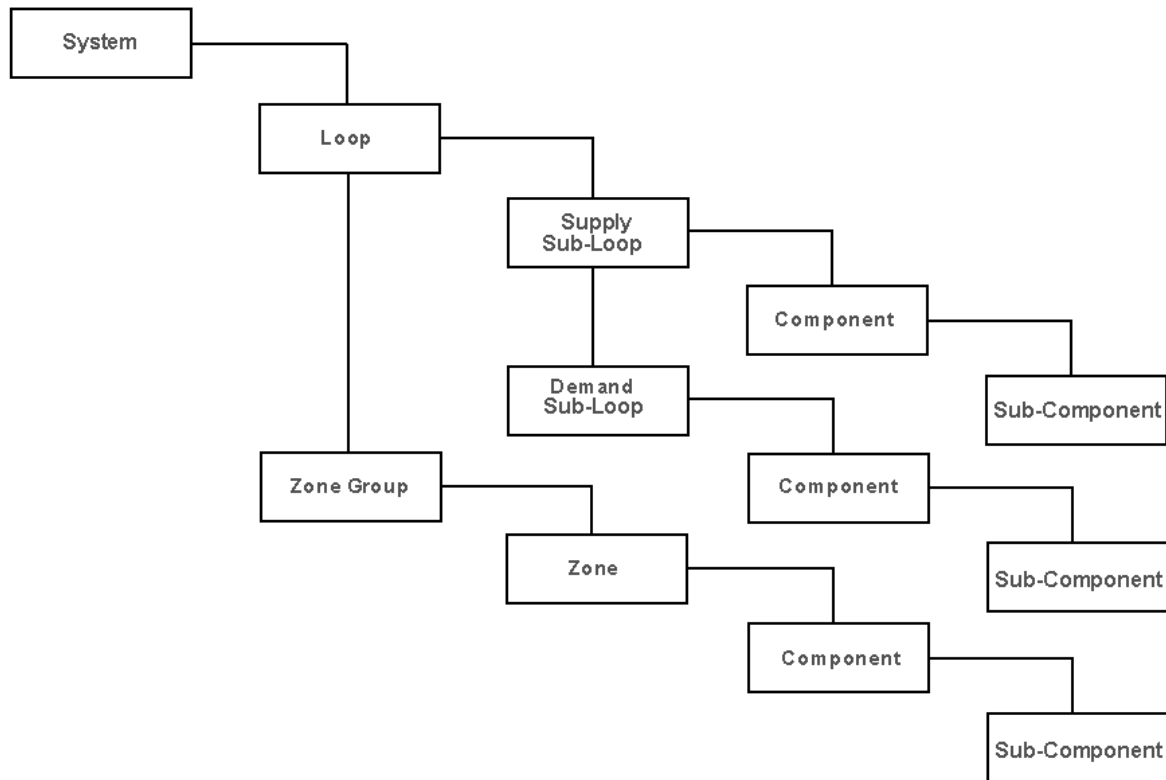
When data is written to the EnergyPlus IDF file prior to simulation, all constituent zones, zone equipment and zone equipment connections are written out separately.

See also:

[Edit HVAC zone data.](#)

HVAC Component Hierarchy And Navigation

HVAC systems are structured using a similar hierarchical system to that used by the DesignBuilder building model:



Methods of navigating the HVAC system are the similar to those used for navigating the building model (see [Navigating Building Models](#) for further information). To move down to an object lower down in the hierarchy, you can either select the object graphically on the Edit screen and double-click on it or alternatively, just select it from the navigator tree list at the left of the screen. To move up a level, you can either press the F2 key, or select the required level object from the Navigator.

After moving down to an object, only that object and any related components in other objects will be active, all other objects in the hierarchy will become inactive and displayed in grey on the Edit screen.

Note: DesignBuilder HVAC does not use the same [hierarchical data inheritance](#) system as the building model. In particular [HVAC zone groups](#) are used to define default data for newly added [HVAC zones](#) but their data is not inherited down to its HVAC zones.

HVAC System Control - Setpoint Managers

The [setpoint manager](#) is the main mechanism through which the behaviour of HVAC system loops is controlled. All HVAC loops require at least one setpoint manager to specify the supply sub-loop outlet temperature and these are automatically included in all pre-defined HVAC loops. Setpoint managers will place a calculated or scheduled setpoint value at the point or connection in the loop at which the setpoint manager control line has been placed. No two setpoint managers should be located on the same connection for like control types otherwise an error message will be displayed and the simulation aborted.

There 15 different types of setpoint manager available. However only certain types of setpoint manager are available for different types of air loops and plant/condenser loops:

- [1-Scheduled](#) (All loops).
- [2-Warmest](#) ([Generic](#) and [dual duct](#) air loops only).
- [3-Coldest](#) ([Generic](#) and [dual duct](#) air loops only).
- [4-Multi-zone minimum humidity average](#) (Air loops only).
- [5-Multi-zone maximum humidity average](#) (Air loops only).
- [6-Multi-zone humidity minimum](#) (Air loops only).
- [7-Multi-zone humidity maximum](#) (Air loops only).
- [8-Single zone humidity minimum](#) (Air loops only).
- [9-Single zone humidity maximum](#) (Air loops only).
- [10-Outdoor air reset](#) (Generic and dual duct air loops, plant and condenser loops).
- [11-Single zone heating](#) (Air loops only).
- [12-Single zone cooling](#) (Air loops only).
- [13-Multi-zone cooling average](#) (Air loops only).
- [14- Multi-zone heating average](#) (Air loops only).
- [15-Follow ground temperature](#) (Condenser loops only)

Refer to the [Setpoint manager](#) section for information regarding specific setpoint manager types.

Setpoint Manager Used With Draw-Through Fans

Adjusting setpoints for fan pickup

When placing Setpoint managers on a node *upstream of a draw-through fan* you may need to manually adjust the setpoint to take into account the temperature rise that will take place as the air flows across the fan.

Mixed air Setpoint Managers

The Mixed Air Setpoint Manager is a special type of setpoint manager which is automatically added to air handling units that incorporate coils located before the supply fan and where no setpoint manager has been added explicitly before the fan. This type of setpoint manager works in conjunction with the AHU outdoor air controller (see Air Handling Unit Outdoor Air System). The Mixed Air Setpoint Manager is used to establish a temperature setpoint at the mixed air node. The AHU outdoor air controller then operates the outdoor air damper to attempt to meet this setpoint. The main function of the Mixed Air Setpoint Manager is to pick up a temperature setpoint provided by a setpoint manager located downstream of the supply fan and then set this value as the off-coil setpoint, having first subtracted the calculated temperature rise across the supply fan.

Note: A Mixed Air Setpoint manager is only included in systems that incorporate 'draw-through' coils. Also, if a setpoint manager is added in front of a draw-through fan, the Mixed Air Setpoint manager is omitted.

Working At HVAC System Level

The HVAC system level is at the top of the hierarchy of the building HVAC data. At HVAC system level, you can:

1. Use the [Load HVAC template](#) dialog to load a fully connected HVAC system. When there are no HVAC loops or HVAC zone groups defined already and you navigate to the HVAC system level, this dialog is opened automatically.
2. Use the [Save HVAC Template](#) tool to save the currently defined system as a template for future use.
3. [Add](#) and edit [air, plant and condenser loops](#).
4. [Add](#) and [edit zone groups](#).
5. Delete one or more existing HVAC components. Either select the component(s) for deletion and use the **Delete** toolbar icon or, to delete all components, use the Delete tool with no components selected.

Load HVAC Template Dialog



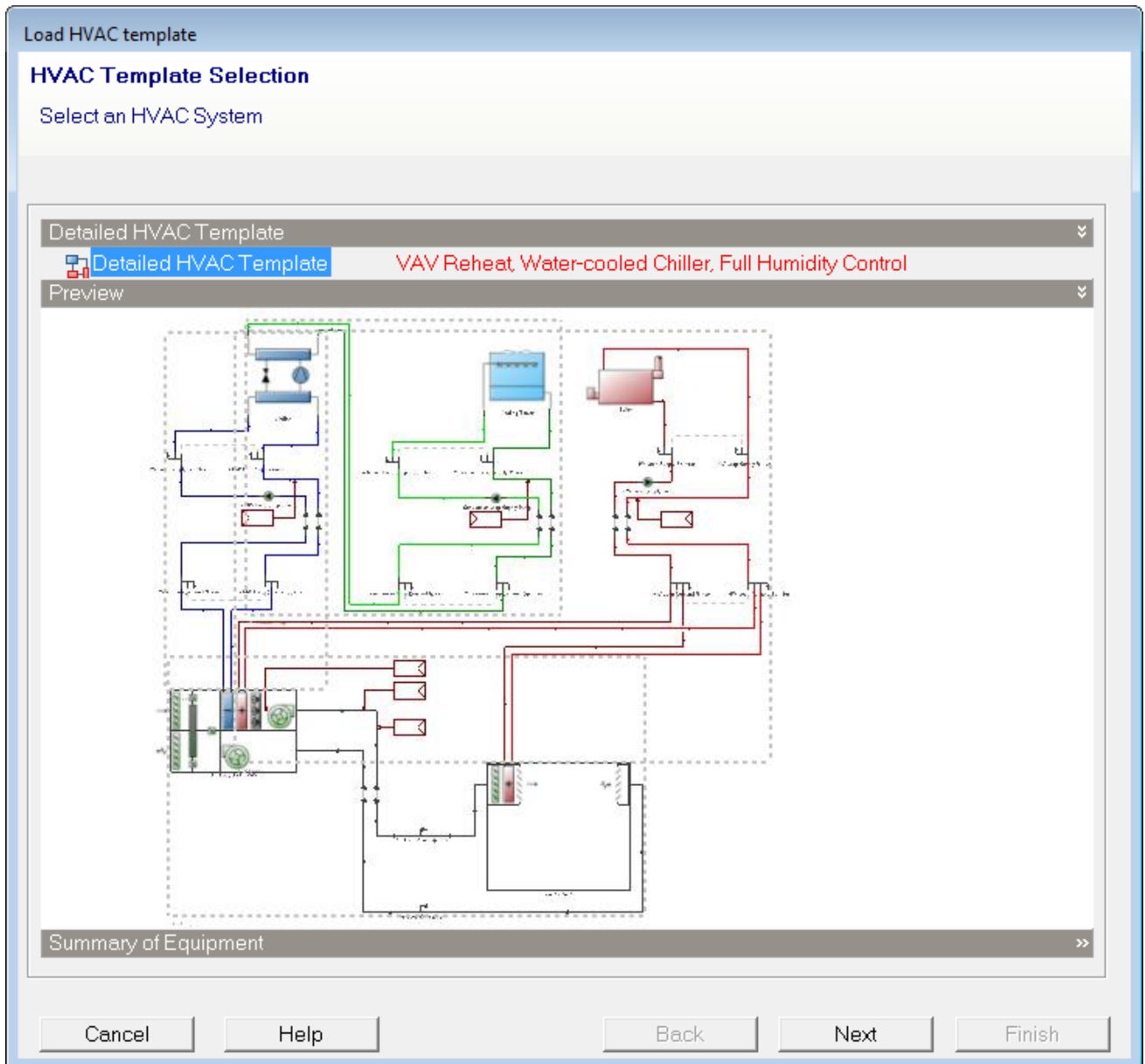
You can use the Load HVAC Template dialog to load a fully connected HVAC system to the model either in addition to any system data already defined or to replace it. To load an HVAC template, first go to the [HVAC System level](#) then click on the **Load HVAC Template** tool.

Note: You can find a summary description of all the available HVAC templates in the [HVAC Templates Overview](#) Guide.

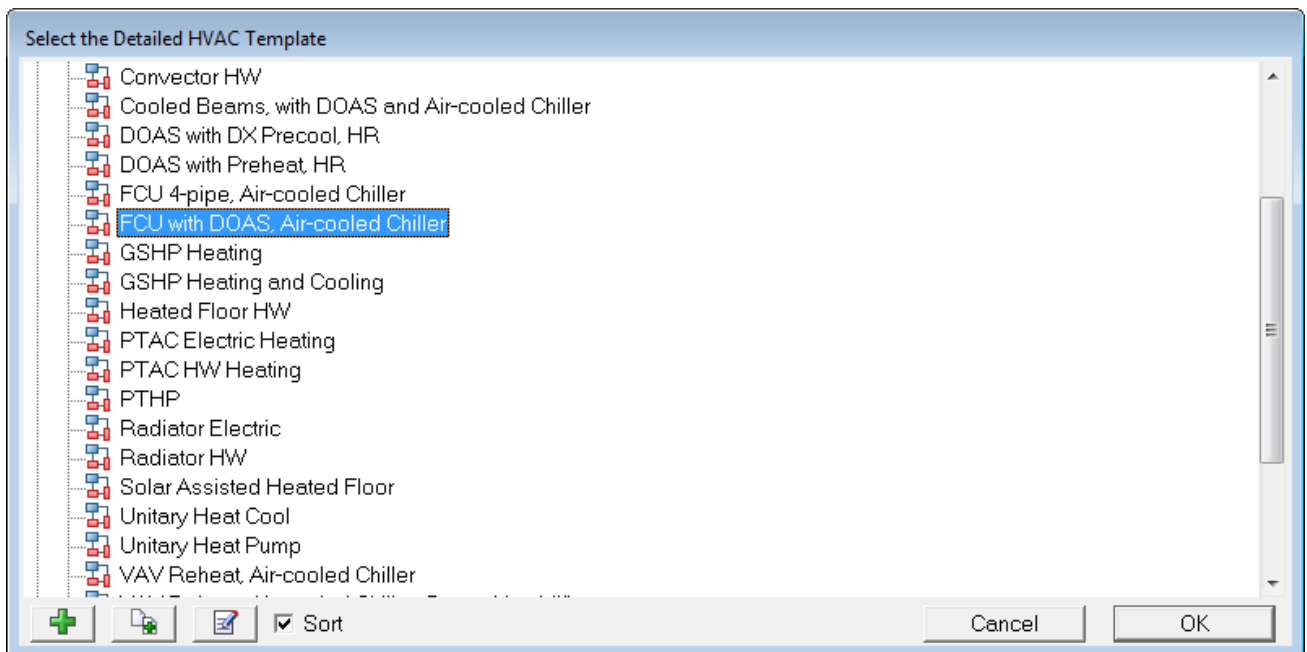
Note: When there are no HVAC loops or HVAC zone groups defined already and you navigate to the HVAC system level, this dialog is opened automatically.

Select The HVAC System Template

The first page on the **Load HVAC Template** dialog allows you to select the predefined HVAC system you would like to load. The thumbnail image shows an image of the system layout currently selected.



To select a different HVAC system, click on the Detailed HVAC template browse item to reveal the browse button and click on the browse button to bring up a list of available templates. With this list open simply select the HVAC system you would like and press the OK button.



Once you have the right HVAC system selected click on the Next button in the **Load HVAC Template** dialog.

Select Building Zones

On the second page you can select the building zones to be served by the selected HVAC template. By default all zones that are currently not already served are selected. You will normally want to unselect any unoccupied zones or other zones that are not served by this system.

Load HVAC template

VAV Reheat, Water-cooled Chiller, Full Humidity Control: Select the zones served by this HVAC System

Use the check boxes to select the zones to be served by this HVAC system.

Zones

- ☒ Training Building
 - ☒ L0
 - ☒ Cell Office NE
 - ☒ Circulation
 - ☒ Open Office N
 - ☒ Open Office S
 - ☒ Open Office W
 - ☒ Reception
 - ☒ Store
 - ☒ Toilet F
 - ☒ L1
 - ☒ Open Office E
 - ☒ Open Office N
 - ☒ Open Office S
 - ☒ Open Office W
 - ☒ Roof
 - ☐ Roof Void

Options

- ☒ Replace all existing HVAC systems
- ☒ Override template defaults

Cancel Help Back Next Finish

Replace all existing HVAC systems

On this page of the dialog you can choose to **Replace all existing HVAC systems** which, as the name suggests, removes any existing HVAC components before placing the selected HVAC system and allows all zones in the building to be selected as being served by the selected HVAC template.

Override template defaults

This option allows you to override the default settings saved with the template using your own values on further pages on this dialog.

Air System Data

If the HVAC template includes an air system and you chose to override template defaults then on this page you get the opportunity to provide key settings for the air system.

Load HVAC template

VAV Reheat, Water-cooled Chiller, Full Humidity Control: Air Handling Unit

Enter details on the Air System

AHU Fans		<<
Supply Fan		<<
Pressure rise (inH2O)	2.411	
Fan efficiency	0.700	
☒ Part-load performance	Fan Part-Load Power, Inlet Vane Dampers	
Extract Fan		<<
☐ Include extract fan		
Recirculation		<<
☒ On		
Economiser (Free Cooling)		<<
Economiser control type	1-No economizer	
Heat Recovery		<<
☒ On		
Sensible effectiveness	0.750	
Latent effectiveness	0.000	

Cancel Help Back Next Finish

Hot Water System Data

If the HVAC template includes a hot water system and you chose to override template defaults then on this page you get the opportunity to provide key settings for the hot water system.

Load HVAC template

VAV Reheat, Water-cooled Chiller, Full Humidity Control: Hot Water System
Enter details on the Hot Water System

Pump Type

Rated pump head (ftH2O)

6.691000

Plant loop flow type

2-Variable flow

☒ Performance curve template

Pump Part-Load Power, Default Variable Speed

Boiler

 Boiler template

Gas-fired condensing boiler

Boiler capacity (Btu/h)

Autosize

Cancel

Help

Back

Next

Finish

Chilled Water System Data

If the HVAC template includes a chilled water system and you chose to override template defaults then on this page you get the opportunity to provide key settings for the chilled water system.

Load HVAC template

VAV Reheat, Water-cooled Chiller, Full Humidity Control: Chilled Water System

Enter details on the Chilled Water System

Pump Type	
Rated pump head (ftH2O)	6.691000
Plant loop flow type	2-Variable flow
☒ Performance curve template	Pump Part-Load Power, Default Variable Speed
Chiller	
 Chiller template	DOE-2 Centrifugal/5.50COP

Loading The Template

When you have entered all of the necessary data click on the Finish button. If there is no existing HVAC data or you chose to replace all existing HVAC data then the components associated with the template are placed automatically on the HVAC layout screen. Alternatively, if the HVAC template is being used to add components to an existing system, you should use the mouse to position the new components away from the existing ones and click with the mouse to actually place them. You may need to zoom out to find a free area to place the new components.

Tip: See the [Save HVAC Template](#) page for specific guidance on [loading custom HVAC templates](#)

Note: If you click on an area where the new HVAC components would overlap with any that already exist then the load is cancelled.

See the [Introduction to Detailed HVAC](#) tutorial

Save HVAC Template

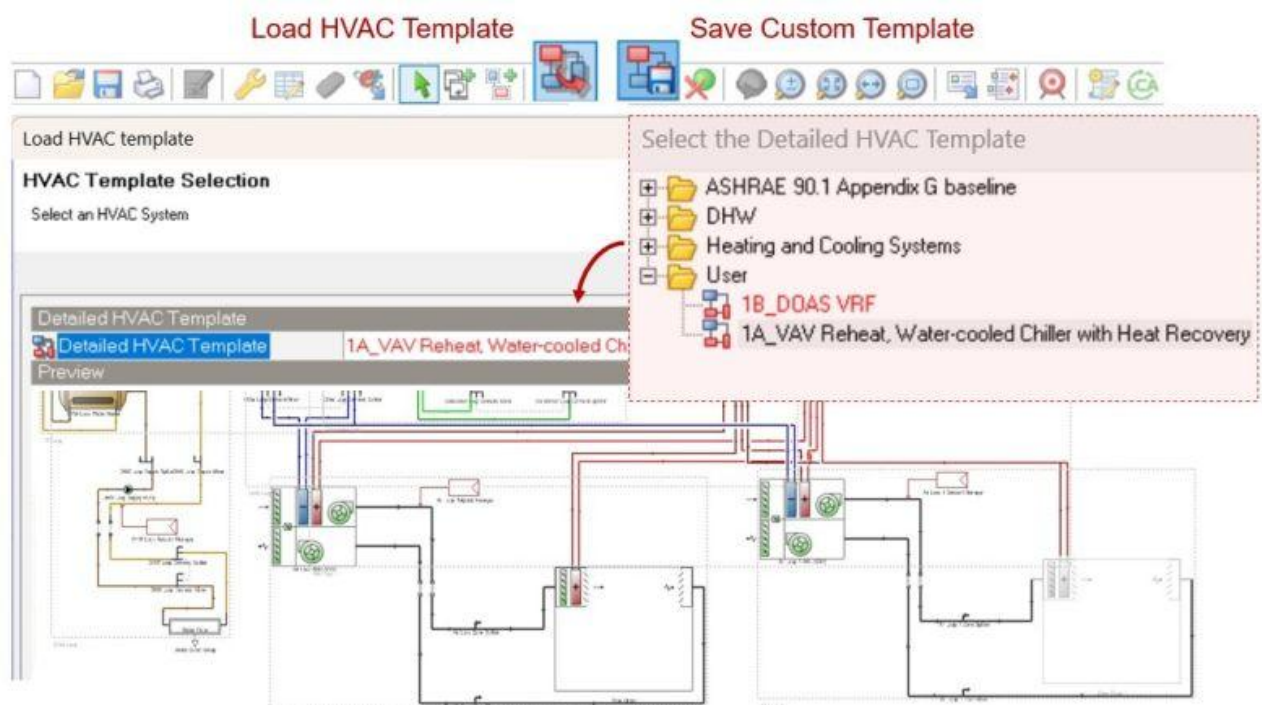


Use the **Save HVAC Template** tool if you would like to save the HVAC system defined in your model for loading to this or other models in the future.

Summary - how to save a custom HVAC template

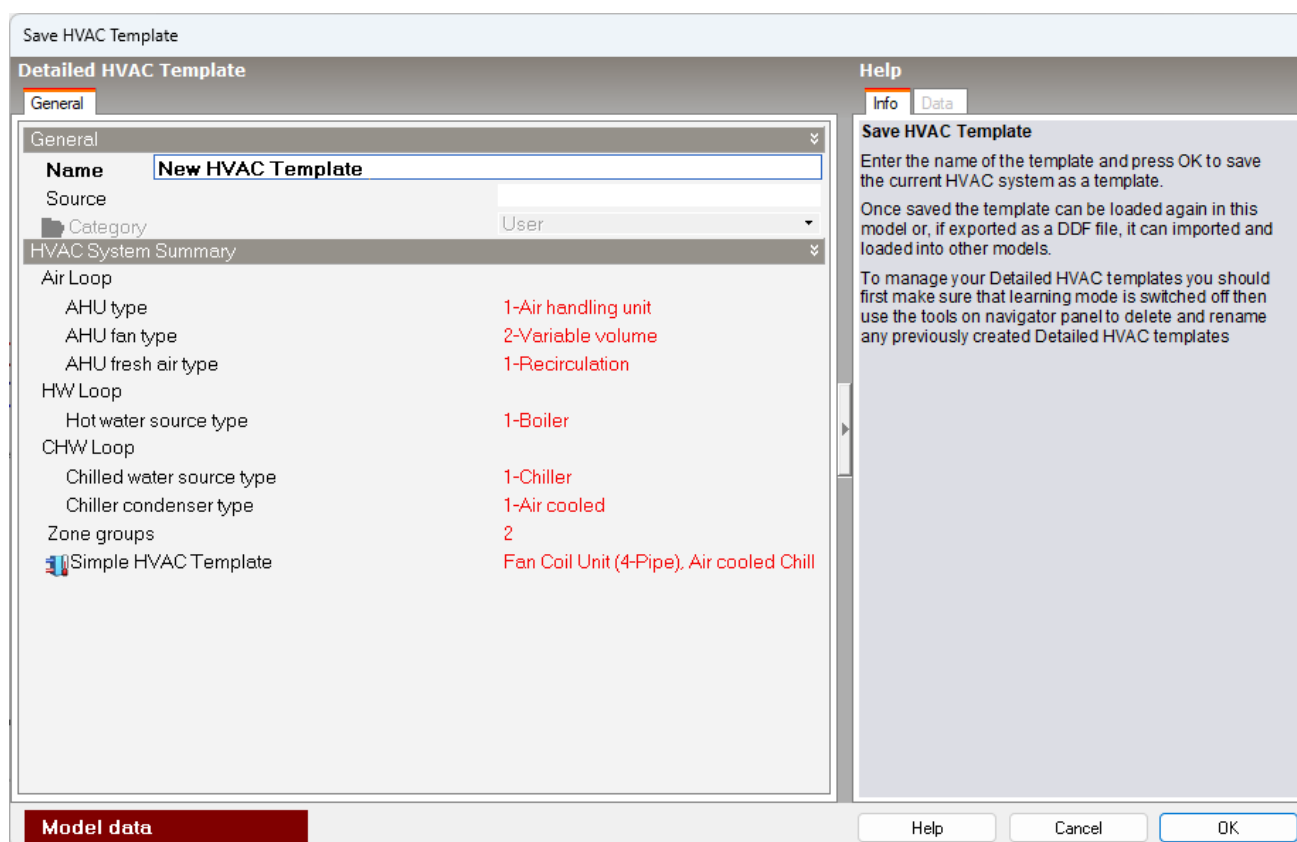
To save a custom HVAC template, follow these steps:

1. Go to the [HVAC System level](#) by click on the **<HVAC System>** item in the Navigator.
2. Open the [Save HVAC Template dialog](#) by clicking on the **Save HVAC Template** icon on the toolbar.
3. On the dialog, enter a descriptive name for your template.
4. Press the OK button to save the template.



Save HVAC Template dialog

The Save HVAC Template dialog allows you to name the HVAC template and provides a summary of the sub-systems that will included with the saved template.



When you press the OK button to save the template the HVAC data is stored in the template library.

Note: The whole HVAC system, as seen at HVAC system level, is saved and there is no way to selectively save particular parts of the system.

Loading custom HVAC templates

Any HVAC systems saved as a template will be available for subsequent loading using the [Load HVAC Template](#) dialog. The HVAC template library is global to the program and so any HVAC templates that you save will automatically be available in other projects on your machine.

To load your custom HVAC template:

1. Go to the [HVAC System level](#) by click on the **<HVAC System>** item in the Navigator.
2. Load the template by clicking the **Load HVAC Template** icon to launch the [Load HVAC Template](#) dialog.
3. Select your custom template by clicking the ellipsis (...) to the right of the **Detailed HVAC Template** control and navigate to your template in the **User** folder.
4. Select the zones to include in the first zone group of this template. See note below.
5. Click the Finish button to load the system.

Note: When loading custom templates saved from more complex HVAC layouts with more than 1 zone group, only the first zone group in the system can be populated with zones from the [Load HVAC Template](#) dialog. However, you can easily assign zones to the other zone groups from the [HVAC zone group](#) dialog as required after the template has been loaded .

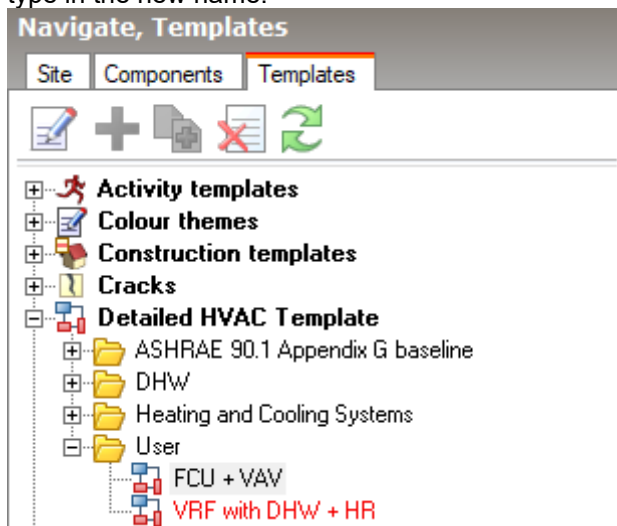
Exporting custom HVAC template(s)

You can [export](#) your HVAC template library in DDF form just like any other type of template or component. This is the most secure method to store your HVAC templates as the DDF file can be loaded to other machines and in future installations.

Managing custom HVAC templates

To manage your custom Detailed HVAC templates, follow these steps:

1. Ensure DesignBuilder is in [non-learning mode](#). In this case all templates are listed under the **Templates** tab on the Navigator.
2. Open the **Detailed HVAC Templates** list item and then open the **User** category item below that to display the list of custom HVAC templates
3. Use the icons at the top of the Navigator panel to delete or view custom templates in the list.
4. To rename a template, click on the template in the list, then click again to open the list label editor, and type in the new name.



See the [Introduction to Detailed HVAC](#) tutorial

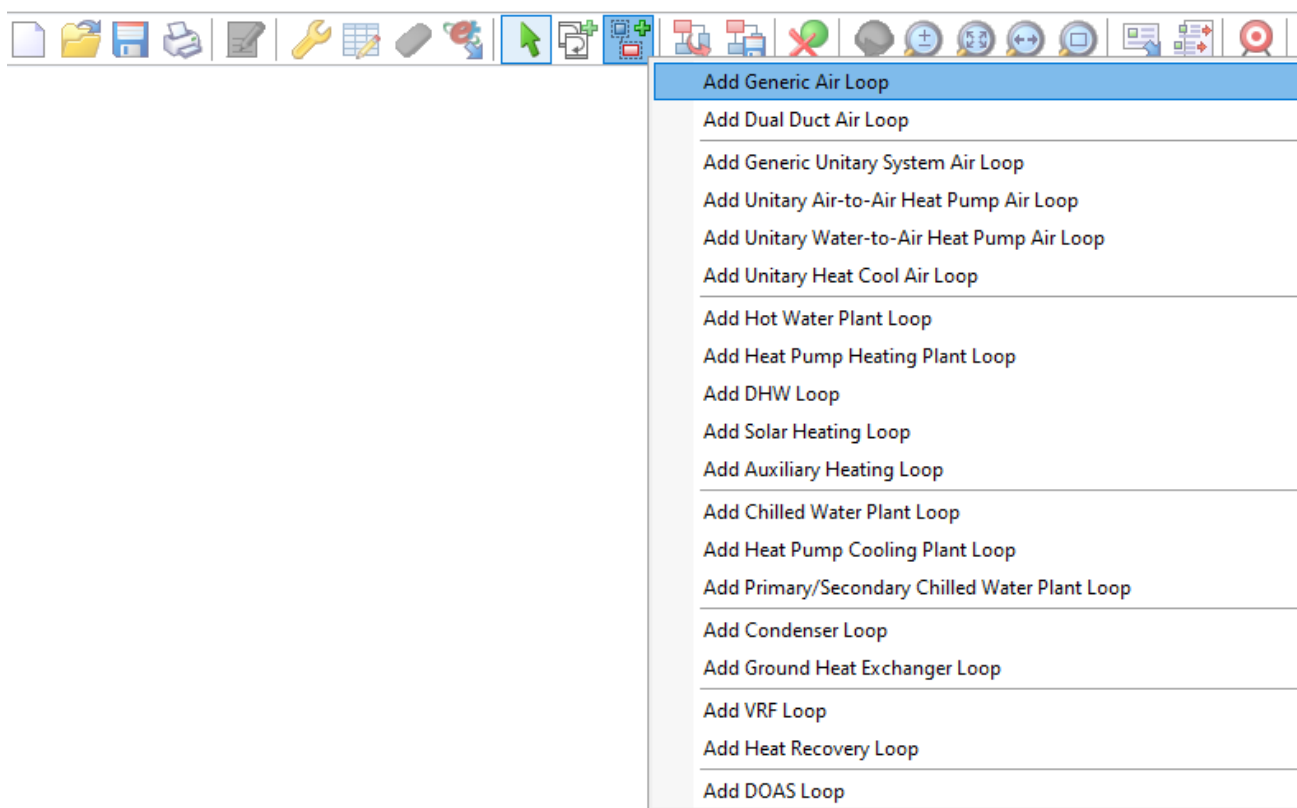
Add Zone Group Tool



To place an [HVAC zone group](#), first go to the [HVAC System level](#) and click on the Add zone group tool. To place a zone group, simply move the cursor to the required position on the Edit screen and click the mouse button to place the zone group.

See the [HVAC zone groups and components](#) tutorial

Add Loop Tool



To place an HVAC loop, first go to the HVAC System level and click on the Add loop tool which will display a drop-menu of available pre-defined loops. You can then select the type of loop you require from the menu.

Having selected the loop type, to place it, move the cursor to the required position on the Edit screen and click the mouse button.

See the [Adding loops and connecting components](#) tutorial

Auto Connect Tool

This tool can be used to boost your productivity when connecting HVAC plant loops to equipment in Zone groups and Air loops. To connect one or more Plant loops to one or more HVAC loops and Zone groups:

1. Navigate to HVAC System level.
2. Select all loops and Zone groups to be connected. Use <Ctrl> + mouse click or the rubber band selection tool to select more than one object.
3. Click on the [Connect components](#) toolbar icon to automatically make the connections.

The diagram below illustrates the process.

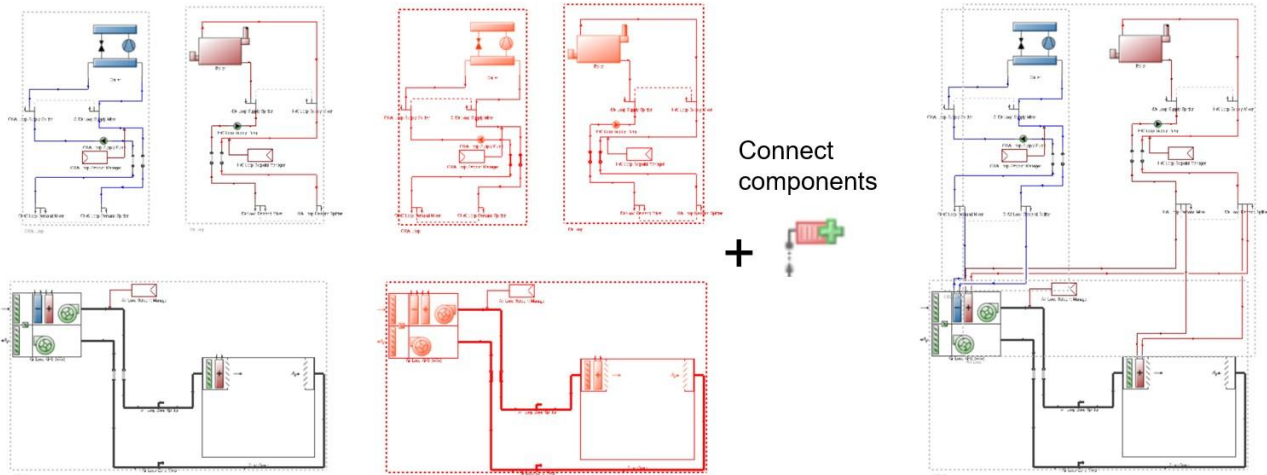
Not yet connected



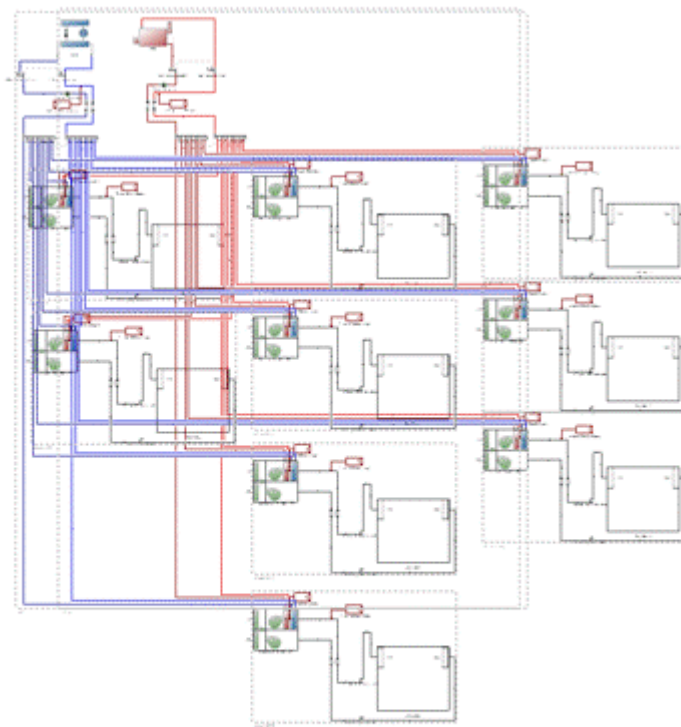
Select Loops and Zone Groups Together



Connected with a single click



The animation below shows a specific example.



With a **single click**, DesignBuilder created 32 pipe connections across eight additional AHUs

Tip: The initial positioning of loops and zone groups can significantly affect the layout of connections generated. The Auto connect tool doesn't try to avoid line intersections, so sometimes the connection layout won't look quite as you would draw it yourself. However if you're not happy with the result you can always undo the connection, reposition the objects and try again.

Note: There are some limitations in what can be automatically connected to what. For example chilled water loops or hot water loops won't be automatically connected to heat pump condensers. Any connections that were not made automatically can be completed manually using the [Connect components](#) tool in the usual way.

See also: "Connect Components Tool" on page 767

Working At Sub-Loop Level

At the sub-loop level, you can:

1. [Add](#) and [Edit](#) Boilers (heating plant supply sub-loop)
2. [Add](#) and [Edit](#) Chillers (chilled water plant supply sub-loop)
3. [Add](#) and [Edit](#) Cooling towers (condenser supply sub-loop)
4. [Add](#) and [Edit](#) Water heaters (domestic hot water plant supply sub-loop)
5. [Add](#) and [Edit](#) Pumps (all plant and condenser supply sub-loops)
6. [Add](#) and [Edit](#) Heating coils (dual-duct air loop supply sub-loop)
7. [Add](#) and [Edit](#) Cooling coils (dual-duct air loop supply sub-loop)
8. [Add](#) and [Edit](#) Humidifiers (dual-duct air loop supply sub-loop)
9. [Add](#) and [Edit](#) Water heater heat pumps.
10. [Add](#) and [Edit](#) Setpoint managers (all supply sub-loops)
11. [Add](#) Mixers and Splitters (all sub-loops)
12. [Connect components](#) (all sub-loops)
13. [Edit air handling units](#) (air loop supply sub-loop)

Note: Sub loops have no attribute data of their own so there is no sub-loop dialog.

Add Boiler Tool



[Boilers](#) can only be incorporated within the supply sub-loops of hot water plant loops so you first need to go to the supply sub-loop of the hot water plant loop to which the boiler is to be added. The Add boiler tool icon will then appear on the toolbar. To place a boiler, click on the Add boiler tool and then move the mouse cursor to the required position and click the mouse button to place the boiler.

Add Chiller Tool



[Chillers](#) can only be incorporated within the supply sub-loops of chilled water plant loops so you first need to go to the supply sub-loop of the chilled water plant loop to which the chiller is to be added. The Add chiller tool icon will then appear on the toolbar. To place a chiller, click on the Add chiller tool and then move the mouse cursor to the required position and click the mouse button to place the chiller.

Add Cooling Tower Tool



[Cooling towers](#) can only be incorporated within the supply sub-loops of condenser loops so you first need to go to the supply sub-loop of the condenser loop to which the cooling tower is to be added. The Add Cooling Tower tool icon will then appear on the toolbar. To place a cooling tower, click on the Add Cooling Tower tool and then move the mouse cursor to the required position and click the mouse button to place the cooling tower.

Add Water Heater Tool



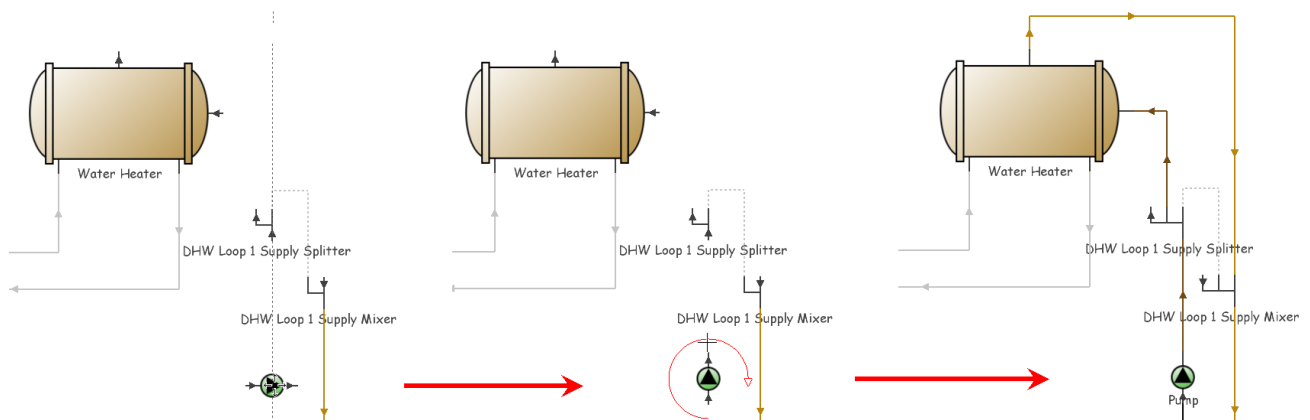
[Water heaters](#) can only be incorporated within the supply sub-loops of domestic hot water plant loops so you first need to go to the supply sub-loop of the domestic hot water plant loop to which the water heater is to be added. The Add Water Heater tool icon will then appear on the toolbar. To place a water heater, click on the Add Water Heater tool and then move the mouse cursor to the required position and click the mouse button to place the water heater.

Add Pump Tool



Pumps can only be included in the supply sub-loops of plant and condenser loops. EnergyPlus requires that the pump must be the first component in the sub-loop. Consequently, a pump must be connected directly to the sub-loop inlet node. Only one pump can be added to each plant/condenser supply sub-loop and so the tool will only appear on the toolbar at the plant/condenser sub-loop level if you delete the existing pump in order to change its orientation.

To place a pump, click on the Add pump tool and click the mouse button when the pump is at the required location. After clicking the mouse button, you can then rotate the pump by moving the mouse in a circular direction around the centre of the pump to obtain the required orientation:

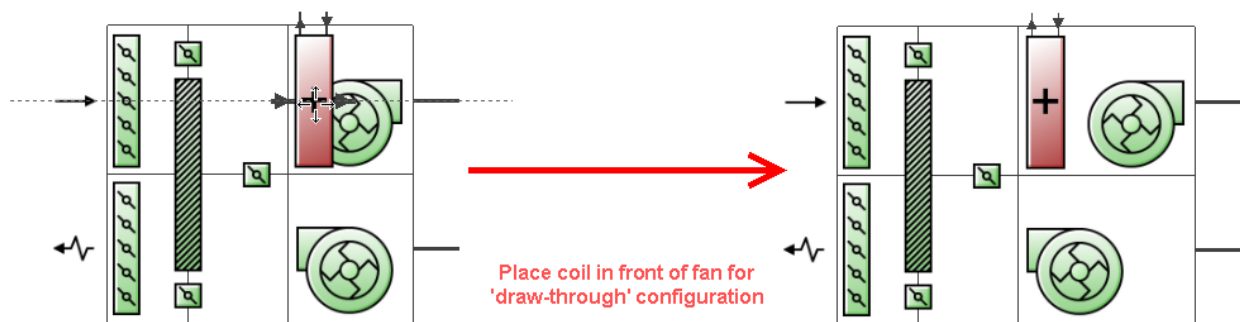


Add Heating Coil Tool



Heating coils can only be incorporated within the supply sub-loops of air loops so you first need to go to the supply sub-loop of the air loop to which the coil is to be added. The Add heating coil tool icon will then appear on the toolbar. To place a heating coil, click on the Add heating coil tool.

Heating coils can be placed anywhere in the sub-loop but are more commonly incorporated within an air handling unit. To add a heating coil to an air handling unit, after clicking on the Add heating coil tool, move the cursor (with the coil attached to it) into the supply section of the air handling unit and click the mouse button to place the coil at the required location. Coils may be placed in front of the supply fan for a 'draw-through' configuration or after the fan for a 'blow-through' configuration:

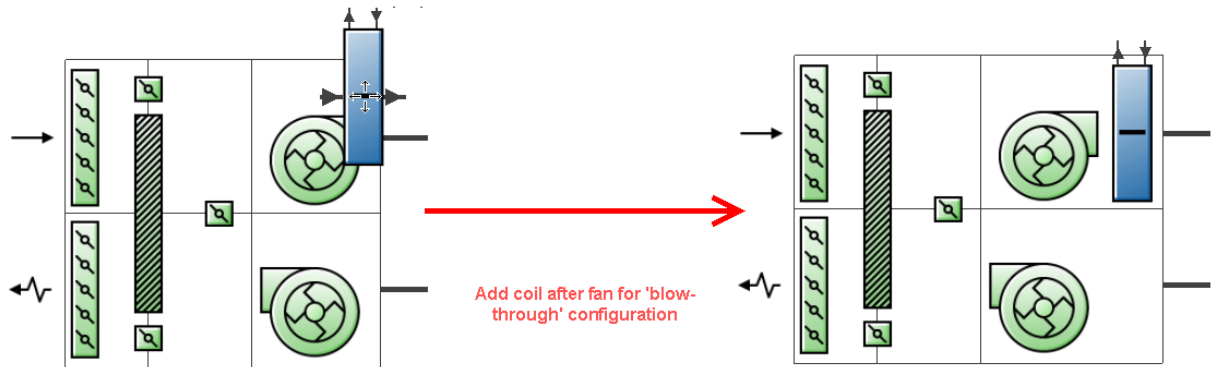


Add Cooling Coil Tool



Cooling coils can only be incorporated within the supply sub-loops of air loops so you first need to go to the supply sub-loop of the air loop to which the coil is to be added. The Add cooling coil tool icon will then appear on the toolbar. To place a cooling coil, click on the Add cooling coil tool and then select the required coil from the drop-list.

Cooling coils can be placed anywhere in the sub-loop but are more commonly incorporated within an air handling unit. To add a cooling coil to an air handling unit, after clicking on the Add cooling coil tool, move the cursor (with the coil attached to it) into the supply section of the air handling unit and click the mouse button to place the coil at the required location. Coils may be placed in front of the supply fan for a 'draw-through' configuration or after the fan for a 'blow-through' configuration:

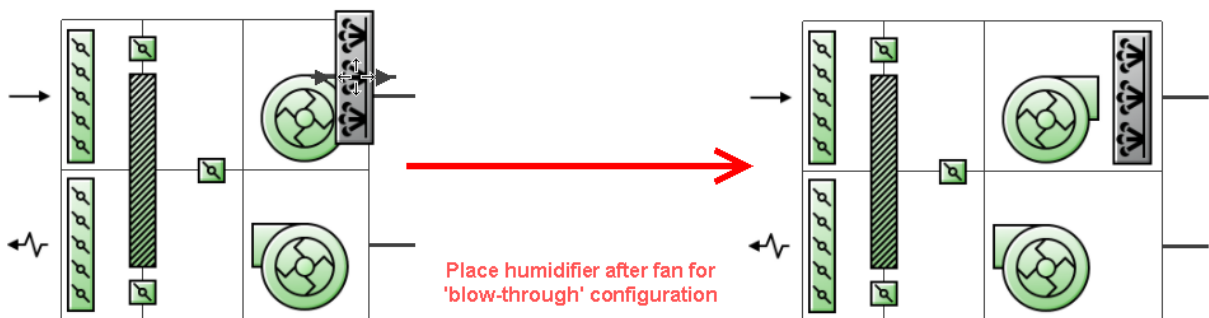


Add Humidifier Tool

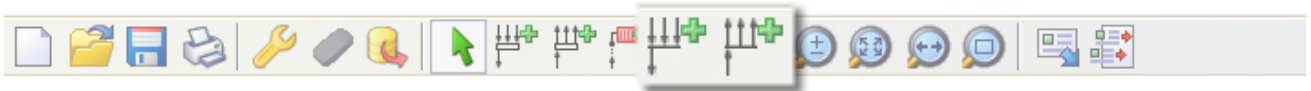


Humidifiers can only be incorporated within the supply sub-loops of air loops so you first need to go to the supply sub-loop of the air loop to which the humidifier is to be added. The Add humidifier tool icon will then appear on the toolbar. To place a humidifier, click on the Add humidifier tool.

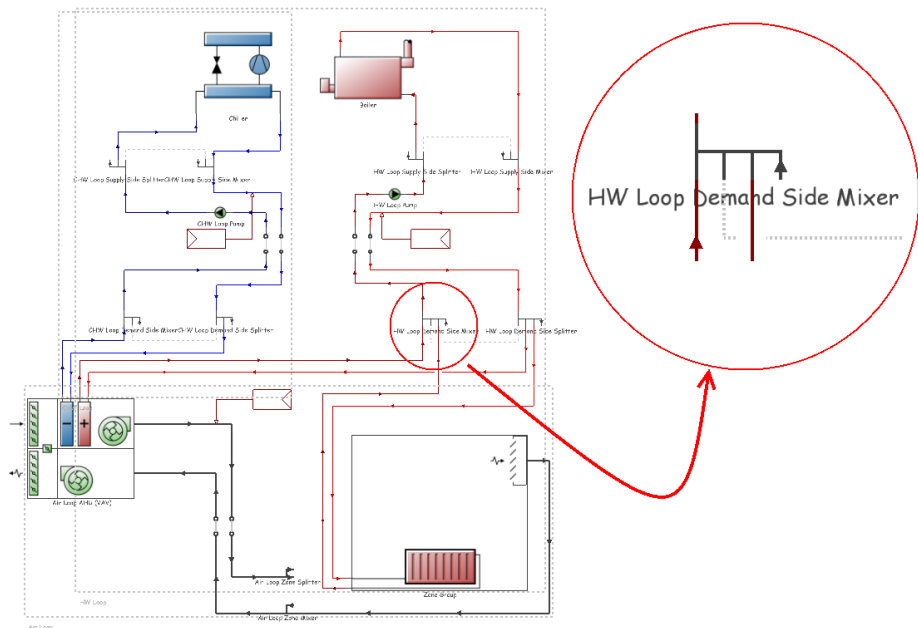
Humidifiers can be placed anywhere in the sub-loop but are more commonly incorporated within an air handling unit. To add a humidifier to an air handling unit, after clicking on the Add humidifier tool, move the cursor (with the humidifier attached to it) into the supply section of the air handling unit and click the mouse button to place the humidifier at the required location. Humidifiers may be placed in front of the supply fan for a 'draw-through' configuration or after the fan for a 'blow-through' configuration:



Add Splitter/Add Mixer Tool



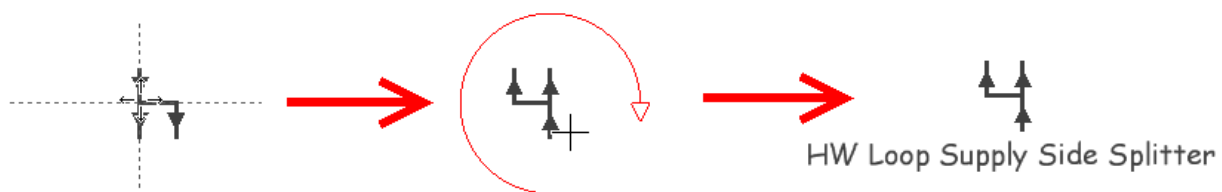
Splitter and mixers are used in HVAC in order to connect multiple items of equipment to a sub-loop by means of parallel branches:



Note: Only one splitter and one mixer can be incorporated within both the supply and demand sub-loops of plant and condenser loops and consequently the Add splitter and Add mixer tool will only appear on the toolbar if an existing splitter and mixer have been removed.

Air loops are slightly different in that air supply sub-loops comprise series component connections and consequently do not require any splitter or mixer with the exception of dual-duct systems which incorporate a single splitter to enable the flow to be split between hot and cold ducts. Air demand sub-loops are similar to plant and condenser sub-loops in that they can only contain a single mixer but can incorporate an additional splitter to cater for dual duct systems.

To place a splitter or mixer, click on the Add splitter or Add mixer tool and click the mouse button when the splitter/mixer is at the required location. After clicking the mouse button, you can then rotate the splitter/mixer by moving the mouse in a circular direction around the centre of the component to obtain the required orientation:



Add Supply/Return Plenum Tool



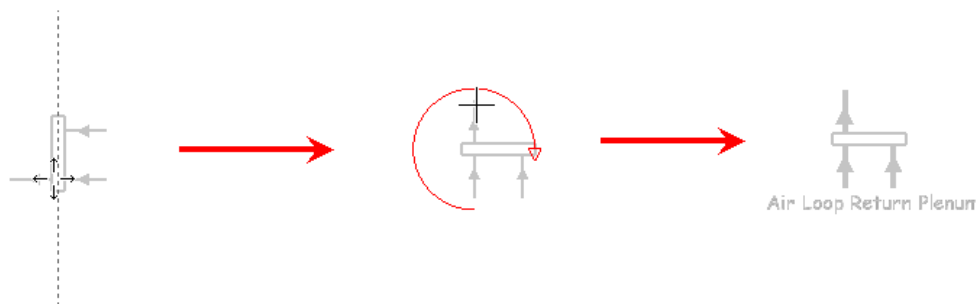
Plenums may be used to model building features such as ceiling or floor voids which are used to supply or return centrally conditioned air. Supply and return plenums are essentially uncontrolled building zones incorporated within the supply and return paths of air loop demand side sub-loops. Plenums can account for building side heat balance, that is they can transfer heat with other zones through surfaces, but they cannot incorporate HVAC equipment.

HVAC air plenums are special types of component since there is a heat balance connection and building description and a system airflow connection that transfers the airflow directly from a supply plenum to a controlled zone or from a controlled zone to a return plenum in the system air simulation.

Placing plenums

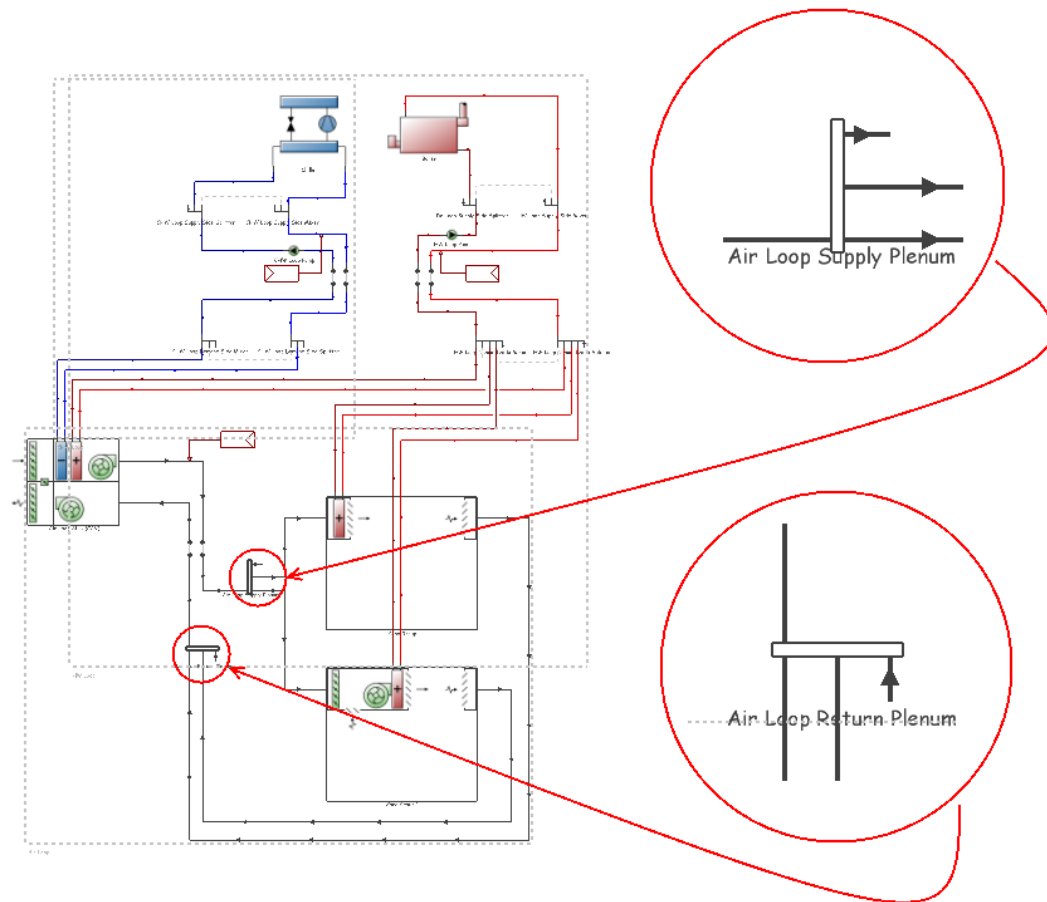
When an air loop is first placed, a splitter and mixer component is automatically included on the supply and return sides respectively. The normal way to place a supply plenum is to first delete the original splitter component and replace it with a supply plenum component. Likewise to include a return plenum, first delete the original mixer component and replace it with a return plenum component.

To place a supply or return plenum, click on the **Add supply plenum** or **Add return plenum** tool and click the mouse button when the plenum is at the required location. After clicking the mouse button, you can then rotate the plenum to its final orientation by moving the mouse in a circular direction around the centre of the component:



Connecting plenums

When you first place a plenum it will be shown on the HVAC layout diagram in light grey indicating that it does not yet have a building zone allocated to it and so cannot yet be connected up. So the first thing to do having placed a supply or return plenum is to [assign a building zone to the Plenum component](#).



Finished system with supply and return plenums

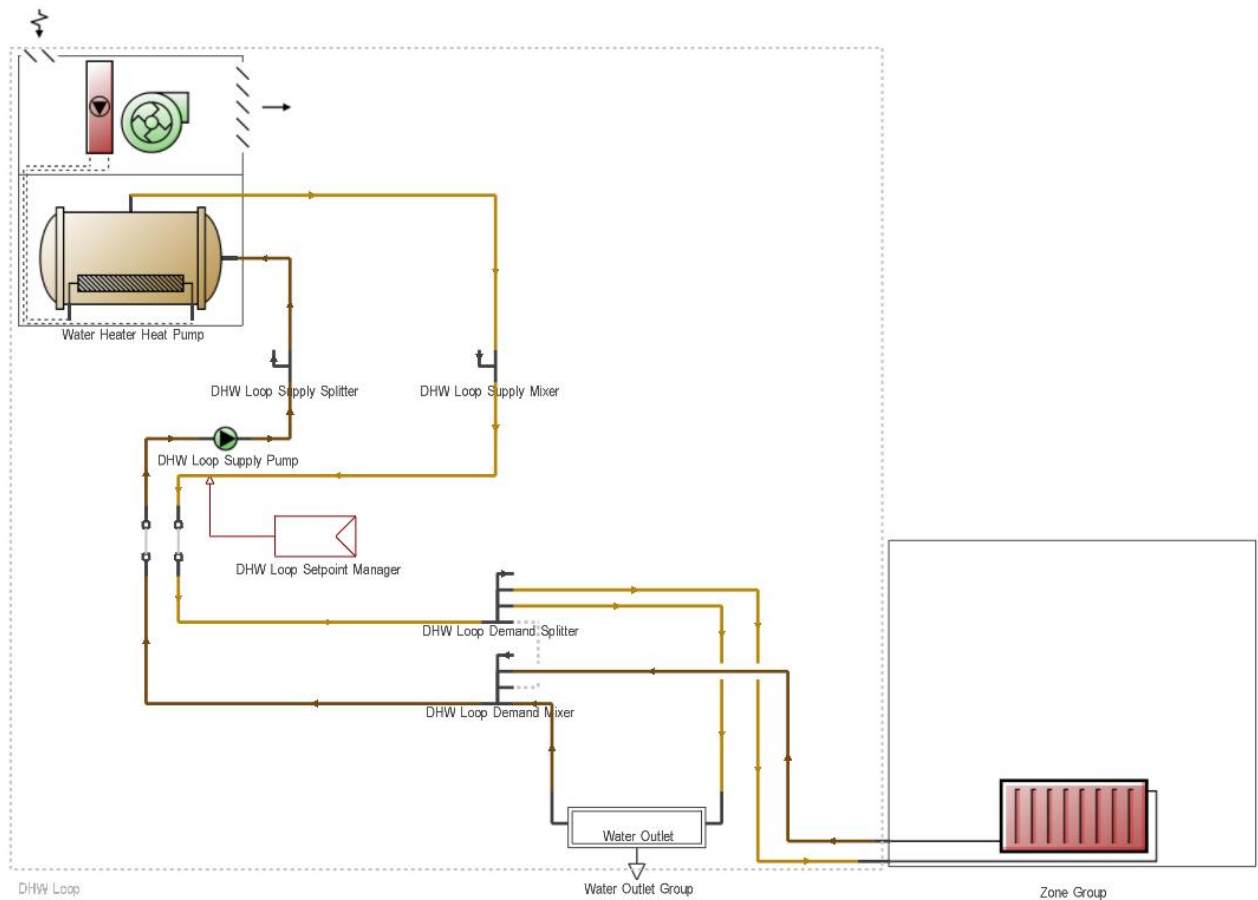
Note: Each air loop demand sub-loop can contain a single return plenum but can incorporate an additional supply plenum to cater for dual duct systems.

Add Water Heater Heat Pump



[Water heater heat pumps](#) (aka Air to Water Heat Pumps or ASHPs) are normally placed on the Supply side of DHW and Auxiliary loops. To do this, navigate to the Supply side of the DHW or Auxiliary loop, click on the **Add Water Heater Heat Pump** tool and left click with the mouse to place it.

In the example below, a Water heater heat pump is placed on the Supply side of a DHW loop. On the Demand side of that loop, water is supplied to a Water heater and to a Hot water radiator within an HVAC zone.



Note: Water heater heat pumps can also be added at the HVAC zone level, in which case, they are referred to as "Zone water heater heat pumps". See: [Add Zone Water Heater Heat Pump](#)

Add Setpoint Manager



Setpoint Managers are one of the high-level control constructs in EnergyPlus. A Setpoint Manager (SPM) is able to access data from any of the HVAC system nodes and use this data to calculate a setpoint (often a temperature setpoint) for one or more other HVAC system nodes.

Setpoint managers are added from the sub-loop level by clicking on the **Add Setpoint manager** toolbar icon. The process is summarised below,

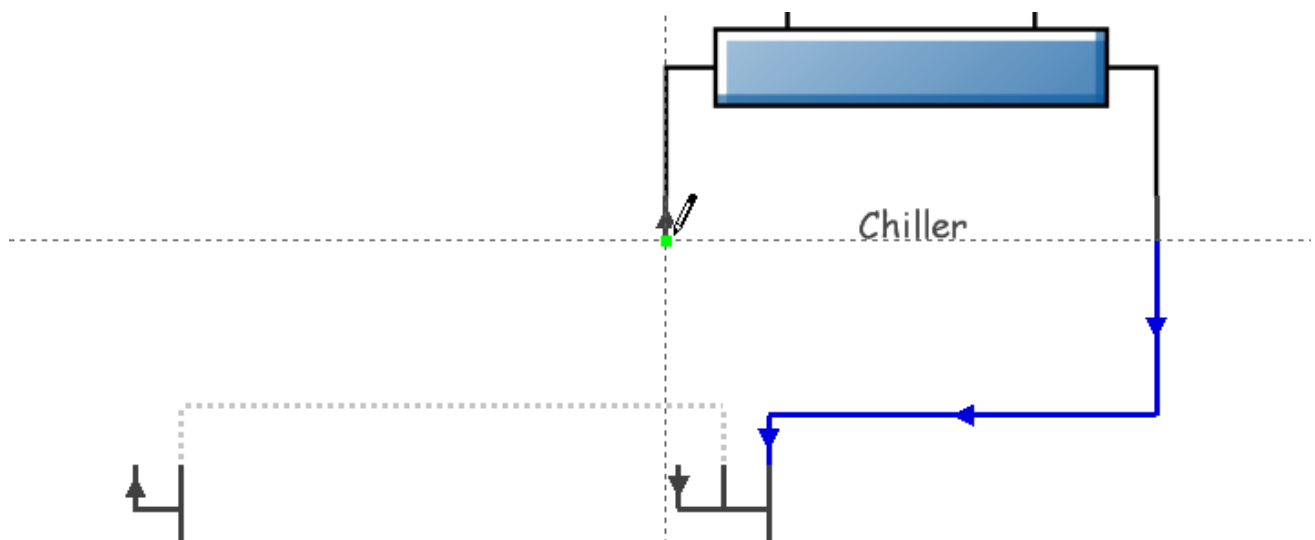
1. Navigate to the sub-loop where the SPM is to be added.
2. Click on the **Add Setpoint manager** toolbar icon to start the command.
3. Move the mouse cursor to the sensor point on the sub loop layout schematic, i.e. to the point where the SPM will sense the conditions as in input to the control logic. A red marker appears on the line at this point.
4. Click with the mouse to define the sensor point. A "tentative" SPM schematic object now appears on the screen, moving as you move the mouse cursor.
5. Move the mouse to point on the diagram where you would like the SPM itself to be placed and click with the mouse to place it.

6. Select the SPM so it is highlighted in red and right click on it with the mouse to open the context menu. Now click on the **Edit** menu item to open the [Edit Setpoint Manager](#) dialog. Here you can enter the SPM details such as type, control variable etc.

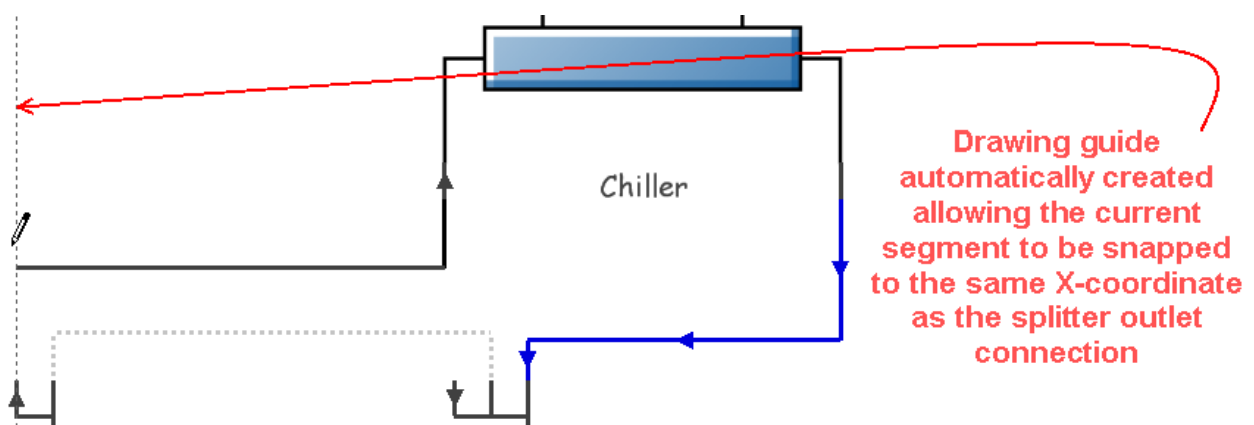
Connect Components Tool



The **Connect components** tool is used to create connections between the inlets and outlets of both air-side and water-side components. The tool can only be used at the sub-loop level. HVAC connections are created in the form of a number of segments or steps, each segment snapped to the X or Y axis. To create a connection between two components, first click on the Connect components tool and move the cursor to the required component inlet/outlet. When the cursor is over the required inlet/outlet, a green connection marker will be displayed to indicate that the beginning of the connection can be snapped to the inlet/outlet:



You can then click the mouse button to start the connection. Having snapped the beginning of the first connection segment to the required inlet/outlet, as you move the mouse cursor away, the cursor will automatically snap to the X or Y axis depending on the position of the cursor relative to the connection point. As you move the cursor across the screen, drawing guides are displayed to indicate that the current cursor position has been automatically snapped to align with existing connections in the X and Y axes:



When the cursor snaps to a required drawing guide, you can then click the mouse button to place the end point of the current connection segment. Finally, as you move the cursor towards the component inlet/outlet to which the connection is to be created, a green connection marker will again be displayed to indicate that the end of the connection can be snapped into position by clicking the mouse cursor button.

See also "Auto Connect Tool" on page 757.

See the [Adding loops and connecting components](#) tutorial

Working At HVAC Zone Group Level

When at the HVAC zone group level, you can edit the current zone group if no HVAC zones are selected by clicking on the Edit icon in the toolbar. Alternatively you can edit HVAC Zone data by selecting the HVAC zone in the Edit screen - it will be highlighted in red. Then right-click and in the menu select the Edit option.

- [Edit HVAC zone data.](#)

Working At HVAC Zone Level

When at the HVAC zone level, if no zone components are selected, you can edit the current HVAC zone by clicking on the Edit icon in the toolbar. Also, at the HVAC zone level, you can:

1. [Add](#) and [Edit](#) Air distribution units (VAV boxes, Chilled beams etc.).
2. [Add](#) and [Edit](#) Forced air units (Fan-coil units, etc.).
3. [Add](#) and [Edit](#) Radiant surfaces.
4. [Add](#) and [Edit](#) Heated floors.
5. [Add](#) and [Edit](#) Chilled ceilings.
6. [Add](#) and [Edit](#) Radiators/convectors.
7. [Add](#) and [Edit](#) High temperature radiant heaters.
8. [Add](#) and [Edit](#) Water heater heat pumps.
9. [Add](#) a Zone extract.
10. [Add](#) and [edit](#) Zone exhaust fan.

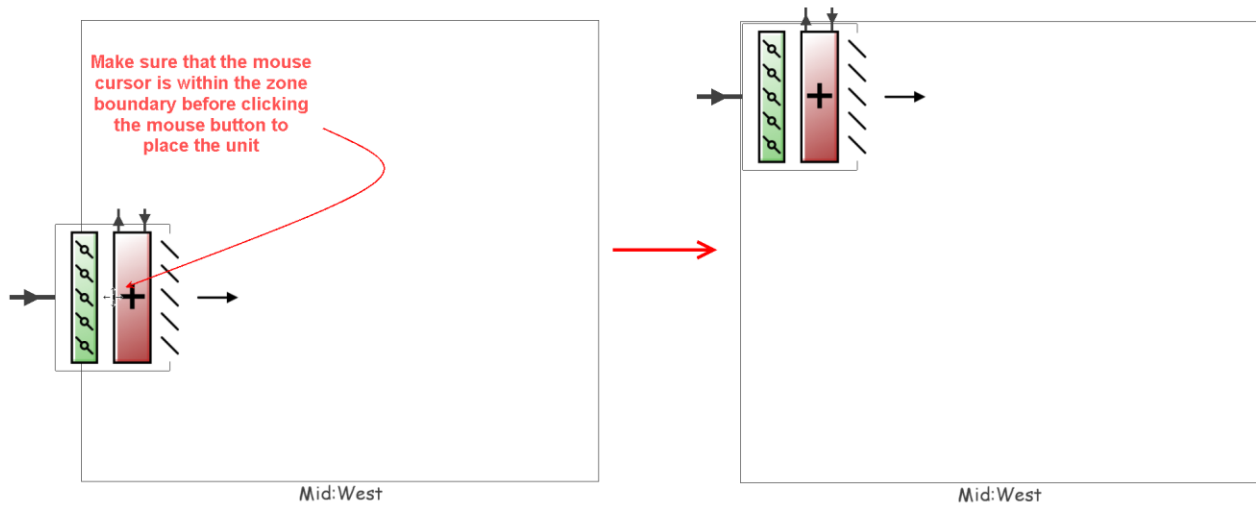
See also:

"Editing HVAC Zone Data" on page 926

Add Zone Air Distribution Unit Tool



[Zone air distribution units](#) are placed at the HVAC zone level. To place an ADU, navigate to the HVAC zone, click on the **Add air distribution unit** tool and then select the required unit from the drop-list. Then, making sure that the cursor is inside the boundary of the zone, left click with the mouse to place the ADU. The ADU will be automatically positioned at the top left of the zone:



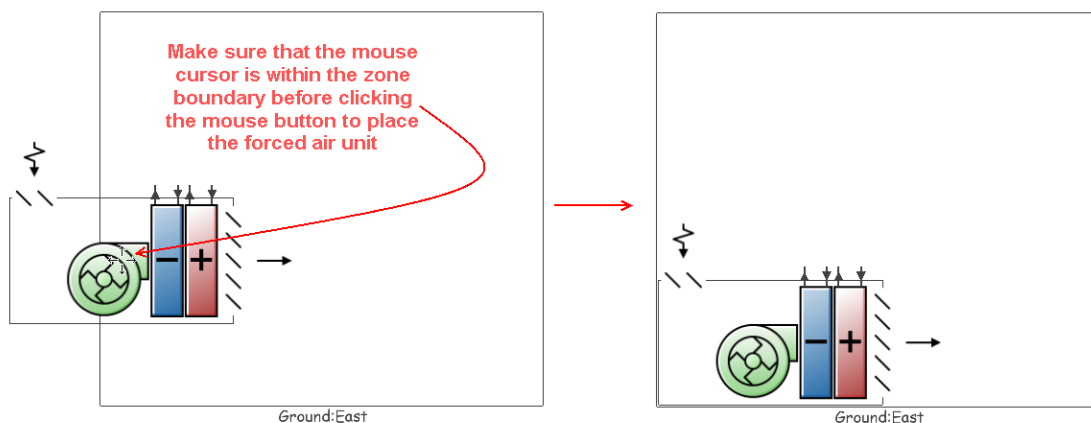
You should make sure to add a distribution unit that is compatible with the type of air loop you plan to connect to. The constant volume ADUs available in DesignBuilder can connect to both CAV and VAV air loops, while Variable volume ADUs can only connect to VAV air loops.

Note You must also add a zone [extract unit](#) to a zone that contains an ADU in order to connect up the air return path from the zone.

Add Zone Forced Air Unit Tool



[Forced air units](#) are placed at the HVAC zone level. To place a forced air unit, navigate to the HVAC zone, click on the **Add forced air unit** tool and then select the required unit from the drop-list. Then, making sure that the cursor is inside the boundary of the required zone, left click with the mouse to place the unit. The unit will be automatically positioned at the bottom left of the zone:

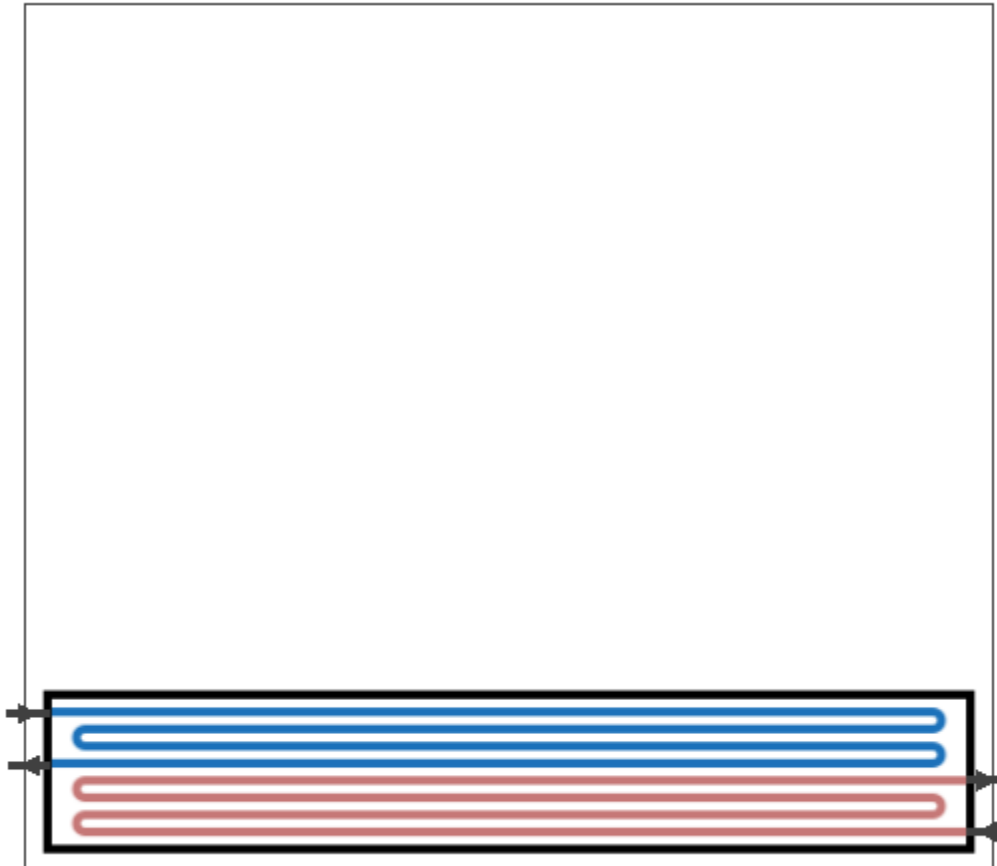


Add Zone Radiant Surface Tool



[Radiant Surfaces](#) are placed at the HVAC zone level. To place a Radiant Surface, navigate to the HVAC zone, click on the **Add Radiant Surface** tool and then, making sure that the cursor is inside the boundary of the required zone, left click with the mouse to place it.

The Radiant Surface will be automatically positioned at the bottom of the zone:

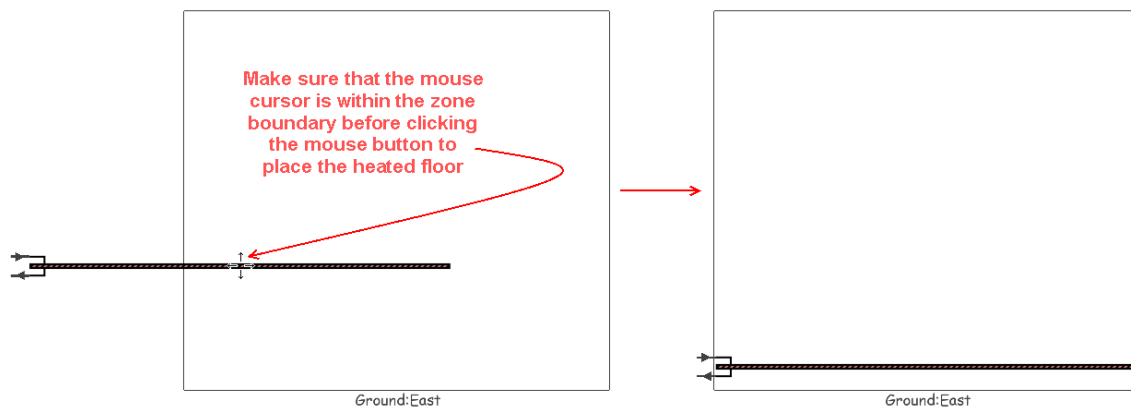


Add Zone Heated Floor Tool



[Heated floors](#) are placed at the HVAC zone level. To place a heated floor, navigate to the HVAC zone, click on the **Add heated floor** tool and then, making sure that the cursor is inside the boundary of the required zone, left click with the mouse to place the heated floor.

The heated floor will be automatically positioned at the bottom of the zone:

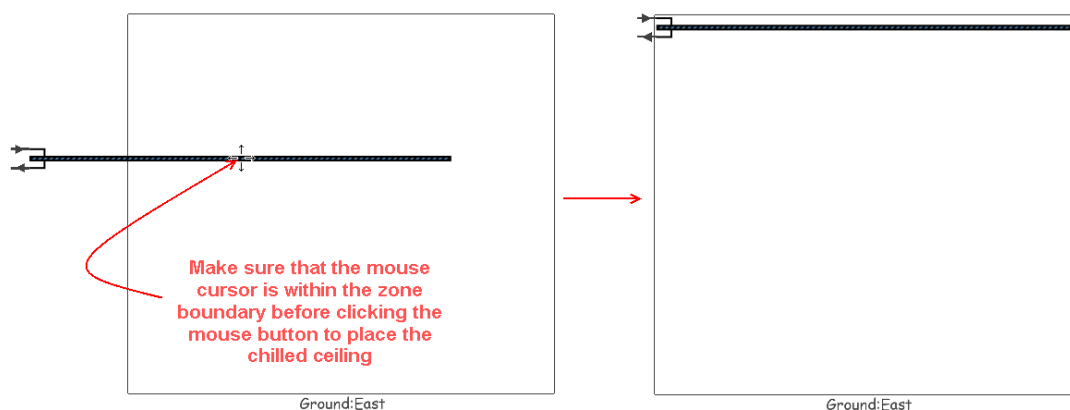


Add Zone Chilled Ceiling Tool



[Chilled ceilings](#) are placed at the HVAC zone level. To place a chilled ceiling, navigate to the HVAC zone, click on the **Add chilled ceiling** tool and then, making sure that the cursor is inside the boundary of the required zone, left click with the mouse to place the chilled ceiling.

The chilled ceiling will be automatically positioned at the top of the zone:



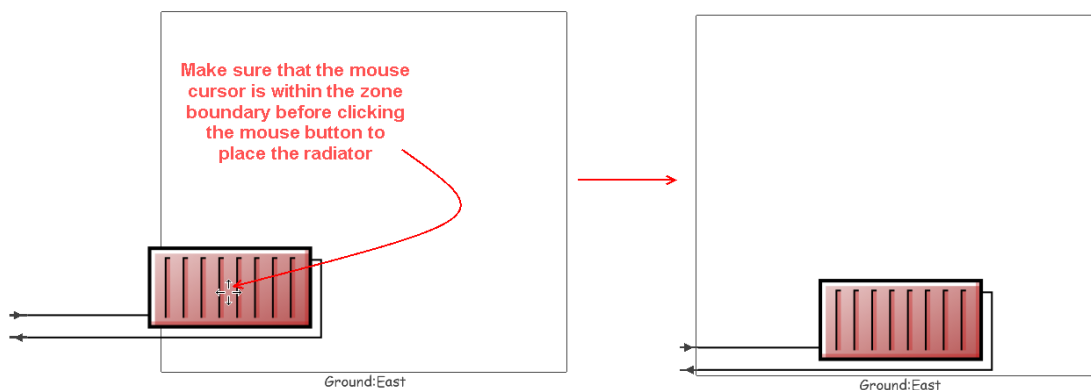
Note: Where chilled ceilings are added to zones with internal ceilings, the floor surface of the associated adjacent upper zone must have a construction with an internal source to represent the embedded piping system of the chilled ceiling. Similarly, in the case of zones with external flat roofs, the roof construction must contain an internal source. Refer to the documentation on [Constructions](#) for further information on defining [internal sources](#) for constructions.

Add Zone Radiative-Convective Unit Tool



[Radiators and convectors](#) are placed at the HVAC zone level. To place a radiator or convector, navigate to the HVAC zone, click on the **Add radiative-convective unit** tool and then select the required unit from the drop-list. Then, making sure that the cursor is inside the boundary of the required zone, left click with the mouse to place the radiator/convector.

The unit will be automatically positioned at the bottom centre of the zone:

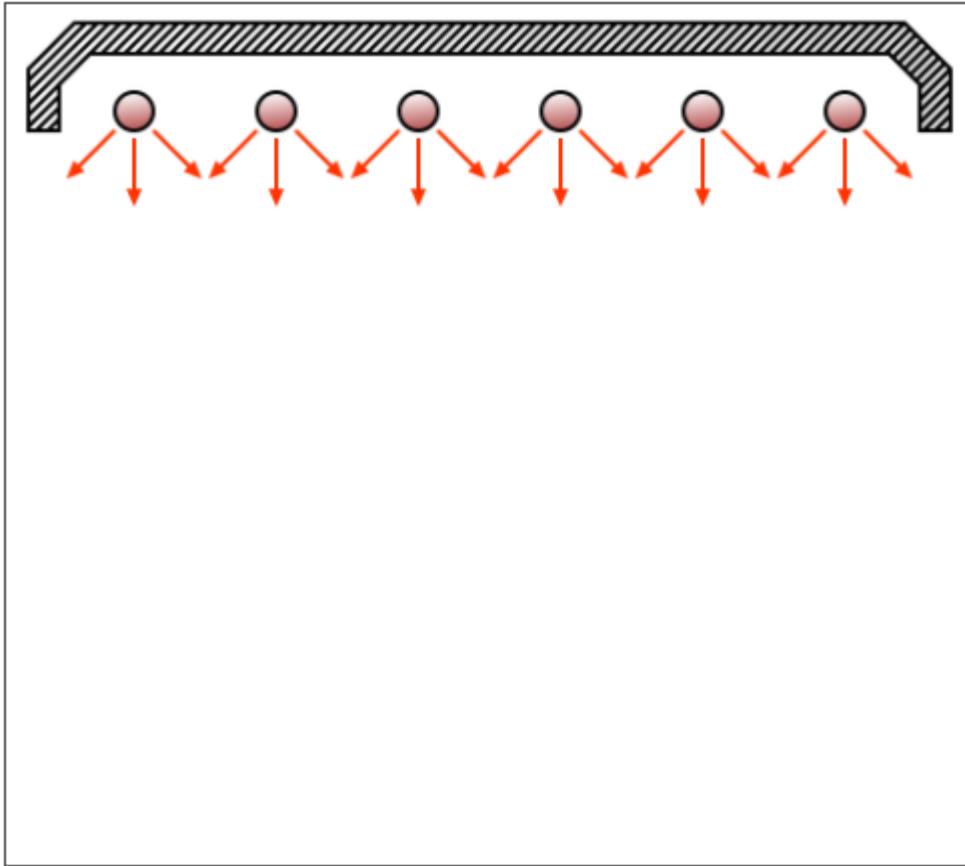


Add Zone High Temperature Radiant Heating



[High temperature radiant heaters](#) are placed at the HVAC zone level. To place a High temperature radiant heater, navigate to the HVAC zone, click on the **Add High temperature radiant heating** tool and then, making sure that the cursor is inside the boundary of the required zone, left click with the mouse to place it.

The High temperature radiant heating component will be automatically positioned at the top of the zone:

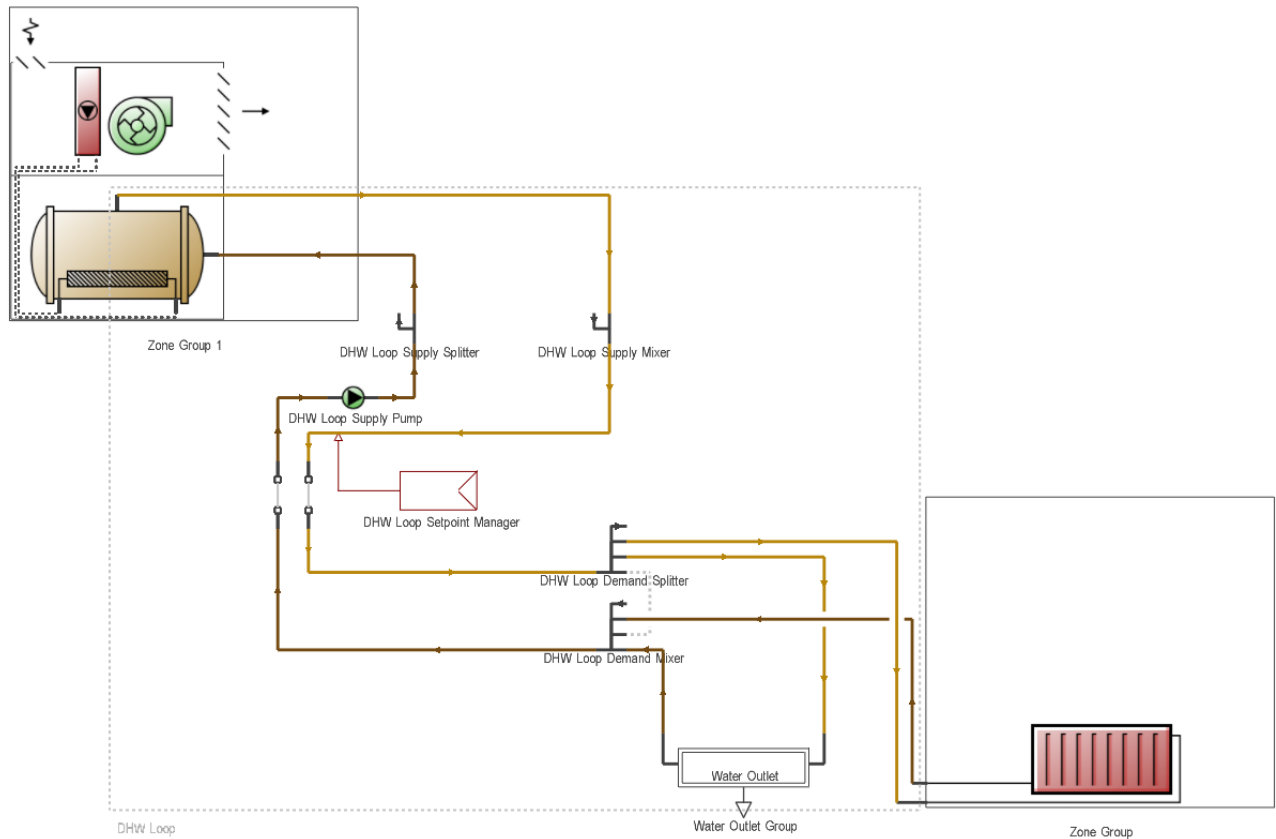


Add Zone Water Heater Heat Pump



[Zone water heater heat pumps](#) (aka Air to Water Heat Pumps or ASHPs) are placed directly in an HVAC zone. In this case, you must also connect it to an Auxiliary heating or DHW loop.

The application below represents a Water heater heat pump located inside a zone and connected to a DHW loop. An example of this application is an apartment building or hotel where the heat pump water heater is installed in a plant or utility room. The unit draws heat from the air in that room and uses it to heat the domestic hot water, so its operation can cool the room and affect the zone heat balance. In the model, the ASHP is therefore placed in the HVAC zone to represent this air-side interaction, while its water side is connected to the DHW loop supplying the building's hot-water outlets.



Note: In this case, the Zone Water Heater Heat Pump must be connected to a *single zone* HVAC group. You should create a separate HVAC Group for each zone that includes a Zone Water Heater Heat Pump. Placing the zone water heat pump in the HVAC zone without connecting it to an Auxiliary heating or DHW loop is not allowed.

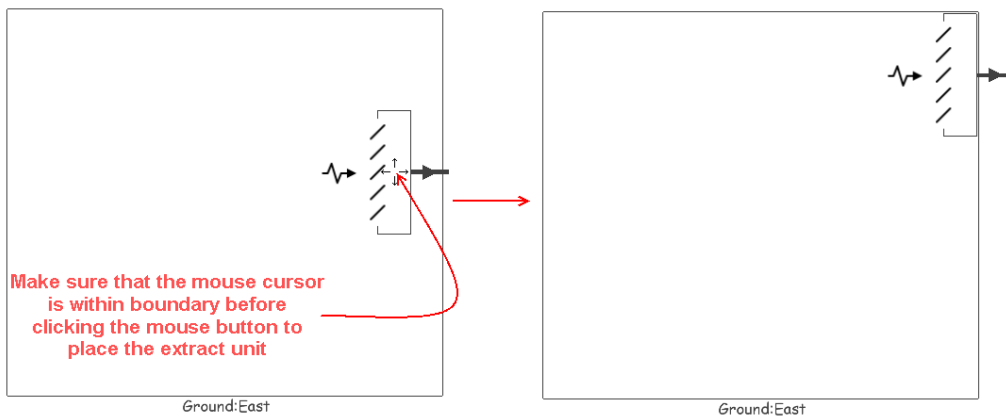
Note: Water heater heat pumps can also be added on the Supply side of a DHW or Auxiliary loop. See: [Add Water Heater Heat Pump](#)

Add Zone Extract Tool



Zone extracts are placed at the HVAC zone level and are used to return air back to the AHU. To place an extract, navigate to the HVAC zone, click on the **Add zone extract tool**. Then, making sure that the cursor is inside the boundary of the zone, left click with the mouse to place the extract.

The extract unit will be automatically positioned near the top right of the zone:



Note: A zone extract is required in an [HVAC Zone](#) if an [ADU](#) has been placed.

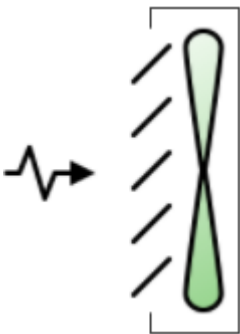
The Zone extract does not require any data to be entered and so does not appear on the Navigator list. It is not to be confused with the [Zone exhaust fan](#) which is used to extract air direct to outside and can be linked to the Airflow Network.

Add Zone Exhaust Fan Tool



Zone exhaust are placed at the HVAC zone level and are used to extract air to outside. To place an extract, navigate to the HVAC zone, click on the **Add zone exhaust fan** tool and then making sure that the cursor is inside the boundary of the zone just click the mouse button to place the extract.

The exhaust fan will be automatically positioned at the top right of the HVAC zone:

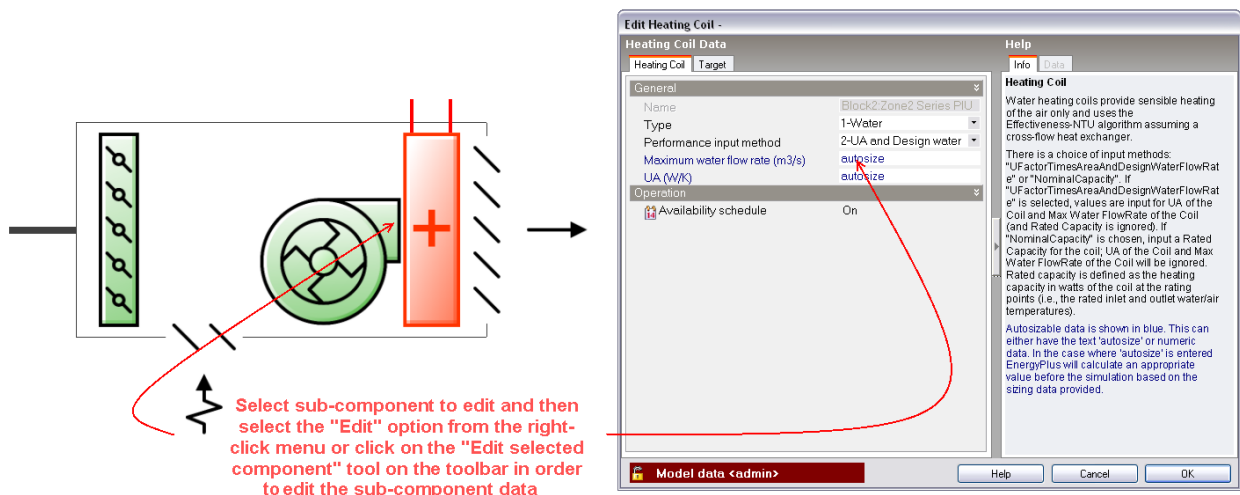
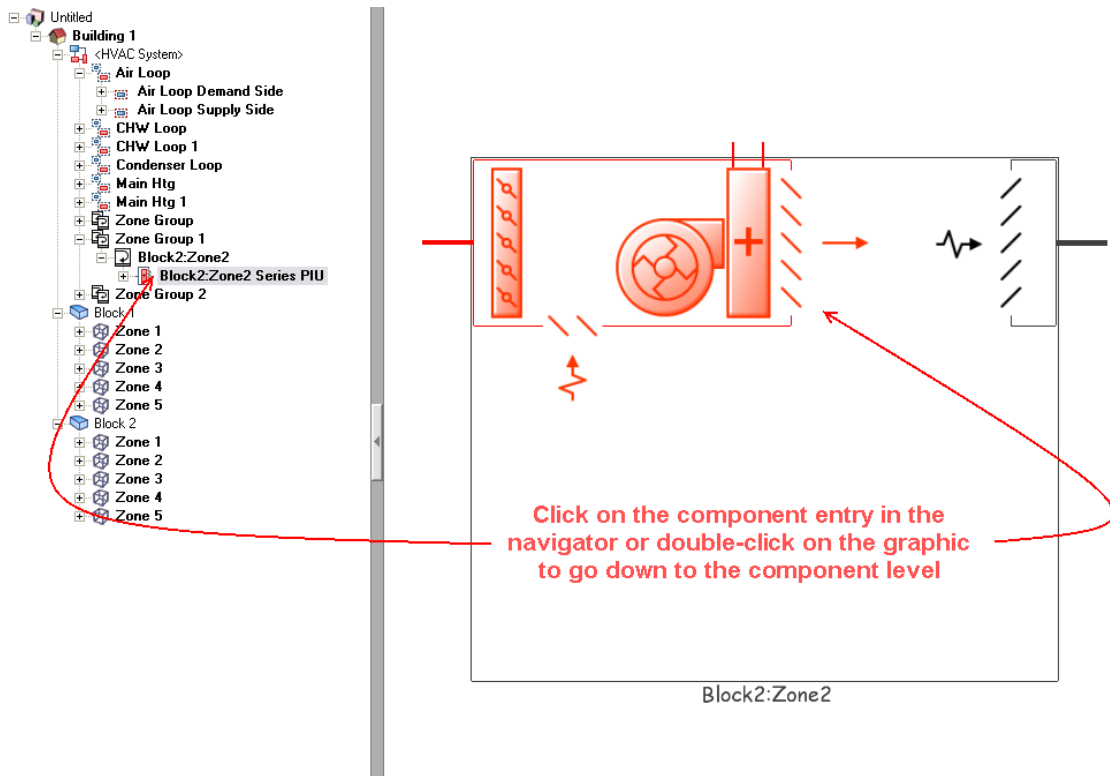


Note: A zone exhaust fan can be added in addition an extract in an [HVAC Zone](#) if an [ADU](#) has been placed. Alternatively it can included without an ADU and extract.

Data for the zone exhaust fan data can be entered on the [Zone exhaust fan](#) dialog. It is not to be confused with the Zone extract which is used to return air back to the AHU.

Working At HVAC Component Level

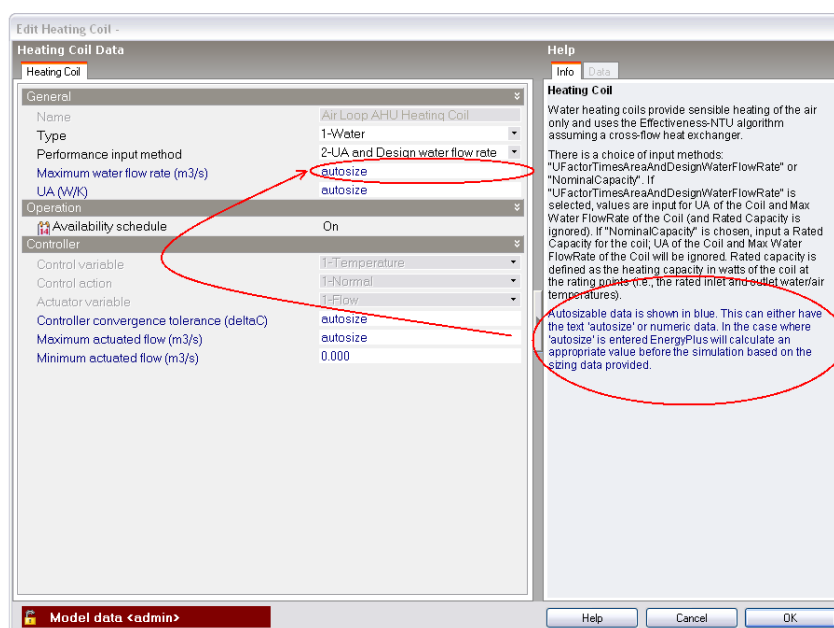
Many components in DesignBuilder HVAC are composite components and are themselves composed of a number of sub-components. Examples of composite components include air handling units and various items of zone equipment such as forced air units (fan-coil units, etc.) and air distribution units (VAV terminal boxes with reheat, etc.). In order to access and edit the data associated with each sub-component of a composite component you first need to go to the component level (by selecting it from the navigator or double-clicking on it with the mouse) and then you can select and edit any of the appropriate sub-components:



Autosizing HVAC Components

In many cases, HVAC component sizes can be autosized by EnergyPlus. All settings that can be autosized are displayed in dark blue on the component edit dialogs. The special keyword **Autosize** must be typed into the edit control on the component edit dialog if a setting is to be autosized.

For example, for a [hot water coil](#), the Maximum water flow rate can be autosized:



Autosizing In EnergyPlus

EnergyPlus is able to automatically size many HVAC components based on user-specified building configuration and design external conditions.

By default DesignBuilder automatically provides data on 2 sizing periods, one for winter, the other for summer for sizing heating and cooling equipment respectively. If [multiple summer design months](#) have been selected these will all be used in the summer simulation cooling autosizing as well. EnergyPlus uses this design day weather data to specify outside conditions when autosizing HVAC components.

Information on how zone load sizing is to be carried out can be entered for each zone (on the [HVAC zone dialog](#)). Also data governing how air loops are to be sized can be entered for each air system on the [Air loop dialog](#). Likewise sizing data for plant and condenser loops is entered on [Plant loop dialogs](#).

Other than zone thermostat setpoints, the sizing calculations generally know nothing about the system control inputs defined on [Setpoint managers](#) and availability schedules. You must coordinate the sizing inputs on the HVAC zone and air loop dialogs with the actual simulation control inputs.

The sizing calculations only recognize the presence of central heating and cooling coils, preheat and precool coils and reheat coils. These are assumed to deliver the various supply temperatures specified in the [Air loop](#) and [HVAC Zone](#) objects. The impact of their sub-components such as heat recovery, dehumidifiers, fans, and pumps are not accounted for in the sizing calculations.

Note: The Detailed HVAC autosizing calculations referred to on this page are not to be confused with [Heating](#) and [Cooling design](#) calculations which use separate simpler data entry from the [HVAC tab](#). **No Detailed HVAC data is used for Heating and Cooling design calculations**, which are based on the EnergyPlus Ideal loads system.

Schedules and Sizing

Close attention should be paid to [Schedules](#). When using [7/12 schedules](#) you must make sure that the appropriate [Design day definition](#) is made to ensure that they reflect appropriate summer and winter design days respectively. Likewise [Compact schedules](#) provide the **SummerDesignDay** and **WinterDesignDay** day types and these must be set up correctly.

Some commonly used applications of summer and winter design days in schedules are:

1. Setting internal loads (lights, equipment, occupancy) to maximum all day for cooling and to zero all day for heating;
2. Setting heating and cooling thermostat set points to constant values (no set up or set back);
3. Setting heating and cooling equipment to be always on for design days.

None of these applications are necessarily recommended but these and other uses of the special summer/winter design day schedules may prove useful for specific situations.

Some examples of common mistakes with design day data in schedules are:

1. Using a schedule such as the predefined **On** schedule for operating internal gains equipment such as [computers](#) or [lights](#). Internal gains must not be included in heating sizing calculations because the size of the heating equipment must be calculated without consideration of any contribution from internal gains. Including such gains usually results in undersizing of heating zone equipment and/or plant. Internal gains must of course be included in cooling sizing so should operate for the **SummerDesignDay**.
2. Selecting a [Heating operation schedule](#) on the [HVAC tab](#) with options indicating the schedule to be off for the **WinterDesignDay**. Or selecting a [Cooling operation schedule](#) on the [HVAC tab](#) with options indicating the schedule to be off for the **SummerDesignDay**. Likewise when using the [2-Detailed HVAC activity data](#) model option the heating and cooling setpoint schedules selected on the HVAC zone dialog must operate on their respective design day periods.

Sizing Factors

There are generally 2 areas of input where the user can impose *sizing factors*.

1. In [HVAC zone sizing data](#), the user can specify heating and cooling sizing factors for a specific zone. These factors are applied to the calculated zone design loads and air flow rates.
2. For some plant components (all central [chillers](#), [boilers](#) and [cooling towers](#)) the user can specify a sizing factor that modifies the autosized component capacity and flow rates. These factors are applied after zone sizing factors. They are primarily used to split the design load between multiple components. These sizing factors can change the autosizing of the associated loops and pumps. The following rules define the effect of plant component sizing factors on loops and pumps.
 - a. For supply side branches, the sizing factors of all components in series on the branch are summed and the result becomes the branch sizing factor. If there is a branch pump, its autosized design flow rate is multiplied by the branch sizing factor.
 - b. For each loop, if the average of the branch sizing factors is less than 1, the loop sizing factor is set equal to the sum of the branch sizing factors. If the average is greater than 1, the loop sizing factor is set equal to the maximum of the branch sizing factors. The loop sizing factor is applied to the loop design flow rate (if autosized) and to the loop pump flow rate (if autosized).

Mixing User-Specified and Autosized Inputs

Mixed user-specified and autosized inputs can be successfully used if the following points and suggestions are followed.

1. Each component is autosized independently. Thus user input for a flow rate in one component will have no effect on other components' autosized flow rates. For instance, specifying the chilled water loop pump's rated flow rate will have no effect on the autosizing of the chiller's design evaporator flow rate or on the plant loop's autosized maximum loop flow rate.
2. Within a component it is best to autosize all inputs or enter specified values for all inputs. For example, in a chiller, if only the nominal capacity is user-specified, the autosized chilled water flow rate may not be consistent with the specified capacity.
3. Sizing information flows only from the sizing data on the [HVAC zone](#), [Air loop](#) and [Plant loop](#) dialogs to the components. The sizing calculations have no knowledge of user-specified values in a component. The only exception to this rule is that plant loop sizing will collect all component design water flow rates whether autosized or user-specified.
4. To specify a particular zone or system air flow rate use the [HVAC Zone](#) and [Air Loop](#) data rather than in the individual components.
5. The plant loop flow rates are sized from the total design demand of the components connected to each loop. The components demanding water need not be autosized for the plant loop autosizing to work successfully. So the user could specify all the air side components and autosize all the plant loops and

plant components. Or specify the chilled water loop flow rate, chilled water pump inputs and chiller inputs and let the condenser loop and tower autosize.

Zone design flow rate inputs

In EnergyPlus the autosizing calculations start with a calculation of the zone design air flow rates using zone by zone design day simulations. The resulting zone design air flow rates and daily air flow sequences are used in the subsequent HVAC and central plant air and fluid flow design calculations and in the component autosizing calculations. The user can override or change the calculated zone design air flow rates in various ways.

1. The user can specify a separate zone level [Zone sizing factor](#) for [heating](#) and [cooling](#) for each [HVAC zone](#) object.
2. For each zone the user can input a [Cooling design air flow rate](#) and/or a [Heating design air flow rate](#) (and specify [Cooling design air flow method](#) = **1-Flow/Zone** and [Heating design air flow method](#) = **1-Flow/Zone**). These user inputs override the calculated values. The program divides the user input cooling or heating design air flow rate by the calculated values and uses the result as a zone sizing factor to multiply all the elements in the design heating and cooling air flow and load sequences. From this point the design calculations proceed as usual.

System design flow rate inputs

Using the results of the zone design air flow rate calculation (including any user input or altered flow rates) EnergyPlus proceeds to calculate central air system flow rates and cooling and heating loads. The results of this calculation can be overridden by setting the Air loop [Cooling design air flow method](#) to be **2-Flow/System** and input a value for [Cooling design air flow rate](#). Similarly for heating specify the [Heating design air flow method](#) to be **2-Flow/System** and input a value for [Heating design air flow rate](#).

HVAC Sizing Tips

To help achieve successful autosizing of HVAC equipment, note the following general guidelines:

- Begin with everything fully autosized (no user-specified values) and get a working system before trying to control any specifically sized components.
- You must coordinate system controls with sizing inputs. For example, if the Air loop [Central cooling design supply air temperature](#) is set to 13°C, you must make sure that the corresponding setpoint manager for the central cooling coil controls to the same 13°C as design conditions. EnergyPlus does not cross-check these inputs. The HVAC autosizing calculations use the sizing information in the [Air loop](#) and [HVAC zone](#) data. The simulation uses the information in controllers and setpoint managers.
- User-specified flow rates will only impact the sizing calculations if entered in the [Air loop](#) and [HVAC zone](#) data. Sizing information flows only from the sizing objects to the components. The sizing calculations have no knowledge of user specified values in a component. The only exception to this rule is that plant loop sizing will collect all component design water flow rates whether autosized or user specified.
- The zone thermostat schedules determine the times at which design loads will be calculated. All zone-level schedules (such as lights, electric equipment, infiltration) are active during the sizing calculations (using the day type specified for the sizing period). System and plant schedules (such as availability managers and component schedules) are unknown to the sizing calculations. To exclude certain times of day from the sizing load calculations, use the thermostat setpoint schedules for **SummerDesignDay** and/or **WinterDesignDay**. For example, setting the cooling setpoint schedule to 99°C during night-time hours for the **SummerDesignDay** day type will turn off cooling during those hours.

Differences Between Autosizing Simulations And Cooling Design Calculations

In general the zone heating and cooling loads displayed in the "HVAC Sizing Summary" table for the autosizing simulations should match closely with the equivalent values displayed on the Cooling design screen on the Summary tab. However some differences may be observed due to one or more of:

1. When using Detailed HVAC and the [Detailed HVAC activity data](#), the simulation setpoints and schedules are defined separately on the HVAC zone dialog and can easily be different to the setpoints

- defined on the Activity tab and to the schedules defined on the HVAC tab. When using the default Simple HVAC activity data option this difference is much less likely to occur.
2. The mechanical ventilation settings are defined on the HVAC tab for heating and cooling design calculations but they are defined on the Air loop dialog for autosizing simulations.
 3. The conditions of the supplied air is defined on the HVAC zone dialog for autosizing simulations whereas the data on the HVAC tab under [Supply air conditions](#) headers is used for heating and cooling design calculations.
 4. Cooling design calculations are run using 2 timesteps per hour whereas the cooling autosizing simulations are run using the number of timesteps selected for the simulation. The timestep difference doesn't arise for heating design where the inside and outside conditions are static.
 5. The **WinterDesignDay** settings control the operation for the heating autosizing simulations, whereas for Heating design calculations, fixed values are always used and there is less scope for obtaining "surprising" results. E.g. there is no way to include internal gains from occupants and equipment, whereas it is possible to include these gains (deliberately or inadvertently) in heating autosizing simulations through compact schedule settings.
 6. When the Calculated natural ventilation model option is selected this is used in autosizing simulations (if the relevant schedules allow it) whereas for Cooling design calculations the scheduled natural ventilation settings are used (if natural ventilation is enabled).
 7. Heat recovery can be activated or not for Cooling design calculations whereas it is *not* included when sizing zone loads in autosizing simulations.

Humidity Control

It is usually easiest to select an appropriate HVAC Template as the starting point when setting up humidity control for Detailed HVAC systems. However in some cases you may need to define humidity control manually and this page describes the steps involved.

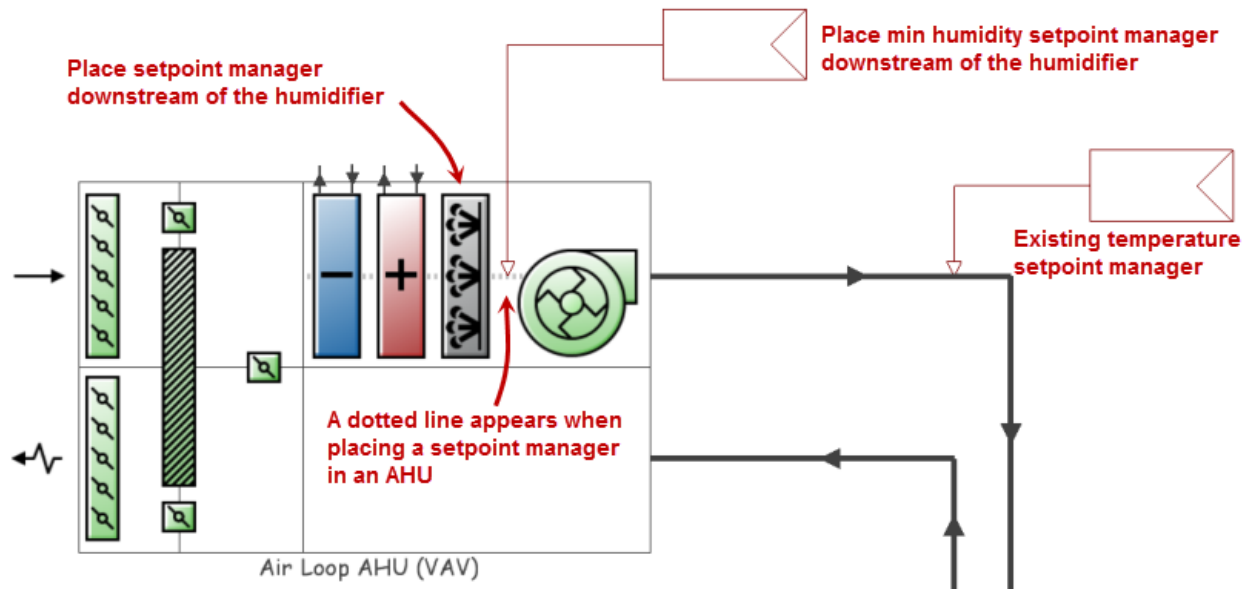
The process depends on the base system type. The instructions below are for the [Simple HVAC activity data](#) option. When the [Detailed HVAC activity data](#) is used the process is similar but [humidistat setpoints](#) come from schedules defined on the HVAC zone dialog instead of [setpoint values](#) defined on the Activity tab and demand schedules on the HVAC zone dialog.

VAV With Reheat

To add humidification/dehumidification control to a VAV system, follow these steps:

Humidification

1. Add a [humidifier](#) to the AHU, typically as the rightmost component in the AHU (nearest to the supply end).
2. Set any sizing and capacity parameters on the [humidifier dialog](#).
3. On the supply loop level of the air loop add a [setpoint manager](#) *immediately downstream* of the humidifier and edit the setpoint manager type to use one of the "humidity minimum" setpoint managers. For example the [6-Multi-zone humidity minimum controller](#) calculates the supply air minimum humidity ratio based on the zone with the highest humidity ratio setpoint to maintain the zone RH levels near their respective humidifying setpoints. Setting up the setpoint manager in this way ensures that air leaving the humidifier has a minimum level of humidity required to meet the setpoint in the zone with the lowest RH. The screenshot below illustrates the components involved.



- When using one of the "humidity minimum" controllers you must switch on the [humidistat](#) option on the [HVAC zone](#) dialog and define RH setpoint schedules. Humidistats can provide a fine degree of humidification control within each zone. To define RH setpoint schedules enter the zone [Humidification setpoint](#) model data and [Humidifying demand schedule](#) on the HVAC zone dialog (when using Simple HVAC Activity Data) or the HVAC zone [Humidifying RH setpoint schedule](#) (when using Detailed HVAC Activity Data).
- An alternative approach which does not provide close control of zone humidity is to use a [1-Scheduled](#) Setpoint manager type and the [6-Minimum humidity ratio](#) Control variable. This simply ensures that the supply air has a minimum RH. In this case the HVAC zone humidifying setpoint settings are not used.

Dehumidification

- On the AHU [cooling coil](#) dialog when using water coils, to control both temperature and humidity, set the [Controller control variable](#) to **3-Temperature and Humidity Ratio**, or if no temperature control is required, select the **2-Humidity Ratio option**.

Edit Water Cooling Coil -

Water Cooling Coil

Cooling Coil

General	>>
Operation	>>
Controller	>>
Control variable	3-Temperature and humidity ratio
Control action	2-Reverse
Actuator variable	1-Flow
Controller convergence tolerance (deltaC)	Autosize
Maximum actuated flow (m3/s)	Autosize
Minimum actuated flow (m3/s)	0

Edit Water Cooling Coil -

Water Cooling Coil

Cooling Coil

General	>>
Operation	>>
Controller	>
Control variable	2-Humidity ratio
Control action	2-Reverse
Actuator variable	1-Flow
Controller convergence tolerance (deltaC)	Autosize
Maximum actuated flow (m3/s)	Autosize
Minimum actuated flow (m3/s)	0

- Where a DX cooling coil is to conduct both temperature and humidity control, set the Coil Type to [2-Two stage with dehumidification](#) and set the [Dehumidification control type](#) to **2-Multi mode** or **3-Cool reheat**. For all dehumidification controls, one of the maximum humidity setpoint managers on the control node are used. The temperature and humidity setpoint managers must both be located on the same node. Assuming that the default temperature setpoint manager is already placed downstream of the AHU, add a new [setpoint manager](#) downstream of the AHU and edit the setpoint manager [Type](#) to use one of the "humidity maximum" setpoint managers. For example the [7-Multi-zone humidity maximum](#) controller calculates the supply air humidity ratio based on the zone with the lowest humidity ratio setpoint to maintain the zone RH levels near their respective humidity setpoints. Setting up the setpoint manager in this way ensures that air leaving the cooling coil has a maximum level of humidity, but on its own does not provide close control humidity in the zones.

Edit DX Cooling Coil -

DX cooling coil

DX Cooling Coil

General	
Name	Air Loop AHU Unitary Cooling Coil
Coil type	2-Two stage with dehumidification
2017 Rated evaporator fan power per volume flow rate (W/(m3/s))	773.300
2023 Rated evaporator fan power per volume flow rate (W/(m3/s))	934.400
Number of capacity stages	1
Dehumidification control type	3-Cool Reheat
☒ Enhanced dehumidification mode	
Operation >>	
Normal Mode Stage 1 >>	
Dehumidification Mode Stage 1 >>	
Crankcase Heater >>	
Run Settings >>	
DOAS Settings >>	

- To provide a finer degree of dehumidification control within each zone, you must switch on the [humidistat](#) option on the [HVAC zone](#) dialog and define any time-varying demand schedules.

Edit HVAC Zone -

HVAC Zone

General Sequence Target

General >>

Thermostat Schedules >>

Comfort PMV Setpoint Schedules >>

Humidistat Control >

☒ Humidistat control

Humidifying RH setpoint schedule Relative humidity setpoint schedule

De-humidifying RH setpoint schedule Relative humidity setpoint schedule

CO2 and Contaminant Control >>

Zone Air Distribution Effectiveness >>

Sizing >>

- For all dehumidification systems currently supported by DesignBuilder it is essential to include some form of zone reheat to ensure that "overcooled" air from the AHU does not cause the zone to be overcooled. The zone reheat can be a standard [reheat ADU](#) or it can be some other form of [zone heater](#) such as a convector or radiator.

Note: The temperature setpoint managers [11-Single Zone Heating](#), [12-Single Zone Cooling](#), [13-Multi Zone Cooling Average](#) and [14-Multi Zone Heating Average](#) are not suitable for humidity control. For humidity control applications, use the [1-Scheduled](#), [2-Warmest](#), [3-Coldest](#), or [10-Outdoor Air Reset](#) instead.

Note: In cases where a VAV system uses a *water* cooling coil for dehumidification in the AHU, it is sometimes not possible to control upper zone humidity levels accurately. This situation is more likely under low load conditions when the VAV terminal box dampers are at minimum position and the water cooling coil cannot drop the air stream temperature to the level required for high humidity control. In such cases you may find that swapping the water coil for a [Cooling Coil - Two Stage with Humidity Control DX](#) helps because the lower coil temperatures and humidity control capability of this DX coil allow it to drop out more moisture from the air stream, helping to maintain the zone maximum humidity setpoint. Another possible solution to this problem is to increase the minimum flow fraction for the ADU to a higher than normal value, e.g. 0.5. This allows a greater volume of dry air from the AHU to be introduced into the zones, even when sensible loads are low.

The Humidification and Dehumidification set ups described above can be used together to achieve full humidity control.

HVAC Templates Illustrating Humidity Control

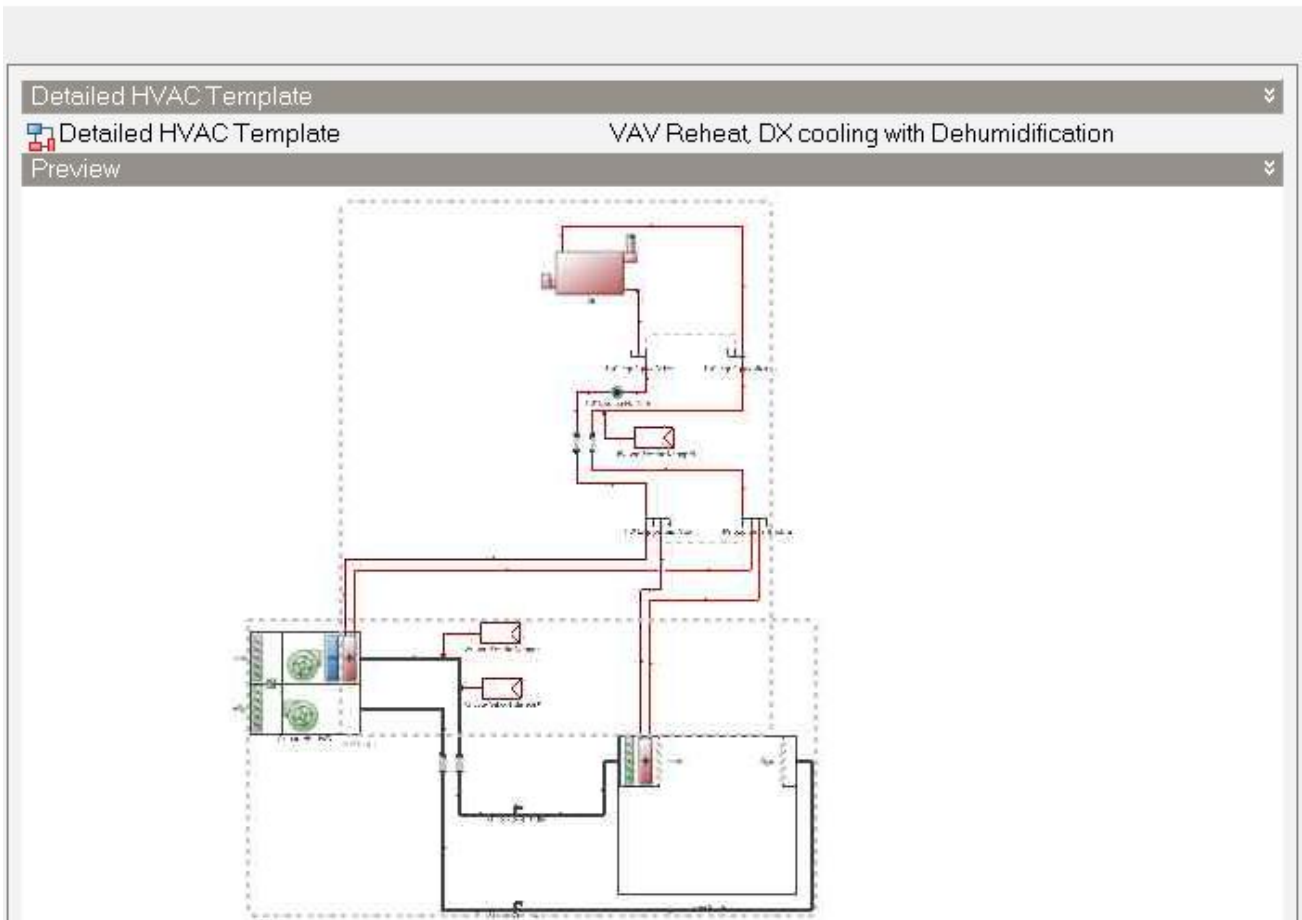
Three Detailed HVAC templates demonstrate how to model VAV systems with humidity control using different cooling approaches:

VAV Reheat, DX cooling with Dehumidification

This template uses a VAV reheat air handling unit with direct expansion (DX) cooling and dedicated dehumidification. The DX coil cools and dehumidifies the air, while a boiler supplies hot water to the reheat coils to maintain supply air temperature. The system includes separate setpoint managers for temperature and humidity, delivering conditioned air to the zone through a ducted VAV configuration.

HVAC Template Selection

Select an HVAC System

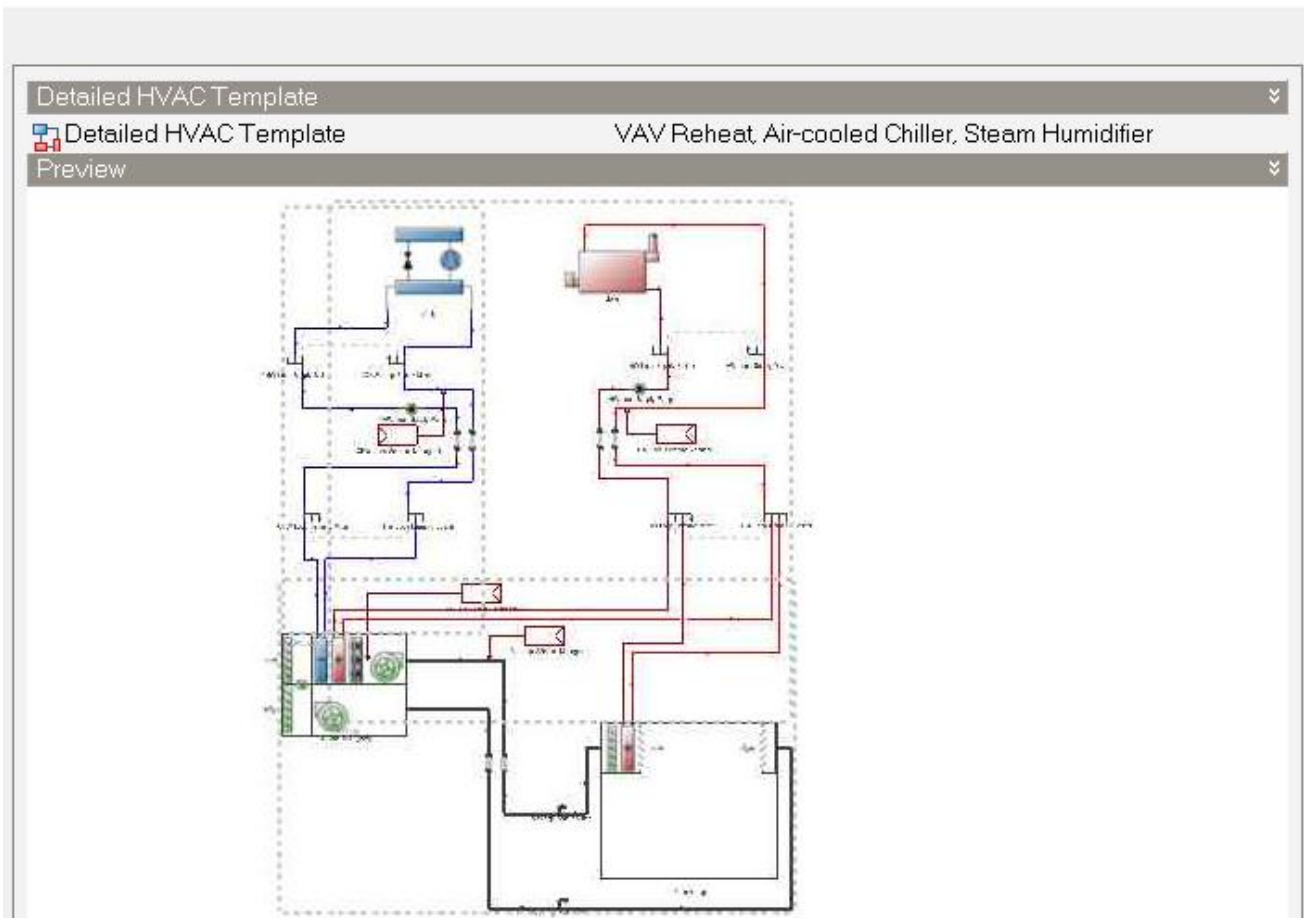


VAV Reheat, Air-cooled Chiller, Steam Humidifier

This template features a Variable Air Volume (VAV) reheat air handling unit with both cooling and humidity control. An air-cooled chiller supplies chilled water for cooling, while a boiler provides hot water for reheat coils. A steam humidifier is included in the air loop to maintain humidity levels. The air loop distributes conditioned air to the zone, ensuring both temperature and humidity control.

HVAC Template Selection

Select an HVAC System

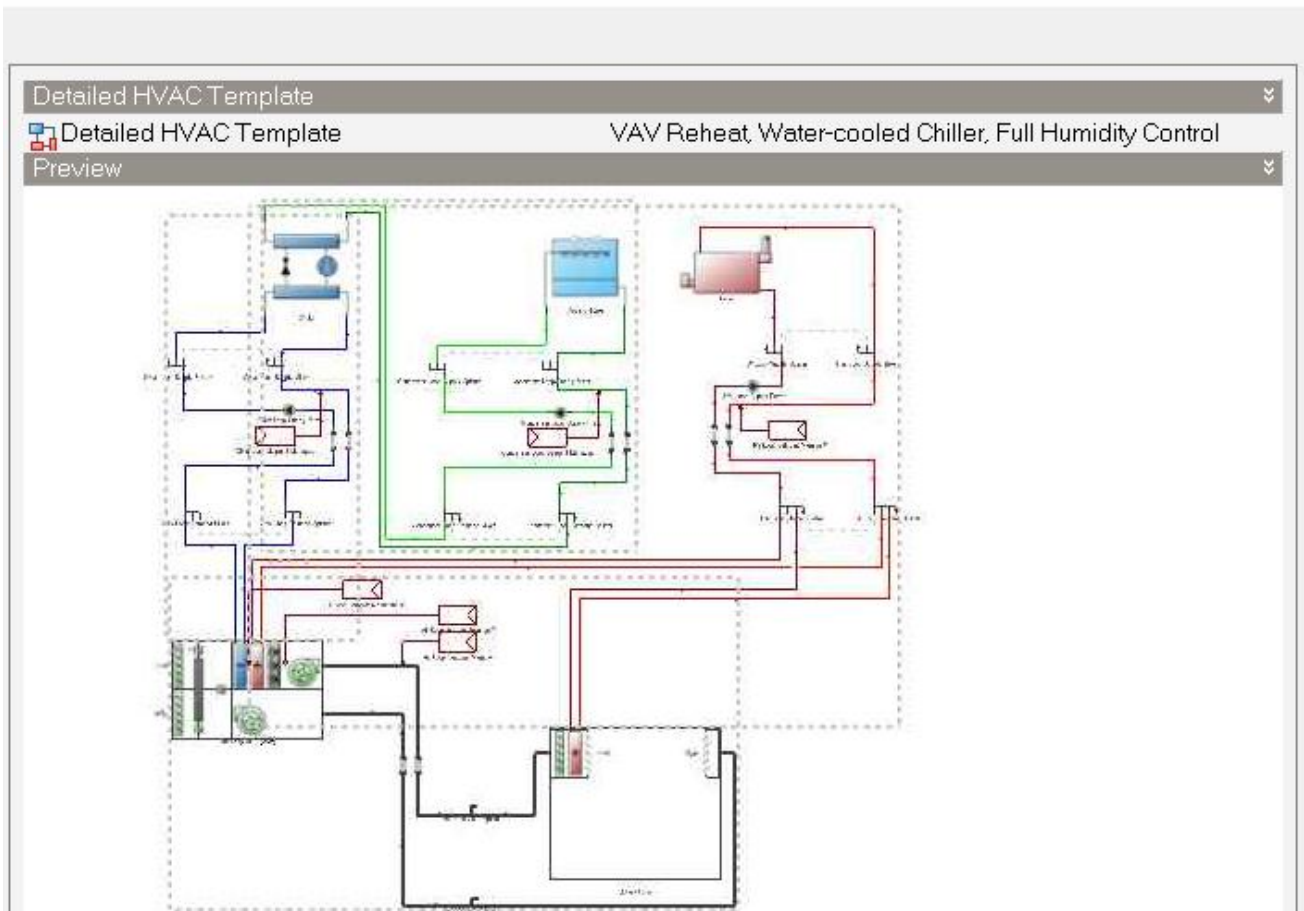


VAV Reheat, Water-cooled Chiller, Full Humidity Control

This template demonstrates how to set up a VAV reheat system with water coils for full humidity control (humidification and dehumidification).

HVAC Template Selection

Select an HVAC System



To load an HVAC template, first go to the [HVAC System level](#) then click on the [Load HVAC Template](#) tool.



CAV With Reheat

The procedure for setting up humidity control in CAV systems is similar to that described for VAV systems above, except that there is less likely to be issues with achieving dehumidification setpoints during times of low sensible load. This is because there is no damper to reduce flow at the terminal unit so zones get access to the full flow of air from the AHU.

Unitary Heat Cool And Unitary Heat Pump Humidity Control

To add humidification/dehumidification control to a unitary heat cool system, follow these steps:

1. For each zone supplied by the unitary heat cool AHU, switch humidity control on by checking the [Humidistat control](#) check box on the HVAC zone dialog and define any time-varying demand schedules.

Dehumidification

1. Go down to the air loop supply sub-loop and edit the [AHU](#), changing the [Dehumidification control type](#) to **2-CoolReheat**.

2. Add a Maximum humidity [setpoint manager](#) downstream of the AHU (**5-Multi-zone maximum humidity average** or **7-Multi-zone humidity maximum** or **9-Single zone humidity maximum**).

Humidification

1. Add a [Humidifier](#) component to the AHU.
2. Add a Minimum humidity [setpoint manager](#) immediately downstream of the Humidifier (**4-Multi-zone minimum humidity average** or **6-Multi-zone humidity minimum** or **8-Single zone humidity minimum**).

HVAC Components

A range of component types are available with Detailed HVAC:

- "Loop Data" on page 727
- "Air Loop Components" on page 787
- "Plant Loop Heat Pumps" on page 862
- "Air to Water Heat Pump / ASHP Data" on page 884
- "Performance Curves" on page 896
- "Editing Zone Group Data" on page 924
- "Editing HVAC Zone Data" on page 926
- "Setpoint Manager" on page 938
- "DHW" on page 948
- "Solar Hot Water" on page 973
- "Plant and Condenser Loop Equipment Data" on page 978
- "Zone Unit Data" on page 1112
- "Heating and Cooling Coil Data" on page 1212
- "Generators" on page 1247

Air Loop Components

The following air loop components can be placed on the supply side of an air loop:

- "Air Handling Units" on page 787
- "Heating Coils " on page 1212
- "Cooling Coils" on page 1228
- "Plenum Data" on page 827
- "Fans" on page 828
- "Humidifier - Steam Electric" on page 844
- "Evaporative Cooler Data" on page 846

Air Handling Units

Air handling units (AHUs) are components that are automatically included in the supply sub-loops of all air loops. AHUs are composite units incorporating all of the components that comprise central air handling plant such as outdoor air mixers, fans, heating/ cooling coils and humidifiers. There are two main types of AHU in DesignBuilder HVAC, generic and unitary AHUs. The main difference between the two units is that unitary AHUs incorporate an EnergyPlus unitary system such as air-to-air heat pump or heat/cool unit.

To edit the data associated with an AHU, you first need to be at the supply sub loop level of the Air loop. Select the AHU component by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the Edit selected component tool from the toolbar.

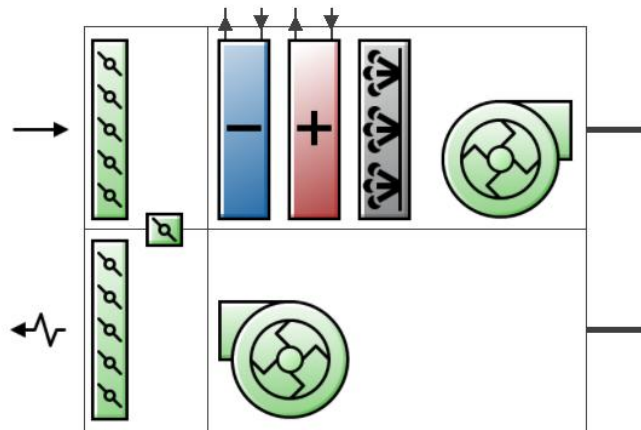
The following AHU types are available:

- [Generic AHU](#)
- "Unitary Generic AHU" on page 808
- [Unitary Heat Cool AHU](#)
- [Unitary Air to Air Heat Pump AHU](#)
- "Unitary Water To Air Heat Pump AHU" on page 822
- "DOAS AHU" on page 826

See the [Adding air loops and AHUs](#) tutorial

Generic Air Handling Unit (AHU)

The basic generic AHU incorporates an outdoor air system and a supply fan. There are also options for night cycle operation, a heat recovery system, an economiser, pre-heat and pre-cool coils and an extract fan. Additional components such as heating coils, cooling coils and humidifiers may also be added to generic AHUs.



See the [Adding air loops and AHUs](#) tutorial

General

Name

This is a unique, read-only label that is automatically generated by the software and which incorporates the name of the air loop in which the AHU is located.

Fan type

This setting is used to indicate a Constant Air Volume (CAV) or Variable Air Volume (VAV) system. It can take one of two values:

- **1-Constant volume** or
- **2-Variable volume.**

Note 1: Once an air loop has been connected to a zone ADU, the **Fan type** setting is fixed (greyed out) preventing an invalid connection and remains greyed out until the air loop is disconnected from all ADUs. So to change an air loop from VAV to CAV or vice-versa using this control you must first disconnect the demand side of the air loop from all zone ADUs. You can do this by deleting connecting branches.

Note 2: Once the AHU fan type has been set, the air loop containing the AHU can only be connected to valid [zone air distribution units](#) (ADU), i.e. CAV air loops can only be connected to CAV ADUs and similarly VAV systems can only be connected to VAV ADUs.

Design supply air flow rate

This is the air loop primary air design volumetric flow rate (in m³/s or ft³/min). This setting is auto-sizeable.

Operation

Availability schedule

This schedule contains information on the availability of the AHU for operation. A schedule value greater than 0 (1 is usually used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

When this unit is unavailable (schedule values of 0) then all equipment contained within this AHU will also be switched off.

Night Cycle

This setting can be used to enable cycling of an air system when one or more zones become too hot or too cold. A common requirement for this mechanism is where the AHU is turned off at night. However if the building gets too cold there might be condensation on the walls and other damage. So control systems are usually programmed to turn the system on either if a specified control thermostat or any thermostat shows a zone temperature of less than a night time set point. Similarly, there might be a concern about a building getting too hot. Again the control system is programmed to turn the AHU back on if one or any zone temperature exceeds a night time cooling set point.

This mechanism offers flexibility in determining how the nighttime on/off decision will be made. The temperature in one specific zone may be used or the temperatures in all the zones connected to the AHU may be sampled. You can specify a temperature tolerance and a run time for the system once it switches on. There is also an applicability schedule for scheduling when this mechanism may be applied.

Applicability schedule (night cycle)

This schedule determines whether or not for a given time period this mechanism is to be applied. Schedule values greater than zero (usually 1 is used) indicate the night cycle mechanism is to be applied, whereas schedule values less than or equal to zero (usually 0 is used) denote that it is not used for this time period.

Control type

The possible inputs are:

- **1-Stay off** means the night cycle mechanism will have no effect – AHU on/off will be determined by the fan schedule
- **2-Cycle on any** means that if any zone served by the air loop incorporating this AHU has an air temperature outside the cooling or heating set points the central fan will turn on even though the fan schedule indicates the fan is off.
- **3-Cycle on control zone** means the same thing except that the temperature in only the specified control zone is used.
- **4-Cycle on any zone fans only** is the same as **2-Cycle on any** except that only the zone fans are cycled on and the AHU fan is left off.

Important Note: Plant loops and any required zone units must be available during the night cycle by amending the availability schedules appropriately. The night cycle availability manager only automatically overrides the air loop fan schedule.

Thermostat tolerance

This is the amount (in °C or °F) by which the zone temperature must exceed the cooling set point or fall below the heating set point in order for the night cycle mechanism to signal that the system should turn on.

Cycling run time

The time in seconds for which the system will run after it has cycled on. The default is 3600 seconds (1 hour).

Control zone

For the Control Type option **3-Cycle on control zone** select the control zone.

Extract Fan

Include extract fan

An extract fan is included within all AHUs by default. This setting may be used to remove the extract fan from the AHU.

Note: If the AHU is connected to a zone that incorporates a Variable Air Volume with Reheat and Variable Speed Fan ADU, the AHU extract fan should be omitted otherwise EnergyPlus will raise an error and the simulation will be aborted.

Mixed Mode

You can set up Mixed mode controls for the AHU to model optimal interaction between the natural ventilation system (Scheduled or Calculated) and the AHU.

Refer to the main HVAC tab [Mixed Mode](#) section for details on the options available.

Note: If mixed mode is activated here on the AHU dialog, then any mixed mode settings for connected zones on the HVAC tab will be overridden by these mixed mode settings.

Outdoor Air System (Common To All AHUs)

The outdoor air system incorporates an air mixer, an optional pre-cool coil, an optional pre-heat coil, an optional heat recovery device and an outdoor air controller. The purpose of the outdoor air controller is to provide outdoor air for ventilation and also provide free cooling (through additional outdoor air and/or bypassing an air-to-air heat exchanger) whenever possible. The outdoor air controller includes a number of user-selectable limit controls. If any of the selected limits are exceeded, the outdoor airflow rate is set to the minimum.

If all the limits are satisfied, the outdoor air controller does the following for continuous air flow systems: if the outdoor air temperature is greater than or equal to the mixed air temperature setpoint, the outdoor air flow rate is set to the maximum; if the outdoor air temperature is less than the mixed air temperature setpoint, the outdoor air controller will modulate the outdoor air flow so that the mixed air temperature will match the mixed air setpoint temperature.

A time-of-day schedule may also be used to simulate an increase in outdoor air flow rate for “push-button” type economiser applications. When the schedule permits (i.e., schedule values are greater than 0), the outdoor air flow rate is increased to the user-specified maximum outdoor air flow rate.

The outdoor air controller can also account for changes in the outdoor air flow rate during times when indoor humidity levels are high. A zone humidistat must be used with this control option. During high indoor humidity, the outdoor air flow rate is modified in response to a high indoor humidity condition. If high humidity control is based on the outdoor air humidity ratio and the outdoor humidity ratio is greater than the indoor humidity ratio, high humidity control is terminated. When the economiser is used in conjunction with the high humidity control option, high humidity control has priority and controls the change in air flow rates. If the AHU Night Cycle option is switched on, it has priority over high humidity control and will use the controller’s maximum outdoor air flow rate when the Night Cycle mechanism cycles the fan on.

Optional pre-cool coil, pre-heat coil and heat recovery may be specified upstream of the mixer. When this is the case, any modulation will be determined by the conditions at the inlet node of the mixer rather than the outdoor air. This means that the controller will account for the heat recovery device or pre-heating/pre-cooling coils that may modify the condition of outdoor air before it reaches the mixer. If all the limits are satisfied, the outdoor air controller does the following for cycling fan systems: the outdoor air flow rate is set to the maximum when the fan cycles on. If the limits are not satisfied, the outdoor air flow rate is at the minimum when the fan cycles on.

DOAS

Air handling unit

General
Outdoor Air System

DOAS ⌵

☒ Dedicated outdoor air supply

Recirculation ⌵

☒ On

Minimum outdoor air flow rate (m3/s) Autosize

Maximum outdoor air flow rate (m3/s) Autosize

Minimum limit type 2-Fixed minimum ⌵

Economiser (Free Cooling) ⌵

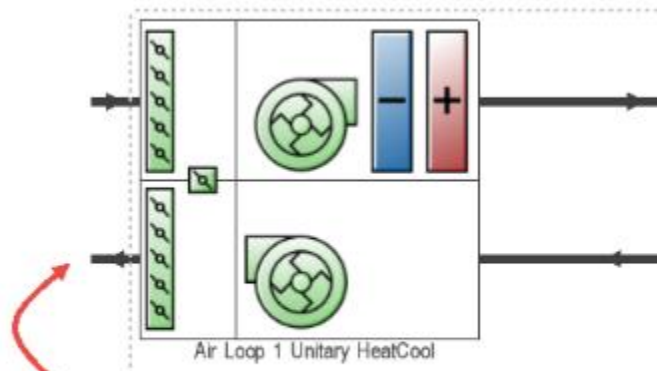
Economiser control type 1-No economizer ⌵

Outdoor Air Schedules ⌵

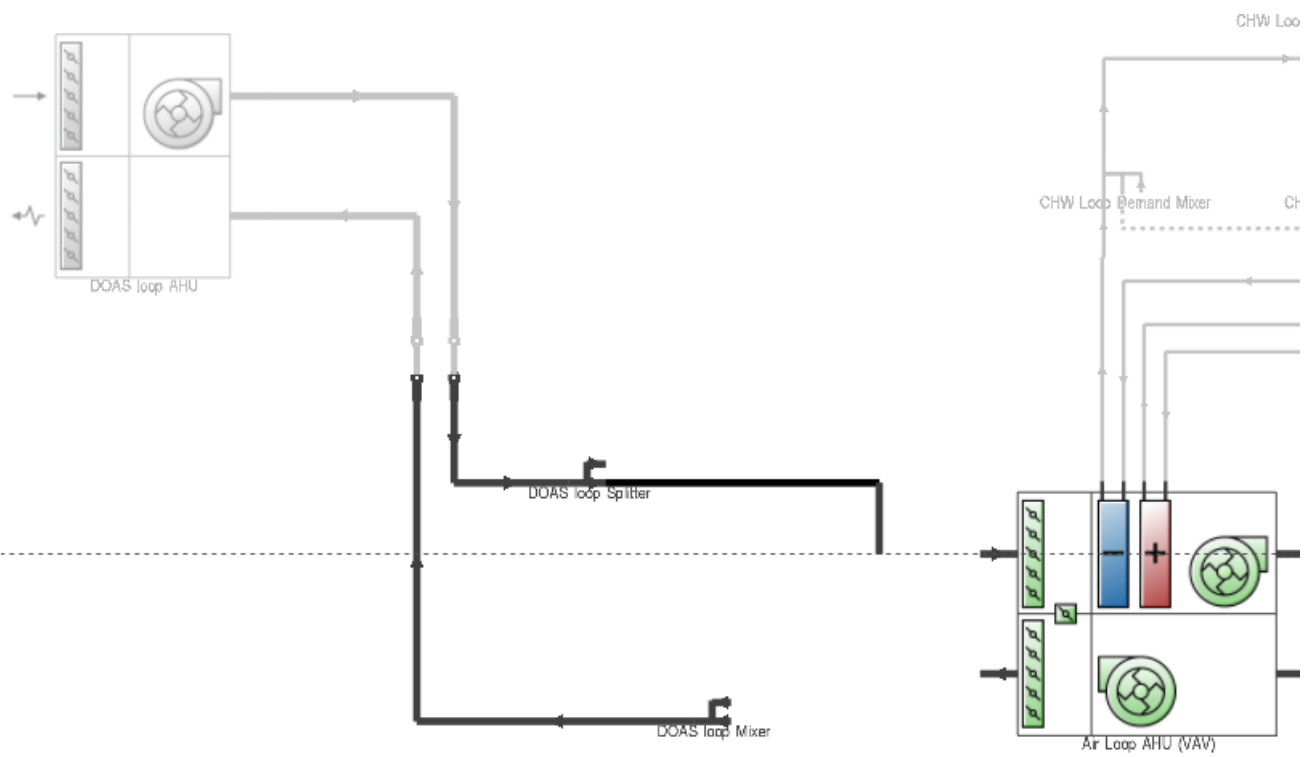
Demand Controlled Ventilation ⌵

Dedicated outdoor air supply

If this AHU is to receive outdoor air from an upstream DOAS loop then this checkbox should be checked. This enables a connection to be made between the supply side of the DOAS loop and the outdoor air connections in this AHU. See below.



DOAS Connections



Recirculation

Air handling unit

General Outdoor Air System

DOAS

☒ Dedicated outdoor air supply

Recirculation

☒ On

Minimum outdoor air flow rate (m3/s) Autosize

Maximum outdoor air flow rate (m3/s) Autosize

Minimum limit type 2-Fixed minimum

Economiser (Free Cooling)

Economiser control type 1-No economizer

Outdoor Air Schedules

☐ Apply minimum outdoor air schedule

☐ Apply minimum fraction of outdoor air schedule

☐ Apply maximum fraction of outdoor air schedule

Demand Controlled Ventilation

☐ Demand controlled ventilation

Heat Recovery

☐ On

If this option is checked, the AHU will re-circulate a proportion of the air delivered by the AHU, otherwise the system will operate as a full fresh air system.

Full Fresh Air with Economiser

If you need to model a full fresh air system with no recirculation but need some of the functionality under the Recirculation check box, such as an economiser, you should check the Recirculation checkbox and make the following settings:

- Check the [Apply minimum fraction of outdoor air schedule](#) checkbox and select the **On 24/7** schedule as the [Minimum fraction of outdoor air schedule](#).
- Check the [Apply maximum fraction of outdoor air schedule](#) checkbox and select the **On 24/7** schedule as the [Maximum fraction of outdoor air schedule](#).

The screenshot below shows how the data should look when you are finished.

Outdoor Air Schedules

☐ Apply minimum outdoor air schedule

☒ Apply minimum fraction of outdoor air schedule

 Minimum fraction of outdoor air schedule On 24/7

☒ Apply maximum fraction of outdoor air schedule

 Maximum fraction of outdoor air schedule On 24/7

These settings ensure that the fraction of fresh air is 1 at all times and that there is no recirculation.

Minimum outdoor air flow rate

This is the minimum outdoor air flow rate that the system can supply (in m3/s or ft3/min).

This value should be set to zero when using DCV and to **Autosize** when the Minimum outdoor air flow rate is to be autosized.

Maximum outdoor air flow rate

This is the maximum outdoor air flow rate for the system (in m³/s or ft³/min).

Minimum limit type

Choices for this field are:

- **1-Proportional minimum** which means the minimum outdoor airflow rate varies in proportion to the total system air flow rate
- **2-Fixed minimum** which means that the minimum outdoor airflow rate is fixed no matter what the actual system flow rate is.

Economiser

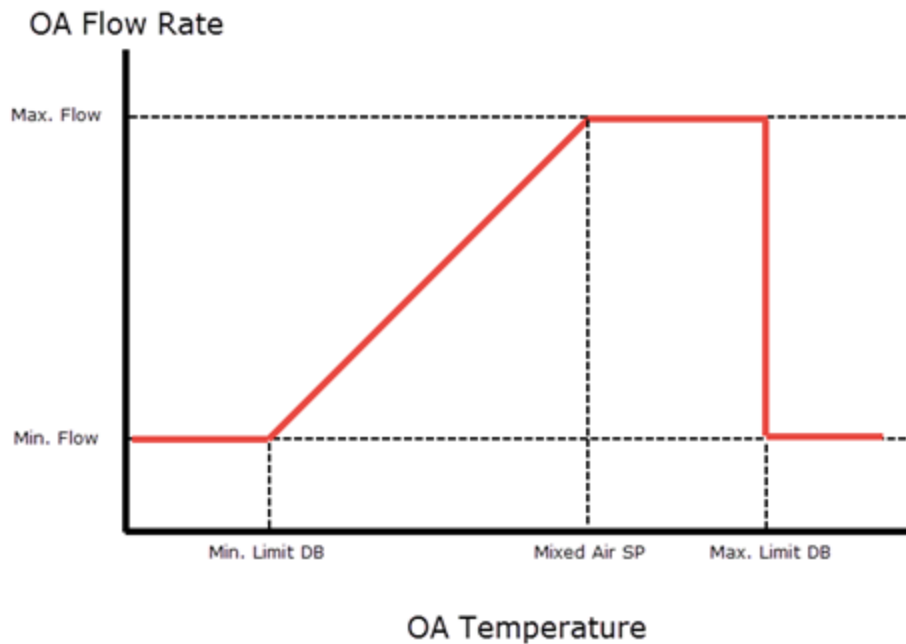
An economiser can be included to provide free cooling by increasing the flow of outdoor air into the AHU mixing box when conditions are favourable. The types of economiser available are:

- **1-No Economiser** - the economiser will not operate and the outdoor airflow rate will be at the minimum for the entire simulation.
- **2-Fixed Dry-Bulb** - the economiser will set the outdoor airflow rate at minimum if the outdoor air temperature is higher than a specified dry-bulb temperature limit.
- **3-Differential Dry-Bulb** - will trigger the outdoor airflow to minimum when the dry-bulb temperature of outdoor air is higher than the dry-bulb temperature of the return air.
- **4-Fixed Enthalpy** - checks the upper limit of the enthalpy given as a field input against the enthalpy content of outdoor air and will set the outdoor airflow rate to minimum if the latter is greater than the former.
- **5-Differential Enthalpy** does the same as **4-Fixed Enthalpy** but compares the return air enthalpy with the enthalpy of outdoor air. When the enthalpy of outdoor air is greater than the enthalpy of the return air, the outdoor air flow rate is set to minimum.
- **6-Electronic Enthalpy** enables the simulation to calculate the humidity ratio limit of outdoor air based on the dry-bulb temperature of outdoor air and a quadratic/cubic curve, and compare it to the actual outdoor air humidity ratio. If the actual outdoor humidity ratio is greater than the calculated humidity ratio limit, then the outdoor airflow rate is set to minimum.
- **7-Fixed Dew-Point and Dry-Bulb** compares both the outdoor dew-point temperature and the outdoor dry-bulb temperature to their specified high limit values. If either outdoor temperature exceeds the high limit value, the outdoor airflow rate is set to minimum.
- **8-Differential Dry-Bulb and Enthalpy** enables the control strategy to be based on both the **3-Differential Dry Bulb** and **5-Differential Enthalpy** economiser control strategies.

In addition to all economiser control types listed above, each control type checks for user-entered values for the upper limit of dry-bulb temperature, enthalpy limit, humidity ratio limit and dew-point limit. The outdoor air flow rate is set to minimum if any of these entered limits are exceeded.

Note: The economiser acts to control the *temperature* of the mixed air stream regardless of the type of economiser.

The way that the economiser air flow rate is controlled is illustrated in the diagram below.



Economiser optimiser control action

- The **Min. Limit DB** temperature on the graph above is set using the [Economiser minimum limit dry-bulb temperature](#) setting and when the **OA Temperature** is below this value, the **OA Flow Rate** is set to minimum.
- The **Mixed Air SP** temperature is obtained by EnergyPlus from a mixed air [Setpoint Manager](#) which is automatically created by DesignBuilder using the setpoint defined for the first cooling SPM downstream from the AHU air mixing box.
- When the **OA Temperature** is less than the **Mixed Air SP**, the controller modulates the OA flow rate so as to maintain the **Mixed Air SP**.
- The **Max. Limit DB** is set using the [Economiser maximum limit dry-bulb temperature](#) setting (which is only be available on the dialog for **2-Fixed Dry Bulb** and **7-Fixed Dew Point And Dry Bulb** economiser control types).
- When the **OA Temperature** is between the **Mixed Air SP** and the **Max Limit DB**, the OA controller sets the OA flow rate to maximum.
- When the OA temperature reaches the **Max. Limit DB**, the OA controller sets the **OA Flow Rate** to minimum.

When the economiser control type is set to **4-Fixed Enthalpy**, the **Max. Limit DB** would be replaced by the enthalpy value entered for the [Economiser maximum limit enthalpy](#) setting, i.e. instead of switching the OA flow to minimum when the OA temperature reaches the specified maximum, it would switch to minimum when the OA enthalpy reaches the specified maximum. The optimiser is still working though to maintain the setpoint temperature for the mixed air stream.

Note: A cooling or scheduled temperature setpoint manager must be included on the supply side of the air loop when an economiser is selected. This will typically be downstream of the AHU.

Economiser control action type

There are two choices:

- **1-Modulate flow** which means that the outdoor air flow rate will be increased to meet the mixed air setpoint temperature, subject to the limits imposed via other inputs for this object (e.g., Economizer Maximum Limit Dry-Bulb Temperature, Maximum Outdoor Air Flow Rate, etc.).
- **2-Minimum flow with bypass**, which is only available if Heat Recovery is selected. This option is used in conjunction with a heat recovery device for providing free cooling operation in the absence of a conventional air-side economizer (i.e., when outdoor air flow rate is not increased during economizer mode). It forces the outdoor air flow rate to always remain at the minimum. However, when high humidity control is used, the outdoor air flow rate is set to the product of the maximum outdoor air flow

rate multiplied by the high humidity outdoor air flow ratio. The heat exchanger uses the limit checking in the outdoor air controller to decide whether or not to bypass the outdoor air around the heat exchanger – or turn off the wheel motor in the case of a rotary heat exchanger. Heat exchange is also suspended when high humidity control is active. When **2-Minimum flow with bypass** is used and the [Time of day economizer control schedule](#) value is 0, then the flow is set to the minimum. If the time of day schedule is not used, then flow is always at minimum. If there is no economizer, the flow is always at minimum. If there is an economizer, this data either maintains flow at minimum when bypassing or not.

Lockout type

Choices for this setting are:

- **1-No lockout**,
- **2-Lockout with heating**, which means that if the packaged unit is in heating mode, the economiser is locked out – i.e. the economizer dampers are closed and there is minimum outdoor air flow.
- **3-Lockout with compressor**. This setting only applies to unitary systems with DX coils and means that in addition to locking out the economiser when the unit is in heating mode the economiser is locked out when the DX unit compressor is operating to provide cooling. In other words, the economiser must meet the entire cooling load – it isn't allowed to operate in conjunction with the DX cooling coil. The **3-Lockout with compressor** option is sometimes called a “non-integrated” economiser.

Economiser minimum limit dry-bulb temperature

This is an optional outdoor air temperature low limit (in °C or °F) for economiser operation. If the outdoor air temperature is below this limit, the outdoor airflow rate will be set to the minimum. This limit applies to the conditions at the Actuator Node regardless of whether or not there are any other components in the outdoor air path upstream of the mixer.

Economiser maximum limit dry-bulb temperature

This is an optional outdoor air temperature high limit (in °C or °F) for economiser operation. If the outdoor air temperature is above this limit, the outdoor airflow rate will be set to the minimum. This limit applies to the conditions at the Actuator Node regardless of whether or not there are any other components in the outdoor air path upstream of the mixer.

Economiser maximum limit enthalpy

This is an optional outdoor air enthalpy limit (in J/kg or Btu/lb) for economiser operation. If the outdoor air enthalpy is above this value, the outdoor airflow rate will be set to the minimum. This field is required if the Economiser type is set to **4-Fixed enthalpy**. This limit applies to the conditions at the Actuator Node regardless of whether or not there are any other components in the outdoor air path upstream of the mixer.

Economiser maximum limit dew-point temperature

This is an optional outdoor air dew-point limit (in °C or °F) for economiser operation. If the outdoor air dew-point temperature is above this value, the outdoor airflow rate will be set to the minimum. This limit applies to the conditions at the Actuator Node regardless of whether or not there are any other components in the outdoor air path upstream of the mixer.

Electronic enthalpy limit curve

This is an optional [Quadratic](#) or [Cubic](#) curve which provides the maximum outdoor air humidity ratio (function of outdoor air dry-bulb temperature) for economiser operation. If the outdoor air humidity ratio is greater than the curve's maximum humidity ratio (evaluated at the outdoor air dry-bulb temperature), the outdoor air flow rate will be set to the minimum. This limit applies to the conditions at the Actuator node regardless of whether or not there are any other components in the outdoor air path upstream of the mixer.

Time of day economiser control schedule

This is an optional schedule which controls the outdoor air flow rate based on a time-of-day economiser. Schedule values equal to 0 disable this feature. A schedule value of greater than 0 causes the outdoor air flow rate to increase to the maximum. When an economiser is used in conjunction with the high humidity control option, high humidity control has priority.

High humidity control

This setting establishes whether or not the outdoor air flow rate is modified in response to high indoor relative humidity. If **2-Yes** is selected, the outdoor air flow rate may be modified when the indoor relative humidity is above the humidistat setpoint. If **1-No** is selected, this option is disabled and the following three settings are not displayed.

Humidistat Control Zone

This is the zone name where the humidistat is located. This setting is only required when the High Humidity Control is set to **2-Yes**.

High humidity outdoor air flow ratio

This is the ratio of the modified outdoor air flow rate to the maximum outdoor air flow rate. When the high humidity control algorithm determines that the outdoor air flow rate will be changed (i.e. increased or decreased), the operating outdoor air flow rate is equal to the maximum outdoor air flow rate multiplied by this ratio. The minimum value for this setting is 0. This setting is only required when the [High humidity control](#) is set to **2-Yes**. When an economiser is used in conjunction with the high humidity control option, high humidity control has priority.

Control high indoor humidity based on outdoor humidity ratio

This setting determines if high humidity control is activated based on high indoor relative humidity alone or is activated only when the indoor relative humidity is above the humidistat setpoint and the indoor humidity ratio is greater than the outdoor humidity ratio. Valid choices are **1-Yes** and **2-No**. If **2-No** is selected, high humidity control is active any time the zone humidistat senses a moisture load. If **1-Yes** is selected, the model also verifies that the outdoor humidity ratio is less than the humidistat's zone air humidity ratio. This setting is only required when the High Humidity Control is set to **2-Yes**.

Note: While conditions may be favourable for economizer operation, it does not guarantee that the air-side economizer has increased outdoor air flow above the minimum level since the actual outdoor air flow rate is also governed by other controls (e.g., mixed air set point temperature, time of day economizer control, maximum humidity setpoint, etc).

Outdoor Air Schedules

If the minimum outdoor air flow rate is to be scheduled then use one or both of the minimum outdoor air schedule or minimum fraction of outdoor air schedule settings described below.

- **Apply minimum outdoor air schedule** - to apply a schedule whose values are multiplied by the minimum outdoor air flow rate to calculate the outdoor fresh air rate at any time during the simulation. In this case you can select the **Minimum outdoor air schedule** below.
- **Apply minimum fraction of outdoor air schedule** - to apply a schedule whose values are multiplied by the total (design) air flow rate to calculate the outdoor fresh air rate at any time during the simulation. In this case you can select the **Minimum fraction of outdoor air schedule** below.

Scheduling Variable Outdoor Air

The screenshot below shows how to change the AHU Outdoor Air Schedule settings to obtain outdoor air varying between the maximum value defined on the HVAC tab and zero based on occupancy schedule "Office_OpenOff_Occ", in the same way as it does with Simple HVAC.

Outdoor Air Schedules	
☒ Apply minimum outdoor air schedule	
 Minimum outdoor air schedule	Office_OpenOff_Occ
☐ Apply minimum fraction of outdoor air schedule	
☐ Apply maximum fraction of outdoor air schedule	

When outdoor air delivery to zones is to be controlled accurately as described above it is necessary to make 3 further settings:

1. When using the [VAV with Reheat](#) and [VAV with no Reheat](#) terminal units in the way described above then it is also necessary to select the [Control on outdoor air flow](#) option in these 2 dialogs.
2. It is also necessary to use the **2-Fixed minimum** [Minimum limit type](#) on the AHU dialog.
3. Ensure that the [Minimum outdoor air flow rate](#) is set to **Autosize** and not zero. Note that this is the opposite of the requirement for DCV.

It is important to understand that even with the above settings you won't generally obtain exact specified outdoor air flow rates *to individual zones*, however the *total fresh air supply rate provided through the AHU* should be correctly calculated as the sum of the minimum fresh requirements of all zones served by the AHU. This is because the total outdoor air requirement for each zone on the air handler is summed and used to set the minimum total outdoor air flow rate for the air handler. The outdoor air controller sets an outdoor air flow rate for the air handler at each time step during the simulation. After that, each zone receives a pro-rated share of the total outdoor air flow proportioned by the current supply flow rate to each zone, but the exact supply flow rate to each zone will be calculated based on cooling or heating demand not fresh air requirements and there is no further zone-by-zone allocation of outdoor air quantities - which is what happens in a real system.

Apply minimum outdoor air schedule

Check this checkbox if you wish to apply a **Minimum outdoor air schedule** limit in which case the schedule can be selected in the browse list below.

Minimum outdoor air schedule

When the **Apply minimum outdoor air schedule** checkbox is checked you can select this schedule which uses decimal values (0.0 – 1.0). These values are multiplied by the minimum outdoor air flow rate. This schedule is useful for reducing the outdoor air flow rate to zero during unoccupied or start up hours. If this schedule option is not used, the minimum outdoor air flow rate either remains constant during the simulation period ([Minimum limit type](#) = **2-Fixed minimum**) or varies in proportion to the supply air flow rate ([Minimum limit type](#) = **1-Proportional minimum**).

Apply minimum fraction outdoor air schedule

Check this checkbox if you wish to apply a **Minimum fraction of outdoor air schedule** limit in which case the schedule can be selected in the browse list below.

Minimum fraction of outdoor air schedule

When the **Apply minimum fraction of outdoor air schedule** checkbox is checked you can select this schedule which uses decimal values (0.0 – 1.0). These values are multiplied by the design supply air flow rate. This is an alternate method for specifying the minimum outdoor air amount. The other method is the [Minimum outdoor air schedule](#) described above.

Note: If both the **Minimum outdoor air schedule** and the **Minimum fraction of outdoor air schedule** are both selected then for each timestep of the simulation, the higher of the two minimum limits is calculated and applied. Both of these settings act by overriding the [Outdoor air flow rate](#).

Apply maximum fraction of outdoor air schedule

Check this option to limit the maximum outdoor air flow rate.

Maximum fraction of outdoor air schedule

When the above **Apply maximum fraction of outdoor air schedule** option is selected you can also select the corresponding schedule which has fractional values in the range 0.0-1.0. This schedule effectively enables you to limit the maximum amount of outdoor air into the system.

Demand Controlled Ventilation

Ventilation standards provide guidance on appropriate levels of outdoor ventilation air required for acceptable indoor air quality. For example the Ventilation Rate Procedure (VRP) of ASHRAE Standard 62.1-2007/2010 requires outdoor ventilation rates to be determined based on the floor area of each occupied zone plus the number of people in each zone and considers the zone air distribution effectiveness and system ventilation efficiency. The outdoor air ventilation rate can be reset dynamically as operating conditions change (e.g. variations in occupancy).

The VRP option can be used for calculating these outdoor air ventilation requirements and resetting them based on varying occupancy levels and zone diversification. This is particularly useful for large air distribution systems that serve a number of different zone types with varying occupancy levels.

DCV control can also be used to model the Indoor Air Quality Procedure (IAQP) as defined in Standard 62.1

Demand controlled ventilation

Check this option to include DCV controls for this AHU.

System outdoor air method

Select the method to be used to calculate the system minimum outdoor air flow. The following options are available:

- **1-Zone sum** - sums the outdoor air flows across all zones served by the system.
- **2-Standard 62.1 ventilation rate procedure** - uses the multi-zone equations defined in 62.1-2007 to calculate the system outdoor air flow. VRP considers zone air distribution effectiveness and zone diversification of outdoor air fractions.
- **3-Standard 62.1 ventilation rate procedure with limit** - is similar to **2-Standard 62.1 ventilation rate procedure**. The only difference between the two methods is that **3-Standard 62.1 ventilation rate procedure with limit** - doesn't allow the system outdoor air flow to be larger than the design value.
- **4-Indoor Air Quality Procedure (IAQP)** - the other procedure defined in ASHRAE Standard 62.1-2007 for calculating the amount of outdoor air necessary to maintain the levels of indoor air carbon dioxide at or below a setpoint. Setpoints are defined on the Activity tab ([Simple HVAC activity data](#) option) or on the HVAC zone dialog ([Detailed HVAC activity data](#) option). Outdoor air intake and other system design parameters are based on an analysis of contaminant sources, and contaminant concentration targets. EnergyPlus uses the IAQP to control carbon dioxide and allows credit to be taken for controls that can be demonstrated to result in indoor carbon dioxide and generic contaminant concentrations equal to or lower than those achieved using the VRP. The IAQP may also be used where the design is intended to attain specific target carbon dioxide and generic contaminant levels.
- **5-Proportional control based on design occupancy** - where the required outdoor air flow rate varies in proportion to the percentage of the CO2 signal range. This variant of the proportional method uses the design occupancy to calculate outdoor air rate. This is usually a more realistic control strategy for real systems, which don't have access to real time occupancy data. This option should generally be used for design stage simulations.
- **6-Proportional control based on occupancy schedule** described in Appendix A of the ASHRAE 62.1-2010 user's manual discusses another method for implementing CO2-based DCV in a single zone system where the required outdoor air flow rate varies in proportion to the percentage of the CO2 signal range. This variant of the proportional method uses the real occupancy at the current time step to calculate outdoor air rate. This is a good approach to estimate outdoor air rate in special applications where the zone controller has access to the real time occupancy information.
- **7-Indoor Air Quality Procedure (IAQP) generic contaminant** calculates the amount of outdoor air necessary to maintain the levels of indoor air generic contaminant at or below the setpoint defined on the Activity tab ([Simple HVAC activity data](#) option) or on the HVAC zone dialog ([Detailed HVAC activity data](#) option).
- **8-Indoor air quality procedure combined** uses the maximum of the required outdoor airflow for CO2 IAQP and generic contaminant IAQP. Both CO2 and generic contaminant models must be active when selecting this option.
- **9-Proportional control based on design outdoor air rate** - uses design outdoor air flow rate to calculate demand outdoor air flow rate.

Note: If either **1-Zone sum**, **2-Standard 62.1 Ventilation Rate Procedure (VRP)** or **3-Standard 62.1 ventilation rate procedure with limit** options are selected here then the [CO2 and contaminant](#) option must not be selected on the HVAC zone dialog.

Note: If either **4-Indoor Air Quality Procedure (IAQP)**, **7-Indoor Air Quality Procedure (IAQP) generic contaminant** or **8-Indoor air quality procedure combined** are selected then the [CO2 and contaminant](#) option must be selected for at least one HVAC zone connected to the AHU.

Note: When one of the proportional control methods is selected (**5-Proportional control based on design occupancy**, **6-Proportional control based on occupancy schedule** or **9-Proportional control based on design outdoor air rate**) then selection of the [CO2 and contaminant](#) HVAC zone options is optional. If zone minimum CO2 concentration settings for proportional control are to be specified then you should check it, otherwise leave it unchecked.

Note: When using DCV you should ensure that the [Minimum outdoor air flow rate](#) is set to zero and not to **Autosize**. This is the opposite of the requirement when [scheduling outdoor air flow without using DCV](#).

A [summary guide](#) on the steps required to model DCV is provided in the Modelling advice section.

For more details on how DCV is simulated in EnergyPlus see the [DCV page](#) in the EnergyPlus Engineering Reference Guide.

Zone maximum outdoor air fraction

For VAV systems, when a zone requires outdoor air higher than the user specified Zone Maximum Outdoor Air Fraction, the zone supply air flow will be increased (if damper not fully open yet) to cap the outdoor air fraction at the maximum value. This allows the system level outdoor air flow to be reduced while the total supply air flow increases to meet zone outdoor air requirement. Valid values are from 0 to 1.0. Default is 1.0 which indicates zones can have 100% outdoor air maintaining backward compatibility. This inputs work for single and dual duct VAV systems.

Availability schedule

Select the schedule which defines when DCV should operate. If the schedule's value is 0.0, then mechanical ventilation is not available and flow will not be requested. If the schedule's value is > 0.0 (usually 1 is used), mechanical ventilation is available.

Tip: This schedule can be useful for purging the building of contaminants prior to occupancy (i.e., ventilation rate per unit floor area will be provided even if the occupancy is zero).

Interaction with normal outdoor air control

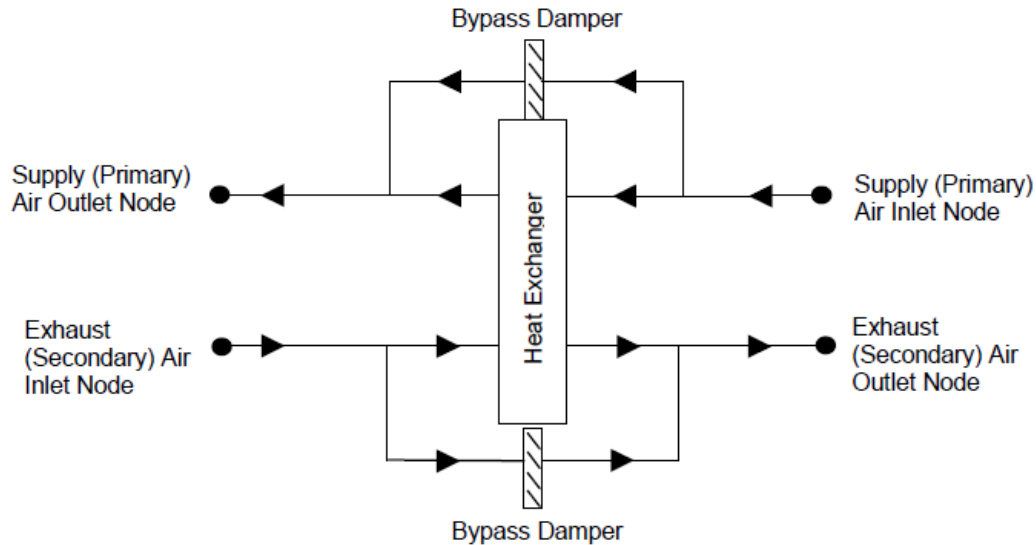
The minimum quantity of outdoor air delivered via the mixed air box will be the greater of:

- The minimum outdoor air flow rate calculated by the normal (non-DCV) outdoor air method based on the settings on the HVAC and Activity tabs, and,
- The outdoor air flow rate calculated by the DCV

The actual outdoor air flow rate may be higher than the minimum if free cooling is available. Regardless, the outdoor air flow rate will not exceed the [Maximum outdoor air flow rate](#)

Heat Recovery (Common To All AHUs)

If this option is checked, the AHU will incorporate a sensible and latent air-to-air heat recovery device capable of both heating and cooling the supply air stream using waste energy from the exhaust side.



Heat exchanger performance can be specified to transfer sensible energy, latent energy or both between the supply and exhaust air streams. The input requires no geometric data. Performance is defined by specifying sensible and/or latent effectiveness at 75% and 100% of the nominal (rated) supply air flow rate at two operating conditions as shown in the following table:

Parameter	Conditions	
	Heating	Cooling
Entering supply air temperature:		
Dry-bulb	1.7°C (35°F)	35°C (95°F)
Wet-bulb	0.6°C (33°F)	26°C (78°F)
Entering exhaust air temperature:		
Dry-bulb	21°C (70°F)	24°C (75°F)
Wet-bulb	14°C (58°F)	17°C (63°F)

Operating Conditions for Defining Heat Exchanger Performance.

Note: Conditions consistent with the Air-Conditioning and Refrigeration Institute's (ARI) Standard 1060-2001

Heat exchange between the supply and exhaust air streams occurs whenever the unit is scheduled to be available (Availability schedule) and supply/exhaust air flow is present. Heat recovery can be used in conjunction with a conventional air-side economiser (select **1-Modulate flow** for the Economiser control action type), whereby heat exchange is suspended whenever the air-side economiser (or high humidity control) is active (i.e. air flow is fully by-passed around a fixed-plate heat exchanger or the rotation of a rotary heat exchanger is stopped). It is also possible to suspend heat exchange for the purpose of providing free cooling operation in the absence of a conventional air-side economizer (i.e. specify **2-Minimum flow with bypass** for the Economiser control action type). During winter weather, humid exhaust air entering the heat exchanger can form frost on the cold heat exchanger surfaces, which can reduce air flow and the amount of energy recovery. Several methods are used to control or eliminate frost formation, and the following types can be modelled for this heat exchanger object: supply air pre-heat, minimum exhaust air temperature, exhaust air recirculation and exhaust only. For preheat frost control, the Pre-heat Coil option must be checked under the Pre-Treatment header. The other frost control types may be selected using the [Frost control type](#) drop-list.

Air-to-air heat exchangers are sometimes controlled to maintain a fixed supply air outlet temperature to avoid overheating. DesignBuilder provides an [option](#) to model this control in EnergyPlus by automatically including a [Setpoint manager](#) to establish a temperature set point at the supply air outlet node of the heat exchanger. Wheel speed modulation or plate supply air bypass is used to control the supply air exiting conditions to this set point. The set point for supply air temperature control should be set at the minimum economizer temperature set point if an air-side economizer is also being used by the air system. If frost control and supply

air outlet temperature control are used, frost control takes precedence over supply air temperature control (e.g. Control defrost time fraction is determined as if wheel speed modulation or plate supply air bypass is not used).

Heat recovery control tips

The 4 methods listed below are can be used to help control heat recovery systems.

Note: Although an economiser is included in some methods, its main purpose is to provide control of the heat recovery and not free cooling. If effective free cooling is required then you should consider using the **1-Modulate flow** economiser option.

Control method 1: Scheduled heat recovery heating setpoint: When using heat recovery with [Supply air outlet temperature control](#) and without an economiser the heat recovery unit heats the outside air up to the heat recovery setpoint and no higher while the outside air temperature is below the setpoint. The heat recovery unit is bypassed when the temperature setpoint is satisfied. When the outside air temperature rises above the heat recovery setpoint the heat exchanger is not bypassed running at full capacity and heating the air as much as possible. This can lead to high supply temperatures when it is warm outside. Overall this control method works well for lower outdoor air temperatures but not so well in summer.

Control method 2: Fixed heating setpoint with economiser: To avoid the overheating caused by Control method 1 an economiser can be introduced with the following settings:

- Economiser control type = **2-Fixed dry bulb**
- Outdoor dry-bulb temperature low limit control = On
- Economiser maximum limit dry-bulb temperature = Fixed heat recovery setpoint temperature, e.g. 12°C.
- Economiser control action type = **2-Minimum flow with bypass**
- Heat recovery bypass control type = **1-Bypass when within economiser limits**
- Heat recovery Supply air outlet temperature control = **2-Yes**

This works well reducing overheating relative to method 1, but it only provides a single fixed heat recovery setpoint temperature for the year which makes it difficult to achieve optimal control.

Control method 3: Heat recovery outdoor air temperature setpoint: Another approach to controlling heat recovery is to use the heat exchanger only when the outside air temperature is below a setpoint, and for warmer outside conditions bypass the heat recovery unit. The following settings achieve this:

- Economiser control type = **2-Fixed dry bulb**
- Outdoor dry-bulb temperature low limit control = On
- Economiser maximum limit dry-bulb temperature = 50 (high value to avoid heat recovery operation during warm conditions)
- Economiser control action type = **2-Minimum flow with bypass**
- Heat recovery bypass control type = **1-Bypass when within economiser limits**
- Heat recovery Supply air outlet temperature control = **1-No**
- Economiser lockout = **1-Yes**

Control method 4: Use EMS. An EMS program can be written to control the heat recovery unit in a more optimal way. An example script is shown below which is based on Control method 1 but overrides the HR setpoint temperature and/or availability schedules to simulate a control system which provides the full benefit of HR in winter and bypasses the heat exchanger in warmer conditions when HR is not required:

```
!- EMS Program: HR Optimal Control Strategy 2 - Stepped HR Setpoint + Zone Temp Sensing
EnergyManagementSystem:Sensor,
Toa,
Environment,
Site Outdoor Air Drybulb Temperature;

EnergyManagementSystem:Sensor,
TempBlock1Zone1, ! Name
Block1:Zone1, ! Zone name
Zone Mean Air Temperature ; ! Output:Variable or Output:Meter Name

EnergyManagementSystem:Actuator,
OverrideHRAvailability, !- Name
HR Availability Schedule, !- Actuated Component Unique Name
```



```

Schedule:Compact, !- Actuated Component Type
Schedule Value; !- Actuated Component Control Type

EnergyManagementSystem:Actuator,
OverrideHRSetpointTemp, !- Name
HR Setpoint Schedule, !- Actuated Component Unique Name
Schedule:Compact, !- Actuated Component Type
Schedule Value; !- Actuated Component Control Type

EnergyManagementSystem:ProgramCallingManager,
Optimise HR, !- Name
BeginTimestepBeforePredictor, !- EnergyPlus Model Calling Point
CalcHRSetpoint; !- Program Name

EnergyManagementSystem:Program,
CalcHRSetpoint,
! Stepped control of HR setpoint temp depending on outside air temp
IF Toa < 8,
Set OverrideHRSetpointTemp = 20.0,
ELSEIF Toa < 12,
Set OverrideHRSetpointTemp = 16.0,
ELSE,
Set OverrideHRSetpointTemp = 12.0,
ENDIF,
IF Toa > OverrideHRSetpointTemp,
Set OverrideHRAvailability = 0, ! Avoid E+ HR control quirk
ELSEIF TempBlock1Zone1 > 23,
Set OverrideHRAvailability = 0, ! avoid HR when room temp more than 23
ELSE,
Set OverrideHRAvailability = 1, ! Otherwise HR is available
ENDIF;

```

Other possibilities include checking for zone heating and/or cooling loads to help ensure optimal HR setpoints.

Nominal supply air flow rate

The nominal primary side (supply) air flow rate (in m3/s or ft3/min). The actual supply and exhaust air flow rates must be between 50% and 130% of this value or a warning will be issued.

Nominal electric power

This is the electric consumption rate of the device (W). Electric power is considered constant whenever the unit operates. This numeric input can be used to model electric power consumption by controls (transformers, relays, etc.) and/or a motor for a rotary heat exchanger. None of this electric power contributes thermal load to the supply or exhaust air streams. The default value for this field is 0.

Heat recovery bypass control type

This selection determines whether specialized control is to be used to optimise the use of heat recovery when combined with an economiser. Options are:

- **1-Bypass when within economizer limits** - The default option where heat recovery is disabled any time the controller determines that the economizer is active (i.e. all economiser controls are within limits). Note that in cases where the economiser is selected but no economiser limits are defined then the heat recovery will always be bypassed.
- **2-Bypass when OA flow greater than minimum** - EnergyPlus checks whether the economizer is active and also whether the outdoor air flow rate is greater than the minimum, if so heat recovery is disabled. This means that heat recovery is active any time the outdoor air flow rate is equal to the minimum even when the economizer requests free cooling. For this case, the use of Supply air outlet temperature control for the heat exchanger is recommended (see below).

Supply air outlet temperature control

This data determines if the heat exchanger's supply air outlet is controlled to a temperature set point. The choices for this input field are **1-Yes** or **2-No**, with the default being **2-No**. When supply air outlet temperature control is used, the wheel rotational speed modulates or supply air is bypassed around the plate heat exchanger to maintain the desired setpoint temperature.

When **1-Yes** is selected you can enter the setpoint temperature to be used in the next text box below. In this case DesignBuilder automatically adds a scheduled [Setpoint manager](#) at the heat recovery supply air outlet node using the entered set point temperature. When an air-side economizer is also being modelled for this air system, the set point for the supply air outlet temperature control is forced to be equal to the [Economizer minimum limit dry-bulb temperature](#).

Supply air outlet setpoint temperature schedule

When **1-Yes** is selected for [Supply air outlet temperature control](#) you can select a time-varying setpoint temperature schedule to be used for controlling the heat recovery unit as described above.

Heat exchanger type

This setting is used to define the type of heat exchanger being modelled. The options are:

- **1-Plate** (e.g., fixed plate) or
- **2-Rotary** (e.g., rotating cylinder or wheel).

Plate heat exchangers can be either sensible only when plates are made of a non-porous material such as aluminium, or they can also exchange latent heat when the plates are made of a porous material.

The heat exchanger type affects the modelling of frost control options and supply air outlet temperature control. For rotary heat exchangers, rotational speed is varied to control frost formation or the supply air outlet temperature. For plate exchangers, air bypass around the heat exchanger is used to obtain the desired effect.

Economiser lockout

This input denotes whether the heat exchanger unit is locked out (bypassed for plate type heat exchangers or the rotation is suspended for rotary type heat exchangers) when the air-side economiser is operating. Both the economiser and High Humidity Control activate the heat exchanger lockout as specified by this input. The input choices are **1-Yes** (meaning locked out) or **2-No**.

Sensible effectiveness at 100% heating air flow

The sensible heat exchange effectiveness at the heating condition defined in the Table above with both the supply and exhaust air volume flow rates equal to 100% of the nominal supply air flow rate specified in the previous input field. The default value for this field is 0.

Latent effectiveness at 100% heating air flow

The latent heat exchange effectiveness at the heating condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 100% of the nominal supply air flow rate. Specify this value as 0.0 if the heat exchanger does not transfer latent energy. The default value for this field is 0.

Sensible effectiveness at 75% heating air flow

The sensible heat exchange effectiveness at the heating condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 75% of the nominal supply air flow rate. The default value for this field is 0.

Latent effectiveness at 75% heating air flow

The latent heat exchange effectiveness at the heating condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 75% of the nominal supply air flow rate. Specify this value as 0.0 if the heat exchanger does not transfer latent energy. The default value for this field is 0.

Sensible effectiveness at 100% cooling air flow

The sensible heat exchange effectiveness at the cooling condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 100% of the nominal supply air flow rate. The default value for this setting is 0.

Latent effectiveness at 100% cooling air flow

The latent heat exchange effectiveness at the cooling condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 100% of the nominal supply air flow rate. Specify this value as 0.0 if the heat exchanger does not transfer latent energy. The default value for this setting is 0.

Sensible effectiveness at 75% cooling air flow

The sensible heat exchange effectiveness at the cooling condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 75% of the nominal supply air flow rate. The default value for this setting is 0.

Latent effectiveness at 75% cooling air flow

The latent heat exchange effectiveness at the cooling condition defined in the Operating Conditions for Defining Heat Exchanger Performance table with both the supply and exhaust air volume flow rates equal to 75% of the nominal supply air flow rate. Specify this value as 0.0 if the heat exchanger does not transfer latent energy. The default value for this field is 0.

Frost control type

There are four options for frost control:

- **1-None**,
- **2-Exhaust air recirculation**, dampers are used to direct exhaust air back into the zone through the supply side of the heat exchanger when the supply (outdoor) air inlet temperature falls below a threshold temperature (defined in the next input field). The fraction of time that exhaust air is circulated through the supply side of the heat exchanger is dependent on the supply (outdoor) air inlet temperature with respect to the threshold temperature, the initial defrost time fraction, and the rate of change of defrost time fraction (see [Rate of defrost time fraction increase](#)). When exhaust air is being re-circulated, no supply (outdoor ventilation) air is being provided through the heat exchanger unit (which may or may not be acceptable regarding ventilation for occupants).
- **3-Exhaust only** (supply air bypass): this control cycles off the supply air flow through the heat exchanger for a certain period of time while the exhaust air continues to flow through the exhaust side of the heat exchanger. The fraction of time that the supply flow through the heat exchanger is cycled off is dependent on the supply (outdoor) air inlet temperature with respect to the threshold temperature, the initial defrost time fraction, and the rate of change of defrost time fraction (see [Rate of defrost time fraction increase](#)). When implemented in real applications, provisions are usually made to avoid building depressurization when this frost control is operating (automatic or pressure-operated dampers, or a bypass air damper around the supply side of the heat exchanger). For this frost control type, it is assumed that the supply air is bypassed around the heat exchanger during frost control operation, i.e. the total supply flow is not reduced during defrost, but merely bypassed around the heat exchanger.
- **4-Minimum exhaust temperature** where the temperature of the exhaust air leaving the heat exchanger is monitored and the heat exchanger effectiveness is decreased (by slowing heat exchanger rotation or bypassing supply air around the plate exchanger) to keep the exhaust air from falling below the threshold temperature.

If a [pre-heat frost control](#) coil is specified, this setting is automatically set to **1-None** and disabled.

Threshold temperature

This numeric field defines the dry-bulb temperature of air which is used to initiate frost control (in °C or °F). The default value is 1.7°C. If **2-Exhaust air recirculation** or **3-Exhaust only** options are selected the [Frost control type](#), the threshold temperature defines the supply (outdoor) air inlet temperature below which frost control is active. For **4-Minimum exhaust temperature** frost control, heat exchanger effectiveness is controlled to keep the exhaust air outlet temperature from falling below this threshold temperature value. The appropriate threshold temperature varies with exhaust (inlet) air temperature and humidity, frost control type, heat exchanger type, and whether the heat exchanger transfers sensible energy alone or both sensible and latent energy (enthalpy). Typical threshold temperatures are provided in the table below. However, it is recommended that the user consult manufacturer's information for the specific air-to-air heat exchanger being modelled.

Frost control type	Heat exchanger type	Energy exchange	Threshold temperature
Exhaust air recirculation	Plate	Sensible-only	-1.1°C (30°F)
		Sensible + latent	-12.2°C (10°F)
	Rotary	Sensible-only	-12.2°C (10°F)
		Sensible + latent	-23.3°C (-10°F)
Exhaust only	Plate	Sensible-only	-1.1°C (30°F)
		Sensible + latent	-12.2°C (10°F)
	Rotary	Sensible-only	-12.2°C (10°F)
		Sensible + latent	-23.3°C (-10°F)
Minimum exhaust temperature	Plate	Sensible-only	1.7°C (35°F)
		Sensible + latent	1.7°C (35°F)
	Rotary	Sensible-only	1.7°C (35°F)
		Sensible + latent	1.7°C (35°F)
Preheat**	Plate	Sensible-only	-1.1°C (30°F)
		Sensible + latent	-12.2°C (10°F)
	Rotary	Sensible-only	-12.2°C (10°F)
		Sensible + latent	-23.3°C (-10°F)

Typical Threshold Temperatures

Source: Indoor Humidity Assessment Tool, U.S. Environmental Protection Agency,

** To model preheat frost control, specify frost control type as **1-None** and select a pre-heat coil for the AHU controlled to keep the air temperature above the frost threshold temperature.

Initial defrost time fraction

This is the fraction of the simulation timestep when frost control will be invoked when the threshold temperature is reached. This setting is only used for the **2-Exhaust air recirculation** and **3-Exhaust only** frost control types. The value for this setting must be ≥ 0 and ≤ 1 . The default time fraction is 0.083 (e.g., 5 min / 60 min) which is typical for Exhaust air recirculation frost control. Higher initial defrost time fractions (e.g., 0.167 = 10 min / 60 min) are typically required for Exhaust only frost control. For best results, you should obtain this information from the manufacturer.

Rate of defrost time fraction increase

This is the rate of increase in the defrost time fraction as the supply (outdoor) air inlet temperature falls below the threshold temperature. This field is only used for the **2-Exhaust air recirculation** and **3-Exhaust only** frost control types. The value for this setting must be ≥ 0 . The default value is 0.012 (e.g., 0.72 min / 60 min per degree C temperature difference) which is typical for Exhaust air recirculation frost control. Higher values (e.g. 0.024 = 1.44 min / 60 min per °C temperature difference) are typically required for Exhaust only frost control. For best results, you should obtain this information from the manufacturer.

The total defrost time fraction is determined as follows:

Total defrost time fraction = Initial Defrost Time Fraction + Rate of Defrost Time Fraction Increase * $(T_{\text{threshold}} - T_{\text{supply air inlet}})$

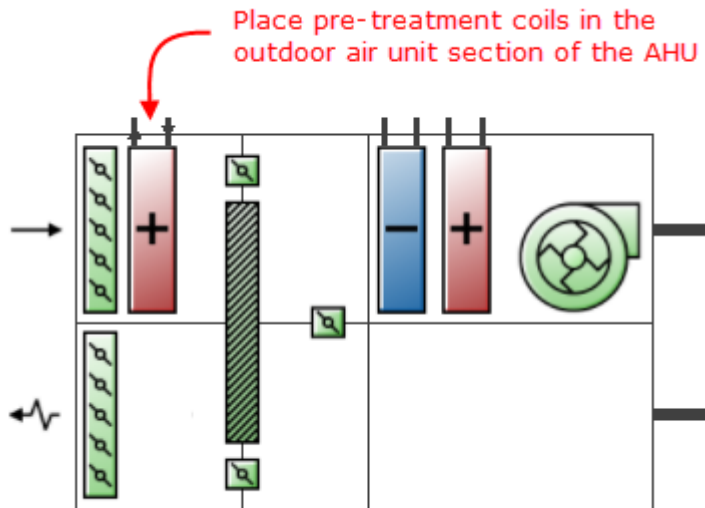
The model does not allow the total defrost time fraction to exceed 1.0 or be less than 0.

Availability schedule

This is the schedule that determines whether or not the heat recovery device is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the device can be on. A value less than or equal to 0 (usually 0 is used) denotes that the device must be off.

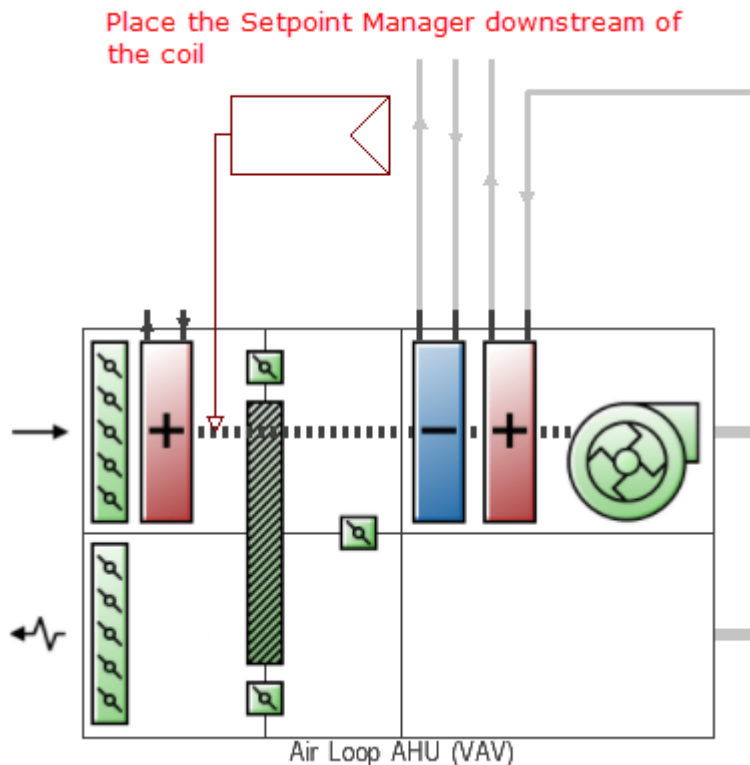
Pre-treatment Coils

Pre-treatment heating and cooling coils, humidifiers and evaporative coolers can be included to ensure that the outside air is pre-conditioned before it enters the air handling unit. These coils are added in the same way as for other AHU coils, by navigating to the AHU object and placing coils in the outdoor air unit section of the AHU. This allows flexibility in the number and type of coils added and also in the way that they are controlled. See screenshot below.



Separate SPMs are required for pre-heat and pre-cool coils to ensure the correct off-coil conditions even if there is a temperature Setpoint Manager placed on the supply air side of the AHU.

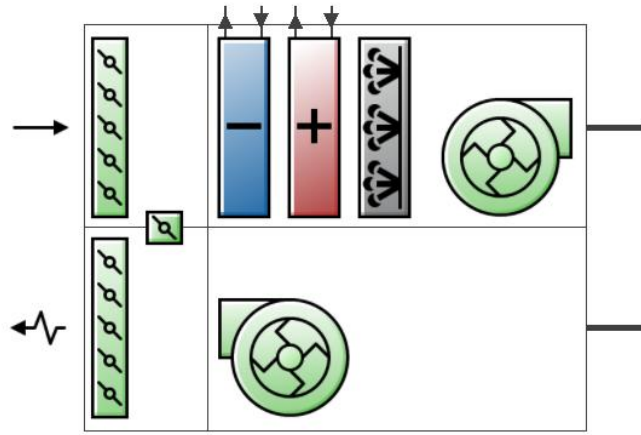
To add SPMs to control pre-treatment coils, starting by navigating up to the supply side of the air loop. Then click on the [Add Setpoint manager](#) toolbar icon to add a SPM. See screenshot below.



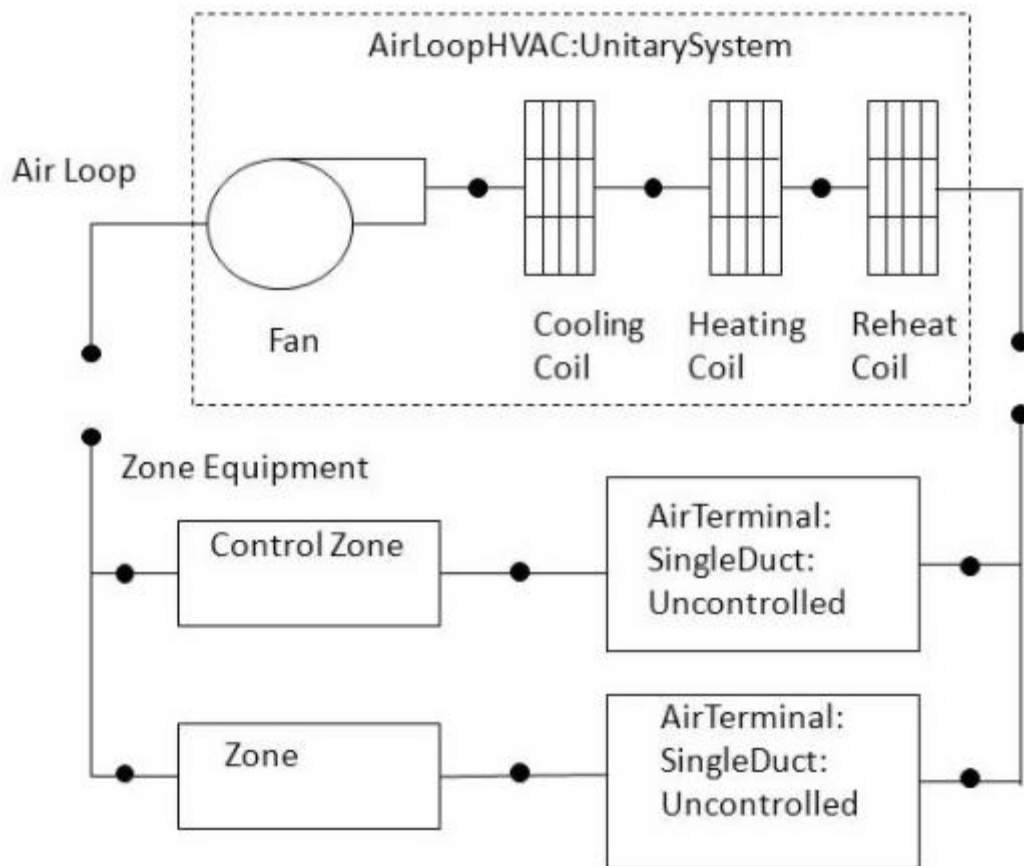
Note: Only **1-Scheduled** type SPMs can be added for pre-treatment coils.

Unitary Generic AHU

The Unitary Generic AHU is intended to replace all other AHUs, although other system types are still available. This system is unique in that it can accommodate all fan and coil types whereas other AHUs are specific to the type of fan and coil available for simulation.



The generic unitary AHU incorporates an outdoor air system and a supply fan (OnOff, ConstantVolume, VariableVolume). There are also options for night cycle operation, a heat recovery system, an economiser, pre-heat and pre-cool coils and an extract fan. Additional components such as heating coils, cooling coils and humidifiers may also be added.



The Unitary AHU consists of a fan component (OnOff, ConstantVolume or VariableVolume), a cooling coil component, a heating coil component, and a reheat coil as shown in the schematic diagram above. For draw-through configurations, the fan is placed directly after the heating coil. If dehumidification control is selected, a reheat coil component is also required. If the reheat coil is present and the [Dehumidification control type](#) input is not specified as **2-Cool reheat**, the reheat coil will not be active.

In addition, the control zone and the system design operating conditions are specified in the unitary system inputs.

General

Name

This field contains the identifying name for the unitary AHU. It is read-only.

Fan type

Select the type of fan for the unitary system from the following options:

- **1-Constant volume**, used when the [Fan operating mode schedule](#) values are never 0 and the fan operates continuously.
- **2-Variable volume**, used for variable air volume systems or multi or variable-speed coils.
- **3-On/Off**, used when the fan cycles on and off with the cooling or heating coil (i.e. [Fan operating mode schedule](#) values are at times 0).

Fan placement

Select from the following options:

1-Blow through, a blow through fan where the unit consists of a fan followed by the main cooling and heating coils and supplemental heating coil. The fan “blows through” the cooling and heating coils.

2-Draw through, a draw through fan where the unit consists of the main cooling/heating coil(s) followed by a fan, with the supplemental heater located at the outlet of the fan. The fan “draws air through” the cooling/heating coil(s).

Fan operating mode schedule

Select a schedule that defines the supply air fan operating mode schedule. The fan operating mode may vary during the simulation based on time of day or with a change of season.

- Schedule values of 0 denote that the unitary system supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (aka AUTO fan).
- Schedule values other than 0 denote that the supply fan runs continuously while the heating or cooling coil cycles to meet the load.

Note: The **3-Single Zone VAV Control type** is only active when the supply air fan runs continuously, and during cycling fan operation the **1-Load** Control type model is used.

Nighttime operation

Unitary systems don't need the night cycle manager used by the standard generic AHU. Instead this functionality is achieved as follows.

1. Select the **On 24/7 Availability schedule** to allow the system to always be on.
2. Use the **Fan operating mode schedule** to switch between:
 - a. continuous fan operation for ventilation during occupied periods, and,
 - b. cycling fan for unoccupied periods to allow the system to cycle on and off as the thermostat requests, and run just enough to meet the load.

Operation

Availability schedule

Select the schedule which contains information on the availability of the unitary system to operate. A schedule value equal to 0 denotes that the unitary system must be off for that time period. A value greater than 0 denotes that the unitary system is available to operate during that time period. This schedule may be used to completely disable the unitary system as required.

Control

Control type

This setting defines the control type for the unitary system, i.e. whether it is load based or setpoint based. Select from these options:

- **1-Load** which requires a Controlling zone to be defined.
- **2- SetPoint** which requires set points at each coil outlet node. A single set point at the outlet of the system is allowed but not recommended. If setpoint control is used and the system represents a heat pump (i.e., cooling and heating coils are both DX coils) then only one of these coils may operate at a time.
- **3-Single Zone VAV** which requires two distinct fan flow rates, namely the Cooling and Heating supply air flow rate and a lower No Load Supply Air Flow Rate which is used during times of reduced cooling or heating loads. This option allows load control at low speed fan until the load exceeds available capacity or the outlet air temperature exceeds the specified limits where the fan speed is then increased. Temperature limits are identified in the input fields for [Minimum](#) and [Maximum supply air temperature](#).

Additionally, specific coil types are required for this control type. Allowed cooling coil types are: **1-Water**

and **3-DX single speed**, while the allowed heating coil types are **1-Water**, **2-Electric**, **3-Fuel** and **4-DX single speed**. If other coil types are used they are modelled using the **1-Load** based control method.

Both **1-Load** and **3-Single Zone VAV** Control types require a [Controlling zone](#) to be defined.

Controlling zone

When [Control type](#) is set to **1-Load** or **3-Single Zone VAV** you must select the zone where the thermostat controlling the unitary system is located.

Dehumidification control type

This setting allows you to select the type of dehumidification control. The following options are available:

- **1-None** - the default option where the AHU meets sensible loads only with no active dehumidification control. This option is required when Control type is set to **3-Single Zone VAV**.
- **2-Cool reheat** - where the AHU cools beyond the dry-bulb temperature set point as required to meet the high humidity setpoint.

Latent load control

This setting defines the latent load control method. Available choices are:

- **1-Sensible only**, the default option, will operate to meet only a sensible load and is also required when the **3-Single Zone VAV** Control type is selected.
- **2-Latent only** will operate to meet only a latent load
- **3-Latent with sensible** will operate to meet the latent load only if there is a sensible load.
- **4-Latent or sensible** will operate to meet either a latent or sensible load.

Supply Air Flow

Minimum supply air temperature

This numeric field contains the design operating air outlet temperature (in °C or °F) when the unitary system is cooling.

When the [Use DOAS DX cooling coil](#) checkbox is checked, this is the DX cooling coil's leaving minimum air temperature for frost control. The DX cooling coil leaving air temperature is not allowed to be lower than this minimum air temperature. The DX cooling coil frost controller adjusts or limits the desired coil outlet air setpoint temperature when the coil outlet temperature exceeds this minimum temperature limit specified. The minimum and maximum values of this input field are 0.0°C and 7.5°C respectively. The default value is 2.0°C.

When using the **3-Single Zone VAV** Control type, enter the minimum air temperature limit for reduced fan speed in cooling mode. In this case the maximum limit for the minimum supply air temperature is 20.0°C. Additionally, for the **3-Single Zone VAV** Control type, this input does not limit the minimum supply air temperature resulting from cooling coil operation at high fan speed.

Maximum supply air temperature

This numeric field contains the design operating air outlet temperature (in °C or °F) when the unitary system is heating. When using the **3-Single Zone VAV** Control type, enter the maximum air temperature limit for reduced fan speed in heating model. In this case, this input does not limit the maximum supply air temperature resulting from heating or supplemental heating coil operation at high fan speed. This field is autosizable.

Cooling supply air flow rate method

This setting defines the supply air flow method during cooling operation. Available choices are:

- **1-Supply air flow rate.**
- **2-Flow per floor area.**
- **3-Fraction of autosized cooling value.**
- **4-Flow per cooling capacity.**

For each of the choices, a corresponding air flow rate for cooling must be specified (see below). If the system does not have a cooling coil a 0 may be entered for cooling air flow rate and/or no load supply air flow rate to turn the fan off when cooling is not required.

Cooling supply air flow rate

This setting defines the supply air flow rate leaving the unitary system (in m³/s or ft³/min) when the cooling coil is operating. It is required when the Cooling supply air flow rate method is **1-Supply air flow rate**. This field is autosizable.

Cooling supply air flow rate per floor area

This setting defines the supply air flow rate per floor area leaving the unitary system (in m³/s or ft³/min) when the cooling coil is operating. It is required when the Cooling supply air flow rate method is **2-Flow per floor area**. This field is autosizable.

Cooling fraction of autosized design cooling supply air flow rate

This setting defines the fraction of autosized supply air flow rate leaving the unitary system when the cooling coil is operating. It is required when the Cooling supply air flow rate method is **3-Fraction of autosized cooling value**.

Cooling supply air flow rate per unit of capacity

This numeric field defines the supply air flow rate per unit of capacity leaving the unitary system when the cooling coil is operating. The units are or m³/s/W or (ft³/min)/(Btu/h). It is required when the Cooling supply air flow rate method is **4-Flow per cooling capacity**.

Heating supply air flow rate method

This setting defines the supply air flow method during heating operation. Available choices are:

- **1-Supply air flow rate**.
- **2-Flow per floor area**.
- **3-Fraction of autosized heating value**.
- **4-Flow per heating capacity**.

For each of the choices, a corresponding air flow rate for heating must be specified (see below). If the system does not have a heating coil a 0 may be entered for heating air flow rate and/or no load supply air flow rate to turn the fan off when heating is not required.

Heating supply air flow rate

This setting defines the supply air flow rate leaving the unitary system (in m³/s or ft³/min) when the heating coil is operating. It is required when the Heating supply air flow rate method is **1-Supply air flow rate**. This field is autosizable.

Heating supply air flow rate per floor area

This setting defines the supply air flow rate per floor area leaving the unitary system (in m³/s or ft³/min) when the heating coil is operating. It is required when the Heating supply air flow rate method is **2-Flow per floor area**. This field is autosizable.

Heating fraction of autosized design heating supply air flow rate

This setting defines the fraction of autosized supply air flow rate leaving the unitary system when the heating coil is operating. It is required when the Heating supply air flow rate method is **3-Fraction of autosized heating value**.

Heating supply air flow rate per unit of capacity

This numeric field defines the supply air flow rate per unit of capacity leaving the unitary system when the heating coil is operating. The units are or m³/s/W or (ft³/min)/(Btu/h). It is required when the Heating supply air flow rate method is **4-Flow per cooling capacity**.

No load supply air flow rate method

This setting defines the supply air flow method when neither cooling or heating is required. Available choices are:

- **1-Supply air flow rate.**
- **2-Flow per floor area.**
- **3-Fraction of autosized cooling value.**
- **4-Fraction of autosized heating value.**
- **5-Flow per cooling capacity.**
- **6-Flow per heating capacity.**

For each of these options, a corresponding air flow rate must be specified. The following "No load supply air flow rate" etc fields are also used to specify the lower air flow rate for the **3-Single Zone VAV** control method with recommendations of greater than or equal to 67% of the Cooling or Heating Supply Air Flow Rate when any DX coil is used and 50% for other coil types.

No load supply air flow rate

This numeric setting defines the supply air flow rate leaving the unitary system (in m³/s or ft³/min) when neither cooling or heating is required (i.e., when the main cooling/heating coils and supplemental heater are off but the supply air fan operates). It is only used when the unitary system operating mode is specified as continuous fan operation. It is required when the No load supply air flow rate method is **1-Supply air flow rate**.

This field is autosizable. If this field is autosized, then it is sized to the minimum of the heating and cooling lowest speed supply air flow rate. If the unitary system operating mode is specified as continuous fan operation and this value is set to zero, then the model assumes that the supply air flow rate when no cooling/heating is needed is equal to the supply air flow rate when the compressor was last operating (for cooling operation or heating operation).

No load supply air flow rate per floor area

This numeric setting defines the supply air flow rate per floor area leaving the unitary system (in m³/s or ft³/min) when neither cooling or heating coil is operating. It is required when the No load supply air flow rate method is **2-Flow per floor area**. This field is autosizable.

No load fraction of autosized cooling supply air flow rate

This numeric field defines the fraction of autosized supply air flow rate leaving the unitary system when neither cooling or heating coil is operating. It is required when the No load supply air flow rate method is **3-Fraction of autosized cooling value**.

No load fraction of autosized heating supply air flow rate

This numeric setting defines the fraction of autosized supply air flow rate leaving the unitary system when the neither cooling or heating coil is operating. It is required when the No load supply air flow rate method is **4-Fraction of autosized heating value**.

No load supply air flow rate per unit of capacity during cooling operation

This numeric setting defines the supply air flow rate per unit of capacity leaving the unitary system when neither cooling or heating is operating. It is required when the No load supply air flow rate method is **5-Flow per cooling capacity**.

No Load Supply Air Flow Rate Per Unit of Capacity During Heating Operation

This numeric setting defines the supply air flow rate per unit of capacity leaving the unitary system when neither cooling or heating is operating. It is required when the No load supply air flow rate method is **6-Flow per heating capacity**.

Extract Fan

Include extract fan

Check the checkbox to include an extract fan.

DX Coil

DX heating coil sizing ratio

This numeric field is used to adjust heat pump heating capacity with respect to DX cooling capacity. It is used only for DX heat pump configurations (i.e., when DX cooling and DX heating coils are used).

Use DOAS DX cooling coil

Important Note: In DesignBuilder v2025.1, this option should be left unchecked due to a bug in EnergyPlus v23.1: [#11378](#).

This input field enables DX Cooling coils to be used for 100% outdoor air dedicated outdoor air system applications.

When this option is checked, the DX coil is used as 100% outdoor DX coil.

When it is unchecked, the DX coil is used as regular DX coil.

This option is not available when the **3-Single Zone VAV Control type** is selected.

Compressor

Maximum cycling rate

This numeric field contains the maximum on-off cycling rate for the compressor, which occurs at 50% run time fraction. Suggested values for Maximum cycling rate are shown below(Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	2.5
Good	2.5
Poor	3.0

Heat Pump

Heat pump time constant

This numeric field contains the time constant for the cooling coil's capacity to reach steady state after startup. Suggested values are shown in the table below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	60
Good	60
Poor	60

Heat pump fan delay time

This numeric field contains the time delay for the heat pump supply air fan to shut off after the compressor cycles off in seconds. This value can be obtained from the manufacturer or the heat pump catalog. Enter a value of zero when the heat pump's fan operating mode is continuous. Suggested value is 60 seconds.

On-Cycle And Off-Cycle Power Use

Fraction of on-cycle power use

This numeric field contains the fraction of on-cycle power use to adjust the part load fraction based on the off-cycle power consumption due to crankcase heaters, controls, fans, and etc. Suggested value values are shown in the table below. (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	2.5
Good	2.5
Poor	3.0

Ancillary on-cycle electric power

This field defines ancillary electrical power (W) consumed during the on-cycle period (i.e., when the cooling or heating coil is operating). The model assumes that this ancillary power does not contribute to heating the supply air.

Ancillary off-cycle electric power

This field defines ancillary electrical power (W) consumed during the off-cycle period (i.e., when the cooling and heating coil are not operating). The model assumes that this ancillary power does not contribute to heating the supply air.

Supplemental Heater

Maximum outdoor dry-bulb temperature for supplemental heater operation

This numeric field defines the outdoor air dry-bulb temperature above which the heat pump supplemental heating coil is disabled. The temperature for this input field must be less than or equal to 21°C.

Mixed Mode Zone Equipment

You can set up Mixed mode controls for the AHU to model optimal interaction between the natural ventilation system (Scheduled or Calculated) and the AHU.

Refer to the main HVAC tab [Mixed Mode](#) section for details on the options available.

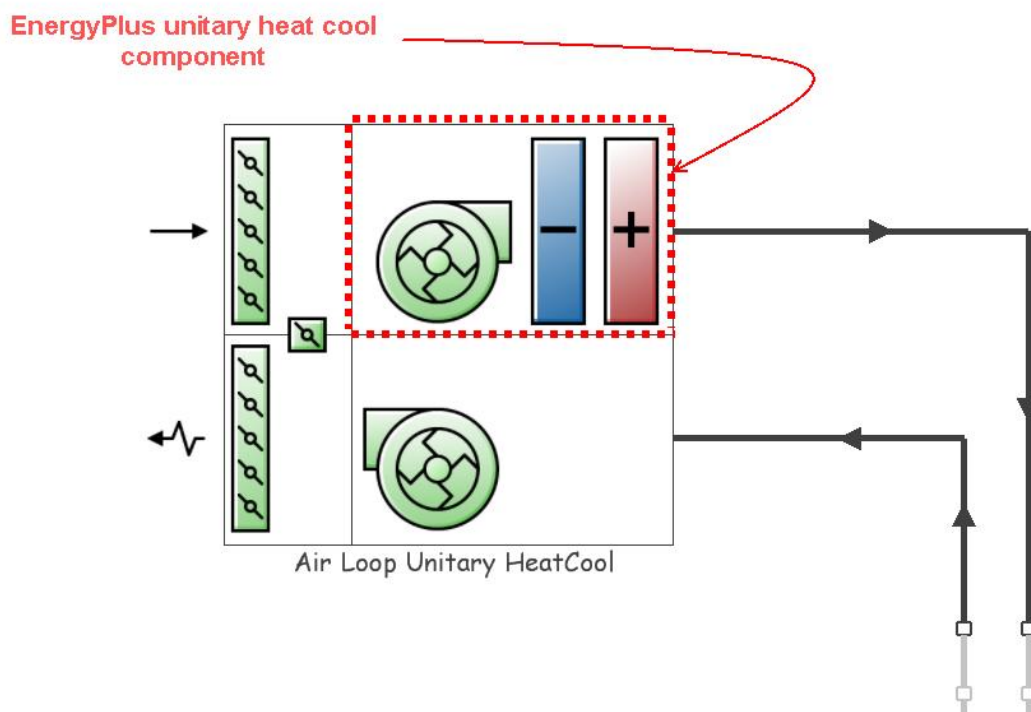
Note: If mixed mode is activated here on the AHU dialog, then any mixed mode settings for connected zones on the HVAC tab will be overridden by these mixed mode settings.

Outdoor Air System

See [Outdoor air system description for Generic AHU](#)

Unitary Heat Cool AHU

The unitary heat cool AHU incorporates an EnergyPlus unitary heat cool component, an outdoor air system (see Air Handling Unit Outdoor Air System), optional night cycle operation, an optional heat recovery system (see Heat Recovery), optional pre-heat and pre-cool coils and an optional extract fan. The unitary heat cool component itself comprises a supply fan, a DX cooling coil component and an electric heating coil component. If dehumidification control is selected, an electric reheat coil is automatically included.



Note: The Unitary Heat Cool AHU can only be added as part of a [Unitary Heat Cool Air loop](#) .

A detailed description of how Unitary air loops work is provided in the [EnergyPlus Engineering Reference](#).

General

Name

This is the name that you assign to the AHU which should be unique. If the supplied name is not unique, the software will automatically append a backslash and integer to ensure that there are no duplicate names.

Operation

Availability schedule

This schedule contains information on the availability of the AHU for operation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

When this unit is unavailable (schedule values of 0) then all equipment contained within this AHU will also be switched off.

Supply Air Temperature

Maximum supply air temperature

This is the design operating air outlet temperature (in °C or °F) when the unitary system is heating.

Supply Air Flow Rates

Design air flow rate

This is the air loop primary air design volumetric flow rate (in m³/s or ft³/min). This setting is auto-sizeable.

Cooling operation air flow rate

This setting defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when the DX cooling coil is operating. Values must be greater than 0 or this field is auto-sizeable.

Heating operation air flow rate

This setting defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when the DX heating coil and/or supplemental heater are operating. Values must be greater than 0 or this field is auto-sizeable.

Supply air flow rate when no cooling or heating is needed

This setting defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when neither cooling nor heating is required (i.e., DX coils and supplemental heater are off but the supply air fan operates). This setting is only used when the heat pump operating mode is specified as continuous fan operation (see Supply Air Fan Operating Mode Schedule). Values must be greater than or equal to zero, or this field is auto-sizeable. If the heat pump operating mode is specified as continuous fan operation and this value is set to zero or this field is left blank, then the model assumes that the supply air flow rate when no cooling/heating is needed is equal to the supply air flow rate when the compressor was last operating (for cooling operation or heating operation).

Supply Fan

Fan type

Two fan types are available for selection in this AHU:

- **1-Constant volume** where the fan delivers a constant flow rate through the unit regardless of whether the unit is heating, cooling and/or providing fresh air. Constant volume fans don't use the [Supply air fan operating mode schedule](#) (below). When this option is selected an [extract fan](#) can optionally also be included in the AHU.
- **3-On/Off** where the [Supply air fan operating mode schedule](#) (below) determines the flow regime. In this case an extract fan cannot be selected.

Supply fan placement

There are two options:

- **1-Blow through** where the supply air fan is before the DX cooling/heating coil.
- **2-Draw through** where the supply air fan is after the DX cooling/heating coil.

Supply air fan operating mode schedule

The supply air fan operating mode schedule can only be selected for On/Off fan types. The supply air fan operating mode may vary during the simulation based on time-of-day or with a change of season. Schedule

values of 0 indicate that the unitary system supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (sometimes referred to as an "AUTO" fan). Schedule values other than 0 denote that the supply fan runs continuously while the heating or cooling coil cycles to meet the load.

The default schedule is "Fan operation mode - Continuous" with a constant value of 1, i.e. that the supply fan runs continuously while the heating or cooling coil cycles to meet the load at all times. To obtain the AUTO fan configuration, select the "Fan operation mode - Cycling" schedule which has a constant value of 0.

Extract Fan

Include extract fan

An extract fan is included within all AHUs by default. This setting may be used to remove the extract fan from the AHU.

Note: This option is only available for **Constant volume** fan types. **On/Off** fans do not allow an extract fan to be included in the AHU and when this fan type is selected the **Include extract fan** check box will be disabled.

Thermostat Location

Controlling zone or thermostat location

While the unitary heat cool AHU may be configured to serve multiple zones, system operation is controlled by a thermostat located in a single "control" zone. Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone supplied by the AHU where the thermostat controlling the unitary heat cool AHU is located.

Dehumidification

Dehumidification control type

There are two options for dehumidification control:

1-None - meet sensible load only, no active de-humidification control;

2-Cool Reheat - cool beyond the dry-bulb temperature set point as required to meet the high humidity setpoint.

Note: If **2-Cool Reheat** is used, a humidistat must be defined in one of the zones supplied by the AHU.

More information on controlling humidity is available in the [Humidity control](#) section.

Night Cycle

See [Night cycle description for Generic AHU](#)

Mixed Mode

You can set up Mixed mode controls for the AHU to model optimal interaction between the natural ventilation system (Scheduled or Calculated) and the AHU.

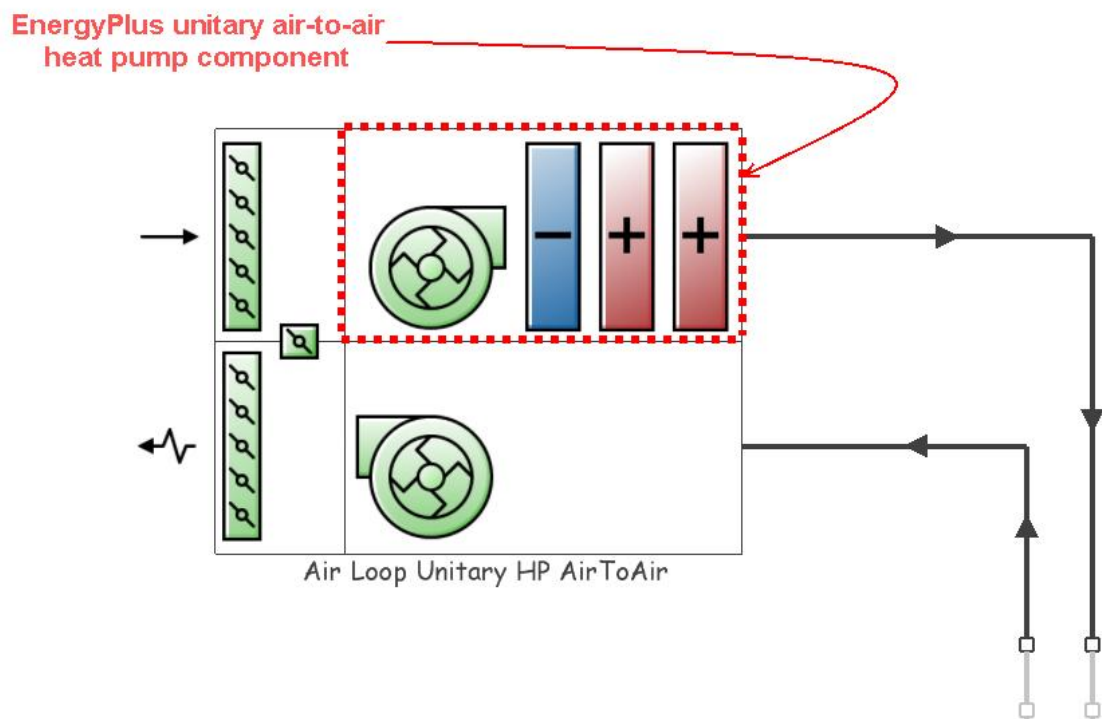
Refer to the main HVAC tab [Mixed Mode](#) section for details on the available options.

Outdoor Air System

See [Outdoor air system description for Generic AHU](#)

Unitary Air To Air Heat Pump AHU

The unitary air-to-air heat pump AHU incorporates an EnergyPlus unitary air-to-air heat pump component, an outdoor air system, optional night cycle operation, an optional heat recovery system, optional pre-heat and pre-cool coils and an optional extract fan. The unitary air-to-air heat pump component itself comprises a supply fan, a DX cooling coil component, a DX heating coil component, and an electric, hot water or fuel supplementary heating coil component.



Tip: The Unitary Air to Air Heat Pump AHU can be used to model ASHRAE 90.1 Appendix G System 4 PSZ-HP.

Note: The Unitary Air to Air Heat Pump AHU can only be added as part of a [Unitary Air to Air Heat Pump Air loop](#).

A detailed description of how Unitary air loops work is provided in the [EnergyPlus Engineering Reference](#).

General

Name

This is the name that you assign to the AHU which should be unique. If the supplied name is not unique, the software will automatically append a backslash and integer to ensure that there are no duplicate names.

Operation

Availability schedule

This schedule contains information on the availability of the AHU for operation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

When this unit is unavailable (schedule values of 0) then all equipment contained within this AHU will also be switched off.

Supply Air Flow Rates

Cooling operation air flow rate

This setting defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when the DX cooling coil is operating. Values must be greater than 0 or this field is autosizable.

Heating operation air flow rate

This setting defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when the DX heating coil and/or supplemental heater are operating. Values must be greater than 0 or this field is auto-sizable.

Supply air flow rate when no cooling or heating is needed

This setting defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when neither cooling nor heating is required (i.e., DX coils and supplemental heater are off but the supply air fan operates). This setting is only used when the heat pump operating mode is specified as continuous fan operation (see Supply Air Fan Operating Mode Schedule). Values must be greater than or equal to zero, or this field is auto-sizable. If the heat pump operating mode is specified as continuous fan operation and this value is set to zero or this field is left blank, then the model assumes that the supply air flow rate when no cooling/heating is needed is equal to the supply air flow rate when the compressor was last operating (for cooling operation or heating operation).

Supply Fan

Fan type

Two fan types are available for selection in this AHU:

- **1-Constant volume** where the fan delivers a constant flow rate through the unit regardless of whether the unit is heating, cooling and/or providing fresh air. Constant volume fans don't use the [Supply air fan operating mode schedule](#) (below). When this option is selected an [extract fan](#) can optionally also be included in the AHU.
- **3-On/Off** where the [Supply air fan operating mode schedule](#) (below) determines the flow regime. In this case an extract fan cannot be selected.

Supply fan placement

There are two options:

- **1-Blow through** where the supply air fan is before the DX cooling/heating coil and the supplementary heating coil.
- **2-Draw through** where the supply air fan is between the DX cooling/heating coil and the supplementary heating coil.

Supply air fan operating mode schedule

The supply air fan operating mode schedule can only be selected for On/Off fan types. The supply air fan operating mode may vary during the simulation based on time-of-day or with a change of season. Schedule

values of 0 indicate that the unitary system supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (sometimes referred to as an "AUTO" fan). Schedule values other than 0 denote that the supply fan runs continuously while the heating or cooling coil cycles to meet the load.

The default schedule is "Fan operation mode - Continuous" with a constant value of 1, i.e. that the supply fan runs continuously while the heating or cooling coil cycles to meet the load at all times. To obtain the AUTO fan configuration, select the "Fan operation mode - Cycling" schedule which has a constant value of 0.

Extract Fan

Include extract fan

An extract fan is included within all AHUs by default. This setting may be used to remove the extract fan from the AHU.

Note: This option is only available for **Constant volume** fan types. **On/Off** fans do not allow an extract fan to be included in the AHU and when this fan type is selected the **Include extract fan** check box will be disabled.

Thermostat Location

Controlling zone or thermostat location

While the heat pump may be configured to serve multiple zones, system operation is controlled by a thermostat located in a single "control" zone. Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone supplied by the AHU where the thermostat controlling the heat pump is located.

Dehumidification

Dehumidification control type

There are two options for dehumidification control:

1-None - meet sensible load only, no active de-humidification control;

2-Cool Reheat - cool beyond the dry-bulb temperature set point as required to meet the high humidity setpoint.

Note: If the **2-Cool Reheat** option is selected then some further settings are also required:

a) a **humidistat** must be defined on the HVAC zone dialog of the control zone supplied by the AHU.

b) DesignBuilder v6 requires that a SPM be added to the outlet of the Unitary HP AirToAir AHU. Valid SPM options are: **5-Multi-zone maximum humidity average**, **7-Multi-zone humidity maximum** and **9-Single-zone humidity maximum**.

Tip: To provide humidification, a humidifier should be added to the AHU and a setpoint manager added downstream of the AHU with one of the "minimum humidity" control options.

Supplemental Heater

If the heat pump's DX heating coil output at full load is insufficient to meet the entire heating load, the part load ratio is set equal to 1.0 (compressor and fan are not cycling) and the remaining heating load is passed to the supplemental heating coil. If the outdoor air temperature is below the minimum outdoor air temperature for

compressor operation, the compressor is turned off and the entire heating load is passed to the supplemental gas or electric heating coil.

Maximum supply air temperature from supplemental heater

This setting defines the maximum allowed supply air temperature exiting the heat pump supplemental heating coil.

Maximum outdoor dry-bulb temperature for supplemental heater operation

This setting defines the outdoor air dry-bulb temperature above which the heat pump supplemental heating coil is disabled. The temperature for this input field must be less than or equal to 21°C.

Night Cycle

See [Night cycle description for Generic AHU](#)

Mixed Mode

You can set up Mixed mode controls for the AHU to model optimal interaction between the natural ventilation system (Scheduled or Calculated) and the AHU.

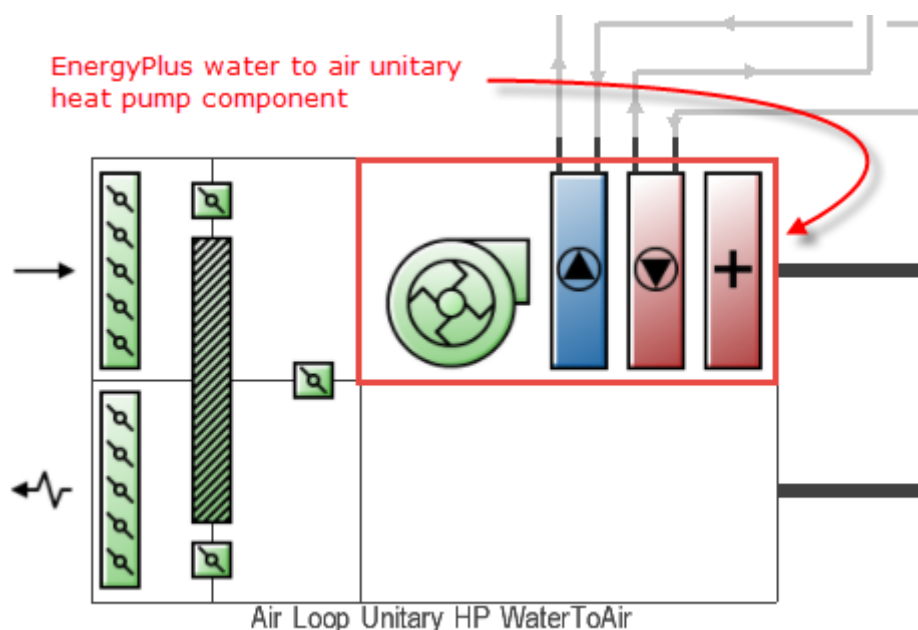
Refer to the main HVAC tab [Mixed Mode](#) section for details on the available options.

Outdoor Air System

See [Outdoor air system description for Generic AHU](#)

Unitary Water To Air Heat Pump AHU

The unitary water-to-air heat pump AHU incorporates an EnergyPlus unitary water-to-air heat pump component, an outdoor air system, optional night cycle operation, an optional heat recovery system, optional pre-heat and pre-cool coils and an optional extract fan. The unitary water-to-air heat pump component itself comprises a [supply fan](#), a [water-to-air heat pump cooling coil](#) component, a [water-to-air heat pump heating coil](#) component, and an electric, hot water or fuel supplementary heating coil component.



The unitary water-to-air heat pump is similar to the [unitary air-to-air heat pump](#) except water is used on the source side, connected to a condenser loop with a [heat exchanger](#) (ground heat exchanger or other type) or a plant loop with a heating source such as a boiler and a cooling source such as a chiller or cooling tower.

Note: The Unitary Water to Air Heat Pump AHU can only be added as part of a [Unitary Water to Air Heat Pump Air loop](#).

A detailed description of how Unitary air loops work is provided in the [EnergyPlus Engineering Reference](#).

General

Name

This is the name that you assign to the AHU which should be unique. If the supplied name is not unique, the software will automatically append a backslash and integer to ensure that there are no duplicate names.

Operation

Availability schedule

This schedule contains information on the availability of the AHU for operation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

When this unit is unavailable (schedule values of 0) then all equipment contained within this AHU will also be switched off.

Supply Air Flow Rates

Supply air flow rate

This setting defines the design supply air flow rate through the heat pump (in m³/s or ft³/min). This volume flow rate is only used when the cooling and heating coil object type is Parameter Estimation. Although a value greater than 0 is required (input cannot be blank or 0), this input is not used for the Equation Fit coil model. Instead, the supply air flow rate is determined by the input in the corresponding Equation Fit coil objects.

Supply Fan

Supply fan placement

There are two options:

- **1-Blow through** where the supply air fan is before the DX cooling/heating coil and the supplementary heating coil.
- **2-Draw through** where the supply air fan is between the DX cooling/heating coil and the supplementary heating coil.

Supply air fan operating mode schedule

The supply air fan operating mode may vary during the simulation based on time-of-day or with a change of season. Schedule values of 0 indicate that the unitary system supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (sometimes referred to as an "AUTO" fan). Schedule values other than 0 denote that the supply fan runs continuously while the heating or cooling coil cycles to meet the load.

The default schedule is "Fan operation mode - Continuous" with a constant value of 1, i.e. that the supply fan runs continuously while the heating or cooling coil cycles to meet the load at all times. To obtain the AUTO fan configuration, select the "Fan operation mode - Cycling" schedule which has a constant value of 0.

Supply fan delay time

This is the time delay for the heat pump supply air fan to shut off after the compressor cycles off in seconds. This value can be obtained from the manufacturer or the heat pump catalog. Enter a value of zero when the heat pump's fan operating mode is continuous. The suggested value is 60 seconds.

Thermostat Location

Controlling zone or thermostat location

While the heat pump may be configured to serve multiple zones, system operation is controlled by a thermostat located in a single "control" zone. Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone supplied by the AHU where the thermostat controlling the heat pump is located.

On Cycle Power Use

Fraction on cycle power use

Enter the fraction of on-cycle power use to adjust the part load fraction based on the off-cycle power consumption due to crankcase heaters, controls, fans etc. Some suggested value values are provided below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	0.01
Good	0.01
Poor	0.03

Dehumidification

Dehumidification control type

There are two options for dehumidification control:

1-None - meet sensible load only, no active de-humidification control;

2-Cool Reheat - cool beyond the dry-bulb temperature set point as required to meet the high humidity setpoint.

Note: If **2-Cool Reheat** is used, a humidistat must be defined in one of the zones supplied by the AHU.

Tip: To provide humidification, a humidifier should be added to the AHU and a [setpoint manager](#) added downstream of the AHU with one of the "minimum humidity" control options.

Cooling Coil

Cooling coil time constant

Enter the time constant for the cooling coil's capacity to reach steady state after startup. Suggested values are shown below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	60
Good	60
Poor	60

Compressor

Maximum cycling rate

Enter the maximum on-off cycling rate for the compressor, which occurs at 50% run time fraction. Suggested values are shown below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	2.5
Good	2.5
Poor	3.0

Supplemental Heater

If the heat pump's DX heating coil output at full load is insufficient to meet the entire heating load, the part load ratio is set equal to 1.0 (compressor and fan are not cycling) and the remaining heating load is passed to the supplemental heating coil. If the outdoor air temperature is below the minimum outdoor air temperature for compressor operation, the compressor is turned off and the entire heating load is passed to the supplemental gas or electric heating coil.

Maximum supply air temperature from supplemental heater

This setting defines the maximum allowed supply air temperature exiting the heat pump supplemental heating coil.

Maximum outdoor dry-bulb temperature for supplemental heater operation

This setting defines the outdoor air dry-bulb temperature above which the heat pump supplemental heating coil is disabled. The temperature for this input field must be less than or equal to 21°C.

Coil Water Flow Rate

Heat pump coil water flow mode

This data specifies the way in which water flow through the heat pump coils will be modelled. It is only used when WatertoAirHeatPump:EquationFit coils are used. There are three options:

- **1-Cycling** which varies water flow through the coil based on the heat pump Part Load Ratio. This control method is appropriate for modelling heat pumps that are outfitted with a solenoid valve which allows water to flow through the coil only when the compressor is active. This is the default setting.
- **2-Constant** which provides a constant water flow regardless of heat pump operation. Remember that EnergyPlus has two coils (a heating coil and a cooling coil) to approximate the operation of one coil that can operate in either heating mode or cooling mode. Therefore, when the water flow mode is constant, there will be full flow through either the heating coil or the cooling coil, but not both at the same time.
- **3-Constant on demand** which provides full flow through the coil whenever there is a load. When there is no load, there is zero flow through the coil.

Night Cycle

See [Night cycle description for Generic AHU](#)

Mixed Mode

You can set up Mixed mode controls for the AHU to model optimal interaction between the natural ventilation system (Scheduled or Calculated) and the AHU.

Refer to the main HVAC tab [Mixed Mode](#) section for details on the available options.

Advanced

Cooling convergence

This numeric value allows the user to determine how close the air side has to be controlled. The lower the value of convergence the better the control of air side conditions and the less the zone temperature fluctuates. However in a poorly designed system, a lower convergence might result in warnings which are caused due to the iteration limit for run time fraction calculation is limited to 20.

Heating convergence

This numeric value allows the user to determine how close the air side has to be controlled. The lower the value of convergence the better the control of air side conditions and the less the zone temperature fluctuates. However in a poorly designed system, a lower convergence might result in warnings which are caused due to the iteration limit for run time fraction calculation is limited to 20.

Outdoor Air System

See [Outdoor air system description for Generic AHU](#)

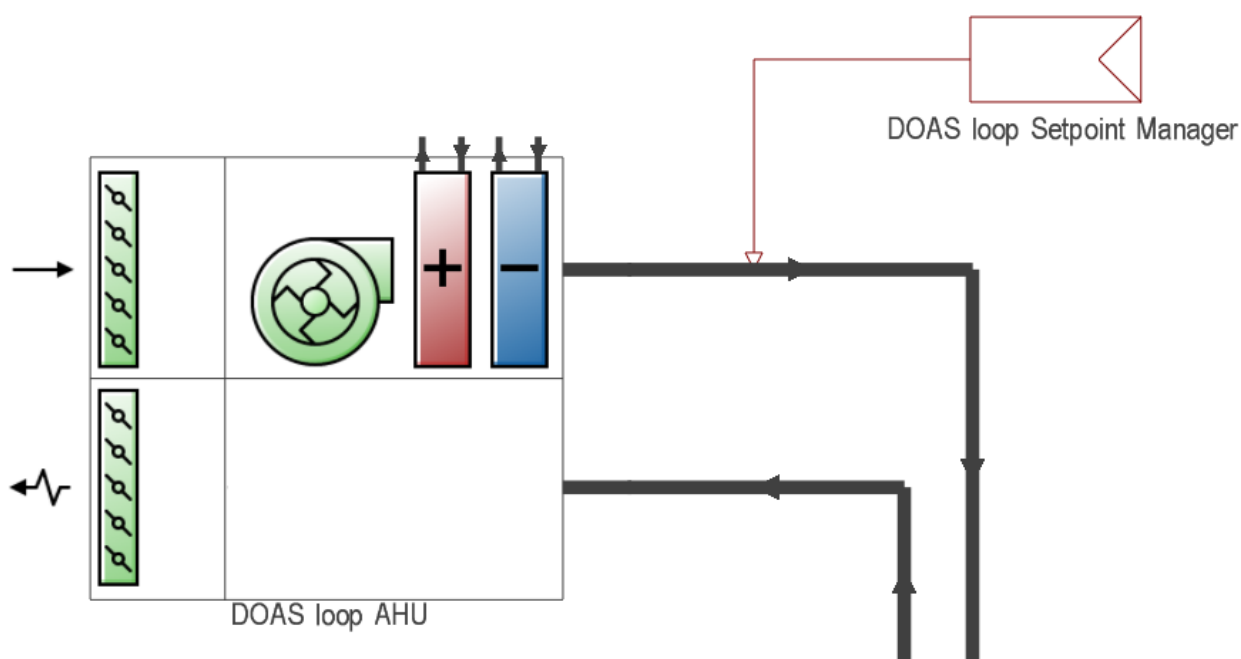
DOAS AHU

The Dedicated Outdoor Air System (DOAS) AHU is a central dedicated outdoor unit that typically delivers outdoor air to multiple air loops. The amount of delivered outdoor air is based on a sum of outdoor air flow rate demands from the downstream AHU outdoor air nodes.

Central DOAS systems can also pre-treat outdoor air before distributing it the connected AHUs, with given precool and preheat air conditions. This is achieved by adding components such as heating and cooling coils and humidifiers to the DOAS AHU in the same way as for standard AHUs.

One restriction with DOAS AHUs is that the fan type must be of the **4-System model** type.

The DOAS AHU dialog currently has 2 fields, only one of which is user-definable. These are described below.



General

Name

This is a unique, system-defined, non-editable name assigned to the AHU.

Operation

Availability schedule

This schedule contains information on the availability of the DOAS AHU for operation. A schedule value greater than 0 (1 is usually used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

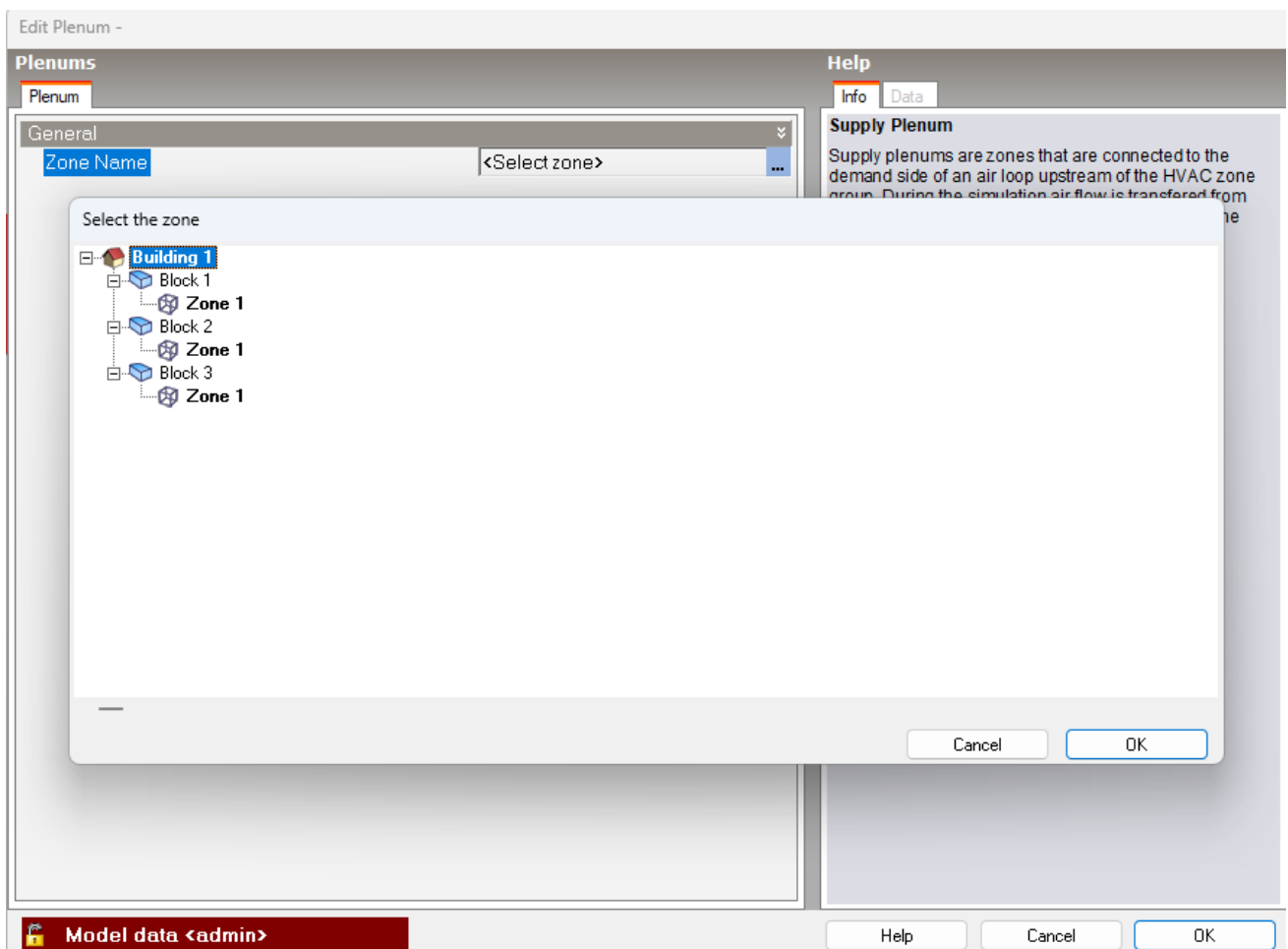
When this unit is unavailable (schedule values of 0) then any equipment contained within this AHU will also be switched off.

Connecting To Standard AHUs

To allow a DOAS AHU to be connected to standard AHUs, it is necessary to enable the DOAS connections in the standard AHUs.

Plenum Data

Supply and Extract plenums are HVAC components much like Splitters and Mixers but they can be associated with a building zone which is used as the supply or extract plenum.



The only item of data required for an HVAC plenum is the name of the associated building zone.

Zone name

If no zone has previously been allocated to the plenum, click on the ellipsis button adjacent to the <Select zone> text to bring up a zone selection control. Only zones that have not already been associated with zone groups or other plenums are available for selection.

See also [Add Supply/Return Plenum Tool](#)

Fans

The following fan types are available when using Detailed HVAC:

- [Fan - Constant Volume](#)
- [Fan - Variable Volume](#)
- [Fan - On/Off](#)
- [Fan - Zone Exhaust](#)

Fan - Constant Volume



Fan:ConstantVolume

Used in:

- Air Handling Unit (Constant Volume)
- Fan Coil Unit

- PTAC,
- PTHP,
- Unitary Heat Cool,
- Unitary Heat Pump

This object models a constant air volume fan that is intended to operate continuously based on a time schedule. This fan will not cycle on and off based on cooling/heating load or other control signals.

General

Name

A unique auto-assigned name for the Fan. Any reference to this fan by another object will use this name.

Type

The type of the constant volume fan is fixed as:

- **2-Constant volume**

Note: To change from constant volume to variable volume you must open the parent [AHU](#) dialog and change the AHU [Fan type](#) to **2-Variable volume**. To allow the AHU Fan type to be edited you must first disconnect the demand side of the air loop from all zone ADUs by deleting connecting branches.

Fan total efficiency

This is the overall efficiency of the fan, i.e. the ratio of the power delivered to the fluid to the electrical input power. It is the product of the motor efficiency and the impeller efficiency. The motor efficiency is the power delivered to the shaft divided by the electrical power input to the motor. The impeller efficiency is the power delivered to the air divided by the shaft power. The power delivered to the air is the mass flow rate of the air multiplied by the pressure rise divided by the air density. This efficiency must be between 0 and 1.

Pressure rise

The pressure rise (in Pascals or inH2O) at full flow and standard (sea level) conditions (20°C and 101325 Pa).

The required pressure rise across an AHU fan is dependent on the duct network supplied by the AHU. For example, if a duct network was sized using a constant pressure loss of 1 Pa/m and the index run of the network (route of highest pressure drop) was say 300 m and the fitting losses were say an additional 20%, the required pressure rise across the fan would be $300 \times 1 \times 1.2 = 360$ Pa.

End-use subcategory

Allows you to specify a user-defined end-use subcategory, e.g., "Central System", etc. A new meter for reporting is created for each unique subcategory (ref: Report Meter). Subcategories are also reported in the ABUPS table. If this field is omitted or blank, the fan will be assigned to the "General" end-use subcategory.

Flow Rates

Maximum flow rate

The full load air volumetric flow rate (m3/sec or ft3/min) at standard temperature and pressure (dry air at 20°C dry-bulb). EnergyPlus uses local barometric pressure to account for altitude using equation for "standard atmospheric" pressure on p 6.1 of the ASHRAE 1997 HOF (SI edition) to initialize the air systems being simulated.

$$p = 101325 * (1 - 2.25577E-05 * Z)^{5.2559}$$

where p =pressure in Pa and Z =altitude in m

Motor

Motor efficiency

The shaft power divided by the electrical power consumed. Must be between 0 and 1.

Motor in airstream fraction

The fraction of the motor heat that is added to the air stream. A value of 0 means that the motor is completely outside the air stream. A value of 1 means that all of the motor heat loss will go into the air stream and act to cause a temperature rise. Must be between 0 and 1.

Operation

Availability schedule

Select the [Schedule](#) that defines whether the fan can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the fan can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the fan is off.

Algorithm

The way that fan and motor efficiency values affect the condition of the air at the fan outlet is given by the following equations:

$$\begin{aligned}Q_{\text{tot}} &= m \cdot \delta P / (e_{\text{tot}} \cdot \rho_{\text{air}}) \\Q_{\text{shaft}} &= e_{\text{motor}} \cdot Q_{\text{tot}} \\Q_{\text{toair}} &= Q_{\text{shaft}} + (Q_{\text{tot}} - Q_{\text{shaft}}) \cdot f_{\text{motortoair}} \\h_{\text{out}} &= h_{\text{in}} + Q_{\text{toair}} / m \\w_{\text{out}} &= w_{\text{in}} \\T_{\text{out}} &= \text{PsyTdbFnHW}(h_{\text{out}}, w_{\text{out}})\end{aligned}$$

Nomenclature:

$f_{\text{motortoair}}$ is the fraction of motor heat generated passed into the air stream
 Q_{tot} is fan power in W
 m is mass flow in kg/s
 δP is the design pressure increase
 e_{tot} is fan total efficiency
 e_{motor} is the motor efficiency
 ρ_{air} is air density at standard conditions (kg/m³)
 Q_{shaft} is the fan shaft power in W
 Q_{toair} is the power entering the air in W
 $h_{\text{out}}, h_{\text{in}}$ are the outlet and inlet specific enthalpies in J/kg
 $w_{\text{out}}, w_{\text{in}}$ are the inlet and outlet air stream humidity ratios
PsyTdbFnHW is the EnergyPlus psychrometric routine relating enthalpy and humidity ratio to temperature.

Fan - Constant Volume Outputs

HVAC, Average, Fan Electric Power[W]
HVAC, Average, Fan Delta Temp[C]
HVAC, Sum, Fan Electric Consumption[J]

Fan Electric Power [W]

This output field contains the average electricity consumption rate for the fan in Watts for the timestep being reported.

Fan Delta Temp [C]

This output field contains the average rise in air temperature across the fan (outlet air temperature minus inlet air temperature) (in °C or °F) for the timestep being reported.

Fan Electric Consumption [J]

This output field contains the electricity consumption of the fan in Joules for the timestep being reported. This output is also added to a report meter with Resource Type = Electricity, End Use Key= Fans, Group Key= System (ref. Report Meter).

Fan - Variable Volume



Fan:VariableVolume

Used in:

- **Air Handling Unit (Variable Volume)**

General

Name

A unique auto-generated name for the fan. Any reference to this fan by another object will use this name.

Type

The type of the variable volume fan is fixed as:

- **2-Variable volume**

Note: To change from variable volume to constant volume you must open the parent [AHU](#) dialog and change the AHU [Fan type](#) to **1-Constant volume**. To allow the AHU Fan type to be edited you must first disconnect the demand side of the air loop from all zone ADUs by deleting connecting branches.

Fan total efficiency

This is the overall efficiency of the fan, i.e. the ratio of the power delivered to the fluid to the electrical input power. It is the product of the motor efficiency and the impeller efficiency. The motor efficiency is the power delivered to the shaft divided by the electrical power input to the motor. The impeller efficiency is the power delivered to the air divided by the shaft power. The power delivered to the air is the mass flow rate of the air multiplied by the pressure rise divided by the air density. This efficiency must be between 0 and 1.

Pressure rise

The pressure rise (in Pascals or inH2O) at full flow and standard (sea level) conditions (20°C and 101325 Pa).

The required pressure rise across an AHU fan is dependent on the duct network supplied by the AHU. For example, if a duct network was sized using a constant pressure loss of 1 Pa/m and the index run of the network (route of highest pressure drop) was say 300 m and the fitting losses were say an additional 20%, the required pressure rise across the fan would be $300 \times 1 \times 1.2 = 360$ Pa.

End-use subcategory

Allows you to specify a user-defined end-use subcategory, e.g., "Central System", etc. A new meter for reporting is created for each unique subcategory (ref: Report Meter). Subcategories are also reported in the ABUPS table. If this field is omitted or blank, the fan will be assigned to the "General" end-use subcategory.

Flow Rates

Fan minimum flow rate input method for fan power

This field is a key/choice field that tells which of the next two fields is filled and is descriptive of how the minimum flow rate is specified for calculating the fan power. The key/choices are:

- **1-Fraction** where the fan power will be calculated using the value specified in the **Minimum flow fraction for fan power** below.
- **2-Fixed flow rate** where the fan power is calculated using the value specified in the **Minimum air flow rate for fan power** below.

Minimum flow fraction for fan power

The minimum air volumetric flow rate for fan power, specified as a fraction of maximum system air flow rate. Must be between 0 and 1. Note that this field is only used to calculate the fan power. This field does not enforce the system air flow rate during simulation.

Minimum air flow rate for fan power

The minimum air volumetric flow rate for fan power, specified as a constant minimum air flow rate (m3/sec or ft3/min). Note that this field is only used to calculate the fan power. This field does not enforce the system air flow rate during simulation.

Maximum flow rate

The full load air volumetric flow rate (m3/sec or ft3/min) at standard temperature and pressure (dry air at 20°C dry-bulb). The program does use local barometric pressure to account for altitude using equation for "standard atmospheric" pressure on p 6.1 of the ASHRAE 1997 HOF (SI edition) to initialize the air systems being simulated.

$$p=101325*(1-2.25577E-05*Z)^{5.2559}$$

where p=pressure in Pa and Z=altitude in m

Minimum flow rate

The minimum air volumetric flow rate (m3/sec or ft3/min) for the fan at standard temperature and pressure (see Maximum Flow Rate field above for details).

Motor

Motor efficiency

The shaft power divided by the electrical power consumed. Must be between 0 and 1.

Motor in airstream fraction

The fraction of the motor heat that is added to the air stream. A value of 0 means that the motor is completely outside the air stream. A value of 1 means that all of the motor heat loss will go into the air stream and act to cause a temperature rise. Must be between 0 and 1.

Fan Coefficients

Fan coefficient data under this header provides the coefficients C2 to C5 in the fourth order polynomial curve giving the fraction of full load power (PLF) as a function of flow fraction (FF). Flow fraction is the air mass flow rate divided by the maximum air mass flow rate.

$$PLF = C_1 + C_2 \cdot FF + C_3 \cdot FF^2 + C_4 \cdot FF^3 + C_5 \cdot FF^4$$

Fan coefficient 1

The constant coefficient C_1 in the above curve

Fan coefficient 2

The linear coefficient C_2 in the above curve

Fan coefficient 3

The quadratic coefficient C_3 in the above curve

Fan coefficient 4

The cubic coefficient C_4 in the above curve

Fan Coefficient 5

The coefficient C_5 in the above curve

Operation

Availability schedule

[Schedule](#) that defines whether the fan can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the fan can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the fan is off.

Algorithm

The algorithm used to calculate energy consumption and air outlet conditions in variable flow fans are as follows.

$$\begin{aligned} f_{\text{flow}} &= m / m_{\text{max}} \\ f_{\text{pl}} &= c_1 + c_2 \cdot f_{\text{flow}} + c_3 \cdot f_{\text{flow}}^2 + c_4 \cdot f_{\text{flow}}^3 + c_5 \cdot f_{\text{flow}}^4 \\ Q_{\text{tot}} &= f_{\text{pl}} \cdot m \cdot \delta P / (e_{\text{tot}} \cdot \rho_{\text{air}}) \\ Q_{\text{shaft}} &= e_{\text{motor}} \cdot Q_{\text{tot}} \\ Q_{\text{toair}} &= Q_{\text{shaft}} + (Q_{\text{tot}} - Q_{\text{shaft}}) \cdot f_{\text{motortair}} \\ h_{\text{out}} &= h_{\text{in}} + Q_{\text{toair}} / m \\ w_{\text{out}} &= w_{\text{in}} \\ T_{\text{out}} &= \text{PsyTdbFnHW}(h_{\text{out}}, w_{\text{out}}) \end{aligned}$$

Nomenclature:

f_{flow} is the flow fraction of design or maximum flow
 $f_{\text{motortair}}$ is the fraction of motor heat generated passed into the air stream
 f_{pl} is the part load factor
 Q_{tot} is fan power in W
 m is mass flow in kg/s
 δP is the design pressure increase
 e_{tot} is fan total efficiency
 e_{motor} is the motor efficiency
 ρ_{air} is air density at standard conditions (kg/m³)
 Q_{shaft} is the fan shaft power in W
 Q_{toair} is the power entering the air in W
 $h_{\text{out}}, h_{\text{in}}$ are the outlet and inlet specific enthalpies in J/kg
 $w_{\text{out}}, w_{\text{in}}$ are the inlet and outlet air stream humidity ratios

PsyTdbFnHW is the EnergyPlus psychrometric routine relating enthalpy and humidity ratio to temperature.

Fan - Variable Volume Outputs

HVAC,Average,Fan Electric Power[W]
HVAC,Average,Fan Delta Temp[C]
HVAC,Sum,Fan Electric Consumption[J]

Fan Electric Power [W]

This output field contains the average electricity consumption rate for the fan in Watts for the timestep being reported.

Fan Delta Temp [C]

This output field contains the average rise in air temperature across the fan (outlet air temperature minus inlet air temperature) (in °C or °F) for the timestep being reported.

Fan Electric Consumption [J]

This output field contains the electricity consumption of the fan in Joules for the timestep being reported. This output is also added to a report meter with Resource Type = Electricity, End Use Key= Fans, Group Key= System (ref. Report Meter).

Fan - On/Off



Used in:

- Fan Coil Unit
- PTAC,
- PTHP,
- Unitary Heat Cool,
- Unitary Heat Pump

The On/off fan models a constant air volume fan that is intended to cycle on and off in tandem with a cooling or heating system (i.e., AUTO fan control mode). The fan can also operate continuously like a [Constant volume fan](#). If modelling continuous operation and this object is used as part of a system that utilizes Coil:Heating:Gas, Coil:Cooling:DX:SingleSpeed or Coil:Heating:DX:SingleSpeed, the user should confirm proper air flow rates (coil and fan max load fraction is less than or equal to 1 for all values of coil part-load ratio).

Multi-speed fan operation can also be modelled when the On/off fan is included as part of a compound object that allows multiple fan speeds (e.g., Unitary Heat Cool, PTAC, etc.). In this case, the ratio of the compound object air flow rate to the fan's maximum air flow rate is used to determine the power at alternate fan speeds. The input for Fan Power Ratio Function of Speed Ratio Curve Name must be entered to model multi-speed fan operation. An optional fan total efficiency ratio curve is also available to model efficiency differences at alternate fan speeds.

Name

A unique system assigned name for an instance of an On/off fan. Any reference to this fan by another object will use this name.

Fan total efficiency

This is the overall efficiency of the fan, i.e. the ratio of the power delivered to the fluid to the electrical input power. It is the product of the motor efficiency and the impeller efficiency. The motor efficiency is the power

delivered to the shaft divided by the electrical power input to the motor. The impeller efficiency is the power delivered to the air divided by the shaft power. The power delivered to the air is the mass flow rate of the air multiplied by the pressure rise divided by the air density. This efficiency must be between 0 and 1.

Pressure rise

The pressure rise (in Pascals or inH2O) at full flow and standard (sea level) conditions (20°C and 101325 Pa).

The required pressure rise across an AHU fan is dependent on the duct network supplied by the AHU. For example, if a duct network was sized using a constant pressure loss of 1 Pa/m and the index run of the network (route of highest pressure drop) was say 300 m and the fitting losses were say an additional 20%, the required pressure rise across the fan would be $300 \times 1 \times 1.2 = 360$ Pa.

End-use subcategory

Allows you to specify a user-defined end-use subcategory, e.g., "Central System", etc. A new meter for reporting is created for each unique subcategory (ref: Report Meter). Subcategories are also reported in the ABUPS table. If this field is omitted or blank, the fan will be assigned to the "General" end-use subcategory.

Maximum flow rate

The full load air volumetric flow rate (m3/sec or ft3/min) at standard temperature and pressure (dry air at 20°C dry-bulb). EnergyPlus uses local barometric pressure to account for altitude using equation for "standard atmospheric" pressure on p 6.1 of the ASHRAE 1997 HOF (SI edition) to initialize the air systems being simulated.

$$p = 101325 * (1 - 2.25577E-05 * Z)^{5.2559}$$

where p=pressure in Pa and Z=altitude in m

Motor efficiency

The shaft power divided by the electrical power consumed. Must be between 0 and 1.

Motor in airstream fraction

The fraction of the motor heat that is added to the air stream. A value of 0 means that the motor is completely outside the air stream. A value of 1 means that all of the motor heat loss will go into the air stream and act to cause a temperature rise. Must be between 0 and 1.

Fan power ratio function of speed ratio curve

Select an [Exponent Performance curve](#). This curve is used to simulate multi-speed fan motors, representing the ratio of actual fan power to rated fan power when a change in fan speed occurs.

Fan efficiency ratio function of speed ratio curve

Enter the name of a [Quadratic](#) or [Cubic](#) Performance curve. This curve is used to simulate multi-speed fan motors, representing the ratio of actual fan total efficiency to rated fan total efficiency when a change in fan speed occurs.

Operation

Availability schedule

Select the [Schedule](#) that defines whether the fan can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the fan can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the fan is off.

Algorithm

The on/off fan model is similar to the simple constant volume fan model with the exception that the on/off fan may cycle on and off during a simulation time step. The cycling rate of the fan is known as the run time fraction. The calculation of run time fraction accounts for the part-load losses of other equipment used in the HVAC system. A part-load factor (a.k.a. part-load ratio) is first calculated for the fan as the ratio of the actual operating mass flow rate to the maximum fan mass flow rate. The run time fraction is then calculated as the part-load factor divided by the part-load fraction. The part-load fraction is determined by other HVAC equipment in the simulation.

$$f_{\text{flow}} = m / m_{\text{max}}$$
$$\text{RTF} = f_{\text{flow}} / \text{PLF}$$

The total fan power is then calculated as the maximum fan power multiplied by the run time fraction.

$$Q_{\text{tot}} = \text{RTF} \cdot [m \cdot \delta P / (e_{\text{tot}} \cdot \rho_{\text{air}})]$$
$$Q_{\text{shaft}} = e_{\text{motor}} \cdot Q_{\text{tot}}$$
$$Q_{\text{toair}} = Q_{\text{shaft}} + (Q_{\text{tot}} - Q_{\text{shaft}}) \cdot f_{\text{motortoair}}$$
$$h_{\text{out}} = h_{\text{in}} + Q_{\text{toair}} / m$$
$$w_{\text{out}} = w_{\text{in}}$$
$$T_{\text{out}} = \text{PsyTdbFnHW}(h_{\text{out}}, w_{\text{out}})$$

Nomenclature:

f_{flow} is the flow fraction or part load ratio
 $f_{\text{motortoair}}$ is the fraction of motor heat generated passed into the air stream
 Q_{tot} is fan power in W
 m is mass flow in kg/s
 m_{max} is the maximum or design mass flow in kg/s
RTF is runtime fraction
PLF is part load factor
 δP is the design pressure increase
 e_{tot} is fan total efficiency
 e_{motor} is the motor efficiency
 ρ_{air} is air density at standard conditions (kg/m³)
 Q_{shaft} is the fan shaft power in W
 Q_{toair} is the power entering the air in W
 $h_{\text{out}}, h_{\text{in}}$ are the outlet and inlet specific enthalpies in J/kg
 $w_{\text{out}}, w_{\text{in}}$ are the inlet and outlet air stream humidity ratios
PsyTdbFnHW is the EnergyPlus psychrometric routine relating enthalpy and humidity ratio to temperature.

Fan - On/Off Outputs

HVAC, Average, Fan Electric Power[W]
HVAC, Average, Fan Delta Temp[C]
HVAC, Sum, Fan Electric Consumption[J]
HVAC, Average, On/Off Fan Runtime Fraction

Fan Electric Power [W]

This output field contains the average electricity consumption rate for the fan in Watts for the timestep being reported.

Fan Delta Temp [C]

This output field contains the average rise in air temperature across the fan (outlet air temperature minus inlet air temperature) (in °C or °F) for the timestep being reported.

Fan Electric Consumption [J]

This output field contains the electricity consumption of the fan in Joules for the timestep being reported. This output is also added to a report meter with Resource Type = Electricity, End Use Key= Fans, Group Key= System (ref. Report Meter).

On/Off Fan Runtime Fraction

This output field contains the fraction of time that this fan operated for the timestep being reported

Fan - System Model

The System Model fan models fans of various types using a relatively simple engineering model. It can be used in variable air volume, constant volume, on-off cycling, two-speed, or multi-speed applications. It was designed as a replacement for all of the other fan types: [Constant Volume](#), [On/Off](#), and [Variable Volume](#).

The electric power consumed by the fan can be directly input or autosized using one of three optional methods. For fans that can vary the volume flow rate the performance can be described using a separate performance curve.

General

Name

A unique auto-generated name for the fan. Any reference to this fan by another object will use this name.

Type

The type of the System Model fan is fixed as:

- **4-System model**

Design pressure rise

This is the total system design pressure rise experienced by the fan (in Pascals or inH₂O) at full flow rate and altitude-adjusted standard density of dry air at 20°C drybulb.

Electric Power

Design electric power consumption method

Select from:

- **1-Autosize**, where EnergyPlus will autosize the [Electric power per unit flow rate](#) using one of the methods listed below under [Design power sizing method](#).
- **2-Manual input**, where the [Design electric power consumption](#) is entered manually in the setting below.

Design electric power consumption

This setting is available when the [Design electric power consumption method](#) is set to **2-Manual input**. It is the electric power consumption at the full [Maximum flow rate](#) and [Pressure rise](#) (in W). The value entered in this field is used to determine the fan efficiency.

Design power sizing method

This setting is used when the [Design electric power consumption method](#) is set to **1-Autosize** to select how the fan's Design electric power consumption is sized. There are three choices:

- **1-Power per flow**, where the value entered for [Electric power per unit flow rate](#) is used to size the Design electric power consumption. This method is useful during early-phase design modelling when little information is available for determining the Design pressure rise. Although the pressure rise is not

used to size the [Design electric power consumption](#) it is still used to determine the heat added to the air stream as a result of the work done by the fan.

- **2-Power per flow per pressure**, where the value entered for [Electric power per unit flow rate per unit pressure](#) is used to size the [Design electric power consumption](#). This method takes into account the Design Pressure Rise when sizing the Design Electric Power Consumption.
- **3-Total efficiency and pressure**, where the values entered for [Fan total efficiency](#) and [Design pressure rise](#) are used to size the power.

Electric power per unit flow rate

This setting is used when the [Design electric power consumption method](#) is set to **1-Autosize** and the [Design power sizing method](#) is set to **1-Power per flow**. This value has units of W/(m³/s) and is used to scale the [Design electric power consumption](#) directly from the [Maximum flow rate](#). It is defined such that

Design electric power consumption = (Electric power per unit flow rate) * (Maximum flow rate).

Electric power per unit flow rate per unit pressure

This setting is used when the [Design electric power consumption method](#) is set to **1-Autosize** and the [Design power sizing method](#) is set to **1-Power per flow per pressure**. This value has units W/((m³/s)-Pa) and is used to scale the [Design electric power consumption](#) from the [Maximum flow rate](#) and the [Pressure rise](#). It is defined such that

Design electric power consumption = (Electric power per unit flow rate per unit pressure) * (Maximum flow rate) * Design pressure rise).

The default is 1.66667.

Fan total efficiency

This is the overall efficiency of the fan, i.e. the ratio of the power delivered to the fluid to the electrical input power. It is the product of the motor efficiency and the impeller efficiency. It is used when the [Design electric power consumption method](#) is set to **1-Autosize** and the [Design power sizing method](#) is set to **3-Total efficiency and pressure**. This value is used to determine the [Design electric power consumption](#) from the [Maximum flow rate](#) and the [Pressure rise](#). It is defined such that

Design electric power consumption = (Maximum flow rate) * (Design pressure rise) / (Fan Total Efficiency).

The default is 0.7.

Flow Rates

Maximum flow rate

Enter the full load air volumetric flow rate (m³/sec or ft³/min) installed in the HVAC system. This volume flow rate will be converted to a mass flow rate using an altitude-adjusted standard density of dry air at 20 °C drybulb. This full-speed flow rate and serves as the upper limit for fans that can vary their flow rate.

This field can be autosized and is greyed out if the parent HVAC component (e.g. AHU) defines the design air flow rate.

Motor

Motor efficiency

The fan shaft power divided by the electrical power consumed. Must be between 0 and 1.

Motor in airstream fraction

This is the fraction of the motor heat that is added to the air stream. The value must be between 0 and 1. A value of 0 means fan motor is located completely outside of the air stream and none of the motor's heat is added to the air stream. A value of 1.0 means the motor is located completely inside of air stream and all of the motor's heat is added to the air stream.

Note: Regardless of the value here there will be heat added to the air stream as a result of the work done to move the air. This setting only describes what happens to the heat generated as a result of the motor's inefficiency. The heat lost from the motor that is not added to the air stream can be added to the surrounding thermal zone where the motor is located using the settings below.

Motor loss added to zone

The heat lost from the motor that is not added to the air stream can be added to the thermal zone where the motor is located by checking this checkbox and selecting the zone in the setting below. If the fan is outdoors, or the motor's thermal losses are not to be modeled then leave this unchecked.

Motor loss zone

When the **Motor loss added to zone** checkbox is checked, you must select the zone in which the fan motor is located. In this case, the portion of the motor's thermal losses that are not added to the air stream are added to the surrounding thermal zone as internal heat gains.

Motor loss radiative fraction

When the **Motor loss added to zone** checkbox is checked, you can define the split between thermal radiation and convection for the heat losses from the fan motor. If this field is left at the default zero value all the losses will be convective. Values should be between 0 and 1.

Speed Control

Speed control method

This setting is used to select how the fan speed can be varied. There are two choices:

- **1-Continuous**, which indicates that the fan speed is variable and can change smoothly up to the **Maximum flow rate**.
- **2-Discrete**, which indicates that the fan can operate only at specific speed settings and cannot be continuously varied.

This input informs the program how power should be calculated with Discrete control using time-weighted averaging and Variable control using flow-weighted averaging.

Tip: For constant volume or on-off fans use **2-Discrete** with [Number of speeds](#) set to 1. For variable air volume fans use **1-Continuous**, in which case, a fan performance curve must be selected for [Electric power function of flow fraction curve](#).

Electric power minimum flow rate fraction

This setting is only available when the [Speed control method](#) is set to **1-Continuous**. It is used to describe how low a variable speed fan can be operated. This value is used to calculate the fan power at low flow rates but it does not enforce a lower end of the air flow during simulation. The value is a fraction of the [Maximum flow rate](#) and should be between 0 and 1.

Electric power function of flow fraction curve

Select the performance curve that describes how electric power consumption varies with air flow rate. The independent “x” variable of the performance curve is a normalized flow fraction defined as the current flow rate divided by the [Maximum flow rate](#). The model actually uses the ratio of (moist) air mass flow rates with the numerator taking account of humidity and barometric pressure. The dependent variable that is the result of the performance curve or lookup table is a fraction that is multiplied by the [Design electric power consumption](#) to determine the electric power use as a function of flow rate. Any of the single-independent-variable curves can be used.

This field is available if the [Speed control method](#) is set to **1-Continuous**. It is also used when the [Speed control method](#) is set to **2-Discrete** and the [Number of speeds](#) is greater than 1 and [Electric power fraction from curve](#) is checked.

Number of speeds

This numeric field is used to specify the number of different speed levels available when Speed Control Method is set to Discrete. This field and the remaining field sets are not used when the [Speed control method](#) is set to **1-Continuous**. For a constant volume fan enter a value of 1.0. A value of 1.0 will use the fan's maximum design and no additional field sets are needed. When set to a value greater than 1 then a pair of flow and power fraction inputs are provided for each speed in the remaining input fields.

Electric power fraction from curve

Check this checkbox is available when [Speed control method](#) is set to **2-Discrete** and the [Number of speeds](#) is greater than 1. In this case checking this checkbox indicates that EnergyPlus should calculate the electric power fraction for each discrete speed.

Paired values of Speed flow fraction and Speed electric power fraction

Enter a pair of values for the flow fraction and electric power fraction for each speed. The sets should be defined in increasing order so that the flow fractions become larger in subsequent field sets. Typically the highest speed level will match the design maximum and have fractions of 1.0.

Speed <#> Flow fraction

This is the flow fraction for the fan speed. It is multiplied by the [Maximum flow rate](#) to obtain the flow rate when operating at this speed.

Speed <#> Electric power fraction

This is the electric power fraction for the fan speed. It is multiplied by the [Design electric power consumption](#) to obtain the power consumption when operating at this speed. This field is available only if the [Electric power fraction from curve](#) checkbox is unchecked. Otherwise, if it is checked, the power at this speed will be determined using the performance curve selected for [Electric power function of flow fraction curve](#).

Operation

Availability schedule

This is the [Schedule](#) that defines whether the fan can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the fan can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the fan is off.

This setting is non-editable in cases where the operation availability schedule is defined by the parent HVAC component (e.g. AHU).

Fan - Zone Exhaust

The zone exhaust fan provides a way to extract zone air to outside. It is a standalone component and unlike the other Detailed HVAC fans it does not link in with the main Air loop. It can also impact air flows in central air handlers by decreasing the flow of return air and sometimes increasing the outdoor air flow rate.

There are several control options available for the exhaust fan including an on/off availability schedule, interaction with system availability managers, minimum zone air temperature control limits and a variable flow fraction schedule. The way in which the exhaust fan impacts central air system can be controlled by declaring what portion of the flow has been “balanced” by simple airflow from infiltration, ventilation, or mixing. However it is important to note that presence of an exhaust fan does not by itself drive any simple airflow such as infiltration, ventilation, or zone mixing.

Note: When used with **Calculated natural ventilation**, Zone exhaust fans can take part in the Airflow Network causing a negative pressure in the zone which can be balanced by flows into the zone through other openings and cracks. In this case a single corresponding [vent](#) from the **Zone Exhaust Fans** category must be placed on an *external* surface of the building zone.

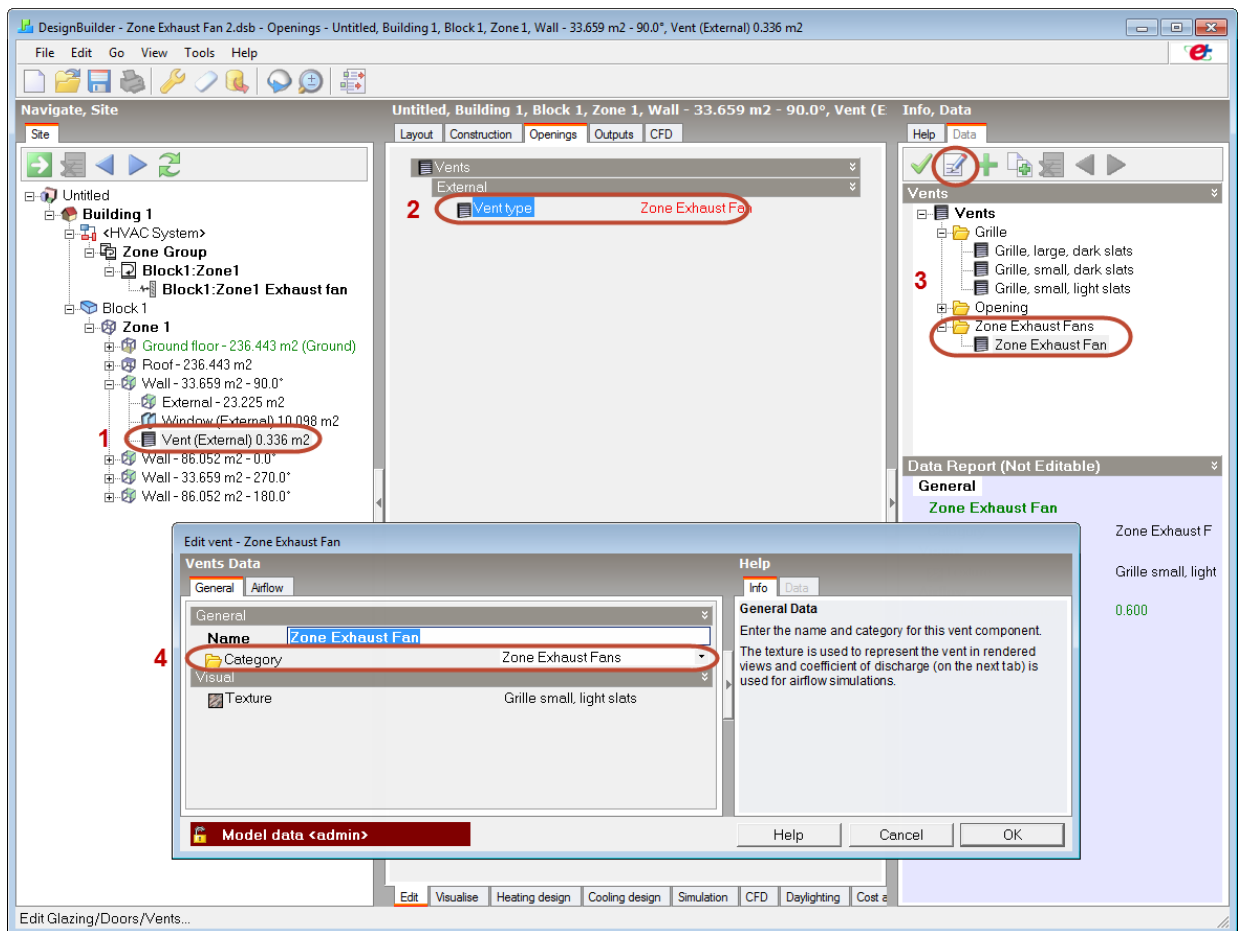
Note: To obtain balancing when using **Scheduled natural ventilation** the main natural ventilation airflow must be coordinated with the flows defined for the exhaust fan as there is no comprehensive automatic mass balancing between air system flows, exhaust flows, and the separate simple airflows.

Zone exhaust fans differ from the other fans in EnergyPlus HVAC in that they can stand on their own in a zone rather than serving as one part of an HVAC air system.

Summary of zone exhaust fan data entry

A zone exhaust fan is defined differently depending on whether the Calculated natural ventilation model option is selected or not.

- **When *not* using Calculated natural ventilation** simply add an Exhaust fan to the HVAC zone and make any settings on the dialog.
- **When using Calculated natural ventilation** then as well as the Exhaust fan in the HVAC zone you must also include a single external *rectangular* vent from the **Zone Exhaust Fans** category somewhere in the zone by drawing it at surface level on an *exterior* surface. The purpose of the vent is to include the zone exhaust fan as an airflow path in the EnergyPlus Airflow Network. The numbered steps that follow refer to the screenshot below. Having drawn the vent object at surface level:
 1. Navigate to the vent object.
 2. Select the **vent type** model data on the Openings tab.
 3. Select the pre-defined **Zone Exhaust Fan** vent component.
 4. Alternatively create a new vent component and set its category to **Zone Exhaust Fans**.



The end result should be that each zone with an HVAC Exhaust fan has exactly 1 *rectangular* vent from the **Zone Exhaust Fans** category placed on an *exterior* surface.

Tip: It is possible to simply add a single zone exhaust fan to a zone group without any other HVAC equipment to simulate cases where no HVAC equipment (apart from the exhaust fan) is required. Zone exhaust fans can equally be mixed with other zone equipment data including standard extract components.

Name

A unique user-assigned name for an instance of a zone exhaust fan. Any reference to this fan by another object will use this name.

End-Use subcategory

Allows you to specify a user-defined end-use subcategory, e.g., "Kitchen Exhaust", "Fume Hoods", etc. A new meter for reporting is created for each unique subcategory (ref: Output:Meter object). Subcategories are also reported in the ABUPS table. If this field is omitted or blank, the fan will be assigned to the "General" end-use subcategory.

Fan Data

Fan total efficiency

This is the overall efficiency of the fan, i.e. the ratio of the power delivered to the fluid to the electrical input power. It is the product of the motor efficiency and the impeller efficiency. The motor efficiency is the power delivered to the shaft divided by the electrical power input to the motor. The impeller efficiency is the power delivered to the air divided by the shaft power. The power delivered to the air is the mass flow rate of the air

multiplied by the pressure rise divided by the air density. This efficiency must be between 0 and 1. The default is 0.6.

Pressure rise

The pressure rise (in Pascals or inH2O) at full flow and standard (sea level) conditions (20°C and 101325 Pa).

Maximum flow rate

The full load air volumetric flow rate (in m3/sec or ft3/min) at standard temperature and pressure (dry air at 20°C drybulb). The program does use local barometric pressure to account for altitude using equation for "standard atmospheric" pressure on p 6.1 of the ASHRAE 1997 HOF (SI edition) to initialize the air systems being simulated.

$$p = 101325 * (1 - 2.25577E-05 * Z)^{5.2559}$$

where p=pressure in Pa and Z=altitude in m

System availability manager coupling mode

This setting can be used to control whether the exhaust fan should operate independently or not. For example, when a night cycle availability manager turns on the central air system for freeze protection, this field can be used to control if the zone exhaust fans should also run at the same time or not. The options are:

- **1-Coupled** which indicates that the exhaust fan should be integrated with the system availability manager so that the fan runs when the air system is forced to run.
- **2-Decoupled** which indicates that the exhaust fan should operate on its own and ignore the system availability manager's requests so that the exhaust fan can remain off when the air system runs.

The default is **1-Coupled**.

Apply minimum zone temperature limit schedule

If you would like to define a minimum zone temperature limit schedule (below) then check this option.

Minimum zone temperature limit schedule

This field is optional. If it is not used then there will be no temperature-related control over the operation of the exhaust fan. If the option is used, then enter the name of a schedule with values for zone temperature values (in °C). The fan's control will be based on a comparison between the current zone air temperature and the schedule values. If the zone is warmer than the scheduled limit, then the fan will operate. When balancing with Scheduled natural ventilation, this feature can be used to coordinate exhaust fan operation with the main natural ventilation controls for minimum indoor temperature.

Apply balanced exhaust fraction schedule

If you would like to define a balanced exhaust fraction schedule (below) then check this option.

Balanced exhaust fraction schedule

This field is optional. If it is not used, then all the exhaust air flow is assumed to be unbalanced by any simple airflows, such as infiltration, ventilation, or zone mixing. Unbalanced exhaust is then modelled as being provided by the outdoor air system in the central air system. The modelling of unbalanced will reduce the flow rates at the zone's return air node by the flow rate that is being exhausted and will insure that the outdoor air flow rate is sufficient to serve the exhaust. If this field is used, then enter the name of a schedule with fractional values between 0.0 and 1.0, inclusive. This fraction is applied to the exhaust fan flow rate and the model tracks the portion of the exhaust that is "balanced." Balanced exhaust is then modelled as being provided by simple airflows and does not impact the central air system return air or outdoor air flow rates. For example, if a kitchen zone with an exhaust fan is designed to draw half of its make up air from a neighbouring dining room and the other half from the outdoor air system, then a schedule value of 0.5 could be used here.

Operation

Availability schedule

The availability schedule denotes whether the fan can run during a given time period. A schedule value of 0 indicates that the fan is off for that time period. A schedule value greater than 0 indicates that the fan can operate during the time period.

Apply flow fraction schedule

If you would like to define a Flow fraction schedule (below) then check this option.

Flow fraction schedule

This field is optional. If it is not used then the fan operates at the maximum flow rate. If a schedule is selected here, then it should contain fractional values between 0.0 and 1.0, inclusive. During the simulation the flow rate provided by the fan will be this fraction times the maximum flow rate. This allows a variable speed exhaust fan to be modelled according to a schedule.

Humidifier - Steam Electric



Humidifier:Steam:Electric

Used in:

- Air Handling Units

The electric steam humidifier is a component that represents an electrically heated, self contained steam humidifier. The component uses electrical energy to convert ordinary tap water to steam which it then injects into the supply air stream by means of a blower fan. The actual unit might be an electrode-type humidifier or a resistance-type humidifier.

The humidifier model includes local control of the humidifier unit to meet a humidity ratio setpoint on its air outlet node.

Note: To ensure that the humidifier will add moisture to meet the humidity ratio setpoint, a Minimum humidity [Setpoint manager](#) must be added *immediately downstream* of the Humidifier (**4-Multi-zone minimum humidity average** or **6-Multi-zone humidity minimum** or **8-Single zone humidity minimum**).

More information on controlling humidity is available in the [Humidity control](#) section.

General

Name

A unique system assigned name for a particular humidifier unit. Any reference to this unit by another object will use this name.

Rated capacity

The nominal full output water addition rate of the unit (in m³/sec or gal/min of water at 5.05°C).

Note: The Rated capacity *cannot* be autosized in current versions of EnergyPlus, but if you are not sure of the correct value to use, it is possible to enter a very high value (e.g. 1m³/s) to ensure that humidification loads can always be met.

Rated power

The nominal full output power consumption of the unit (in W), exclusive of the blower fan power consumption and any standby power. This field can be autosized. When it is autosized, the calculations are based on the [rated capacity](#) and the enthalpy rise of the feed water from the reference temperature of liquid water at 20°C to a saturated steam at 100°C.

Rated fan power

The nominal full output power consumption of the blower fan (in W).

Standby power

The standby power consumption (in W). This amount of power will be consumed whenever the unit is available (as defined by the availability schedule).

Operation

Availability schedule

[Schedule](#) that defines whether the unit can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the fan can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

Humidifier - Steam Electric Outputs

HVAC,Average,Humidifier Water Consumption Rate[m3/s]
 HVAC,Sum,Humidifier Water Consumption[m3]
 HVAC,Average,Humidifier Electric Power[W]
 HVAC,Sum,Humidifier Electric Consumption[J]
 Zone,Meter,Humidifier:Water [m3]
 Zone,Meter,Humidifier:Electricity [J]
 HVAC,Average,Humidifier Water From Storage Tank Rate [m3/s]
 HVAC,Sum,Humidifier Water From Storage Tank [m3]
 HVAC,Average,Humidifier Water Starved By Storage Tank Rate [m3/s]
 HVAC,Sum,Humidifier Water Starved By Storage Tank [m3]
 Zone,Meter,Humidifier:MainsWater [m3]
 HVAC,Sum,Humidifier Water Mains Draw [m3]

Humidifier Water Consumption Rate[m3/s]

This field reports the water consumption rate of the steam humidifier in cubic meters of water per second.

Humidifier Water Consumption[m3]

This output is the cubic meters of water consumed by the steam humidifier over the timestep being reported.

Humidifier Electric Power[W]

This output is the electricity consumption rate in Watts of the steam humidifier.

Humidifier Electric Consumption[J]

This is the electricity consumption in Joules of the steam humidifier over the timestep being reported.

Humidifier:Water [m3]

This meter output contains the sum of the water consumed (in cubic meters of water during the report timestep) by all the steam humidifiers at the HVAC level in the simulation.

Humidifier:Electricity [J]

This meter output contains the sum of the electricity consumed (in Joules during the report timestep) by all the steam humidifiers at the HVAC level in the simulation.

Humidifier Water From Storage Tank Rate [m3/s]

Humidifier Water From Storage Tank [m3]

These outputs contain the rate and volume of water obtained from water storage tank. These are only present if the humidifier is connected to a Water Storage Tank for its water supply.

Humidifier Water Starved By Storage Tank Rate [m3/s]

Humidifier Water Starved By Storage Tank [m3]

These outputs contain the rate and volume of water that could not be obtained from the water storage tank. The component will still operate as if it did get all the water with the balance obtained directly from the mains

Humidifier Water Mains Draw [m3]

This output contains the volume of water obtained from the mains.

Evaporative Cooler Data

An evaporative cooler is a type of cooling component that provides free adiabatic cooling to an air stream purely through the evaporation of water. Evaporative cooling differs from typical air conditioning systems which use vapour-compression or absorption refrigeration cycles. Evaporative cooling works by employing water's large enthalpy of vaporisation. The temperature of dry air can be dropped significantly through the phase transition of liquid water to water vapour, which requires much less energy than refrigeration. In dry climates, it also has the added benefit of conditioning the air with more moisture for the comfort of occupants. Unlike refrigeration, it requires a water source, and must continually consume water to operate.

As water is evaporated, energy is lost from the air, reducing its temperature. The lowest temperature that an evaporative cooler can cool to is the incoming air wet-bulb temperature.

Two types of evaporative cooler are supported by DesignBuilder EnergyPlus: direct and indirect systems.

- **Direct** evaporative cooling (open circuit) is used to lower the temperature of air by using latent heat of evaporation, changing liquid water to water vapour. In this process, the energy in the air does not change. Warm dry air is changed to cool moist air. The heat of the outside air is used to evaporate water.
- **Indirect** evaporative cooling (closed circuit) is similar to direct evaporative cooling, but uses some type of heat exchanger. The cooled moist air never comes in direct contact with the conditioned environment.

In DesignBuilder evaporative coolers can be placed within an AHU in much the same way that water and DX cooling coils are placed. They can also be selected as outdoor air [pretreatment](#) components within the AHU dialogs. Five types of evaporative coolers are provided any of which can be selected when placing the component or from the Evaporative cooler dialog:

- [1-Direct CelDekPad](#)
- [2-Direct Research Special](#)
- [3-Indirect CelDekPad](#)
- [4-Indirect Wet Coil](#)
- [5-Indirect Research Special](#)

Evaporative Cooler - Direct CelDekPad

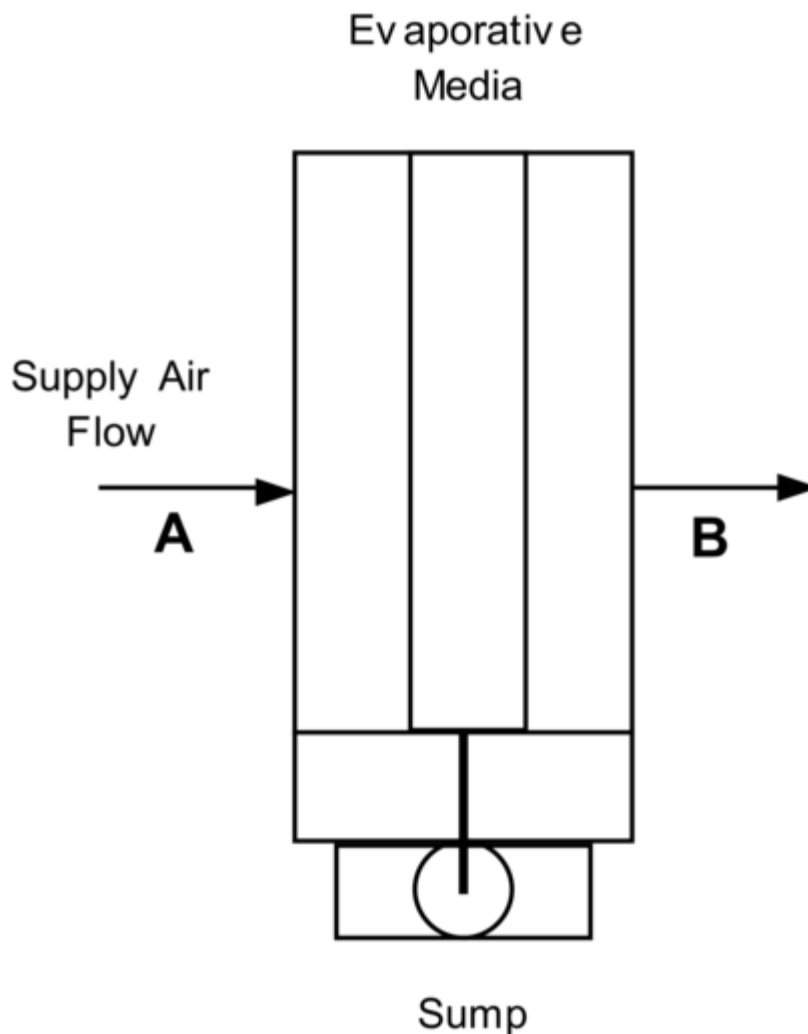


EvaporativeCooler:Direct:CelDekPad

Used in:

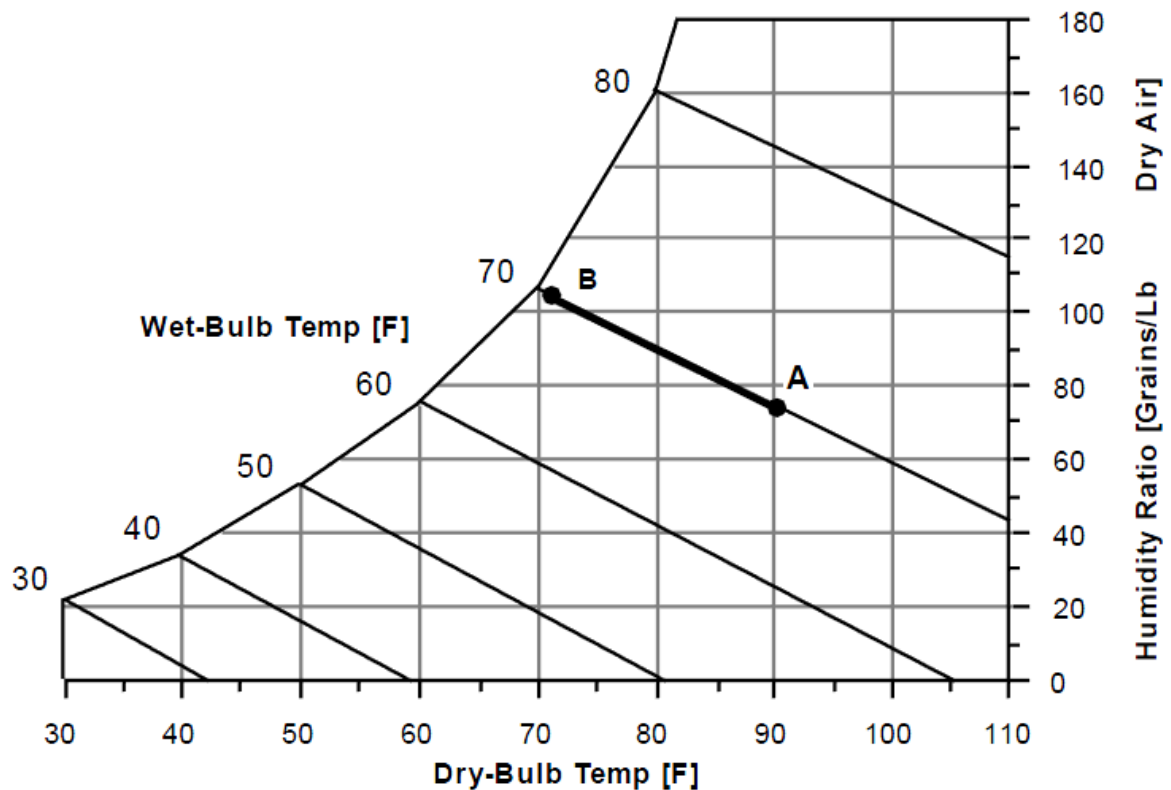
- Air Handling Units

The direct stage, shown in the figure below, consists of a rigid media evaporative pad, with water recirculated from a reservoir. The water is pumped from the reservoir to a water distribution header, for water feed by gravity from above the media. The evaporative pad provides the area for the adiabatic saturation of the air. While the process provides a lower dry-bulb temperature, the moisture content of the leaving air is higher than the entering condition. The direct stage is used for comfort cooling in a building where adding humidity to the air can be tolerated.



Direct Stage Evaporative Cooler

The thermodynamic process is a simultaneous heat and mass transfer, or adiabatic cooling, and follows a constant enthalpy line on the psychrometric chart, it is shown in the figure below as a process from A to B. Since the deviation of the constant wet-bulb line and the constant enthalpy line is small, it is assumed that the wet-bulb temperature is constant across the direct evaporative stage.



Psychrometric Chart - Constant Enthalpy

If the direct evaporative process were 100% efficient, the leaving dry-bulb temperature would equal the entering wet-bulb temperature. The efficiency of the direct evaporative process is less than 100% and by defining saturation efficiency (ϵ_{se}) for the direct stage or evaporative pad, the leaving dry-bulb temperature can be expressed by the following equation.

$$T_{db\ supout} = T_{db\ supin} - \epsilon_{se} \cdot (T_{db\ supin} - T_{wb\ supin})$$

General

Name

A unique name for an instance of an evaporative cooler which is predetermined by DesignBuilder.

Type

Select the type of evaporative cooler from the list of options:

- [1-Direct CelDekPad](#), the cooler described here, a direct evaporative cooler with rigid media evaporative pad and water recirculated from a reservoir.
- [2-Direct Research Special](#), similar to [1-Direct CelDekPad](#) but with methods to specify variable effectiveness based on primary air flow rate and a way to control the cooler operating range based on temperature limits.
- [3-Indirect CelDekPad](#), an indirect evaporative cooler with a rigid media pad, similar to the direct evaporative cooler.
- [4-Indirect Wet Coil](#), an indirect evaporative cooler where water is sprayed directly on the tubes of the heat exchanger.
- [5-Indirect Research Special](#), similar to the other indirect evaporative coolers, but it provides additional flexibility including a way to specify the source of secondary air.

Settings

Direct pad area

The face area of the evaporative pad (in m² or ft²). With the area and mass flow rate, the air velocity is calculated and is used to determine the saturation efficiency.

Direct pad depth

The depth of the evaporative pad (in m or in). The pad depth is used to determine the saturation efficiency.

Recirculating water pump power consumption

This field is used to specify the power consumed by the evaporative cooler recirculating pump (in W).

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the evaporative cooler can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

Evaporative Cooler - Direct Research Special



EvaporativeCooler:Direct:ResearchSpecial

Used in:

- Air Handling Units

This cooler is similar in principal to the [Direct CelDekPad](#). The model differs in that it gives the user a simple way of specify the cooler effectiveness. Using the Research Special input object also allows the cooler to control the amount of cooling based on node setpoints, controlled by [Setpoint managers](#). This avoids problems from over cooling when conditions are such that loads are low and cooling power is high. Water pump power is assumed to vary linearly when the cooler is operating at less than full capacity.

The model allows the effectiveness to be vary depending on the primary air flow rates. The design effectiveness can be modified by multiplying by an Effectiveness flow fraction modifier curve. The flow fraction is the ratio of the current primary airflow rate to the design flow rate. The recirculating and spray water pump power is assumed to vary with the primary air flow. The design pump power is modified using user specified pump modifier curve value. The normalized pump power modifier curve is a function of primary air flow fraction as a independent variable.

Also the direct evaporative cooler operating range can be controlled depending on the entering air dry bulb and wet bulb temperatures. The operating range can be controlled based on minimum and maximum inlet node air temperature limits. The evaporative cooler can be turned on or off depending user specified minimum and maximum temperature limits. If the inlet node entering air temperature is lower or higher than the minimum and maximum limits, respectively, then the evaporative cooler is turned off. The operating range control feature is primarily intended for application in data centres.

General

Name

A unique name for an instance of an evaporative cooler which is predetermined by DesignBuilder.

Type

Select the type of evaporative cooler from the list of options:

- [1-Direct CelDekPad](#), a direct evaporative cooler with rigid media evaporative pad and water recirculated from a reservoir.
- [2-Direct Research Special](#), the cooler described here which is similar to [1-Direct CelDekPad](#) but with methods to specify variable effectiveness based on primary air flow rate and a way to control the cooler operating range based on temperature limits.
- [3-Indirect CelDekPad](#), an indirect evaporative cooler with a rigid media pad, similar to the direct evaporative cooler.
- [4-Indirect Wet Coil](#), an indirect evaporative cooler where water is sprayed directly on the tubes of the heat exchanger.
- [5-Indirect Research Special](#), similar to the other indirect evaporative coolers, but it provides additional flexibility including a way to specify the source of secondary air.

Settings

Primary design air flow rate

This autosizable setting is the primary air design air flow rate (in m³/s or ft³/min). If the evaporative cooler is on main air loop branch, the design flow rate is the same as branch design flow rate, or else if it is on outdoor air system it will be the maximum of the outdoor air design flow rate and the half of the primary air flow rate on the main air loop branch.

Cooler design effectiveness

This setting specifies the effectiveness of the cooler at design flow rate that is applied to the wet-bulb depression to determine the conditions leaving the cooler. This model assumes that the effectiveness can vary with supply air flow rate. For effectiveness variation with supply air flow fraction enter the [Effectiveness flow ratio modifier curve](#) below. The flow fraction is the ratio of the sum of current primary air and secondary air sides flow rates and the sum of the design flow rates.

Effectiveness flow ratio modifier

If an Effectiveness flow ratio modifier curve (below) is to be specified then check this checkbox. If this option is not checked then the effectiveness is assumed to be constant.

Effectiveness flow ratio modifier curve

This curve modifies the [Cooler effectiveness](#) design value (above) by multiplying the value by the result of this curve. The modifying curve is a function of flow fraction, which is the ratio of the current primary air flow rates divided by the design primary air flow rates. Any curve with one independent variable can be used. Select a curve from one of these categories:

- [Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

Drift loss fraction

This field is optional and can be used to model additional water consumed by the cooler from drift. Drift is water that leaves the cooling media as droplets and does not evaporate into the process air stream. For example, water may get blown off the evaporative media by winds and escape the air system. The value entered here is a simple fraction of the water consumed by the cooler for normal process evaporation. The amount of drift is this fraction times the water evaporated for the normal cooling process. This field can be left blank and then there will be no added water consumption from drift.

Recirculating Water

Recirculating water pump design power

This is the nominal water recirculating and spray pump power of evaporative cooler at primary air design flow rates and cooler design effectiveness. This setting is autosizable.

Water pump power sizing factor

This numeric input field value is recirculating water pump sizing factor (in W/(m³/s) or W/gpm). This field is used when the previous field is set to autosize. The pump design electric power is scaled with Secondary Design Air Flow Rate. The default value is 90.0 W/(m³/s), i.e. Pump power / Primary air design flow rate = 90 W/(m³/s). Typical values range from 55.0 to 150.0 W/(m³/s).

Water pump power modifier

Select this option to apply a [Water pump power modifier curve](#). If this option is not checked then the pump power is assumed to vary linearly with the load.

Water pump power modifier curve

Select a dimensionless normalized pump power modifying curve. This curve modifies the pump electric power by multiplying the design power by the result of this curve. The normalized curve is a function of the primary air flow fraction as independent variable. The curve generates a value of 1.0 at a flow fraction of 1.0. The flow fraction is the ratio of the primary air during current operation divided by [Primary design air flow rate](#). Any curve with one independent variable can be used. Select a curve from one of these categories:

- [Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

Temperature Limits

Evaporative operation minimum dry-bulb temperature

To define a dry-bulb temperature lower limit for evaporative cooler operation check this checkbox. Otherwise, if this option is unchecked then there is no dry-bulb temperature lower limit for evaporative cooler operation.

Minimum limit dry-bulb temperature

This setting defines the evaporative cooler inlet node dry-bulb temperature minimum limit (in °C or °F). The evaporative cooler will be turned off when evaporator cooler air inlet node dry-bulb temperature falls below this value. The typical minimum value is 16°C.

Evaporative operation maximum limit wet-bulb temperature

To define a wet-bulb temperature upper limit for evaporative cooler operation check this checkbox. Otherwise, if this option is unchecked then there is no wet-bulb temperature upper limit for evaporative cooler operation.

Minimum limit dry-bulb temperature

This setting defines the evaporative cooler air inlet node air wet-bulb temperature maximum limit (in °C or °F). When the evaporative cooler air inlet node air wet-bulb temperature exceeds this limit, then the evaporative cooler is turns off. The typical maximum value is 24°C.

Evaporative operation maximum limit dry-bulb temperature

To define a dry-bulb temperature upper limit for evaporative cooler operation check this checkbox. Otherwise, if this option is unchecked then there is no dry-bulb temperature upper limit for evaporative cooler operation.

Maximum limit dry-bulb temperature

This setting defines the evaporative cooler air inlet node dry-bulb temperature maximum limit (in °C or °F). The evaporative cooler will be turned off when the evaporative cooler air inlet node dry-bulb temperature exceeds this value. The typical maximum value is 28°C.

Blowdown

Blowdown On

Select this option to model additional water consumed by the cooler from blowdown. Blowdown is water that is intentionally drained from the cooler's sump to offset the build up of solids in the water that would otherwise occur because of evaporation.

Blowdown concentration ratio

The value entered here is dimensionless. It can be characterized as the ratio of solids in the blowdown water to solids in the make up water. Typical values are 3 to 5. The default is 3.0.

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the evaporative cooler can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

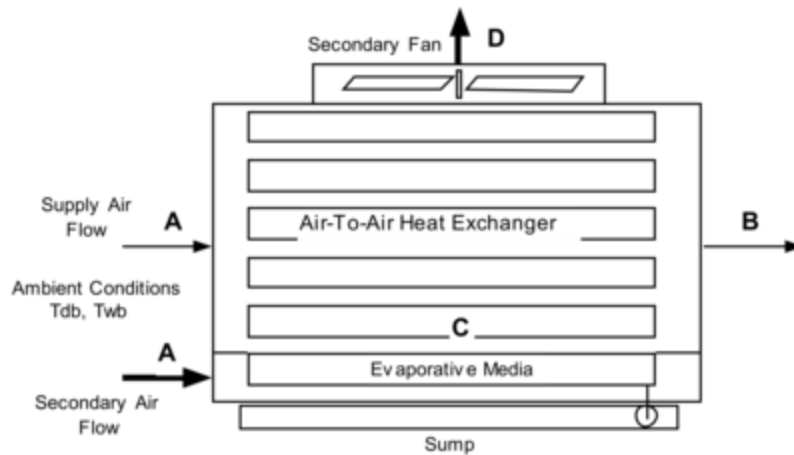
Evaporative Cooler - Indirect CelDekPad



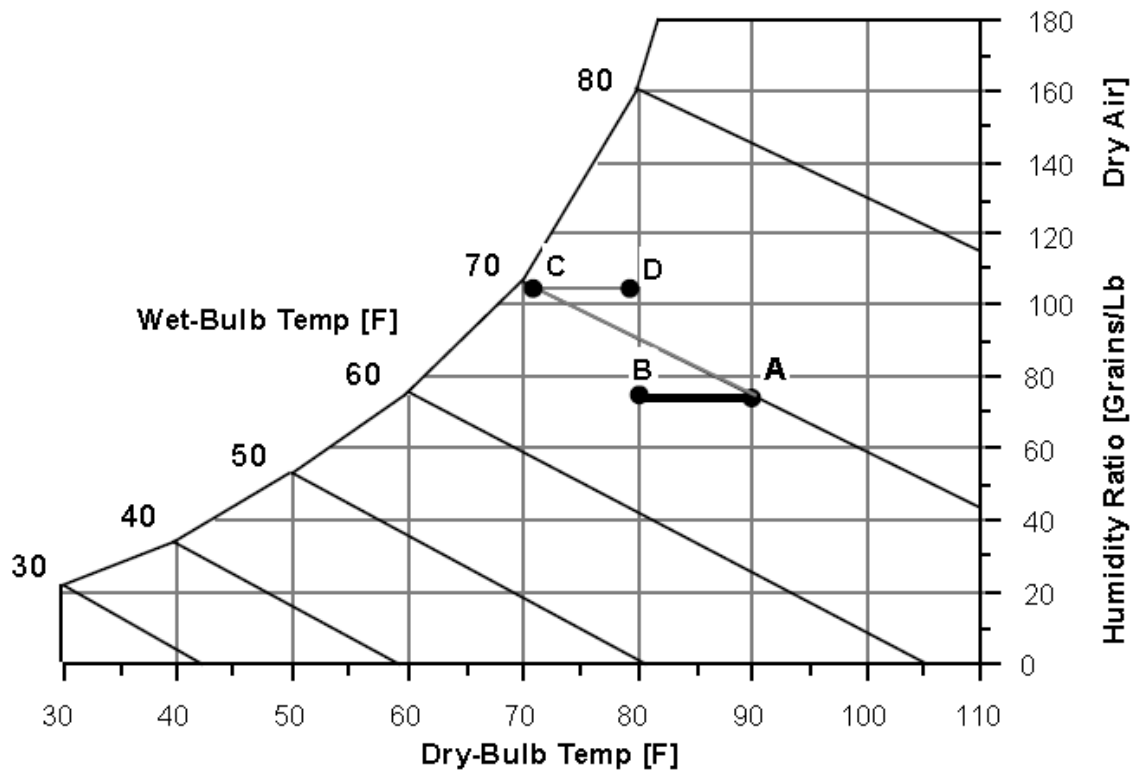
Used in:

- Air Handling Units

The dry coil indirect evaporative cooler has a rigid media pad, similar to the direct evaporative stage, where the adiabatic cooling takes place. The secondary air enters the evaporative cooler at ambient conditions, is passed into the rigid media pad and enters an air to air heat exchanger where it cools the supply air flowing through the heat exchanger tubes (see diagram below). The moist secondary air is then exhausted to the environment. The secondary air stream has its own fan and consists of a rigid media evaporative pad with water recirculated from a reservoir. The water is pumped from the reservoir to a water distribution header for water feed by gravity from above the media. The evaporative pad provides the area for the adiabatic saturation of the air.



The process that the secondary air goes through, A to C to D, is shown by the dashed lines in the following figure. Process A to C is adiabatic cooling in the rigid media pad. Then the air enters the shell side of the heat exchanger and is sensibly heated from C to D by the warm supply air passing through the tube side. The secondary air inlet is modeled as a separate stream of outdoor air and the user has the option of defining the name of an outdoor air node.



The advantage of the dry coil heat exchanger is that the heat exchanger does not have the evaporation taking place on the outside of the tubes, thus no mineral deposits are left on the heat exchange surface to reduce the efficiency of the heat exchanger. The rigid media pads are designed to flush the mineral deposits to the sump, so the saturation efficiency of the pad stays relatively constant.

General

Name

A unique name for an instance of an evaporative cooler which is predetermined by DesignBuilder.

Type

Select the type of evaporative cooler from the list of options:

- [1-Direct CelDekPad](#), a direct evaporative cooler with rigid media evaporative pad and water recirculated from a reservoir.
- [2-Direct Research Special](#), similar to [1-Direct CelDekPad](#) but with methods to specify variable effectiveness based on primary air flow rate and a way to control the cooler operating range based on temperature limits.
- [3-Indirect CelDekPad](#), the indirect evaporative cooler described here with a rigid media pad, similar to the direct evaporative cooler.
- [4-Indirect Wet Coil](#), an indirect evaporative cooler where water is sprayed directly on the tubes of the heat exchanger.
- [5-Indirect Research Special](#), similar to the other indirect evaporative coolers, but it provides additional flexibility including a way to specify the source of secondary air.

Settings

Direct pad area

The face area of the evaporative pad (in m² or ft²). With the area and mass flow rate, the air velocity is calculated and is used to determine the saturation efficiency on the secondary side of the evaporative cooler.

Direct pad depth

The depth of the evaporative pad in meters. The pad depth is used to determine the saturation efficiency on the secondary side of the evaporative cooler.

Indirect heat exchanger effectiveness

This field is used to specify the effectiveness of the indirect heat exchanger between the primary and secondary air flow.

Recirculating water pump power consumption

This field is used to specify the power consumed by the evaporative cooler recirculating pump (in W).

Secondary Fan

The secondary flow is sourced from outside air and blown over the

Secondary fan flow rate

This field is used to specify the secondary fan flow rate and is specified in m³/s or ft³/min.

Secondary fan efficiency

This field is used to specify the total efficiency of the fan and is used to calculate the power consumed by the evaporative cooler secondary fan. Input values should be between 0 and 1.

Secondary fan delta pressure

This field is used to specify the delta pressure across the secondary stage of the evaporative cooler (in Pa or in H₂O).

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the evaporative cooler can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

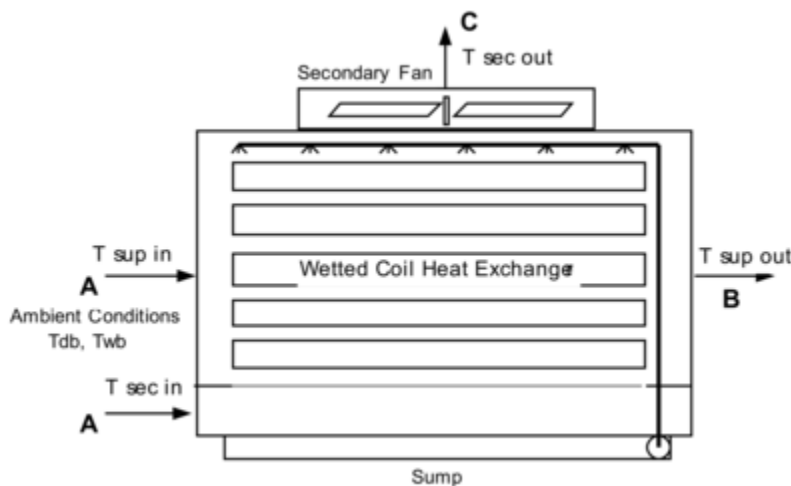
Evaporative Cooler - Indirect Wet Coil



Used in:

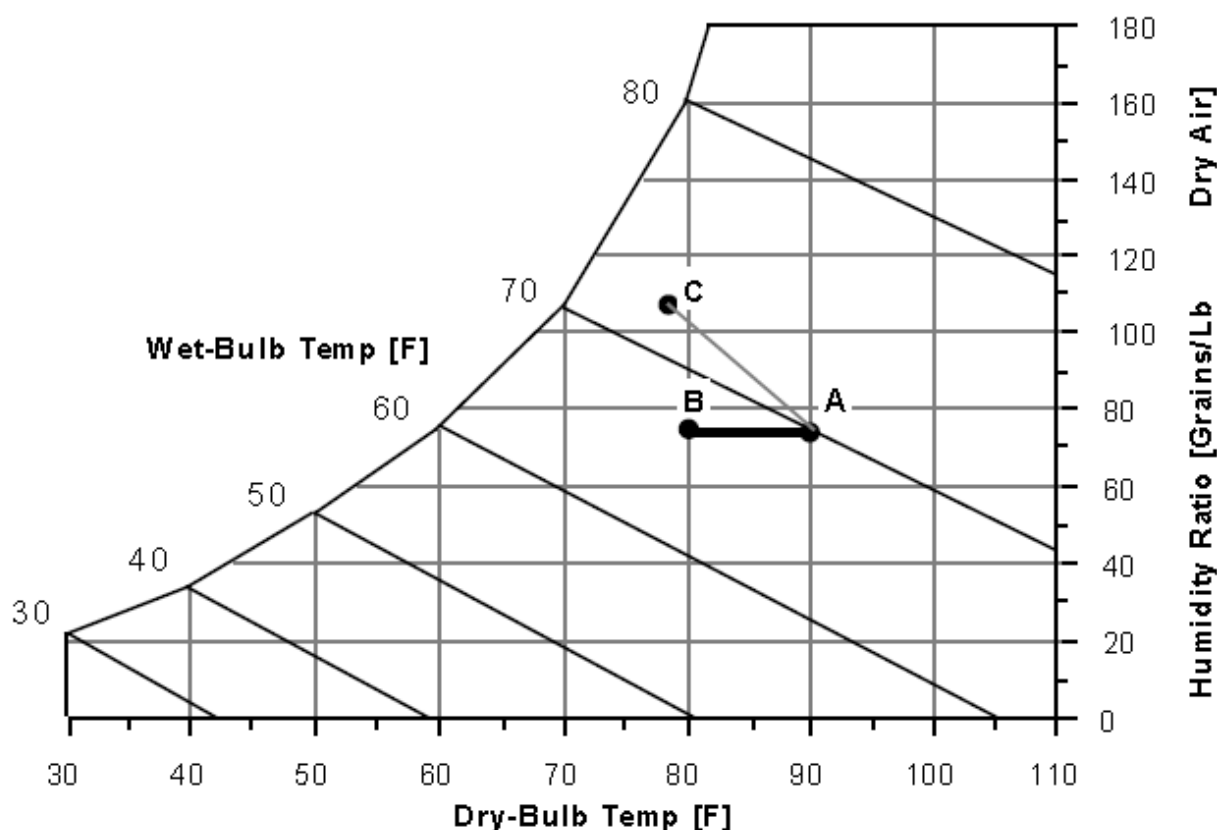
- Air Handling Units

The wetted coil evaporative cooler shown in the figure below, has water sprayed directly on the tubes of the heat exchanger where latent cooling takes place. The vaporization of the water on the outside of the heat exchanger tubes allows the simultaneous heat and mass transfer which removes heat from the supply air on the tube side. Secondary air at ambient conditions is passed over the wet coil and is exhausted to outside. The secondary air stream has its own fan.



Evaporative Cooler – Indirect Wet Coil

The process that the secondary air goes through, A to C on the following figure, is a path of simultaneous heat and mass transfer, but it does not follow a line of constant enthalpy as in the direct stage. The process is not adiabatic due to the heat gain from the supply air flowing through the tubes of the heat exchanger.



Secondary Air Process – Indirect Wet Coil Evaporative Cooler

General

Name

A unique name for an instance of an evaporative cooler which is predetermined by DesignBuilder.

Type

Select the type of evaporative cooler from the list of options:

- [1-Direct CelDekPad](#), a direct evaporative cooler with rigid media evaporative pad and water recirculated from a reservoir.
- [2-Direct Research Special](#), similar to [1-Direct CelDekPad](#) but with methods to specify variable effectiveness based on primary air flow rate and a way to control the cooler operating range based on temperature limits.
- [3-Indirect CelDekPad](#), an indirect evaporative cooler with a rigid media pad, similar to the direct evaporative cooler.
- [4-Indirect Wet Coil](#), the indirect evaporative cooler described here where water is sprayed directly on the tubes of the heat exchanger.
- [5-Indirect Research Special](#), similar to the other indirect evaporative coolers, but it provides additional flexibility including a way to specify the source of secondary air.

Settings

Coil maximum efficiency

The maximum efficiency of the stage is a combination of the efficiency due to the simultaneous heat and mass transfer on the outside of the tube and the efficiency of the heat exchanger. This value can be higher than the dry coil overall efficiency since the convective coefficients on the outside of the tube are larger.

Coil flow ratio

The Coil flow ratio is determined from performance data. The Coil Flow Ratio tells how quickly the efficiency of the stage would decrease with a mismatch of the supply and secondary flows.

Recirculating water pump power consumption

This field is used to specify the power consumed by the water pump that circulates water (in W).

Secondary Fan

Secondary fan flow rate

This field is used to specify the secondary fan flow rate and is specified in m3/s or ft3/min.

Secondary fan total efficiency

This field is used to specify the total efficiency of the fan and is used to calculate the power consumed by the evaporative cooler secondary fan. Input values should be between 0 and 1.

Secondary fan delta pressure

This field is used to specify the delta pressure across the secondary stage of the evaporative cooler (in Pa or in H2O).

Blowdown

Blowdown On

Select this option to model additional water consumed by the cooler from blowdown. Blowdown is water that is intentionally drained from the cooler's sump to offset the build up of solids in the water that would otherwise occur because of evaporation.

Blowdown concentration ratio

The value entered here is dimensionless. It can be characterized as the ratio of solids in the blowdown water to solids in the make up water. Typical values are 3 to 5. The default is 3.0.

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the evaporative cooler can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

Evaporative Cooler - Indirect Research Special



EvaporativeCooler:Indirect:ResearchSpecial

Used in:

- Air Handling Units

The EnergyPlus Indirect Research Special is similar in principal to the [Indirect CelDekPad](#) and [Indirect WetCoil](#) evaporative coolers, but it provides some additional flexibility. The model differs in that it gives the user more flexibility to specify the source of secondary air. The cooler effectiveness with respect to wet-bulb depression is allowed to go beyond 1.0. Using the Research Special input object also allows the cooler to control the amount of cooling based on node setpoints (controlled by Setpoint Managers). This avoids

problems from over cooling when conditions are such that loads are low and cooling power is high. Fan power is assumed to vary linearly when the cooler is operating at less than full capacity.

The indirect evaporative cooler research special calculation procedure allows accounting for dry and wet effectiveness value variation with flow fraction. Two effectiveness modifier curves are included as optional user inputs for this purpose. Effectiveness modifier curves operate on the design dry and wet effectiveness values. The flow fraction is calculated as a ratio of the sum of current primary and secondary air flow rates to the sum of the design flow rates. Model also accounts for fan and recirculation water pump power variation with secondary air flow rates using pump power modifying curve. The fan power is calculated by multiplying the design fan power using fan power modify curve value evaluated at current secondary air flow fraction. Similarly, recirculating pump power is calculated by multiplying the design pump power by pump power modifier curve value evaluated at current secondary air flow fraction. If the secondary air fan and recirculating pump power modifier curves are not specified, then fan and pump power are assumed to vary linearly with part load fraction.

The cooler effectiveness with respect to wet-bulb depression is allowed to go beyond 1. Fan power is assumed to vary linearly when the cooler is operating at less than full capacity.

General

Name

A unique name for an instance of an evaporative cooler which is predetermined by DesignBuilder.

Type

Select the type of evaporative cooler from the list of options:

- [1-Direct CelDekPad](#), a direct evaporative cooler with rigid media evaporative pad and water recirculated from a reservoir.
- [2-Direct Research Special](#), similar to [1-Direct CelDekPad](#) but with methods to specify variable effectiveness based on primary air flow rate and a way to control the cooler operating range based on temperature limits.
- [3-Indirect CelDekPad](#), an indirect evaporative cooler with a rigid media pad, similar to the direct evaporative cooler.
- [4-Indirect Wet Coil](#), an indirect evaporative cooler where water is sprayed directly on the tubes of the heat exchanger.
- [5-Indirect Research Special](#), the model described here, similar to the other indirect evaporative coolers, but it provides additional flexibility including a way to specify the source of secondary air.

Settings

Primary design air flow rate

This autosizable setting is the primary air design air flow rate (in m³/s or ft³/min). If the evaporative cooler is on main air loop branch, the design flow rate is the same as branch design flow rate, or else if it is on outdoor air system it will be the maximum of the outdoor air design flow rate and the half of the primary air flow rate on the main air loop branch.

Cooler wet-bulb design effectiveness

This setting specifies the design effectiveness that is applied to the wet-bulb depression to determine the conditions leaving the cooler. This effectiveness is a complicated function of the efficiency with which heat and mass are transferred on the secondary side and the efficiency of heat exchange between the secondary and primary flows. The model assumes that the effectiveness a function of flow fraction. The flow fraction is the ratio of the sum of primary air and secondary air current flow rates and the sum of the primary air and secondary air design flow rates.

Wet-bulb effectiveness flow ratio modifier

Check this option to allow the Cooler wet-bulb design effectiveness specified above to be modified by a curve based on flow fraction. If this option is not checked then the wet-bulb effectiveness is assumed to be the constant value entered above.

Wet-bulb effectiveness flow ratio modifier curve

This curve modifies the wet-bulb effectiveness design value specified above by multiplying it by the result of this curve. The modifying curve is a function of flow fraction, which is the ratio of the sum of the primary and secondary flow rates divided by the sum of the design flow rates. Any curve with one independent variable can be used. Select a curve from one of these categories:

- [Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

Cooler dry-bulb design effectiveness

Enter the dry-bulb design effectiveness of the evaporative cooler. This is the nominal design dry bulb effectiveness with respect to dry bulb temperature difference, i.e., dry operation and at design air flow rates, and no water evaporation or spraying on the secondary side.

Dry-bulb effectiveness flow ratio modifier

Check this option to allow the Cooler dry-bulb design effectiveness specified above to be modified by a curve based on flow fraction. If this option is not checked then the dry-bulb effectiveness is assumed to be the constant value entered above.

Dry-bulb effectiveness flow ratio modifier curve

The dry-bulb effectiveness entered in the previous field can be modified by multiplying by the result of this curve. The curve is evaluated using flow fraction as the independent variable. The flow fraction is the ratio of sum of the primary and secondary flow rates divided by the sum of the design flow rates. Any curve with one independent variable can be used. Select a curve from one of these categories:

- [Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

Dew-point effectiveness factor

This field specifies an effectiveness that is applied to the dew-point depression to determine a bound for the conditions leaving the cooler. The model uses the warmer of the two temperatures determined from wet-bulb depression and dew-point depression.

Drift loss fraction

This field is optional and can be used to model additional water consumed by the cooler from drift. Drift is water that leaves the cooling media as droplets and does not evaporate into the process air stream. For example, water may get blown off the evaporative media by winds and escape the air system. The value entered here is a simple fraction of the water consumed by the cooler for normal process evaporation. The amount of drift is this fraction times the water evaporated for the normal cooling process. This field can be left blank and then there will be no added water consumption from drift.

Recirculating Water

Recirculating water pump design power

This autosizable setting is the recirculating pump electric power at Secondary design air flow rate (in W). It is the nominal design pump power water recirculation and spray for evaporation at design secondary air flow rates and cooler design effectiveness.

Water pump power sizing factor

This numeric input field value is recirculating water pump sizing factor (in W/(m³/s) or W/gpm). This field is used when the previous field is set to autosize. The pump design electric power is scaled with Secondary Design Air Flow Rate. The default value is 90.0 W/(m³/s), i.e. Pump power / Primary air design flow rate = 90 W/(m³/s). Typical values range from 55.0 to 150.0 W/(m³/s).

Water pump power modifier

Select this option to apply a [Water pump power modifier curve](#). If this option is not checked then the pump power is assumed to be constant.

Water pump power modifier curve

Select a dimensionless normalized pump power modifying curve. The pump electric power is modified by multiplying the design power by the result of this curve. The normalized curve is a function of the secondary side flow fraction as independent variable. The curve generates a value of 1.0 at a flow fraction of 1.0. The flow fraction is the ratio of the primary air during current operation divided by [Primary design air flow rate](#). Any curve with one independent variable can be used. Select a curve from one of these categories:

- [Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

Secondary Air

Secondary air design flow rate

This field is used to specify the secondary air fan flow rate and is specified (in m³/s or ft³/min). This flow rate would typically be similar in magnitude to the flow through the primary side. The flow rate is used to determine parasitic fan energy and cooler effectiveness. The flow rate (and fan power) is effectively reduced by cycling when the amount of cooling needs to be restricted for control purpose. This field can be autosized. When this input is autosized, the program calculates it by scaling the [Primary design air flow rate](#) using secondary air scaling factor specified in the input field below.

Secondary air flow scaling factor

This setting is used to scale the secondary air design flow rate and is dimensionless. It is used when the previous field is set to autosize. The [Primary design air flow rate](#) is scaled using this factor to calculate the secondary design air flow rate.

Secondary air fan design power

This autosizable setting is the fan electric power at [Secondary design air flow rate](#). It is the nominal design electric power at full speed of the secondary air fan.

Secondary air fan sizing specific power

This input field value is secondary air fan sizing specific power (in W/(m³/s) or W/gpm). It is used when the previous field is set to autosize. The fan power is scaled with [Secondary design air flow rate](#).

Secondary air fan power modifier

The [Secondary air fan design power](#) above can be modified by a curve using the secondary side flow fraction as the independent variable. Check this option to modify the secondary fan power in this way. If this option is unchecked then the fan power is assumed to be constant.

Secondary air fan power modifier curve

The [Secondary air fan design power](#) above can be modified by multiplying by the result of this dimensionless normalized curve. The curve is a function of the secondary side flow fraction as the independent variable. The curve generates a value of 1.0 at a flow fraction of 1.0. The flow fraction is the secondary air flow rate during operation divided by [Secondary design air flow rate](#). Any curve with one independent variable can be used. Select a curve from one of these categories:

- [Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

Temperature Limits

Evaporative operation minimum dry-bulb temperature

To define a dry-bulb temperature lower limit for evaporative cooler operation check this checkbox. Otherwise, if this option is unchecked then there is no dry-bulb temperature lower limit for evaporative cooler operation.

Minimum limit dry-bulb temperature

This setting defines the evaporative cooler inlet node dry-bulb temperature minimum limit (in °C or °F). The evaporative cooler will be turned off when evaporator cooler air inlet node dry-bulb temperature falls below this value. The typical minimum value is 16°C.

Evaporative operation maximum limit wet-bulb temperature

To define a wet-bulb temperature upper limit for evaporative cooler operation check this checkbox. Otherwise, if this option is unchecked then there is no wet-bulb temperature upper limit for evaporative cooler operation.

Minimum limit dry-bulb temperature

This setting defines the evaporative cooler air inlet node air wet-bulb temperature maximum limit (in °C or °F). When the evaporative cooler air inlet node air wet-bulb temperature exceeds this limit, then the evaporative cooler is turns off. The typical maximum value is 24°C.

Evaporative operation maximum limit dry-bulb temperature

To define a dry-bulb temperature upper limit for evaporative cooler operation check this checkbox. Otherwise, if this option is unchecked then there is no dry-bulb temperature upper limit for evaporative cooler operation.

Maximum limit dry-bulb temperature

This setting defines the evaporative cooler air inlet node dry-bulb temperature maximum limit (in °C or °F). The evaporative cooler will be turned off when the evaporative cooler air inlet node dry-bulb temperature exceeds this value. The typical maximum value is 28°C.

Blowdown

Blowdown On

Select this option to model additional water consumed by the cooler from blowdown. Blowdown is water that is intentionally drained from the cooler's sump to offset the build up of solids in the water that would otherwise occur because of evaporation.

Blowdown concentration ratio

The value entered here is dimensionless. It can be characterized as the ratio of solids in the blowdown water to solids in the make up water. Typical values are 3 to 5. The default is 3.0.

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the evaporative cooler can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

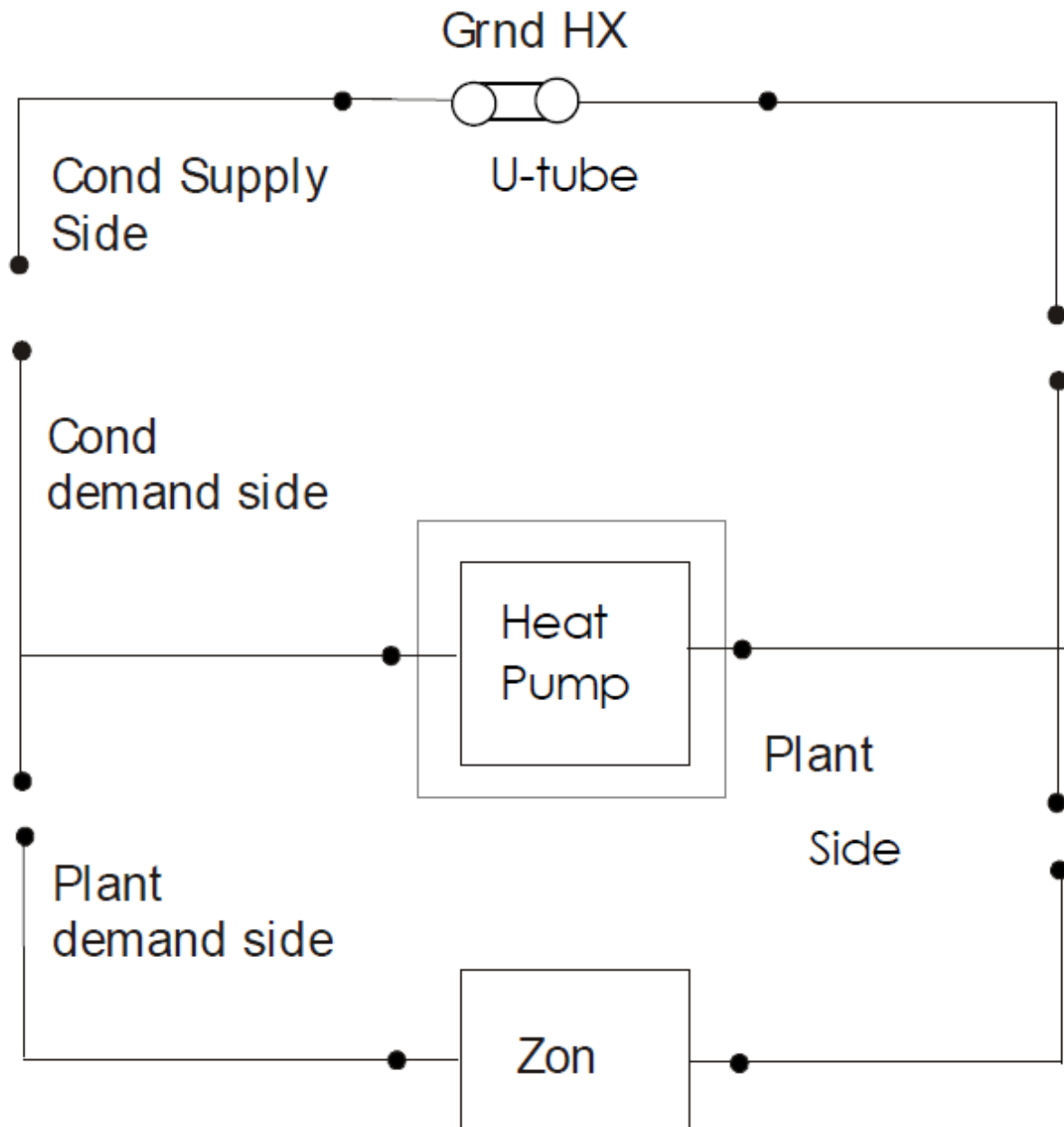
Plant Loop Heat Pumps

A single physical water-to-water heat pump could potentially provide either heating only, cooling only or both heating and cooling. These 3 heat pump configuration types can be represented in DesignBuilder using **Heat pump - heating** and/or **Heat pump - cooling** components connected to hot and chilled water plant loops respectively. Where both heating and cooling are provided by the same physical heat pump both heating and cooling heat pumps must be included in the HVAC layout and the data entry for the heat pumps (especially flow rates), must be coordinated.

The Supply side of the heat pump may be connected to one of the following components:

- [Ground Heat Exchanger](#)
- [Cooling tower](#)
- [Fluid cooler](#)
- [Heat exchanger](#)

The schematic diagram below shows the layout and piping diagram of the water-to-water heat pump.



GSHP Loops

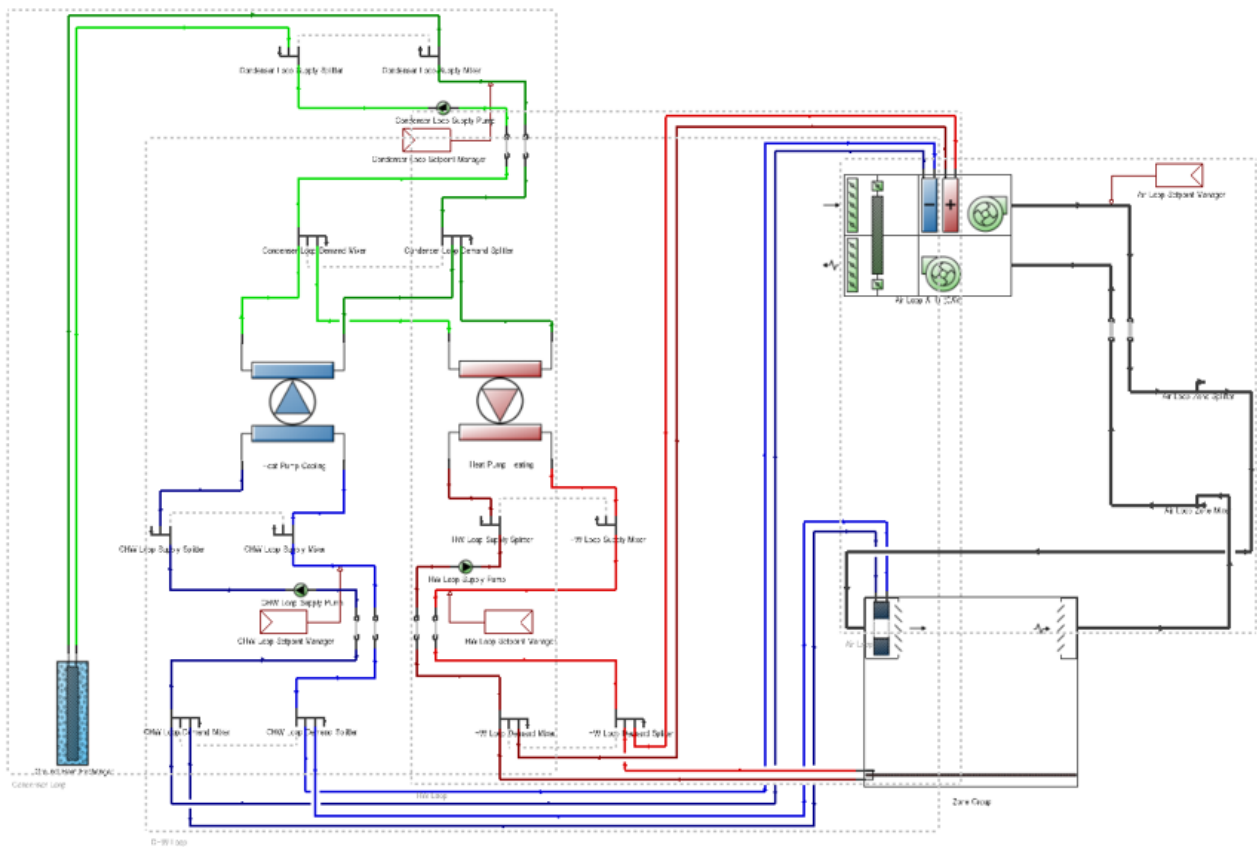
Ground source heat pumps systems are essentially made up of two loops (heating or cooling only cases) or three loops (combined heating and cooling):

- [HW loop](#) (for heating applications) - Hot water loop (Water-to-water heat pump heating)
- [CHW loop](#) (for cooling applications) - Chilled water loop (Water-to-water heat pump cooling)
- [Condenser loop](#) - typically a [Ground heat exchanger loop](#) (exchanges heat with ground via either surface, pond or vertical borehole types)

In addition to the above mandatory loops, the system may also require extra loops such as air loops for specific room/block applications.

Tip: You are advised to work through the [GSHP Example](#) before attempting to model these systems for the first time.

The screenshot below shows a typical GSHP layout diagram.



You can learn more about practical aspects of GSHP design and simulation for project work by watching our webinar: [Design and Modelling of Ground Source Heat Pump Systems](#)

Component descriptions:

- [Water to Water Heat Pump - Heating](#)
- [Water to Water Heat Pump - Cooling](#)

Configuring GSHPs In DesignBuilder

Because GSHPs cannot be autosized in EnergyPlus some care is required in setting up these systems. This section describes a process which ensures correct sizing and provides for correct interaction between the various components involved. The process is as follows:

1. Run DesignBuilder [Heating design](#) and [Cooling design](#) calculations in order to get the maximum heating and cooling loads for the system.
2. Choose/load a correctly sized water-to-water heat pump model from DesignBuilder database by matching the peak heating/cooling loads with heat pump's 'Rated heating/cooling capacity'. Or alternatively, creating and applying your own heat pump data if its catalogue data is available (using procedure described in the).
3. Select ground heat exchanger (vertical) borehole numbers from that quick/simple sizing table; set flow rate equal to the sum of 'Rated source side flow rate' of heat pumps connecting to it, or use your own tool to generate (and apply) a vertical ground heat exchanger case for specific conditions or detailed/precise sizing.
4. Condenser loop: set flow rate equal to or greater than the flow rate of ground heat exchanger (vertical) calculated above.

For more detail on these processes see the [GSHP Example](#).

What about the EnergyPlus ParameterEstimation model?

The EnergyPlus **ParameterEstimation** model limits the refrigerant used to R22 and the compressor to the reciprocating compressor type. However almost all water-to-water heat pumps presently on the market use refrigerants other than R22 (R22 has been phased out recently for environmental reasons) and equipped with scroll and other compressor types. So catalogue data is no longer available for the **ParameterEstimation** model.

Also EnergyPlus strongly recommend using the **EquationFit** model instead of **ParameterEstimation** for efficient and reliable modelling,

For these reasons DesignBuilder currently only provides the **EquationFit** water-to-water heat pump model.
Equation Fit Heat Pumps Technical

The Equation Fit model uses four non-dimensional equations or curves to predict the heat pump performance. The same method is used in both cooling and heating modes.

The performance data used in the Equation Fit method is derived from manufacturers data through a pre-process step in which the generalized least square method is used to generate a set of performance coefficients from manufacturer's catalogue data at indicated reference conditions. This is done in an EnergyPlus spreadsheet. Then the calculated coefficients and indicated reference conditions are entered on the heat pump dialog in the model to simulate the heat pump performance. A library of templates containing pre-defined manufacturers heat pump data is provided to allow an early stage analysis to be carried out without specific project equipment being specified.

The variables that influence the water-to-water heat pump performance include load side entering water temperature, source side entering water temperature, source side water flow rate and load side water flow rate. The governing equations for the cooling and heating modes are as follows:

Cooling Mode

$$\frac{Q_c}{Q_{c,ref}} = A1 + A2 \left[\frac{T_{L,in}}{T_{ref}} \right] + A3 \left[\frac{T_{S,in}}{T_{ref}} \right] + A4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + A5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

$$\frac{Power_c}{Power_{c,ref}} = B1 + B2 \left[\frac{T_{L,in}}{T_{ref}} \right] + B3 \left[\frac{T_{S,in}}{T_{ref}} \right] + B4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + B5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

Heating Mode:

$$\frac{Q_h}{Q_{h,ref}} = C1 + C2 \left[\frac{T_{L,in}}{T_{ref}} \right] + C3 \left[\frac{T_{S,in}}{T_{ref}} \right] + C4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + C5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

$$\frac{Power_h}{Power_{h,ref}} = D1 + D2 \left[\frac{T_{L,in}}{T_{ref}} \right] + D3 \left[\frac{T_{S,in}}{T_{ref}} \right] + D4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + D5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

Assuming no losses, the source side heat transfer rate for cooling and heating mode is calculated as:

$$Q_{source,c} = Q_c + Power_c$$

$$Q_{source,h} = Q_h - Power_h$$

Where:

A1-D5 EquationFit curve coefficients for the cooling and heating mode

T_{ref} Reference temperature (using fixed 283.15K)

$T_{L,in}$ Load side entering water temperature (K)

$T_{S,in}$ Source side entering water temperature (K)

$\dot{V}_L$ Load side volumetric flow rate (m3/s)

V_s Source side volumetric flow rate (m³/s)
 $V_{L,ref}$ Reference load side volumetric flow rate (m³/s)
 $V_{S,ref}$ Reference source side volumetric flow rate (m³/s)
 Q_c Load side heat transfer rate (cooling mode) (W)
 $Power_c$ Power consumption (cooling mode) (W)
 $Q_{source,c}$ Source side heat transfer rate (cooling mode) (W)
 Q_h Load side heat transfer rate (heating mode) (W)
 $Q_{h,ref}$ Reference load side heat transfer rate (heating mode) (W)
 $Power_h$ Power consumption (heating mode) (W)
 $Power_{h,ref}$ Reference power consumption (heating mode) (W)
 $Q_{source,h}$ Source side heat transfer rate (heating mode) (W)

Heat Pump - Cooling

General

Name

This field contains the unique identifying name for the cooling side of the plant loop heat pump.

Type

DesignBuilder provides the following plant loop heat pump types:

- [1-Equation fit water to water heat pump](#) - corresponds to the EnergyPlus Equation fit model.
- [2-Plant loop EIR heat pump](#) - for modelling water to water or air to water heat pumps based on the EIR formulation.

Tip: For both these heat pump types, it is important to use manufacturer's performance data to ensure the accuracy of your simulations.

Equation Fit Water to Water Heat Pump

The Equation fit model uses four non-dimensional equations (curves) to predict the heat pump performance in cooling and heating modes. The methodology involves using the generalised least square method to generate a set of performance coefficients from the catalogue data at indicated reference conditions. Then the respective coefficients and indicated reference conditions are used in the model to simulate the heat pump performance. The variables that influenced the water-to-water heat pump performance are load side inlet water temperature, source side inlet temperature, source side water flow rate and load side water flow rate.

General

Template

You can load data to the dialog from a pre-defined heat pump template as a starting point for your particular heat pump component. A sample list of manufacturers data is provided in the templates and instructions on how to create your own templates from manufacturers data.

Heat pumps are usually selected using their Rated cooling capacity.

Reference COP

This setting is required if the [Reference cooling power consumption](#) is set to autosize. The nominal COP is defined by the [Reference cooling capacity](#) divided by the corresponding Reference cooling power consumption and is non-dimensional. This field is only used for sizing; if the Reference power consumption is set to a fixed value then the COP of the component during simulation will be determined by the ratio of

Reference cooling capacity divided by the corresponding Reference cooling power consumption and not by the value in this field. This COP does not include power for fluid circulation pumps, it is just for the heat pump itself.

Sizing factor

This setting allows a sizing factor for this component to be specified. The sizing factor is used when the component design inputs are autosized: the autosizing calculations are performed as usual and the results are multiplied by the sizing factor. The Sizing factor allows a component to be sized to meet part of the plant loop's design load while continuing to use the autosizing feature. For example if there are two heat pumps on the supply side, each one could be sized to be half of the design load.

Rated Flow Rate And Capacity

Reference load side flow rate

This autosizable setting contains the rated volumetric flow rate on the load side of the heat pump in (m³/s or gal/min). This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Reference source side flow rate

This autosizable setting contains the rated volumetric flow rate on the source side of the heat pump in (m³/s or gal/min). This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Reference cooling capacity

This autosizable setting contains the rated cooling capacity of the heat pump in W. This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Reference cooling power consumption

This autosizable setting contains the rated power consumption of the heat pump in W. This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Performance Curves

Cooling capacity curve

Select a [Quad Linear](#) Performance curve to be used to define the cooling capacity of this heat pump at runtime.

More details on how these coefficients are used are provided in the [technical section](#) of the Plant loop heat pumps page and in the [Engineering Reference](#).

Cooling compressor power curve

Select a [Quad Linear](#) Performance curve to define the Cooling compressor power curve. More details on how these coefficients in the curve are used are described in the [Engineering Reference](#).

Companion Component

Companion heating heat pump

This optional selection allows you to specify the companion heating water to water heat pump for this cooling component. This should be the name of an Equation Fit heat pump heating component on the same plant loop. Because the cooling and heating models are often intended to represent separate operating modes of one single reversible heat pump, it is useful to identify the corresponding heating heat pump. This is used with autosizing so that the sizing calculations can be coordinated ensuring that the two companion heat pumps get the same design reference flow rates.

Plant loop EIR Heat Pump

The plant loop EIR model is based on the same formulation used in the EIR chiller model. This allows curves specifying capacity and energy usage to be defined generically, enabling them to be applicable to a wide variety of components. In this model, the curve can be a quadratic function of temperatures or an even more generic lookup table.

Tip: Note: The EIR model is more flexible than the Equation fit model. It can be operated either as a water-sourced heat pump with the source side connected to another plant loop or as an air-sourced heat pump with the source side interacting with the outdoor air.

Data for this component is also considered to be easier to obtain than for the Equation fit model.

Note: This component is currently a constant flow device (it will always request its full design flow from the plant). A future addition is planned to make it respond to outlet setpoint in order to vary the flow request.

General

Reference COP

This setting defines the reference COP for this unit, which is used when calculating the power required for the heat pump under simulation conditions. The COP is defined as the load side energy transfer divided by the power input, with dimensionless units of [W/W].

Sizing factor

This setting allows the sizing of this unit to be scaled relative to the plant-determined autosized flow and heat capacity.

Flow Rates And Capacity

Load side reference flow rate

This autosizable setting defines the reference load side flow rate for the heat pump. This field is used as the nominal flow request for this heat pump unit during the simulation. The units for this field are (m³/s or gal/min).

Source side reference flow rate

This autosizable setting defines the reference source side flow rate for the heat pump. This field is used as the nominal flow request for this heat pump unit during the simulation. The units for this field are (m³/s or gal/min).

Important Note: For *air source* heat pumps, the **Source side reference flow rate** is an air flow rate, whereas for *water to water* heat pumps it is a water flow rate. Water source side reference flow rates will typically be much lower than Air source reference air flow rates.

Reference capacity

This autosizable setting defines the heat pump's reference load side cooling capacity (in W).

Performance Curves

Capacity modifier function of temperature curve

Select a [Bivariate](#) curve or table that defines an available capacity modifier of the unit as a function of the load side outlet temperature and the source side inlet temperature. The temperatures are in °C when used in the function and the output of the function is multiplied by the reference capacity to get a current available capacity.

Electric input to output ratio modifier function of temperature curve

Select a [Bivariate](#) curve or table that defines an EIR (1/COP) modifier as a function of the load side outlet temperature and the source side inlet temperature. The temperatures are in °C when used in the function. The output of this function and the output of the EIR Modifier Function of PLR are multiplied by the reference EIR to get a current EIR .

Electric input to output ratio modifier function of part load ratio curve

Select a [Univariate](#) curve or table that defines an EIR modifier as a function of the current part load ratio. The output of this function and the output of the EIR Modifier Function of Temperature are multiplied by the reference EIR to get a current EIR .

Condenser

Condenser type

Select the heat pump's condenser type. Options are:

- **1-Air Source** - for an air to water heat pump
- **2-Water Source** for a water to water heat pump

Heat Pump - Heating

General

Name

This field contains the unique identifying name for the heating side of the plant loop heat pump.

Type

DesignBuilder provides the following plant loop heat pump types:

- [1-Equation fit water to water heat pump](#) - corresponds to the EnergyPlus Equation fit model.
- [2-Plant loop EIR heat pump](#) - for modelling water to water or air to water heat pumps based on the EIR formulation.

Tip: For both these heat pump types, it is important to use manufacturer's performance data to ensure the accuracy of your simulations.

Equation Fit Water to Water Heat Pump

The Equation fit model uses four non-dimensional equations (curves) to predict the heat pump performance in cooling and heating modes. The methodology involves using the generalised least square method to generate a set of performance coefficients from the catalogue data at indicated reference conditions. Then the respective coefficients and indicated reference conditions are used in the model to simulate the heat

pump performance. The variables that influenced the water-to-water heat pump performance are load side inlet water temperature, source side inlet temperature, source side water flow rate and load side water flow rate.

General

Template

You can load data to the dialog from a pre-defined heat pump template as a starting point for your particular heat pump component. A sample list of manufacturers data is provided in the templates and instructions on how to create your own templates from manufacturers data.

Heat pumps are usually selected using their Rated heating capacity.

Reference COP

This setting is used if the [Reference heating power consumption](#) is set to autosize. The nominal COP is defined by the Reference heating capacity divided by the corresponding Reference heating power consumption and is non-dimensional. This field is only used for sizing; if the Reference power consumption is set to a fixed value then the COP of the component during simulation will be determined by the ratio of Reference heating capacity divided by the corresponding Reference heating power consumption and not by the value in this field. This COP does not include power for fluid circulation pumps, it is just for the heat pump itself.

Sizing factor

This setting allows a sizing factor for this component to be specified. The sizing factor is used when the component design inputs are autosized: the autosizing calculations are performed as usual and the results are multiplied by the sizing factor. The Sizing factor allows a component to be sized to meet part of the plant loop's design load while continuing to use the autosizing feature. For example if there are two heat pumps on the supply side, each one could be sized to be half of the design load.

Rated Flow Rate And Capacity

Reference load side flow rate

This autosizeable setting contains the rated volumetric flow rate on the load side of the heat pump in (m³/s or gal/min). This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Reference source side flow rate

This autosizeable setting contains the rated volumetric flow rate on the source side of the heat pump in (m³/s or gal/min). This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Reference heating capacity

This autosizeable setting contains the rated heating capacity of the heat pump in W. This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Reference heating power consumption

This autosizeable setting contains the rated power consumption of the heat pump in W. This corresponds to the highest load side heat transfer rate listed in the catalogue data.

Performance Curves

Heating capacity curve

Select a [Quad Linear](#) Performance curve to be used to define the heating capacity of this heat pump at runtime.

More details on how these coefficients are used are provided in the [technical section](#) of the Water to water heat pumps page and in the [Engineering Reference](#).

Heating compressor power curve

Select a [Quad Linear](#) Performance curve to define the Heating compressor power curve. More details on how these coefficients in the curve are used are described in the [Engineering Reference](#).

Companion Component

Companion cooling heat pump

This optional selection allows you to specify the companion cooling water to water heat pump for this heating component. This should be the name of an Equation Fit heat pump cooling component on the same plant loop. Because the cooling and heating models are often intended to represent separate operating modes of one single reversible heat pump, it is useful to identify the corresponding cooling heat pump. This is used with autosizing so that the sizing calculations can be coordinated ensuring that the two companion heat pumps get the same design reference flow rates.

Plant loop EIR Heat Pump

The plant loop EIR model is based on the same formulation used in the EIR chiller model. This allows curves specifying capacity and energy usage to be defined generically, enabling them to be applicable to a wide variety of components. In this model, the curve can be a quadratic function of temperatures or an even more generic lookup table.

Tip: Note: The EIR model is more flexible than the Equation fit model. It can be operated either as a water-sourced heat pump with the source side connected to another plant loop or as an air-sourced heat pump with the source side interacting with the outdoor air.

Data for this component is also considered to be easier to obtain than for the Equation fit model.

Note: This component is currently a constant flow device (it will always request its full design flow from the plant). A future addition is planned to make it respond to outlet setpoint in order to vary the flow request.

General

Reference COP

This setting contains the reference COP for this unit, which is used when calculating the power required for the heat pump under simulation conditions. The COP is defined as the load side energy transfer divided by the power input, with dimensionless units of [W/W].

Sizing factor

This setting allows the the sizing of this unit to be scaled relative to the plant-determined autosized flow and heat capacity.

Flow Rates And Capacity

Load side reference flow rate

This autosizable setting defines the reference load side flow rate for the heat pump. This field is used as the nominal flow request for this heat pump unit during the simulation. The units for this field are (m3/s or gal/min).

Source side reference flow rate

This autosizable setting defines the reference source side flow rate for the heat pump. This field is used as the nominal flow request for this heat pump unit during the simulation. The units for this field are (m3/s or gal/min).

Important Note: For *air source* heat pumps, the **Source side reference flow rate** is an air flow rate, whereas for *water to water* heat pumps it is a water flow rate. Water source side reference flow rates will typically be much lower than Air source reference air flow rates.

Reference capacity

This autosizable setting defines the heat pump's reference load side heating capacity (in W).

Performance Curves

Capacity modifier function of temperature curve

Select a [Bivariate](#) curve or table that defines an available capacity modifier of the unit as a function of the load side outlet temperature and the source side inlet temperature. The temperatures are in °C when used in the function and the output of the function is multiplied by the reference capacity to get a current available capacity.

Electric input to output ratio modifier function of temperature curve

Select a [Bivariate](#) curve or table that defines an EIR (1/COP) modifier as a function of the load side outlet temperature and the source side inlet temperature. The temperatures are in °C when used in the function. The output of this function and the output of the EIR Modifier Function of PLR are multiplied by the reference EIR to get a current EIR .

Electric input to output ratio modifier function of part load ratio curve

Select a [Univariate](#) curve or table that defines an EIR modifier as a function of the current part load ratio. The output of this function and the output of the EIR Modifier Function of Temperature are multiplied by the reference EIR to get a current EIR .

Condenser

Condenser type

Select the heat pump's condenser type. Options are:

- **1-Air Source** - for an air to water heat pump
- **2-Water Source** for a water to water heat pump

Generating Heat Pump Coefficients From Manufacturers Data

In order to allow manufacturer's water-to-water and water-to-air heat pump catalogue data to be used, EnergyPlus have provided spreadsheets to generate heating/cooling capacity coefficients (1 to 5) and compressor power coefficients (1 to 5) for each heat pump model. The spreadsheets are available from the [EnergyPlus support desk](#), or from [DesignBuilder](#).

Processing Catalogue Data

The following briefly introduces how to use spreadsheets mentioned above to generate datasets in the required formats from manufacturers catalogue data.

Manufacturer's catalogues normally provides more data points than are needed to generate coefficients for the EnergyPlus heat pumps. The EnergyPlus spreadsheet only requires 7 fields:

1. i.e., source side entering water temperature (EWT),
2. source side flow rate,
3. load side EWT
4. load side flow rate,
5. output (heating/cooling output),
6. input (compressor power input) and
7. heat of extraction (HE in heating mode)/heat of rejection (HR in cooling mode).

These seven fields must be rearranged as columns in the order specified by the EnergyPlus spreadsheet before attempting to generate the relevant coefficients.

The more data points used, the more accurate the calculation of the coefficients. As EnergyPlus states, the minimum data points for EF based model are 5. However, the data points must involve various inlet conditions (e.g. water flow rates, inlet water temperatures, etc.) that cover the entire range of the heat pump operating conditions.

Some catalogue data is in IP units, however the EnergyPlus spreadsheets can take data in either SI or IP units (data in IP units will be converted to SI units after a button is clicked in the spreadsheets). All processing is carried out using SI units.

The figure below shows how the relevant catalogue data is rearranged to the format EnergyPlus spreadsheets require and how the links between the two are established starting with the manufacturer's table on the right table and copying relevant data to the spreadsheet shown on the left). The example is based on a Carrier heat pump model 50PSW180 heating mode .

Required input data table

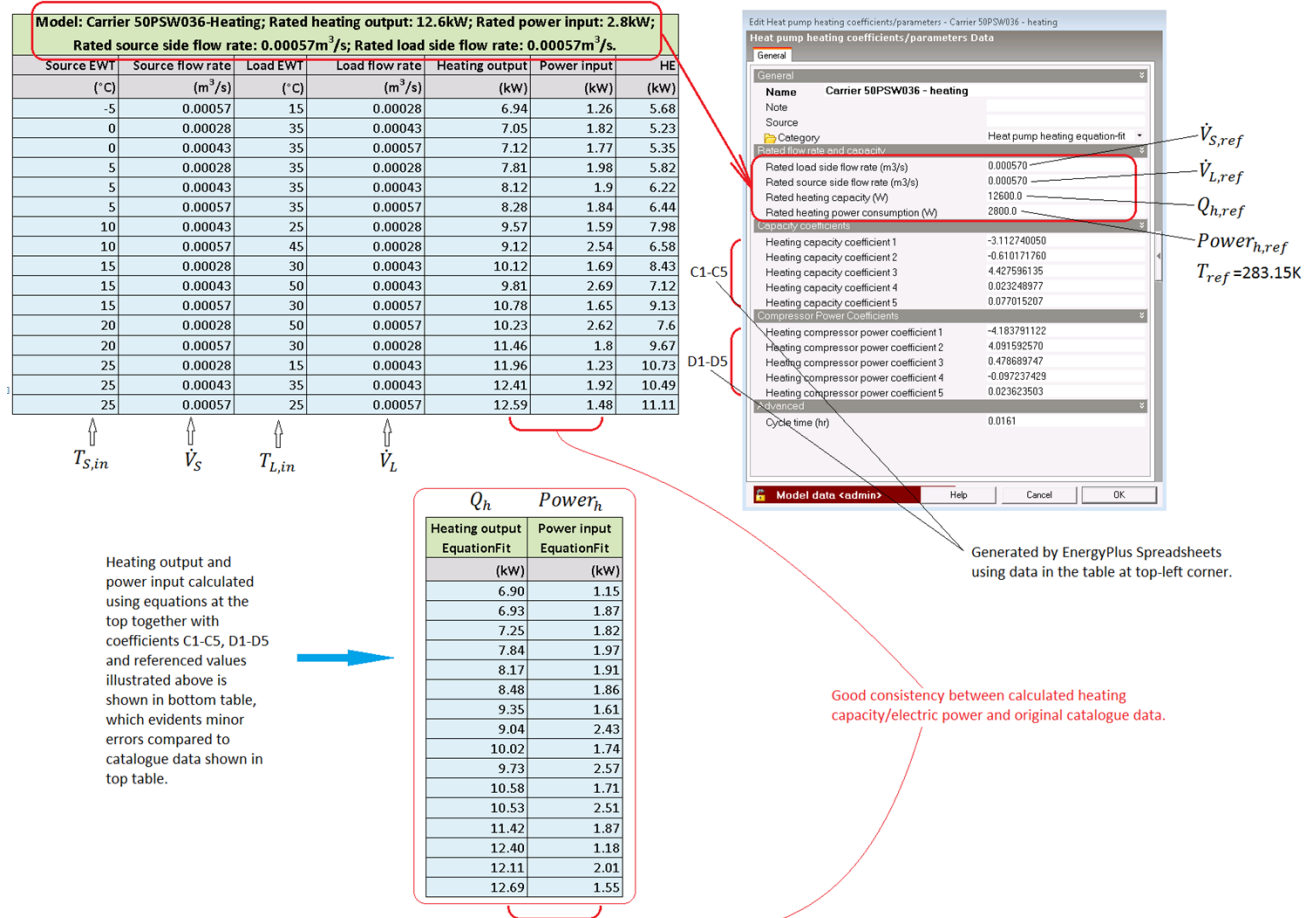
Source EWT (deg-C)	Source flow rate (m3/s)	Load EWT (deg-C)	Load flow rate (m3/s)	Output (kW)	Input (kW)	HE (kW)
15	0.00114	20	0.00114	37.7	6.37	31.4
25	0.00114	25	0.00114	38.6	6.9	31.7
35	0.00114	30	0.00114	37	7.32	29.7
45	0.00114	35	0.00114	36.8	7.83	29
55	0.00114	40	0.00114	35.3	8.59	26.7
65	0.00114	45	0.00114	35	9.31	25.7
75	0.00114	50	0.00114	34.1	10.06	24.1
85	0.00114	55	0.00114	33.3	10.86	22.5
95	0.00114	60	0.00114	32.6	11.69	21
105	0.00114	65	0.00114	32	12.54	19.5
115	0.00114	70	0.00114	31.4	13.41	18
125	0.00114	75	0.00114	30.8	14.3	16.5
135	0.00114	80	0.00114	30.2	15.24	15
145	0.00114	85	0.00114	29.7	16.19	13.5
155	0.00114	90	0.00114	29.2	17.17	12
165	0.00114	95	0.00114	28.7	18.18	10.5
175	0.00114	100	0.00114	28.2	19.22	9
185	0.00114	105	0.00114	27.7	20.29	7.5
195	0.00114	110	0.00114	27.2	21.39	6
205	0.00114	115	0.00114	26.7	22.52	4.5
215	0.00114	120	0.00114	26.2	23.68	3
225	0.00114	125	0.00114	25.7	24.87	1.5
235	0.00114	130	0.00114	25.2	26.09	0
245	0.00114	135	0.00114	24.7	27.34	-1.5
255	0.00114	140	0.00114	24.2	28.62	-3
265	0.00114	145	0.00114	23.7	29.93	-4.5
275	0.00114	150	0.00114	23.2	31.27	-6
285	0.00114	155	0.00114	22.7	32.64	-7.5
295	0.00114	160	0.00114	22.2	34.04	-9
305	0.00114	165	0.00114	21.7	35.47	-10.5
315	0.00114	170	0.00114	21.2	36.93	-12
325	0.00114	175	0.00114	20.7	38.42	-13.5
335	0.00114	180	0.00114	20.2	39.94	-15
345	0.00114	185	0.00114	19.7	41.49	-16.5
355	0.00114	190	0.00114	19.2	43.07	-18
365	0.00114	195	0.00114	18.7	44.68	-19.5
375	0.00114	200	0.00114	18.2	46.32	-21
385	0.00114	205	0.00114	17.7	47.99	-22.5
395	0.00114	210	0.00114	17.2	49.69	-24
405	0.00114	215	0.00114	16.7	51.42	-25.5
415	0.00114	220	0.00114	16.2	53.18	-27
425	0.00114	225	0.00114	15.7	54.97	-28.5
435	0.00114	230	0.00114	15.2	56.79	-30
445	0.00114	235	0.00114	14.7	58.64	-31.5
455	0.00114	240	0.00114	14.2	60.52	-33
465	0.00114	245	0.00114	13.7	62.43	-34.5
475	0.00114	250	0.00114	13.2	64.37	-36
485	0.00114	255	0.00114	12.7	66.34	-37.5
495	0.00114	260	0.00114	12.2	68.34	-39
505	0.00114	265	0.00114	11.7	70.37	-40.5
515	0.00114	270	0.00114	11.2	72.43	-42
525	0.00114	275	0.00114	10.7	74.52	-43.5
535	0.00114	280	0.00114	10.2	76.64	-45
545	0.00114	285	0.00114	9.7	78.79	-46.5
555	0.00114	290	0.00114	9.2	80.97	-48
565	0.00114	295	0.00114	8.7	83.18	-49.5
575	0.00114	300	0.00114	8.2	85.42	-51
585	0.00114	305	0.00114	7.7	87.69	-52.5
595	0.00114	310	0.00114	7.2	89.99	-54
605	0.00114	315	0.00114	6.7	92.32	-55.5
615	0.00114	320	0.00114	6.2	94.68	-57
625	0.00114	325	0.00114	5.7	97.07	-58.5
635	0.00114	330	0.00114	5.2	99.49	-60
645	0.00114	335	0.00114	4.7	101.94	-61.5
655	0.00114	340	0.00114	4.2	104.42	-63
665	0.00114	345	0.00114	3.7	106.93	-64.5
675	0.00114	350	0.00114	3.2	109.47	-66
685	0.00114	355	0.00114	2.7	112.04	-67.5
695	0.00114	360	0.00114	2.2	114.64	-69
705	0.00114	365	0.00114	1.7	117.27	-70.5
715	0.00114	370	0.00114	1.2	119.93	-72
725	0.00114	375	0.00114	0.7	122.62	-73.5
735	0.00114	380	0.00114	0.2	125.34	-75
745	0.00114	385	0.00114	-0.3	128.09	-76.5
755	0.00114	390	0.00114	-0.8	130.87	-78
765	0.00114	395	0.00114	-1.3	133.68	-79.5
775	0.00114	400	0.00114	-1.8	136.52	-81
785	0.00114	405	0.00114	-2.3	139.39	-82.5
795	0.00114	410	0.00114	-2.8	142.29	-84
805	0.00114	415	0.00114	-3.3	145.22	-85.5
815	0.00114	420	0.00114	-3.8	148.18	-87
825	0.00114	425	0.00114	-4.3	151.17	-88.5
835	0.00114	430	0.00114	-4.8	154.19	-90
845	0.00114	435	0.00114	-5.3	157.24	-91.5
855	0.00114	440	0.00114	-5.8	160.32	-93
865	0.00114	445	0.00114	-6.3	163.43	-94.5
875	0.00114	450	0.00114	-6.8	166.57	-96
885	0.00114	455	0.00114	-7.3	169.74	-97.5
895	0.00114	460	0.00114	-7.8	172.94	-99
905	0.00114	465	0.00114	-8.3	176.17	-100.5
915	0.00114	470	0.00114	-8.8	179.43	-102
925	0.00114	475	0.00114	-9.3	182.72	-103.5
935	0.00114	480	0.00114	-9.8	186.04	-105
945	0.00114	485	0.00114	-10.3	189.39	-106.5
955	0.00114	490	0.00114	-10.8	192.77	-108
965	0.00114	495	0.00114	-11.3	196.18	-109.5
975	0.00114	500	0.00114	-11.8	199.62	-111
985	0.00114	505	0.00114	-12.3	203.09	-112.5
995	0.00114	510	0.00114	-12.8	206.59	-114
1005	0.00114	515	0.00114	-13.3	210.12	-115.5
1015	0.00114	520	0.00114	-13.8	213.68	-117
1025	0.00114	525	0.00114	-14.3	217.27	-118.5
1035	0.00114	530	0.00114	-14.8	220.89	-120
1045	0.00114	535	0.00114	-15.3	224.54	-121.5
1055	0.00114	540	0.00114	-15.8	228.22	-123
1065	0.00114	545	0.00114	-16.3	231.93	-124.5
1075	0.00114	550	0.00114	-16.8	235.67	-126
1085	0.00114	555	0.00114	-17.3	239.44	-127.5
1095	0.00114	560	0.00114	-17.8	243.24	-129
1105	0.00114	565	0.00114	-18.3	247.07	-130.5
1115	0.00114	570	0.00114	-18.8	250.93	-132
1125	0.00114	575	0.00114	-19.3	254.82	-133.5
1135	0.00114	580	0.00114	-19.8	258.74	-135
1145	0.00114	585	0.00114	-20.3	262.69	-136.5
1155	0.00114	590	0.00114	-20.8	266.67	-138
1165	0.00114	595	0.00114	-21.3	270.68	-139.5
1175	0.00114	600	0.00114	-21.8	274.72	-141
1185	0.00114	605	0.00114	-22.3	278.79	-142.5
1195	0.00114	610	0.00114	-22.8	282.89	-144
1205	0.00114	615	0.00114	-23.3	287.02	-145.5
1215	0.00114	620	0.00114	-23.8	291.18	-147
1225	0.00114	625	0.00114	-24.3	295.37	-148.5
1235	0.00114	630	0.00114	-24.8	299.59	-150
1245	0.00114	635	0.00114	-25.3	303.84	-151.5
1255	0.00114	640	0.00114	-25.8	308.12	-153
1265	0.00114	645	0.00114	-26.3	312.43	-154.5
1275	0.00114	650	0.00114	-26.8	316.77	-156
1285	0.00114	655	0.00114	-27.3	321.14	-157.5
1295	0.00114	660	0.00114	-27.8	325.54	-159
1305	0.00114	665	0.00114	-28.3	329.97	-160.5
1315	0.00114	670	0.00114	-28.8	334.43	-162
1325	0.00114	675	0.00114	-29.3	338.92	-163.5
1335	0.00114	680	0.00114	-29.8	343.44	-165
1345	0.00114	685	0.00114	-30.3	347.99	-166.5
1355	0.00114	690	0.00114	-30.8	352.57	-168
1365	0.00114	695	0.00114	-31.3	357.18	-169.5
1375	0.00114	700	0.00114	-31.8	361.82	-171
1385	0.00114	705	0.00114	-32.3	366.49	-172.5
1395	0.00114	710	0.00114	-32.8	371.19	-174
1405	0.00114	715	0.00114	-33.3	375.92	-175.5
1415	0.00114	720	0.00114	-33.8	380.68	-177
1425	0.00114	725	0.00114	-34.3	385.47	-178.5
1435	0.00114	730	0.00114	-34.8	390.29	-180
1445	0.00114	735	0.00114	-35.3	395.14	-181.5
1455	0.00114	740	0.00114	-35.8	400.02	-183
1465	0.00114	745	0.00114	-36.3	404.93	-184.5
1475	0.00114	750	0.00114	-36.8	409.87	-186
1485	0.00114	755	0.00114	-37.3	414.84	-187.5
1495	0.00114	760	0.00114	-37.8	419.84	-189
1505	0.00114	765	0.00114	-38.3	424.87	-190.5
1515	0.00114	770	0.00114	-38.8	429.93	-192
1525	0.00114	775	0.00114	-39.3	435.02	-193.5
1535	0.00114	780	0.00114	-39.8	440.14	-195
1545	0.00114	785	0.00114	-40.3	445.29	-196.5
1555	0.00114	790	0.00114	-40.8	450.47	-198
1565	0.00114	795	0.00114	-41.3	455.68	-199.5
1575	0.00114	800	0.00114	-41.8	460.92	-201
1585	0.00114	805	0.00114	-42.3	466.19	-202.5
1595	0.00114	810	0.00114	-42.8	471.49	-204
1605	0.00114	815	0.00114	-43.3	476.82	-205.5
1615	0.00114	820	0.00114	-43.8	482.18	-207
1625	0.00114	825	0.00114	-44.3	487.57	-208.5
1635	0.00114	830	0.00114	-44.8	493.0	-210
1645	0.00114					

Reference Condition Explanation And EquationFit Functionality Verification

Using Carrier water-to-water heat pump model 50PSW036 (heating) sample data as an example, the figure below describes where the reference data used in equations is allocated. Applying coefficients and referenced data into equations, the heating output and power (electric) input figures can be calculated (two columns at the bottom of the figure). These calculated figures show a good consistency with original catalogue data, proving a working functionality of EquationFit model.

$$\frac{Q_h}{Q_{h,ref}} = C1 + C2 \left[\frac{T_{L,in}}{T_{ref}} \right] + C3 \left[\frac{T_{S,in}}{T_{ref}} \right] + C4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + C5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

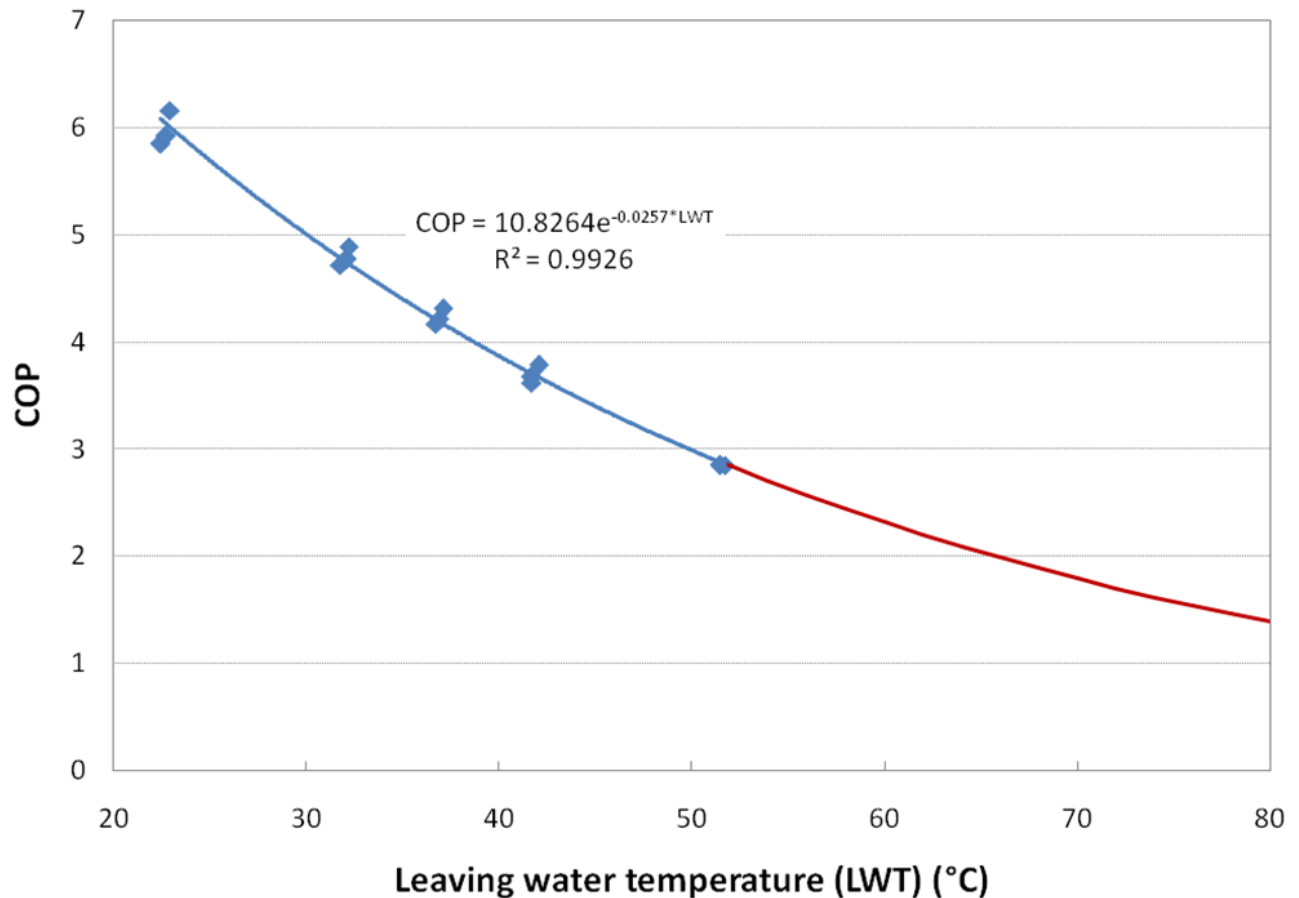
$$\frac{Power_h}{Power_{h,ref}} = D1 + D2 \left[\frac{T_{L,in}}{T_{ref}} \right] + D3 \left[\frac{T_{S,in}}{T_{ref}} \right] + D4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + D5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$



Recommendation for efficient use when heating

The highest supply water temperature provided in catalogue data is normally between 50°C-60°C. This is not normally hot enough to supply hot water radiators and heating coils and you are advised not to connect to water radiators to these systems. However, EnergyPlus does not limit the output water temperature to the catalogue maximum, so in practice, if setpoints allow it, supply water temperatures can achieve the typical hot water radiator/baseboard temperature levels of 75°C-80°C. The figure below uses Carrier model 50PSW036 (highest leaving water temperature 51.8°C) heating as an example to show the impact of using heat pumps in this way. Extrapolation of catalogue data is not normally recommended however the trend line (following red coloured line) shows its COP value is much reduced at 1.38 when output water temperature reaches 80°C. So

this sort of system configuration is clearly not nearly as energy efficient as a low temperature system such as a heated floor which require supply temperatures around 35°C. Underfloor heating is therefore recommended for use with GSHP.



Ground Source Heat Pump (GSHP) Example

Create a building model first then follow the steps below to set up a water to water (ground source) heat pump (GSHP) system using vertical boreholes to provide both heating through heated floors and cooling through chilled beams.

Note: This example is currently based on using Equation fit heat pumps and has not yet been updated to use the more up date EIR heat pumps.

1. Run DesignBuilder Heating design calculation to obtain peak heating load.

Air Temperature (°C)	22.00
Radiant Temperature (°C)	18.59
Operative Temperature (°C)	20.30
Outside Dry-Bulb Temperature (°C)	-4.80
Glazing (kW)	-6.57
Walls (kW)	-2.88
Ground Floors (kW)	-0.28
Roofs (kW)	-7.07
External Infiltration (kW)	-26.03
External Vent. (kW)	-42.45
Zone Sensible Heating (kW)	85.15

Edit	Visualise	Heating design	Cooling design	Simulation	CFD	Daylighting	Cost and Carbon
------	-----------	----------------	----------------	------------	-----	-------------	-----------------

In this case, peak heating load is 85 kW.

2. Run DesignBuilder Cooling design calculation to obtain peak cooling load.

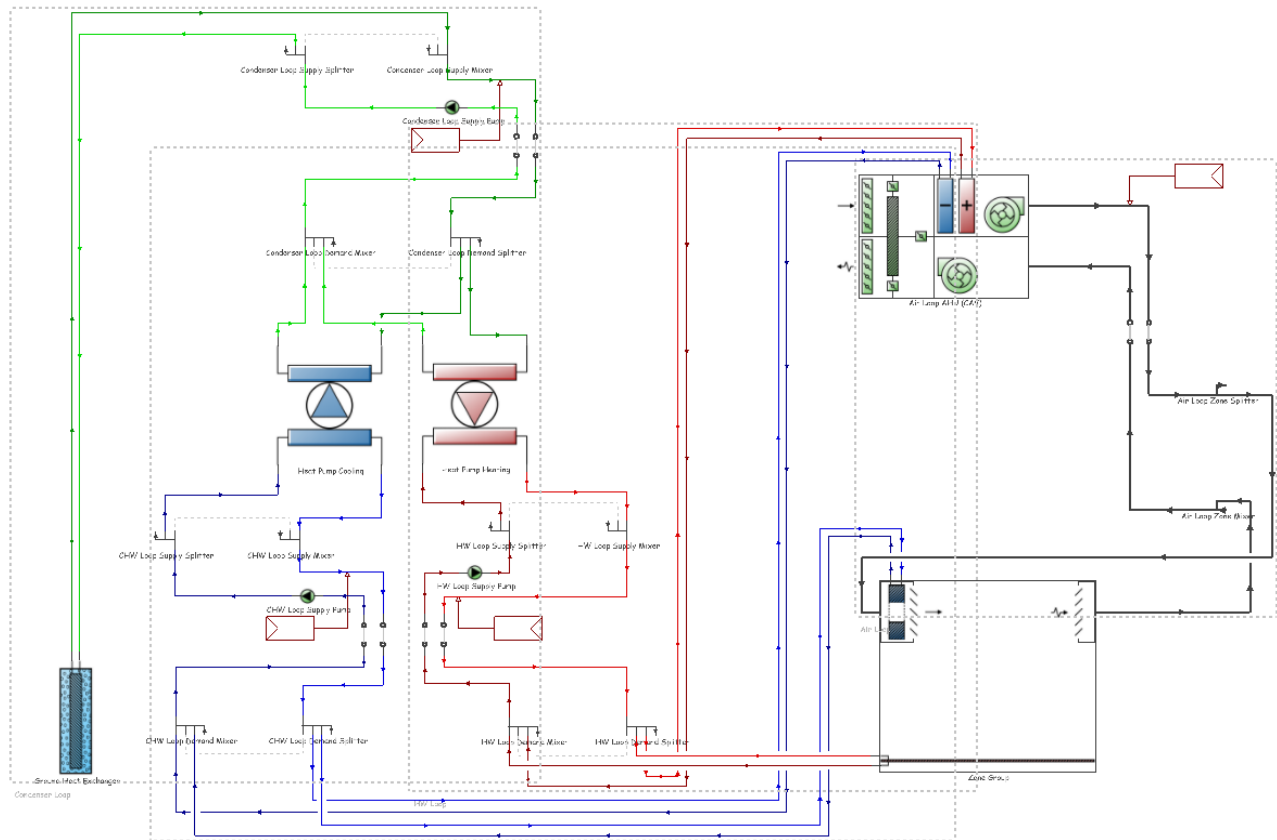
Air Temperature (°C)	25.23	24.61	24.00	24.00	24.00	24.00	24.00	24.00	24.00	28.11	27.01
Radiant Temperature (°C)	26.69	26.06	26.39	28.24	29.61	30.27	30.81	30.92	30.41	29.55	28.40
Operative Temperature (°C)	25.96	25.34	25.20	26.12	26.81	27.13	27.40	27.46	27.20	28.83	27.71
Outside Dry-Bulb Temperature (°C)	18.08	17.49	17.49	19.82	23.31	25.74	27.00	26.42	24.67	22.15	20.40
Glazing (kW)	-2.48	-2.46	-8.93	-9.05	-7.98	-5.61	-7.09	-8.02	-8.03	-2.35	-2.43
Walls (kW)	3.60	3.66	2.33	-1.22	-2.41	-2.38	-2.35	-1.65	-0.27	0.52	2.14
Ground Floors (kW)	4.20	5.13	-0.10	-6.55	-8.45	-7.91	-9.31	-8.12	-5.48	-0.87	0.92
Roofs (kW)	-0.79	-1.62	-6.05	-6.64	1.25	8.24	9.75	10.44	9.23	6.46	2.58
External Infiltration (kW)	-6.41	-6.39	-5.85	-3.72	-0.61	1.52	2.61	2.11	0.59	-5.27	-5.88
External Vent. (kW)	0.00	0.00	0.00	-3.04	-0.99	1.86	4.25	3.43	0.24	0.00	0.00
General Lighting (kW)	0.00	0.00	0.00	21.73	21.73	21.73	21.73	21.73	21.73	0.00	0.00
Computer + Equip (kW)	0.64	0.64	0.64	12.79	12.79	12.79	12.79	12.79	12.79	0.64	0.64
Occupancy (kW)	0.00	0.00	0.00	4.13	8.26	6.19	8.26	8.26	2.06	0.00	0.00
Solar Gains Exterior Windows (kW)	0.00	0.00	14.38	18.29	19.76	17.19	19.55	18.63	15.08	0.00	0.00
Zone Sensible Cooling (kW)	0.00	0.00	-5.06	-35.71	-52.48	-60.29	-69.12	-68.90	-56.49	0.00	0.00
Sensible Cooling (kW)	0.00	0.00	-5.06	-32.64	-51.48	-62.23	-73.57	-72.49	-56.75	0.00	0.00
Total Cooling (kW)	0.00	0.00	-6.16	-38.51	-61.89	-71.75	-84.50	-83.50	-61.95	0.00	0.00
Relative Humidity (%)	49.83	51.71	52.19	50.51	50.98	50.03	50.24	50.21	48.81	40.42	44.52
Mech Vent + Nat Vent + Infiltration (ac/h)	0.72	0.72	0.72	1.88	3.02	2.43	3.00	3.00	1.28	0.71	0.72

Edit	Visualise	Heating design	Cooling design	Simulation	CFD	Daylighting	Cost and Carbon
------	-----------	----------------	----------------	------------	-----	-------------	-----------------

In this case, (coincidentally) the peak cooling load is also 85 kW.

3. Create detailed HVAC system of ground source heat pump model.

The layout below shows a typical heating and cooling application. This was loaded from the [GSHP - Heated Floor and Chilled Beams](#) HVAC template but you can create a similar layout from scratch to suit your particular configuration.



4. Set heat pump heating component data.

Navigate to the Heat pump heating component and edit it. Load an appropriate heat pump template using the heat pump [Rated heating capacity](#) data to match the heat pump with the design heating load calculated in step 1. You can see in the example below a heat pump having 104.6 kW has been selected to meet the peak heating load of 85 kW giving some spare capacity.

Alternatively you can create your own heating capacity coefficients and heating compressor power coefficients together with rated flow rates, rated heating capacity and rated heating power consumption from catalogue data.

Note that the rated load side flow rate for the selected heat pump is 0.00442 m³/s.

Edit Water-to-Water Heat Pump Heating -

Water-to-Water Heat Pump Heating Data

General

Name	Heat Pump Heating
Template	ClimateMaster TMW340 - heating
Type	1-Equation Fit
Rated load side flow rate (m3/s)	0.004420
Rated source side flow rate (m3/s)	0.004420
Rated heating capacity (W)	104600.0
Rated heating power consumption (W)	25800.0

Heating Capacity Coefficients

Heating capacity coefficient 1	-1.73958113
Heating capacity coefficient 2	-1.06993246
Heating capacity coefficient 3	3.52744356
Heating capacity coefficient 4	0.09384548
Heating capacity coefficient 5	0.07982408

Heating Compressor Power Coefficients

Heating compressor power coefficient 1	-3.33782041
Heating compressor power coefficient 2	3.33442764
Heating compressor power coefficient 3	0.47274952
Heating compressor power coefficient 4	0.01351532
Heating compressor power coefficient 5	0.01442905

Advanced

Cycle time (hr)	0.01610
-----------------	---------

Help

Info Data

Heat pump heating coefficients/parameters

- ClimateMaster TMW036 - heating
- ClimateMaster TMW060 - heating
- ClimateMaster TMW120 - heating
- ClimateMaster TMW170 - heating
- ClimateMaster TMW340 - heating
- ClimateMaster TMW360 - heating
- ClimateMaster TMW600 - heating
- ClimateMaster TMW840 - heating
- Trane EXWE060 - heating
- Trane EXWE120 - heating

Data Report (Not Editable)

General

ClimateMaster TMW340 - heating

Note

Source

Category Heat pump h

Rated flow rate and capacity

Rated load side flow rate (m3/s)	0.004420
Rated source side flow rate (m3/s)	0.004420
Rated heating capacity (W)	104600.0
Rated heating power consumption (W)	25800.0

Capacity coefficients

Heating capacity coefficient 1	-1.739581126
Heating capacity coefficient 2	-1.069932459
Heating capacity coefficient 3	3.527443560
Heating capacity coefficient 4	0.093845484
Heating capacity coefficient 5	0.079824077

Model data <admin>

Help Cancel OK

5. Set heat pump cooling component data.

Navigate to the Heat pump cooling component and edit it. Load an appropriate heat pump model from template using the heat pump [Rated cooling capacity](#) data to match the heat pump with the design cooling load calculated in step 2. You can see in the example below a heat pump having 102 kW has been selected to meet the peak cooling load of 85 kW giving some spare capacity.

Alternatively you can create your own cooling capacity coefficients and cooling compressor power coefficients together with rated flow rates, rated heating capacity and rated cooling power consumption from catalogue data.

Note that the rated load side flow rate for the selected heat pump is 0.00442 m3/s.

Edit Water-to-Water Heat Pump Cooling -

Water-to-Water Heat Pump Cooling Data

General

Name: Heat Pump Cooling
 Template: ClimateMaster TMW340 - coolin ...
 Type: 1-Equation Fit

Rated load side flow rate (m3/s): 0.004420
 Rated source side flow rate (m3/s): 0.004420
 Rated cooling capacity (W): 102000.0
 Rated cooling power consumption (W): 24000.0

Cooling Capacity Coefficients

Cooling capacity coefficient 1: -1.43192302
 Cooling capacity coefficient 2: 3.34478686
 Cooling capacity coefficient 3: -1.25554159
 Cooling capacity coefficient 4: 0.10386455
 Cooling capacity coefficient 5: 0.04784334

Cooling Compressor Power Coefficients

Cooling compressor power coefficient 1: -3.01836980
 Cooling compressor power coefficient 2: 0.30928663
 Cooling compressor power coefficient 3: 3.33456029
 Cooling compressor power coefficient 4: 0.00974232
 Cooling compressor power coefficient 5: -0.13184990

Advanced

Cycle time (hr): 0.01610

Heat pump cooling coefficients/parameters

- ClimateMaster TMW036 - cooling
- ClimateMaster TMW060 - cooling
- ClimateMaster TMW120 - cooling
- ClimateMaster TMW170 - cooling
- ClimateMaster TMW340 - cooling**
- ClimateMaster TMW360 - cooling
- ClimateMaster TMW600 - cooling
- ClimateMaster TMW840 - cooling
- Trane EXWE060 - cooling
- Trane EXWE120 - cooling

Data Report (Not Editable)

General

ClimateMaster TMW340 - cooling

Note
 Source
 Category: Heat pump c

Rated flow rate and capacity

Rated load side flow rate (m... 0.004420
 Rated source side flow rate (... 0.004420
 Rated heating capacity (W) 102000.0
 Rated heating power consu... 24000.0

Capacity coefficients

Cooling capacity coefficient 1 -1.431923021
 Cooling capacity coefficient 2 3.344786860
 Cooling capacity coefficient 3 -1.255541585
 Cooling capacity coefficient 4 0.103864549
 Cooling capacity coefficient 5 0.047843336

Model data <admin> Help Cancel OK

6. Set ground heat exchanger data.

Open the Ground heat exchanger dialog and use the quick sizing table (below) to decide which template to load. Rounding up design heating and cooling loads from 85 kW to 90 kW you can see that 40 boreholes are required to meet this combination of loads.

		Peak cooling load (kW)									
		10	20	30	40	50	60	70	80	90	100
Peak heating load (kW)	10	4	4	8	8	8	12	12	12	16	16
	20	8	8	8	12	12	12	12	12	16	16
	30	12	12	12	12	16	16	16	16	16	16
	40	16	16	16	16	16	24	24	24	24	24
	50	24	24	24	24	24	24	24	24	24	24
	60	24	24	24	24	24	24	24	24	24	40
	70	40	40	40	40	40	40	40	40	40	40
	80	40	40	40	40	40	40	40	40	40	40
	90	40	40	40	40	40	40	40	40	40	40
	100	40	40	40	40	40	40	40	40	40	40

Change both **Design flow rate** and **Maximum flow rate** to 0.00884 which is the sum of 0.00442 + 0.00442 for the heat pump heating and heat pump cooling components respectively. Refer to steps 4 and 5. If you have used your own specific heat pump heating/cooling data generated from catalogue data you should alter this flow rate figure accordingly.

Edit Ground heat exchanger -

Ground heat exchanger Data

General

G Function Data

General

Name

Ground Heat Exchanger

Template

U-tube 76m 40-boreholes

Type

1-Vertical

Ground heat exchanger - vertical

Ground temperature (°C)

13.375

Flow rate

Design flow rate (m3/s)

0.00884

Maximum flow rate (m3/s)

0.00884

Borehole and pipe geometry

Number of boreholes

40

Borehole length (m)

76.0

Borehole radius (m)

0.063508

Pipe outer diameter (m)

0.0266667

U-tube distance (m)

0.0253977

Pipe thickness (m)

0.00241285

Ground, grout and pipe properties

Ground thermal conductivity (W/m-K)

0.692626

Ground thermal heat capacity (J/m3-K)

2347000.00

Grout thermal conductivity (W/m-K)

0.692626

Pipe thermal conductivity (W/m-K)

0.391312

Others

Maximum length of simulation

1

G-function reference ratio

0.000836

Help

Info

Data

Ground heat exchanger - vertical

Quick/Simple sizing

Generally, there is a quick/simple way in doing ground loop heat exchanger sizing. The procedure is to use DesignBuilder 'Heating Design' and 'Cooling Design' calculations to obtain peak heating and cooling loads in kW, then look at the table below to get a minimum borehole number for the model.

		Peak cooling load (kW)									
		10	20	30	40	50	60	70	80	90	100
Peak heating load (kW)	10	4	4	8	8	8	12	12	12	16	16
	20	8	8	8	12	12	12	12	12	16	16
	30	12	12	12	12	16	16	16	16	16	16
	40	16	16	16	16	16	24	24	24	24	24
	50	24	24	24	24	24	24	24	24	24	24
	60	24	24	24	24	24	24	24	24	24	40
	70	40	40	40	40	40	40	40	40	40	40
	80	40	40	40	40	40	40	40	40	40	40
	90	40	40	40	40	40	40	40	40	40	40
	100	40	40	40	40	40	40	40	40	40	40

Recommended flow rate for ground heat exchanger is the maximum of sum of rated flow rates from all connected heat pumps for heating or heat pumps for cooling.

Detailed/Precise sizing
The net heating or cooling of the ground over each season clearly depends on the accumulated heat rejection/extraction, and therefore on the building loads throughout the whole year (not only the peak loads mentioned above), which might be available through simulation based on a very simple HVAC setting in DesignBuilder.

The solution for above relies on using other ground loop heat exchanger tools (not supplied by DesignBuilder or EnergyPlus), which are capable of doing detailed/precise sizing.

Load vertical ground heat exchanger data from a file
When using vertical ground heat exchanger, you can load settings and g-functions from idf files stored on your local drive.

[Load vertical ground heat exchanger data](#)

Model data <admin>

Import

Help

Cancel

OK

7. Set condenser loop flow rate.

Change **Maximum loop flow rate** from **Autosize** to 0.00884, the same as (or greater than) ground heat exchanger flow rate, derived in step 6.

Important note: You must not use Autosize for condensers connected to GSHPs.

Edit Plant loop -

Plant loop Data	
General	Plant Equipment Operation
General	
Name	Condenser Loop
Fluid type	1-Water
Plant loop volume (m3)	Autocalculate
Flow Type	
Plant loop flow type	2-Variable flow
Temperature	
Maximum loop temperature (°C)	50.00
Minimum loop temperature (°C)	5.00
Flow Rate	
Maximum loop flow rate (m3/s)	0.008840
Minimum loop flow rate (m3/s)	0.000000
Load distribution scheme	1-Sequential
Plant loop demand calculation scheme	1-SingleSetPoint
Sizing	
Design loop exit temperature (°C)	29.00
Loop design temperature difference (deltaC)	5.00
Operation	
Availability schedule	On
Outside Temperature Operation	
☐ Outside temperature operation	

Model data <admin>

Help

Info Data

Condenser Loop

The Condenser loop consists of:

- Supply sub loop which contains one or more cooling towers, a pump and a setpoint controller
- Demand sub loop which distributes the water to the condenser of a water-cooled chiller.

This dialog covers the sizing and operation details of the overall loop.

Load Distribution Scheme

The Load Distribution Scheme selects the algorithm used to sequence equipment operation in order to meet the plant loop demand. There are 3 options:

- 'Sequential' uses each piece of equipment to its maximum part load ratio and will operate the last required piece of equipment between its minimum and maximum part load ratio in order to meet the loop demand.
- 'Optimal' operates each piece of equipment at its optimal part load ratio and will operate the last component between its minimum and maximum part load ratio in order to meet the loop demand.
- 'Uniform' evenly distributes the loop demand amongst all available components on the equipment list for a given load range.

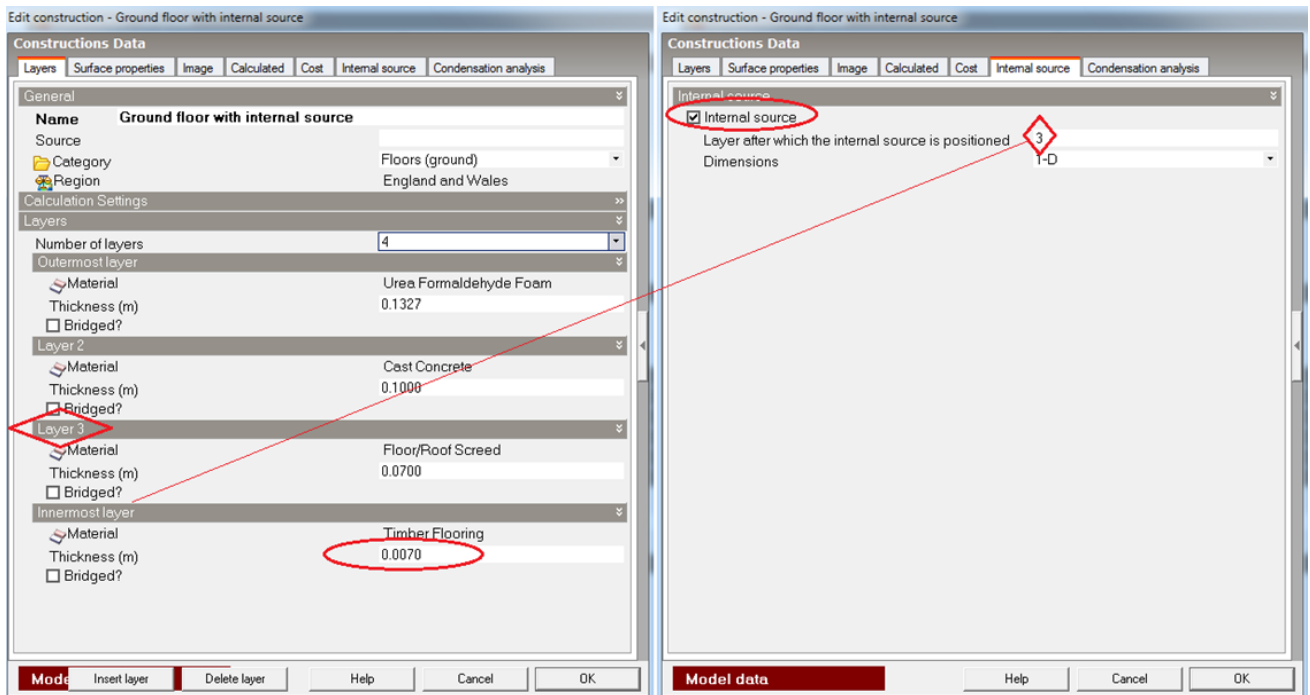
Help Cancel OK

8. Turn on ground floor's internal source in order to apply underfloor heating system.

Edit any Ground floor, Internal floor or External floor constructions where the underfloor heating applies to ensure that an internal source is included in the construction.

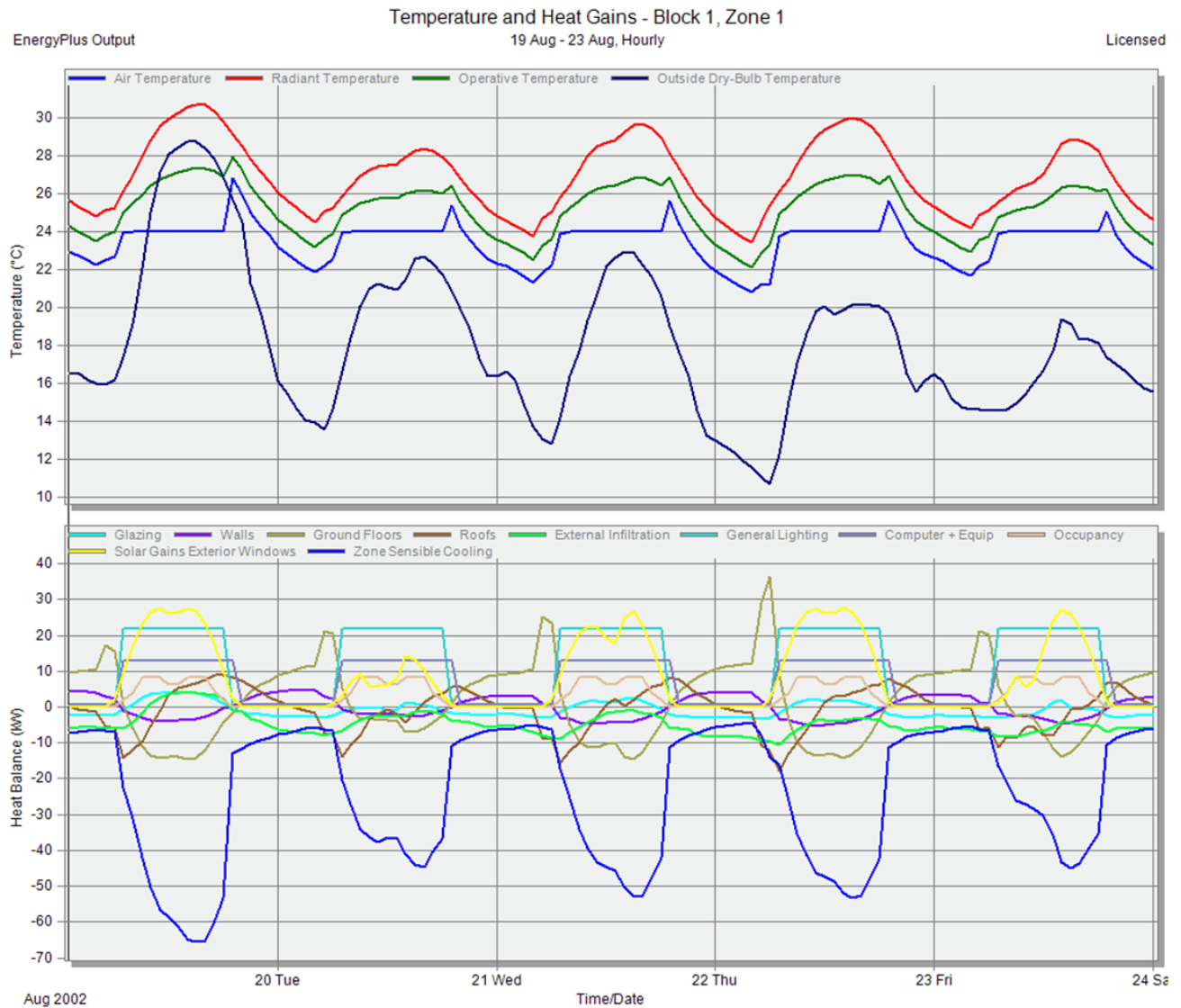
To ensure the best heat transfer from the embedded hot water pipes to the room occupied region, the innermost layer (top layer) of the construction should have a relatively small thickness using thermally conductive timber flooring or tiles rather than a more insulating material such as carpet.

On the [Internal source](#) tab, check the [Internal source](#) option and assign the position of the internal source within the construction layers. The internal source layer should normally be placed at one layer below (inside) innermost layer.



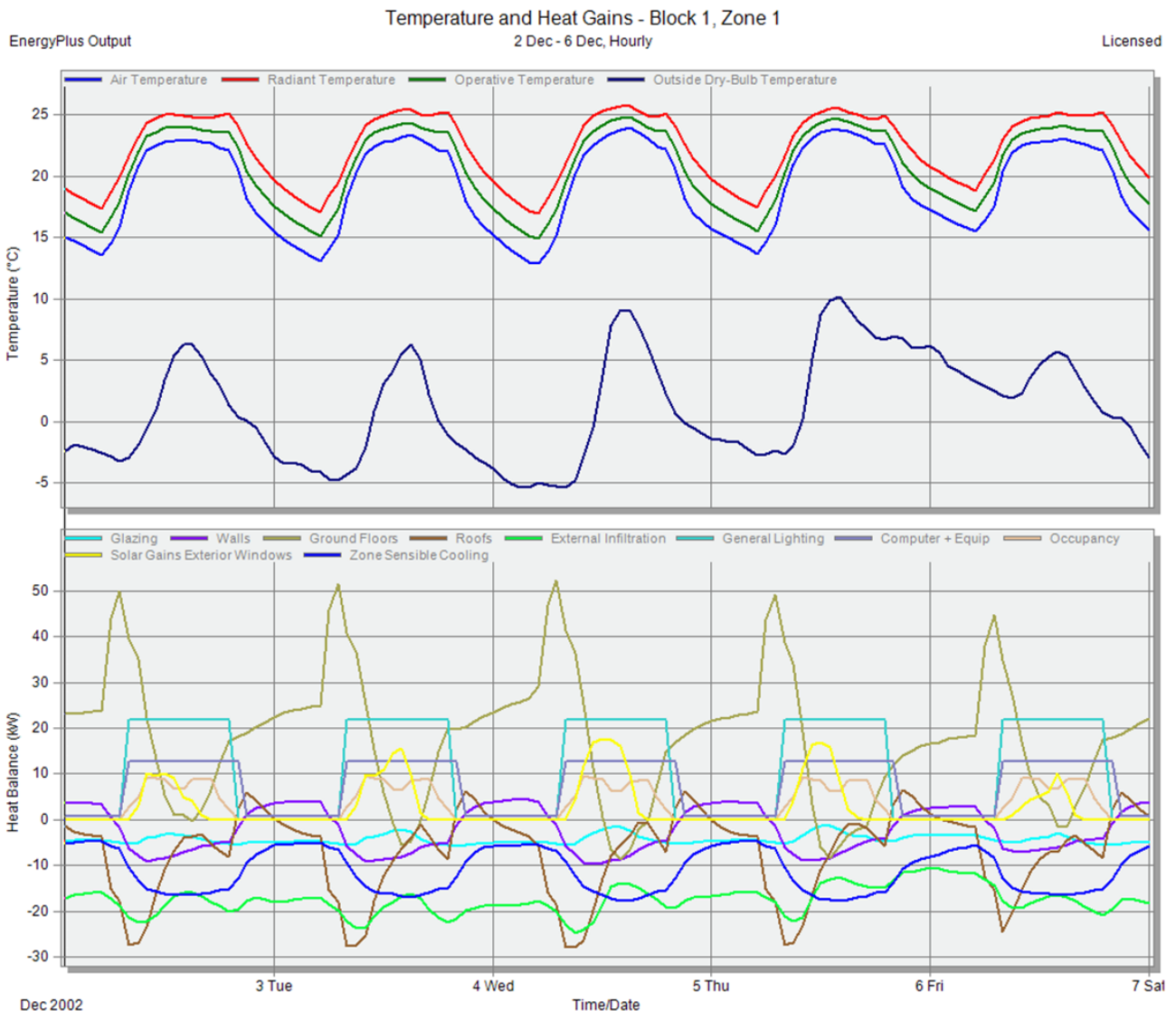
9. Run simulation and check the results.

(1) Summer modelling results:



During occupied hours each day in summer, room air temperature was controlled well at 24°C (cooling setpoint temperature is 24°C).

(2) Winter modelling results:



During occupied hours each day in winter, room air temperature was controlled at about 23°C (heating setpoint temperature is 22°C).

Air To Water Heat Pump / ASHP Data

The air to water heat pump, aka air source heat pump (ASHP), is a compound object consisting of a [water heater](#) tank, [a direct expansion \(DX\) "coil"](#) (i.e., an air-to-water DX compression system which includes a water heating coil, air coil, compressor, and water pump), and a fan to provide air flow across the air coil associated with the DX compression system. These objects work together to model a system which heats water using zone air, outdoor air, or a combination of zone and outdoor air as the primary heat source.

Note: The terms "Water heater heat pump", "Air to water heat pump", "Air source heat pump" and "ASHP" are used interchangeably in the documentation.

Various configurations of tank location, inlet air source, and DX coil compressor location can be modelled.

Air to water heat pumps can be added in 2 main locations:

1. Supply side of an auxiliary heating loop.
2. Supply side of DHW loop.

Compressor Settings

Compressor setpoint temperature schedule

This [schedule](#) defines the set point (or “cut-out”) temperature for the heat pump compressor. Temperature values used in this schedule should be in degrees Celsius. The heat pump compressor cycles off when the tank water reaches this set point temperature. Once the heat pump compressor cycles off, the tank water temperature drops until it falls below the set point temperature minus the dead band temperature difference defined below (i.e., the “cut-in” temperature). At this point, the heat pump compressor cycles on and remains on until the heat pump compressor set point temperature is reached.

Important Note: The overall heat pump compressor “cut-in” temperature (**Compressor setpoint temperature schedule** value minus the [dead band temperature difference](#) below) must be greater than the [Water Heater Setpoint temperature schedule](#) (“cut-out” temperature) in order for the heat pump to provide heat to the water heater tank.

Dead band temperature difference

This numeric field contains the dead band temperature difference (in °C or °F). The heat pump compressor “cut-in” temperature is defined as the **Compressor set point temperature** defined above minus this dead band temperature difference. The heat pump compressor cycles on when the water temperature in the tank falls below the “cut-in” temperature. The heat pump compressor remains on until the water temperature in the tank rises above the compressor set point (“cut-out”) temperature defined above. The dead band temperature difference must be greater than 0°C and less than or equal to 20°C.

In this model, the heat pump water heater’s DX compression system is considered the primary heat source and the water tank’s heater (element or burner) provides supplemental heat as necessary. Therefore, the cut-in temperature for the heat pump compressor (set point minus dead band temperature difference) is usually higher than the set point temperature for the heater (element or burner) in the associated water heater tank object. At times when the water heater tank set point temperature is greater than the cut-in temperature of the heat pump compressor, the heat pump compressor is disabled and the tank’s heater is used to heat the water.

Minimum inlet air temperature for compressor operation

This numeric field contains the minimum inlet air dry-bulb temperature entering the air coil (evaporator) and fan section (in °C or °F) below which the heat pump compressor does not operate. The minimum inlet air dry-bulb temperature should be greater than or equal to -20°C.

Note: The minimum **Minimum inlet air temperature for compressor operation** setting allowed in DesignBuilder is -20°C, but if you intend to export the model to run outside DesignBuilder in EnergyPlus v8.6, the minimum temperature allowed is higher at -5°C. In this case, if you need to use a value lower than -5°C, you should either copy the idd file stored in the DesignBuilder EnergyPlus folder to the EnergyPlus folder you are using, or manually edit the EnergyPlus idd file. In EnergyPlus v8.9 and later, the minimum limit has been removed so this issue will not arise.

Compressor location

Select the location of the heat pump compressor and the air temperature for this location used to control operation of the compressor’s crankcase heater in the [Air to water heat pump coil](#). Valid entries are:

- **1-Schedule**, in which case a compressor ambient temperature schedule must be defined below.
- **2-Zone** when the crankcase heater operation is controlled based on the air temperature in the zone in which the ASHP is located.
- **3-Outdoors** in which case the crankcase heater operation is controlled based on the outdoor air temperature.

The **2-Zone** option is only available when the ASHP is used as an object placed within an HVAC zone.

Compressor ambient temperature schedule

This schedule is available when the **1-Schedule** option is selected as the Compressor location above. It defines the ambient air temperature surrounding the heat pump compressor, which is used to control the compressor's crankcase heater operation.

Flow Rate Settings

Condenser water flow rate

This numeric field contains the heat pump's condenser water flow rate (in m3/s or gal/min). It is the actual condenser water flow rate to be simulated, which may differ from the [Rated condenser water flow rate](#) specified for the heat pump's DX coil. This water flow rate must be greater than 0 or this field is autoc calculatable. If autocalculated (field value = autocalculate), the condenser water flow rate is set equal to the rated heating capacity of the heat pump's DX coil multiplied by 4.487E-8 m3/s/W. When this flow rate is different from the [Rated condenser water flow rate](#) specified for the heat pump's DX coil, the user should also specify a [Heating capacity function of water flow fraction curve](#) and a [Heating COP function of water flow fraction curve](#) in the associated DX coil object to account for differences in capacity and power consumption at the off-rated water flow rate.

Evaporator air flow rate

This numeric field contains the air flow rate across the heat pump's air coil (evaporator) (in m3/s or ft3/min). It is the actual air flow rate to be simulated, which may differ from the [Rated evaporator air flow rate](#) specified for the heat pump's DX coil. Values must be greater than 0 or this field is autoc calculatable. If autocalculated (field value = autocalculate), the evaporator air flow rate is set equal to the rated heating capacity of the heat pump's DX coil multiplied by 5.035E-5 m3/s/W. When this flow rate is different from the [Rated evaporator air flow rate](#) specified for the heat pump's DX coil, the user should also specify a [Heating capacity function of air flow fraction curve](#) and a [Heating COP function of air flow fraction curve](#) in the associated DX coil object to account for differences in capacity and power consumption at the off-rated water flow rate.

Inlet Air Settings

Inlet air configuration

Select the configuration of the air flow path through the heat pump air coil (evaporator) and fan section. Valid entries are:

- **1-Schedule**, in which case names an inlet air temperature schedule and an inlet air humidity schedule are defined in the fields below.
- **2-Zone air only**, option only available when the ASHP is placed as a zone unit. The inlet air is sourced from the zone in which the ASHP is placed.
- **3-Outdoor air only**.
- **4-Zone and outdoor air**, option only available when the ASHP is placed as a zone unit. The inlet air is sourced from the zone in which the ASHP is placed and mixed with outdoor air based on the [Inlet air mixer schedule](#) below.

Inlet air temperature schedule

Select the schedule used to define the dry-bulb temperature of the inlet air to the heat pump air coil (evaporator) and fan section. Schedule values should be in degrees Celsius. This field is only required when the [Inlet air configuration](#) defined above is specified as **1-Schedule**.

Inlet air humidity schedule

Select the schedule used to define the humidity of the inlet air to the heat pump evaporator and fan section. Schedule values must be entered as relative humidity fraction from 0 to 1 (e.g., a schedule value of 0.5 means 50%RH). This field is only required when the [Inlet air configuration](#) defined above is specified as **1-Schedule**.

Inlet air mixer schedule

If the **Inlet air configuration** is specified as **4-Zone and outdoor air** you must select the Inlet air mixer schedule which defines whether the heat pump draws its inlet air from the zone, outdoors, or a combination of zone and outdoor air. A schedule value equal to 0 indicates that the heat pump draws its inlet air from the zone. A schedule value equal to 1 denotes that the heat pump draws its inlet air from outdoors. Values between 0 and 1 denote a mixture of zone and outdoor air proportional to the schedule value. The **Inlet air mixer schedule** controls both the inlet air mixer and outlet air splitter nodes in unison to ensure that the operation of the heat pump does not contribute to zone pressurization or depressurization. For example if the Inlet air mixer schedule value is 0.4, then the inlet air mixer node is composed of 40% outdoor air and 60% zone air. For this same case, the outlet air splitter directs 60% of the ASHP outlet air back to the zone and 40% of the outlet air flow is exhausted outdoors.

Fan Settings

Fan placement

This Selection defines the placement of the fan in the heat pump water heater. Valid choices are:

- **1-Blow through** (fan upstream of the air coil) and
- **2-Draw through** (fan downstream of the air coil).

Parasitic Load Settings

On cycle parasitic electric load

This numeric field contains the on-cycle parasitic electric power (in W). This is the parasitic electric power consumed by controls or other electrical devices associated with the heat pump water heater. This parasitic electric load is consumed whenever the heat pump compressor is operating and the model assumes that this parasitic power does not contribute to heating the water. This parasitic load does, however, affect the zone air heat balance when the heat pump water heater sends some or all of its outlet air to a zone (i.e., **Inlet air configuration** field specified as **2-Zone air only**, or **4-Zone and outdoor air**) and the **Parasitic heat rejection location** field is specified as **1-Zone**. The minimum value for this field is 0.0, and the default value is also 0.0.

Off cycle parasitic electric load

This numeric field contains the off-cycle parasitic electric power (in W). This is the parasitic electric power consumed by controls or other electrical devices associated with the heat pump compressor. This parasitic electric load is consumed whenever the heat pump water heater is available but the compressor is not operating, and the model assumes that this parasitic power does not contribute to heating the water. This parasitic load does, however, affect the zone air heat balance when the heat pump water heater sends some or all of its outlet air to a zone (i.e., **Inlet air configuration** field specified as **2-Zone air only**, or **4-Zone and outdoor air**) and the **Parasitic heat rejection location** field is specified as **1-Zone**. The minimum value for this field is 0.0, and the default value is also 0.0.

Parasitic heat rejection location

Select where the on-cycle and off-cycle parasitic heat is rejected. Valid choices are:

- **1-Zone** where both the on-cycle and off-cycle parasitic heat is rejected to the zone. This option is only available for ASHP used as zone equipment in which case the heat is rejected to the zone containing the ASHP.
- **2-Exterior** where parasitic heat is rejected outdoors (does not impact the zone air heat balance)

Air To Water Heat Pump Coil

The Air To Water Heat Pump "Coil" described here models an air-to-water DX compression system (which includes a water heating coil, air coil, compressor, and water pump) to determine its air-side and water-side

performance. It is used within the composite EnergyPlus [air to water heat pump](#) which also includes a [water heater](#) tank and a [fan](#) to provide air flow across the air coil associated with the DX compression system. These objects work together to model a system which heats water using zone air, outdoor air, or a combination of zone and outdoor air as the primary heat source.

This DX coil calculates the air-side sensible and latent cooling capacity at the specific operating conditions for each simulation timestep, as well as the condenser's water-side temperature difference at a given condenser water flow rate. The heat pump water heater DX coil model performs the following major functions:

- Calculates the electric consumption of the DX compressor and water pump.
- Calculates the amount of heat delivered to the water tank.
- Calculates the electric consumption of the compressor's crankcase heater.
- Calculates the air-side performance of the DX coil.

The input fields for this object are described below in detail.

General

Name

Enter a unique name for the air to water heat pump coil.

Rated heating capacity

This numeric field defines the DX coil heating capacity (in W or Btu/h) at the rated evaporator inlet air temperatures, rated condenser inlet water temperature, rated evaporator air flow rate, and rated condenser water flow rate specified below. Values must be greater than 0. This value represents water heating capacity, and it may or may not include the impact of condenser pump heat (see field [Condenser pump heat included in rated heating capacity and rated COP](#) below).

Rated COP

This numeric field defines the DX coil's water heating coefficient of performance ($COP = \text{water heating capacity} / \text{electrical power input}$) at rated conditions (rated inlet temperatures and flow rates specified below). This input not only determines the electric energy use of the heat pump DX coil, but also the amount of total air cooling provided by the evaporator. The rated COP includes compressor power, and may or may not include condenser pump power or evaporator fan power (see field [Evaporator fan power included in rated COP](#) and field [Condenser pump power included in rated COP](#)). Values must be greater than 0.

Rated sensible heat ratio

This numeric field defines the air-side sensible heat ratio ($SHR = \text{sensible cooling capacity} / \text{total cooling capacity}$) of the DX coil at rated conditions (rated inlet temperatures and flow rates specified below). This value should not include the effect of evaporator fan heat. Values must be greater than or equal to 0.5, and less than or equal to 1.0. The default value is 0.85.

Evaporator Settings

Rated evaporator inlet air dry-bulb temperature

This numeric field defines the evaporator inlet air dry-bulb temperature (in °C or °F), that corresponds to rated coil performance (heating capacity, COP and SHR). Values must be greater than 5°C. The default value is 19.7°C.

Rated evaporator inlet air wet-bulb temperature

This numeric field defines the evaporator inlet air wet-bulb temperature (in °C or °F) that corresponds to rated coil performance (heating capacity, COP and SHR). Values must be greater than 5°C. The default value is 13.5°C.

Rated evaporator air flow rate

This numeric field defines the evaporator air volume flow rate (m³/s or ft³/min) at rated conditions. Values must be greater than 0. If this field is autocalculated (value = autocalculate), the default value is 5.035E-5 m³/s/W (31.25 cfm/MBH) multiplied by the [Rated heating capacity](#) specified above. When autocalculating the rated evaporator air volumetric flow rate, the zone sizing data is not used.

Evaporator fan power included in rated COP

This choice field specifies if evaporator fan power is included in the rated COP defined above. This input impacts the calculation of compressor electric power and total air cooling provided by the evaporator for each simulation timestep. If this option is selected, the evaporator fan power is subtracted from the total electric heating power when calculating total evaporator cooling capacity. Otherwise it is assumed that the total heating power does not include evaporator fan power. See the Engineering Reference section for Coil:WaterHeating:AirToWaterHeatPump for further details.

Condenser Settings

Rated condenser inlet water temperature

This numeric field defines the condenser inlet water temperature (in °C or °F) that corresponds to rated coil performance (heating capacity, COP and SHR). Values must be greater than 25°C. The default value is 57.5°C.

Rated condenser water flow rate

This numeric field defines the condenser water volumetric flow rate (in m³/s or gal/min) at rated conditions. Values must be greater than 0. If this field is left blank or autocalculated (value = autocalculate), the default value is 4.487E-8 m³/s/W (0.208 gpm/MBH) multiplied by the [Rated heating capacity](#) specified above. When autocalculating the rated condenser water volumetric flow rate, the data in the zone sizing object is not used. A warning message will be issued if the ratio of [Rated condenser water flow rate](#) to [Rated heating capacity](#) is less than 1.79405E-8 m³/s/W (0.083 gpm/MBH) or greater than 8.97024E-8 m³/s/W (0.417 gpm/MBH), but the simulation will continue.

Condenser pump power included in rated COP

This option specifies whether condenser pump power is included in the rated COP defined above. This input impacts the calculation of compressor electric power which then impacts the total air cooling provided by the evaporator for each simulation timestep. If it is selected, the condenser pump power is subtracted from the total electric heating power when calculating total evaporator cooling capacity. Otherwise it is assumed that the total heating power does not include the condenser pump. See Engineering Reference section for Coil:WaterHeating:AirToWaterHeatPump for further details.

Condenser pump heat included in rated heating capacity and rated COP

This option specifies whether condenser pump heat is included in the rated heating capacity and rated COP defined above. It impacts the calculation of compressor electric power and total air cooling provided by the evaporator for each simulation timestep. If it is selected, the condenser pump heat is already included in the rated heating capacity and rated COP. If it isn't selected, it is assumed that the rated heating capacity and rated COP do not include the condenser pump heat, and pump heat is added to the total water heating capacity based on the [Condenser water pump power](#) and [Fraction of condenser pump heat to water](#) fields below. See Engineering Reference section for Coil:WaterHeating:AirToWaterHeatPump for further details.

Condenser water pump power

This numeric field defines the DX coil's condenser pump power (in W or Btu/h). This is the operating pump power as installed. Values must be greater than or equal to 0. A warning message will be issued if the ratio of [Condenser water pump power](#) to [Rated heating capacity](#) exceeds 0.1422 W/W (41.67 Watts/MBH), but the simulation will continue.

Fraction of condenser pump heat to water

This numeric field defines the fraction of condenser pump heat that is transferred to the condenser water. The pump is assumed to be downstream of the condenser water coil, and this field is used to determine the water temperature at the condenser outlet node when the [Condenser pump power included in rated heating capacity and COP](#) option is unchecked. Values must be greater than or equal to 0 and less than or equal to 1. The default value is 0.2.

Crankcase Heater Settings

Crankcase heater capacity

This numeric field defines the compressor's crankcase heater capacity (in W or Btu/h). The crankcase heater only operates when the compressor is off and the air surrounding the compressor is below the **Maximum ambient temperature for crankcase heater operation** specified below.

Maximum ambient temperature for crankcase heater operation

This numeric field defines the maximum ambient temperature for crankcase heater operation (in °C or °F). The crankcase heater only operates when the air surrounding the compressor is below this maximum temperature value and the compressor is off. The ambient temperature surrounding the compressor is set by the Air to water heat pump parent object (**Compressor location**).

Performance Settings

You can use curves to modify the rated capacity and rated COP defined above based on one or more performance curves. To apply a curve first check its associated check box and then select the required curve.

Note: to use the rated capacity and rated COP without modification you should uncheck all of the check boxes in this section.

Evaporator air temperature type for curves

This drop list selection specifies the air temperature type used for the heating capacity and COP modifier curve objects below. The valid selections are:

- **1-Dry-bulb temperature** in which case the inlet air dry-bulb temperature entering the heat pump DX coil and fan section is used to evaluate the curve objects.
- **2-Wet-bulb temperature** where the inlet air wet-bulb temperature entering the heat pump DX coil and fan section is used to evaluate the curve objects.

If the curves are not defined then this field is not used.

Heating capacity function of temperature curve

This [Bi-quadratic](#) or [Cubic](#) performance curve defines the variation in DX coil heating capacity as a function of inlet fluid (air and water) temperatures. The Bi-quadratic curve uses evaporator inlet air temperature (dry-bulb or wet-bulb temperature based on the field [Evaporator air temperature type for curves](#) defined above) and condenser inlet water temperature as the independent variables. The cubic curve uses evaporator inlet air (dry-bulb or wet-bulb) temperature as the independent variable. The output of this curve is multiplied by the rated heating capacity to give the heating capacity at specific operating conditions (i.e., at temperatures different from the rating point temperatures). The curve should be normalized to have the value of 1.0 at the rating point temperatures.

If the associated checkbox is unchecked, the heating capacity remains constant (curve value assumed to be 1.0 for all conditions).

Heating capacity function of air flow fraction curve

This [Quadratic](#) or [Cubic](#) performance curve defines the variation in DX coil heating capacity as a function of the ratio of actual air flow rate across the evaporator coil to the rated evaporator air flow rate. The output of this curve is multiplied by the Rated heating capacity and the Heating capacity modifier curve (function of temperature) to give the DX coil heating capacity at the specific inlet fluid temperatures and air flow rate at which the coil is operating. The curve should be normalized to have the value of 1.0 at the rated evaporator air flow rate (air flow fraction of 1.0).

If the associated checkbox is unchecked the heating capacity remains constant (curve value assumed to be 1.0 for all air flow rates).

Heating capacity function of water flow fraction curve

This [Quadratic](#) or [Cubic](#) performance curve defines the variation in DX coil heating capacity as a function of the ratio of actual water flow rate through the condenser to the rated condenser water flow rate. The output of this curve is multiplied by the rated heating capacity and the output from the two other heating capacity modifier curves (function of temperature and function of air flow fraction) to give the DX coil heating capacity at the specific inlet fluid temperatures and flow rates at which the coil is operating. The curve should be normalized to have the value of 1.0 at the rated condenser water flow rate (water flow fraction of 1.0).

If the associated checkbox is unchecked the heating capacity remains constant (curve value assumed to be 1.0 for all water flow rates).

Heating COP function of temperature curve

This [Bi-quadratic](#) or [Cubic](#) performance curve defines the variation in DX coil heating COP as a function of inlet fluid (air and water) temperatures. The biquadratic curve uses evaporator inlet air temperature (dry-bulb or wet-bulb temperature based on the field [Evaporator air temperature type for curves](#) defined above) and condenser inlet water temperature as the independent variables. The cubic curve uses evaporator inlet air (dry-bulb or wet-bulb) temperature as the independent variable. The output of this curve is multiplied by the rated COP to give the heating COP at specific operating conditions (i.e., at temperatures different from the rating point temperatures). The curve should be normalized to have the value of 1.0 at the rating point temperatures.

If the associated checkbox is unchecked the COP remains constant (curve value assumed to be 1.0 for all conditions).

Heating COP function of air flow fraction curve

This [Quadratic](#) or [Cubic](#) performance curve defines the variation in DX coil heating COP as a function of the ratio of actual air flow rate across the evaporator coil to the rated evaporator air flow rate. The output of this curve is multiplied by the rated COP and the heating COP modifier curve (function of temperature) to give the heating COP at the specific inlet fluid temperatures and air flow rate at which the coil is operating. The curve should be normalized to have the value of 1.0 at the rated evaporator air flow rate (air flow fraction of 1.0).

If the associated checkbox is unchecked the heating COP remains constant (curve value assumed to be 1.0 for all air flow rates).

Heating COP function of water flow fraction curve

This [Quadratic](#) or [Cubic](#) performance curve defines the variation in DX coil heating COP as a function of the ratio of actual water flow rate through the condenser to the rated condenser water flow rate. The output of this curve is multiplied by the rated COP and the output from the two other heating COP modifier curves (function of temperature and function of air flow fraction) to give the DX coil heating COP at the specific inlet fluid temperatures and flow rates at which the coil is operating. The curve should be normalized to have the value of 1.0 at the rated condenser water flow rate (water flow fraction of 1.0).

If the associated checkbox is unchecked the heating COP remains constant (curve value assumed to be 1.0 for all water flow rates).

Part load fraction correlation curve

This [Quadratic](#) or [Cubic](#) performance curve parameterizes the variation of electrical power input to the DX unit as a function of the part load ratio (PLR, sensible cooling load/steady-state sensible cooling capacity). The product of the rated EIR and EIR modifier curves is divided by the output of this curve to give the “effective” EIR for a given simulation timestep. The part load fraction (PLF) correlation accounts for efficiency losses due to compressor cycling.

The part load fraction correlation should be normalized to a value of 1.0 when the part load ratio equals 1.0 (i.e., no efficiency losses when the compressor(s) run continuously for the simulation timestep). For PLR values between 0 and 1 ($0 \leq \text{PLR} < 1$), the following rules apply:

$\text{PLF} \geq 0.7$ and $\text{PLF} \geq \text{PLR}$

If $\text{PLF} < 0.7$ a warning message is issued, the program resets the PLF value to 0.7, and the simulation proceeds. The runtime fraction of the coil is defined as PLR/PLF . If $\text{PLF} < \text{PLR}$, then a warning message is issued and the runtime fraction of the coil is limited to 1.0.

A typical part load fraction correlation for a conventional, single-speed DX cooling coil (e.g., residential unit) would be:

$$\text{PLF} = 0.85 + 0.15(\text{PLR})$$

If the user wishes to model no efficiency degradation due to compressor cycling, the part load fraction correlation should be defined as follows:

$$\text{PLF} = 1.0 + 0.0(\text{PLR})$$

Air To Water Heat Pump Example

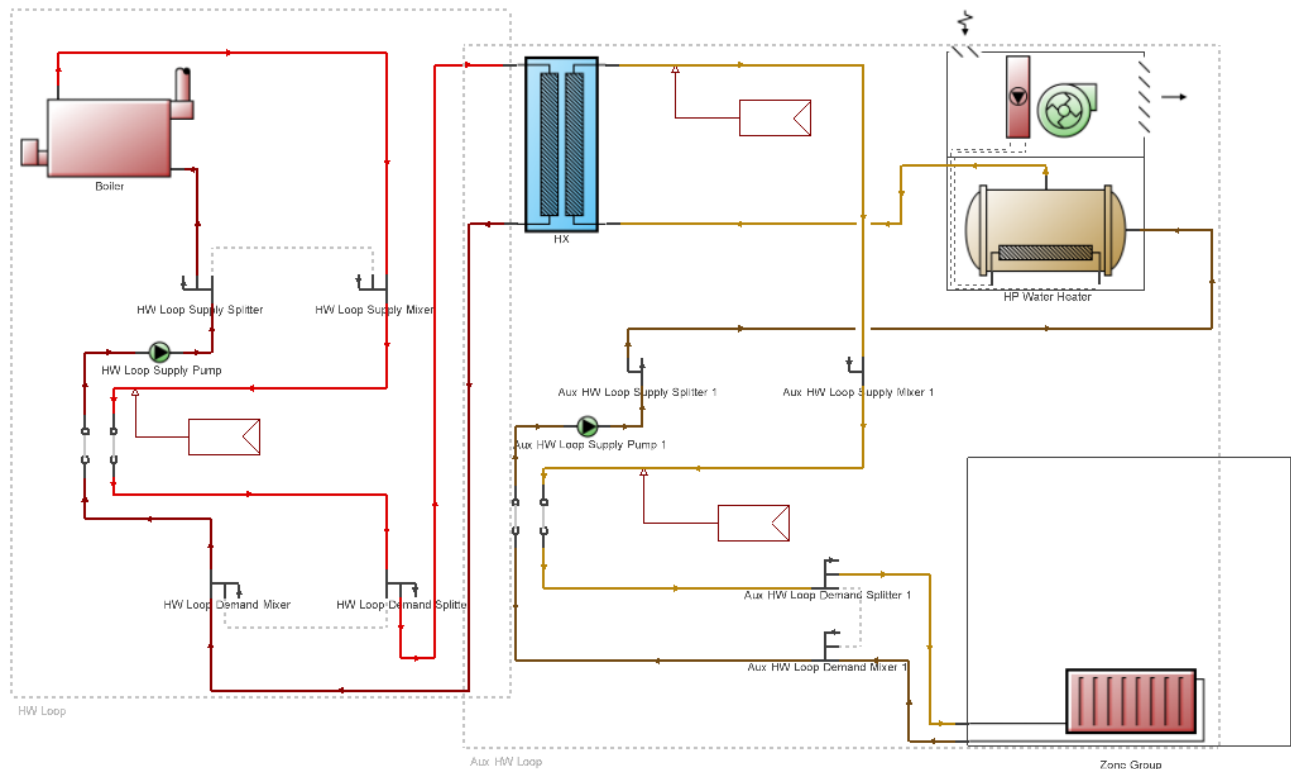
This example explains the main steps involved in setting up an ASHP system. It involves loading an ASHP Detailed HVAC template and modifying some settings to achieve a high-performance low temperature ASHP system with gas-fired boiler supplementary heating.

The steps are as follows.

1. Create a new file located in London Gatwick and add a building to the site with a simple square block having dimensions 10m x 10m. Use default Model options and template settings.

Note: Although this example is based on a simple single zone building model, you could follow the same steps on your own multi-zone model.

2. Set the [Detailed HVAC](#) model option.
3. Load the [ASHP Air-to-water Heat Pump, Supplementary Boiler, Water Convactor](#) Detailed HVAC template to the occupied zone(s).



This provides an air to water heat pump with gas boiler supplementary heater which provides high temperature water.

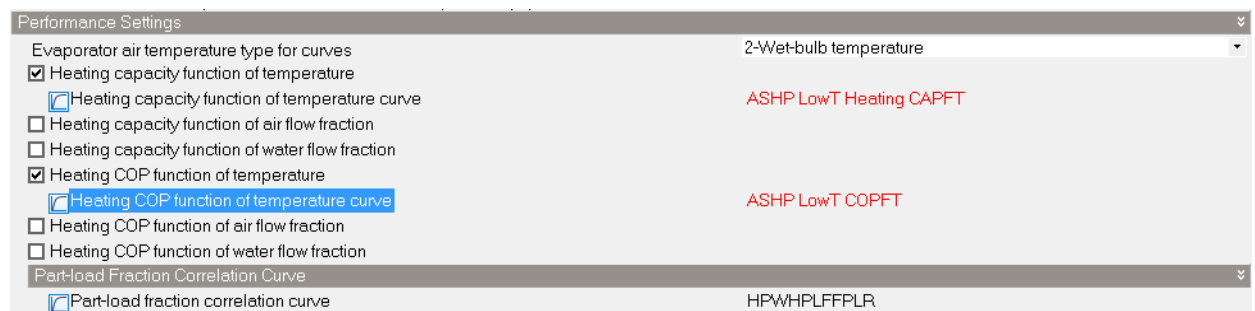
4. Run the [Heating design calculations](#) to obtain the total heating [Design Capacity](#) for the building and hence the required heating capacity of the ASHP system.

5. Edit the [Air to water heat pump coil](#)

5.1. Change the default high temperature model loaded from the template to use low temperature outlet from the HP. To do this, change the heating capacity and COP function of temperature performance curve settings as follows:

- Change [Heating capacity function of temperature curve](#) to **ASHP LowT COPFT**, and
- Change [Heating COP function of temperature curve](#) to **ASHP LowT Heating CAPFT**.

The curves section of the dialog should now be similar to the screenshot below.



5.2. Enter the [Rated heating capacity](#) of the ASHP. Enter 90% of the design heating load calculated in step 4. The system will rely on the gas boiler to boost overall system capacity to achieve the full capacity.

5.3. To ensure that the rated conditions are consistent with the new curves selected in step 5.1, you should change the [Rated evaporator inlet air dry and wet-bulb temperatures](#) to 16°C and 12°C respectively. See below.

Evaporator Settings	
Rated evaporator inlet air dry-bulb temperature (°C)	16.00
Rated evaporator inlet air wet-bulb temperature (°C)	12.00
Rated evaporator air flow rate (m3/s)	Autocalculate

6. Next move up a level in the HVAC system hierarchy and edit the [Air to water heat pump](#) (Water heater heat pump) and:

- Create and select a new schedule to set the water outlet temperature setpoint to be 58°C. To do that duplicate **LowT heat pump water supply temperature: Always 55.00** to create **LowT heat pump water supply temperature: Always 58.00**. Apply the new schedule to the [Compressor setpoint temperature schedule](#) field.
- Ensure that the [Minimum inlet air temperature for compressor operation](#) is set to -10°C to allow operation on cold days but prevent operation on days colder than -10°C.
- Change the [Deadband temperature difference](#) to 2°C.

7. Edit the [Water heater](#) and verify some key inputs:

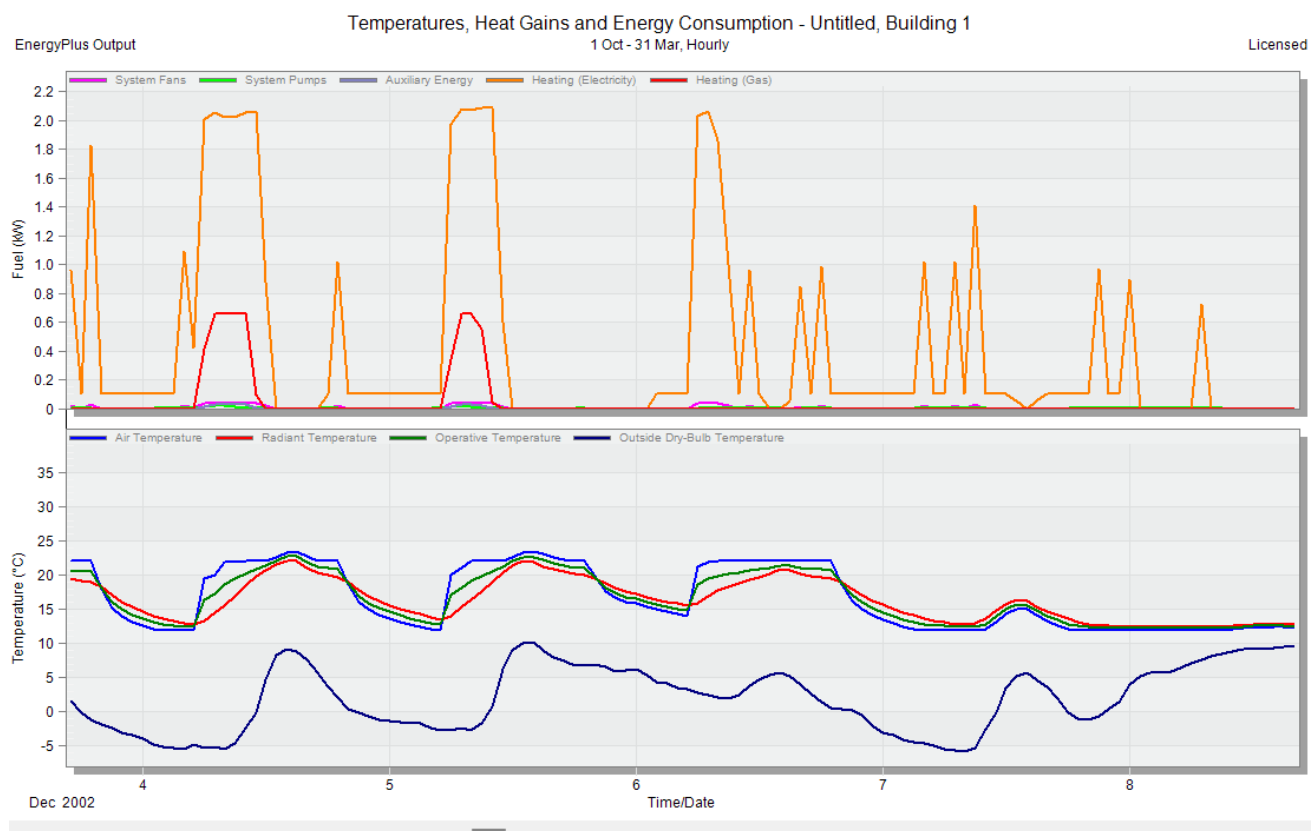
- On the **Water heater** tab, ensure that the [Tank volume](#) is set to **Autosize**.
- On the **Sizing** tab, check that [Design mode](#) is set to **1-Peak draw** and the [Time for tank recovery](#) is set to 0.6hr.

9. Edit the [Boiler](#) and change the capacity of the boiler to provide the remaining 10% of the design heating load.

10. Check that the [SPM](#) downstream of the supply side of the HX controls the supply water temperature to 55°C.

11. Change the Pressure rise of the [ASHP fan](#) to 100Pa.

12. Finally, run a simulation of the model over the winter to ensure that the overall system is able to meet the heating loads even on the coldest days without excessive use of the supplementary gas-fired boiler. You should see results similar to those shown below.



Note how the gas boiler only needs to fire on the coldest periods.

District Heating



Used in:

- Hot water loop, supply side

The district heating component is used to provide heat to the supply side of a hot water loop. It can be used to model these systems:

- a. Centralised sources of hot water (district heating systems), or,
- b. Cases where it is not necessary to simulate the plant., e.g. where the focus of the study is on the air side of the system or the building zones.

General

Name

The auto-generated name of the district cooling system can be edited.

Nominal capacity

This numeric field contains the nominal demand (in W or Btu/h) that the district heating will meet. This field is autosizable.

Capacity fraction schedule

Select the schedule that describes how the nominal capacity varies over time. Values must non-negative. The capacity at a given point in time is determined by the product of the **Nominal capacity** (above) and the value in this schedule. The default schedule is **On 24/7** i.e. a value of 1 all the time.

Performance Curves

Curve:Linear

Curve:QuadLinear Used in many HVAC components as well as electrical storage devices

Curve:QuintLinear

Curve:Quadratic

Curve:Cubic

Curve:Quartic

Curve:Exponent

Curve:Bicubic

Curve:Biquadratic

Curve:RectangularH

yperbola2

Curve:DoubleExpon

entialDecay

In EnergyPlus HVAC, performance curves are the mathematical mechanism that adjusts rated equipment performance (capacity, efficiency, airflow effects, etc.) as operating conditions change based on actual operating conditions. They are needed because most HVAC equipment does not operate at its rated capacity or efficiency except at laboratory reference conditions. In more detail, performance curves:

- Modify capacity, efficiency and other equipment characteristics based on temperatures, flow rates etc.
- Capture part-load performance and cycling losses.
- Apply manufacturer catalogue data to allow real equipment to be modelled.
- Allow generalised HVAC component models to behave like specific real equipment.

DesignBuilder performance curves consist of a range of different polynomial curve types that are used to characterise the performance of HVAC equipment. Several other non-polynomial curves are also included to characterize the performance of pumps and fans. The curves are usually derived from fits or regressions to data covering a limited range. Results for independent variable values outside this range are likely to be invalid, so curve input always contains a range of validity (maximum and minimum permitted values) for each independent variable and can optionally have limits on the curve output. No error or warning message is issued if an independent variable is outside the range. Instead, EnergyPlus uses the minimum value if an independent variable is less than the minimum, and the maximum if a variable exceeds the maximum. Similarly, no error or warning message is issued if the curve output is outside the range of the optional minimum and maximum curve output limits. Instead, EnergyPlus uses the minimum and maximum curve limits to cap the output of the performance curve.

Tip: For guidance on how to generate the data required for Performance curves see: "Performance Curve Coefficient Tools" on page 914

General Tab

On the **General** tab you can set the curve type and enter the curve coefficients.

The curve data follow a similar pattern and all have the same first 4 data items:

Name

A user assigned unique name for a curve. When a curve is used, it is referenced by this name.

Note

Specific information about the curve.

Source

The source setting defines the origin of the curve. For most library curves provided with the software, the source is usually either "EnergyPlus", meaning it comes from the EnergyPlus curves database, which is largely derived from the DOE-2 database, or "DesignBuilder" meaning that the curve is provided by DesignBuilder Software.

Category

The category can be one of:

- [Linear](#)
- [Quad Linear](#)
- [Quint Linear](#)
- [Quadratic](#)
- [Cubic](#)
- [Exponent](#)
- [Bi-Quadratic](#)
- [Bi-Cubic](#)
- [Quartic](#)
- [Rectangular Hyperbola 2](#)
- [Double Exponential Decay](#)

The rest of the data depends on the curve **Category** setting, as described below.

Linear

Input for the linear curve consists of a curve name, the two coefficients, and the maximum and minimum valid independent variable values. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g. limit extrapolation). The equation represented by the linear curve is:

$$y = C_1 + C_2 * x$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 x

The linear coefficient (C_2) in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select Dimensionless which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Quad Linear

Input consists of the curve name, the five coefficients, and maximum and minimum values for each of the independent variables. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g., limit extrapolation). The equation is represented by this curve:

$$y = C_1 + C_2 * w + C_3 * x + C_4 * y + C_5 * z$$

Name

A user assigned unique name for an instance of this curve. When a curve of this type is used, it is referenced by this name.

Coefficient1 Constant

The constant coefficient (C_1) in the equation.

Coefficient2 w

The coefficient (C_2) in the equation.

Coefficient3 x

The coefficient (C_3) in the equation.

Coefficient4 y

The coefficient (C_4) in the equation.

Coefficient5 z

The coefficient (C_5) in the equation.

MinimumW

The minimum allowable value of w. Values of w less than the minimum will be replaced by the minimum.

MaximumW

The maximum allowable value of w. Values of w greater than the maximum will be replaced by the maximum.

MinimumX

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

MaximumX

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

MinimumY

The minimum allowable value of y. Values of y less than the minimum will be replaced by the minimum.

MaximumY

The maximum allowable value of y. Values of y greater than the maximum will be replaced by the maximum.

MinimumZ

The minimum allowable value of z. Values of z less than the minimum will be replaced by the minimum.

MaximumZ

The maximum allowable value of z. Values of z greater than the maximum will be replaced by the maximum.

Input unit type for W

This field is used to indicate the kind of units that may be associated with the w values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of w** and **Maximum value of w**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for X

This field is used to indicate the kind of units that may be associated with the x values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for Y

This field is used to indicate the kind of units that may be associated with the y values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of y** and **Maximum value of y**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for Z

This field is used to indicate the kind of units that may be associated with the z values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of z** and **Maximum value of z**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Quint Linear

Input consists of the curve name, the six coefficients, and maximum and minimum values for each of the independent variables. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g., limit extrapolation). The equation is represented by this curve:

$$y = C_1 + C_2 * v + C_3 * w + C_4 * x + C_5 * y + C_6 * z$$

Name

A user assigned unique name for an instance of this curve. When a curve of this type is used, it is referenced by this name.

Coefficient1 Constant

The constant coefficient (C_1) in the equation.

Coefficient2 v

The coefficient (C_2) in the equation.

Coefficient3 w

The coefficient (C_3) in the equation.

Coefficient4 x

The coefficient (C_4) in the equation.

Coefficient5 y

The coefficient (C_5) in the equation.

Coefficient6 z

The coefficient (C_6) in the equation.

MinimumV

The minimum allowable value of v. Values of v less than the minimum will be replaced by the minimum.

MaximumV

The maximum allowable value of v. Values of v greater than the maximum will be replaced by the maximum.

MinimumW

The minimum allowable value of w. Values of w less than the minimum will be replaced by the minimum.

MaximumW

The maximum allowable value of w. Values of w greater than the maximum will be replaced by the maximum.

MinimumX

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

MaximumX

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

MinimumY

The minimum allowable value of y. Values of y less than the minimum will be replaced by the minimum.

MaximumY

The maximum allowable value of y. Values of y greater than the maximum will be replaced by the maximum.

MinimumZ

The minimum allowable value of z. Values of z less than the minimum will be replaced by the minimum.

MaximumZ

The maximum allowable value of z. Values of z greater than the maximum will be replaced by the maximum.

Input unit type for V

This field is used to indicate the kind of units that may be associated with the v values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of v** and **Maximum value of v**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for W

This field is used to indicate the kind of units that may be associated with the w values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of w** and **Maximum value of w**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for X

This field is used to indicate the kind of units that may be associated with the x values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**

- **5-Distance**
- **6-Power**

Input unit type for Y

This field is used to indicate the kind of units that may be associated with the y values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of y** and **Maximum value of y**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input Unit Type for Z

This field is used to indicate the kind of units that may be associated with the z values. It is used by the editor to display the appropriate SI and IP units for the **Minimum value of z** and **Maximum value of z**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Quadratic

Input for a quadratic curve consists of the curve name, the three coefficients, and the maximum and minimum valid independent variable values. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g. limit extrapolation). The equation represented by the quadratic curve is:

$$y = C_1 + C_2 * x + C_3 * x^2$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 x

The linear coefficient (C_2) in the equation.

Coefficient 3 x²

The quadratic coefficient (C_3) in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Cubic

Input for a cubic curve consists of the curve name, the 4 coefficients, and the maximum and minimum valid independent variable values. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g. limit extrapolation). The equation represented by the cubic curve is:

$$y = C_1 + C_2 * x + C_3 * x^2 + C_4 * x^3$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 x

The linear coefficient (C_2) in the equation.

Coefficient 3 x²

The quadratic coefficient (C_3) in the equation.

Coefficient 4 x³

The quadratic coefficient (C_4) in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Exponent

Input for a exponent curve consists of the curve name, the 3 coefficients, and the maximum and minimum valid independent variable values. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g. limit extrapolation). The equation represented by the exponent curve is:

$$y = C_1 + C_2 * x^{C_3}$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 constant

The linear coefficient (C_2) in the equation.

Coefficient 3 constant

The exponent coefficient (C_3) in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Bi-quadratic

This curve is a function of two independent variables. Input consists of the curve name, the six coefficients, and min and max values for each of the independent variables. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g. limit extrapolation). The equation represented by the biquadratic curve is:

$$z = C_1 + C_2 * x + C_3 * x^2 + C_4 * y + C_5 * y^2 + C_6 * xy$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 x

The linear coefficient (C_2) in the equation.

Coefficient 3 x^2

The quadratic coefficient (C_3) in the equation.

Coefficient 4 y

The coefficient C_4 in the equation.

Coefficient 5 y^2

The coefficient C_5 in the equation.

Coefficient 6 xy

The coefficient C_6 in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum value of y

The minimum allowable value of y. Values of y less than the minimum will be replaced by the minimum.

Maximum value of y

The maximum allowable value of y. Values of y greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of x** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for y

This field is used to indicate the kind of units that may be associated with the y values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of y** and **Maximum value of y**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Bi-cubic

This curve type is a function of two independent variables. Input consists of the curve name, the ten coefficients, and the minimum and maximum values for each of the independent variables. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g. limit extrapolation). The equation represented by the bicubic curve is:

$$z = C_1 + C_2 * x + C_3 * x^2 + C_4 * y + C_5 * y^2 + C_6 * xy + C_7 * x^3 + C_8 * y^3 + C_9 * x^2y + C_{10} * xy^2$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 x

The linear coefficient (C_2) in the equation.

Coefficient 3 x^2

The quadratic coefficient (C_3) in the equation.

Coefficient 4 y

The coefficient C_4 in the equation.

Coefficient 5 y^2

The coefficient C_5 in the equation.

Coefficient 6 xy

The coefficient C_6 in the equation.

Coefficient 7 x^3

The coefficient C_7 in the equation.

Coefficient 8 y^3

The coefficient C_8 in the equation.

Coefficient 9 x^2y

The coefficient C_9 in the equation.

Coefficient 10 xy^2

The coefficient C_{10} in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum value of y

The minimum allowable value of y. Values of y less than the minimum will be replaced by the minimum.

Maximum value of y

The maximum allowable value of y. Values of y greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by editor to display the appropriate SI and IP units for the **Minimum value of X** and **Maximum value of x**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Input unit type for y

This field is used to indicate the kind of units that may be associated with the y values. It is used by editor to display the appropriate SI and IP units for the Minimum Value of y and Maximum Value of y. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**

- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Quartic

Input for a Quartic (fourth order polynomial) curve consists of the curve name, the five coefficients, and the maximum and minimum valid independent variable values. Optional inputs for curve minimum and maximum may be used to limit the output of the performance curve (e.g., limit extrapolation). The equation represented by the quartic curve is:

$$y = C_1 + C_2 * x + C_3 * x^2 + C_4 * x^3 + C_5 * x^4$$

Coefficient 1 constant

The constant coefficient (C_1) in the equation.

Coefficient 2 x

The linear coefficient (C_2) in the equation.

Coefficient 3 x^2

The quadratic coefficient (C_3) in the equation.

Coefficient 4 x^3

The cubic coefficient (C_4) in the equation.

Coefficient 5 x^4

The fourth-order coefficient (C_5) in the equation.

Minimum value of x

The minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum curve output

The minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum value of x

The maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This field is used to indicate the kind of units that may be associated with the x values. It is used by IDF Editor to display the appropriate SI and IP units for the Minimum Value of X and Maximum Value of x. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select Dimensionless which will have no unit conversion.

- **1-Dimensionless**
- **2-Temperature**
- **3-VolumetricFlow**
- **4-MassFlow**
- **5-Distance**
- **6-Power**

Output unit type

This field is used to indicate the kind of units that may be associated with the output values. It is used by the editor to display the appropriate SI and IP units for the **Minimum curve output** and **Maximum curve output**. The unit conversion is not applied to the coefficients. The available options are shown below. If none of these options are appropriate, select **1-Dimensionless** which will have no unit conversion.

- **1-Dimensionless**
- **2-Capacity**
- **3-Power**

Rectangular Hyperbola 2

Input for the single rectangular hyperbola type 2 curve consists of the curve name, the three coefficients, and the maximum and minimum valid independent variable values. Optional inputs for the curve minimum and maximum may be used to limit the output of the performance curve (e.g., limit extrapolation). The equation is:

$$y = (C_1 * x) / (C_2 + x) + C_3 * x$$

Coefficient 1 C_1

The required numeric constant coefficient C_1 in the equation.

Coefficient 2 C_2

The required numeric constant coefficient C_2 in the equation.

Coefficient 3 C_3

The required numeric constant coefficient C_3 in the equation.

Minimum value of x

The required numeric minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum.

Maximum value of x

The required numeric maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum.

Minimum curve output

The optional numeric minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum.

Maximum curve output

The optional numeric maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

This optional field is provided for future purposes so that DesignBuilder could display the appropriate SI or IP units for the **Minimum value of x** and **Maximum value of x** (currently dimensionless). At this time, only the Dimensionless option is provided so that no unit conversion is used for this curve.

Output unit type

This optional field is provided for future purposes so that DesignBuilder could display the appropriate SI or IP units for the **Minimum curve output** and **Maximum curve output** (currently dimensionless). At this time, only the **1-Dimensionless** option is provided so that no unit conversion is used for this curve.

Double Exponential Decay

Input for the double exponential decay curve consists of the curve name, the five coefficients, and the maximum and minimum valid independent variable values. Optional inputs for the curve include the minimum and maximum output of the performance curve. The equation is:

$$y = C_1 + C_2 \exp(C_3 x) + C_4 \exp(C_5 x)$$

Coefficient 1 C_1

The required numeric constant coefficient C_1 in the equation.

Coefficient 2 C_2

The required numeric constant coefficient C_2 in the equation.

Coefficient 3 C_3

The required numeric constant coefficient C_3 in the equation.

Coefficient 4 C_4

The required numeric constant coefficient C_4 in the equation.

Coefficient 5 C_5

The required numeric constant coefficient C_5 in the equation.

Minimum value of x

The required numeric minimum allowable value of x. Values of x less than the minimum will be replaced by the minimum

Maximum value of x

The required numeric maximum allowable value of x. Values of x greater than the maximum will be replaced by the maximum

Minimum curve output

The optional numeric minimum allowable value of the evaluated curve. Values less than the minimum will be replaced by the minimum

Maximum curve output

The optional numeric maximum allowable value of the evaluated curve. Values greater than the maximum will be replaced by the maximum.

Input unit type for x

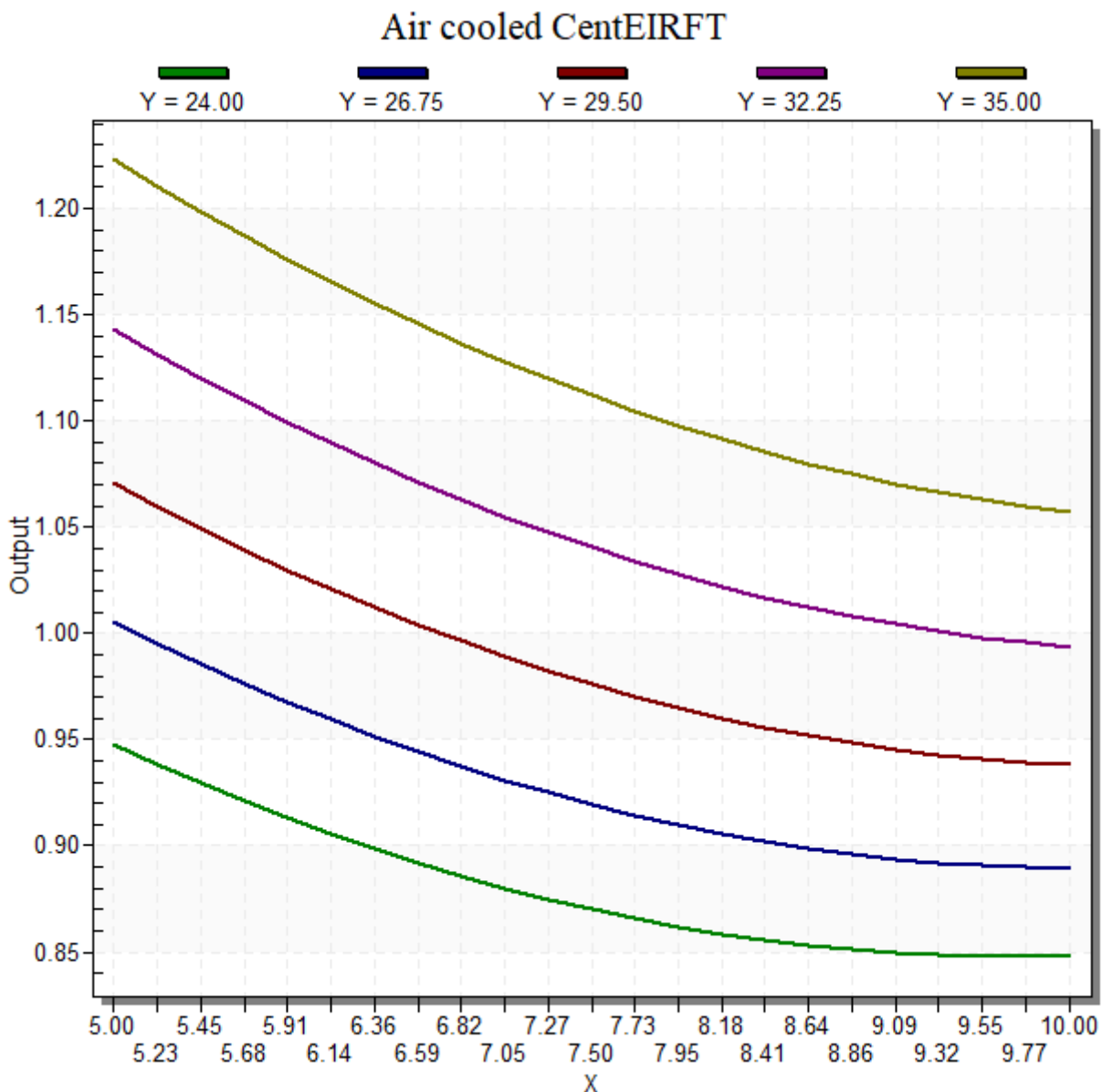
The optional field is provided for future purposes so that DesignBuilder could display the appropriate SI or IP units for the **Minimum** and **Maximum value of x**. Currently, only Dimensionless option is provided so that no unit conversion is used for this curve

Output unit type

The optional field is provided for future purposes so that DesignBuilder could display the appropriate SI or IP units for the **Minimum** and **Maximum curve output**. Currently, only the **1-Dimensionless** option is provided so that no unit conversion is used for this curve.

Curve Plot Tab

Click on the **Curve Plot** tab to view the performance curve data in graphical form.



Performance Curve Coefficient Tools

A range of tools are available to help set up performance curves in a form that can be used in DesignBuilder EnergyPlus simulations based on equipment manufacturers' data sheets:

- **DesignBuilder Curve Generator Tool:** Accessed from the Info panel of the Curves dialog.
- **VRF Curve Generation Utilities:** Specialised spreadsheet tools for generating EnergyPlus curves for accurate variable refrigerant flow (VRF) system simulation.
- **EnergyPlus Curve Fit Tool:** Used to generate HVAC performance curve coefficients for any HVAC equipment modelled with Biquadratic, Cubic, or Quadratic curves using manufacturer data sheets.
- **Chiller Performance Curve Coefficients Tool:** Used to generate performance curve coefficients for chillers and any other HVAC equipment modelled with Biquadratic, Cubic, or Quadratic curves using manufacturer data sheets.
- **Ice Storage Curve Fit Tool:** Assists with generating performance curve coefficients for Ice Thermal Storage charge and discharge cycles.

All of the above are freely available utilities for generating performance curve coefficients used in HVAC equipment modelling within DesignBuilder/EnergyPlus. They enable users to input performance data at various operating conditions and automatically generate the polynomial coefficients needed to create the curves for accurate equipment simulation. They are commonly used for chillers, heat pumps, DX coils, fans, pumps, and other HVAC equipment.

Notes on the "Chiller Performance Curve Coefficients" and the "EnergyPlus Curve Fit" Tools

Both these tools provided by EnergyPlus generate the same curve types (Biquadratic, Cubic, and Quadratic) using identical mathematical principles. Each tool is optimised for its primary workflow, but both offer flexibility for adaptation. Neither tool is strictly limited to a single equipment type or medium (e.g., chillers, heat pumps, air-source vs water-source systems, or water pumps).

Both tools are flexible enough to allow you to adapt the independent variable inputs to match your equipment's actual operating conditions. For example, you can adjust the labels and input ranges to represent water temperatures rather than air temperatures, making them suitable for water-to-water systems. See unmet hours article: <https://unmethours.com/question/2059/air-source-heat-pump-curves>. You can also generate the cubic curves required for water pump part-load performance modelling.

Fundamentally, the labels are simply examples tied to the tools' original design intent, while the mathematical curve fitting capability is what makes them generally applicable. Once this principle is clear, you can use either tool (or any other available) for any application requiring Biquadratic, Cubic, or Quadratic curve fitting, regardless of the specific equipment type or operating medium.

For more information on all performance curve types, see: [Performance Curve Data](#).

How DesignBuilder Plots Performance Curves

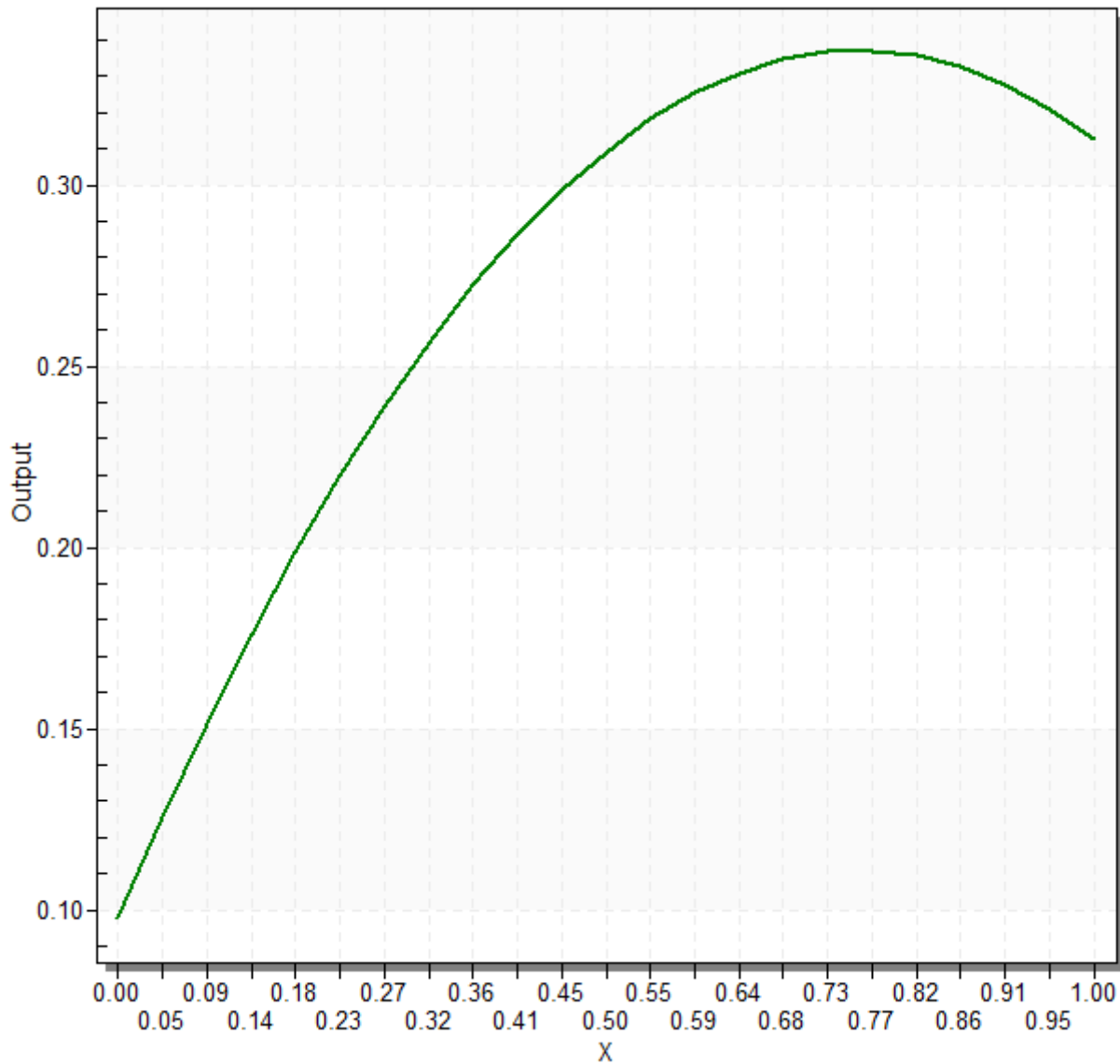
From the Curves dialog, you can view the performance data as graphical curves on the [Curve Plot](#) tab of the dialog.

When DesignBuilder plots a performance curve, the axes and legend representation depends on whether the curve is single-variable or multi-variable:

Single-variable curves (e.g., Linear, Quadratic, Cubic, Quartic):

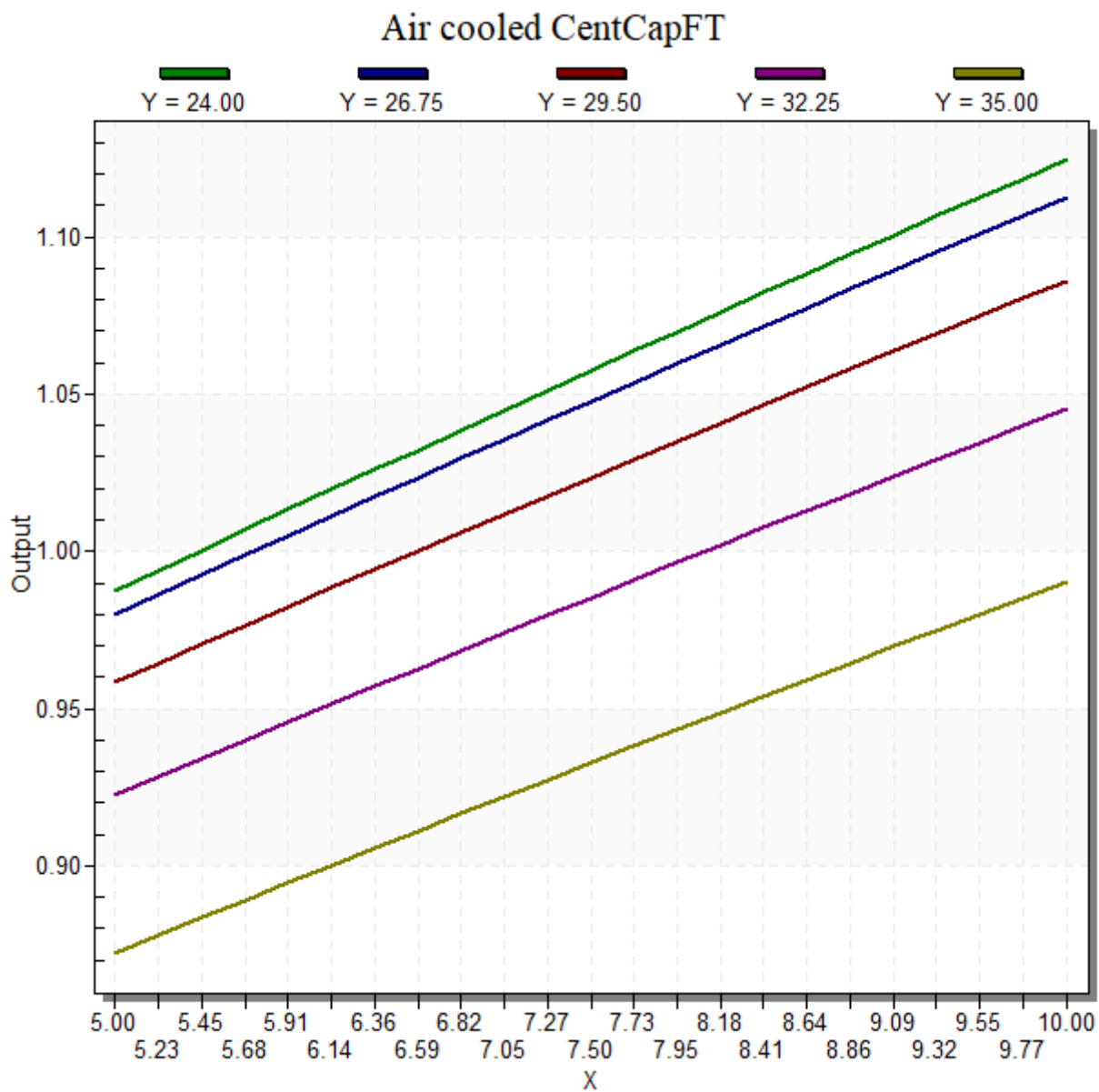
- Horizontal axis (X): The independent variable (e.g., part-load ratio, flow fraction, temperature)
- Vertical axis (Output): The curve's output value (e.g., power modifier, efficiency modifier)

BG Shaft Power Curve



Multi-variable curves (e.g., Biquadratic, Bicubic):

- Horizontal axis (X): The first independent variable (e.g., leaving chilled water temperature)
- Legend entries (Y, colored lines): The second independent variable held constant for each series (e.g., outdoor/source temperature at different fixed values)
- Vertical axis (Output): The curve's output value (e.g., EIR modifier, capacity modifier)



The variable assignment is determined by the order in which you enter your independent variables in the Performance Curve Coefficients tool input table.

For example, when using the **DesignBuilder Curve Generator**:

- Column x (first variable) → Horizontal axis (X)
- Column y (second variable) → Legend (Y)

The screenshot shows the "Curve Generator Tool for DesignBuilder" window. It features two main tables at the top. The left table has columns for x, y, Value, and Adjusted Value. The right table has columns for Coefficient and Value. Below these tables are several control elements: buttons for "Save Curve File", "Clear Table", "Open Curve File", and "Paste from SpreadSheet"; a checkbox labeled "Remove worst outlier data point"; a dropdown menu currently set to "BiQuadratic" which displays the equation $z = c_1 + c_2x + c_3x^2 + c_4y + c_5y^2 + c_6xy$; buttons for "About", "Calculate", and "Copy to DB"; a "Normalize by:" label followed by a numeric input field containing "1"; and a "Copy Clipboard" button. The window title bar includes standard minimize, maximize, and close icons.

Note: Changing the minimum and maximum ranges of X and Y in the Edit Curve will not change which variable is assigned to the horizontal axis or the legend. This only changes the range over which those variables are evaluated.

Edit Curve - Air cooled CentCapFT

Curves

General Curve Plot

General

Name Air cooled CentCapFT

Note

Source EnergyPlus

Category Bi-Quadratic

Coefficients

Coefficient 1	0.25789600000000000000
Coefficient 2	0.03890160000000000000
Coefficient 3	-0.00021708000000000000
Coefficient 4	0.04686840000000000000
Coefficient 5	-0.00094284000000000000
Coefficient 6	-0.00034344000000000000

Range

MinimumX	5.0000000000
MaximumX	10.0000000000
MinimumY	24.0000000000
MaximumY	35.0000000000
Input unit type for X	
Input unit type for Y	

Example: Chiller EIR Modifier For Temperature Curve (Biquadratic)

For a typical chiller Energy Input Ratio (EIR) modifier curve as a function of two temperatures, consider an example where you have performance data at various leaving chilled water temperatures and entering condenser water temperatures. This section demonstrates both how the plot appears in DesignBuilder and

how the coefficient generation is validated across the **Curve Generator** and **Chiller Performance Curve Coefficients** tools.

When modelling a chiller's EIR performance, you need to capture how its efficiency changes with:

- First independent variable (x): Supply/Leaving water temperature (CWS)
- Second independent variable (y): Entering condenser water temperature (ECT)
- Value (z) = EIR = 1/COP

EIR modifier as a function of temperature: $z = C_1 + C_2 \cdot x + C_3 \cdot x^2 + C_4 \cdot y + C_5 \cdot y^2 + C_6 \cdot x \cdot y$

You collect performance data at various combinations of these temperatures and enter them into the performance curve tool (e.g., Curve Generator or Chiller_PerformanceCurve_Coefficients) with their corresponding normalised EIR values (relative to rated conditions).

Capacity rated (kW)	Capacity (kW)	CWS (°C)	ECT (°C)	Power Input (kW)	EIR
78.6	83.8	5	25	24.1	0.287589499
78.6	79.1	5	30	26.2	0.331226296
78.6	74.1	5	35	28.5	0.384615385
78.6	68.7	5	40	31.2	0.454148472
78.6	62.8	5	45	32.2	0.512738854
78.6	88.8	7	25	24.6	0.277027027
78.6	83.9	7	30	26.7	0.318235995
78.6	78.6	7	35	29	0.368956743
78.6	72.8	7	40	31.6	0.434065934
78.6	66.5	7	45	34.6	0.520300752
78.6	96.7	10	25	25.3	0.261633919
78.6	91.4	10	30	27.5	0.300875274
78.6	85.6	10	35	29.8	0.348130841
78.6	79.4	10	40	32.4	0.408060453
78.6	72.6	10	45	35.4	0.487603306
78.6	111.3	15	25	26.7	0.239892183
78.6	105.2	15	30	28.9	0.274714829
78.6	98.6	15	35	31.3	0.317444219
78.6	91.4	15	40	33.9	0.370897155
78.6	83.6	15	45	36.8	0.440191388
78.6	121.5	18	25	27.3	0.224691358
78.6	114.5	18	30	29.7	0.259388646
78.6	107	18	35	32.3	0.301869159
78.6	99.1	18	40	34.9	0.352169526
78.6	90.6	18	45	37.7	0.41611479

Note: The performance curve tools require that your input data be normalised by dividing the performance value at each condition by the reference/nominal value (e.g., actual EIR / rated EIR). This normalisation is essential because it ensures the resulting curve has a value of 1.0 at reference conditions, which is how DesignBuilder/EnergyPlus interprets the curve during simulation. In **Curve Generator** this can be done by entering the reference value in the **Normalize by** field.

How to Generate Curves Using the Curve Generator

1. Open the **Curve Generator tool** in DesignBuilder, accessed from the Info panel of the **Curves** dialog.
2. Select **BiQuadratic** from the **Curve Category** dropdown
3. In the input data table, enter:
 - x column: Leaving chilled water temperatures (e.g., 5, 7, 10, 15, 18°C)
 - y column: Entering condenser water temperatures (e.g., 25, 30, 35, 40, 45°C)
 - Value column: Corresponding EIR values at each x, y combination
4. In the **Normalize by** field, enter the reference EIR value (e.g., if rated COP = 2.71, then reference EIR = $1/2.71 = 0.369$)
5. Click on the **Calculate** button to generate the coefficients
6. The tool in the right panel displays:
 - Six biquadratic coefficients (c1 through c6). These correspond to the biquadratic curve equation:

$$z = c1 + c2 \cdot x + c3 \cdot x^2 + c4 \cdot y + c5 \cdot y^2 + c6 \cdot x \cdot y$$
 - The minimum and maximum x and y values based on the input data range
 - R² value showing goodness-of-fit
7. Click the **Copy to DB** button to prepare for pasting the coefficients to the DesignBuilder Curve Editor, or **Save Curve File** for later entry, or **Copy to Clipboard** to paste the coefficients to a spreadsheet.

Curve Generator Tool for DesignBuilder

x	y	Value	Adjusted Value
5	25	0.779466711	0.777835059601672
5	30	0.897737478	0.894349283103707
5	35	1.042440318	1.04104520020574
5	40	1.230898961	1.21792281090777
5	45	1.389699099	1.42498211520981
7	25	0.75083877	0.751285014292494
7	30	0.862529284	0.86278536785385
7	35	1	1.00446741501521
7	40	1.176468359	1.17633115577656
7	45	1.410194452	1.37837659013792
10	25	0.70911814	0.712274549427212
10	30	0.815475741	0.816254098077551
10	35	0.943554625	0.950415340327891
10	40	1.105984539	1.11475827617823
10	45	1.321573098	1.30928290562857
15	25	0.650190538	0.649429382914373

Coefficient	Value
c1	0.653591825669105
c2	-0.00139203028168329
c3	5.43068732324135E-05
c4	-0.007390083289254
c5	0.000603633871999997
c6	-0.000501386994067798
xmin	5
xmax	18
ymin	25
ymax	45
R2	0.997695553253508

Save Curve File Clear Table ☐ Remove worst outlier data point BiQuadratic $z = c1 + c2 \cdot x + c3 \cdot x^2 + c4 \cdot y + c5 \cdot y^2 + c6 \cdot x \cdot y$ Copy to DB

Open Curve File Paste from Spreadsheet About Calculate Normalize by: Copy Clipboard

How to Generate Curves Using the Chiller Performance Curve Coefficients Tool

1. Open the **Chiller Performance Curve Coefficients** spreadsheet tool
2. In the performance data table, enter:
 - CAP_NOM (kW): Rated capacity
 - CAP (kW): Actual capacity at each condition
 - CWS (°C): Leaving chilled water temperatures (e.g., 5, 7, 10, 15, 18°C)
 - ECT (°C): Entering condenser water temperatures (e.g., 25, 30, 35, 40, 45°C)
 - UNIT RATE (kW): Power input
 - Rated EIR: The EIR in nominal conditions
3. The spreadsheet calculates the performance metrics: EIR(Normalized), EIR(Normalized EIR_predicted), COP, and EIR.
4. In the "Coefficients for the Energy Input Ratio Function of Temperature curve (EIRFT)" table, the tool displays:

- The calculated coefficient values labelled as (a, b, c, d, e, f). These correspond to the standard biquadratic form coefficients in the equation:

$$z = a + b \cdot x + c \cdot x^2 + d \cdot y + e \cdot y^2 + f \cdot x \cdot y$$
- R² value showing goodness-of-fit
- Other statistical measures from the regression

CAP NOM(kW)	CAP(kW)	CWS(°C)	CWS*CWS	ECT(°C)	ECT*ECT	CWS*ECT	EIR(Normalize)	EIR(Normalize)	EIR_predicted	UNIT RATE (k)	COP	Rated EIR	EIR
78.6	83.8	5	25	25	625	125	0.77946671	0.77783506	0.28698749	24.1	3.47717842	0.36895674	0.2875895
78.6	79.1	5	25	30	900	150	0.89773748	0.89434928	0.3299762	26.2	3.01908397	0.36895674	0.3312263
78.6	74.1	5	25	35	1225	175	1.04244032	1.0410452	0.38410065	28.5	2.6	0.36895674	0.38461538
78.6	68.7	5	25	40	1600	200	1.23089896	1.21792281	0.44936083	31.2	2.20192308	0.36895674	0.45414847
78.6	62.8	5	25	45	2025	225	1.3896991	1.42498212	0.52575676	32.2	1.95031056	0.36895674	0.51273885
78.6	88.8	7	49	25	625	175	0.75083877	0.75128501	0.27719167	24.6	3.6097561	0.36895674	0.27702703
78.6	83.9	7	49	30	900	210	0.86252928	0.86278537	0.31833048	26.7	3.1423221	0.36895674	0.318236
78.6	78.6	7	49	35	1225	245	1	1.00446741	0.37060503	29	2.71034483	0.36895674	0.36895674
78.6	72.8	7	49	40	1600	280	1.17646836	1.17633116	0.43401531	31.6	2.30379747	0.36895674	0.43406593
78.6	66.5	7	49	45	2025	315	1.41019445	1.37837659	0.50856134	34.6	1.92196532	0.36895674	0.52030075
78.6	96.7	10	100	25	625	250	0.70911814	0.71227455	0.2627985	25.3	3.82213439	0.36895674	0.26163392
78.6	91.4	10	100	30	900	300	0.81547574	0.8162541	0.30116245	27.5	3.32363636	0.36895674	0.30087527
78.6	85.6	10	100	35	1225	350	0.94355462	0.95041534	0.35066215	29.8	2.87248322	0.36895674	0.34813084
78.6	79.4	10	100	40	1600	400	1.10598454	1.11475828	0.41129758	32.4	2.45061728	0.36895674	0.40806045
78.6	72.6	10	100	45	2025	450	1.3215731	1.30928291	0.48306876	35.4	2.05084746	0.36895674	0.48760331
78.6	111.3	15	225	25	625	375	0.65019054	0.64942938	0.23961135	26.7	4.16853933	0.36895674	0.23989218
78.6	105.2	15	225	30	900	450	0.74457192	0.74087426	0.27335055	28.9	3.64013841	0.36895674	0.27471483
78.6	98.6	15	225	35	1225	525	0.8603833	0.86250082	0.31822549	31.3	3.15015974	0.36895674	0.31744422
78.6	91.4	15	225	40	1600	600	1.00525919	1.01430909	0.37423618	33.9	2.69616519	0.36895674	0.37089716
78.6	83.6	15	225	45	2025	675	1.19307045	1.19629904	0.4413826	36.8	2.27173913	0.36895674	0.44019139
78.6	121.5	18	324	25	625	450	0.60899106	0.61302565	0.22617995	27.3	4.45054945	0.36895674	0.22469136
78.6	114.5	18	324	30	900	540	0.70303268	0.69694972	0.2571443	29.7	3.85521886	0.36895674	0.25938865
78.6	107	18	324	35	1225	630	0.81816951	0.81105548	0.29924439	32.3	3.3126935	0.36895674	0.30186916
78.6	99.1	18	324	40	1600	720	0.95450085	0.95534294	0.35248022	34.9	2.83954155	0.36895674	0.35216953
78.6	90.6	18	324	45	2025	810	1.12781457	1.12981208	0.41685179	37.7	2.40318302	0.36895674	0.41611479

Coefficients for the Energy Input Ratio Function of Temperature curve (EIRFT)					
f	e	d	c	b	a
-0.00050139	0.00060363	-0.00739008	5.4307E-05	-0.00139203	0.65359182
7.276E-05	5.9747E-05	0.00427281	0.00015557	0.0044303	0.07869377
0.99769555	0.01249693	#N/A	#N/A	#N/A	#N/A
1645.18585	19	#N/A	#N/A	#N/A	#N/A
1.28467049	0.00296729	#N/A	#N/A	#N/A	#N/A

How to Enter Curve Coefficients into DesignBuilder and View the Curve Plot

Once you have generated the coefficients using either tool, you need to enter them into DesignBuilder to use them in your simulation:

1. Navigate to the Detailed HVAC system
2. Select the HVAC component (e.g., [Chiller](#), [Plant Loop Heat Pumps](#))
3. Go to the Performance Curves section
4. Create a copy of an existing Curve or add a new one.
5. Edit the Curve to enter the performance curve coefficients
6. Click the [Curve Plot](#) tab to visualize the curve

Example "Energy Input Ratio Modifier Function of Temperature Curve":

- Create a copy of an existing Curve or add a new one and name it "Example EIR Modifier Function of Temperature Curve"
- Open the Curve editor
- In the Curve editor choose the appropriate Curve type (e.g., Bi-Quadratic) and enter/paste the coefficients
- Set the curve Range (these should match your xmin, xmax, ymin, ymax values from the Performance Curve Coefficients Tools)

Edit Curve - Example EIR Modifier Function of Temperature Curve

Curves

General **Curve Plot**

General

Name Example EIR Modifier Function of Temperature Curve

Note

Source

Category Bi-Quadratic

Coefficients

Coefficient 1	0.6535918249
Coefficient 2	-0.0013920305
Coefficient 3	0.0000543069
Coefficient 4	-0.0073900832
Coefficient 5	0.0006036339
Coefficient 6	-0.0005013870

Range

MinimumX	5.0000000000
MaximumX	18.0000000000
MinimumY	25.0000000000
MaximumY	45.0000000000

Input unit type for X

Input unit type for Y

Output

Help

Info Data

Performance curves

EnergyPlus performance curves are generally polynomials that are used to characterize the performance of HVAC equipment. One other non-polynomial curve is also included to characterize the performance of pumps and fans.

The curves are usually derived from fits or regressions to data covering a limited range. Results for independent variable values outside this range are likely to be invalid, so curve input always contains a range of validity (maximum and minimum permitted values) for each independent variable and can optionally have limits on the curve output. No error or warning message is issued if an independent variable is outside the range. Instead, the curve manager uses the minimum value if an independent variable is less than the minimum, and the maximum if a variable exceeds the maximum. Similarly, no error or warning message is issued if the curve output is outside the range of the optional minimum and maximum curve output limits. Instead, the curve manager uses the minimum and maximum curve limits to cap the output of the performance curve.

[Paste curve coefficients](#)

[Paste curve coefficients in reverse order](#)

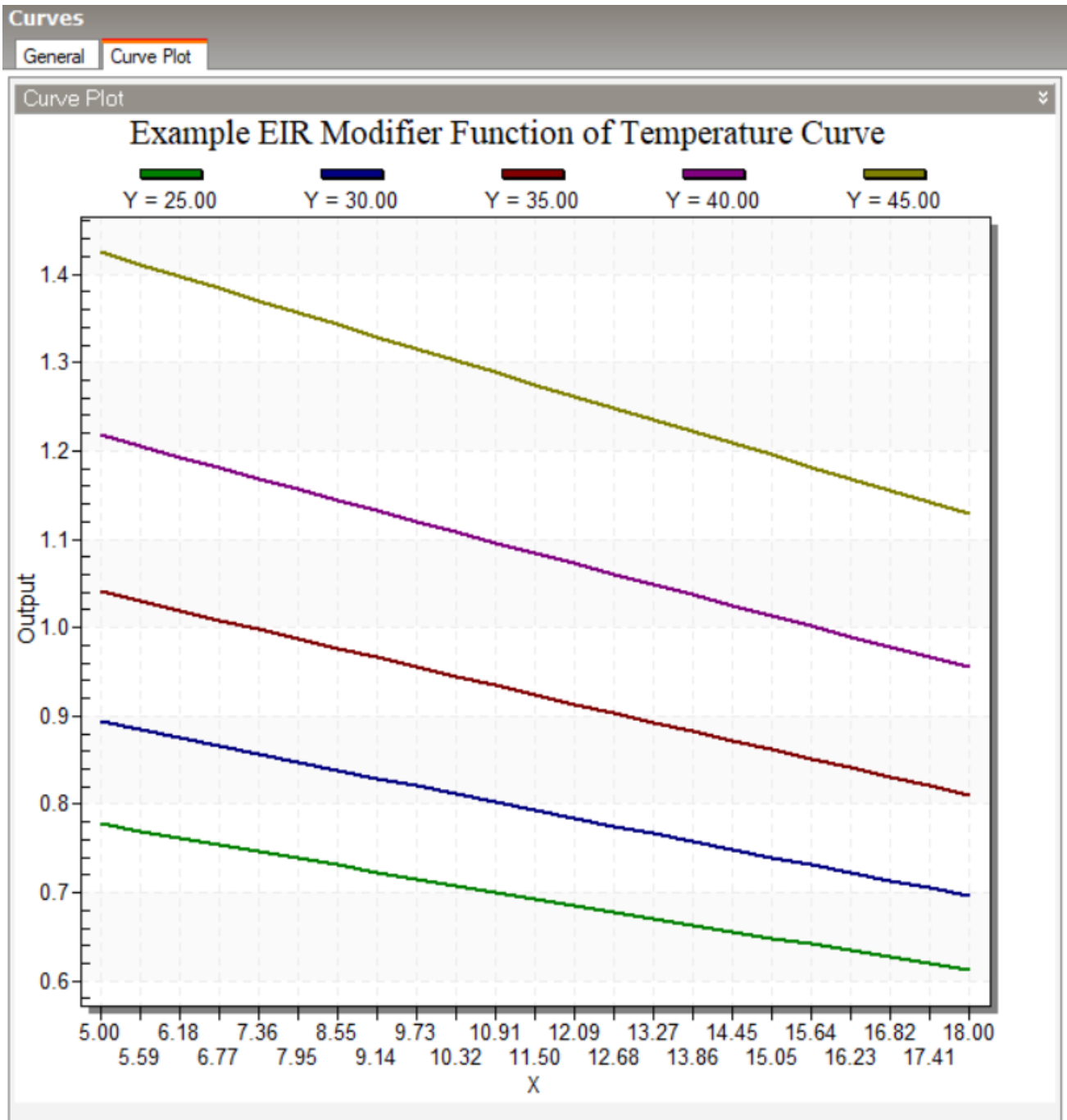
[Run curve generator tool](#)

Model data

Help Cancel OK

What you will see in the DesignBuilder [Curve Plot](#) tab:

- Horizontal axis: Supply water temperatures (e.g., 5°C to 18°C representing leaving water temperature)
- Legend: Outdoor/condenser temperature held constant for each curve (e.g., Y=25, Y=30, Y=35, Y=40, Y=45 representing entering condenser water)
- Vertical axis: The curve output (e.g., EIR modifier value)



Each coloured line shows the multiplier (e.g. EIR multiplier) at different leaving water temperatures while holding the entering condenser water temperature constant. For example, the green line (Y=25°C) shows how EIR varies as leaving chilled water temperature changes from 5°C to 18°C when the condenser temperature is at 25°C.

Important Checks When Using Any Performance Curve Coefficients Tool

To ensure accurate curve generation, always verify the following:

1. Column Order and Axis Assignment

Ensure each independent variable in your dataset is mapped to the proper axis. The order of your columns determines which variable appears where on the plot:

- Check that your first independent variable corresponds to what should be on the horizontal axis
- Check that your second independent variable corresponds to what should be in the legend

- Refer to DesignBuilder documentation or the [EnergyPlus Input/Output Reference](#) to confirm the expected variable order for your specific curve type

2. Normalisation Value

Enter the correct reference value to normalise the curve output. Normalisation is essential for ensuring that the curve has a value of 1.0 at reference conditions. This value depends on both the curve type and the equipment being modelled.

Why this matters: If you enter the wrong normalisation value, your curve will not equal 1.0 at reference conditions, which will cause simulation errors or incorrect equipment performance predictions. DesignBuilder/EnergyPlus uses the curve output of 1.0 at reference conditions as the baseline for scaling equipment performance during simulation. The specific reference value to use (e.g., rated capacity, reference EIR) should be determined based on your equipment data and curve type.

Additional Information

For additional information, refer to the following documentation:

1. [Performance Curve Data](#) - General information on performance curves and the Curve Generator tool
2. [Heat Pump - Cooling](#) / [Heat Pump -Heating](#) - Application of performance curves for heat pumps
3. [Chiller:Electric:EIR Technical Description](#) - Detailed technical description of chiller EIR model and its performance curves
4. [EnergyPlus Documentation](#) - You can download all the EnergyPlus documentation from here
5. [Unmet Hours Article: Determine COP Performance Curves in DesignBuilder for an ASHP](#) - Community discussion on generating and validating heat pump performance curves

EnergyPlus Curve Fit Tool

The EnergyPlus Curve Fit spreadsheet Tool can be used to generate HVAC performance curves in EnergyPlus curve object format. For each set of performance data entered, Capacity and EIR performance curves are generated, and these curves are generated either as a function of temperature(s) or flow fraction. The Capacity and EIR of Cooling DX Coils as a function of temperatures require only Biquadratic curve whereas Capacity and EIR of Heating DX Coils may use Biquadratic, Cubic and Quadratic curves. The selection of either of these curves is dependent on availability of performance data. The Capacity and EIR as a function of flow fraction allows either Cubic or Quadratic curve type.

The curve coefficients calculated by the tool can be copied over manually into the relevant DesignBuilder dialogs.

Note: The curve fit spreadsheet can typically be found in a folder like **C:\EnergyPlusV23-1-0\PreProcess\HVACCurveFitTool** after installing the standalone version of EnergyPlus.

The curve types allowed are:

Cooling Capacity as a Function of Temperatures: Biquadratic Curve
 $cCapacity = a0 + a1*X + a2*X^2 + a3*Y + a4*Y^2 + a5*X*Y$

Heating Capacity as a Function of Temperatures: Biquadratic curve
 $hCapacity = a0 + a1*X + a2*X^2 + a3*Y + a4*Y^2 + a5*X*Y$

Heating Capacity as a Function of Temperatures: Cubic Curve
 $hCapacity = a0 + a1*X + a2*X^2 + a3*x^3$

Energy Input Ratio (EIR) as a Function of Temperatures: Biquadratic Curve

$$\text{EIR} = a_0 + a_1 \cdot X + a_2 \cdot X^2 + a_3 \cdot Y + a_4 \cdot Y^2 + a_5 \cdot X \cdot Y$$

Energy Input Ratio (EIR) as a Function of Temperatures: Cubic Curve

$$\text{EIR} = a_0 + a_1 \cdot X + a_2 \cdot X^2 + a_3 \cdot x^3$$

Capacity as a Function of Flow Fraction: Cubic Curve

$$\text{EIR} = a_0 + a_1 \cdot X + a_2 \cdot X^2 + a_3 \cdot x^3$$

Capacity as a Function of Flow Fraction: Quadratic Curve

$$\text{EIR} = a_0 + a_1 \cdot X + a_2 \cdot X^2$$

Energy Input Ratio (EIR) as a Function of Flow Fraction: Cubic Curve

$$\text{EIR} = a_0 + a_1 \cdot X + a_2 \cdot X^2 + a_3 \cdot x^3$$

Energy Input Ratio (EIR) as a Function of Flow Fraction: Quadratic Curve

$$\text{EIR} = a_0 + a_1 \cdot X + a_2 \cdot X^2$$

These performance curves as a function of temperatures are generated for a given set of input data at a given speed. The curves as a function of flow fraction are generated at the rated temperature conditions. The rated test condition is the AHRI standard rated test condition. The AHRI standard test condition may vary by the equipment type. For multiple speeds or multiple stage DX Coils, different curve sets can be generated by entering a different set of data for each speed or stage at a time. This tool can generate the following four curve types: (1) Capacity as a function of temperatures, (2) Capacity as a function of flow fraction, (3) EIR as a function of temperatures, and (4) EIR as a function of flow fraction. The tool automatically populates the labels for each data inputs variable when users select the Coil Type, Independent Variables, Curve Type, and Units as shown on INPU tab. The labels shown in the INPU tab are selected for DX Coil to generate capacity and EIR biquadratic curves as function of temperatures. The tool can be used for the DX coil types: Coil:Cooling:DX:SingleSpeed, Coil:Heating:DX:SingleSpeed, Coil:Cooling:DX:TwoSpeed (high and low speed) , CoilPerformance:DX:Cooling (each stage), and any HVAC equipment that use Biquadratic, Cubic or Quadratic curves. To add this flexibility generic input data labels can be populated by selecting "Other" for DX Coil Type input field, located in Cell B3 in the INPUT tab.

Editing Zone Group Data

Zone group data can be accessed as follows:

- [General](#) tab, where the building zones to be included in the zone group are selected.
- [Heating and Cooling Sequence](#) tab where the order of simulation is defined for each item of zone equipment.
- [Zone defaults](#) tab which is hidden by default but can be displayed when the [Show HVAC zone default data on the HVAC zone group dialog](#) Program option is selected.

See the [HVAC zone groups and components](#) tutorial

See also [Understanding HVAC Zone Groups](#).

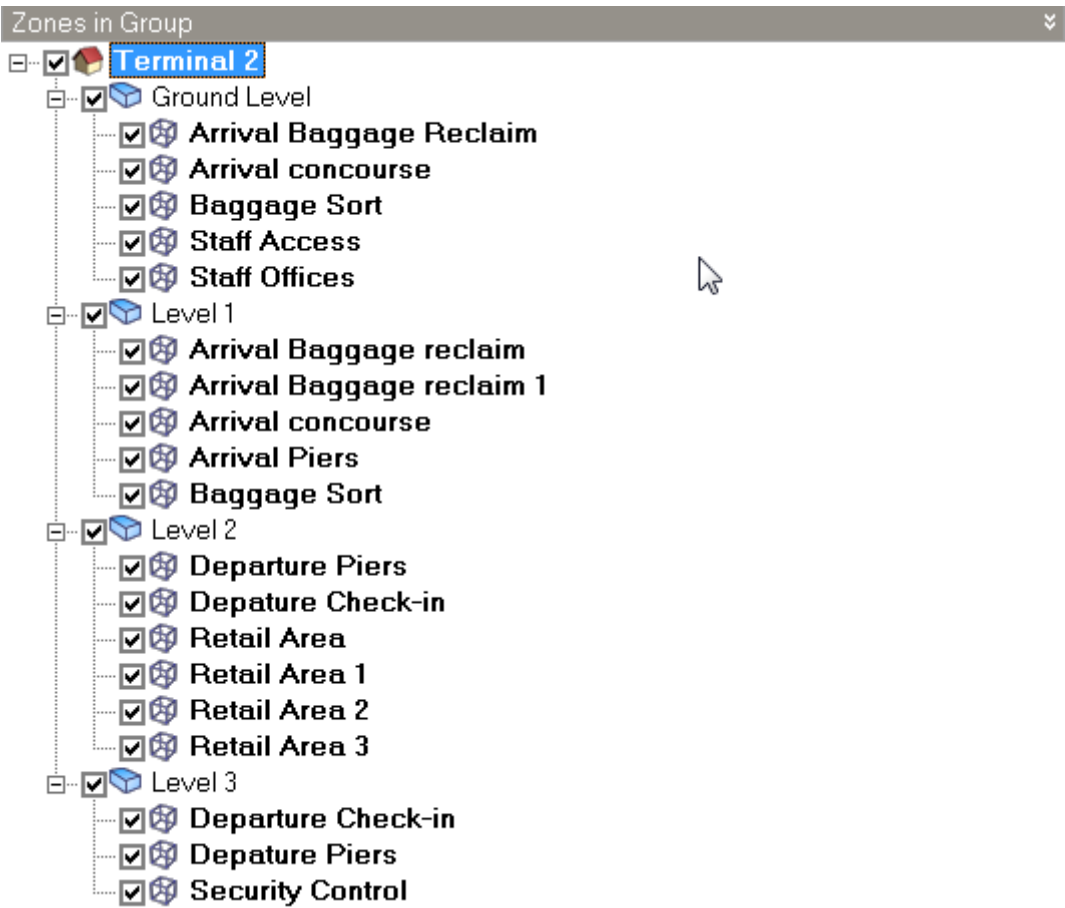
General Tab

Title

This is the name that you assign to the zone group. The name must be unique and if a duplicate name is entered, the software will automatically append an integer in order to ensure a unique name.

Zones in Group

This is a tree control with checkboxes that you can use to define which building zones are to be added to the zone group. Only zones that have not already been assigned to other zone groups or plenums are made available for selection.



Tip: You can use the checkboxes at building and block levels as a fast way to select/deselect all of the available zones in the building or block.

Heating And Cooling Calculation Sequence Tab

Heating and cooling sequences are used to specify the order in which items of equipment are simulated by EnergyPlus in order to satisfy a heating or cooling load. Select a value of 1 against the system that should be simulated first. In the example below for a DOAS air system and a FCU, the Direct air ADU has been set up to be calculated before the FCU to ensure that any loads introduced by the fresh air can be met by the FCU.

Heating Calculation Sequence	
Block1:Zone1 Fan Coil Unit	2
Block1:Zone1 Direct Air	1
Cooling Calculation Sequence	
Block1:Zone1 Fan Coil Unit	2
Block1:Zone1 Direct Air	1

If the calculation sequence had been set up the other way round then the FCU is unlikely to have been able to accurately meet heating and cooling setpoints since it would not be accounting for the load introduced by the ADU.

Note 1: The heating and cooling calculation sequence can only be assigned after equipment has been added to one of the constituent HVAC zones.

Note 2: When a Zone group includes both heating and/or cooling zone equipment with an ADU providing ventilation air, DesignBuilder will by default set up the calculation sequence so that the air system is simulated first and other zone equipment after that. This will generally be the correct setting as mentioned above, it ensures that any air load resulting from the supply of centrally delivered air to the zone can be met by the other zone equipment. However you can change that behaviour manually by selecting zone equipment to simulate first.

Zone Defaults

When the [Show HVAC zone default data on the HVAC zone group dialog](#) Program option is selected you can enter default settings which will be used to initialise any HVAC zones *subsequently added* to this HVAC zone group. Making changes to these zone defaults values do not affect HVAC zones once they have been created.

Tip: The most important application of HVAC zone group default data is when [saving HVAC templates](#) where specific HVAC zone settings are required. Any HVAC zone settings made in the source model are not saved with an HVAC template, but the settings on the HVAC zone group are saved and applied to zones added to the HVAC zone group when the template is subsequently loaded in the destination model.

Editing HVAC Zone Data

Zone data is divided using 3 tabs:

- [General](#)
- [Sequence](#)
- [Target](#)

The [General tab](#) is used to define temperature and humidity control together with zone sizing information (Cooling Sizing, Heating Sizing and Outdoor Air Sizing). The [Cooling sizing](#) and [Heating sizing](#) information provides the data needed to perform a zone design air flow calculation for a single zone. This calculation assumes a variable amount of supply air at a fixed temperature and humidity. The information needed consists of the zone inlet supply air conditions: temperature and humidity ratio for heating and cooling.

The calculation is done for every design day included in the input. The maximum cooling load and air flow and the maximum heating load and air flow are then saved for the system level design calculations and for the component automatic sizing calculations. The [Outdoor air sizing](#) information is used to define the design outdoor air flow rate. This can be specified in a number of ways as described under [Outdoor air method](#). This data is then used in the EnergyPlus system sizing calculation for sizing zone components that use outside air. You can also place limits on the heating and design cooling air flow rates. See [Heating design air flow method](#) and [Cooling design air flow method](#) and the explanations of the various heating and cooling flow input fields.

See also:

- The [Sequence](#) tab
- The [Target tab](#) is used to specify which other zones in the zone group the changes will be applied to.
- [Autosizing HVAC Components](#)

Edit HVAC Zone Data - General Tab

Data on this tab is used for controlling the zone internal conditions during the simulation and also for autosizing heating, cooling and outside air equipment related to the zone.

General

Name

This is the name of the building zone with which the current HVAC zone is linked.

Thermostat Schedules

Detailed HVAC Activity data set to **2-Detailed HVAC**

If the **2-Detailed HVAC** [Detailed HVAC Activity data](#) model option is set then you can select schedules defining zone heating and cooling setpoint temperatures on this dialog. Otherwise, if the **1-Simple HVAC** option is selected then zone setpoint temperatures and schedules are defined on the Activity and HVAC model data tabs respectively and the Thermostat schedule data is not displayed on this dialog.

Note: The temperatures defined in these schedules must be in °C regardless of whether you are using [IP](#) or [SI](#) units in the interface.

Thermostat heating setpoint schedule

This is the zone heating temperature schedule. Values in the schedule are temperatures (in °C).

Thermostat cooling setpoint schedule

This is the zone cooling temperature schedule. Values in the schedule are temperatures (in °C).

Comfort PMV Setpoint Schedules

Detailed HVAC Activity data set to **2-Detailed HVAC**

If the **2-Detailed HVAC** [Detailed HVAC Activity data](#) model option is set then you can select schedules which define zone heating and cooling setpoint [PMV values](#) on this dialog. Otherwise, if the **1-Simple HVAC** option is selected then zone setpoint PMV values and schedules are defined on the Activity and HVAC model data tabs respectively and the Thermostat schedule data is not displayed on this dialog.

The [Thermal comfort calculator tool](#) can be used to derive PMV values based on perhaps more familiar parameters.

Heating PMV setpoint schedule

This is the zone heating PMV schedule. Values in the schedule are PMV indices, typically in the range -3 to 3.

Cooling PMV setpoint schedule

This is the zone cooling PMV schedule. Values in the schedule are PMV indices, typically in the range -3 to 3.

Humidistat Control

Humidistat control

This check-box is used to specify whether or not the zone has humidity control.

Detailed HVAC Activity data is **1-Simple HVAC**

If the **1-Simple HVAC** option is selected then zone RH setpoints on the Activity tab are used and the demand schedule data on this dialog defines the times when these setpoints operate.

Humidifying demand schedule

This is a schedule that defines the times when humidification may be required. The schedule should have only values of 0 (for times when no humidification is required) or 1 (when humidification to the humidification setpoint defined on the Activity tab is required).

Dehumidifying demand schedule

This is a schedule that defines the times when dehumidification may be required. The schedule should have only values of 0 (for times when no dehumidification is required) or 1 (when dehumidification to the dehumidification setpoint defined on the Activity tab is required).

Detailed HVAC Activity data is **2-Detailed HVAC**

If the **2-Detailed HVAC** [Detailed HVAC Activity data](#) model option is set then you can select schedules defining zone relative humidity setpoints on this dialog.

Humidifying RH setpoint schedule

This is a schedule that defines the humidifying relative humidity setpoint, expressed as a percentage (0-100), for each timestep of the simulation.

Dehumidifying RH setpoint schedule

This is a schedule that defines the dehumidifying relative humidity setpoint, expressed as a percentage (0-100), for each timestep of the simulation.

More information on controlling humidity is available in the [Humidity control](#) section.

The rest of the General tab of the HVAC Zone dialog is related to [Detailed HVAC autosizing](#) calculations only.

CO2 And Contaminant Control

CO2 and contaminant control

This check-box is used to specify whether or not the zone has CO2 or contaminant control. This must be selected if the zone is to have maximum CO2 or generic contaminant concentration controlled through one of the IAQP DCV methods.

Note: When the AHU DCV [System outdoor air method](#) has been set to one of **1-Zone sum**, **2-Standard 62.1 Ventilation Rate Procedure (VRP)** or **3-Standard 62.1 ventilation rate procedure with limit** then this option **must not be selected**.

Note: When one of the proportional DCV [System outdoor air methods](#) is selected on the AHU dialog (**5-Proportional control based on design occupancy**, **6-Proportional control based on occupancy schedule** or **9-Proportional control based on design outdoor air rate**) then selection of the **CO2 and contaminant control** checkbox on this dialog is optional. If you wish to specify zone minimum CO2 concentration settings for proportional control then you should check this checkbox, otherwise leave it unchecked, in which case outdoor air CO2 concentration will be used as the minimum instead. You can find details of the calculations used to calculate the target indoor CO2 concentration in the [Engineering reference](#) under Proportional Control.

Note: When one of the IAQP DCV [System outdoor air methods](#) are selected (**4-Indoor Air Quality Procedure (IAQP)** or **7-Indoor Air Quality Procedure (IAQP) generic contaminant**) then the **CO2 and contaminant** option must be selected for at least one HVAC zone connected to the AHU.

More information on DCV is available in the [DCV control](#) section of the AHU help page.

Detailed HVAC Activity data is 1-Simple HVAC

If the **1-Simple HVAC** option is selected then fixed contaminant setpoints from the Activity tab are used and no further data needs to be entered here.

Detailed HVAC Activity data is 2-Detailed HVAC

If the **2-Detailed HVAC** [Detailed HVAC Activity data](#) model option is set then you can select schedules defining time-varying CO2 and contaminant setpoints on this dialog.

CO2 control availability schedule

This is a schedule that defines the availability of CO2 control for each timestep of the simulation.

CO2 setpoint schedule

This is a schedule that defines the CO2 setpoint in ppm, for each timestep of the simulation.

Minimum CO2 concentration schedule

This is a schedule that defines the minimum concentration of CO2 in ppm, for each timestep of the simulation.

Generic contaminant control availability schedule

This is a schedule that defines the availability of Generic contaminant control for each timestep of the simulation.

Generic contaminant setpoint schedule

This is a schedule that defines the Generic contaminant setpoint in ppm, for each timestep of the simulation. Enter values between 0 and 1.

The rest of the General tab of the HVAC Zone dialog is related to [Detailed HVAC autosizing](#) calculations only.

Zone Air Distribution Effectiveness

ASHRAE Standard 62.1 defines Zone air distribution effectiveness (Ez) as a measure of how effectively the zone air distribution system uses the supply air to maintain acceptable air quality in the occupied zone (also known as the breathing zone). In an imperfect system the fresh air introduced through supply diffusers may not mix fully with the rest of the room air before being extracted. These settings allow the distribution effectiveness to be defined separately for heating and cooling operation modes.

Method

You can choose to either define fixed zone air distribution effectiveness values for heating and cooling or select a schedule with time-varying values to cover both heating and cooling together. Select from:

- **1-Separate heating and cooling** where fixed zone air distribution effectiveness values are entered separately for heating and cooling.
- **2-Scheduled** where a schedule is selected which defines how the zone air distribution effectiveness for both heating and cooling varies with time.

Effectiveness in cooling mode

Enter the fixed zone air distribution effectiveness when the zone is in cooling mode. ASHRAE Standard 62.1-2010 provides typical values (see table below). The default value 1.0.

Effectiveness in heating mode

Enter the fixed zone air distribution effectiveness when the zone is in heating mode. ASHRAE Standard 62.1-2010 provides typical values (see table below). The default value 1.0.

Zone Air Distribution Effectiveness (TABLE 6-2 ASHRAE 62.1)	
Air Distribution Configuration	Ez
Ceiling supply of cool air.	1.0
Ceiling supply of warm air and floor return.	1.0
Ceiling supply of warm air 15°F (8°C) or more above space temperature and ceiling return.	0.8
Ceiling supply of warm air less than 15°F (8°C) above space temperature and ceiling return provided that the 150 fpm (0.8 m/s) supply air jet reaches to within 4.5 ft (1.4 m) of floor level. Note: For lower velocity supply air, Ez = 0.8.	1.0
Floor supply of cool air and ceiling return provided that the 150 fpm (0.8 m/s) supply jet reaches 4.5 ft (1.4 m) or more above the floor. Note: Most underfloor air distribution systems comply with this proviso.	1.0
Floor supply of cool air and ceiling return, provided low-velocity displacement ventilation achieves unidirectional flow and thermal stratification.	1.2
Floor supply of warm air and floor return.	1.0
Floor supply of warm air and ceiling return.	0.7
Make-up supply drawn in on the opposite side of the room from the exhaust and/or return.	0.8
Make-up supply drawn in near to the exhaust and/or return location.	0.5
1. "Cool air" is air cooler than space temperature. 2. "Warm air" is air warmer than space temperature. 3. "Ceiling" includes any point above the breathing zone. 4. "Floor" includes any point below the breathing zone. 5. As an alternative to using the above values, Ez may be regarded as equal to air change effectiveness determined in accordance with ANSI/ASHRAE Standard	

Recirculation fraction

The non-negative numeric input for this field is the fraction of a zone's recirculation air that does not directly mix with the outdoor air. The zone secondary recirculation fraction Er is determined by the designer based on system configuration. For plenum return systems with secondary recirculation (e.g., fan-powered VAV with plenum return) Er is usually less than 1.0, although values may range from 0.1 to 1.2 depending upon the location of the ventilation zone relative to other zones and the air handler. For ducted return systems with secondary recirculation (e.g., fan-powered VAV with ducted return), Er is typically 0.0, while for those with system-level recirculation (e.g, dual-fan dual-duct systems with ducted return) Er is typically 1.0. For other system types, Er is typically 0.75. Minimum is 0.0, and default is 0.0 for single-path systems (also to maintain backward compatibility). For parallel fan-powered VAV systems, the secondary ventilation path only functions (Er > 0.0) when the fans in the VAV boxes operate, which is during heating. The local ventilation path and the benefits of secondary recirculation disappear during cooling, when the local parallel fans are off (Er = 0.0).

Sizing

Cooling Sizing

Cooling design supply air temperature input method

This option determines how the cooling design air flow rate is calculated in the system autosizing calculations:

- **1- Supply air temperature** means that the **Zone cooling design supply air temperature** (below) will be used to determine the zone cooling design air flow rate.
- **2-Temperature difference** where the **Zone cooling design supply air temperature difference** (below) is used to determine the zone cooling design air flow rate.

Cooling design supply air temperature

This is the supply air temperature (in °C or °F) for the zone cooling air flow rate calculation. Air is supplied to the zone at this temperature during the cooling design day simulation and the zone load is met by varying the zone air flow rate. The maximum zone flow rate is used as the zone cooling design air flow rate.

This data is only available when the Cooling design supply air temperature input method (above) is **1-Supply air temperature**.

Cooling design supply air temperature difference

The temperature difference between cooling design supply air temperature and room air temperature (in delta°C or delta°F) for the zone cooling design air flow rate calculation. Air is supplied to the zone with this temperature difference during the cooling design day autosizing simulation. The zone load is met by varying the zone air flow rate. The maximum zone flow rate is used as the zone cooling design air flow rate.

This data is only available when the Cooling design supply air temperature input method (above) is **2-Temperature difference**.

Cooling design supply air humidity ratio

This is the humidity ratio in kilograms of water per kilogram of dry air of the supply air in the zone cooling autosizing air flow rate calculation.

Cooling minimum air flow fraction

This is the minimum zone design cooling volumetric flow rate expressed as a fraction of the zone design cooling volumetric flow rate for VAV systems only. In all cases the maximum flow derived from [Cooling minimum flow per zone floor area](#), [Cooling minimum air flow](#), and [Cooling minimum air flow fraction](#) is used to set a minimum supply air flow rate for the zone for VAV systems. The default is zero. This input is currently used in sizing the fan minimum flow rate. It does not affect other component auto-sizing.

Note: This value is only used when the zone is connected to a VAV Air loop, in which case this data is used for sizing and not for the simulation itself.

Zone cooling sizing factor

This is the zone level cooling sizing ratio. The zone design cooling air flow rates and loads will be multiplied by the number input in this field. The default value comes from the [Zone group dialog](#) whose default **Zone cooling sizing factor** is the ASHRAE recommended cooling oversizing factor of 1.15.

Cooling design air flow method

There are 3 options:

- **1-Design day** means the program will calculate the zone design cooling air flow rate using the zone cooling sizing information and a design day simulation without imposing any limits other than those set by the minimum outside air requirements.
- **2-Flow/zone** means that the program will use the [Cooling design air flow rate](#) as the zone design cooling air flow rate.
- **3-Design day with limit** means that the maximum from [Cooling minimum flow per zone floor area](#) and [Cooling minimum air flow](#) will set a lower limit on the design maximum cooling air flow rate.

Cooling design air flow rate

This is the design zone cooling air flow rate (in m³/s or ft³/min). This setting is only required if [Cooling design air flow method](#) is specified as **2-Flow/zone**. This value will be multiplied by the global or zone sizing factor and by zone multipliers.

Cooling minimum flow per zone floor area

This is the minimum zone cooling volumetric flow rate per square meter (units are m³/s-m²). This setting is only required if the [Cooling design air flow method](#) is specified as **3-Design day with limit**. In this case it sets a lower bound on the zone design cooling air flow rate. In all cases the maximum flow derived from [Cooling minimum flow per zone floor area](#), [Cooling minimum air flow](#), and [Cooling minimum air flow fraction](#) is used to set a minimum supply air flow rate for the zone for VAV systems. The default is 0.000762 m³/s-m², corresponding to 0.15 cfm/ft². The applicable sizing factor is not applied to this value.

Cooling minimum air flow

This is the minimum zone cooling volumetric flow rate (in m³/s). This field is used when [Cooling design air flow method](#) is specified as **3-Design day with limit**. In this case it sets a lower bound on the zone design cooling air flow rate. In all cases the maximum flow derived from [Cooling minimum flow per zone floor area](#), [Cooling minimum air flow](#), and [Cooling minimum air flow fraction](#) is used to set a minimum supply air flow rate for the zone for VAV systems. The default is zero. The applicable sizing factor is not applied to this value.

Heating Sizing

Heating design supply air temperature input method

This option determines how the heating design air flow rate is calculated in the system autosizing calculations:

- **1- Supply air temperature** means that the **Zone heating design supply air temperature** (below) will be used to determine the zone heating design air flow rate.
- **2-Temperature difference** where the **Zone heating design supply air temperature difference** (below) is used to determine the zone heating design air flow rate.

Heating design supply air temperature

This is the supply air temperature (in °C or °F) for the zone heating design air flow rate calculation. Air is supplied to the zone at this temperature during the heating design day simulation. The zone load is met by varying the zone air flow rate. The maximum zone flow rate is used as the zone heating design air flow rate.

Heating design supply air temperature difference

The temperature difference between heating design supply air temperature and room air temperature (in delta°C or delta°F) for the zone heating design air flow rate calculation. Air is supplied to the zone with this temperature difference during the heating design day autosizing simulation. The zone load is met by varying the zone air flow rate. The maximum zone flow rate is used as the zone heating design air flow rate.

Heating design supply air humidity ratio

This is the humidity ratio in kilograms of water per kilogram of dry air of the supply air in the zone heating design air flow rate calculation.

Zone heating sizing factor

This is the zone level heating sizing ratio. The zone design heating air flow rates and loads will be multiplied by the number input in this field. The default value comes from the [Zone group dialog](#) whose default **Zone heating sizing factor** is the ASHRAE recommended heating oversizing factor of 1.25.

Heating design air flow method

There are three options:

- **1-Design day** means the program will calculate the zone design heating air flow rate using the zone heating sizing information and a design day simulation without imposing any limits other than those set by the minimum outside air requirements.
- **2-Flow/zone** means that the program will use the [Heating design air flow rate](#) as the zone design heating air flow rate.
- **3-Design day with limit** means that the maximum from [Heating maximum flow per zone floor area](#) and [Heating maximum air flow](#) will set a lower limit on the design maximum heating air flow rate.

Heating design air flow rate

This is the design zone heating air flow rate (in m³/s or ft³/min). This setting is only required if [Heating design air flow method](#) is specified as **2-Flow/zone**. This value will be multiplied by the global or zone sizing factor and by zone multipliers.

Heating maximum flow per zone floor area

This is the maximum zone heating volumetric flow rate per square metre (units are m³/s-m²). This setting is only required if the [Heating design air flow method](#) is specified as **3-DesignDayWithLimit**. In this case it sets an upper bound on the zone design heating air flow rate. In all cases the maximum flow derived from [Heating maximum flow per zone floor area](#), [Heating maximum air flow](#), and [Heating maximum air flow fraction](#) is used to set a maximum heating supply air flow rate for the zone for VAV systems. The default is 0.002032 m³/s-m², corresponding to .40 cfm/ft². This input is not currently used for auto-sizing any of the components.

Heating maximum air flow

This is the maximum zone heating volumetric flow rate in m³/s. This setting is only required if the [Heating design air flow method](#) is specified as **3-Design day with limit**. In this case it sets an upper bound on the zone design heating air flow rate. In all cases the maximum flow derived from [Heating maximum flow per zone floor area](#), [Heating maximum air flow](#), and [Heating maximum air flow fraction](#) is used to set a maximum heating supply air flow rate for the zone for VAV systems. The default is 0.14158 m³/s, corresponding to 300 cfm. This input is not currently used for auto-sizing any of the components.

Outdoor Air Sizing

When the [Detailed HVAC Activity data](#) is set to **1-Simple HVAC** then all data under the **Outdoor Air Sizing** header is disabled and the outdoor air requirement settings are copied from the corresponding building zones' Activity and/or HVAC model data as described below.

When the [Detailed HVAC Activity data](#) is set to **2-Detailed HVAC** then the outdoor air requirement is defined here on the HVAC zone dialog.

Outdoor air method

There are five options:

- **1-Flow/Zone**, means that the program will use the [Outdoor air flow per zone](#) as the zone design outdoor air flow rate.
- **2-Flow/Person**, means the program will use the input from the field [Outdoor air flow per person](#) and the [maximum occupancy](#) to calculate a zone design outdoor air flow rate.
- **3-Flow/Area**, means that the flows calculated from the fields [Outdoor air flow per zone floor area](#) used to obtain the zone design outdoor air flow rate.
- **4-Sum**, means that the flows calculated from the fields [Outdoor air flow per person](#), [Outdoor air flow per zone floor area](#) and [Outdoor air flow per zone](#) will be added to obtain the zone design outdoor air flow rate.

- **5-Maximum** means that the maximum flow derived from [Outdoor air flow per person](#), [Outdoor air flow per zone floor area](#) and [Outdoor air flow per zone](#) will be used for the zone design outdoor air flow rate.

Note: If the [Detailed HVAC Activity data](#) is set to **1-Simple HVAC** then this option is disabled and the **Outdoor air method** is copied from the corresponding building zones' HVAC model data.

Outdoor air flow per person

This is the design outdoor air flow rate per person for this zone (in m³/s or ft³/min). This setting is only required if either one of the **2-Flow/person**, **4-Sum**, or **5-Maximum** options have been selected for the [Outdoor air method](#). The default is 0.00944 m³/s (20 cfm per person). An outdoor air flow rate is calculated based on the total [number of people](#) assigned to the zone. Occupancy schedule values are not applied.

Note: If the [Detailed HVAC Activity data](#) is set to **1-Simple HVAC** then this option is disabled and the **Outdoor air flow per person** is copied from [Outdoor air flow per person](#) from the corresponding building zones' Activity model data.

Outdoor air flow per zone floor area

This is the design outside air volumetric flow rate per square meter of floor area (in m³/s-m²). This setting is only required if either one of the **3-Flow/area**, **4-Sum**, or **5-Maximum** options have been selected for the [Outdoor air method](#). This value is multiplied by the zone floor area to obtain a flow rate.

Note: If the [Detailed HVAC Activity data](#) is set to **1-Simple HVAC** then this option is disabled and the [Outdoor air flow per zone floor area](#) data is copied from the corresponding building zones' Activity model data.

Outdoor air flow per zone

This is the design outdoor air flow rate for this zone (in m³/s or ft³/min). This setting is only used if either one of the **1-Flow/zone**, **4-Sum**, or **5-Maximum** options have been selected for the [Outdoor air method](#).

Note: If the [Detailed HVAC Activity data](#) is set to **1-Simple HVAC** then this option is disabled and the flow per zone is calculated from the ac/h value set on the HVAC model data tab and displayed here.

Dedicated Outdoor Air System Sizing

Account for Dedicated Outdoor Air System

Checking this option means that the zone sizing calculation will use the inputs below to calculate the heat gain or loss imposed on the zone by a Dedicated Outdoor Air System (DOAS). This heat gain/loss is added to the zone design heat gain for the zone and the zone design air flow rate is adjusted to meet the DOAS heat gain plus the zone design heat gain.

Field: Dedicated Outdoor Air System Control Strategy

Select from three control strategies for the outdoor air system. The choices are:

- **1-Neutral supply air**, the default option in which ventilation air supplied to the zone will cause little heating or cooling. The air will be heated or cooled to keep it between the low and high temperature setpoints specified in the subsequent two fields. A good choice for these fields might be 21.1 and 23.9°C.
- **2-Neutral dehumidified supply air**, where the ventilation air will be cooled and dehumidified and then reheated to a neutral temperature. The ventilation air is cooled to the lower setpoint temperature (if necessary) and reheated to the upper setpoint temperature. A good choice for the setpoints would be 14.4 and 22.2°C, or
- **3-Cold supply air** where the ventilation air will be used to supply cooling to the zone. Cold outside air is heated to the upper setpoint; warm outside air is cooled to the lower setpoint. A good choice for the setpoints would be 12.2 and 14.4°C.

Low temperature setpoint for design

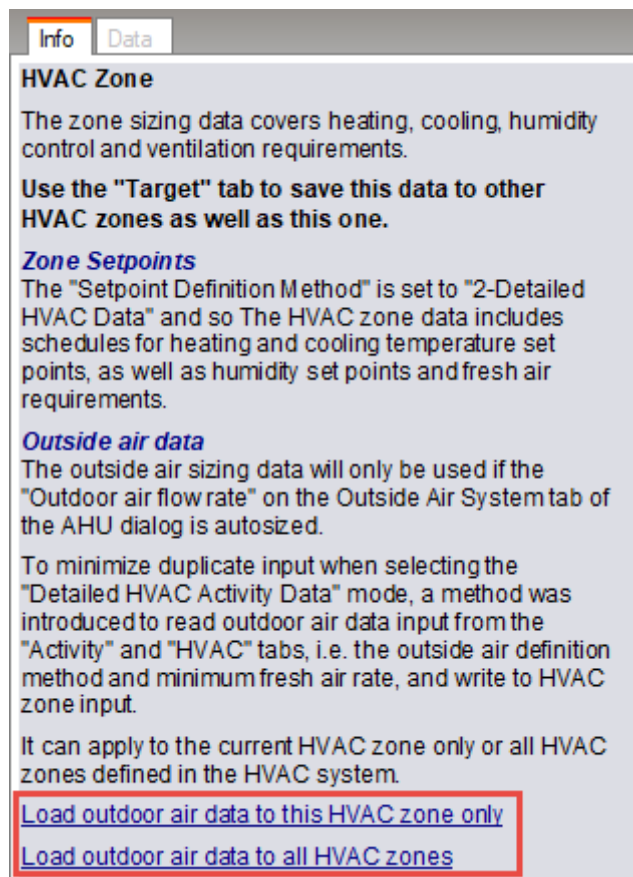
The lower setpoint temperature to be used with the DOAS design control strategy (in °C or °F). The default is autosized to the values given above for the three design control strategies.

High temperature setpoint for design

The higher setpoint temperature to be used with the DOAS design control strategy (in °C or °F). The default is autosized to the values given above for the three design control strategies.

Load Outdoor Air Model Data

When using the [Detailed HVAC Activity data](#) model option, 2 links are displayed on the Info panel to facilitate the copying of relevant outdoor air data from the HVAC and Activity model data tabs to the [Outdoor air sizing](#) section of the HVAC zone dialog.



Load outdoor air data to this HVAC zone only

When the [Load outdoor air data to this HVAC zone only](#) Info panel link is clicked, model data from the associated building zone is copied from the Activity and HVAC tabs to this dialog.

For example, if the Mechanical Ventilation [Outside air definition method](#) on the HVAC tab is set to **4-Min fresh air (Sum per person + per area)**, DesignBuilder will make the following settings under the **Outside Air Sizing** header of this dialog.

- [Outside air method](#) is set to **4-Sum**.
- [Outside air flow per person](#) is set based on the setting on the [Minimum fresh air per person](#) setting on the Activity tab.
- [Outside air flow per zone floor area](#) is set based on the setting on the [Mech vent per area](#) setting on the Activity tab.

Load outdoor air data to all HVAC zones

When the [Load outdoor air data to all HVAC zones](#) Info panel link is clicked, model data from the associated building zone is copied from the Activity and HVAC tabs to this dialog. Then if/when you press the OK button to confirm settings for this HVAC zone, the Outside Air Sizing data on this dialog is also copied to all other HVAC zones in the model.

Edit HVAC Zone Data - Sequence

Data on the [Sequence](#) tab defines the load distribution scheme, heating and cooling calculation sequences and sequential heating and cooling fraction schedules.

Load Distribution Scheme

Load Distribution Scheme

Select the load distribution scheme to be used for this HVAC Zone. Choose from the following options:

- **1-Sequential Load** - loads each piece of available equipment sequentially with the full remaining load.
- **2-Uniform Load** - evenly distributes the total load equally among all available components on the equipment list for the current load type.
- **3-Uniform PLR** - evenly distributes the total load among all available components on the equipment list for the current load type at a uniform part load ratio (PLR).
- **4-Sequential Uniform PLR** - determines how many of the available components are required to meet the current load, then distributes the load at a uniform part load ratio (PLR).

Heating Calculation Sequence

Each item of zone heating equipment associated with the current HVAC zone is listed under the **Heating Calculation Sequence** header. Here you can define the order that each item of equipment is simulated and hence the sequence that each item of equipment will try to meet the demand with its available capacity.

Equipment is simulated in the order specified, depending on the current thermostat request. For equipment of similar type, assign sequence 1 to the first system intended to serve that type of load, assign sequence 2 to the next system, and so on. The #1 sequence equipment will try to meet the entire demand with its available capacity and then pass the results on to #2 to meet any remaining load and so on.

For situations where one or more equipment types has limited capacity or limited control capability, order the sequence so that the most controllable piece of equipment runs last. For example, with a dedicated outdoor air system (DOAS), the air terminal for the DOAS should be assigned a sequence of 1. Any other equipment should be assigned sequence 2 or higher so that it will see the net load after the DOAS air is added to the zone.

Cooling Calculation Sequence

Each item of zone cooling equipment associated with the current HVAC zone is listed under the **Cooling Calculation Sequence** header. Here you can define the order that each item of equipment is simulated and hence the sequence that each item of equipment will try to meet the demand with its available capacity.

Equipment is simulated in the order specified, depending on the current thermostat request. For equipment of similar type, assign sequence 1 to the first system intended to serve that type of load, assign sequence 2 to the next system, and so on. The #1 sequence equipment will try to meet the entire demand with its available capacity and then pass the results on to the #2 to meet any remaining load and so on.

For situations where one or more equipment types has limited capacity or limited control capability, order the sequence so that the most controllable piece of equipment runs last. For example, with a dedicated outdoor air system (DOAS), the air terminal for the DOAS should be assigned a sequence of 1. Any other equipment should be assigned sequence 2 or higher so that it will see the net load after the DOAS air is added to the zone.

Sequential Heating Fraction Schedules

For each item of zone heating equipment associated with the current HVAC zone, select a schedule that specifies the fraction of the remaining heating load this equipment will attempt to serve. This setting is only available if the Load distribution scheme is set to **1-Sequential Load**.

Sequential Cooling Fraction Schedules

For each item of zone cooling equipment associated with the current HVAC zone, select a schedule that specifies the fraction of the remaining cooling load this equipment will attempt to serve. This setting is only available if the Load distribution scheme is set to **1-Sequential Load**.

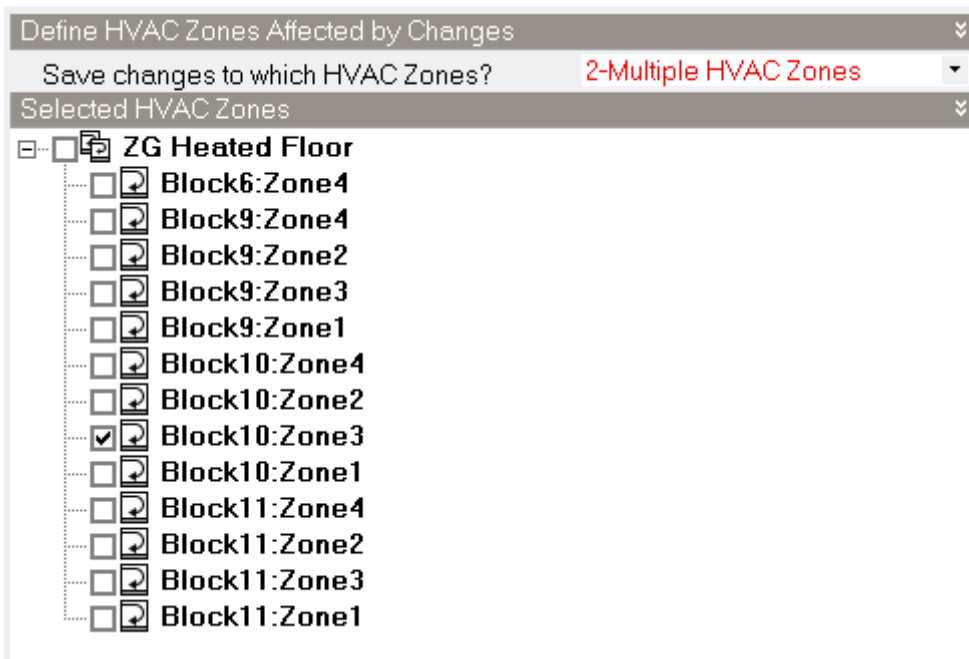
Edit HVAC Zone Data - Target

Define HVAC Zone Affected By Changes

HVAC Zones target

By default any edits made to the HVAC Zone dialog will apply only to the current HVAC Zone. However it is often convenient to make the same changes to one or more other zones in the same HVAC Zone group. This selection allows you to control how changes are applied. Options are:

- **1-This HVAC Zone only** - the data on the dialog will be saved for the current HVAC zone only.
- **2-Multiple (all settings)** - all of the data on the dialog will be saved for all HVAC zones in the group
- **3-Multiple (modified settings only)** - only the changed settings on the dialog (shown in red) will be saved for all HVAC zones in the group.



Note: If you would like to change the default behaviour so that the equivalent objects in other HVAC zones in the same group are all ticked when dialogs are first opened then you should check the [Target all HVAC zones in group by default](#) Program option.

Setpoint Manager

To edit the data associated with a setpoint manager, you first need to select it by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the Edit selected component tool from the toolbar.

Name

This is the name that you assign to the setpoint manager which should be unique. If the supplied name is not unique, the software will automatically append a backslash and number to ensure that there are no duplicate names.

Type

There several different types of setpoint manager available. However only certain types of setpoint manager are available for different types of loop:

- [1-Scheduled](#) (All loops).
- [2-Warmest](#) ([Generic](#) and [dual duct](#) air loops only).
- [3-Coldest](#) ([Generic](#) and [dual duct](#) air loops only).
- [4-Multi-zone minimum humidity average](#) (Air loops only).
- [5-Multi-zone maximum humidity average](#) (Air loops only).
- [6-Multi-zone humidity minimum](#) (Air loops only).
- [7-Multi-zone humidity maximum](#) (Air loops only).
- [8-Single zone humidity minimum](#) (Air loops only).
- [9-Single zone humidity maximum](#) (Air loops only).
- [10-Outdoor air reset](#) ([Generic](#) and [dual duct](#) air loops, plant and condenser loops).
- [11-Single zone heating](#) (Air loops only).
- [12-Single zone cooling](#) (Air loops only).
- [13-Multi-zone cooling average](#) (Air loops only).

- [14- Multi-zone heating average](#) (Air loops only).
- [15-Follow ground temperature](#) (Condenser loops only)
- [16-Follow outdoor air temperature](#)
- [17-Dual setpoint](#) (Plant loops only)

1-Scheduled Setpoint Manager

Scheduled setpoint managers are the simplest type of setpoint manager and work by placing a scheduled setpoint value at the point or connection in the loop at which the setpoint manager control line has been placed. This allows you to specify either a constant, a seasonal, or other time-varying scheduled reset control strategy.

Control variable

The control variable can be one of:

- **1-Temperature** - defines temperature of fluid at node (°C)
- **2-Maximum temperature** - defines the maximum temperature of fluid at node (°C)
- **3-Minimum temperature** - defines minimum temperature of fluid at node (°C)
- **4-Humidity ratio** - defines the humidity ratio of fluid at node (kg water/ kg dry air)
- **5-Maximum humidity ratio** - defines maximum humidity ratio of fluid at node (kg water/ kg dry air)
- **6-Minimum humidity ratio** - defines minimum humidity ratio of fluid at node (kg water/ kg dry air)
- **7-Mass flow rate** - defines mass flow rate of fluid at node (kg/s)
- **8-Maximum mass flow rate** - defines maximum mass flow rate of fluid at node (kg/s)
- **9-Minimum mass flow rate** - defines minimum mass flow rate of fluid at node (kg/s)

Setpoint variable schedule

Select a schedule whose values are to be used as setpoints on the node or node list. The schedule value for each time period is the setpoint for this type of setpoint manager. The schedule must have values that are in the appropriate units as defined in the previous field.

2-Warmest Setpoint Manager

Warmest setpoint managers are available only in Generic and Dual Duct air loops. The warmest setpoint manager resets the cooling supply air temperature of a central forced air HVAC system according to the cooling demand of the warmest zone. For each zone in the system at each system timestep, the manager calculates a supply air temperature that will meet the zone cooling load at the maximum zone supply air flow rate. The lowest of the possible supply air temperatures becomes the new supply air temperature setpoint, subject to minimum and maximum supply air temperature constraints. The resulting temperature setpoint is the highest supply air temperature that will meet the cooling requirements of all the zones. When compared to a fixed cooling supply air temperature setpoint, this strategy minimises zone reheat coil energy (or overcooling) and central chiller energy consumption (if the chilled water temperature is also reset) at the cost of possible increased fan energy.

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum setpoint temperature

This is the minimum allowed setpoint temperature (in °C or °F). If the calculated setpoint temperature is less than this minimum, the setpoint is set to the minimum.

Maximum setpoint temperature

This is the maximum allowed setpoint temperature (in °C or °F). If the calculated setpoint is greater than this value the setpoint is set to the maximum. This value is also used as the setpoint temperature when none of the zones have a cooling load.

3-Coldest Setpoint Manager

Coldest setpoint managers are available only in Generic and Dual Duct air loops. The coldest setpoint manager is used in dual duct systems to reset the setpoint temperature of the air in the heating supply duct. Usually it is used in conjunction with a Warmest Setpoint Manager resetting the temperature of the air in the cooling supply duct. For each zone in the system at each system timestep, the manager calculates a supply air temperature that will meet the zone heating load at the maximum zone supply air flow rate. The highest of the possible supply air temperatures becomes the new supply air temperature setpoint, subject to minimum and maximum supply air temperature constraints. The resulting temperature setpoint is the lowest supply air temperature that will meet the heating requirements of all the zones. When compared to a fixed heating supply air temperature setpoint, this strategy minimises central boiler energy consumption (if the hot water temperature is also reset or there are variable speed pumps) at the cost of possible increased fan energy (if there is variable volume control in the air system).

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum setpoint temperature

This is the minimum allowed setpoint temperature (in °C or °F). If the calculated setpoint temperature is less than this minimum, the setpoint is set to the minimum. This value is also used as the setpoint temperature when none of the zones have a heating load.

Maximum setpoint temperature

This is the maximum allowed setpoint temperature (in °C or °F). If the calculated setpoint is greater than this value the setpoint is set to the maximum.

4-Multi-Zone Minimum Humidity Average Setpoint Manager

This setpoint manager allows for controlling the minimum humidity level in multiple zones served by a central forced air HVAC system (air loop). This setpoint manager, together with humidistats which must be defined for the controlled zones, detects the air humidity level in multiple controlled zones served by the HVAC air loop and uses air/moisture mass balances to calculate an average supply air humidity ratio needed to maintain the zone relative humidity levels near their respective humidifying setpoints. The calculated humidity ratio is then used as the minimum humidity ratio setpoint for the designated setpoint node(s). A humidification component (e.g. humidifier) placed upstream of the setpoint node can then use the minimum humidity ratio setpoint to control humidification rate.

Note: This setpoint manager requires that humidistats are specified for the controlled zones.

Minimum setpoint humidity ratio

This is the minimum humidity ratio (kg/kg) that is allowed by this setpoint manager. If the calculated setpoint humidity ratio is less than this value, then the setpoint is set to this minimum value.

Maximum setpoint humidity ratio

This is the maximum humidity ratio (kg/kg) that is allowed by this setpoint manager. If the calculated setpoint humidity ratio is greater than this value, then the setpoint is set to this maximum value.

5-Multi-Zone Maximum Humidity Average Setpoint Manager

This setpoint manager allows for controlling the maximum humidity level in multiple zones served by a central forced air HVAC system (air loop). This setpoint manager, together with humidistats which must be defined for the controlled zones, detects the air humidity level in multiple controlled zones served by the HVAC air loop and uses air/moisture mass balances to calculate an average supply air humidity ratio needed to maintain the

zone relative humidity levels near their respective dehumidifying setpoints. The calculated humidity ratio is then used as the maximum humidity ratio setpoint for the designated setpoint node(s). A dehumidification component (e.g. water cooling coil) placed upstream of the setpoint node can then use the maximum humidity ratio setpoint to control its moisture removal rate.

In the case of a chilled water coil which is used for both temperature and high humidity control, this setpoint manager works in conjunction with the coil controller to determine the supply air temperature required to meet both the temperature (sensible) and humidity (latent) load in the control zone.

Note: This setpoint manager requires that humidistats are specified for the controlled zones.

Minimum setpoint humidity ratio

This is the minimum humidity ratio (kg/kg) that is allowed by this setpoint manager. If the calculated setpoint humidity ratio is less than this value, then the setpoint is set to this minimum value.

Maximum setpoint humidity ratio

This is the maximum humidity ratio (in kg/kg) that is allowed by this set point manager. If the calculated setpoint humidity ratio is greater than this value, then the setpoint is set to this maximum value.

6-Multi-Zone Humidity Minimum Setpoint Manager

This setpoint manager allows for controlling the minimum humidity level in multiple zones served by a central forced air HVAC system (air loop). This setpoint manager, together with humidistats which must be defined for the controlled zones, detects the air humidity level in multiple controlled zones served by the HVAC air loop and uses air/moisture mass balances to calculate a supply air minimum humidity ratio based on a zone with the critical humidification requirement (i.e., a zone with the highest humidity ratio setpoint) to maintain the zone relative humidity levels near their respective humidifying setpoints. The calculated humidity ratio is then used as the minimum humidity ratio setpoint for the designated setpoint node(s). A humidification component (e.g. humidifier) placed upstream of the setpoint node can then use the minimum humidity ratio setpoint to control humidification rate. If a humidistat is defined for only one of the zones served by the air loop then this setpoint manager behaves in the same way as the Single Zone Humidity Minimum setpoint manager.

Note: This setpoint manager requires that humidistats are specified for the controlled zones.

Minimum setpoint humidity ratio

This is the minimum humidity ratio (kg/kg) that is allowed by this setpoint manager. If the calculated setpoint humidity ratio is less than this value, then the setpoint is set to this minimum value.

Maximum setpoint humidity ratio

This is the maximum humidity ratio (kg/kg) that is allowed by this setpoint manager. If the calculated setpoint humidity ratio is greater than this value, then the setpoint is set to this maximum value.

7-Multi-Zone Humidity Maximum Setpoint Manager

This setpoint manager allows for controlling the maximum humidity level in multiple zones served by a central forced air HVAC system (air loop). This setpoint manager, used in conjunction with one or more Zone humidistat objects, detects the air humidity level in multiple controlled zones served by the HVAC air loop and uses air/moisture mass balances to calculate the maximum supply air humidity ratio based on a zone with the critical dehumidification requirement (i.e., a zone with the lowest humidity ratio setpoint) to maintain the zone relative humidity levels near their respective dehumidifying setpoints. The calculated humidity ratio is then used as the maximum humidity ratio setpoint for the designated setpoint node(s). A dehumidification component (e.g., desiccant dehumidifier) placed upstream of the setpoint node can then use the maximum humidity ratio setpoint to control its moisture removal rate.

The use of this object requires that a Zone Humidistat object be specified with a dehumidifying relative humidity schedule for one or more controlled zones served by the HVAC air loop. If the Zone Humidistat is defined for only one of the zones served by the air loop then this Setpoint Manager performs as a [Single zone humidity maximum setpoint manager](#).

In the case of a chilled water coil which is used for both temperature and high humidity control, this setpoint manager works in conjunction with the controller associated with the [Cooling coil - water](#) object to determine the supply air temperature required to meet both the temperature (sensible) and dehumidification (latent) load in the control zone.

See object Controller:WaterCoil in the EnergyPlus Engineering Reference for a detailed discussion of how this is achieved.

Minimum setpoint humidity ratio

The minimum humidity ratio (in kg/kg) that is allowed by this setpoint manager. If the calculated setpoint humidity ratio is less than this value, then the setpoint is set to this minimum value. The default value is 0.008 kg/kg.

Maximum setpoint humidity ratio

The maximum humidity ratio (in kg/kg) that is allowed by this set point manager. If the calculated setpoint humidity ratio is greater than this value, then the setpoint is set to this maximum value. The default value is 0.015 kg/kg.

8-Single Zone Humidity Minimum Setpoint Manager

The Single Zone Minimum Humidity Setpoint Manager allows the control of a single zone minimum humidity level. This setpoint manager detects the humidity level in a control zone and, using air and moisture mass balance, calculates the supply air humidity ratio needed to maintain the zone relative humidity at or above a given setpoint. The calculated supply air humidity ratio is then used as the setpoint for the designated setpoint node. A humidifier component placed upstream of the setpoint node can then use the humidity ratio setpoint to control its moisture addition rate.

Note: This setpoint manager requires that a humidistat be specified for the controlled zone.

Control zone

Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone for humidity control.

9-Single Zone Humidity Maximum Setpoint Manager

The Single Zone Maximum Humidity Setpoint Manager allows the control of a single zone maximum humidity level. This setpoint manager, together with a humidistat which must be defined for the controlled zone, detects the air humidity level in a single control zone and uses air/moisture mass balances to calculate the supply air humidity ratio needed to maintain the zone relative humidity at or below a given setpoint. The calculated supply air humidity ratio is then used as the setpoint for the designated setpoint node. A dehumidification component placed upstream of this node can then use the humidity ratio setpoint to control its moisture removal rate (e.g. water cooling coil).

In the case of a chilled water coil which is used for both temperature and high humidity control, this setpoint manager works in conjunction with the coil controller to determine the minimum supply air temperature required to meet both the temperature (sensible) and humidity (latent) load in the control zone.

Note: This setpoint manager requires that a humidistat be specified for the controlled zone.

Control zone

Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone for humidity control.

10-Outdoor Air Reset Setpoint Manager

The Outdoor Air Reset Setpoint Manager sets the supply temperature according to the outdoor air temperature using a reset rule. The reset rule is determined by 2 points: the supply setpoint temperature at the outdoor high temperature (SATO_H) and the supply setpoint temperature at the outdoor low temperature (SATO_L). If the outdoor temperature is above the outdoor high temperature, the supply temperature is set to SATO_H. If the outdoor temperature is below the outdoor low temperature, the supply temperature is set to SATO_L. If the outdoor temperature is between the outdoor high and outdoor low temperatures, the supply air temperature is linearly interpolated between SATO_H and SATO_L.

An optional second reset rule may be defined.

Note: Normally SATO_H will be lower than SATO_L in which case EnergyPlus interpolates to calculate the supply temperature. If SATO_H is higher than SATO_L then EnergyPlus sets the supply air temperature to the average of the SATO_H and SATO_L.

The default Outdoor air reset settings for various loop types are:

Loop Type	Outdoor Low Temp (°C)	Setpoint for Outdoor Low Temp (°C)	Outdoor High Temp (°C)	Setpoint for Outdoor High Temp (°C)	Comment
Air loop	0	16	15	12	
Hot water loop	5	80	15	60	
Chilled water loop	15.56	12.22	26.67	6.67	ASHRAE 90.1 Appendix G
Condenser loop	0	33	15	29	
DHW loop	5	60	15	50	

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Setpoint at outdoor low temperature

This is the setpoint (°C or °F) at the outdoor low temperature for the first reset rule.

Outdoor low temperature

This is the outdoor air low temperature (°C or °F) for the first supply air temperature reset rule. Generally, at this outdoor air temperature the supply temperature is at its maximum.

Setpoint at outdoor high temperature

This is the setpoint temperature (°C or °F) at the outdoor high temperature for the first reset rule.

Outdoor high temperature

This is the outdoor air high temperature (°C or °F) for the first supply air temperature reset rule. Generally, at this outdoor air temperature the supply temperature is at its minimum.

Second reset rule

This check-box allows you to define a second reset rule.

Second reset schedule

This is the schedule whose values indicate which reset rule to use for each time over the year. Schedule values of 1 indicate that the first reset rule will be used. Schedule values of 2 select the second reset rule.

Important Note: The schedule selected here must only include values of 1 or 2.

Second reset rule setpoint at outdoor low temperature

This is the setpoint temperature (°C or °F) at the outdoor low temperature for the second reset rule.

Second reset rule outdoor low temperature

This is the outdoor air low temperature (°C) for the second supply air temperature reset rule. Generally, at this outdoor air temperature the supply temperature is at its maximum.

Second reset rule setpoint at outdoor high temperature

This is the setpoint temperature (°C or °F) at the outdoor high temperature for the second reset rule.

Second reset rule outdoor high temperature

This is the outdoor air high temperature (°C) for the second supply air temperature reset rule. Generally, at this outdoor air temperature the supply temperature is at its minimum.

11-Single Zone Heating Setpoint Manager

The Single Zone Heating Setpoint Manager allows components to be controlled based on the load required to meet the zone heating setpoint. This setpoint manager detects the control zone load to meet the current heating setpoint, zone inlet node flow rate, and zone node temperature, and calculates a setpoint temperature for the supply air that will satisfy the zone heating load for the control zone. This setpoint manager creates a variable temperature system and is normally applied only to constant volume air loops.

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum supply air temperature

The minimum supply air temperature (in °C or °F) that is allowed for this system (as set by this setpoint manager).

Maximum supply air temperature

The maximum supply air temperature (in °C or °F) that is allowed for this system (as set by this setpoint manager)

Control zone

Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone for single zone heating control.

Combining 11-Single Zone Heating and 12-Single Zone Cooling Setpoint Managers

Note: A variable temperature, constant volume heating and cooling system can be simulated by combining a **11-Single Zone Heating Setpoint Manager** with a **12-Single Zone Cooling Setpoint Manager** together on the supply side of a loop.

In this case, the temperature range setpoints entered for both heating and cooling SPMs should be identical.

The way this is done depends on whether the AHU has a blow-through or draw-through fan configuration.

Blow-through: When the fan is placed upstream of the heating and cooling coils in the AHU then the **11-Single Zone Heating Setpoint Manager** is placed directly after the heating coil, between the heating coil and the cooling coil and the **12-Single Zone Cooling Setpoint Manager** is placed downstream of the AHU.

Draw-through: When the fan is placed downstream of the heating and cooling coils in the AHU then both the **11-Single Zone Heating Setpoint Manager** and **12-Single Zone Cooling Setpoint Manager** are placed downstream of the AHU.

12-Single Zone Cooling Setpoint Manager

The Single Zone Cooling Setpoint Manager allows components to be controlled based on the load required to meet the zone cooling setpoint. This setpoint manager detects the control zone load to meet the current cooling setpoint, zone inlet node flow rate, and zone node temperature, and calculates a setpoint temperature for the supply air that will satisfy the zone cooling load for the control zone. This setpoint manager creates a variable temperature system and is normally applied only to constant volume air loops.

Note: See also the note above on [Combining 11-Single Zone Heating and 12-Single Zone Cooling Setpoint Managers](#).

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum supply air temperature

The minimum supply air temperature (in °C or °F) that is allowed for this system (as set by this setpoint manager).

Maximum supply air temperature

The maximum supply air temperature (in °C or °F) that is allowed for this system (as set by this setpoint manager).

Control zone

Click on the <Select zone> label and then click on the displayed ellipsis button to bring up the zone selector dialog. Select the zone for single zone cooling control.

13-Multi Zone Cooling Average Setpoint Manager

This SPM is commonly used in conjunction with a multi-zone heating average SPM for systems where both heating and cooling is to be provided by the central AHU with no zone reheat. It is used to establish a supply

air temperature setpoint for a central forced air HVAC system (air loop) based on the predicted sensible cooling loads and actual supply air mass flow rates for all zones served by the system. For all controlled zones in the air loop (i.e., zones with a thermostat object), the setpoint manager calculates an average supply air temperature that will meet the zone cooling loads based on the actual zone supply air mass flow rates (lagged by one time step). The calculated setpoint temperature is subject to the minimum and maximum setpoint temperature constraints specified by the user. When compared to a fixed cooling supply air temperature setpoint, this strategy may reduce zone reheat coil energy (or overcooling) and central chiller energy consumption (if the chilled water temperature is also reset) at the cost of possible increased fan energy.

Note: this SPM is intended for use on constant volume air loops only

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum setpoint temperature

The minimum supply air temperature (in °C or °F) that is allowed for this system. If the calculated setpoint temperature is less than this minimum, the setpoint is set to the minimum. The default value is 12°C.

Maximum setpoint temperature

The maximum supply air temperature (in °C or °F) that is allowed for this system. If the calculated setpoint temperature is greater than this value, the setpoint is set to this maximum value. The default value is 18°C.

Note: The same notes for combining single zone heating and cooling SPMs apply to the multizone equivalents. See [Combining 11-Single Zone Heating and 12-Single Zone Cooling Setpoint Managers](#).

14-Multi Zone Heating Average Setpoint Manager

This setpoint manager is used to establish a supply air temperature setpoint for a central forced air HVAC system (air loop) based on the predicted sensible heating loads and actual supply air mass flow rates for all zones served by the system. For all controlled zones in the air loop (i.e., zones with a thermostat object), the setpoint manager calculates an average supply air temperature that will meet the zone heating loads based on the actual zone supply air mass flow rates (lagged by one time step). The calculated setpoint temperature is subject to the minimum and maximum setpoint temperature constraints specified by the user. When compared to a fixed heating supply air temperature setpoint, this strategy may reduce central boiler energy consumption (if the hot water temperature is also reset or there are variable speed pumps) at the cost of possible increased fan energy (if there is variable volume control for the air system).

Note: this SPM is intended for use on constant volume air loops only

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum setpoint temperature

The minimum supply air temperature (in °C or °F) that is allowed for this system. If the calculated setpoint temperature is less than this minimum, the setpoint is set to the minimum. The default value is 20°C.

Maximum setpoint temperature

The maximum supply air temperature (in °C or °F) that is allowed for this system. If the calculated setpoint temperature is greater than this value, the setpoint is set to this maximum value. The default value is 50°C.

Note: The same notes for combining single zone heating and cooling SPMs apply to the multizone equivalents. See [Combining 11-Single Zone Heating and 12-Single Zone Cooling Setpoint Managers](#).

15-Follow Ground Temperature

This setpoint manager provides a temperature setpoint on a system node that is derived from a current ground temperature. The ground temperatures are specified in at site level and used during the simulation. This setpoint manager is primarily intended for condenser or plant loops using some type of ground heat exchanger.

When this SPM is selected the following other fields should be defined.

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum setpoint temperature

The minimum supply air temperature (in °C or °F) that is allowed for this system. If the calculated setpoint temperature is less than this minimum, the setpoint is set to the minimum. The default value is 11°C.

Maximum setpoint temperature

The maximum supply air temperature (in °C or °F) that is allowed for this system. If the calculated setpoint temperature is greater than this value, the setpoint is set to this maximum value. The default value is 16°C.

Reference ground temperature

This setting specifies the type of ground temperature to be used by the setpoint manager. There are four options:

- 1-Deep** - uses the [Deep monthly temperatures](#) defined at site level.
- 2-Shallow** - uses the [Shallow monthly temperatures](#) defined at site level.
- 3-Building surface** - uses the [Monthly ground temperatures](#) defined at site level.

Tip: Generally the deep ground temperatures are the most useful for a plant loop serving a vertical borehole ground heat exchanger.

Offset temperature difference

This data provides a temperature offset that will be applied to the value of the ground temperature. If this value is zero, and the limits are met, then the resulting setpoint will be exactly the same as the ground temperature. The sign convention is that a positive value here will increase the resulting setpoint to higher than the ground temperature.

16-Follow Outdoor Air Temperature

This setpoint manager is used to place a temperature setpoint on a system node that is derived from the current outdoor air environmental conditions. The outdoor air conditions are obtained from the weather information during the simulation.

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Minimum setpoint temperature

This is the minimum allowed setpoint temperature (in °C or °F). If the calculated setpoint temperature is less than this minimum, the setpoint is set to the minimum.

Maximum setpoint temperature

This is the maximum allowed setpoint temperature (in °C or °F). If the calculated setpoint is greater than this value the setpoint is set to the maximum. This value is also used as the setpoint temperature when none of the zones have a cooling load.

Offset temperature difference

This is a temperature offset that will be applied to the value of the reference temperature (outdoor air wetbulb/drybulb). If this value is zero, and the limits are met, then the resulting setpoint will be exactly the same as the outdoor air wetbulb/drybulb temperature. The sign convention is that a positive value here will increase the resulting setpoint to higher than the outdoor air wetbulb/drybulb.

17-Dual Setpoint

This setpoint manager places a high and low schedule value on one or more nodes. The input consists of the Setpoint Manager name, the control variable, the high and low set point schedule names.

The Dual Setpoint manager can be used with Plant loops only and can be used to create a mixed water loop to give both heating and cooling setpoints on the same loop. In this case settings on the [Plant Equipment Operation](#) tab of the Plant loop dialog define which equipment has priority depending on whether there is a heating or cooling load.

The Dual Setpoint manager should be used together with the **2-DualSetPoint** [Plant loop demand calculation scheme](#) option.

When this SPM is selected the following other fields should be defined.

Control variable

The type of variable that will be controlled. There is only one choice for this type of setpoint manager: **1-Temperature**.

Low setpoint schedule

Select the schedule that contains the low setpoint values. The schedule value for each time period is the low setpoint.

High setpoint schedule

Select the schedule that contains the high setpoint values. The schedule value for each time period is the high setpoint.

DHW

Domestic Hot Water (DHW) is provided in DesignBuilder Detailed HVAC through [DHW loops](#), [Water heaters](#), [Water outlet groups](#) and [Water outlet](#) components.

DHW loops are added at the HVAC System level. On the demand side of the DHW sub loops, Water outlet groups can be added. However a default Water outlet group is provided with the initial DHW loop so in many cases you will not need to add a new Water outlet group. Instead simply navigate to the default Water outlet

group and add one or more Water outlet components to define hot water consumption and any heat transfer to a building zone.

If you are working with the default **1-Simple HVAC** [Detailed HVAC Activity data](#) model option then simply associate the zones to be connected on the Zones served tab of the dialog. In many cases this would be all occupied zones in the building. In this case the amount of DHW consumed is calculated as a function of the activity in the zone. For example, there is a demand assumed to arise from the occupants of an office for activities such as washing hands and washing up cups. This demand is associated with the office rather than the toilet or tea room. So the demand from each space needs to be assigned to a DHW system even if the DHW delivery (tap, shower etc) is not present in the space.

On the other hand, when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#) model option it would normally make more sense to model the actual water draw off rates in the rooms where hot water is consumed (toilet, tea room, shower room etc).

Tip: For consistency with Simple HVAC use the default **1-Simple HVAC** [Detailed HVAC Activity data](#) model option then create a single DHW loop, use the default [Water outlet group](#) and [Water outlet](#) provided with the [DHW loop](#) and select all zones in the building on the [Zones served](#) tab of the Water outlet.

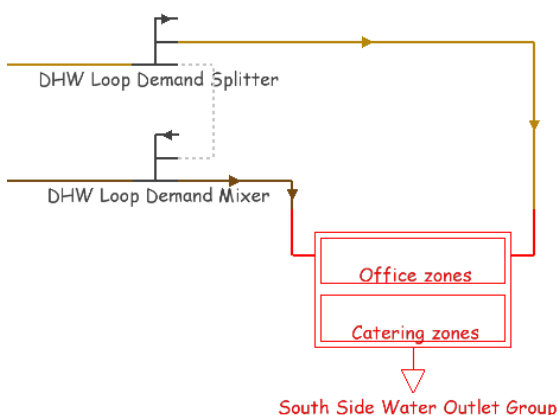
The DHW-related Detailed HVAC components are:

- [Water Outlet Group](#)
- [Water Outlet](#)
- [Water Heater Data](#)

Water Outlet Group

The Water outlet group component is used to group Water outlet components and connect them to a Water heater. Water outlet groups are placed on the demand sub loop of a DHW loop as shown in the screenshot above.

To add new [Water outlets](#) to a Water outlet group first navigate to the Water outlet group and then use the Add water outlet toolbar option. You can edit the details for each Water outlet by right clicking on it in the Water outlet group screen image and selecting the Edit drop down menu option.



Drain Water Heat Exchanger

Drain water heat exchanger type

The heat exchanger type to be used for drain water heat recovery. The options are:

1-None indicating no heat recovery is to be simulated.

2-Ideal sets a heat exchanger effectiveness of 1.0.

3-Counter flow where effectiveness is calculated dynamically

4-Cross flow where effectiveness is calculated dynamically.

One popular type of drain water heat exchanger is the Gravity-Film Heat Exchanger (GFX). The GFX is most closely approximated with the **3-Counter flow** option, but keep in mind that the UA varies more with flow rate for the GFX than for most traditional heat exchangers.

Drain water heat exchanger destination

The heat exchanger configuration to be used for drain water heat recovery. The configuration determines where the heat will be used after it is recovered. The options are:

- **1-Plant** which indicates that the all of the recovered heat will be sent to the return flow at the outlet node of the [Water outlets](#) to preheat the make-up cold water from the water mains.
- **2-Equipment**, indicating that the all of the recovered heat will be used to preheat the cold water flow side of the [Water outlets](#) listed in this Water outlet group
- **3-Plant and Equipment**, indicating that the recovered heat will be divided between the plant and the [Water outlets](#), as described above. This is the only option where the flow rates are equal in the drain and the heat exchanger.

Drain water heat exchanger UA

The UA is the heat transfer coefficient (in W/K or Btu/h-F) for the heat exchanger. It is the product of U, the overall heat transfer coefficient, and A, the heat exchanger surface area.

Water Outlet

The Water outlet component defines hot water consumption, delivery temperatures as well as any heat conduction and latent gains to zones. One or more Water outlets can be associated with a previously defined [Water outlet group](#).

The way that Water outlets works in very different depending on whether you are using the Simple or Detailed HVAC [Detailed HVAC Activity data](#) setting.

Simple HVAC Activity data

When using Simple HVAC activity data one Water outlet component can be used to serve multiple zones very easily simply by selecting the zones on the [Zones served](#) tab. In this case DHW demand data from [Activity](#) and [HVAC](#) model data tabs is used to define the total hot water flow rate and schedule of consumption for the zones referenced in the Zones served tab.

Detailed HVAC Activity data

When using Detailed HVAC activity data option the Water outlet hot water consumption peak flow rate and time varying schedule are defined independently of any model data DHW settings and you must provide the peak hot water flow rate and schedule of operation for the Water outlet. This gives you the flexibility to define DHW consumption in various ways:

- **Use one Water outlet for multiple zones.** This approach can be used in cases where the usage pattern (schedule) is the same in each of the zones. In this case the total peak hot water flow rate for all the zones to be included can be calculated in a spreadsheet (for example) and the peak flow for all zones entered here on the Water outlet dialog.
- **Use one Water outlet for each zone.** This approach might be preferred when either the patterns of usage (schedules) for each zone is different and needs to be defined separately or when you prefer a more explicit separation of zones for a clearer representation in the model, rather than lumping all flows into a single Water outlet.
- **Use one Water outlet for each physical water outlet in the building**, e.g. one Water outlet for each hot water tap, shower etc. In this case you would need to calculate expected peak flow and schedules of usage for each outlet.

In all of the above cases it is usually more work to set up DHW input data when using the Detailed HVAC activity data than for the Simple HVAC activity alternative.

General

Name

Enter a unique name for the water outlet.

Target temperature schedule

Select a schedule defining the temperature of the water delivered from taps etc.

Flow Rate

Note: The data under the **Flow Rate** header is only shown when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#) setting.

Peak flow rate

The peak demanded hot water flow rate (in m³/s or gal/min). This value is multiplied by the **Flow rate fraction schedule** (below) to determine the actual volumetric flow rate at any time in the simulation. This value is only required when using the [Detailed HVAC data](#) model option.

Flow rate fraction schedule

Select the schedule specifying the flow rate fraction relative to the **Peak flow rate** (above). This value is only required when using the [Detailed HVAC data](#) model option.

Heat Gain To Zone

Heat gain to zone

Check this option if you wish to model sensible and/or latent heat transfer from the Water outlet to a zone in the building model.

Zone

Select the zone to receive sensible and/or latent loads from this water outlet.

Sensible Fraction Schedule Name

Select the schedule specifying the fraction of the maximum possible sensible heat gain (based on inlet water conditions and ambient zone conditions) that is added to the zone. Select schedule **Off** for no sensible load.

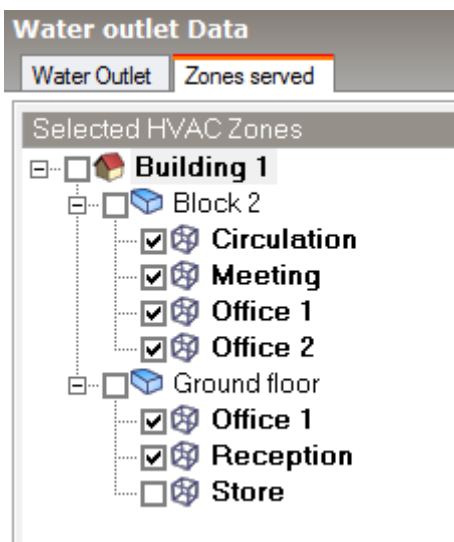
Latent Fraction Schedule Name

Select the schedule specifying the fraction of the maximum possible latent heat gain (based on inlet water conditions and ambient zone conditions) that is added to the zone. Select schedule **Off** for no latent load.

Zones Served

The **Zones served** tab on the Water outlet dialog is only visible when using the default **1-Simple HVAC** [Detailed HVAC Activity data](#) setting. It allows you to select the zones served by this Water outlet. For these zones the actual hot water consumption rates used in the simulation are generated based on the [DHW consumption rate](#) set on the Activity tab and the [DHW Operation schedule](#) on the HVAC tab in the same way as for Simple HVAC.

Check the boxes of the zones to be served by this Water outlet as shown below.



Tip: You can use the checkboxes at building and block levels as a fast way to select/deselect all of the available zones in the building or block.

Note: Any zones selected on the **Zones served** tab will not consume hot water in the simulation if their [DHW consumption](#) rate is 0. This means that you can select all zones in a building without concern for generating DHW demand for unoccupied zones.

Water Heater



WaterHeater:Mixed
WaterHeater:Stratified
WaterHeater:Sizing

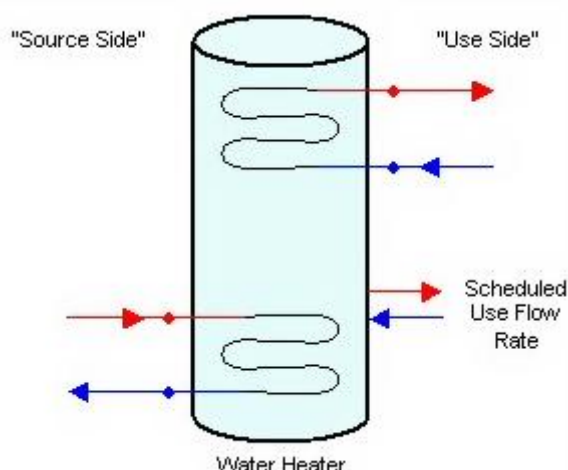
Used in:

Supply side of [DHW Loops](#)

Water heaters are DHW loop components and can be used for simulating many types of water heaters and storage tanks, including gas and electric residential water heaters, and a variety of large commercial water heaters. They model stand-alone operation, on- and off-cycle parasitic loads and thermal losses to the zone and instantaneous/tankless water heaters. Typical applications include domestic hot water heating, low-temperature radiant space heating, and energy storage for solar hot water systems or waste heat recovery.

Water heaters have an inlet node and outlet node on the “source side” and an inlet node and outlet node on the “use side”. The source side typically draws cold water from the tank and returns warmer water, for instance, from solar hot water systems or waste heat recovery systems. The use side typically draws hot water from the tank and returns cooler water from the cold water supply mains or from the outlet of a heating system. The distinction between source and use sides is merely a convenience for reporting. They can actually be used interchangeably.

For a water heater that is indirectly heated (e.g. with a separate boiler), the source side can be used to provide remotely heated water to the tank. The source side is configured to operate as a component on the demand side of a plant loop. The design flow rate through the source side can be set by the user or autosized. The water heater input includes an additional design parameter that describes how rapidly the tank can recover.



Water heater configuration

There are currently two types of water heater:

- [1-Mixed](#).
- [2-Stratified](#).

The **1-Mixed** type simulates a well-mixed, single-node water tank. The **2-Stratified** type simulates a stratified, multi-node water tank. Both water heaters can be appropriate for simulating many types of water heaters and storage tanks, including gas and electric residential water heaters, and a variety of large commercial water heaters. Both types share similar features, such as on- and off-cycle parasitic loads and thermal losses to the zone. However, each type has its advantages which may make one more appropriate than the other depending on the application.

Advantages of **1-Mixed** type:

- can simulate instantaneous/tankless water heaters
- requires less input than the stratified tank
- faster execution time than the stratified tank
- adequate for modeling gas water heaters with no source connections.

Advantages of **2-Stratified** type:

- better modelling of electric water heaters with two heating elements
- better modelling of thermal storage applications which rely on stratification to improve heat transfer performance.

Standard Ratings

The EIO file reports the industry standard ratings of Recovery Efficiency and Energy Factor for water heaters. The rating method is based on the GAMA and 10CFR430 test procedures. Under certain input parameters, the rating method will not succeed and a warning message will be generated. Problems occur when inputs do not allow the tank to recover to the setpoint temperature within the test period. This can occur if the maximum heater capacity is undersized, or if the deadband temperature difference is large enough that the first draw of the test does not trigger the heater to come on. In either case, the Recovery Efficiency test will not compute properly because recovery to the setpoint was not achieved.

Standard ratings for storage-only water tanks (Heater Maximum Capacity = 0) cannot be calculated and do not report anything in the EIO file.

Water Heater Outputs

The following Water heater output variables are available:

HVAC,Average,Water Heater Tank Temperature [C]

HVAC,Average,Water Heater Final Tank Temperature [C]
 HVAC,Average,Water Heater Loss Rate [W]
 HVAC,Sum,Water Heater Loss Energy [J]
 HVAC,Average,Water Heater Use Flow Rate [kg/s]
 HVAC,Average,Water Heater Use Inlet Temperature [C]
 HVAC,Average,Water Heater Use Outlet Temperature [C]
 HVAC,Average,Water Heater Use Rate [W]
 HVAC,Sum,Water Heater Use Energy [J]
 HVAC,Average,Water Heater Source Flow Rate [kg/s]
 HVAC,Average,Water Heater Source Inlet Temperature [C]
 HVAC,Average,Water Heater Source Outlet Temperature [C]
 HVAC,Average,Water Heater Source Rate [W]
 HVAC,Sum,Water Heater Source Energy [J]
 HVAC,Average,Water Heater Off-Cycle Parasitic Heat Rate To Tank [W]
 HVAC,Sum,Water Heater Off-Cycle Parasitic Heat Energy To Tank [J]
 HVAC,Average,Water Heater On-Cycle Parasitic Heat Rate To Tank [W]
 HVAC,Sum,Water Heater On-Cycle Parasitic Heat Energy To Tank [J]
 HVAC,Average,Water Heater Total Demand Rate [W]
 HVAC,Sum,Water Heater Total Demand Energy [J]
 HVAC,Average,Water Heater Heating Rate [W]
 HVAC,Sum,Water Heater Heating Energy [J]
 HVAC,Average,Water Heater Unmet Demand Rate [W]
 HVAC,Sum,Water Heater Unmet Demand Energy [J]
 HVAC,Average,Water Heater Venting Rate [W]
 HVAC,Sum,Water Heater Venting Energy [J]
 HVAC,Average,Water Heater Net Heat Transfer Rate [W]
 HVAC,Sum,Water Heater Net Heat Transfer Energy [J]
 HVAC,Sum,Water Heater Cycle On Count []
 HVAC,Average,Water Heater Runtime Fraction []
 HVAC,Average,Water Heater Part Load Ratio []
 HVAC,Average,Water Heater <Fuel Type> Consumption Rate[W]
 HVAC,Sum,Water Heater <Fuel Type>Consumption [J]
 HVAC,Average,Water Heater Off-Cycle Parasitic <Fuel Type> Consumption Rate[W]
 HVAC,Sum,Water Heater Off-Cycle Parasitic <Fuel Type> Consumption [J]
 HVAC,Average,Water Heater On-Cycle Parasitic <Fuel Type> Consumption Rate[W]
 HVAC,Sum,Water Heater On-Cycle Parasitic <Fuel Type> Consumption [J]
 HVAC,Average,Water Heater Water Consumption Rate [m3/s]
 HVAC,Sum,Water Heater Water Consumption [m3]

Water Heater Tank Temperature [C]

The average water tank temperature.

Water Heater Final Tank Temperature [C]

The final water tank temperature at the end of the system timestep. If reported at the "Detailed" interval, this report variable can be used to verify an exact energy balance on the water heater. Also see the report variable: Water Heater Net Heat Transfer Energy.

Water Heater Loss Rate [W]

The average heat loss rate due to the off- and on-cycle loss coefficients to the ambient temperature.

Water Heater Loss Energy [J]

The heat loss energy due to the off- and on-cycle loss coefficients to the ambient temperature.

Water Heater Use Flow Rate [kg/s]

The use side mass flow rate. If stand-alone, this is the scheduled use flow rate.

Water Heater Use Inlet Temperature [C]

The inlet temperature on the use side.

Water Heater Use Outlet Temperature [C]

The outlet temperature on the use side.

Water Heater Use Rate [W]

The average heat transfer rate between the use side water and the tank water.

Water Heater Use Energy [J]

The heat transfer energy between the use side water and the tank water.

Water Heater Source Flow Rate [kg/s]

The source side mass flow rate. If in stand-alone operation, this is 0.

Water Heater Source Inlet Temperature [C]

The inlet temperature on the source side.

Water Heater Source Outlet Temperature [C]

The outlet temperature on the source side.

Water Heater Source Rate [W]

The average heat transfer rate between the source side water and the tank water.

Water Heater Source Energy [J]

The heat transfer energy between the source side water and the tank water.

Water Heater Off-Cycle Parasitic Heat Rate To Tank [W]

The average heat gain rate to the tank water due to off-cycle parasitics.

Water Heater Off-Cycle Parasitic Heat Energy To Tank [J]

The heat gain energy to the tank water due to off-cycle parasitics.

Water Heater On-Cycle Parasitic Heat Rate To Tank [W]

The average heat gain rate to the tank water due to on-cycle parasitics.

Water Heater On-Cycle Parasitic Heat Energy To Tank [J]

The heat gain energy to the tank water due to on-cycle parasitics.

Water Heater Total Demand Rate [W]

The average heating rate demanded to maintain the setpoint temperature.

Water Heater Total Demand Energy [J]

The heating energy demanded to maintain the setpoint temperature.

Water Heater Heating Rate [W]

The average heating rate supplied by the heater element or burner.

Water Heater Heating Energy [J]

The heating energy supplied by the heater element or burner.

Water Heater Unmet Demand Rate [W]

The average heating rate unmet by the heater element or burner. The difference between the Total Demand Rate and the Heating Rate.

Water Heater Unmet Demand Energy [J]

The heating energy unmet by the heater element or burner. The difference between the Total Demand Energy and the Heating Energy.

Water Heater Venting Rate [W]

The average venting rate to keep the tank below the Maximum Temperature Limit.

Water Heater Venting Energy [J]

The venting energy to keep the tank below the Maximum Temperature Limit.

Water Heater Net Heat Transfer Rate [W]

The average net heat transfer rate when considering all losses and gains.

Water Heater Net Heat Transfer Energy [J]

The net heat transfer energy when considering all losses and gains.

Water Heater Cycle On Count []

The number of times that the heater turned on in the time period.

Water Heater Runtime Fraction []

The fraction of the time period that the heater was running.

Water Heater Part Load Ratio []

The fraction of the Heater Maximum Capacity.

Water Heater <Fuel Type> Consumption Rate [W]

The average fuel consumption rate for the heater element or burner.

Water Heater <Fuel Type> Consumption Energy [J]

The fuel consumption energy for the heater element or burner.

Water Heater Off-Cycle <Fuel Type> Consumption Rate [W]

The average fuel consumption rate for the off-cycle parasitics.

Water Heater Off-Cycle <Fuel Type> Consumption Energy [J]

The fuel consumption energy for the off-cycle parasitics.

Water Heater On-Cycle <Fuel Type> Consumption Rate [W]

The average fuel consumption rate for the on-cycle parasitics.

Water Heater On-Cycle <Fuel Type> Consumption Energy [J]

The fuel consumption energy for the on-cycle parasitics.

Water Heater Water Consumption Rate [m3/s]

The water consumption rate for the use side, if in stand-alone operation.

Water Heater Water Consumption [m3]

The water consumption for the use side, if in stand-alone operation.

Water Heater - Mixed



WaterHeater:Mixed
WaterHeater:Sizing

Used in:

Supply side of [DHW Loops](#)

The **1-Mixed** water heater simulates a well-mixed, single-node water tank. The model solves the differential equation governing the energy balance of the water tank. Within a timestep, conditions are solved separately for when the heater element or burner is “on” (on-cycle) and when it is “off” (off-cycle). This approach allows ambient losses and parasitic loads to be divided into on-cycle and off-cycle effects and accounted for in detail.

For losses to the ambient environment, the ambient air temperature can be taken from a schedule, a zone, or the exterior. When used with a zone, a fraction of the skin losses can be added to the zone heat balance as internal heat gains.

Control options allow the heater to cycle or modulate to meet the load. When cycling, the heater element or burner is either on or off. The heater remains fully on while heating the tank up to the setpoint temperature. When the setpoint is reached, the heater turns off. The heater remains off until the tank temperature falls below the “cut-in” temperature, i.e., the setpoint temperature minus the deadband temperature difference. The heater continuously cycles on and off to maintain the tank temperature within the deadband. Most storage-tank water heaters cycle.

When modulating, the heater power varies between the maximum and minimum heater capacities. The heater stays on as long as the required total demand is above the minimum capacity. Below the minimum capacity, the heater will begin to cycle on and off based on the deadband temperature difference. Equipment is usually designed and rated to avoid this condition. Most tankless/instantaneous water heaters modulate.

See also:

- [Water heater overview](#)
- [Stratified Water heater](#)

General

Name

The name of the Water heater.

Type

Select the type of water heater from the list:

- **1-Mixed.**
- **2-Stratified.**

For a discussion of the applications, advantages and disadvantages of the two water heater types see the [Water heater overview](#) topic.

Tank volume

The volume of the storage tank (in m3 or ft3). This field is autosizable based on the [Sizing data](#). Although this field is allowed to go down to zero, even so-called "tankless" water heaters have some volume of water that is maintained around the heating elements or in the heat exchanger, typically around 0.00379 m3 (1 gallon).

External Heating Plant Connection

External heating plant connection

Check this option if the water heater is to be supplied with heat from a Hot water loop. In this case inlet and outlet connections are included on the bottom of the water heater allowing the connection to be made to the demand side of the Hot water plant loop.

Indirect water heating recovery time

This field is only required when the External heating plant option is checked, i.e. when the water heater is connected to the demand side of a plant loop. It is used to provide a design parameter for autosizing design flow rates. The recovery time is expressed in hours. This is the time that the entire volume of the tank can be heated from 14.4°C to 57.2°C (58°F to 135°F) with an inlet temperature defined as the exit temperature in the associated Plant Sizing object. The default is 1.5 hours. The calculation is based on log-mean temperature difference (LMTD) and includes the heat transfer effectiveness factor entered above.

Temperature Settings

Setpoint temperature schedule

Select the [Schedule](#) that defines the hot water temperature setpoint (in °C or °F). Also known as the "cut-out" temperature.

Deadband temperature difference

The delta temperature difference (in Δ°C or Δ°F) between the setpoint and the "cut-in" temperature at which the heater will turn on. In other words, the "cut-in" temperature is Setpoint – Deadband.

Maximum temperature limit

The temperature (in °C or °F) at which the tank water becomes dangerously hot and is vented through boiling or an automatic safety. The tank temperature will never exceed the maximum. Any extra heat added to the tank is immediately vented. The maximum temperature must be greater than the setpoint temperature at all times.

Ambient Temperature Settings

Ambient temperature indicator

The Ambient temperature indicator specifies how the temperature of the air surrounding the Water heater is to be defined. These options are available:

- **1-Schedule** where the Ambient temperature schedule field defines the ambient temperature.
- **2-Zone** where the zone air temperature of the zone specified by Ambient temperature zone, below, provides the ambient temperature
- **3-Outdoors** where the outdoor dry-bulb air temperature provides the ambient temperature for the water heater.

Ambient temperature schedule

Select the schedule specifying the ambient air temperature around the tank for skin losses. This field is only required if [Ambient temperature indicator](#) is **1-Schedule**.

Ambient temperature zone

The building zone specifying the ambient air temperature around the tank for skin losses. This field is only required if [Ambient temperature indicator](#) is **2-Zone**.

On-Cycle loss coefficient to ambient temperature

The loss coefficient (in W/K or Btu/h-F) to the ambient air temperature. Often this coefficient is identical to the "UA" for skin losses. If the loss effects of the flue are being modelled in the Off-Cycle Loss Coefficient, than this field would have a different value accounting only for the skin losses.

Off-Cycle loss coefficient to ambient temperature

The loss coefficient (in W/K or Btu/h-F) to the ambient air temperature. Often this coefficient is identical to the "UA" for skin losses. However, it can also be used to model the loss effects of the flue in a combustion water heater, in addition to the skin losses.

On-Cycle loss fraction to zone

If the [Ambient temperature indicator](#) is **2-Zone**, this field adds the specified fraction of the on-cycle losses to the zone heat balance as an internal gain.

Off-Cycle loss fraction to zone

If the Ambient temperature indicator is **2-Zone**, this field adds the specified fraction of the off-cycle losses to the zone heat balance as an internal gain.

Internal Heating Element

Internal heating element

If the water heater includes an internal heating element then checking this checkbox provides access to the element settings.

Internal Heating Element Settings

Heater control type

The control type can be one of:

- **1-Cycle** for most storage tank water heaters, or,
- **2-Modulate** for most instantaneous, tank-less water heaters.

Heater maximum capacity

The maximum heat rate (in W or btu/h) that can be supplied to the water and typically the same as the "nominal" capacity.

Note: This field is autosizable however auto-sizing of the heating element will generally only work when the tank sizing [Design mode](#) is set to **1-Peak draw** and the water heater has significant capacity. In this case, the element should be sized using the tank capacity in conjunction with the [Time for tank recovery](#) setting. In a situation where there is an external plant connection, the [Nominal tank volume for autosizing plant connections](#) setting is also used.

Heater minimum capacity

The minimum heat rate (in W or btu/h) that can be supplied to the water. This field is only used when the [Heater control type](#) is **2-Modulate**. If the total demand rate for heating is less than the minimum, even a modulating water heater will begin to cycle.

Heater fuel type

The type of fuel used for heating. The fuel type can be one of:

- **1-Electricity,**
- **2-Natural gas,**
- **3-Propane gas,**
- **4-Fuel oil 1,**
- **5-Fuel oil 2,**
- **6-Coal,**
- **7-Diesel,**
- **8-Gasoline,**
- **9-Steam,**
- **10-District heating.**
- **11-Other fuel 1**
- **12-Other fuel 2**

11-Other fuel 1 and **12-Other fuel 2** can be used for representing biofuels such as biomass and biogas for example.

Heater thermal efficiency

The thermal conversion efficiency from fuel energy to heat energy for the heater element or burner. This is not the same as the overall efficiency of the water heater.

Heater part load factor curve

Select the [Quadratic](#) or [Cubic](#) curve that relates the overall efficiency of the water heater to the runtime fraction (if Control Type **1-Cycle**) or part load ratio (if Control type is **2-Modulate**). This is an additional multiplier applied to the [Heater thermal efficiency](#) above, to compute fuel energy use. The Part load factor curve should not have a value less than 0.1 in the domain from 0 to 1.

Note: If the curve accounts for ambient losses and/or parasitic fuel consumption, these effects should not also be input into the related fields in this object as that would result in double-counting.

On-Cycle Settings

On-Cycle parasitic fuel consumption rate

On-cycle parasitics include parts of the water heater that consume fuel when the heater is on, for example, an induction fan, or stand-by electronic control circuits. The fuel consumption rate (in W or Btu/h) is the total fuel that is consumed by all of the on-cycle parasitics.

On-Cycle parasitic fuel type

The type of fuel used by the on-cycle parasitics. The fuel type can be

- **1-Electricity,**
- **2-Natural gas,**
- **3-Propane gas,**
- **4-Fuel oil #1,**
- **5-Fuel oil #2,**
- **6-Coal,**
- **7-Diesel,**
- **8-Gasoline,**
- **9-Steam, or**
- **10-District heating.**

The fuel type can be the same or different from the Heater Fuel Type.

On-Cycle parasitic heat fraction to tank

The fraction of on-cycle parasitic fuel energy that is converted to heat energy that ends up in the tank water. For example, an induction fan might (maybe) deliver a small fraction of its energy to the tank water for a value of 0.05. Electronic control circuits, on the other hand, do not add any heat to the tank and should be 0.

Off-Cycle Settings

Off-Cycle Parasitic Fuel Consumption Rate

Off-cycle parasitics include parts of the water heater that consume fuel when the heater is off, for example, a pilot light, or stand-by electronic control circuits. The fuel consumption rate (in W or Btu/h) is strictly the total fuel that is consumed by all of the off-cycle parasitics.

Off-Cycle Parasitic Fuel Type

The type of fuel used by the off-cycle parasitics. The fuel type can be one of

- 1-Electricity,
- 2-Natural gas,
- 3-Propane gas,
- 4-Fuel oil #1,
- 5-Fuel oil #2,
- 6-Coal,
- 7-Diesel,
- 8-Gasoline,
- 9-Steam, or
- 10-District heating.

The fuel type can be the same or different from the Heater fuel type.

Off-Cycle parasitic heat fraction to tank

The fraction of off-cycle parasitic fuel energy that is converted to heat energy that ends up in the tank water. For example, a pilot light would deliver most of its heat to the tank water, as long as the thermal conversion efficiency must be taken into account, so perhaps 0.80 is reasonable. Electronic control circuits, on the other hand, do not add any heat to the tank and should be 0.

Use-Side Settings

Use side effectiveness

This field specifies the heat transfer effectiveness between the use side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the use side water and the tank water. If the effectiveness is lower, then the use side outlet water temperature will not be as hot as the tank water, simulating a heat exchanger.

Use side design flow rate

This field is optional and is used to specify the design flow rate through the Use Side of the water heater. The volumetric design flow rate is specified in m³/s or gal/min. The field is needed when the Use side is connected to a plant loop. The field can be autosized. Sizing results are reported in the EIO file.

Source-Side Settings

Source side effectiveness

This field specifies the heat transfer effectiveness between the source side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the source side water

and the tank water. If the effectiveness is lower, then the source side outlet water temperature will not be as hot as the tank water, simulating a heat exchanger.

Source side design flow rate

This field is optional and is used to specify the design flow rate through the Source Side of the water heater. The volumetric design flow rate is specified in m3/s or gal/min. The field is needed when the Source Side is connected to a plant loop. The field can be autosized. Sizing results are reported in the EIO file.

Source side flow control mode

This field is optional and is used to provide control over the logic used by the source side of the water heater to request flow. There are three choices for different modes:

- **1-IndirectHeatPrimarySetpoint**, where the water heater will request flow at the source side when the main [Setpoint temperature schedule](#) and deadband call for the tank to be heated. This mode is typical for a water heater indirectly heated by a boiler.
- **2-IndirectHeatAlternateSetpoint** is similar but it bases its control decisions on an [alternate setpoint](#) selected in the following field. This mode is useful when the indirect source of heat may not satisfy the load and an internal heater is used for backup.
- **3-StorageTank** is for a passive tank and it always requests flow unless the tank temperature is equal to or higher than the maximum limit given in the input field called [Maximum temperature limit](#).

Indirect alternate setpoint temperature schedule

This field is required when using the **2-IndirectHeatAlternateSetpoint** Source side flow control mode (above) and is used to provide a schedule with alternate setpoints. The Schedule specifies the hot water temperature setpoint (in °C) to use as the “cut-out” temperature for control logic at the source side.

Sizing

The Water heater sizing data is entered on the Sizing tab.

Design mode

This field describes the method to be used for sizing the water heater. There are six choices:

- **1-Peak draw**. This design method uses the design flow rates of all the different demands placed on the water heater. The tank size is based on how long it can meet the demand and how quickly it can recover. The user enters the time in hours that the water heater can meet the demands. Only the hot water uses connected to an individual water heater are included in that water heater's peak draw.
- **2-Residential HUD-FHA minimum** This design method is based on minimum permissible water heater sizes (established by HUD-FHA in its Minimum Property Standards for One- and Two-Family Living Units, No. 4900.1-1982). The user enters the number of bathrooms and bedrooms in this input object. The smallest allowable water heater sizes are used.
- **3-Per person** This design method scales sizes based on the total number of people in all zones in the building. Each water heater in the model will be sized using the total (peak, design) number of people for the entire model. The number of people is determined from People objects defined elsewhere in the input file
- **4-Per floor area** This design method scales sizes based on the total floor area in all the zones in the building. Each water heater in the model will be sized using all the floor area in the model. The floor areas are determined from the geometry input elsewhere in the input file.
- **5-Per unit** This design method scales sizes based on an arbitrary number of units. This can be used, for example, to size based on the number of rooms in a lodging building. The user provides the number of units in an input field in this object.
- **6-Per solar collector area** This design method scales tank volume based on the collector area for a solar hot water collector. The collector area is summed for all the collectors in the model and each tank is sized for the total. The collector area is determined from input for Solar Collectors defined elsewhere in the input file.

Time storage can meet peak draw

This field provides the time, in hours, that the tank's volume can sustain a peak draw. It is used to size the tank's volume which is the simple product of peak draw volume flow rate and the draw time. There is no assurance that the water will be at the desired temperature for the entire draw. This field is only used if the [Design mode](#) is **1-Peak draw**. The water heater must be connected to a full plant loop and be on the supply side.

Time for tank recovery

This field provides the time, in hours, that tank's heater needs to recover the volume of the tank. The temperatures used to define recovery are a starting temperature of 14.4°C (58°F) and a final temperature of 57.2°C (135°F). Time for tank recovery data is only required if the [Design mode](#) is **1-Peak draw**.

Nominal tank volume for autosizing plant connections

This field is used in case the water heater is indirectly heated by its source side connections and they are also autosized. The units are m3 or ft3. Because of the complexity of such a water heater and the timing for when sizing calculation happen inside EnergyPlus, the Source side connection flow rates need to be reported before the tank's volume can be sized to meet Peak draw. This input field is used to provide a nominal tank volume to use temporarily while the flow connections are sized. This field is only required if the [Design mode](#) is **1-Peak draw** and the water heater has autosized plant connections on the demand side.

Number of bedrooms

This field is used to enter the number of bedrooms in the model. It is only required if the [Design mode](#) is **2-Residential HUD-FHA minimum**.

Number of bathrooms

This field is used to enter the number of bathrooms in the model. It is only required if the [Design mode](#) is **2-Residential HUD-FHA minimum**.

Storage capacity per person

This field is used to enter the tank's storage volume on per-person basis. The units are m3/person or ft3/person. The storage capacity per person is only required if the [Design mode](#) is **3-Per person**.

Recovery capacity per person

This field is used to enter the recovery capacity per person in units of m3/person-hr or ft3/person-h. This is the volume of water the heater can recover in one hour per person. Recovery is heating water from a starting temperature of 14.4°C (58°F) to a final temperature of 57.2°C (135°F). The Recovery capacity per person is only required if the [Design mode](#) is **3-Per person**.

Storage capacity per floor area

This field is used to enter the tank's storage volume on a per-floor-area basis. The units are m3 /m2 or ft3/ft2. The Recovery capacity per person is only required if the [Design mode](#) is **3-Per person**.

Recovery capacity per floor area

This field is used to enter the recovery capacity per floor area in units of m3/h-m2 or ft3/h-ft2. This is the volume water the heater can recover in an hour per floor area. Recovery is heating water from a starting temperature of 14.4°C (58°F) to a final temperature of 57.2°C (135°F). The Recovery capacity per floor area is only required if the [Design mode](#) is **4-Per floor area**.

Number of units

This field is used to enter the number of Units for use in sizing on per-Unit basis with the next two fields. The Number of units data only required if the [Design mode](#) is **5-Per unit**. This can be used to account for any arbitrary item such as lodging rooms, desks, water fixtures, restrooms, etc.

Storage capacity per unit

This field is used to enter the tanks' storage volume on per-Unit basis. The units are m3/h or ft3/h. The number of Units is entered in the previous field. The Storage capacity per unit is only required if the [Design mode](#) is **5-Per unit**.

Recovery capacity per unit

This field is used to enter the recover capacity per Unit in units of m3/h or ft3/h. This is the volume of water the heater can recover in an hour per Unit. Recovery is heating water from a starting temperature of 14.4°C (58°F) to a final temperature of 57.2°C (135°F). This field is only required if the [Design mode](#) is **5-Per unit**.

Storage capacity per collector area

This field is used to enter the tank's storage volume on per-solar-collector-area basis. The units are m3/m2 or f3/m2. The Recovery capacity per unit is only required if the [Design mode](#) is **6-Per solar collector area**.

Water Heater - Stratified



WaterHeater:Stratified
WaterHeater:Sizing

Used in:

Supply side of [DHW Loops](#)

The **2-Stratified** water heater simulates a stratified, multi-node water tank. The model divides the water tank into multiple nodes of equal volume. The nodes are coupled by vertical conduction effects, internode fluid flow, and temperature inversion mixing. The water heater simultaneously solves the differential equations governing the energy balances on the nodes using a numerical method. The system timestep is divided into many small substeps that allow the simulation to capture events that occur on a very short time scale. This approach allows ambient losses and parasitic loads to be divided into on-cycle and off-cycle effects and accounted for in detail.

For losses to the ambient environment, the ambient air temperature can be taken from a schedule, a zone, or the exterior. When used with a zone, a fraction of the skin losses can be added to the zone heat balance as internal heat gains.

The Stratified water heater allows two heating elements to be simulated. The two elements can cycle on and off to maintain the node temperature within the deadband. The **Heater priority control** setting determines how the heaters work together. There are two priority control options: 1-Master Slave or 2-Simultaneous. In the Master slave option, the main Heater is the master and the auxiliary heater is the slave. That is, both heaters are not allowed to turn on at the same time. If the thermostats ask for heat from both heaters, only the main heater will turn on. Once the main heater has met the set point, it turns off and the auxiliary heater can turn on, if necessary. In the Simultaneous priority control option, both heaters can turn on and off independently. Autosizing is available for only for the main Heater.

General

Name

The name of the Water heater.

Type

Select the type of water heater from the list:

- **1-Mixed**,
- **2-Stratified**.

For a discussion of the applications, advantages and disadvantages of the two water heater types see the [Water heater overview](#) topic.

Tank volume

The volume of the storage tank (in m³ or ft³). Although this field is allowed to go down to zero, even so-called "tankless" water heaters have some volume of water that is maintained around the heating elements or in the heat exchanger, typically around 0.00379 m³ (1 gallon).

This field is autosizable based on the [Sizing data](#), however we are aware of some issues with the EnergyPlus autosizing method for this field and so we recommend that the tank volume is hard set wherever possible.

Tank height

The height of the tank (in m or in).

For the **2-Horizontal cylinder** tank shape (see below) the height of the tank is the measure in the axial direction, i.e., the height if you were to stand the cylinder up on its end.

This field is autosizable based on the [Sizing data](#), however we are aware of some issues with the EnergyPlus autosizing method for this field and so we recommend that the tank height is hard set wherever possible.

Tank shape

The tank shape determines the size and skin losses of the stratified nodes. There are two options:

- **1-Vertical Cylinder**, which describes most upright residential water heaters.
- **2-Horizontal Cylinder**, which describes a few speciality water heaters and large commercial storage tanks. This shape can also be used to model an outdoor storage tank located above a solar collector in a thermosiphon configuration. **2-Horizontal cylinder** implies that the tank is divided into nodes of equal mass, but not equal height.

External Heating Plant Connection

External heating plant connection

Check this option if the water heater is to be supplied with heat from a Hot water loop. In this case inlet and outlet connections are included on the bottom of the water heater allowing the connection to be made to the demand side of the Hot water plant loop.

Indirect water heating recovery time

This field is only required when the External heating plant option is checked, i.e. when the water heater is connected to the demand side of a plant loop. It is used to provide a design parameter for autosizing design flow rates. The recovery time is expressed in hours. This is the time that the entire volume of the tank can be heated from 14.4°C to 57.2°C (58°F to 135°F) with an inlet temperature defined as the exit temperature in the associated Plant Sizing object. The default is 1.5 hours. The calculation is based on log-mean temperature difference (LMTD) and includes the heat transfer effectiveness factor entered above.

Temperature Settings

Maximum temperature limit

The temperature (in °C or °F) at which the tank water becomes dangerously hot and is vented through boiling or an automatic safety. The tank temperature will never exceed the maximum. Any extra heat added to the tank is immediately vented.

Also, when [Source side flow control mode](#) is set to **3-StorageTank** the controls always request flow unless the tank temperature is equal to or higher than the maximum limit provided here.

Note: The **Maximum temperature limit** must be greater than the setpoint temperature at all times.

Setpoint temperature schedule

Select the schedule specifying the hot water temperature set point (in °C) for the main heater. Also known as the “cut-out” temperature.

Deadband temperature difference

The delta temperature difference ($\Delta^{\circ}\text{C}$ or $\Delta^{\circ}\text{F}$) between the set point and the “cut-in” temperature at which the main heater will turn on. In other words, the “cut-in” temperature is Set Point – Deadband.

Ambient Heat Transfer Settings

Ambient temperature indicator

The Ambient temperature indicator specifies how the temperature of the air surrounding the Water heater is to be defined. These options are available:

- **1-Schedule** where the Ambient temperature schedule field defines the ambient temperature.
- **2-Zone** where the zone air temperature of the zone specified by Ambient temperature zone, below, provides the ambient temperature
- **3-Outdoors** where the outdoor dry-bulb air temperature provides the ambient temperature for the water heater.

Ambient temperature schedule

Select the schedule specifying the ambient air temperature around the tank for skin losses. This field is only required if [Ambient temperature indicator](#) is **1-Schedule**.

Ambient temperature zone

The building zone specifying the ambient air temperature around the tank for skin losses. This field is only required if [Ambient temperature indicator](#) is **2-Zone**.

Off-cycle flue loss coefficient to ambient temperature

The off-cycle flue loss coefficient to the ambient air temperature (in W/K or Btu/h-F). This setting mainly applies to gas water heaters that have a flue.

Uniform skin loss coefficient per unit area to ambient temperature

This is the uniform skin loss coefficient (or U-Value) of the tank to the ambient air temperature (in W/m²-K or Btu/hr-ft²-F). The uniform skin loss accounts for the tank insulation and applies during both off- and on-cycle operation. The overall losses at any particular node can be further modified using the [Additional loss coefficient](#) settings to account for thermal shorting due to pipe penetrations, water heater feet, and any other loss effects.

On-Cycle loss fraction to zone

If the [Ambient temperature indicator](#) is **2-Zone**, this field adds the specified fraction of the on-cycle losses to the zone heat balance as an internal gain.

Off-Cycle loss fraction to zone

If the Ambient temperature indicator is **2-Zone**, this field adds the specified fraction of the off-cycle losses to the zone heat balance as an internal gain.

Skin loss fraction to zone

If the Ambient temperature indicator is **2-Zone**, this field adds the specified fraction of the skin losses to the zone heat balance as an internal gain.

Internal Heating Element

Internal heating element

If the water heater includes an internal heating element then checking this checkbox provides access to the element settings.

Internal Heating Element Settings

Heater fuel type

The type of fuel used for heating. The fuel type can be one of:

- **1-Electricity**,
- **2-Natural gas**,
- **3-Propane gas**,
- **4-Fuel oil 1**,
- **5-Fuel oil 2**,
- **6-Coal**,
- **7-Diesel**,
- **8-Gasoline**,
- **9-Steam**,
- **10-District heating**.
- **11-Other fuel 1**
- **12-Other fuel 2**

11-Other fuel 1 and **12-Other fuel 2** can be used for representing biofuels such as biomass and biogas for example.

Heater thermal efficiency

The thermal conversion efficiency from fuel energy to heat energy for the heater element or burner. This is not the same as the overall efficiency of the water heater.

Heater capacity

The heat rate (in W) supplied to the water for the main heater, probably the same as the “nominal” capacity. For residential electric water heaters, heating elements are usually 4500 W. This field is autosizable.

Height

The height of the main heating element in the tank (in m or in).

Auxiliary Heating Element

Auxiliary heating element

If a second "auxiliary" heater is to be included in the water heater then this checkbox should be checked.

Auxiliary Heating Element Settings

Heater priority control

The heater priority control determines how the main heater and the auxiliary heater work together. Note these heaters are called Heater 1 and Heater 2 in the EnergyPlus documentation. There are two options:

- **1-Master slave**, where the main heater is the master and the auxiliary heater is the slave. In most residential electric water heaters, the heaters operate in a Master-slave relationship. That is, both heaters are not allowed to turn on at the same time. If the thermostats ask for heat at both the main and auxiliary heaters, only the main heater will turn on. Once the main heater has met the set point, it turns off and the auxiliary heater can turn on, if necessary. In other words, only one heater can be on at any time, and the main heater 1 always has priority over the auxiliary heater.
- **2-Simultaneous**, where the main and auxiliary heaters can turn on and off independently.

Heater capacity

The heat rate (in W) supplied to the water for the auxiliary heater.

Height

The height of the auxiliary heating element in the tank (in m or in).

Setpoint temperature schedule

Select the schedule specifying the hot water temperature set point (in °C) for the auxiliary heater. Also known as the “cut-out” temperature.

Deadband temperature difference

The delta temperature difference ($\Delta^{\circ}\text{C}$ or $\Delta^{\circ}\text{F}$) between the set point and the “cut-in” temperature at which the auxiliary heater will turn on. In other words, the “cut-in” temperature is Set Point – Deadband.

On-Cycle Settings

On-Cycle parasitic fuel consumption rate

On-cycle parasitics include parts of the water heater that consume fuel when the heater is on, for example, an induction fan, or stand-by electronic control circuits. The fuel consumption rate (in W or Btu/h) is the total fuel that is consumed by all of the on-cycle parasitics.

On-Cycle parasitic fuel type

The type of fuel used by the on-cycle parasitics. The fuel type can be

- **1-Electricity**,
- **2-Natural gas**,
- **3-Propane gas**,
- **4-Fuel oil #1**,
- **5-Fuel oil #2**,
- **6-Coal**,
- **7-Diesel**,
- **8-Gasoline**,
- **9-Steam**, or
- **10-District heating**.

The fuel type can be the same or different from the Heater Fuel Type.

On-Cycle parasitic heat fraction to tank

The fraction of on-cycle parasitic fuel energy that is converted to heat energy that ends up in the tank water. For example, an induction fan might (maybe) deliver a small fraction of its energy to the tank water for a value of 0.05. Electronic control circuits, on the other hand, do not add any heat to the tank and should be 0.

On-Cycle parasitic height

The height (in m or in) where any on-cycle parasitic heat gains are added to the tank.

Off-Cycle Settings

Off-Cycle Parasitic Fuel Consumption Rate

Off-cycle parasitics include parts of the water heater that consume fuel when the heater is off, for example, a pilot light, or stand-by electronic control circuits. The fuel consumption rate (in W or Btu/h) is strictly the total fuel that is consumed by all of the off-cycle parasitics.

Off-Cycle Parasitic Fuel Type

The type of fuel used by the off-cycle parasitics. The fuel type can be one of

- **1-Electricity,**
- **2-Natural gas,**
- **3-Propane gas,**
- **4-Fuel oil #1,**
- **5-Fuel oil #2,**
- **6-Coal,**
- **7-Diesel,**
- **8-Gasoline,**
- **9-Steam,** or
- **10-District heating.**

The fuel type can be the same or different from the Heater fuel type.

Off-Cycle parasitic heat fraction to tank

The fraction of off-cycle parasitic fuel energy that is converted to heat energy that ends up in the tank water. For example, a pilot light would deliver most of its heat to the tank water, as long as the thermal conversion efficiency must be taken into account, so perhaps 0.80 is reasonable. Electronic control circuits, on the other hand, do not add any heat to the tank and should be 0.

Off-Cycle parasitic height

The height (in m or in) where any off-cycle parasitic heat gains are added to the tank.

Inlet Mode

Inlet mode

The inlet mode of entering fluid from the use and source sides. There are two options:

- **1- Fixed**, the default mode, where the fluid enters at the fixed heights specified above.
- **2-Seeking** where the fluid “seeks out” the stratified node that is closest to the inlet temperature and adds all flow to that node. The Seeking mode provides maximum stratification.

Use-Side Settings

Use side effectiveness

This field specifies the heat transfer effectiveness between the use side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the use side water and the tank water. If the effectiveness is lower, then the use side outlet water temperature will not be as hot as the tank water, simulating a heat exchanger.

Use side design flow rate

This field is optional and is used to specify the design flow rate through the Use Side of the water heater. The volumetric design flow rate is specified in m³/s or gal/min. The field is needed when the Use side is connected to a plant loop. The field can be autosized. Sizing results are reported in the EIO file.

Use side inlet height

The height of the use side inlet to the tank (in m or in). The default value of 0 means the bottom of the tank. The inlet height cannot be higher than the tank height.

Use side outlet height

The height of the use side outlet from the tank (in m or in). If autocalculate, the inlet defaults to the top of the tank. The outlet height cannot be higher than the tank height.

Source-Side Settings

Source side effectiveness

This field specifies the heat transfer effectiveness between the source side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the source side water and the tank water. If the effectiveness is lower, then the source side outlet water temperature will not be as hot as the tank water, simulating a heat exchanger.

Source side design flow rate

This field is optional and is used to specify the design flow rate through the Source Side of the water heater. The volumetric design flow rate is specified in m³/s or gal/min. The field is needed when the Source Side is connected to a plant loop. The field can be autosized. Sizing results are reported in the EIO file.

Source side flow control mode

This field is optional and is used to provide control over the logic used by the source side of the water heater to request flow. There are three choices for different modes:

- **1-IndirectHeatPrimarySetpoint**, where the water heater will request flow at the source side when the main [Setpoint temperature schedule](#) and deadband call for the tank to be heated. This mode is typical for a water heater indirectly heated by a boiler.
- **2-IndirectHeatAlternateSetpoint** is similar but it bases its control decisions on an [alternate setpoint](#) selected in the following field. This mode is useful when the indirect source of heat may not satisfy the load and an internal heater is used for backup.
- **3-StorageTank** is for a passive tank and it always requests flow unless the tank temperature is equal to or higher than the maximum limit given in the input field called [Maximum temperature limit](#).

Indirect alternate setpoint temperature schedule

This field is required when using the **2-IndirectHeatAlternateSetpoint** Source side flow control mode (above) and is used to provide a schedule with alternate setpoints. The Schedule specifies the hot water temperature setpoint (in °C) to use as the “cut-out” temperature for control logic at the source side.

Source side inlet height

The height of the source side inlet to the tank (in m or in). If autocalculate, the inlet defaults to the top of the tank. The inlet height cannot be higher than the tank height.

Source side outlet height

The height of the source side outlet from the tank (in m or in). The outlet height cannot be higher than the tank height.

Stratification Settings

Additional destratification conductivity

An additional destratification conductivity (in W/m-K or Btu-in/h-ft²-°F) is added to the fluid conductivity of water (0.6 W/m-K) to account for vertical conduction effects along the inside of the tank wall, and perhaps other vertical components such as the flue, the cold water inlet pipe (dip tube), and the anode rod.

Number of stratified nodes

The number of stratified nodes in the tank. There must be at least one node. The maximum number of nodes is 12.

Additional Loss Coefficient for node 1-12

An additional loss coefficient (in W/K or Btu/h-°F) added to the skin losses for a given node to account for thermal shorting due to pipe penetrations, water heater feet, and any other loss effects.

Sizing

The Water heater sizing data is entered on the Sizing tab.

Design mode

This field describes the method to be used for sizing the water heater. There are six choices:

- **1-Peak draw** This design method uses the design flow rates of all the different demands placed on the water heater. The tank size is based on how long it can meet the demand and how quickly it can recover. The user enters the time in hours that the water heater can meet the demands. Only the hot water uses connected to an individual water heater are included in that water heater's peak draw.
- **2-Residential HUD-FHA minimum** This design method is based on minimum permissible water heater sizes (established by HUD-FHA in its Minimum Property Standards for One- and Two-Family Living Units, No. 4900.1-1982). The user enters the number of bathrooms and bedrooms in this input object. The smallest allowable water heater sizes are used.
- **3-Per person** This design method scales sizes based on the total number of people in all zones in the building. Each water heater in the model will be sized using the total (peak, design) number of people for the entire model. The number of people is determined from People objects defined elsewhere in the input file.
- **4-Per floor area** This design method scales sizes based on the total floor area in all the zones in the building. Each water heater in the model will be sized using all the floor area in the model. The floor areas are determined from the geometry input elsewhere in the input file.
- **5-Per unit** This design method scales sizes based on an arbitrary number of units. This can be used, for example, to size based on the number of rooms in a lodging building. The user provides the number of units in an input field in this object.
- **6-Per solar collector area** This design method scales tank volume based on the collector area for a solar hot water collector. The collector area is summed for all the collectors in the model and each tank is sized for the total. The collector area is determined from input for Solar Collectors defined elsewhere in the input file.

Time storage can meet peak draw

This field provides the time, in hours, that the tank's volume can sustain a peak draw. It is used to size the tank's volume which is the simple product of peak draw volume flow rate and the draw time. There is no assurance that the water will be at the desired temperature for the entire draw. This field is only used if the [Design mode](#) is **1-Peak draw**. The water heater must be connected to a full plant loop and be on the supply side.

Time for tank recovery

This field provides the time, in hours, that tank's heater needs to recover the volume of the tank. The temperatures used to define recovery are a starting temperature of 14.4°C (58°F) and a final temperature of 57.2°C (135°F). Time for tank recovery data is only required if the [Design mode](#) is **1-Peak draw**.

Nominal tank volume for autosizing plant connections

This field is used in case the water heater is indirectly heated by its source side connections and they are also autosized. The units are m3 or ft3. Because of the complexity of such a water heater and the timing for when sizing calculation happen inside EnergyPlus, the Source side connection flow rates need to be reported before the tank's volume can be sized to meet Peak draw. This input field is used to provide a nominal tank volume to use temporarily while the flow connections are sized. This field is only required if the [Design mode](#) is **1-Peak draw** and the water heater has autosized plant connections on the demand side.

Number of bedrooms

This field is used to enter the number of bedrooms in the model. It is only required if the [Design mode](#) is **2-Residential HUD-FHA minimum**.

Number of bathrooms

This field is used to enter the number of bathrooms in the model. It is only required if the [Design mode](#) is **2-Residential HUD-FHA minimum**.

Storage capacity per person

This field is used to enter the tank's storage volume on per-person basis. The units are m3/person or ft3/person. The storage capacity per person is only required if the [Design mode](#) is **3-Per person**.

Recovery capacity per person

This field is used to enter the recovery capacity per person in units of m3/person-hr or ft3/person-h. This is the volume of water the heater can recover in one hour per person. Recovery is heating water from a starting temperature of 14.4°C (58°F) to a final temperature of 57.2°C (135°F). The Recovery capacity per person is only required if the [Design mode](#) is **3-Per person**.

Storage capacity per floor area

This field is used to enter the tank's storage volume on a per-floor-area basis. The units are m3 /m2 or ft3/ft2. The Recovery capacity per person is only required if the [Design mode](#) is **3-Per person**.

Recovery capacity per floor area

This field is used to enter the recovery capacity per floor area in units of m3/h-m2 or ft3/h-ft2. This is the volume water the heater can recover in an hour per floor area. Recovery is heating water from a starting temperature of 14.4°C (58°F) to a final temperature of 57.2°C (135°F). The Recovery capacity per floor area is only required if the [Design mode](#) is **4-Per floor area**

Number of units

This field is used to enter the number of Units for use in sizing on per-Unit basis with the next two fields. The Number of units data only required if the [Design mode](#) is **5-Per unit**. This can be used to account for any arbitrary item such as lodging rooms, desks, water fixtures, restrooms, etc.

Storage capacity per unit

This field is used to enter the tanks' storage volume on per-Unit basis. The units are m³/h or ft³/h. The number of Units is entered in the previous field. The Storage capacity per unit is only required if the [Design mode](#) is **5-Per unit**.

Recovery capacity per unit

This field is used to enter the recover capacity per Unit in units of m³/h or ft³/h. This is the volume of water the heater can recover in an hour per Unit. Recovery is heating water from a starting temperature of 14.4°C (58°F) to a final temperature of 57.2°C (135°F). This field is only required if the [Design mode](#) is **5-Per unit**.

Storage capacity per collector area

This field is used to enter the tank's storage volume on per-solar-collector-area basis. The units are m³/m² or ft³/m². The Recovery capacity per unit is only required if the [Design mode](#) is **6-Per solar collector area**.

Height aspect ratio

This field is used to scale the height of a stratified tank to preserve relative geometry for different size tanks. It is defined at the length scale in the vertical direction (height) divided by the length scale in the horizontal direction (diameter). It is only used if the [Tank height](#) has been set to Autosize, and the [Tank shape](#) is set to **1-Vertical Cylinder**. This field is used for all Design modes.

Solar Hot Water

Some typical solar hot water loop configurations are shown on the [Solar and Auxiliary Heating Loops](#) help page. This page provides more information on how to configure and control solar hot water systems.

Solar Heating System Control

To provide control over the collector loop, a differential thermostat option is incorporated within the solar loop dialog. The differential thermostat compares the temperature in the water heater to the temperature in a collector so that the loop pump is only turned on when there is a useful heat gain.

Plant loop Data

General | Plant Equipment Operation

General

Name: Solar Loop

Fluid type: 2-EthyleneGlycol

Glycol concentration: 0.500

Plant loop volume (m3): 0.500

Flow Type

Plant loop flow type: 2-Variable flow

Temperature

Maximum loop temperature (°C): 100.00

Minimum loop temperature (°C): 0.00

Flow Rate

Maximum loop flow rate (m3/s): Autosize

Minimum loop flow rate (m3/s): 0.000000

Load distribution scheme: 1-Sequential

Plant loop demand calculation scheme: 1-SingleSetPoint

Temperature Protection

☐ Low temperature protection

☒ High temperature protection

Temperature above which system is off (°C): 90.00

Differential Thermostat

☒ Differential thermostat

Temperature difference to turn system on (deltaC): 10.00

Temperature difference to turn system off (deltaC): 2.00

Sizing

There is an option on the solar collector dialog to allow you to select which collector will act as the sensor for the thermostat. The outlet temperature from the water heater is used as the location for the other thermostat sensor.

Solar collector Data

Solar collector

General

Name: Solar Collector

Maximum flow rate (m3/s): 0.0001380

Solar Collector Surface

Solar collector surface: Solar collector.5

Control

☒ Differential thermostat hot node sensor

Performance

Solar collector performance template: ACR Solar International 10-01

Gross area (reference use only) (m2): 0.932500

Test flow rate (m3/s): 0.000032

Test correlation type: 1-Inlet

Efficiency Equation Coefficients

Coefficient 1: 0.603000

Coefficient 2 (W/m2-K): -3.8665000

Coefficient 3 (W/m2-K2): 0.0015000

Incident Angle Modifier Coefficients

Coefficient 1: 1

Coefficient 2: -0.194400

Coefficient 3: -0.018600

Note: If the two temperature differences on the solar collector dialog are too close, it is possible for the system to turn on and off rapidly without much useful heat gain. This can also occur if the flow rate through the collector is too high. Without flow the fluid in the collector heats up more quickly; when high flow is turned on, all of the hot fluid is removed and the temperature drops, forcing the system off again.

Freeze Prevention

In climates with a cold season, the solar heating system must be designed to avoid the risk of fluid freezing in the solar collector or exposed pipes and causing damage. There are several strategies that can minimise the risk.

1. Seasonal schedule. The simplest strategy is to not use the system during the cold season. However, this is problematic because it requires the collector to be manually drained of all fluid. The benefits of the solar heating system are also lost during this time. This can be simulated using an appropriate pump flow schedule for the collector system.
2. Antifreeze. The freezing point of the liquid is decreased by adding antifreeze (glycol) to the water or using a different heat transfer liquid with a lower freezing point. This can be achieved by selecting a glycol fluid for the loop fluid type on the solar loop dialog, this is the default setting.
3. Drain-back system. This strategy automatically empties the collector when the pump is not running. This scenario is modelled by default in EnergPylus, although the extra pump energy required to start the system is not taken into account.
4. Re-circulation system. This strategy automatically re-circulates warm liquid from the storage tank back through the collector to maintain the system above the freezing point. There are system losses using this method. This can be simulated by using the Low temperature protection setting on the solar loop dialog to force the system to turn on when the outdoor air temperature or collector outlet temperature falls below a specified minimum.

Edit Plant loop -

Plant loop Data

General | Plant Equipment Operation

General

Name	Solar Loop
Fluid type	2-EthyleneGlycol
Glycol concentration	0.500
Plant loop volume (m3)	0.500

Flow Type

Plant loop flow type	2-Variable flow
----------------------	-----------------

Temperature

Maximum loop temperature (°C)	100.00
Minimum loop temperature (°C)	0.00

Flow Rate

Maximum loop flow rate (m3/s)	Autosize
Minimum loop flow rate (m3/s)	0.000000
Load distribution scheme	1-Sequential
Plant loop demand calculation scheme	1-SingleSetPoint

Temperature Protection

☒ Low temperature protection	
Temperature below which system is on (°C)	0.00
☒ High temperature protection	
Temperature above which system is off (°C)	90.00

Additional Controls

In addition to freeze prevention, it is also necessary to prevent the system from becoming too hot. This is usually a safety issue for the water heater. For this case it is important to have a high temperature cut-off to stop the pump before damaging the water heater. This is accomplished using the High temperature protection setting on the solar loop dialog (see above).

Solar Collector Data



SolarCollector:FlatPlate:
Water

Used in:

- **Solar hot water loop, demand side**

Solar hot water collectors are thermal devices that convert solar energy into thermal energy by raising the temperature of a circulating heat transfer fluid (which must be water in current versions of EnergyPlus). The fluid is used to heat water for DHW usage and/or space heating. In EnergyPlus solar collectors are components that are connected to a solar plant loop. A solar heating system can be constructed with a combination of solar collectors, pumps, and hot water tanks.

The EnergyPlus model is based on the equations found in the ASHRAE standards and Duffie and Beckman (1991). This model applies to glazed and unglazed flat-plate collectors, as well as banks of tubular, i.e. evacuated tube, collectors.

The solar collector object references a [solar collector surface](#) which is defined in the building model and defines the gross area, position, orientation and tilt of the collector allowing it to participate normally in detailed solar and shading calculations if either of the **2-Full exterior** or **3-Full interior and exterior** [Solar distribution](#) options are selected. Solar radiation incident on the collector surface includes beam and diffuse radiation, as well as radiation reflected from the ground and adjacent surfaces. Shading of the collector by other surfaces, such as nearby buildings or trees, is also taken into account. Likewise, the collector surface can shade other surfaces, for example, reducing the incident radiation on the roof beneath it.

The thermal and optical properties of the collector module can be loaded from a Solar collector template.

General

Name

The unique name of the Solar hot water collector.

Maximum flow rate

The maximum flow rate (in m3/s or gal/min) allowed through the collector.

Solar Collector Surface

Solar collector surface

Select a building solar collector surface to define the gross area, position, orientation and tilt of the collector.

Control

Differential thermostat hot node sensor

Check this option to identify this Solar collector as the one to be used as a reference point for the [differential thermostat](#) selected on the Solar loop dialog.

Performance

Solar collector performance template

A Solar collector performance template which defines the thermal and optical properties of the collector.

DesignBuilder provides a database of Solar collector templates containing the thermal and optical performance parameters for a single collector module. These parameters are based on the testing methodologies described in ASHRAE Standards 93 and 96. The Solar Rating and Certification Corporation (SRCC) applies these standards in their rating procedures of solar collectors. The coefficients for the energy conversion efficiency and incident angle modifier allow first order (linear) or second order (quadratic) correlations. To use a first order correlation, the second order coefficient must be set to zero.

In order for the model to work correctly, the test conditions for which the performance coefficients were measured must be specified in the fields: Test flow rate, and Test correlation type. Currently, only water is allowed as the Test Fluid.

More detailed information about the performance coefficients, can be found in the [EnergyPlus Engineering Reference Document](#).

Gross area

The gross area of the collector module (in m² or ft²). This value is mainly for reference. The area of the associated solar collector surface is used in all calculations.

Test flow rate

The volumetric flow rate during testing (m³/s or gal/min). If the value is available as flow rate per unit area, it is recommended to multiply by the Gross Area of the collector module, not the net aperture area.

Test correlation type

This field specifies type of temperature used to develop the correlation equations. The testing procedure is based on an experimental correlation using either **Inlet**, **Average** or **Outlet** temperature. Enter one of these choices. The ASHRAE Standards 93 and 96 always use Inlet temperature.

Coefficient 1 of efficiency equation

First coefficient of efficiency equation for energy conversion (dimensionless). This is the Y-intercept term.

Coefficient 2 of efficiency equation

Second coefficient of efficiency equation for energy conversion (W/m²-K or Btu/h-ft²-F). This is the first order term.

Coefficient 3 of efficiency equation

Third coefficient of efficiency equation for energy conversion (W/m²-K² or Btu/h-ft²-F²). This is the second-order term. If set to zero, a first-order linear correlation is used.

Coefficient 2 of incident angle modifier

Second coefficient of the incident angle modifier equation. This the first-order term. (There is no Coefficient 1 of Incident Angle Modifier because that number is always 1.0.)

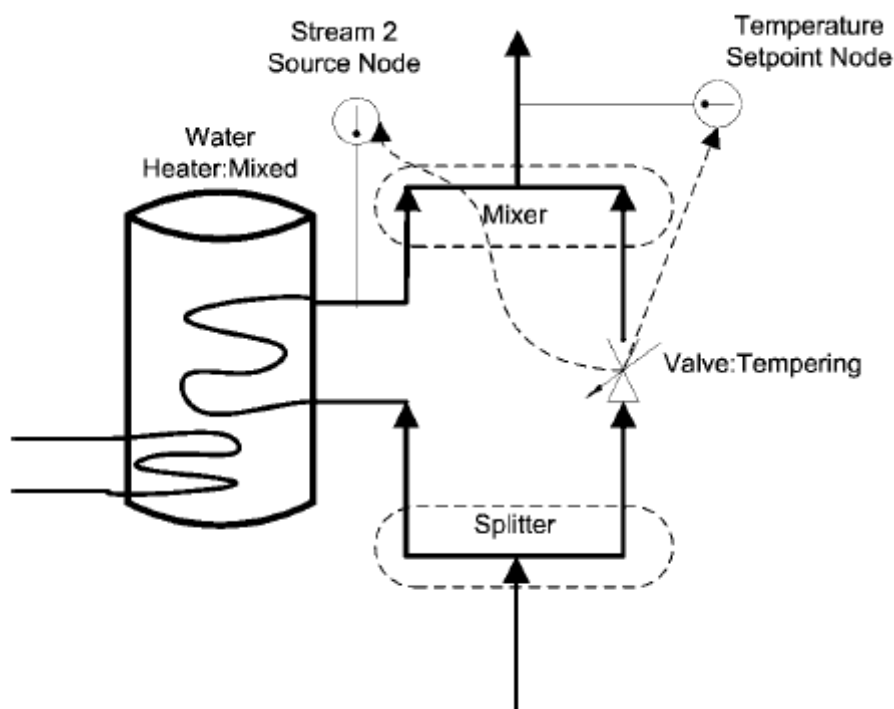
Coefficient 3 of incident angle modifier

Third coefficient of the incident angle modifier equation. This is the second-order term. If set to zero, a first order linear correlation is used.

Tempering Valve

A tempering valve may be used for cases where plant flow control is needed to make efficient use of thermal storage and is required in solar hot water loops to prevent the DHW loop thermal storage tank from becoming warmer than is necessary or allowable for safe use of the hot water. Although real installations of a tempering, or anti-scald valve, would more commonly mix a new stream of mains water with the hot water to achieve a desired outlet temperature, this is difficult to model directly within EnergyPlus because plant loops need to be closed circuits. For installations where the water entering the splitter is directly from the mains, such as make up water entering a water heater tank, the modelling provided with this object will be thermodynamically equivalent.

The Tempering valve models a temperature-controlled diversion valve on a bypass pipe that can open to divert flow around one or more plant components. It can only be used on one of two branches between a Splitter and a Mixer. The figure below shows the use of the tempering valve with a Water heater component on "Stream 2." The tempering valve acts to divert flow through the branch it is on in order to adjust the temperature at the outlet of the mixer. If the temperature at Stream 2 Source Node is warmer than the setpoint and the inlet flow is cooler than the setpoint, then a controller determines how much flow should bypass the storage tank to achieve the desired setpoint.



No data entry is required for tempering valves.

Plant And Condenser Loop Equipment Data

Plant equipment is used in plant and condenser loops:

- "Boilers" on page 979
- "District Heating" on page 896
- "Chillers" on page 985
- "District Cooling" on page 1018
- "Cooling Towers" on page 1019
- "Fluid Cooler" on page 1042
- "Pumps " on page 1046
- "Ground Heat Exchanger" on page 1050
- "VRF Outdoor Unit" on page 1070

- "Fluid To Fluid Heat Exchanger" on page 1087

Boilers



Boiler:HotWater

Used in:

- **Hot water loop, supply side**

Boilers provide hot water to hot water heating coils, hot water radiators, heated floors, baseboards etc. The fuel consumed by the boiler is based on a nominal thermal efficiency value. A normalised efficiency performance curve can additionally be used to more accurately represent the performance of non-electric boilers, but is not a required input.

Boilers can be placed using the [Add Boiler](#) tool when you are located in a hot water supply loop.

To edit an existing boiler go to the **Hot water supply loop** (HVAC System > Hot water loop > Supply loop) level where the boiler is placed, click on the boiler icon to highlight it then either right click and select the **Edit selected component** menu option or, when using [Learning](#) mode, click on the Edit icon in at the top of the info panel.

General

Name

The auto-generated name of the boiler can be edited.

Boiler template

Use this browse option to select a boiler from the EnergyPlus boilers database.

Fuel type

Select the type of fuel used by the boiler. The fuel type can be one of:

- **1-Natural gas,**
- **2-Electricity,**
- **3-Propane gas,**
- **4-Fuel oil 1,**
- **5-Fuel oil 2,**
- **6-Coal,**
- **7-Diesel,**
- **8-Gasoline,**
- **9-Other fuel 1**
- **10-Other fuel 2**

9-Other fuel 1 and **10-Other fuel 2** can be used for representing biofuels such as biomass and biogas for example.

Nominal capacity

This numeric field contains the nominal operating capacity of the boiler (in W or Btu/h). The boiler nominal capacity may be autosized.

Boiler flow mode

This choice field determines how the boiler operates with respect to the intended fluid flow through the device. There are three choices:

- **1-Constant flow** is the default option for constant volume chilled water loops and is useful for constant speed pumping arrangements where the boiler's request for flow is stricter and can increase the overall loop flow.
- **2-Leaving setpoint modulated** changes the boiler model to internally vary the flow rate so that the temperature leaving the boiler matches a setpoint.
- **3-Not modulated** is the default option for variable volume hot water loops and is useful for either variable or constant speed pumping arrangements where the boiler is passive in the sense that although it makes a nominal request for its design flow rate, it can operate at varying flow rates.

In all cases the operation of the external plant system can also impact the flow through the boiler - for example if the relative sizes and operation are such that flow is restricted and the requests cannot be met.

For *variable* flow chilled water loops these options are available:

- **2-Leaving setpoint modulated** and
- **3-Not modulated**.

For *constant* flow chilled water loops these options are available:

- **1-Constant flow** and
- **3-Not modulated**.

The type of loop (variable/constant flow) can be changed by modifying the [Plant loop flow type](#) on the [Hot water plant loop dialog](#).

Note: When the **2-Leaving setpoint modulated** option is selected then you must add an extra Setpoint manager immediately downstream of the boiler hot water outlet to define the temperature of the water supplied.

Parasitic electric load

This optional numeric field contains the parasitic electric power (in W or Btu/h) consumed by a forced draft fan or other electrical device associated with the boiler. This parasitic electric load is consumed whenever the boiler is operating, and the model assumes that this parasitic power does not contribute to heating the water. The minimum value for this field is 0.0.

Sizing factor

The sizing factor is used when the boiler design inputs are autosized. In this case the autosizing results are multiplied by this additional sizing factor. The usual value to enter is 1.0.

The inputs that would be altered by the sizing factor are: **Nominal capacity** and **Design water flow rate**.

The most common use of the sizing factor is for sizing boilers to meet only part of the design load while continuing to use the autosizing feature. For example when a set of boilers is chained together to supply hot water to a plant loop, this sizing factor can be used to indicate the proportion of the load to be met by each boiler.

See also the section on [Autosizing HVAC Components](#)

Efficiency

Fuel use equation

The EnergyPlus boiler model is based the following two equations:

Theoretical Fuel Consumption = Boiler Load / Nominal Thermal Efficiency

Fuel Used = Theoretical Fuel Consumption / Normalised Boiler Efficiency Curve Output

Nominal thermal efficiency

This is the heating efficiency (as a fraction between 0 and 1) of the boiler's burner relative to the higher heating value (HHV) of fuel at a part load ratio of 1.0. Manufacturers typically specify the efficiency of a boiler using the higher heating value of the fuel. For the rare occurrences when a manufacturers (or particular data set) thermal efficiency is based on the lower heating value (LHV) of the fuel, multiply the thermal efficiency by the lower-to-higher heating value ratio. For example, assume a fuel's lower and higher heating values are approximately 45,450 and 50,000 kJ/kg, respectively. For a manufacturers thermal efficiency rating of 0.90 (based on the LHV), the nominal thermal efficiency entered here is 0.82 (i.e. 0.9 multiplied by 45,450/50,000).

Note: The overall boiler efficiency used during the simulation will be the product of the Nominal thermal efficiency and the output of Normalized boiler efficiency performance curve (below).

Efficiency curve temperature evaluation variable

This field is used to control which value for hot water temperature is used when evaluating the **Normalized boiler efficiency performance curve** specified in the next field (if applicable). There are two options:

- **1-Leaving boiler** which indicates that the efficiency curves will be evaluated using the temperature at the boiler outlet.
- **2-Entering boiler** which indicates that the efficiency curves will be evaluated using the temperature at boiler inlet node

This field is only used if type of curve is one that uses temperature as a independent variable.

Normalized boiler efficiency performance curve

This field contains the [Curve](#) which describes the normalized heating efficiency (as a fraction of nominal thermal efficiency) of the boiler's burner. If the <None> linear curve is selected, the nominal thermal efficiency is assumed to be constant (i.e., Fuel Used is equal to the Theoretical Fuel Use in the equation above). When a boiler efficiency curve is used, the curve may be any valid curve with 1 (PLR) or 2 (PLR and boiler outlet water temperature) independent variables. A tri-quadratic curve object is not allowed since it uses 3 independent variables. The linear, quadratic, and cubic curve types may be used when the boiler efficiency is only a function of boiler part-load ratio (PLR). When this type of curve is used, the boiler should operate at (or very near) the design boiler water outlet temperature. Other curve types may be used when the boiler efficiency is a function of both PLR and boiler water temperature. Examples of valid single and dual independent variable equations are shown below. For all curve types PLR is always the x independent variable. When using 2 independent variables, the boiler outlet water temperature (T_{outlet}) is always the y independent variable.

Linear: $\text{Eff} = A0 + A1 \cdot \text{PLR}$

Quadratic: $\text{Eff} = A0 + A1 \cdot \text{PLR} + A2 \cdot \text{PLR}^2$

Cubic: $\text{Eff} = A0 + A1 \cdot \text{PLR} + A2 \cdot \text{PLR}^2 + A3 \cdot \text{PLR}^3$

BiQuadratic: $\text{Eff} = A0 + A1 \cdot \text{PLR} + A2 \cdot \text{PLR}^2 + A3 \cdot T_w + A4 \cdot T_w^2 + A5 \cdot \text{PLR} \cdot T_w$

QuadraticLinear: $\text{Eff} = A0 + A1 \cdot \text{PLR} + A2 \cdot \text{PLR}^2 + A3 \cdot T_w + A4 \cdot \text{PLR} \cdot T_w + A5 \cdot \text{PLR}^2 \cdot T_w$

BiCubic:

$\text{Eff} = A0 + A1 \cdot \text{PLR} + A2 \cdot \text{PLR}^2 + A3 \cdot T_w + A4 \cdot T_w^2 + A5 \cdot \text{PLR} \cdot T_w + A6 \cdot \text{PLR}^3 + A7 \cdot T_w^3 + A8 \cdot \text{PLR}^2 \cdot T_w + A9 \cdot \text{PLR} \cdot T_w^2$

where:

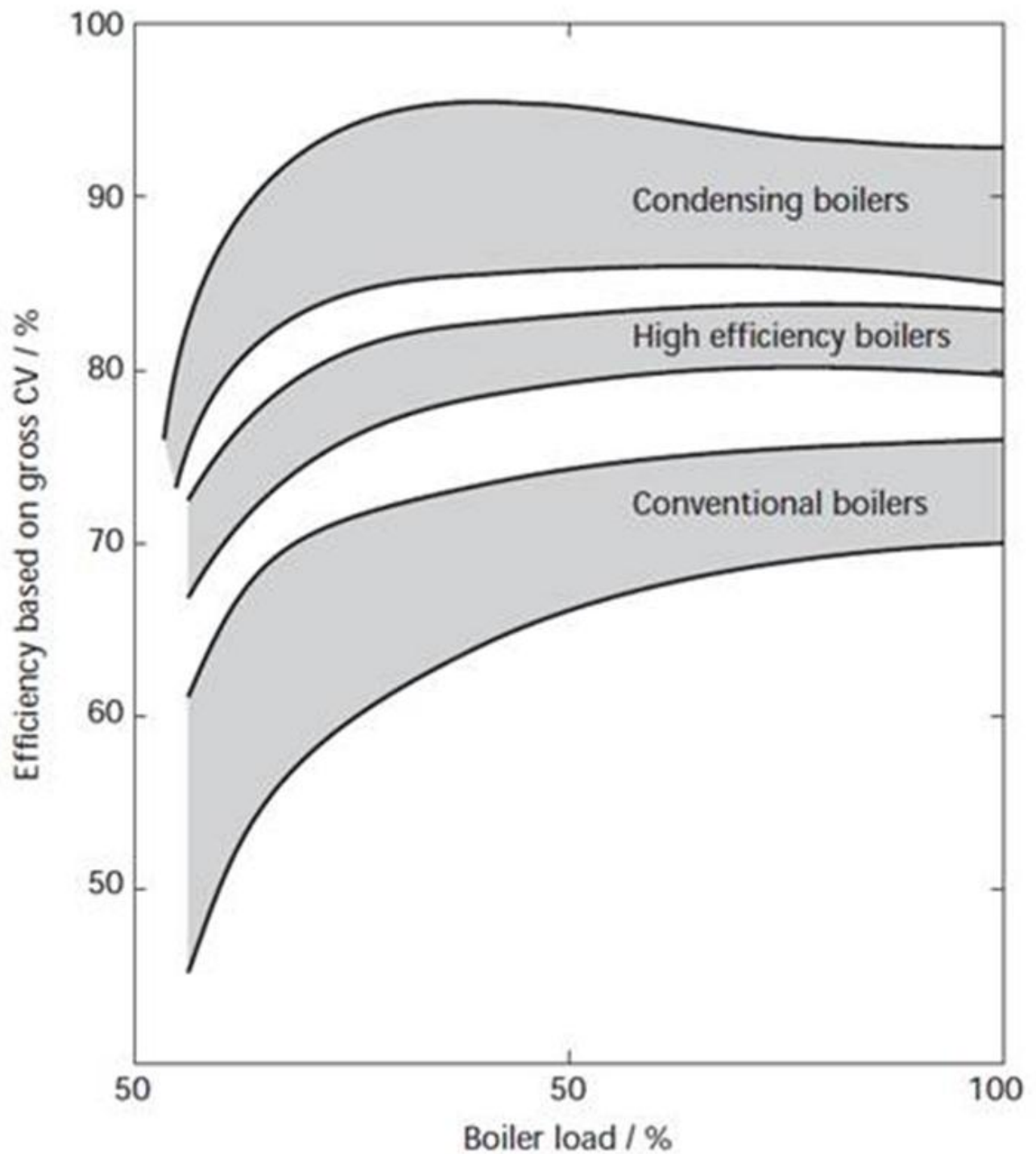
Eff = normalized boiler efficiency

PLR = boiler part-load ratio

T_w = boiler (outlet) water temperature [°C]

When using the normalized efficiency performance curve, if all coefficients are not required, simply set the unused coefficients to 0. For example, an electric boiler could be modelled by setting the nominal thermal efficiency to a value in the range of 0.96 to 1.0. **Coefficient 0** in the normalized efficiency performance curve would equal 1 and all other coefficients would be set to 0. Coefficients for other types of non-electric boilers would set a combination of the available coefficients to non-zero values.

The graph below (reproduced with permission from CIBSE) illustrates the range of typical efficiency for different types of boiler.



CIBSE Guide B Fig 1.4

Water Outlet

Design water flow rate

This numeric field contains the maximum design water volumetric flow rate (in m³/sec or gal/min). This should be the largest flow rate than can be heated to the design outlet temperature. This field is autosizable.

Water outlet upper temperature limit

This numeric field contains the outlet temperature upper limit (in °C or °F). It is used to cap the output from the boiler to avoid the boiler water outlet temperature from getting too high when the boiler is operating at less than the design flow rate, or when the return water temperature is relatively high. It is similar to the [Leaving chilled water lower temperature limit](#) for chillers.

Warning: The **Water outlet upper temperature limit** must be at least 5°C higher than the design supply loop outlet heating setpoint temperature to avoid the boiler cutting out during the simulation. Using temperatures equal to the supply loop design outlet temperature typically prevents boiler operation completely. **You are advised to leave this setting at 100°C even when supplying hot water at much lower temperatures.**

Part Load Ratios

Minimum part load ratio

This numeric field contains the minimum part load ratio. If the ratio of demand to boiler nominal capacity is less than the minimum part load ratio, then the Min PLR will determine the operating PLR. The expected range is between 0 and 1.

Maximum part load ratio

This numeric field contains the maximum part load ratio. If the ratio of demand to boiler nominal capacity is greater than the maximum part load ratio, then the Max PLR will determine the operating PLR. This value may exceed 1, but the normal range is between 0 and 1.1.

Optimum part load ratio

This numeric field contains the optimum part load ratio, i.e. is the part load ratio at which the boiler performs at its maximum efficiency. It is used in the load management calculations when the [Plant loop Load distribution scheme](#) is set to **2-Optimal**. A typical value will be 1, but for condensing gas boilers, the optimum part load ratio is much lower at about 0.3.

Boiler Outputs

HVAC,Average,Boiler Heating Output Rate [W]
 HVAC,Sum,Boiler Heating Output Energy [J]
 Zone,Meter,Boilers:EnergyTransfer [J]
 HVAC,Average,Boiler Water Inlet Temp [C]
 HVAC,Average,Boiler Water Outlet Temp [C]
 HVAC,Average,Boiler Water Mass Flow Rate [kg/s]
 HVAC,Average,Boiler Parasitic Electric Power [W]
 HVAC,Sum,Boiler Parasitic Electric Consumption [J]
 HVAC,Average,Boiler Part-Load Ratio

One of the following blocks will be applicable based on fuel type:

HVAC,Average,Boiler Electric Consumption Rate [W]
 HVAC,Sum,Boiler Electric Consumption [J]
 Zone,Meter, Electricity:Plant [J]
 Zone,Meter,Heating:Electricity [J]

HVAC,Average,Boiler Gas Consumption Rate [W]
 HVAC,Sum,Boiler Gas Consumption [J]
 Zone,Meter,Gas:Plant [J]
 Zone,Meter,Heating:Gas [J]

HVAC,Average,Boiler Propane Consumption Rate [W]

HVAC,Sum,Boiler Propane Consumption [J]
 Zone,Meter, Propane:Plant [J]
 Zone,Meter,Heating:Propane [J]

HVAC,Average,Boiler FuelOil#1 Consumption Rate [W]
 HVAC,Sum,Boiler FuelOil#1 Consumption [J]
 Zone,Meter, FuelOil#1:Plant [J]
 Zone,Meter,Heating:FuelOil#1 [J]

HVAC,Average,Boiler FuelOil#2 Consumption Rate [W]
 HVAC,Sum,Boiler FuelOil#2 Consumption [J]
 Zone,Meter, FuelOil#2:Plant [J]
 Zone,Meter,Heating:FuelOil#2 [J]

HVAC,Average,Boiler Coal Consumption Rate [W]
 HVAC,Sum,Boiler Coal Consumption [J]
 Zone,Meter,Coal:Plant [J]
 Zone,Meter,Heating:Coal [J]

HVAC,Average,Boiler Diesel Consumption Rate [W]
 HVAC,Sum,Boiler Diesel Consumption [J]
 Zone,Meter, Diesel:Plant [J]
 Zone,Meter,Heating:Diesel [J]

HVAC,Average,Boiler Gasoline Consumption Rate [W]
 HVAC,Sum,Boiler Gasoline Consumption [J]
 Zone,Meter, Gasoline:Plant [J]
 Zone,Meter,Heating:Gasoline [J]

Boiler Heating Output Rate [W]

Boiler Heating Output Energy [J]

These outputs are the heating output (load) of the boiler. Energy is metered on Boilers:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Boiler Water Inlet Temp [C]

Boiler Water Outlet Temp [C]

Boiler Water Mass Flow Rate [kg/s]

These outputs are the hot water inlet and outlet temperatures and flow rate for the boiler.

Boiler Part-Load Ratio

This output is the operating part-load ratio of the boiler. The part-load ratio is calculated as the boiler load divided by the boiler nominal capacity. The part-load ratio is limited by the minimum and maximum part-load ratio inputs specified by the user.

Boiler <FuelType> Consumption Rate [W] Boiler <FuelType> Consumption [J]

These outputs are the energy input to the boiler. Valid fuel types are: Electric, Gas (natural gas), Propane, FuelOil#1, FuelOil#2, Coal, Diesel, and Gasoline. Consumption is metered on Heating:<FuelType>, <FuelType>:Plant, and <FuelType>:Facility.

Boiler Parasitic Electric Power [W] Boiler Parasitic Electric Consumption [J]

These outputs are the parasitic electric power and parasitic electric consumption associated with the boiler. Used when simulating a forced draft fan or other electric component. Consumption is metered on Heating:Electricity and Electricity:Plant.

Chillers

Chillers provide chilled water to cooling coils, chilled beams and cooled ceilings. They can be placed using the [Add Chiller](#) command when you are located in a Chilled water supply loop.

When using Detailed HVAC, to edit an existing chiller go to the **Chilled water supply loop** (HVAC System > Chilled water loop > Supply loop) level where the chiller is placed, click on the chiller icon to highlight it then either right click and select the **Edit selected component** menu option or, when using [Learning](#) mode, click on the Edit icon in at the top of the info panel.

These chiller types are available:

- [1-Constant CoP](#) - a simple model with constant efficiency and no performance curves.
- [Chiller - EIR](#) - a more detailed model using 3 performance curves to determine cooling capacity and CoP from the chiller load and outside air temperature.
- [3-Reformulated EIR](#) - an empirical model similar to the Chiller EIR model.
- [4-Absorption](#) - the basic absorption chiller model.
- [5-Indirect absorption](#) - a more advanced absorption chiller model with more flexible performance curves and ability to model the effect of varying evaporator, condenser, and generator temperatures.

Chiller - Constant CoP



Chiller:ConstantCOP

Used in:

- Chilled water loop, supply side

This chiller model is based on a simple, constant COP simulation of the chiller. In this case, performance does not vary with chilled water temperature or condenser conditions. Such a model can be useful when detailed performance data is not available.

General

Name

The auto-generated name of the chiller can be edited.

Chiller type

The type of chiller can be one of these options:

- [1-Constant CoP](#) - the simple constant CoP model described on this page.
- [2-Chiller EIR](#) - a more detailed model using 3 performance curves to determine cooling capacity and CoP from the chiller load and outside air temperature as described on this page.
- [3-Reformulated EIR](#) - an empirical model similar to the EIR model
- [4-Absorption](#) - the basic absorption chiller model
- [5-Indirect absorption](#) - a more advanced absorption chiller model with more flexible performance curves and ability to model the effect of varying evaporator, condenser, and generator temperatures.

Note: The Chiller type cannot be edited directly. To help you to ensure that appropriate performance curves are used, you must either load a new chiller template of the type required or add a new chiller, selecting the appropriate type from the drop list.

Nominal capacity

This numeric field contains the nominal cooling capacity of the chiller (in W or Btu/h). Alternatively, this field can be autosized.

Nominal COP

This numeric field contains the chiller's coefficient of performance to be used for all operating conditions in the simulation. Unlike Chiller EIR, the CoP entered for Constant CoP chillers should include the energy consumed by the condenser fans for air and evaporatively cooled chillers.

Chiller flow mode

This choice field determines how the chiller operates with respect to the intended fluid flow through the device's evaporator. There are three choices for specifying operating modes for the intended flow behaviour:

- **1-Constant flow** is the default option for constant volume chilled water loops and is useful for constant speed pumping arrangements where the chiller's request for flow is stricter and can increase the overall loop flow.
- **2-Leaving setpoint modulated** is the default option for variable volume chilled water loops and changes the chiller model to internally vary the flow rate so that the temperature leaving the chiller matches a setpoint at the evaporator outlet.
- **3-Not modulated** is useful for either variable or constant speed pumping arrangements where the chiller is passive in the sense that although it makes a nominal request for its design flow rate, it can operate at varying flow rates.

In all cases the operation of the external plant system can also impact the flow through the chiller - for example if the relative sizes and operation are such that flow is restricted and the requests cannot be met.

For *variable* flow chilled water loops these options are available: **2-Leaving setpoint modulated** and **3-Not modulated**.

For *constant* flow chilled water loops these options are available: **1-Constant flow** and **3-Not modulated**.

The type of loop (variable/constant flow) can be changed by modifying the [Plant loop flow type](#) on the [Chilled water plant loop dialog](#).

Note: When the **2-Leaving setpoint modulated** option is selected then you must add an extra Setpoint manager immediately downstream of the chiller chilled water outlet to define the temperature of the water supplied.

Sizing factor

The sizing factor is used when the chiller design inputs are autosized. In this case the autosizing results are multiplied by this additional sizing factor. The usual value to enter is 1.0.

The inputs that would be altered by the sizing factor are: [Nominal capacity](#), [Design chilled water flow rate](#) and [Design condenser water flow rate](#).

The most common use of the sizing factor is for sizing chillers to meet only part of the design load while continuing to use the autosizing feature. For example when a set of chillers is chained together to supply chilled water to a plant loop, this sizing factor can be used to indicate the proportion of the load to be met by each chiller.

See also the section on [Autosizing HVAC Components](#)

Condenser

Condenser type

The condenser type determines what type of condenser will be included with this chiller. Valid condenser types are:

- **1-Air cooled,**
- **2-Water cooled,**
- **3-Evaporatively cooled**

The default is **2-Water cooled** which requires the full specification of the Condenser loop and its associated equipment. **1-Air cooled** and **3-Evaporatively cooled** do not require a Condenser loop to be specified.

Flow Rates

Design chilled water flow rate

For a variable flow chiller this is the maximum water flow rate and for a constant flow chiller this is the design water flow rate through the chiller's evaporator. The units are (in m3/s or gal/min). This numeric input field must be greater than zero, or it can be autosized.

Design condenser fluid flow rate

This numeric field contains the chiller's operating condenser water flow rate (in m3/s or gal/min). This field can be autosized.

This field is not used **Air cooled** and **Evaporatively cooled** [Condenser types](#).

Basin Heater

Basin heater capacity

This numeric field contains the capacity of the DX coil's electric evaporative cooler basin heater in (W/K or Btu/h-F). This field only applies for [Condenser type](#) = **Evaporatively cooled**. This field is used in conjunction with the Basin Heater Setpoint Temperature described in the following field. The basin heater electric power is equal to this field multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the chiller is off, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero.

Basin heater setpoint temperature

This numeric field contains the set point temperature (in °C or °F) for the basin heater described in the previous field. This field only applies for [Condenser type](#) = **Evaporatively cooled**. The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the chiller is off. This set point temperature must be greater than or equal to 2°C.

Basin heater operating schedule

This alpha field contains the name of the basin heater operating schedule. This field only applies for [Condenser type](#) = **Evaporatively cooled**. The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin heater operates when scheduled on and the outdoor air dry-bulb temperature is below the set point temperature described in the previous field. Regardless of this schedule, the basin heater can only operate when the chiller is off.

Generic Chiller Outputs

Many output variable names are common across all chiller types. These generic chiller output names all begin with the word "Chiller". Certain chiller types have additional output variables which are specific to that type of chiller. Specific chiller output names begin with the chiller type, for example, "Gas Absorption Chiller Heating Energy [J]." Chiller energy use is added to the appropriate plant-level meters as a cooling end-use.

HVAC,Average,Chiller Electric Power [W]
HVAC,Sum,Chiller Electric Consumption [J]
Zone,Meter,Electricity:Plant [J]
Zone,Meter,Cooling:Electricity [J]
HVAC,Average,Chiller Evap Heat Trans Rate [W]
HVAC,Sum,Chiller Evap Heat Trans [J]
Zone,Meter,EnergyTransfer:Plant [J]
Zone,Meter,Chillers:EnergyTransfer [J]
HVAC,Average,Chiller Evap Water Inlet Temp [C]
HVAC,Average,Chiller Evap Water Outlet Temp [C]
HVAC,Average,Chiller Evap Water Mass Flow Rate [kg/s]
HVAC,Average,Chiller Cond Heat Trans Rate [W]
HVAC,Sum,Chiller Cond Heat Trans [J]
Zone,Meter,HeatRejection:EnergyTransfer [J]
HVAC,Average,Chiller COP [W/W]

The following output is applicable only for air-cooled or evap-cooled chillers:

HVAC,Average,Chiller Cond Air Inlet Temp [C]

The following outputs are applicable only for evap-cooled chillers:

HVAC,Average,Chiller Basin Heater Electric Power [W]
HVAC,Average,Chiller Basin Heater Electric Consumption [J]

The following three outputs are only available for water-cooled chillers:

HVAC,Average,Chiller Cond Water Inlet Temp [C]
HVAC,Average,Chiller Cond Water Outlet Temp [C]
HVAC,Average,Chiller Cond Water Mass Flow Rate [kg/s]
HVAC,Average,Chiller Shaft Power [W]

Chiller Electric Power [W]

Chiller Electric Consumption [J]

These outputs are the electric power input to the chiller. In the case of steam or fuel-powered chillers, this represents the internal chiller pumps and other electric power consumption. Consumption is metered on Cooling:Electricity, Electricity:Plant, and Electricity:Facility.

Chiller Evap Heat Trans Rate [W]

Chiller Evap Heat Trans [J]

These outputs are the evaporator heat transfer which is the cooling delivered by the chiller. Chiller Evap Heat Trans is metered on Chillers:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Chiller Evap Water Inlet Temp [C]

Chiller Evap Water Outlet Temp [C]

Chiller Evap Water Mass Flow Rate [kg/s]

These outputs are the evaporator (chilled water) inlet and outlet temperatures and flow rate.

Chiller COP [W/W]

This output is the coefficient of performance for the chiller during cooling operation. It is calculated as the evaporator heat transfer rate (Chiller Evap Heat Trans Rate) divided by the “fuel” consumption rate by the chiller. For the constant COP and electric chillers, the “fuel” is electricity so the divisor is Chiller Electric Power [W]. For the absorption chiller, the “fuel” is steam so the divisor is Steam Consumption Rate [W].

For the engine driven chiller and combustion turbine chiller, the output variable is renamed as Chiller Fuel COP to clarify that the primary energy input to the chiller is a gaseous or liquid fuel (natural gas, diesel, gasoline, etc.). The divisor is the appropriate fuel consumption rate (Chiller [fuel type] Consumption Rate).

For the direct fired absorption chiller, this variable is renamed as Direct Fired Absorption Chiller Cooling Fuel COP and the divisor is Direct Fired Absorption Chiller Cooling Gas Consumption Rate.

Note that this variable is reported as zero when the chiller is not operating. When reported for frequencies longer than "detailed" (such as timestep, hourly, daily, monthly or environment), this output will only be meaningful when the chiller is operating for the entire reporting period. To determine an average COP for a longer time period, compute the COP based on total evaporator heat transfer divided by total electric or fuel input over the desired period.

Chiller Part-Load Ratio

This output is the operating part-load ratio of the indirect absorption chiller. This output may fall below the minimum part-load ratio specified in the input. For this case, the Chiller Cycling Fraction is used to further define the performance of the indirect absorption chiller.

Chiller Cycling Fraction

This output is the fraction of the timestep the indirect absorption chiller operates. When the chiller operates above the minimum part-load ratio, a chiller cycling fraction of 1 is reported. When the chiller operates below the minimum part-load ratio, the chiller cycling fraction reports the fraction of the timestep the indirect absorption chiller operates.

Chiller Cond Heat Trans Rate [W]

Chiller Cond Heat Trans [J]

These outputs are the condenser heat transfer which is the heat rejected from the chiller to either a condenser water loop or through an air-cooled condenser. Chiller Cond Heat Trans is metered on HeatRejection:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Chiller Cond Air Inlet Temp [C]

This output is the condenser (heat rejection) inlet temperature for air-cooled or evap-cooled chillers. For an air-cooled chiller, this output would be the dry-bulb temperature of the air entering the condenser coil. For an evap-cooled chiller, this output would be the wet-bulb temperature of the air entering the evaporatively-cooled condenser coil.

Chiller Basin Heater Electric Power [W]

Chiller Basin Heater Electric Consumption [J]

These outputs are the electric power input to the chiller's basin heater (for evaporativelycooled condenser type). Consumption is metered on Chillers:Electricity, Electricity:Plant, and Electricity:Facility

Chiller Cond Water Inlet Temp [C]

Chiller Cond Water Outlet Temp [C]

Chiller Cond Water Mass Flow Rate [kg/s]

These outputs are the condenser (heat rejection) inlet and outlet temperatures and flow rate for water-cooled chillers.

Chiller Shaft Power [W]

Chiller Shaft Energy [J]

For engine-driven and turbine-driven chillers, these outputs are the shaft power produced by the prime mover and transferred to the chiller compressor.

Chiller Lube Heat Recovery Rate [W]

Chiller Lube Heat Recovery [J]

Chiller Jacket Heat Recovery Rate [W]

Chiller Jacket Heat Recovery [J]

Chiller Exhaust Heat Recovery Rate [W]

Chiller Exhaust Heat Recovery [J]

Chiller Total Heat Recovery Rate [W]

Chiller Total Heat Recovery [J]

For chillers with heat recovery, such as engine-driven chillers, these outputs are the components of recoverable energy available. For a given chiller type, one or more of the following components may be applicable: Lube (engine lubricant), Jacket (engine coolant), Exhaust (engine exhaust), and Total. Chiller Lube Heat Recovery, Chiller Jacket Heat Recovery, and Chiller Exhaust Heat Recovery are metered on HeatRecovery:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Chiller Exhaust Stack Temp [C]

This is the exhaust temperature leaving an engine chiller.

Chiller Heat Recovery Inlet Temp [C]

Chiller Heat Recovery Outlet Temp [C]

Chiller Heat Recovery Mass Flow Rate [kg/s]

These outputs are the heat recovery inlet and outlet temperatures and flow rate for chillers with heat recovery such as engine-driven and gas turbine chillers.

Chiller <FuelType> Consumption Rate [W]**Chiller <FuelType> Consumption [J]****Chiller <FuelType> Mass Flow Rate [kg/s]****Chiller Gas Mass Consumption [kg] (Gas Turbine Chiller only)**

These outputs are the steam or fuel input for steam or fuel-fired chillers. Valid fuel types depend on the type of chiller. <FuelType> may be one of: Gas (natural gas), Steam, Propane, Diesel, Gasoline, FuelOil#1, and FuelOil#2. Consumption is metered on Cooling:<FuelType>, <FuelType>:Plant, and <FuelType>:Facility.

Chiller EIR

Chiller:Electric:EIR

Used in:

- Chilled water loop, supply side

The Chiller EIR model is the empirical model used in the DOE-2.1 building energy simulation program. It uses performance information at reference conditions along with three curve fits for cooling capacity and efficiency to determine chiller operation at off-reference conditions. Chiller performance curves can be generated by fitting manufacturer's catalog data or measured data. Performance curves for more than 300 chillers, including the default DOE-2.1E reciprocating and centrifugal chillers, are provided as templates for selection within the chiller dialog. This data comes from in the EnergyPlus Reference DataSets (Chillers.idf).

See the [Chilled water loops chillers and fan coil units](#) tutorial

Note: Chiller EIR components and their associated performance curve objects are developed using performance information for a specific chiller and should normally be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with caution.

General**Name**

The auto-generated name of the chiller can be edited.

Chiller template

Use this browse option to select a chiller from the EnergyPlus chiller database whose performance data you wish to copy to your chiller.

Chiller type

The type of chiller can be one of these options:

- [1-Constant CoP](#) - a simple model with constant efficiency and no performance curves.
- [2-Chiller EIR](#) - a more detailed model using 3 performance curves to determine cooling capacity and CoP from the chiller load and outside air temperature as described on this page.
- [3-Reformulated EIR](#) - an empirical model similar to the Chiller EIR model.
- [4-Absorption](#) - the basic absorption chiller model.

- **5-Indirect absorption** -a more advanced absorption chiller model with more flexible performance curves and ability to model the effect of varying evaporator, condenser, and generator temperatures.

Note: The Chiller type cannot be edited directly. To help you to ensure that appropriate performance curves are used, you must either load a new chiller template of the type required or add a new chiller, selecting the appropriate type from the drop list.

Reference capacity

This numeric field contains the reference cooling capacity of the chiller (in W or Btu/h). This should be the capacity of the chiller at the reference temperatures and water flow rates defined below. Alternatively, this field can be autosized.

Reference COP

This numeric field contains the chiller's coefficient of performance which is multiplied by the output of the [chiller performance curves](#) described below. This value should *not* include energy use due to pumps, evaporated or air-cooled condenser fans, or cooling tower fans. This COP should be at the [reference temperatures](#) and [water flow rates](#) defined below.

Compressor motor efficiency

This numeric input represents the fraction of compressor electrical energy consumption that must be rejected by the condenser. Enter a value of 1.0 when modelling hermetic chillers. This value must be between 0.0 and 1.0, with a default value of 1.0.

Chiller flow mode

This option determines how the chiller operates with respect to the intended fluid flow through the device's evaporator. There are three choices for specifying operating modes for the intended flow behaviour:

- **1-Constant flow** is the default option for constant volume chilled water loops and is useful for constant speed pumping arrangements where the chiller's request for flow is stricter and can increase the overall loop flow.
- **2-Leaving setpoint modulated** is the default option for variable volume chilled water loops and changes the chiller model to internally vary the flow rate so that the temperature leaving the chiller matches a setpoint at the evaporator outlet.
- **3-Not modulated** is useful for either variable or constant speed pumping arrangements where the chiller is passive in the sense that although it makes a nominal request for its design flow rate, it can operate at varying flow rates.

In all cases the operation of the external plant system can also impact the flow through the chiller - for example if the relative sizes and operation are such that flow is restricted and the requests cannot be met.

For *variable* flow chilled water loops these options are available: **2-Leaving setpoint modulated** and **3-Not modulated**.

For *constant* flow chilled water loops these options are available: **1-Constant flow** and **3-Not modulated**.

The type of loop (variable/constant flow) can be changed by modifying the [Plant loop flow type](#) on the [Chilled water plant loop dialog](#).

Note: When the **2-Leaving setpoint modulated** option is selected then you must add an extra Setpoint manager immediately downstream of the chiller chilled water outlet to define the temperature of the water supplied.

Sizing factor

The sizing factor is used when the chiller design inputs are autosized. In this case the autosizing results are multiplied by this additional sizing factor. The usual value to enter is 1.0.

The inputs that would be altered by the sizing factor are: [Reference capacity](#), [Reference chilled water flow rate](#) and [Reference condenser water flow rate](#).

The most common use of the sizing factor is for sizing chillers to meet only part of the design load while continuing to use the autosizing feature. For example when a set of chillers is chained together to supply chilled water to a plant loop, this sizing factor can be used to indicate the proportion of the load to be met by each chiller.

See also the section on [Autosizing HVAC Components](#)

Condenser

Condenser type

The condenser type determines what type of condenser will be included with this chiller. Valid condenser types are:

- **1-Air cooled**,
- **2-Water cooled**,
- **3-Evaporatively cooled**

The default is **2-Water cooled** which requires the full specification of the Condenser loop and its associated equipment. **1-Air cooled** and **3-Evaporatively cooled** do not require a Condenser loop to be specified.

Note: **Condenser type** cannot be edited directly. To help you to ensure that appropriate performance curves are used, load a chiller of the correct type from the [Chiller template](#).

Condenser fan power ratio

This data is used to model condenser fan power associated with air-cooled or evaporatively cooled condensers. Enter the ratio of the condenser fan power to the reference chiller cooling capacity in W/W.

During the simulation the condenser fan power for air-cooled or evaporatively cooled chillers is calculated as total chiller power consumption multiplied by this ratio.

Temperatures

Reference leaving chilled water temperature

This numeric field contains the chiller's reference leaving chilled water temperature (in °C or °F). The default value is 6.67°C.

Reference entering condenser fluid temperature

This numeric field contains the chiller's reference entering condenser fluid temperature (in °C or °F). The default value is 29.4°C. For water-cooled chillers this is the water temperature entering the condenser (e.g., leaving the cooling tower). For air- or evap-cooled condensers this is the entering outdoor air dry-bulb or wet-bulb temperature, respectively.

Leaving chilled water lower temperature limit

This numeric field contains the lower limit for the leaving chilled water temperature (in °C or °F). This temperature acts as a cut off for heat transfer in the evaporator, so that the fluid doesn't get too cold. The default value is 2°C.

Note: This input field is currently unused.

Flow Rates

Reference chilled water flow rate

For a variable flow chiller this is the maximum water flow rate and for a constant flow chiller this is the operating water flow rate through the chiller's evaporator. The units are (in m³/s or gal/min). This numeric input field must be greater than zero, or it can be autosized.

Reference condenser fluid flow rate

This numeric field contains the chiller's operating condenser water flow rate (in m³/s or gal/min). This field can be autosized.

Note: For water cooled chillers, EnergyPlus sets the condenser fluid flow rate to the maximum value whenever the chiller is operating. The [Chiller flow mode](#) only affects the chiller evaporator flow rate.

Performance Curves

Cooling capacity function of temperature curve (CAPFT)

The [Bi-quadratic performance curve](#) parameterizes the variation of the cooling capacity as a function of the leaving chilled water temperature and the entering condenser fluid temperature. The output of this curve is multiplied by the reference capacity to give the cooling capacity at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified above. The bi-quadratic curve should be valid for the range of water temperatures anticipated for the simulation.

Electric input to cooling output ratio function of temperature curve (EIRFT)

The [Bi-quadratic performance curve](#) parameterizes the variation of the energy input to cooling output ratio (EIR) as a function of the leaving chilled water temperature and the entering condenser fluid temperature. The EIR is the inverse of the COP. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) to give the EIR at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified above. The bi-quadratic curve should be valid for the range of water temperatures anticipated for the simulation.

Electric input to cooling output ratio function of part load ratio curve (EIRPLR)

The [Quadratic performance curve](#) parameterizes the variation of the energy input ratio (EIR) as a function of the part-load ratio. The EIR is the inverse of the COP, and the part-load ratio is the actual cooling load divided by the chiller's available cooling capacity.

This curve is generated by dividing the operating electric input power by the available full-load capacity (do not divide by load) at the specific operating temperatures. The curve output should decrease from 1 towards 0 as part-load ratio decreases from 1 to 0. The output of this curve is multiplied by the reference full-load EIR (inverse of the reference COP) and the [Electric input to cooling output ratio function of temperature curve \(EIRFT\)](#) to give the EIR at the specific temperatures and part-load ratio at which the chiller is operating. This curve should have a value of 1.0 when the part-load ratio equals 1.0.

An ideal chiller with the same efficiency at all part-load ratio's would use a performance curve that has a value of 0 when the part-load ratio equals 0 (i.e., a line connecting 0,0 and 1,1 when plotted as EIR versus PLR), however, actual systems can have part-load EIR's slightly above or below this line (i.e., part-load efficiency often differs from rated efficiency). The quadratic curve should be valid for the range of part-load ratios anticipated for the simulation.

Tip: If you cannot obtain manufacturers data for this curve then a reasonable estimate can be obtained by using a straight line from 0,0 to 1,1. That is for the quadratic curve, $C_0 = 0$, $C_1 = 1$ and $C_2 = 0$.

Advice on chiller performance curves

During the simulation the electric power consumption of the compressor is calculated using:

$$Q_{\text{compressor}} = (Q_{\text{ref}} / \text{CoP}_{\text{ref}}) * \text{CAPFT} * \text{EIRFT} * \text{EIRPLR}$$

Where CAPFT, EIRFT and EIRPLR are the outputs from the 3 curves above.

If you have manufacturers data you can generate the inputs for these 3 curves using the [Chiller PerformanceCurve Coefficients.xls](#) spreadsheet provided by EnergyPlus. When using this spreadsheet to generate inputs for EIRPLR note that the coefficients are read from the spreadsheet into the DesignBuilder Quadratic Curve dialog as follows:



Of course the performance and capacity of a chiller depends on various factors, the most important of these being part load and temperature chilled water and the Chiller EIR provides an accurate representation taking these factors into account. However in some cases you may only have summary manufacturers data. For example you may only have information on the variation of CoP with part load ratio. In this case you can define the EIRPLR curve in the usual way using your summary manufacturer's data and use constant values for the CAPFT and EIRFT curves which both generate a constant value of 1.0. Because CAPFT and EIRFT are both [Bi-quadratic](#) curves you cannot use a constant Linear curve here so you must choose inputs to a special simple Bi-quadratic curve that generates an output of 1 regardless of inputs. To achieve this simply

create and select a new Bi-quadratic curve with Coefficient 1 = 1 and all other coefficients = 0 for CAPFT and EIRFT.

Part Load Settings

Minimum part load ratio

This numeric field contains the chiller's minimum part-load ratio. The expected range is between 0 and 1. Below this part-load ratio, the compressor cycles on and off to meet the cooling load. The **Minimum part load ratio** must be less than or equal to the **Maximum part load ratio**. The default value is 0.1.

Note: Setting the Minimum part load ratio to zero for air cooled chillers, will cause the condenser fan to run constantly even at zero load which is likely to lead to high chiller fan consumption. This happens because the condenser fan cannot cycle off even at zero chiller load.

Maximum part load ratio

This numeric field contains the chiller's maximum part-load ratio. This value may exceed 1, but the normal range is between 0 and 1.0. The **Maximum part load ratio** must be greater than or equal to the **Minimum part load ratio**. The default value is 1.0.

Optimum part load ratio

This numeric field contains the chiller's optimum part-load ratio. This is the part-load ratio at which the chiller performs at its maximum COP. The **optimum part-load ratio** must be greater than or equal to the **Minimum part load ratio**, and less than or equal to the **Maximum part load ratio**. The default value is 1.0.

Minimum unloading ratio

This numeric field contains the chiller's minimum unloading ratio. The expected range is between 0 and 1. The minimum unloading ratio is where the chiller capacity can no longer be reduced by unloading and must be false loaded to meet smaller cooling loads. A typical false loading strategy is hot-gas bypass. The **minimum unloading ratio** must be greater than or equal to the **Minimum part load ratio**, and less than or equal to the **Maximum part load ratio**. The default value is 0.2.

False loading

Different chiller types (reciprocating, centrifugal etc) have very different operating characteristics. The point at which they switch off at low load (cycle on and off to meet small loads) will vary between about 0.25 for older reciprocating chillers and 0.1 for modern centrifugal chillers which have much better part-load performance. Sometimes a manufacturer or installer may need to introduce a mechanism to stop the chiller cycling on and off too frequently (frequent cycling causes excessive wear and reduces the equipment life). To stop this happening some of the hot gas from the compressor outlet can be re-injected back into the return side of the compressor so that the compressor inlet senses a higher return gas temperature and is 'fooled' into thinking that there is a higher cooling load on the system than there actually is (this basically mimics a higher evaporator leaving temperature) so it needs to work a little harder to meet the load and the load is therefore maintained above the point at which the compressor would normally switch off. This strategy is called false-loading and it is clear that bypassing hot gas from the discharge to the suction side of the compressor results in higher energy consumption so is generally not considered to be an acceptable control strategy and actually contravenes some energy codes (e.g. Part L2 in UK). This strategy is sometimes used to overcome poor design and control problems.

You can avoid false loading in your model by setting the **Minimum unloading ratio** equal to the **Minimum part load ratio**, so when the chiller reaches its minimum part-load ratio it just switches off and false-loading cannot occur.

Heat Recovery

Heat recovery

For water cooled chillers, if heat is to be recovered from the condenser check this checkbox and ensure that a heat recovery loop is specified and attached to the chiller.

Note: Use the **Add heat recovery** loop command at HVAC system level to create a DHW loop that can be connected to a water-cooled chiller with heat recovery. This allows the waste heat from the chiller to be used to heat up a water tank which can be used to provide hot water for DHW, zone heaters or any other heating applications.

Design heat recovery water flow rate

For water cooled chillers with heat recovery enter the design heat recovery water flow rate (in m3/s or gal/min). This field is autosizable. When autosizing, the flow rate is simply the product of the design condenser flow rate and the condenser heat recovery relative capacity fraction set in the field below.

Condenser heat recovery relative capacity fraction

This field is used to describe the physical size of the heat recovery portion of a split bundle condenser section. This fraction describes the relative capacity of the heat recovery bundle of a split condenser compared to the nominal, full load heat rejection rate of the chiller. This fraction will be applied to the full heat rejection when operating at nominal capacity and nominal COP to model a capacity limit for the heat rejection. The default capacity fraction is 1.

Inlet high temperature limit

Check this checkbox if you would like to define an upper limit for the return fluid temperatures entering the chiller at the heat recovery inlet node (see below)

Heat recovery inlet high temperature limit schedule

If the Inlet high temperature limit checkbox is checked then you must select the schedule to be used to control heat recovery operation of the chiller. The schedule should contain temperature values, in °C, that describe an upper limit for the return fluid temperatures entering the chiller at the heat recovery inlet node. If the fluid temperature is too high, then the heat recovery will not operate. This is useful to restrict the chiller lift from becoming too high and to avoid overheating the hot water loop.

Generic Chiller Outputs

Many output variable names are common across all chiller types. These generic chiller output names all begin with the word "Chiller". Certain chiller types have additional output variables which are specific to that type of chiller. Specific chiller output names begin with the chiller type, for example, "Gas Absorption Chiller Heating Energy [J]." Chiller energy use is added to the appropriate plant-level meters as a cooling end-use.

```
HVAC,Average,Chiller Electric Power [W]
HVAC,Sum,Chiller Electric Consumption [J]
Zone,Meter,Electricity:Plant [J]
Zone,Meter,Cooling:Electricity [J]
HVAC,Average,Chiller Evap Heat Trans Rate [W]
HVAC,Sum,Chiller Evap Heat Trans [J]
Zone,Meter,EnergyTransfer:Plant [J]
Zone,Meter,Chillers:EnergyTransfer [J]
HVAC,Average,Chiller Evap Water Inlet Temp [C]
HVAC,Average,Chiller Evap Water Outlet Temp [C]
HVAC,Average,Chiller Evap Water Mass Flow Rate [kg/s]
HVAC,Average,Chiller Cond Heat Trans Rate [W]
HVAC,Sum,Chiller Cond Heat Trans [J]
Zone,Meter,HeatRejection:EnergyTransfer [J]
```

HVAC,Average,Chiller COP [W/W]

The following output is applicable only for air-cooled or evap-cooled chillers:

HVAC,Average,Chiller Cond Air Inlet Temp [C]

The following outputs are applicable only for evap-cooled chillers:

HVAC,Average,Chiller Basin Heater Electric Power [W]

HVAC,Average,Chiller Basin Heater Electric Consumption [J]

The following three outputs are only available for water-cooled chillers:

HVAC,Average,Chiller Cond Water Inlet Temp [C]

HVAC,Average,Chiller Cond Water Outlet Temp [C]

HVAC,Average,Chiller Cond Water Mass Flow Rate [kg/s]

HVAC,Average,Chiller Shaft Power [W]

Chiller Electric Power [W]

Chiller Electric Consumption [J]

These outputs are the electric power input to the chiller. In the case of steam or fuel-powered chillers, this represents the internal chiller pumps and other electric power consumption. Consumption is metered on Cooling:Electricity, Electricity:Plant, and Electricity:Facility.

Chiller Evap Heat Trans Rate [W]

Chiller Evap Heat Trans [J]

These outputs are the evaporator heat transfer which is the cooling delivered by the chiller. Chiller Evap Heat Trans is metered on Chillers:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Chiller Evap Water Inlet Temp [C]

Chiller Evap Water Outlet Temp [C]

Chiller Evap Water Mass Flow Rate [kg/s]

These outputs are the evaporator (chilled water) inlet and outlet temperatures and flow rate.

Chiller COP [W/W]

This output is the coefficient of performance for the chiller during cooling operation. It is calculated as the evaporator heat transfer rate (Chiller Evap Heat Trans Rate) divided by the “fuel” consumption rate by the chiller. For the constant COP and electric chillers, the “fuel” is electricity so the divisor is Chiller Electric Power [W]. For the absorption chiller, the “fuel” is steam so the divisor is Steam Consumption Rate [W].

For the engine driven chiller and combustion turbine chiller, the output variable is renamed as Chiller Fuel COP to clarify that the primary energy input to the chiller is a gaseous or liquid fuel (natural gas, diesel, gasoline, etc.). The divisor is the appropriate fuel consumption rate (Chiller [fuel type] Consumption Rate).

For the direct fired absorption chiller, this variable is renamed as Direct Fired Absorption Chiller Cooling Fuel COP and the divisor is Direct Fired Absorption Chiller Cooling Gas Consumption Rate.

Note that this variable is reported as zero when the chiller is not operating. When reported for frequencies longer than "detailed" (such as timestep, hourly, daily, monthly or environment), this output will only be meaningful when the chiller is operating for the entire reporting period. To determine an average COP for a longer time period, compute the COP based on total evaporator heat transfer divided by total electric or fuel input over the desired period.

Chiller Part-Load Ratio

This output is the operating part-load ratio of the indirect absorption chiller. This output may fall below the minimum part-load ratio specified in the input. For this case, the Chiller Cycling Fraction is used to further define the performance of the indirect absorption chiller.

Chiller Cycling Fraction

This output is the fraction of the timestep the indirect absorption chiller operates. When the chiller operates above the minimum part-load ratio, a chiller cycling fraction of 1 is reported. When the chiller operates below the minimum part-load ratio, the chiller cycling fraction reports the fraction of the timestep the indirect absorption chiller operates.

Chiller Cond Heat Trans Rate [W]

Chiller Cond Heat Trans [J]

These outputs are the condenser heat transfer which is the heat rejected from the chiller to either a condenser water loop or through an air-cooled condenser. Chiller Cond Heat Trans is metered on HeatRejection:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Chiller Cond Air Inlet Temp [C]

This output is the condenser (heat rejection) inlet temperature for air-cooled or evap-cooled chillers. For an air-cooled chiller, this output would be the dry-bulb temperature of the air entering the condenser coil. For an evap-cooled chiller, this output would be the wet-bulb temperature of the air entering the evaporatively-cooled condenser coil.

Chiller Basin Heater Electric Power [W]

Chiller Basin Heater Electric Consumption [J]

These outputs are the electric power input to the chiller's basin heater (for evaporativelycooled condenser type). Consumption is metered on Chillers:Electricity, Electricity:Plant, and Electricity:Facility

Chiller Cond Water Inlet Temp [C]

Chiller Cond Water Outlet Temp [C]

Chiller Cond Water Mass Flow Rate [kg/s]

These outputs are the condenser (heat rejection) inlet and outlet temperatures and flow rate for water-cooled chillers.

Chiller Shaft Power [W]

Chiller Shaft Energy [J]

For engine-driven and turbine-driven chillers, these outputs are the shaft power produced by the prime mover and transferred to the chiller compressor.

Chiller Lube Heat Recovery Rate [W]

Chiller Lube Heat Recovery [J]

Chiller Jacket Heat Recovery Rate [W]

Chiller Jacket Heat Recovery [J]

Chiller Exhaust Heat Recovery Rate [W]

Chiller Exhaust Heat Recovery [J]

Chiller Total Heat Recovery Rate [W]

Chiller Total Heat Recovery [J]

For chillers with heat recovery, such as engine-driven chillers, these outputs are the components of recoverable energy available. For a given chiller type, one or more of the following components may be applicable: Lube (engine lubricant), Jacket (engine coolant), Exhaust (engine exhaust), and Total. Chiller Lube Heat Recovery, Chiller Jacket Heat Recovery, and Chiller Exhaust Heat Recovery are metered on HeatRecovery:EnergyTransfer, EnergyTransfer:Plant, and EnergyTransfer:Facility.

Chiller Exhaust Stack Temp [C]

This is the exhaust temperature leaving an engine chiller.

Chiller Heat Recovery Inlet Temp [C]

Chiller Heat Recovery Outlet Temp [C]

Chiller Heat Recovery Mass Flow Rate [kg/s]

These outputs are the heat recovery inlet and outlet temperatures and flow rate for chillers with heat recovery such as engine-driven and gas turbine chillers.

Chiller <FuelType> Consumption Rate [W]

Chiller <FuelType> Consumption [J]

Chiller <FuelType> Mass Flow Rate [kg/s]

Chiller Gas Mass Consumption [kg] (Gas Turbine Chiller only)

These outputs are the steam or fuel input for steam or fuel-fired chillers. Valid fuel types depend on the type of chiller. <FuelType> may be one of: Gas (natural gas), Steam, Propane, Diesel, Gasoline, FuelOil#1, and FuelOil#2. Consumption is metered on Cooling:<FuelType>, <FuelType>:Plant, and <FuelType>:Facility.

Chiller Electric EIR Outputs

These reports are available only for Electric EIR Chillers.

Chiller Condenser Fan Electric Power [W]

Chiller Condenser Fan Electric Consumption [J]

These outputs are for the electric power consumption of the chiller condenser fan and are applicable to air- or evaporatively-cooled chillers. This output is also added to a report meter with Resource Type = Electricity, End Use Key = Chillers, Group Key = Plant (Ref. Report Meter).

ChillerCapFTemp

This is the output of the curve object Cooling Capacity Function of Temperature Curve.

ChillerEIRFTemp

This is the output of the curve object Electric Input to Cooling Output Ratio Function of Temperature Curve.

ChillerEIRFPLR

This is the output of the curve object Electric Input to Cooling Output Ratio Function of Part Load Curve.

Chiller Part Load Ratio

This output is the ratio of the evaporator heat transfer rate plus the false load heat transfer rate (if applicable) to the available chiller capacity. This value is used to determine ChillerEIRFPLR.

Chiller Cycling Ratio

The cycling ratio is the amount of time the chiller operates during each simulation timestep. If the chiller part-load ratio falls below the minimum part-load ratio, the chiller cycles on and off to meet the cooling load.

Chiller False Load Heat Trans Rate

Chiller False Load Heat Trans

These outputs are the heat transfer rate and total heat transfer due to false loading of the chiller. When the chiller part-load ratio is below the minimum unloading ratio, the chiller false loads (e.g. hot-gas bypass) to further reduce capacity. The false load heat transfer report variable is not metered.

Chiller:Electric:EIR Technical Description

Electric Chiller Model Based on Condenser Entering Temperature

Overview

This model (EnergyPlus name **Chiller:Electric:EIR**) simulates the performance of an electric liquid chiller. The model is based on the compression chiller model (COMREF) in the DOE-2.1 building energy simulation program. The EnergyPlus model contains all of the features of the DOE-2.1 chiller model, plus additional abilities for modelling evaporatively-cooled condensers and heat recovery for water heating.

This model simulates the thermal performance of the chiller and the power consumption of the compressor(s). It also models the power consumption of condenser fans if modelling an air-cooled or evaporatively-cooled condenser. This model does not simulate the thermal performance or the power consumption of associated [pumps](#) or [cooling towers](#).

Model Description

The chiller model uses user-supplied performance information at reference conditions along with three performance curves (curve objects) for cooling capacity and efficiency to determine chiller operation at off-reference conditions. The three performance curves are:

1. [Cooling Capacity Function of Temperature Curve](#)
2. [Energy Input to Cooling Output Ratio Function of Temperature Curve](#)
3. [Energy Input to Cooling Output Ratio Function of Part Load Ratio Curve](#)

Cooling capacity function of temperature curve

The cooling capacity function of temperature curve is a biquadratic performance curve with two independent variables: the leaving chilled water temperature and the entering condenser fluid temperature. The output of this curve is multiplied by the reference capacity to give the full-load cooling capacity at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified in the input data file by the user. The biquadratic curve should be valid for the range of water temperatures anticipated for the simulation.

$$\text{ChillerCapFTemp} = a + b(T_{\text{cw,l}}) + c(T_{\text{cw,l}})^2 + d(T_{\text{cond,e}}) + e(T_{\text{cond,e}})^2 + f(T_{\text{cw,l}})(T_{\text{cond,e}})$$

where:

ChillerCapFTemp = cooling capacity factor, equal to 1 at reference conditions

$T_{\text{cw,l}}$ = leaving chilled water temperature, °C

$T_{\text{cond,e}}$ = entering condenser fluid temperature, °C. For a water-cooled condenser this will be the water temperature returning from the condenser loop (e.g., leaving the cooling tower). For air- or evap-cooled condensers this will be the entering outdoor air dry-bulb or wet-bulb temperature, respectively.

Energy input to cooling output ratio function of temperature curve

The energy input to cooling output ratio function of temperature curve is a biquadratic performance curve that parameterises the variation of the energy input to cooling output ratio (EIR) as a function of the leaving chilled water temperature and the entering condenser fluid temperature. The EIR is the inverse of the COP. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) to give the full-load EIR at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified in the input data file by the user. The Biquadratic curve should be valid for the range of water temperatures anticipated for the simulation.

$$\text{ChillerEIRFTemp} = a + b(T_{\text{cw,l}}) + c(T_{\text{cw,l}})^2 + d(T_{\text{cond,e}}) + e(T_{\text{cond,e}})^2 + f(T_{\text{cw,l}})(T_{\text{cond,e}})$$

where:

ChillerEIRFTemp = energy input to cooling output factor, equal to 1 at reference conditions

$T_{\text{cw,l}}$ = leaving chilled water temperature, °C

$T_{\text{cond,e}}$ = entering condenser fluid temperature, °C. For a water-cooled condenser this will be the water temperature returning from the condenser loop (e.g., leaving the cooling tower). For air- or evap-cooled condensers this will be the entering outdoor air dry-bulb or wet-bulb temperature, respectively.

Energy input to cooling output ratio function of part-load ratio curve

The energy input to cooling output ratio function of part-load ratio curve is a quadratic performance curve that parameterizes the variation of the energy input ratio (EIR) as a function of the part-load ratio. The EIR is the inverse of the COP, and the part-load ratio is the actual cooling load divided by the chiller's available cooling capacity. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) and the Energy Input to Cooling Output Ratio Function of Temperature Curve to give the EIR at the specific temperatures and part-load ratio at which the chiller is operating. This curve should have a value of 1.0 when the part-load ratio equals 1.0. The quadratic curve should be valid for the range of part-load ratios anticipated for the simulation.

$$\text{ChillerEIRFPLR} = a + b \text{ PLR} + c \text{ PLR}^2$$

where:

ChillerEIRFPLR = energy input to cooling output factor, equal to 1 at reference conditions

PLR = part-load ratio = (cooling load) / (chiller's available cooling capacity)

All three of the performance curves are accessed through EnergyPlus' built-in performance curve equation manager (curve:quadratic and curve:biquadratic). It is not imperative that the user utilize all coefficients in the performance curve equations if their performance equation has fewer terms (e.g., if the user's ChillerEIRFPLR

performance curve is linear instead of quadratic, simply enter the values for a and b, and set coefficient c equal to zero).

Note: Chiller:Electric:EIR objects and their associated performance curve objects are developed using performance information for a specific chiller and should normally be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with caution.

For any simulation time step, the chiller's available cooling capacity is calculated as follows:

$$Q_{\text{avail}} = Q_{\text{ref}} \text{ChillerCapFTemp}$$

where:

Q_{ref} = chiller capacity at reference conditions (reference temperatures and flow rates defined by the user), W
 Q_{avail} = available chiller capacity adjusted for current fluid temperatures, W

The model then calculates the evaporator heat transfer rate required to bring the entering chilled water temperature down to the leaving chilled water setpoint temperature (established using a [Setpoint manager](#) and referenced in the Plant loop). If this calculated heat transfer rate is greater than the heat transfer rate being requested by the plant equipment operation scheme, then the evaporator heat transfer rate is reset to the requested cooling rate.

The evaporator heat transfer rate is then compared to the available capacity. If the available chiller capacity is sufficient to meet the evaporator heat transfer rate, the leaving chilled water temperature is set equal to the chilled water setpoint temperature. If the requested evaporator heat transfer rate is larger than the available capacity the chilled water leaving the evaporator is allowed to float upward. For this case, the exiting chilled water temperature is calculated based on the water temperature entering the evaporator, the available cooling capacity, and the evaporator mass flow rate.

Chiller - Reformulated EIR



Chiller:Electric:ReformulatedEIR

Used in:

- Chilled water loop, supply side

The Reformulated EIR chiller is an empirical model similar to the [Chiller EIR](#). The model uses performance information at reference conditions along with three curve fits for cooling capacity and efficiency to determine chiller operation at off-reference conditions. The model has the same capabilities as the Chiller EIR, but can potentially provide significant accuracy improvement for variable-speed compressor drive and variable condenser water flow applications. Chiller performance curves can be generated by fitting manufacturer's catalog data or measured data.

Tip: Performance curves developed from manufacturer's performance data are provided in the Chiller templates.

This chiller model can be used to predict the performance of various chiller types (e.g., reciprocating, screw, scroll, and centrifugal) with water-cooled condensers. The main difference between this model and the [Chiller EIR](#) model is the condenser fluid temperature used in the associated performance curves: the Chiller Reformulated EIR model uses the **leaving** condenser water temperature while the Chiller EIR model uses the **entering** condenser water temperature. Although this model can provide more accurate prediction than the Chiller EIR model, it requires more performance data to develop the associated performance curves (at least 12 points from full-load performance and 7 points from part-load performance).

Note: Reformulated Chiller EIR components and their associated performance curve objects are developed using performance information for a specific chiller and should normally be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with caution. For example, if the user wishes to model a chiller size that is different from the reference capacity, it is highly recommended that the reference flow rates be scaled proportionately to the change in reference capacity.

General

Name

The auto-generated name of the chiller can be edited.

Chiller template

Use this browse option to select a chiller from the EnergyPlus chiller database whose performance data you wish to copy to your chiller.

Chiller type

The type of chiller can be one of these options:

- [1-Constant CoP](#) - a simple model with constant efficiency and no performance curves.
- [2-Chiller EIR](#) - a more detailed model using 3 performance curves to determine cooling capacity and CoP from the chiller load and outside air temperature as described on this page.
- [3-Reformulated EIR](#) - an empirical model similar to the EIR model described on this page.
- [4-Absorption](#) - the basic absorption chiller model
- [5-Indirect absorption](#) - a more advanced absorption chiller model with more flexible performance curves and ability to model the effect of varying evaporator, condenser, and generator temperatures.

Note: The Chiller type cannot be edited directly. To help you to ensure that appropriate performance curves are used, you must either load a new chiller template of the type required or add a new chiller, selecting the appropriate type from the drop list.

Reference capacity

This numeric field contains the reference cooling capacity of the chiller (in W or Btu/h). This should be the capacity of the chiller at the reference temperatures and water flow rates defined below. Alternatively, this field can be autosized.

Reference COP

This numeric field contains the chiller's coefficient of performance which is multiplied by the output of the chiller performance curves described below. This value should *not* include energy use due to pumps or cooling tower fans. This COP should be at the reference temperatures and water flow rates defined below.

Fraction of compressor electric power rejected by condenser

This numeric input represents the fraction of compressor electrical energy consumption that must be rejected by the condenser. Enter a value of 1.0 when modelling hermetic chillers. For open chillers, enter the compressor motor efficiency. This value must be greater than 0.0 and less than or equal to 1.0, with a default value of 1.0.

Chiller flow mode

This option determines how the chiller operates with respect to the intended fluid flow through the device's evaporator. There are three choices for specifying operating modes for the intended flow behaviour:

- **1-Constant flow** is the default option for constant volume chilled water loops and is useful for constant speed pumping arrangements where the chiller's request for flow is stricter and can increase the overall loop flow.
- **2-Leaving setpoint modulated** is the default option for variable volume chilled water loops and changes the chiller model to internally vary the flow rate so that the temperature leaving the chiller matches a setpoint at the evaporator outlet.
- **3-Not modulated** is useful for either variable or constant speed pumping arrangements where the chiller is passive in the sense that although it makes a nominal request for its design flow rate, it can operate at varying flow rates.

In all cases the operation of the external plant system can also impact the flow through the chiller - for example if the relative sizes and operation are such that flow is restricted and the requests cannot be met.

For *variable* flow chilled water loops these options are available: **2-Leaving setpoint modulated** and **3-Not modulated**.

For *constant* flow chilled water loops these options are available: **1-Constant flow** and **3-Not modulated**.

The type of loop (variable/constant flow) can be changed by modifying the [Plant loop flow type](#) on the [Chilled water plant loop dialog](#).

Note: When the **2-Leaving setpoint modulated** option is selected then you must add an extra Setpoint manager immediately downstream of the chiller chilled water outlet to define the temperature of the water supplied.

Sizing factor

The sizing factor is used when the chiller design inputs are autosized. In this case the autosizing results are multiplied by this additional sizing factor. The usual value to enter is 1.0.

The inputs that would be altered by the sizing factor are: **Reference capacity**, **Reference chilled water flow rate** and **Reference condenser water flow rate**.

The most common use of the sizing factor is for sizing chillers to meet only part of the design load while continuing to use the autosizing feature. For example when a set of chillers is chained together to supply chilled water to a plant loop, this sizing factor can be used to indicate the proportion of the load to be met by each chiller.

See also the section on [Autosizing HVAC Components](#)

Temperatures

Reference leaving chilled water temperature

This numeric field contains the chiller's reference leaving chilled water temperature (in °C or °F). The default value is 6.67°C.

Reference leaving condenser water temperature

This numeric field contains the chiller's reference leaving condenser water temperature (in °C or °F). The default value is 35°C.

Leaving chilled water lower temperature limit

This numeric field contains the lower limit for the leaving chilled water temperature (in °C or °F). This temperature acts as a cut off for heat transfer in the evaporator, so that the fluid doesn't get too cold. The default value is 2°C.

Flow Rates

Reference chilled water flow rate

For a variable flow chiller this is the maximum water flow rate and for a constant flow chiller this is the operating water flow rate through the chiller's evaporator. The units are (in m3/s or gal/min). This numeric input field must be greater than zero, or it can be autosized.

Reference condenser water flow rate

This numeric field contains the chiller's operating condenser water flow rate (in m3/s or gal/min). This numeric input field must be greater than zero, or it can be autosized.

Note: For water cooled chillers, EnergyPlus sets the condenser fluid flow rate to the maximum value whenever the chiller is operating. The [Chiller flow mode](#) only affects the chiller evaporator flow rate.

Performance Curves

Cooling capacity function of temperature curve (CAPFT)

The [Bi-quadratic performance curve](#) parameterizes the variation of the cooling capacity as a function of the leaving chilled water temperature and the leaving condenser fluid temperature. The output of this curve is multiplied by the reference capacity to give the cooling capacity at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified above. The bi-quadratic curve should be valid for the range of water temperatures anticipated for the simulation.

Electric input to cooling output ratio function of part load ratio curve type

Note: This option is not shown on the DesignBuilder dialog but may be in the future.

In EnergyPlus this option determines which type of the Electric Input to Cooling Output Ratio Function of Part Load Ratio Curve is used in the chiller modelling. Two curve types are available:

- **LeavingCondenserWaterTemperature** which is based on the leaving condenser water temperature. This is the only option used by DesignBuilder
- **Lift** is based on the normalized lift, which is the temperature difference between the leaving condenser water temperature and the leaving evaporator water temperature.

Electric input to cooling output ratio function of temperature curve

The [Bi-quadratic or Bi-cubic performance curve](#) is used to define the Electric Input to Cooling Output Ratio Function of Part Load Ratio. The form of this curve is based on the input for Electric Input to Cooling Output RatioFunction of Part Load Ratio Curve Type. The curve object type should be that parameterizes the variation of the energy input to cooling output ratio (EIR) as a function of the leaving chilled water temperature and the leaving condenser water temperature. The EIR is the inverse of the COP.

The output of this curve is multiplied by the reference EIR (inverse of the reference COP) to give the EIR at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified above. The curve should be valid for the range of water temperatures anticipated for the simulation (otherwise the program issues warning messages).

Electric input to cooling output ratio function of part load ratio curve (EIRfTPLR)

This [Bi-quadratic performance curve](#) parameterizes the variation of the energy input ratio (EIR) as a function of the leaving condenser water temperature and the part-load ratio (EIRfTPLR). The EIR is the inverse of the COP, and the part-load ratio is the actual cooling load divided by the chiller's available cooling capacity.

This curve is generated by dividing the operating electric input power by the available full-load capacity (do not divide by load) at the specific operating temperatures. The curve output should decrease from 1 towards 0 as part-load ratio decreases from 1 to 0. The output of this curve is multiplied by the reference full-load EIR (inverse of the reference COP) and the Energy Input to Cooling Output Ratio Function of Temperature Curve to give the EIR at the specific temperatures and part-load ratio at which the chiller is operating. This curve should have a value of 1.0 at the reference leaving condenser water temperature with the part-load ratio equal to 1.0.

An ideal chiller with the same efficiency at all part-load ratio's would use a performance curve that has a value of 0 when the part-load ratio equals 0 (i.e., a line connecting 0,0 and 1,1 when plotted as EIRfTPLR versus PLR), however, actual systems can have part-load EIR's slightly above or below this line (i.e., part-load efficiency often differs from rated efficiency). The bicubic curve should be valid for the range of condenser water temperatures and part-load ratios anticipated for the simulation (otherwise the program issues warning messages).

Note: Although a [bicubic curve](#) requires 10 coefficients, coefficients 7, 9 and 10 are typically not used in the performance curve described here and should be entered as 0 unless sufficient performance data and regression accuracy exist to justify the use of these coefficients. Additionally, coefficients 2, 3, and 6 should not be used unless sufficient temperature data is available to accurately define the performance curve (i.e., negative values may result from insufficient data).

Advice on chiller performance curves

During the simulation the electric power consumption of the compressor is calculated using:

$$Q_{\text{compressor}} = (Q_{\text{ref}} / \text{CoP}_{\text{ref}}) * \text{CAPFT} * \text{EIRFT} * \text{EIRPLR}$$

Where CAPFT, EIRFT and EIRPLR are the outputs from the 3 curves above.

If you have manufacturers data you can generate the inputs for these 3 curves using the [Chiller PerformanceCurve Coefficients.xls](#) spreadsheet provided by EnergyPlus. When using this spreadsheet to generate inputs for EIRPLR note that the coefficients are read from the spreadsheet into the DesignBuilder Quadratic Curve dialog as follows:

Load %	CAP(kW)	UNIT RATE(kW)	COP	EIR	PLR	PLR*PLR	EIR(Normalized)	EIR_p(Normalized)	EIR_p
100	78.4	30.1	2.60465116	0.38392857	1	1	1	1.00014178	0.383983
75	58.9	16.5	3.56969697	0.28013582	0.75	0.5625	0.7296561	0.72923076	0.27997253
50	39.3	9	4.36666667	0.22900763	0.5	0.25	0.596485	0.59691033	0.22917093
25	19.6	4.54	4.31718062	0.23163265	0.25	0.0625	0.60332226	0.60318048	0.23157822
20	NO DATA	NO DATA	NO DATA	NO DATA	0.2	0.04	NO DATA	0.62106538	0.23844474
0	NO DATA	NO DATA	NO DATA	NO DATA	0	0	NO DATA	0	0

Edit Curve - New Chiller EIR Function of PLR Curve

Curves

General Curve Plot

General

Name New Chiller EIR Function of PLR Curve

Note

Source User

Category Quadratic

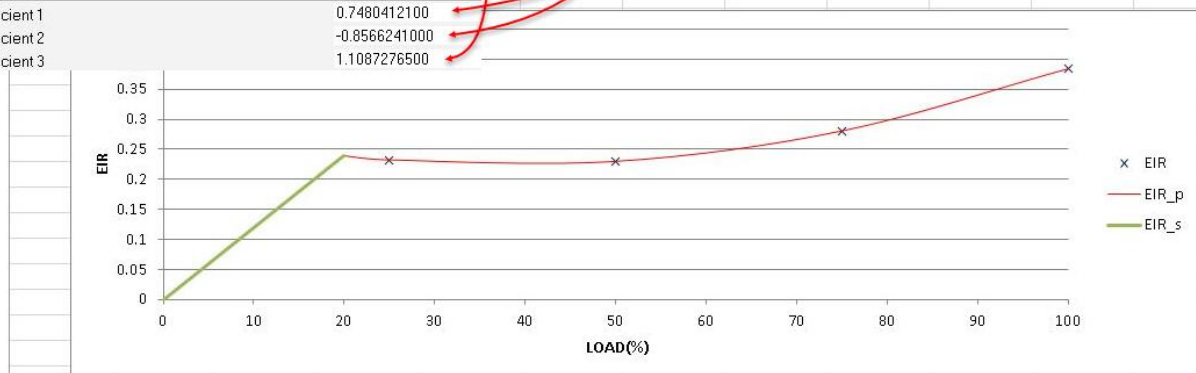
Coefficients

Coefficient 1 0.7480412100

Coefficient 2 -0.8566241000

Coefficient 3 1.1087276500

Coefficients for the Energy Input Ratio Function of Part Load Ratio (EIRFPLR)		
c	b	a
1.10872465	-0.8566241	0.74804121
0.0050724	0.00644115	0.00176512
0.99999623	0.00063405	#N/A
132768.381	1	#N/A
0.10675063	4.0202E-07	#N/A



IF THE CHILLER MODEL HAS 0.2 AS MINIMUM PART LOAD RATIO, THE SIMULATION WILL RIDE THE "EIR_s" CURVE UNTIL PART LOAD RATIO IS 0.2 AND THEN IT WILL RIDE THE "EIR_p" CURVE

Of course the performance and capacity of a chiller depends on various factors, the most important of these being part load and temperature chilled water and the Chiller EIR provides an accurate representation taking these factors into account. However in some cases you may only have summary manufacturers data. For example you may only have information on the variation of CoP with part load ratio. In this case you can define the EIRPLR curve in the usual way using your summary manufacturer's data and use constant values for the CAPFT and EIRFT curves which both generate a constant value of 1.0. Because CAPFT and EIRFT are both Bi-quadratic curves you cannot use a constant Linear curve here so you must choose inputs to a special simple Bi-quadratic curve that generates an output of 1 regardless of inputs. To achieve this simply create and select a new Bi-quadratic curve with Coefficient 1 = 1 and all other coefficients = 0 for CAPFT and EIRFT.

Part Load Settings

Minimum part load ratio

This numeric field contains the chiller's minimum part-load ratio. The expected range is between 0 and 1. Below this part-load ratio, the compressor cycles on and off to meet the cooling load. The **Minimum part load ratio** must be less than or equal to the **Maximum part load ratio**. The default value is 0.1.

Note: Setting the Minimum part load ratio to zero for air cooled chillers, will cause the condenser fan to run constantly even at zero load which is likely to lead to high chiller fan consumption. This happens because the condenser fan cannot cycle off even at zero chiller load.

Maximum part load ratio

This numeric field contains the chiller's maximum part-load ratio. This value may exceed 1, but the normal range is between 0 and 1.0. The **Maximum part load ratio** must be greater than or equal to the **Minimum part load ratio**. The default value is 1.0.

Optimum part load ratio

This numeric field contains the chiller's optimum part-load ratio. This is the part-load ratio at which the chiller performs at its maximum COP. The **optimum part-load ratio** must be greater than or equal to the **Minimum part load ratio**, and less than or equal to the **Maximum part load ratio**. The default value is 1.0.

Minimum unloading ratio

This numeric field contains the chiller's minimum unloading ratio. The expected range is between 0 and 1. The minimum unloading ratio is where the chiller capacity can no longer be reduced by unloading and must be false loaded to meet smaller cooling loads. A typical false loading strategy is hot-gas bypass. The **minimum unloading ratio** must be greater than or equal to the **Minimum part load ratio**, and less than or equal to the **Maximum part load ratio**. The default value is 0.2.

False loading

Different chiller types (reciprocating, centrifugal etc) have very different operating characteristics. The point at which they switch off at low load (cycle on and off to meet small loads) will vary between about 0.25 for older reciprocating chillers and 0.1 for modern centrifugal chillers which have much better part-load performance. Sometimes a manufacturer or installer may need to introduce a mechanism to stop the chiller cycling on and off too frequently (frequent cycling causes excessive wear and reduces the equipment life). To stop this happening some of the hot gas from the compressor outlet can be re-injected back into the return side of the compressor so that the compressor inlet senses a higher return gas temperature and is 'fooled' into thinking that there is a higher cooling load on the system than there actually is (this basically mimics a higher evaporator leaving temperature) so it needs to work a little harder to meet the load and the load is therefore maintained above the point at which the compressor would normally switch off. This strategy is called false-loading and it is clear that bypassing hot gas from the discharge to the suction side of the compressor results in higher energy consumption so is generally not considered to be an acceptable control strategy and actually contravenes some energy codes (e.g. Part L2 in UK). This strategy is sometimes used to overcome poor design and control problems.

You can avoid false loading in your model by setting the **Minimum unloading ratio** equal to the **Minimum part load ratio**, so when the chiller reaches its minimum part-load ratio it just switches off and false-loading cannot occur.

Heat Recovery

Heat recovery

If heat is to be recovered from the water-cooled condenser check this checkbox and ensure that a heat recovery loop is specified and attached to the chiller.

Note: Use the **Add heat recovery** loop command at HVAC system level to create a DHW loop that can be connected to a water-cooled chiller with heat recovery. This allows the waste heat from the chiller to be used to heat up a water tank which can be used to provide hot water for DHW, zone heaters or any other heating applications.

Design heat recovery water flow rate

Enter the design heat recovery water flow rate (in m³/s or gal/min). This field is autosizable. When autosizing, the flow rate is simply the product of the design condenser flow rate and the condenser heat recovery relative capacity fraction set in the field below.

Condenser heat recovery relative capacity fraction

This field is used to describe the physical size of the heat recovery portion of a split bundle condenser section. This fraction describes the relative capacity of the heat recovery bundle of a split condenser compared to the nominal, full load heat rejection rate of the chiller. This fraction will be applied to the full heat rejection when operating at nominal capacity and nominal COP to model a capacity limit for the heat rejection. The default capacity fraction is 1.

Inlet high temperature limit

Check this checkbox if you would like to define an upper limit for the return fluid temperatures entering the chiller at the heat recovery inlet node (see below)

Heat recovery inlet high temperature limit schedule

If the Inlet high temperature limit checkbox is checked then you must select the schedule to be used to control heat recovery operation of the chiller. The schedule should contain temperature values, in °C, that describe an upper limit for the return fluid temperatures entering the chiller at the heat recovery inlet node. If the fluid temperature is too high, then the heat recovery will not operate. This is useful to restrict the chiller lift from becoming too high and to avoid overheating the hot water loop.

Chiller - Absorption



Chiller:Absorption

Used in:

- Chilled water loop, supply side

The Absorption Chiller is nearly identical to the more sophisticated [Absorption indirect chiller](#) model. Since these absorption chiller models are nearly identical (i.e., the performance curves of the enhanced model can be manipulated to produce similar results to the Absorption Chiller model), it is quite probable that the Absorption Chiller model will be deprecated in a future release of EnergyPlus.

Note: Absorption Chillers and their associated performance curve objects are developed using performance information for a specific chiller and should normally be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with caution.

General

Name

The auto-generated name of the chiller can be edited.

Chiller template

Use this browse option to select a chiller from the EnergyPlus chiller database whose performance data you wish to copy to your chiller.

Chiller type

The type of chiller can be one of these options:

- [1-Constant CoP](#) - a simple model with constant efficiency and no performance curves.
- [2-Chiller EIR](#) - a more detailed model using 3 performance curves to determine cooling capacity and CoP from the chiller load and outside air temperature.
- [3-Reformulated EIR](#) - an empirical model similar to the EIR model
- [4-Absorption](#) - the basic absorption chiller model as described on this page.
- [5-Indirect absorption](#) - a more advanced absorption chiller model with more flexible performance curves and ability to model the effect of varying evaporator, condenser, and generator temperatures.

Note: The Chiller type cannot be edited directly. To help you to ensure that appropriate performance curves are used, you must either load a new chiller template of the type required or add a new chiller, selecting the appropriate type from the drop list.

Nominal capacity

This numeric field contains the nominal cooling capability of the chiller (in W or Btu/h).

Nominal pumping power

This numeric field contains the nominal pumping power of the absorber (in W or Btu/h).

Chiller flow mode

This choice field determines how the chiller operates with respect to the intended fluid flow through the device's evaporator. There are three choices for specifying operating modes for the intended flow behaviour:

- **1-Constant flow** is the default option for constant volume chilled water loops and is useful for constant speed pumping arrangements where the chiller's request for flow is stricter and can increase the overall loop flow.
- **2-Leaving setpoint modulated** is the default option for variable volume chilled water loops and changes the chiller model to internally vary the flow rate so that the temperature leaving the chiller matches a setpoint at the evaporator outlet. For the absorption chiller, this mode also affects the flow rate on the generator connection.
- **3-Not modulated** is useful for either variable or constant speed pumping arrangements where the chiller is passive in the sense that although it makes a nominal request for its design flow rate, it can operate at varying flow rates. This flow mode does not impact the condenser loop connection.

In all cases the operation of the external plant system can also impact the flow through the chiller - for example if the relative sizes and operation are such that flow is restricted and the requests cannot be met.

For *variable* flow chilled water loops these options are available: **2-Leaving setpoint modulated** and **3-Not modulated**.

For *constant* flow chilled water loops these options are available: **1-Constant flow** and **3-Not modulated**.

The type of loop (variable/constant flow) can be changed by modifying the [Plant loop flow type](#) on the [Chilled water plant loop dialog](#).

Note: When the **2-Leaving setpoint modulated** option is selected then you must add an extra Setpoint manager immediately downstream of the chiller chilled water outlet to define the temperature of the water supplied.

Sizing factor

This optional numeric field allows the user to specify a sizing factor for this component. The sizing factor is used when the component design inputs are autosized: the autosizing calculations are performed as usual and the results are multiplied by the sizing factor. For this component the inputs that would be altered by the sizing factor are: Nominal Capacity, Nominal Pumping Power, Design Chilled Water Flow Rate, Design Condenser Water Flow Rate and Design Generator Fluid Flow Rate. Sizing factor allows the user to size a component to meet part of the design load while continuing to use the autosizing feature.

Condenser

Condenser type

The condenser type determines what type of condenser will be included with this chiller. Valid condenser types are:

- **1-Air cooled**,
- **2-Water cooled**,
- **3-Evaporatively cooled**

The default is **2-Water cooled** which requires the full specification of the Condenser loop and its associated equipment. **1-Air cooled** and **3-Evaporatively cooled** do not require a Condenser loop to be specified.

Note: **Condenser type** cannot be edited directly. To help you to ensure that appropriate performance curves are used, load a chiller of the correct type from the [Chiller template](#).

Temperatures

Reference entering condenser fluid temperature

This numeric field contains the lower limit for the evaporator outlet temperature (in °C or °F). This temperature acts as a cut off for heat transfer in the evaporator, so that the fluid doesn't get too cold.

Leaving chilled water temperature limit

This numeric field contains the absorption chiller's condenser inlet design temperature (in °C or °F).

Flow Rates

Design chilled water flow rate

For variable volume chiller this is the maximum flow and for constant flow chiller this is the design flow rate. The units are m3/s or gal/min.

Design condenser water flow rate

This numeric field contains the absorption chiller's design condenser water flow rate (in m3/s or gal/min).

Note: For water cooled chillers, EnergyPlus sets the condenser fluid flow rate to the maximum value whenever the chiller is operating. The [Chiller flow mode](#) only affects the chiller evaporator flow rate.

Part Load Settings

Minimum part load ratio

This numeric field contains the absorption chiller's minimum part load ratio. The expected range is between 0 and 1. The minimum part load is not the load where the machine shuts off, but where the amount of power remains constant to produce smaller loads than this fraction.

Maximum part load ratio

This numeric field contains the absorption chiller's maximum part load ratio. This value may exceed 1, but the normal range is between 0 and 1.1.

Optimum part load ratio

This numeric field contains the absorption chiller's optimum part load ratio. This is the part load ratio at which the chiller performs at its maximum COP.

Generator Heat Input Part Load Ratio Curve

The Generator Heat Input Part Load Ratio Curve is a quadratic equation that determines the Ratio of the generator load on the absorber to the demand on the chiller. The defining equation is:

$$\text{GeneratorInputRatio} = C1 * \text{PLR} + C2 + C3 * \text{PLR}$$

The following three fields contain the coefficients for the equation.

Coefficient 1 of the generator part load ratio curve

C1 in the Generator Heat Input Part Load Ratio Curve. This value is obtained by fitting manufacturers' performance data to the curve.

Coefficient 2 of the generator part load ratio curve

C2 in the Generator Heat Input Part Load Ratio Curve. This value is obtained by fitting manufacturers' performance data to the curve.

Coefficient 3 of the generator part load ratio curve

C3 in the Generator Heat Input Part Load Ratio Curve. This value is obtained by fitting manufacturers' performance data to the curve.

Pump Electric Use Part Load Ratio Curve

The Pump Electric Use Part Load Ratio Curve is a quadratic equation that determines the Ratio of the actual absorber pumping power to the nominal pumping power. The defining equation is:

$$\text{ElectricInputRatio} = C1 + C2 * \text{PLR} + C3 * \text{PLR}^2$$

The following three fields contain the coefficients for the equation.

Coefficient 1 of the pump electric use part load ratio curve

C1 in the Pump Electric Use Part Load Ratio Curve. This value is obtained by fitting manufacturers' performance data to the curve.

Coefficient 2 of the pump electric use part load ratio curve

C2 in the Pump Electric Use Part Load Ratio Curve. This value is obtained by fitting manufacturers' performance data to the curve.

Coefficient 3 of the pump electric use part load ratio curve

C3 in the Pump Electric Use Part Load Ratio Curve. This value is obtained by fitting manufacturers' performance data to the curve.

Generator Hot Water Supply

Hot water plant connection

Check this checkbox if there is a hot water plant connection.

Design generator hot water flow rate

This numeric field contains the absorption chiller's design condenser fluid flow rate (m3/s or gal/min).

Chiller - Absorption Indirect



Chiller:Absorption:Indirect

Used in:

- Chilled water loop, supply side

The Absorption Indirect Chiller is an enhanced version of the absorption chiller model found in the Building Loads and System Thermodynamics (BLAST) program. This enhanced model is nearly identical to the [Absorption Chiller](#) model with the exceptions that:

1. The enhanced indirect absorption chiller model provides more flexible performance curves, and,
2. Chiller performance includes the impact of varying evaporator, condenser, and generator temperatures.

Since these absorption chiller models are nearly identical (i.e., the performance curves of the enhanced model can be manipulated to produce similar results to the Absorption Chiller model), it is quite probable that the Absorption Chiller model will be deprecated in a future release of EnergyPlus.

Note: Absorption Indirect Chillers and their associated performance curve objects are developed using performance information for a specific chiller and should normally be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with caution.

General

Name

The auto-generated name of the chiller can be edited.

Chiller template

Use this browse option to select a chiller from the EnergyPlus chiller database whose performance data you wish to copy to your chiller.

Chiller type

The type of chiller can be one of these options:

- [1-Constant CoP](#) - a simple model with constant efficiency and no performance curves.
- [2-Chiller EIR](#) - a more detailed model using 3 performance curves to determine cooling capacity and CoP from the chiller load and outside air temperature as described on this page.
- [3-Reformulated EIR](#) - an empirical model similar to the Chiller EIR model.
- [4-Absorption](#) - the basic absorption chiller model.
- [5-Indirect absorption](#) - a more advanced absorption chiller model with more flexible performance curves and ability to model the effect of varying evaporator, condenser, and generator temperatures, as described on this page.

Note: The Chiller type cannot be edited directly. To help you to ensure that appropriate performance curves are used, you must either load a new chiller template of the type required or add a new chiller, selecting the appropriate type from the drop list.

Nominal capacity

This numeric field contains the nominal cooling capability of the chiller (in W or Btu/h).

Nominal pumping power

This numeric field contains the nominal pumping power of the absorber (in W or Btu/h).

Chiller flow mode

This choice field determines how the chiller operates with respect to the intended fluid flow through the device's evaporator. There are three choices for specifying operating modes for the intended flow behaviour:

- **1-Constant flow** is the default option for constant volume chilled water loops and is useful for constant speed pumping arrangements where the chiller's request for flow is stricter and can increase the overall loop flow.
- **2-Leaving setpoint modulated** is the default option for variable volume chilled water loops and changes the chiller model to internally vary the flow rate so that the temperature leaving the chiller matches a setpoint at the evaporator outlet. For the absorption chiller, this mode also affects the flow rate on the generator connection.
- **3-Not modulated** is useful for either variable or constant speed pumping arrangements where the chiller is passive in the sense that although it makes a nominal request for its design flow rate, it can operate at varying flow rates. This flow mode does not impact the condenser loop connection.

In all cases the operation of the external plant system can also impact the flow through the chiller - for example if the relative sizes and operation are such that flow is restricted and the requests cannot be met.

For *variable* flow chilled water loops these options are available: **2-Leaving setpoint modulated** and **3-Not modulated**.

For *constant* flow chilled water loops these options are available: **1-Constant flow** and **3-Not modulated**.

The type of loop (variable/constant flow) can be changed by modifying the [Plant loop flow type](#) on the [Chilled water plant loop dialog](#).

Note: When the **2-Leaving setpoint modulated** option is selected then you must add an extra Setpoint manager immediately downstream of the chiller chilled water outlet to define the temperature of the water supplied.

Sizing factor

This optional numeric field allows the user to specify a sizing factor for this component. The sizing factor is used when the component design inputs are autosized: the autosizing calculations are performed as usual and the results are multiplied by the sizing factor. For this component the inputs that would be altered by the sizing factor are: Nominal Capacity, Nominal Pumping Power, Design Chilled Water Flow Rate, Design Condenser Water Flow Rate and Design Generator Fluid Flow Rate. Sizing factor allows the user to size a component to meet part of the design load while continuing to use the autosizing feature.

Condenser

Condenser type

The condenser type determines what type of condenser will be included with this chiller. Valid condenser types are:

- **1-Air cooled,**
- **2-Water cooled,**
- **3-Evaporatively cooled**

The default is **2-Water cooled** which requires the full specification of the Condenser loop and its associated equipment. **1-Air cooled** and **3-Evaporatively cooled** do not require a Condenser loop to be specified.

Note: **Condenser type** cannot be edited directly. To help you to ensure that appropriate performance curves are used, load a chiller of the correct type from the [Chiller template](#).

Temperatures

Reference entering condenser fluid temperature

This numeric field contains the chiller's condenser inlet design temperature (in °C or °F). The default value for this field is 30° C and is only used when the [Design chilled water flow rate](#) is autosized.

Entering condenser water temperature limit

This numeric field contains the chiller's lower limit for the condenser entering water temperature (in °C or °F). The default value for this field is 15° C. If this limit is exceeded, a warning message will report the incident. No correction to chiller capacity is made for low condenser entering water temperatures.

Leaving chilled water temperature limit

This numeric field contains the chiller's lower limit for the evaporator leaving water temperature (in °C or °F). The default value for this field is 5° C. If this limit is exceeded, a warning message will report the incident. No correction to chiller capacity is made for low evaporator leaving water temperatures.

Temperature lower limit generator inlet

This numeric field specifies the lower limit of the generator's entering water temperature (in °C or °F).

Flow Rates

Design chilled water flow rate

For variable volume chiller this is the maximum flow and for constant flow chiller this is the design flow rate. The units are m3/s or gal/min.

Design condenser water flow rate

This numeric field contains the absorption chiller's design condenser water flow rate (in m3/s or gal/min).

Note: For water cooled chillers, EnergyPlus sets the condenser fluid flow rate to the maximum value whenever the chiller is operating. The [Chiller flow mode](#) only affects the chiller evaporator flow rate.

Part Load Settings

Minimum part load ratio

This numeric field contains the absorption chiller's minimum part load ratio. The expected range is between 0 and 1. The minimum part load is not the load where the machine shuts off, but where the amount of power remains constant to produce smaller loads than this fraction.

Maximum part load ratio

This numeric field contains the absorption chiller's maximum part load ratio. This value may exceed 1, but the normal range is between 0 and 1.1.

Optimum part load ratio

This numeric field contains the absorption chiller's optimum part load ratio. This is the part load ratio at which the chiller performs at its maximum COP.

Part-Load Settings Curves

Generator heat input function of part load ratio curve

Select the [quadratic](#) or [cubic](#) curve which characterizes the heat input as a function of chiller part-load ratio. The curve output is multiplied by the chiller's nominal capacity and operating part-load ratio or minimum part-load ratio, whichever is greater, to determine the amount of heat input required for the given operating conditions.

Pump electric input function of part load ratio curve

Select the [quadratic](#) or [cubic](#) curve which characterizes the pump electrical power as a function of chiller part-load ratio. The curve output is multiplied by the chiller's nominal pumping power and operating part-load ratio or minimum part-load ratio, whichever is greater, to determine the amount of pumping power required for the given operating conditions.

Correction Curves

Condenser temperature capacity correction

Check this option if you would to model the chiller's evaporator capacity as a function of condenser entering water temperature.

Capacity correction function of condenser temperature curve

Select the [quadratic](#) or [cubic](#) curve which correlates the chiller's evaporator capacity as a function of condenser entering water temperature. This curve is used to correct nominal capacity at off-design condensing temperatures.

Chilled water temperature capacity correction

Check this option if you would like to correlate the chiller's evaporator capacity as a function of evaporator leaving water temperature.

Capacity correction function of chilled water temperature curve

Select the [quadratic](#) or [cubic](#) curve which correlates the chiller's evaporator capacity as a function of evaporator leaving water temperature. This curve is used to correct nominal capacity at off-design evaporator temperatures.

Condenser temperature heat input correction

Check this option if you would like to correlate the chiller's heat input as a function of condenser entering water temperature.

Generator heat input correction function of condenser temperature

Select the [quadratic](#) or [cubic](#) curve which correlates the chiller's heat input as a function of condenser entering water temperature. This curve is used to correct generator heat input at off-design condensing temperatures.

Chilled water temperature generator heat input correction

Check this option if you would like to correlate the chiller's heat input as a function of evaporator leaving water temperature.

Generator heat input correction function of chilled water temperature

Select the [quadratic](#) or [cubic](#) curve which correlates the chiller's heat input as a function of evaporator leaving water temperature. This curve is used to correct generator heat input at off-design evaporator temperatures.

Generator Hot Water Supply

Hot water plant connection

Check this checkbox if there is a hot water plant connection.

Design generator hot water flow rate

This numeric field contains the absorption chiller's design condenser fluid flow rate (m³/s or gal/min).

Generator temperature capacity correction

Checking this option allows you to correlate the chiller's evaporator capacity as a function of generator entering water temperature.

Capacity correction function of generator temperature curve

Select the [quadratic](#) or [cubic](#) curve which correlates the chiller's evaporator capacity as a function of generator entering water temperature. This curve is used to correct nominal capacity at off-design evaporator temperatures.

District Cooling



Used in:

- Chilled water loop, supply side

The district cooling component is used to provide cooling to the supply side of a chilled water loop. It can be used to model these systems:

- a. Centralised sources of cool water (district cooling systems), or,
- b. Cases where it is not necessary to simulate the plant., e.g. where the focus of the study is on the air side of the system or the building zones.

General

Name

The auto-generated name of the district cooling system can be edited.

Nominal capacity

This numeric field contains the nominal demand (in W or Btu/h) that the district cooling will meet. This field is autosizable.

Capacity fraction schedule

Select the schedule that describes how the nominal capacity varies over time. Values must non-negative. The capacity at a given point in time is determined by the product of the **Nominal capacity** (above) and the value in this schedule. The default schedule is **On 24/7** i.e. a value of 1 all the time.

Cooling Towers



CoolingTower:SingleSpeed
CoolingTower:TwoSpeed
CoolingTower:VariableSpeed:Merkel
CoolingTower:VariableSpeed

Used in:

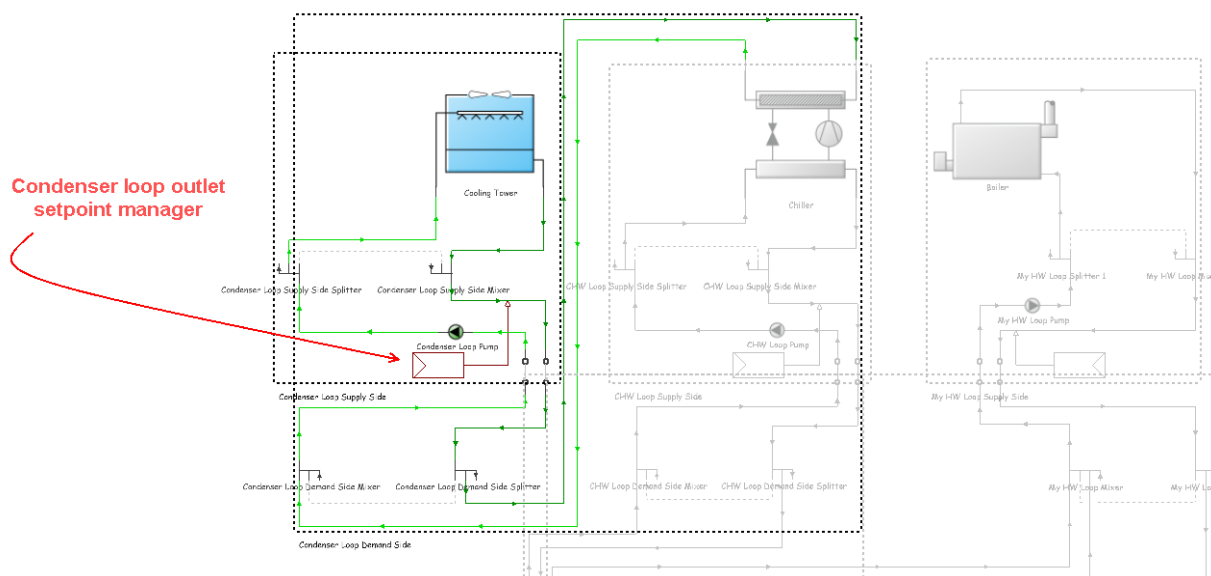
- **Condenser loop, supply side**

Cooling towers are placed on the supply side of condenser loops. The cooling tower is modelled as a counter-flow heat exchanger with options for single-speed, two-speed or variable-speed fan. Specifically, 4 types of cooling tower are available:

- [Single Speed](#)
- [Two Speed](#)
- [Variable Speed \(Merkel\)](#)
- [Variable Speed](#)

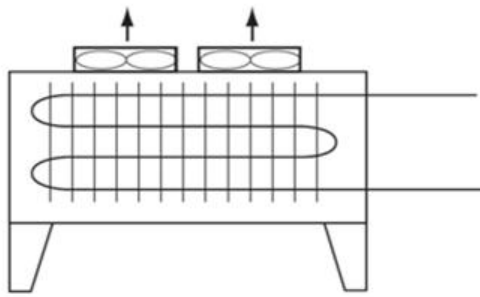
Cooling towers here are “wet” and consume water through evaporation, drift, and blow-down. The model can be used to predict water consumed by the towers.

The cooling tower seeks to maintain the temperature of the water exiting the cooling tower at (or below) a set point. The set point schedule value is defined by the condenser loop outlet setpoint manager:

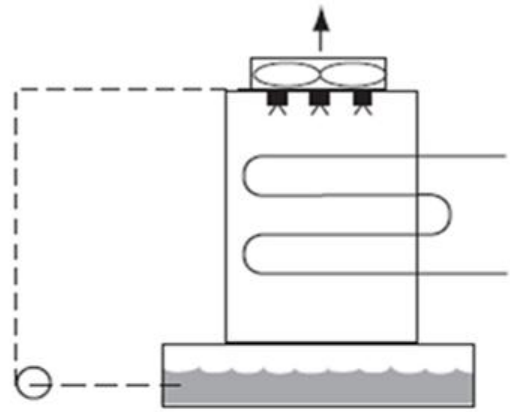


See the [Condenser loops cooling tower](#) tutorial

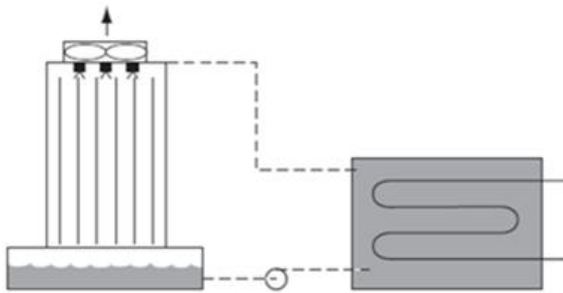
The schematic diagrams below (reproduced with permission from CIBSE) illustrate the various types of Direct & Indirect Heat Rejection used in cooling towers. The condenser process takes the fluid from saturated vapour (point 5) to saturated liquid (point 6) on the graph below.



Direct: Air-cooled

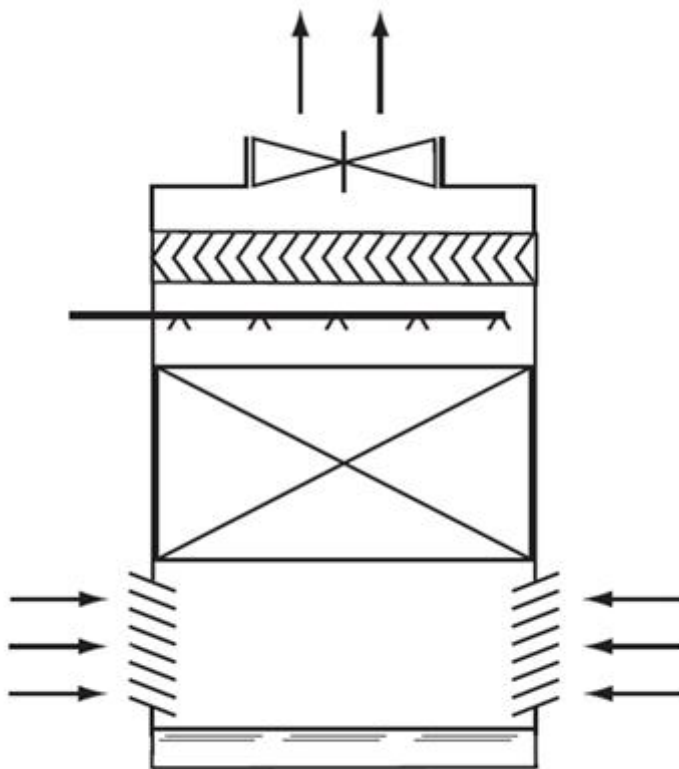


Direct: Evaporatively-cooled



Indirect: Water-cooled

Direct & Indirect Heat Rejection (CIBSE Guide B Table 4.10)



Counterflow

Induced Draft Cooling Tower (CIBSE Guide B Table 4.22)

Cooling Tower - Single Speed And Two Speed



CoolingTower:SingleSpeed
CoolingTower:TwoSpeed

Used in:

- Condenser loop, supply side

Cooling towers are placed on the supply side of condenser loops. The cooling tower is modelled as a counter-flow heat exchanger with a single-speed or two-speed fan (induced draft configuration) based on Merkel's theory. Tower performance is defined via one of two methods: a) design heat transfer coefficient-area product (UA) and design water flow rate, or b) nominal tower capacity at a specific rating point. Regardless of which method is chosen, the design airflow rate and corresponding fan power must be specified (at both high and low speeds when using the two-speed model). The model will also account for tower performance in the "free convection" regime, when the tower fan is off but the water pump remains on and heat transfer still occurs (albeit at a low level). To account for "free convection", you must specify the corresponding airflow rate and heat transfer coefficient-area product (UA), or the nominal tower capacity during this mode of operation.

See the [Condenser loops cooling tower](#) tutorial

The model first checks to determine the impact of "free convection", if specified by the user, on the tower exiting water temperature.

Model Description - Single Speed Cooling Tower

If the exiting water temperature based on "free convection" is at or below the set point, then the tower fan is not turned on. If the exiting water temperature based on "free convection" is below the set point, the tower will operate in [Fluid bypass](#) mode whereby a portion of the water goes through the tower media and gets cooled while the remaining water flow gets bypassed, two water flows then mix together trying to meet the water setpoint temperature. If the exiting water temperature remains above the set point after "free convection" is modelled, then the tower fan is turned on to reduce the exiting water temperature to the set point. If the [capacity control](#) is **1-Fan cycling**, the model assumes that part-load operation is represented by a simple linear interpolation between two steady-state regimes (i.e. the tower fan is on for the entire simulation time-step and the tower fan is off for the entire simulation time-step). Cyclic losses are not taken into account. If the [capacity control](#) is **2-Fluid bypass**, the model determines the fraction of water flow to be bypassed while the remaining water goes through the tower cooling media and gets cooled, then the two water flows mix to meet the setpoint temperature. In this case, the fan runs at full speed for the entire time-step.

Model Description - Two Speed Cooling Tower

If the exiting water temperature based on "free convection" is at or below the set point, then the tower fan is not turned on. If the exiting water temperature remains above the set point after "free convection" is modelled, then the tower fan is turned on at low speed to reduce the exiting water temperature. If operating the tower fan at low speed does not reduce the exiting water temperature to the set point, then the tower fan is increased to its high speed. The model assumes that part-load operation is represented by a simple linear interpolation between two steady-state regimes (i.e., tower fan at high speed for the entire simulation timestep and tower fan at low speed for the entire simulation timestep, or tower fan at low speed for the entire simulation timestep and tower fan off for the entire simulation timestep). Cyclic losses are not taken into account.

Cooling towers here are "wet" and consume water through evaporation, drift, and blow-down. The model can be used to predict water consumed by the towers.

For the operation of multi-cell towers, the first step is to determine the number of cells to operate based on the cell control method, between the minimum number of cells subject to the maximum water flow rate fraction per cell and maximum number of cells subject to the minimum water flow rate fraction per cell. If the calculated cells do not meet the loads, additional cells will be operating to help meet the loads. Inside each cell, the existing capacity controls still apply.

For multi-cell towers, the following inputs are assumed to be for the entire tower including all cells:

- [Design water flow rate](#); [Design air flow rate](#); [Fan power at design air flow rate](#);
- [Air flow rate in free convection regime](#); [Nominal capacity](#); [Free convection capacity](#)
- [Basin heater](#) a common basin is assumed

General

Name

This is the name that you assign to the cooling tower which should be unique. If the supplied name is not unique, the software will automatically append a backslash and integer to ensure that there are no duplicate names.

Cooling tower type

You can select from 4 different types of cooling tower:

- **1-Single speed** (described on this help page)
- **2-Two speed** (described on this help page)
- **3-Variable speed (Merkel)**
- **4-Variable speed**

Evaporation loss mode

This field is used to choose which method is used to model the amount of water evaporated by the cooling tower. There are two options:

- **1-Saturated exit** or
- **2-Loss factor**.

The default is **1-Saturated exit**. The user-defined loss factor is entered under **Evaporation loss factor**, below. By assuming that the air leaving the tower is saturated, the evaporation can be directly calculated using moist air engineering calculations with data available within the cooling tower model (and does not require additional user input).

Evaporation loss factor

This field is used to specify the rate of water evaporated from the cooling tower and lost to the outside air (in %/K or %/F). It is only available if the **Evaporation loss mode** (above) is set to **2-Loss factor**. The evaporation loss is then calculated as a fraction of the circulating condenser water flow and varies with the temperature change in the condenser water. The evaporation rate will equal this value times each degree Kelvin of temperature drop in the condenser water. Typical values are from 0.15 to 0.27 (%/K). The default is 0.2.

Drift loss percent

This field is used to specify the rate of water lost to the exiting air as entrained droplets (%). The drift loss is a percent of the condenser water flow. Typical values for towers with efficient drift eliminators are between 0.002 and 0.2% of the condenser water flow rate. The default value is 0.008%.

Sizing factor

This setting allows the user to specify a sizing factor for this component. The sizing factor is used when the component design inputs are auto-sized. In this case the auto-sizing calculations are performed as usual and the results are multiplied by the sizing factor. Sizing factor allows the user to size a component to meet part of the design load while continuing to use the auto-sizing feature. For this component the inputs that would be altered by the sizing factor are:

1. [Design water flow rate](#).
2. [Design air flow rate](#).
3. [Fan power at design air flow rate](#).
4. [UA at design air flow rate](#).

5. [Air flow rate in free convection regime](#).
6. [UA at free convection air flow rate](#). Note that the U-Factor Times Area Value at Design Air Flow Rate is not multiplied by the Sizing Factor. Instead the design tower load is multiplied by the sizing factor and the design UA then calculated as usual. The U-Factor Times Area Value at Free Convection Air Flow Rate is set to 10% of the new design Tower UA.

See also the section on [Autosizing HVAC Components](#)

Airflow

Design air flow rate

This numeric field contains the design air flow rate induced by the tower fan (in m³/s or ft³/min). A value of greater than zero must be defined regardless of the tower performance input method. Alternately, this field can be autosized. If autosized, the design air flow rate is calculated as follows:

$$\text{Design Air Flow Rate} = \frac{0.5 \rho_{\text{air}} (\text{Fan Power at Design Air Flow Rate})}{190}$$

In which a fan pressure rise of 190 Pa and total fan efficiency of 0.5 are assumed.

Note: For the Two Speed Cooling tower there are 2 Design air flow rates to enter, one for each of the fan speeds.

Design fan power

This is the fan power (W) at the design air flow rate specified under [Design air flow rate](#). A value of greater than zero must be specified regardless of the tower performance input method, or alternatively, this field can be autosized. If autosized, the fan power is calculated as follows:

If [Performance input method](#) is specified as **1-Nominal capacity**, then the fan power is obtained from:

$$\text{Fan Power} = 0.0105 \cdot \text{Tower Nominal Capacity}$$

If [Performance input method](#) is specified as **2-UA and Design water flow rate**, then the fan power is obtained from:

$$\text{Fan Power} = 0.0105 \cdot \rho_{\text{water}} \cdot C_{p,\text{water}} \cdot \text{Design Water Flow Rate} \cdot \text{Design Loop } \Delta T$$

Note: For the Two Speed Cooling tower there are 2 Fan power at design air flow rate settings to enter, one for each of the fan speeds.

Performance Input Method

Performance input method

This setting allows the method by which the user will specify tower performance to be selected. There are two options:

- **1-Nominal Capacity** where the Nominal capacity and Free convection capacity must be entered as described below
- **2-UA and design water flow rate** where you must enter design UA values, design water flow rates and air flow rates

Water Flow

Design water flow rate

This is the design water flow rate through the tower (in m³/s or gal/min). This value is the flow rate of the condenser loop water being cooled by the tower (not the flow rate of water being sprayed on the outside of the heat exchange coil). If the [Performance input method](#) is specified as **2-UA and design water flow rate**, then a water flow rate greater than zero must be defined or the item can be autosized. If autosized, the design water flow rate is derived from the design load to be rejected by the condenser loop and the design loop temperature difference. If [Performance input method](#) is specified as **1-Nominal capacity**, then the model automatically assumes a water flow rate of 5.382E-8 m³/s per W of tower capacity.

U-Factor Times Area Value

UA at design air flow rate

This is the heat transfer coefficient-area product (UA) (in W/K or Btu/h-F) corresponding to the design air and water flow rates specified above. If the Performance Input Method is specified as **2-UA and design water flow rate**, then a UA value greater than zero but less than or equal to 300,000 W/K must be defined, or the setting can be autosized. If autosized, the design tower UA value is derived from the design load to be rejected by the condenser loop and the design loop delta T, assuming a tower water inlet temperature of 35°C and tower inlet air at 35°C dry-bulb/25.6°C wet-bulb.

Note: For the Two Speed Cooling tower there are 2 UA at design air flow rate settings to enter, one for each of the fan speeds.

Capacity

Nominal capacity

Used when the [Performance input method](#) is specified as **1-Nominal capacity**, this is the “nominal” heat rejection capacity of the cooling tower (in W or Btu/h), with entering water at 35°C (95°F), leaving water at 29.4°C (85°F), entering air at 25.6°C (78°F) wet-bulb and 35°C (95°F) dry-bulb temperatures. The design water flow rate is assumed to be 5.382E-8 m³/s per watt of nominal capacity (3 gpm/ton). 125% of this nominal tower capacity gives the actual tower heat rejection at these operating conditions (based on historical assumption that the tower must dissipate 0.25W of compressor heat for every watt of heat removed by the evaporator).

Heat rejection capacity and nominal capacity sizing ratio

Used when the [Performance input method](#) is specified as **1-Nominal capacity**, this is the ratio of actual tower heat rejection to nominal capacity. The ratio is defined at entering water at 35C (95F), leaving water at 29.4C (85F), entering air at 25.6C (78F) wetbulb and 35C (95F) drybulb temperatures. Historically this ratio has been set at 1.25 based on the assumption that the tower must dissipate 0.25 W of compressor power for every what of heat removed at the chiller evaporator. The default is 1.25.

Temperature

Design inlet air dry-bulb temperature

This numeric field specifies the inlet air dry-bulb temperature (in °C or °F) at design conditions. This design temperature should correspond with the design values for range temperature, approach temperature, water flow rate, and air flow rate specified on this dialog. The minimum value for this field is 20°C and the default value is 35°C.

Design inlet air wet-bulb temperature

This numeric field specifies the inlet air wet-bulb temperature (in °C or °F) at design conditions. This design temperature should correspond with the design values for range temperature, approach temperature, water flow rate, and air flow rate specified on this dialog. The minimum value for this field is 20°C and the default value is 25.6°C.

Design approach temperature

This numeric field specifies the tower approach temperature (in °C or °F) at design conditions. The approach temperature is the outlet water temperature minus the inlet air wet-bulb temperature. The design approach temperature should correspond with the design values for inlet air wet-bulb temperature, range temperature, water flow rate, and air flow rate specified for this tower. The value for this field must be greater than 0°C and the default value is 3.9°C.

Design range temperature

This numeric field specifies the range temperature (in °C or °F) at design conditions. The range temperature is defined as the inlet water temperature minus the outlet water temperature. The design range temperature should correspond with the design values for inlet air wet-bulb temperature, approach temperature, water flow rate, and air flow rate specified for this tower. The value for this field must be greater than 0°C and the default value is 5.6°C.

Free Convection Regime

Free convection nominal capacity

This is the “nominal” heat rejection capacity of the cooling tower (in W or Btu/h) when the tower is in the “free convection” regime (water flow exists but tower fan is turned off), with entering water at 35°C (95°F), leaving water at 29.4°C (85°F), entering air at 25.6°C (78°F) wet-bulb and 35°C (95°F) dry-bulb temperatures. The design water flow rate is assumed to be 5.382E-8 m³/s per watt of nominal tower capacity (input field above). 125% of this free convection tower capacity gives the actual tower heat rejection at these operating conditions (based on historical assumption that the tower must dissipate 0.25W of compressor heat for every watt of heat removed by the evaporator). The value specified under this setting must be less than the value specified for the field [Nominal capacity](#). If you do not wish to model “free convection”, then this field should be set to 0.0. If you specify a value greater than zero, then the [Air flow rate in free convection regime](#) setting must contain a value greater than zero.

Free convection nominal capacity sizing factor

This field contains the sizing factor to use when calculating the Free convection nominal capacity above. The default is 0.1.

Air flow rate in free convection regime

This is the air flow rate (in m³/s or ft³/min) when the tower is in the “free convection” regime (water flow exists but tower fan is turned off). This value must be less than the value specified for the [Design air flow rate](#). This field may be autocalculated, in which case it is set to a fraction of the [Design air flow rate](#) determined in the following input field. If you do not wish to model “free convection” and are using the [Performance input method 2-UA and Design water flow rate](#) option, then this field should be set to 0.0. If the user specifies the U-Factor Times Area Value at Free Convection Air Flow Rate or Free Convection Capacity as a value greater than zero, then the free convection air flow rate must be specified greater than 0.0.

Free convection regime air flow rate sizing factor

This numeric field contains the sizing factor to use when calculating the free convection regime air flow rate. The default is 0.1.

UA at free convection air flow rate

This is the heat transfer coefficient-area product (W/K or Btu/h-F) when the tower is in the “free convection” regime (water flow exists but tower fan is turned off). This value must be less than the value specified for the

field U-Factor Times Area Value at Design Air Flow Rate. This field may be autosized, in which case it is set to 10% of the **UA at design air flow rate**. If you do not wish to model “free convection” and are using the [Performance input method 2-UA and design water flow rate](#), then this setting should be set to 0.0. If [Performance input method](#) is specified as **1-Nominal capacity**, then EnergyPlus automatically calculates the tower UA based on the tower capacity specified under the setting [Free convection capacity](#).

Basin Heater Settings

Basin heater capacity

This is the capacity of the tower’s electric basin heater (in W/K or Btu/h-F). It is used in conjunction with the **Basin heater setpoint temperature** (below). The basin heater electric power is equal to this setting multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the tower fan is off and water is not flowing through the tower, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero, with a default value of zero.

Basin heater setpoint temperature

This is the set point temperature (°C or °F) for the tower’s electric basin heater. The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the tower fan is off and water is not flowing through the tower. This set point temperature must be greater than or equal to 2°C, and the default value is 2°C.

Basin heater operating schedule

This is the basin heater operating [schedule](#). The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin heater operates when scheduled on and the outdoor air dry-bulb temperature is below the Basin Heater Setpoint Temperature. Regardless of this schedule, the basin heater may only operate when the cooling tower fan is off and water is not flowing through the tower.

Blowdown

Blow-down calculation mode

This setting specifies which method is used to determine blow-down rates. There are two options:

- **1-Concentration ratio** or
- **2-Scheduled rate**.

The choice will determine which of the two models below is used. The default is **1-Concentration ratio**.

Blow-down concentration ratio

This is used to dynamically adjust the rate of blow-down in the cooling tower as a function of the rate of evaporation. Blow-down is water intentionally drained from the tower in order to offset the build up of solids in the water that would otherwise occur because of evaporation. The value entered here is dimensionless. It can be characterized as the ratio of solids in the blow-down water to solids in the make-up water. Typical values for tower operation are 3 to 5. The default value is 3.

Blow-down makeup water usage schedule

This is the [schedule](#) used to define the amount of water (in m³/s only) flushed from the basin on a periodic basis to purge the tower of mineral scale build-up and other contaminants. This schedule is only used if the Blow-down Calculation Mode is set to **2-Scheduled rate**. The amount of water used due to blow-down depends on the makeup water quality and is specific to each geographical location. Typical values range from 0.0002 to 0.0013 m³/s (17.3 to 112.3 m³/day). This water usage is in addition to the amount of water lost to the atmosphere due to evaporation and/or drift. Since blow-down occurs when the basin water contaminant

concentration is high, blow-down only occurs when the cooling tower is active and water is flowing through the tower (regardless of the water usage defined by this schedule).

Capacity Control

Capacity control

This is the cooling capacity control for the cooling tower. Two choices are available:

- **1-Fan cycling** and
- **2-Fluid bypass.**

During part-load conditions, there are two ways to maintain the exiting water temperature at the setpoint: either cycling the tower fan, or bypassing a portion of the tower water with a three-way valve. For **2-Fluid bypass**, the tower fan still runs at full speed for the entire time-step, but only a portion of the water flow goes through the cooling tower media to get cooled while the remaining portion of the water flow gets bypassed, consequently, two water flows then mix at the common water sump to meet the setpoint temperature.

Multi-Cell Tower Settings

Multi-cell tower

This setting is used to enable a multi-cell tower to be defined.

Number of cells

This is the number of cells in the multi-cell cooling tower.

Cell control

This specifies the method used to control the number of cells used to meet the load, the two choices are:

- **1-Minimal cell** - the program will use the minimal number of cells needed, all other cells will be shut down with no water flow. It will attempt to use as few cells as possible to cool the fluid. In no case, however, will the flow per cell be allowed to exceed its maximum value defined by the [Cell maximum water flow rate fraction](#).
- **2-Maximal cell** - as many cells as possible will be turned on. In no case, however, will the flow per cell be allowed to drop below its minimum value specified by the [Cell minimum water flow rate fraction](#).

Cell minimum water flow rate fraction

This is the allowable smallest fraction of the design water flow rate. Flows less than this value will commonly result in fluid distribution problems; the pressure at each nozzle will be too weak for the fluid to be sprayed out in the correct pattern, not all the fill would be wet. The default value is 0.33.

Cell maximum water flow rate fraction

This numeric field specifies the allowable largest fraction of the design water flow rate. The default value is 2.5.

Cooling Tower - Variable Speed Merkel



CoolingTower:VariableSpeed:Merkel

Used in:

- **Condenser loop, supply side**

The variable speed (Merkel) tower model is based on Merkel's theory and is similar to the single-speed and two-speed tower models. The open wet cooling tower is modelled as a counter flow heat exchanger with a

variable-speed fan drawing air through the tower (induced-draft configuration). The model also includes a “free convection” regime where cooling tower performance modelled with the fan off.

For this model, Merkel's theory is modified to include adjustments developed by Scheier to alter the heat transfer effectiveness based on current wetbulb, air flow rates, and water flow rates. The input requires performance curves or lookup tables to describe these three adjustment factors.

For multi-cell towers, capacity and air/water flow rate inputs are for the entire tower.

General

Name

This is the name that you assign to the cooling tower which should be unique. If the supplied name is not unique, the software will automatically adjust the name to avoid duplicate names.

Cooling tower type

You can select from 4 different types of cooling tower:

- [1-Single speed](#)
- [2-Two speed](#)
- [3-Variable speed \(Merkel\) \(described on this help page\)](#)
- [4-Variable speed](#)

Evaporation loss mode

This field is used to choose which method is used to model the amount of water evaporated by the cooling tower. There are two options:

- [1-Saturated exit](#) or
- [2-Loss factor](#).

The default is [1-Saturated exit](#). The user-defined loss factor is entered under [Evaporation loss factor](#), below. By assuming that the air leaving the tower is saturated, the evaporation can be directly calculated using moist air engineering calculations with data available within the cooling tower model (and does not require additional user input).

Evaporation loss factor

This field is used to specify the rate of water evaporated from the cooling tower and lost to the outside air (in %/K or %/F). It is only available if the [Evaporation loss mode](#) (above) is set to [2-Loss factor](#). The evaporation loss is then calculated as a fraction of the circulating condenser water flow and varies with the temperature change in the condenser water. The evaporation rate will equal this value times each degree Kelvin of temperature drop in the condenser water. Typical values are from 0.15 to 0.27 (%/K). The default is 0.2.

Drift loss percent

This field is used to specify the rate of water lost to the exiting air as entrained droplets (%). The drift loss is a percent of the condenser water flow. Typical values for towers with efficient drift eliminators are between 0.002 and 0.2% of the condenser water flow rate. The default value is 0.008%.

Sizing factor

This field allows you to specify a sizing factor for this component to be applied when the component design inputs are autosized. The autosizing calculations are performed as usual and the results are multiplied by the sizing factor. This allows you to size a component to meet part of the design load while continuing to use the autosizing feature.

See also the section on [Autosizing HVAC Components](#)

Airflow

Design air flow rate

This is the design air flow rate induced by the tower fan at full fan speed (in m³/s or ft³/min). The value can be autocalculated, in which case the air flow rate is based on the scalable sizing factor in the following input field

Design air flow rate per unit of nominal capacity

This is the sizing factor to use when auto-calculating the design air flow rate from the nominal capacity, in units of m³/s/W. The default is 2.76316*10⁵.

Minimum air flow rate ratio

The tower fan is allowed to operate between the minimum air flow rate ratio defined here and a maximum air flow rate ratio of 1.0 (which corresponds to the design [maximum] tower air flow rate). Below this value the tower is assumed to operate in the “free convection” regime with the tower fan off. The minimum air flow rate ratio must be greater than or equal to 0.1 and less than or equal to 0.5.

Performance Input Method

Performance input method

This setting allows the method by which the user will specify tower performance to be selected. There are two options:

- **1-Nominal Capacity** where the [Nominal capacity](#) and [Free convection nominal capacity](#) must be entered.
- **2-UA and design water flow rate** where you must enter design UA values, design water flow rates and air flow rates

Water Flow

Design water flow rate

This is the design water flow rate through the tower (in m³/s or gal/min). This value is the flow rate of the condenser loop water being cooled by the tower (not the flow rate of water being sprayed on the outside of the heat exchange coil). If the [Performance input method](#) is specified as **2-UA and design water flow rate**, then a water flow rate greater than zero must be defined or the item can be autosized. If autosized, the design water flow rate is derived from the design load to be rejected by the condenser loop and the design loop temperature difference. If [Performance input method](#) is specified as **1-Nominal capacity**, then the model automatically assumes a water flow rate of 5.382E-8 m³/s per W of tower capacity.

Design water flow rate per unit of nominal capacity

This is a scalable sizing factor for design water flow rate that scales with nominal capacity, in units of (m³/s/W or (gal/min)/(Btu/h)). The default value is 5.382*10⁸ m³/s/W. This field is only used if the **Design air flow rate** is set to Autosize and If [Performance input method](#) is specified as **1-Nominal capacity**. If the [Performance input method](#) is specified as **2-UA and design water flow rate** then the design water flow rate is obtained from the plant sizing result.

U-Factor Times Area Value

UA at design air flow rate

This is the heat transfer coefficient-area product (UA) (in W/K or Btu/h-F) corresponding to the design air and water flow rates specified above. If the Performance Input Method is specified as **2-UA and design water**

flow rate, then a UA value greater than zero but less than or equal to 300,000 W/K must be defined, or the setting can be autosized. If autosized, the design tower UA value is derived from the design load to be rejected by the condenser loop and the design loop delta T, assuming a tower water inlet temperature of 35°C and tower inlet air at 35°C dry-bulb/25.6°C wet-bulb.

UA modifier function of air flow ratio curve

Select the curve that describes how the UA value varies as a function of air flow rate ratio (air flow rate/design air flow rate). The result of this curve is multiplied by the design UA value to adjust for air flow rate, along with the two other modifiers discussed below. The curve must be for one independent variable and should be normalized to 1.0 at an air flow rate ratio of 1.0.

UA modifier function of wet-bulb temperature difference curve

Select the curve that describes how the UA value varies as a function of the current wetbulb temperature. The result of this curve is multiplied by the design UA value to adjust for wetbulb temperatures that differ from design conditions at 25.56°C (78°F), along with the two other modifiers discussed in the previous and following fields. The independent variable is the design wetbulb minus the current outdoor air wetbulb, in units of degrees Celsius. The curve must be for one independent variable and should be normalized to 1.0 at a wetbulb temperature difference of 0.0.

UA modifier function of water flow ratio curve

Select the curve that describes how the UA value varies as a function of the current water flow rate ratio (water flow rate/design water flow rate). The result of this curve is multiplied by the design UA value to adjust for water flow rates that differ from design level, along with the other two modifiers discussed above. The curve must be for one independent variable and should be normalized to 1.0 at a water flow ratio of 1.0.

Capacity

Nominal capacity

Used when the [Performance input method](#) is specified as **1-Nominal capacity**, this is the “nominal” heat rejection capacity of the cooling tower (in W or Btu/h), with entering water at 35°C (95°F), leaving water at 29.4°C (85°F), entering air at 25.6°C (78°F) wet-bulb and 35°C (95°F) dry-bulb temperatures.

The Heat rejection capacity and nominal capacity sizing ratio (below) is applied to this nominal tower capacity to give the actual tower heat rejection at these operating conditions. This field can be autosized.

Heat rejection capacity and nominal capacity sizing ratio

Used when the [Performance input method](#) is specified as **1-Nominal capacity**, this is the ratio of actual tower heat rejection to nominal capacity. The ratio is defined at entering water at 35C (95F), leaving water at 29.4C (85F), entering air at 25.6C (78F) wetbulb and 35C (95F) drybulb temperatures. Historically this ratio has been set at 1.25 based on the assumption that the tower must dissipate 0.25 W of compressor power for every what of heat removed at the chiller evaporator. The default is 1.25.

Fan Power

Design fan power

This numeric field contains the fan power (in W) at the design (maximum) air flow rate through the tower. A value greater than zero must be specified or this field can be autocalculated. When the field is autocalculated the following input field is used to size the fan power based on nominal capacity.

Design fan power per unit of nominal capacity

This numeric field contains the sizing factor to use when calculating the design fan power from the nominal capacity, in units of W/W. This field is only used if the previous is set to autocalculate. The default values is 0.0105.

Fan power modifier function of air flow rate ratio curve

Select the curve that describes fan power ratio (fan power/design fan power) as a function of air flow rate ratio (air flow rate/design air flow rate). The curve must be for one independent variable, typically a cubic, and should be normalized to 1.0 at an air flow rate ratio of 1.0.

Temperature

Design inlet air dry-bulb temperature

This numeric field specifies the inlet air dry-bulb temperature (in °C or °F) at design conditions. This design temperature should correspond with the design values for range temperature, approach temperature, water flow rate, and air flow rate specified on this dialog. The minimum value for this field is 20°C and the default value is 35°C.

Design inlet air wet-bulb temperature

This numeric field specifies the inlet air wet-bulb temperature (in °C or °F) at design conditions. This design temperature should correspond with the design values for range temperature, approach temperature, water flow rate, and air flow rate specified on this dialog. The minimum value for this field is 20°C and the default value is 25.6°C.

Design approach temperature

This numeric field specifies the tower approach temperature (in °C or °F) at design conditions. The approach temperature is the outlet water temperature minus the inlet air wet-bulb temperature. The design approach temperature should correspond with the design values for inlet air wet-bulb temperature, range temperature, water flow rate, and air flow rate specified for this tower. The value for this field must be greater than 0°C and the default value is 3.9°C.

Design range temperature

This numeric field specifies the range temperature (in °C or °F) at design conditions. The range temperature is defined as the inlet water temperature minus the outlet water temperature. The design range temperature should correspond with the design values for inlet air wet-bulb temperature, approach temperature, water flow rate, and air flow rate specified for this tower. The value for this field must be greater than 0°C and the default value is 5.6°C.

Free Convection Regime

Free convection nominal capacity

This setting contains the “nominal” heat rejection capacity of the cooling tower (in W or Btu/h) when the tower is in the “free convection” regime (water flow exists but tower fan is turned off), with entering water at 35°C (95°F), leaving water at 29.4°C (85°F), entering air at 25.6°C (78°F) wetbulb and 35°C (95°F) drybulb temperatures. The Heat Rejection Capacity and Nominal Capacity Sizing Ratio is applied to this free convection tower capacity to give the actual tower heat rejection at these operating conditions.

If you do not wish to model “free convection”, then this capacity should be set to 0.0. If you set a value greater than zero, then the [Air flow rate in free convection regime](#) must also have a value greater than zero. This field may be autocalculated, in which case it is set to a fraction of the [Nominal capacity](#) determined using the following field.

Warning: Due to a [known bug](#) in EnergyPlus 8.9, when using the **2-UA and Design water flow rate Performance input method** option and [UA at design air flow rate](#) is set to "Autosize" you must set **Free convection nominal capacity** input to “Autocalculate”. You will need to change the [Performance input method](#) to **1-Nominal capacity** to see it, but remember to change it back again to **2-UA and design water flow rate** after checking.

Free convection nominal capacity sizing factor

Enter the sizing factor to use when calculating the free convection capacity. The default is 0.1.

Air flow rate in free convection regime

This is the air flow rate (in m³/s or ft³/min) when the tower is in the “free convection” regime (water flow exists but tower fan is turned off). This value must be less than the value specified for the [Design air flow rate](#). This field may be autocalculated, in which case it is set to a fraction of the [Design air flow rate](#) determined in the following input field.

If you do not wish to model “free convection” and are using the [Performance input method 2-UA and Design water flow rate](#) option, then this field should be set to 0.0. On the other hand, if you set the [UA at free convection air flow rate](#) or [Free convection nominal capacity](#) to a value greater than zero, then the free convection air flow rate must also be greater than 0.0.

Free convection regime air flow rate sizing factor

This numeric field contains the sizing factor to use when calculating the free convection regime air flow rate. The default is 0.1.

UA at free convection air flow rate

This is the heat transfer coefficient-area product (W/K or Btu/h-F) when the tower is in the “free convection” regime (water flow exists but tower fan is turned off). If [Performance input method](#) is set to **2-UA and design water flow rate**, then a UA value greater than zero but less than or equal to 300,000 must be defined, or the field can be autosized. If autosized the design tower UA value is derived from the design load to be rejected by the condenser loop and the design loop delta T, assuming a tower water inlet temperature of 35C and tower inlet air at 35C drybulb/25.6C wetbulb. If [Performance input method](#) is specified as **1-Nominal capacity**, then EnergyPlus automatically calculates the tower UA based on the tower capacity specified under the setting [Free convection capacity](#).

Free convection UA sizing factor

This numeric field contains the sizing factor to use when calculating the free convection regime UA value. The default is 0.1. This field is only used if the previous field is set to autocalculate and [Performance input method](#) is set to **2-UA and Design water flow rate**.

Basin Heater Settings

Basin heater capacity

This is the capacity of the tower's electric basin heater (in W/K or Btu/h-F). It is used in conjunction with the [Basin heater setpoint temperature](#) (below). The basin heater electric power is equal to this setting multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the tower fan is off and water is not flowing through the tower, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero, with a default value of zero.

Basin heater setpoint temperature

This is the set point temperature (°C or °F) for the tower's electric basin heater. The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the tower fan is off and water is not flowing through the tower. This set point temperature must be greater than or equal to 2°C, and the default value is 2°C.

Basin heater operating schedule

This is the basin heater operating [schedule](#). The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin

heater operates when scheduled on and the outdoor air dry-bulb temperature is below the Basin Heater Setpoint Temperature. Regardless of this schedule, the basin heater may only operate when the cooling tower fan is off and water is not flowing through the tower.

Blowdown

Blow-down calculation mode

This setting specifies which method is used to determine blow-down rates. There are two options:

- **1-Concentration ratio** or
- **2-Scheduled rate**.

The choice will determine which of the two models below is used. The default is **1-Concentration ratio**.

Blow-down concentration ratio

This is used to dynamically adjust the rate of blow-down in the cooling tower as a function of the rate of evaporation. Blow-down is water intentionally drained from the tower in order to offset the build up of solids in the water that would otherwise occur because of evaporation. The value entered here is dimensionless. It can be characterized as the ratio of solids in the blow-down water to solids in the make-up water. Typical values for tower operation are 3 to 5. The default value is 3.

Blow-down makeup water usage schedule

This is the [schedule](#) used to define the amount of water (in m³/s only) flushed from the basin on a periodic basis to purge the tower of mineral scale build-up and other contaminants. This schedule is only used if the Blow-down Calculation Mode is set to **2-Scheduled rate**. The amount of water used due to blow-down depends on the makeup water quality and is specific to each geographical location. Typical values range from 0.0002 to 0.0013 m³/s (17.3 to 112.3 m³/day). This water usage is in addition to the amount of water lost to the atmosphere due to evaporation and/or drift. Since blow-down occurs when the basin water contaminant concentration is high, blow-down only occurs when the cooling tower is active and water is flowing through the tower (regardless of the water usage defined by this schedule).

Multi-Cell Tower Settings

Multi-cell tower

This setting is used to enable a multi-cell tower to be defined.

Number of cells

This is the number of cells in the multi-cell cooling tower.

Cell control

This specifies the method used to control the number of cells used to meet the load, the two choices are:

- **1-Minimal cell** - the program will use the minimal number of cells needed, all other cells will be shut down with no water flow. It will attempt to use as few cells as possible to cool the fluid. In no case, however, will the flow per cell be allowed to exceed its maximum value defined by the [Cell maximum water flow rate fraction](#).
- **2-Maximal cell** - as many cells as possible will be turned on. In no case, however, will the flow per cell be allowed to drop below its minimum value specified by the [Cell minimum water flow rate fraction](#).

Cell minimum water flow rate fraction

This is the allowable smallest fraction of the design water flow rate. Flows less than this value will commonly result in fluid distribution problems; the pressure at each nozzle will be too weak for the fluid to be sprayed out in the correct pattern, not all the fill would be wet. The default value is 0.33.

Cell maximum water flow rate fraction

This numeric field specifies the allowable largest fraction of the design water flow rate. The default value is 2.5.

Cooling Tower - Variable Speed



CoolingTower:VariableSpeed

Used in:

- **Condenser loop, supply side**

The variable speed tower model is based on empirical curve fits of manufacturer's performance data or field measurements. Tower performance can be specified at design conditions, and empirical curves are used to determine the approach temperature and fan power at off-design conditions. Tower performance is specified by entering the inlet air wet-bulb temperature, tower range, and tower approach temperature at the design conditions. The corresponding water flow rate (within $\pm 25\%$ of the tower's rated water mass flow rate), air flow rate, and fan power must also be specified. The model will account for tower performance in the "free convection" regime, when the tower fan is off but the water pump remains on and heat transfer still occurs (albeit at a low level). Basin heater operation and makeup water usage (due to evaporation, drift, and blowdown) are also modelled.

The model first checks to determine the impact of "free convection" on the tower exiting water temperature. If the exiting water temperature based on "free convection" is at or below the set point, then the variable-speed tower fan is not turned on. If the exiting water temperature is above the set point after "free convection" is modelled, then the variable-speed tower fan is turned on to reduce the exiting water temperature. Tower fan power is calculated based on the tower air flow rate required to achieve the exiting water set point temperature. Cooling towers here are "wet" and consume water through evaporation, drift, and blowdown. The model can be used to predict water consumed by the towers.

For multi-cell towers, capacity and air/water flow rate inputs are for the entire tower.

General

Name

This is the name that you assign to the cooling tower which should be unique. If the supplied name is not unique, the software will automatically adjust the name to avoid duplicate names.

Cooling tower type

You can select from 4 different types of cooling tower:

- [1-Single speed](#)
- [2-Two speed](#)
- [3-Variable speed \(Merkel\)](#)
- [4-Variable speed \(described on this help page\)](#)

Evaporation loss mode

This field is used to choose which method is used to model the amount of water evaporated by the cooling tower. There are two options:

- **1-Saturated exit** or
- **2-Loss factor.**

The default is **1-Saturated exit**. The user-defined loss factor is entered under **Evaporation loss factor**, below. By assuming that the air leaving the tower is saturated, the evaporation can be directly calculated using moist air engineering calculations with data available within the cooling tower model (and does not require additional user input).

Evaporation loss factor

This field is used to specify the rate of water evaporated from the cooling tower and lost to the outside air (in %/K or %/F). It is only available if the **Evaporation loss mode** (above) is set to **2-Loss factor**. The evaporation loss is then calculated as a fraction of the circulating condenser water flow and varies with the temperature change in the condenser water. The evaporation rate will equal this value times each degree Kelvin of temperature drop in the condenser water. Typical values are from 0.15 to 0.27 (%/K). The default is 0.2.

Drift loss percent

This field is used to specify the rate of water lost to the exiting air as entrained droplets (%). The drift loss is a percent of the condenser water flow. Typical values for towers with efficient drift eliminators are between 0.002 and 0.2% of the condenser water flow rate. The default value is 0.008%.

Sizing factor

This field allows you to specify a sizing factor for this component to be used when the component design inputs are autosized. The autosizing calculations are performed as usual and the results are multiplied by the sizing factor. The Sizing factor allows you to size a component to meet part of the design load while continuing to use the autosizing feature.

For this component the inputs that would be altered by the sizing factor are:

1. Design water flow rate;
2. Design air flow rate;
3. Design fan power.

See also the section on [Autosizing HVAC Components](#)

Airflow

Design air flow rate

This numeric field contains the design (maximum) air flow rate through the tower in m3/s. A value greater than zero must be specified or this field can be autosized. When autosizing the design air flow rate is assumed to be the design fan power multiplied by a total fan efficiency of 0.5 and divided by a fan pressure rise of 190 Pascals. A correction for altitude is also included as follows:

$$\text{Design Air Flow Rate} = \frac{0.5(\text{Fan power at Design Air Flow Rate})}{190.} \left(\frac{101325.}{P_{atm}} \right)$$

Minimum air flow rate ratio

The tower fan is allowed to operate between the minimum air flow rate ratio defined here and a maximum air flow rate ratio of 1.0 (which corresponds to the design [maximum] tower air flow rate). Below this value the tower is assumed to operate either in the “free convection” regime with the tower fan off, or the tower fan is cycled on/off to maintain the exiting water set point temperature. The minimum air flow rate ratio must be greater than or equal to 0.2 and less than or equal to 0.5, with a default value of 0.2.

Model Type

Model type

This setting allows you to select the type of empirical model used to simulate the tower’s thermal performance (approach temperature). Options are:

- **1-Cool tools cross flow**
- **2-York calc**

- **3-Cool tools user defined**
- **4-York calc user defined**

1-Cool tools cross flow and **2-York calc** are empirical models with the equation form and model coefficients already defined within EnergyPlus.

If **3-Cool tools user defined** or **4-York calc user defined** are selected, the user must specify a valid Model Coefficient option in the next input field to reference an appropriate cooling tower performance component.

Model coefficient

Select the Cooling tower performance component that defines the details of the performance characteristics of the cooling tower.

If **3-Cool tools user defined** is selected then a component from the **Cool tools** category should be selected. Otherwise if the model type is **4-York calc user defined** then a component from the **York calc** category must be selected.

Water Flow

Design water flow rate

This is the design water flow rate through the tower (in m3/s or gal/min). The value entered should be within $\pm 25\%$ of the tower's rated water mass flow rate as specified by the manufacturer if Model type is **1-Cool tools cross flow** or **2-York calc**. This constraint may be different for Model type **3-Cool tools user defined** or **4-York calc user defined**.

This value is the flow rate of the condenser loop water being cooled by the tower. A value greater than zero must be specified or this field can be autosized in which case the design water flow rate is derived from the design load to be rejected by the condenser loop and the design loop delta T. The design water flow rate should correspond with the design values specified for the inlet air wet-bulb temperature, approach temperature, range temperature, and air flow rate. When this field is not autosized, the condenser loop flow rate specified in other objects should be within $\pm 25\%$ of the tower's rated water flow rate. A different range is permissible if the Model type is user defined.

Fan Power

Design fan power

This is the fan power (in W) at the design (maximum) air flow rate through the tower. A value greater than zero must be specified or this field can be autosized. If autosized, the fan power is calculated as follows:

$$\text{Design fan power} = 0.0105 \cdot \rho_{\text{water}} \cdot C_{p,\text{water}} \cdot \text{Design Water Flow Rate} \cdot \text{Design Loop } \Delta T$$

Fan power ratio function of air flow rate ratio curve

Select the curve for fan power ratio (fan power/design fan power) as a function of air flow rate ratio (air flow rate/design air flow rate). The curve object must be a cubic curve and should be normalized to 1.0 at an air flow rate ratio of 1.0.

Temperature

Design inlet air wet-bulb temperature

This numeric field specifies the inlet air wet-bulb temperature (in °C or °F) at design conditions. This design temperature should correspond with the design values for range temperature, approach temperature, water flow rate, and air flow rate specified on this dialog. The minimum value for this field is 20°C and the default value is 25.6°C.

Design approach temperature

This numeric field specifies the tower approach temperature (in °C or °F) at design conditions. The approach temperature is the outlet water temperature minus the inlet air wet-bulb temperature. The design approach temperature should correspond with the design values for inlet air wet-bulb temperature, range temperature, water flow rate, and air flow rate specified for this tower. The value for this field must be greater than 0°C and the default value is 3.9°C.

Design range temperature

This numeric field specifies the range temperature (in °C or °F) at design conditions. The range temperature is defined as the inlet water temperature minus the outlet water temperature. The design range temperature should correspond with the design values for inlet air wet-bulb temperature, approach temperature, water flow rate, and air flow rate specified for this tower. The value for this field must be greater than 0°C and the default value is 5.6°C.

Free Convection Regime

Fraction of tower capacity in free convection regime

Enter the fraction of tower capacity available in the free convection regime (i.e., when the tower fan is off but water continues to flow through the tower). The tower fan does not operate when the free convection tower capacity is able to meet or exceed the exiting water set point temperature. The air flow rate through the tower in the free convection regime is assumed to be this same fraction of the tower design air flow rate. The fraction of tower capacity in free convection regime must be greater than or equal to 0 and less than or equal to 0.2, with a default value of 0.125.

Basin Heater Settings

Basin heater capacity

This is the capacity of the tower's electric basin heater (in W/K or Btu/h-F). It is used in conjunction with the [Basin heater setpoint temperature](#) (below). The basin heater electric power is equal to this setting multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the tower fan is off and water is not flowing through the tower, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero, with a default value of zero.

Basin heater setpoint temperature

This is the set point temperature (°C or °F) for the tower's electric basin heater. The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the tower fan is off and water is not flowing through the tower. This set point temperature must be greater than or equal to 2°C, and the default value is 2°C.

Basin heater operating schedule

This is the basin heater operating [schedule](#). The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin heater operates when scheduled on and the outdoor air dry-bulb temperature is below the Basin Heater Setpoint Temperature. Regardless of this schedule, the basin heater may only operate when the cooling tower fan is off and water is not flowing through the tower.

Blowdown

Blow-down calculation mode

This setting specifies which method is used to determine blow-down rates. There are two options:

- **1-Concentration ratio** or

- **2-Scheduled rate.**

The choice will determine which of the two models below is used. The default is **1-Concentration ratio**.

Blow-down concentration ratio

This is used to dynamically adjust the rate of blow-down in the cooling tower as a function of the rate of evaporation. Blow-down is water intentionally drained from the tower in order to offset the build up of solids in the water that would otherwise occur because of evaporation. The value entered here is dimensionless. It can be characterized as the ratio of solids in the blow-down water to solids in the make-up water. Typical values for tower operation are 3 to 5. The default value is 3.

Blow-down makeup water usage schedule

This is the [schedule](#) used to define the amount of water (in m³/s only) flushed from the basin on a periodic basis to purge the tower of mineral scale build-up and other contaminants. This schedule is only used if the [Blow-down calculation mode](#) is set to **2-Scheduled rate**. The amount of water used due to blow-down depends on the makeup water quality and is specific to each geographical location. Typical values range from 0.0002 to 0.0013 m³/s (17.3 to 112.3 m³/day). This water usage is in addition to the amount of water lost to the atmosphere due to evaporation and/or drift. Since blow-down occurs when the basin water contaminant concentration is high, blow-down only occurs when the cooling tower is active and water is flowing through the tower (regardless of the water usage defined by this schedule).

Multi-Cell Tower Settings

Multi-cell tower

This setting is used to enable a multi-cell tower to be defined.

Number of cells

This is the number of cells in the multi-cell cooling tower.

Cell control

This specifies the method used to control the number of cells used to meet the load, the two choices are:

- **1-Minimal cell** - the program will use the minimal number of cells needed, all other cells will be shut down with no water flow. It will attempt to use as few cells as possible to cool the fluid. In no case, however, will the flow per cell be allowed to exceed its maximum value defined by the [Cell maximum water flow rate fraction](#).
- **2-Maximal cell** - as many cells as possible will be turned on. In no case, however, will the flow per cell be allowed to drop below its minimum value specified by the [Cell minimum water flow rate fraction](#).

Cell minimum water flow rate fraction

This is the allowable smallest fraction of the design water flow rate. Flows less than this value will commonly result in fluid distribution problems; the pressure at each nozzle will be too weak for the fluid to be sprayed out in the correct pattern, not all the fill would be wet. The default value is 0.33.

Cell maximum water flow rate fraction

This numeric field specifies the allowable largest fraction of the design water flow rate. The default value is 2.5.

Cooling Tower - Model Coefficients

Variable speed cooling towers can be modelled by EnergyPlus with user-selectable performance based on the CoolTools correlation, YorkCalc correlation, or user-defined coefficients for either the CoolTools or

YorkCalc correlations. The first 2 of these are built into EnergyPlus while the second 2 require additional performance data as explained below.

Cool Tools

The empirical CoolTools tower correlation uses a set of 35 coefficients to model the thermal performance (approach temperature) of a cooling tower based on four independent variables. If you specify the [Model type](#) as **3-Cool Tools cross flow** on the Cooling Tower dialog for a Variable Speed object, then the 35 coefficients derived for the CoolTools simulation model are used and these coefficients are already defined within EnergyPlus. If on the other hand you select the **3-Cool Tools user defined** option, then you must also select a **Cool tools** Cooling Tower Performance component to define the 35 coefficients that will be used by the CoolTools correlation.

To specify the valid range for which the model coefficients were derived, it is necessary to enter the minimum and maximum values for: inlet air wet-bulb temperature, range temperature, approach temperature, and water mass flow rate ratio. For all of these variables, the program issues warnings if the actual values are beyond the minimum/maximum values specified. For inlet air wet-bulb temperature and water mass flow rate ratio, the values of these variables used in the calculation of approach temperature are limited to be within the valid minimum/maximum range specified. For approach and range, the warnings are issued if the values are beyond the specified minimum/maximum range but the actual values are still used.

The CoolTools correlation has four independent variables: inlet air wet-bulb temperature (Twb), tower range temperature (Tr), water flow rate ratio (FRwater), and air flow rate ratio (FRair). Temperatures are in units of °C and flow rate ratios are dimensionless (actual flow rate divided by design flow rate). Using these independent variables, tower approach temperature (°C) is calculated as follows:

$$\begin{aligned} \text{Approach} = & \text{Coeff}(1) + \text{Coeff}(2) \cdot \text{FRair} + \text{Coeff}(3) \cdot (\text{FRair})^2 + \\ & \text{Coeff}(4) \cdot (\text{FRair})^3 + \text{Coeff}(5) \cdot \text{FRwater} + \\ & \text{Coeff}(6) \cdot \text{FRair} \cdot \text{FRwater} + \text{Coeff}(7) \cdot (\text{FRair})^2 \cdot \text{FRwater} + \\ & \text{Coeff}(8) \cdot (\text{FRwater})^2 + \text{Coeff}(9) \cdot \text{FRair} \cdot (\text{FRwater})^2 + \\ & \text{Coeff}(10) \cdot (\text{FRwater})^3 + \text{Coeff}(11) \cdot \text{Twb} + \text{Coeff}(12) \cdot \text{FRair} \cdot \text{Twb} + \\ & \text{Coeff}(13) \cdot (\text{FRair})^2 \cdot \text{Twb} + \text{Coeff}(14) \cdot \text{FRwater} \cdot \text{Twb} + \\ & \text{Coeff}(15) \cdot \text{FRair} \cdot \text{FRwater} \cdot \text{Twb} + \text{Coeff}(16) \cdot (\text{FRwater})^2 \cdot \text{Twb} + \\ & \text{Coeff}(17) \cdot (\text{Twb})^2 + \text{Coeff}(18) \cdot \text{FRair} \cdot (\text{Twb})^2 + \\ & \text{Coeff}(19) \cdot \text{FRwater} \cdot (\text{Twb})^2 + \text{Coeff}(20) \cdot (\text{Twb})^3 + \text{Coeff}(21) \cdot \text{Tr} + \\ & \text{Coeff}(22) \cdot \text{FRair} \cdot \text{Tr} + \text{Coeff}(23) \cdot \text{FRair} \cdot \text{FRair} \cdot \text{Tr} + \\ & \text{Coeff}(24) \cdot \text{FRwater} \cdot \text{Tr} + \text{Coeff}(25) \cdot \text{FRair} \cdot \text{FRwater} \cdot \text{Tr} + \\ & \text{Coeff}(26) \cdot (\text{FRwater})^2 \cdot \text{Tr} + \text{Coeff}(27) \cdot \text{Twb} \cdot \text{Tr} + \\ & \text{Coeff}(28) \cdot \text{FRair} \cdot \text{Twb} \cdot \text{Tr} + \text{Coeff}(29) \cdot \text{FRwater} \cdot \text{Twb} \cdot \text{Tr} + \\ & \text{Coeff}(30) \cdot (\text{Twb})^2 \cdot \text{Tr} + \text{Coeff}(31) \cdot (\text{Tr})^2 + \text{Coeff}(32) \cdot \text{FRair} \cdot (\text{Tr})^2 + \\ & \text{Coeff}(33) \cdot \text{FRwater} \cdot (\text{Tr})^2 + \text{Coeff}(34) \cdot \text{Twb} \cdot (\text{Tr})^2 + \text{Coeff}(35) \cdot (\text{Tr})^3 \end{aligned}$$

It is recommended that a broad set of cooling tower performance data be used to generate these model coefficients. The data set used to create the model coefficients should cover the entire range of water and air flow rate ratios and inlet air wet-bulb, range, and approach temperatures expected during the simulation.

Name

Enter a unique name for the cooling tower performance data.

Source

Enter a description of the source of the data.

Category

This must be **Cool tools** for entering user-defined Cool tools data.

Minimum inlet air wet-bulb temperature

This numeric field contains the minimum inlet air wet-bulb temperature (in °C or °F) to be used by the model (approach temperature correlation). Inlet air wet-bulb temperatures less than this value will not be used;

instead, the minimum inlet air wet-bulb temperature specified here will be used by the correlation and a warning will be issued.

Maximum inlet air wet-bulb temperature

This numeric field contains the maximum inlet air wet-bulb temperature (in °C or °F) to be used by the model (approach temperature correlation). Inlet air wet-bulb temperatures greater than this value will not be used; instead, the maximum inlet air wet-bulb temperature specified here will be used by the correlation and a warning will be issued.

Minimum range temperature

This numeric field contains the minimum range temperature (in Delta °C or Delta °F) (inlet water temperature minus outlet water temperature) to be used by the empirical model. If the range temperature is less than this value the actual range temperature is still passed to the empirical model but a warning will be issued.

Maximum range temperature

This numeric field contains the maximum range temperature (in Delta °C or Delta °F) (inlet water temperature minus outlet water temperature) to be used by the empirical model. If the range temperature is greater than this value the actual range temperature is still passed to the empirical model but a warning will be issued.

Minimum approach temperature

This numeric field contains the minimum approach temperature (in Delta °C or Delta °F) (outlet water temperature minus inlet air wet-bulb temperature) to be used by the empirical model. If the calculated approach temperature is less than this value then the calculated value is still used but a warning will be issued.

Maximum approach temperature

This numeric field contains the maximum approach temperature (in Delta °C or Delta °F) (outlet water temperature minus inlet air wet-bulb temperature) to be used by the empirical model. If the calculated approach temperature is greater than this value then the calculated value is still used but a warning will be issued.

Minimum water flow rate ratio

This numeric field contains the minimum water flow rate ratio (ratio of actual water flow rate to rated water flow rate) to be used by the empirical model. Water flow rate ratios less than this value will not be used; instead, the minimum water flow rate ratio specified here will be used by the model and a warning will be issued.

Maximum water flow rate ratio

This numeric field contains the maximum water flow rate ratio (ratio of actual water flow rate to rated water flow rate) to be used by the empirical model. Water flow rate ratios greater than this value will not be used; instead, the maximum water flow rate ratio specified here will be used by the model and a warning will be issued.

Coefficient 1 to 35

These numeric fields contain the coefficients to be used by the CoolTools approach temperature correlation shown above.

York Calc

The empirical YorkCalc tower correlation uses a set of 27 coefficients to model the thermal performance (approach temperature) of a variable speed cooling tower based on three independent variables. If you specify the [Model type](#) as **2-York calc** on the Cooling Tower dialog for a Variable Speed object, then the 27 coefficients derived for the YorkCalc simulation model are used and these coefficients are already defined within EnergyPlus. If on the other hand you select the [Model type](#) as **4-York calc user defined** then you must also select a Cooling Tower Performance object from the **York calc** category to define the 27 coefficients that will be used by the YorkCalc correlation.

To specify the valid range for which the model coefficients were derived, it is necessary to enter the minimum and maximum values for inlet air wet-bulb temperature, range temperature, approach temperature, and water mass flow rate ratio. It is also necessary to specify the maximum valid liquid-to-gas ratio. For all of these variables, the program issues warnings if the actual values are beyond the minimum/maximum values specified. For inlet air wet-bulb temperature and water mass flow rate ratio, the values of these variables used in the calculation of approach temperature are limited to be within the valid minimum/maximum range specified. For approach, range, and liquid-to-gas ratio the warnings are issued if the values are beyond the specified minimum/maximum range but the actual values are still used.

The YorkCalc correlation has three independent variables: inlet air wet-bulb temperature (Twb), tower range temperature (Tr), and the liquid-to-gas ratio (ratio of water flow rate ratio to air flow rate ratio = LGRatio). Temperatures are in units of °C and liquid-to-gas ratio is dimensionless. Using these independent variables, an approach temperature (°C) is calculated as follows:

$$\begin{aligned} \text{Approach} = & \text{Coeff}(1) + \text{Coeff}(2) \cdot \text{Twb} + \text{Coeff}(3) \cdot \text{Twb}^2 + \text{Coeff}(4) \cdot \text{Tr} + \\ & \text{Coeff}(5) \cdot \text{Twb} \cdot \text{Tr} + \text{Coeff}(6) \cdot \text{Twb}^2 \cdot \text{Tr} + \text{Coeff}(7) \cdot \text{Tr}^2 + \\ & \text{Coeff}(8) \cdot \text{Twb} \cdot \text{Tr}^2 + \text{Coeff}(9) \cdot \text{Twb}^2 \cdot \text{Tr}^2 + \text{Coeff}(10) \cdot \text{LGRatio} + \\ & \text{Coeff}(11) \cdot \text{Twb} \cdot \text{LGRatio} + \text{Coeff}(12) \cdot \text{Twb}^2 \cdot \text{LGRatio} + \\ & \text{Coeff}(13) \cdot \text{Tr} \cdot \text{LGRatio} + \text{Coeff}(14) \cdot \text{Twb} \cdot \text{Tr} \cdot \text{LGRatio} + \\ & \text{Coeff}(15) \cdot \text{Twb}^2 \cdot \text{Tr} \cdot \text{LGRatio} + \text{Coeff}(16) \cdot \text{Tr}^2 \cdot \text{LGRatio} + \\ & \text{Coeff}(17) \cdot \text{Twb} \cdot \text{Tr}^2 \cdot \text{LGRatio} + \text{Coeff}(18) \cdot \text{Twb}^2 \cdot \text{Tr}^2 \cdot \text{LGRatio} + \\ & \text{Coeff}(19) \cdot \text{LGRatio}^2 + \text{Coeff}(20) \cdot \text{Twb} \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(21) \cdot \text{Twb}^2 \cdot \text{LGRatio}^2 + \text{Coeff}(22) \cdot \text{Tr} \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(23) \cdot \text{Twb} \cdot \text{Tr} \cdot \text{LGRatio}^2 + \text{Coeff}(24) \cdot \text{Twb}^2 \cdot \text{Tr} \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(25) \cdot \text{Tr}^2 \cdot \text{LGRatio}^2 + \text{Coeff}(26) \cdot \text{Twb} \cdot \text{Tr}^2 \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(27) \cdot \text{Twb}^2 \cdot \text{Tr}^2 \cdot \text{LGRatio}^2 \end{aligned}$$

It is recommended that a broad set of cooling tower performance data be used to generate these model coefficients. The data set used to create the model coefficients should cover the entire range of water and air flow rate ratios and inlet air wet-bulb, range, and approach temperatures expected during the simulation.

Name

Enter a unique name for the cooling tower performance data.

Source

Enter a description of the source of the data.

Category

This must be **York calc** for entering user-defined York calc data.

Minimum inlet air wet-bulb temperature

This numeric field contains the minimum inlet air wet-bulb temperature (in °C or °F) to be used by the model (approach temperature correlation). Inlet air wet-bulb temperatures less than this value will not be used; instead, the minimum inlet air wet-bulb temperature specified here will be used by the correlation and a warning will be issued.

Maximum inlet air wet-bulb temperature

This numeric field contains the maximum inlet air wet-bulb temperature (in °C or °F) to be used by the model (approach temperature correlation). Inlet air wet-bulb temperatures greater than this value will not be used; instead, the maximum inlet air wet-bulb temperature specified here will be used by the model and a warning will be issued.

Minimum range temperature

This numeric field contains the minimum range temperature (in Delta °C or Delta °F) (inlet water temperature minus outlet water temperature) to be used by the empirical model. If the range temperature is less than this value the actual range temperature is still passed to the empirical model but a warning will be issued.

Maximum range temperature

This numeric field contains the maximum range temperature (in Delta °C or Delta °F) (inlet water temperature minus outlet water temperature) to be used by the empirical model. If the range temperature is greater than this value the actual range temperature is still passed to the empirical model but a warning will be issued.

Minimum approach temperature

This numeric field contains the minimum approach temperature (in Delta °C or Delta °F) (outlet water temperature minus inlet air wet-bulb temperature) to be used by the empirical model. If the calculated approach temperature is less than this value then the calculated value is still used but a warning will be issued.

Maximum approach temperature

This numeric field contains the maximum approach temperature (in Delta °C or Delta °F) (outlet water temperature minus inlet air wet-bulb temperature) to be used by the empirical model. If the calculated approach temperature is greater than this value then the calculated value is still used but a warning will be issued.

Minimum water flow rate ratio

This numeric field contains the minimum water flow rate ratio (ratio of actual water flow rate to rated water flow rate) to be used by the empirical model. Water flow rate ratios less than this value will not be used; instead, the minimum water flow rate ratio specified here will be used by the model and a warning will be issued.

Maximum water flow rate ratio

This numeric field contains the maximum water flow rate ratio (ratio of actual water flow rate to rated water flow rate) to be used by the empirical model. Water flow rate ratios greater than this value will not be used; instead, the maximum water flow rate ratio specified here will be used by the model and a warning will be issued.

Maximum liquid to gas ratio

This numeric field contains the maximum liquid-to-gas ratio (ratio of actual water flow rate ratio [capped to be within the minimum/maximum water flow rate ratio defined above as necessary] to actual air flow rate ratio) to be used by the empirical model. If the liquid-to-gas ratio is greater than this value the actual liquid to gas ratio is still passed to the empirical model but a warning will be issued.

Coefficient 1 to 27

These numeric fields contain the coefficients to be used by the YorkCalc approach temperature correlation shown above.

Fluid Cooler



FluidCooler:SingleSpeed
FluidCooler:TwoSpeed

Used in:

- Condenser loop, supply side

Fluid coolers (aka dry coolers) are used to reject heat from condenser loops in a similar way to cooling towers but with much reduced water consumption. The Fluid cooler is modelled as a cross flow heat exchanger (both streams unmixed) with either single-speed or two-speed fans (induced draft configuration).

Fluid cooler performance is defined through one of the two methods: design heat transfer coefficient-area product (UA) and design water flow rate, or nominal fluid cooler capacity at a specific rating point. Regardless of which method is chosen, the design airflow rate and corresponding fan power must be specified.

The fluid cooler seeks to maintain the temperature of the water exiting the fluid cooler at (or below) a set point. The set point schedule value is defined by the SPM downstream of the condenser supply sub loop.

The single-speed model assumes that part-load operation is represented by a simple linear interpolation between two steady-state regimes (i.e., fluid cooler fan on for the entire simulation timestep and fluid cooler fan off for the entire simulation timestep). Cyclic losses are not taken into account.

The two speed model first runs at low speed and calculates the fluid cooler exiting water temperature. If the exiting water temperature based on “low speed” is at or below the set point, then the fluid cooler fan runs at this speed or below this speed. If the exiting water temperature remains above the set point after “low speed” is modelled, then the fluid cooler fan runs at high speed to reduce the exiting water temperature. The model assumes that part-load operation is represented by a simple linear interpolation between two steady-state regimes (i.e. fluid cooler fan at high and low speeds for the entire simulation timestep). Cyclic losses are not taken into account.

General

Name

This alpha field contains the identifying name for the Fluid Cooler.

Design air flow rate U-factor times area value

This numeric field contains the heat transfer coefficient-area product (UA) (in W/K or Btu/h-F) corresponding to the design air and water flow rates specified above. If the Performance input method (above) is specified as **2-UA and design water flow rate**, then a UA value greater than zero but less than or equal to 300,000 must be defined, or the field can be autosized. If autosized, the Plant sizing data is used and the design fluid cooler UA value is derived from the design load to be rejected by the condenser loop and the design loop delta T. The fluid cooler inlet air dry-bulb and wetbulb temperature are taken from the input. If **Performance input method** is specified as **1-Nominal capacity** then this field is not available since the model automatically calculates the fluid cooler UA based on the fluid cooler capacity and nominal conditions specified below.

Nominal capacity

This numeric input field contains the “nominal” heat rejection capacity of the fluid cooler (in W or Btu/hr), Nominal conditions i.e. entering air dry-bulb temperature, entering air wet-bulb temperature and Entering water temperature should be provided in the input.

Design entering water temperature

This numeric field contains entering water temperature at nominal conditions (in °C or °F). This field must be specified for both the performance input methods. The design entering water temperature must be greater than the design entering air temperature.

Design entering air temperature

This numeric field contains entering air dry-bulb temperature at nominal conditions (in °C or °F). This field must be specified for both the performance input methods. The design entering air temperature must be greater than the design entering air wet-bulb temperature.

Design entering air wet-bulb temperature

This numeric field contains entering air wet-bulb temperature at nominal conditions (in °C or °F). This field must be specified for both the performance input methods. The design entering air wet-bulb temperature must be less than the design entering air (dry-bulb) temperature.

Design water flow rate

This numeric field contains the design water flow rate through the fluid cooler (in m3/s or gal/min). This value is the flow rate of the condenser loop water being cooled by the fluid cooler. A water flow rate greater than zero must be defined or the field can be autosized. If autosized the sizing data on the Plant loop dialog is used

and the design water flow rate is derived from the design load to be rejected by the condenser loop and the design loop delta T.

Fan Settings

Performance input method

Select the method to be used to specify fluid cooler performance:

- **1-Nominal-capacity**, or,
- **2-UA and design water flow rate**

Single-speed fan

Design air flow rate

This numeric field contains the design air flow rate induced by the fluid cooler fan (in m³/s or ft³/min). A value greater than zero must be defined regardless of the fluid cooler performance input method. Alternately, this field can be autosized.

Design air flow rate fan power

This numeric field contains the fan power (in W) at the design air flow rate specified in the previous field. A value greater than zero must be specified regardless of the performance input method, or this field can be autosized. See Engineering ref. for fluid cooler autosizing.

Two-Speed Fan

High Speed Fan Settings

High fan speed air flow rate

This numeric field contains the fluid cooler air flow rate at high fan speed (in m³/s or ft³/min). A value greater than zero must be defined regardless of the fluid cooler performance input method. Alternately, this field can be autosized. See Engineering ref. for fluid cooler autosizing.

High fan speed fan power

This numeric field contains the fan power (in W) at the high-speed air flow rate specified in the previous field. A value greater than zero must be specified regardless of the fluid cooler performance input method, or this field can be autosized. See Engineering ref. for fluid cooler autosizing.

High speed nominal capacity

This numeric input field contains the “nominal” heat rejection capacity of the fluid cooler (in W or Btu/h) under high-speed fan operation, with nominal (design) inputs entering water temperature, entering air temperature and entering air wet-bulb temperature. The design water flow rate is also provided in the input.

High fan speed U-factor times area value

This numeric field contains the heat transfer coefficient-area product (UA) (in W/K or Btu/h-F) corresponding to the high-speed air flow rate and design water flow rate specified above. If the **Performance input method** is specified as **2-UA and design water flow rate** then a UA value greater than zero but less than or equal to 300,000 must be defined, or the field can be autosized. If autosized the sizing data from the Plant loop is used and the fluid cooler UA value at high fan speed is derived from the design load to be rejected by the condenser loop and the design loop delta T. Fluid cooler water inlet temperature, fluid cooler inlet air drybulb and wetbulb are provided in the input. If **Performance input method** is specified as **1-Nominal capacity**, then this field is not available since the model automatically calculates the fluid cooler UA based on the [High speed nominal capacity](#) and nominal conditions specified below.

Low Speed Fan Settings

Low fan speed air flow rate

This numeric field contains the fluid cooler air flow rate at low fan speed (in m³/s or ft³/min). This value must be greater than zero, less than the value specified for the field “Air Flow Rate at High Fan Speed”. This field may be autocalculated, in which case it is set to a fraction of the “Air Flow Rate at High Fan Speed” determined in the following field.

Low fan speed air flow rate sizing factor

This numeric field contains the sizing factor to use when calculating the Low Fan Speed Air Flow Rate. The default is 0.5.

Low fan speed fan power

This numeric field contains the fan power (in W) at the low-speed air flow rate specified in the previous field. This value must be specified greater than zero or the field may be autocalculated, in which case it is set to set to a fraction of the [High speed fan power](#).

Low fan speed fan power sizing factor

This numeric field contains the sizing factor to use when calculating the Low fan speed fan power (above). The default is 0.16.

Low speed nominal capacity

This numeric input field contains the “nominal” heat rejection capacity of the Fluid cooler (in W or Btu/h) under low-speed fan operation, with nominal (design) inputs entering water temperature, entering air temperature and entering air wet-bulb temperature. The design water flow rate is also provided in the input. The value specified for this field must be greater than zero but less than the value specified for the field [High speed nominal capacity](#). This field may be autocalculated, in which case it is set to a fraction of the [High speed nominal capacity](#).

Low speed nominal capacity sizing factor

This numeric field contains the sizing factor to use when calculating the [Low speed nominal capacity](#). The default is 0.5.

Low fan speed U-factor times area value

This numeric field contains the heat transfer coefficient-area product (UA) (in W/K or Btu/h-F) corresponding to the design water flow rate and low-speed air flow rate specified above. If the **Performance input method** is specified as **2-UA and design water flow rate**, this value must be greater than zero but less than or equal to 300,000, less than the value specified for [High fan speed U-factor times area value](#). This field may be autocalculated, in which case it is set to a factor of the [High fan speed U-factor times area value](#). If **Performance input method** is specified as **1-Nominal capacity**, then this field is not available since the model automatically calculates the fluid cooler UA based on the [Low speed nominal capacity](#) and nominal conditions specified below.

Low fan speed U-factor times area sizing factor

This numeric field contains the sizing factor to use when calculating the Low Fan Speed U-factor Times Area Value. The default is 0.6.

Note: There are some restrictions with autosizing Fluid coolers with E+ v8.3 and v8.4. For more information on this please contact the DesignBuilder Support Desk.

Pumps

These pump types are available within Detailed HVAC:

- [Pump - Constant Speed](#)
- [Pump - Variable Speed](#)

See the [Editing HVAC loop and component properties](#) tutorial

Pump - Constant Speed



Pump:ConstantSpeed

Used in:

Plant and Condenser loops

General

Name

The auto-generated name for the pump.

Type

The type of the variable speed pump is fixed as:

- **2-Constant speed**

To change from constant speed to variable speed you must open the parent Plant or Condenser loop dialog and change the **Plant loop flow type** to **1-Variable flow**.

Pump Settings

Design maximum flow rate (hidden)

This numeric field contains the pump's rated volumetric flow rate (in m³/s or gal/min). This setting comes from the Plant loop [Maximum loop flow rate](#) and cannot be accessed.

Design power consumption

This numeric field contains the pump's rated power consumption (in W). If the user is performing a pressure simulation on the loop in which this pump is found, this value would only be used to estimate pump efficiency.

During reported calculations, the pump would use the loop pressure drop and current flow conditions along with efficiency to calculate pump power dynamically.

Design pump head

This numeric field contains the pump's rated head pressure (in Pascals or ftH₂O).

Important Note: The default pump head provided by the Plant and Condenser Loop templates is **20,000 Pa**. This may be suitable for a small building, but it is important that you enter the correctly sized pump head if you wish to simulate pump energy consumption accurately in your model.

Motor efficiency

This numeric field contains the pump's motor efficiency in decimal form (0 = 0%, 1 = 100%).

Fraction of motor inefficiencies to fluid stream

This is the pump's fraction of power loss to the fluid.

Pump control type

This is a choice field of

- **1-Continuous** or
- **2-Intermittent.**

The operation of a constant speed pump is fairly straightforward. If the user designates a constant speed pump that is operating continuously, the pump will run regardless of whether or not there is a load. This may have the net effect of adding heat to the loop if no equipment is turned on. If the pump is constant speed and operates intermittently, the pump will run at its capacity if a load is sensed and will shut off if there is no load on the loop.

Pump - Constant Speed Outputs

HVAC,Average,Pump Electric Power [W]
 HVAC,Sum,Pump Electric Consumption [J]
 HVAC,Average,Pump Shaft Power [W]
 HVAC,Average,Pump Heat To Fluid [W]
 HVAC,Sum,Pump Heat To Fluid Energy [J]
 HVAC,Average,Pump Outlet Temp [C]
 HVAC,Average,Pump Mass Flow Rate [kg/s]

Pump Electric Power [W]

Pump Electric Consumption [J]

These outputs are the electric power input to the pump motor. Consumption is metered on Pumps:Electricity, Electricity:Plant, and Electricity:Facility.

Pump Shaft Power [W]

This is the shaft power delivered from the motor to the pump.

Pump Heat To Fluid [W]

Pump Heat To Fluid Energy [J]

These outputs are the energy added to the fluid as heat. For the current algorithm, this is equal to Pump Shaft Power, because the loops are closed and all energy added to the fluid will ultimately become heat due to friction.

Pump Outlet Temp [C]

Pump Mass Flow Rate [kg/s]

These outputs are the water outlet temperature and mass flow rate.

Pump - Variable Speed



Pump:VariableSpeed

Used in:

Plant and Condenser loops

General

Name

The auto-generated name for the pump.

Type

The type of the variable speed pump is fixed as:

- **2-Variable speed**

To change from variable speed to constant speed you must open the parent Plant or Condenser loop dialog and change the **Plant loop flow type** to **1-Constant flow**.

Pump Settings

Design power consumption

This numeric field contains the pump's rated power consumption (in W). If the user is performing a pressure simulation on the loop in which this pump is found, this value would only be used to estimate pump efficiency. During reported calculations, the pump would use the loop pressure drop and current flow conditions along with efficiency to calculate pump power dynamically.

Design pump head

This numeric field contains the pump's rated head pressure (in Pascals or ftH₂O).

Important Note: The default pump head provided by the Plant and Condenser Loop templates is **20,000 Pa**. This may be suitable for a small building, but it is important that you enter the correctly sized pump head if you wish to simulate pump energy consumption accurately in your model.

Design maximum flow rate (hidden)

This numeric field contains the pump's rated volumetric flow rate (in m³/s or gal/min). This setting comes from the Plant loop [Maximum loop flow rate](#) and cannot be accessed.

Design minimum flow rate

This field contains the minimum volumetric flow rate while operating in variable flow capacity rate (in m³/s or gal/min).

Motor efficiency

This numeric field contains the pump's motor efficiency in decimal form (0 = 0%, 1 = 100%).

Fraction of motor inefficiencies to fluid stream

This numeric field contains the pump's fraction of power loss to the fluid.

Pump control type

This is a choice field of:

- **1-Continuous** or
- **2-Intermittent.**

A variable speed pump is defined with maximum and minimum flow rates that are the physical limits of the device. If there is no load on the loop and the pump is operating intermittently, then the pump can shutdown. For any other condition such as the loop having a load and the pump is operating intermittently or the pump is continuously operating (regardless of the loading condition), the pump will operate and select a flow somewhere between the minimum and maximum limits. In these cases where the pump is running, it will try to meet the flow request made by demand side components.

Part Load Performance

Pump curve template

Use the pump curve template to load some pre-defined pump coefficients to the dialog.

Pump Coefficients

The fraction of full load power is determined during the simulation by the cubic equation:

$$\text{FractionFullLoadPower} = C_1 + C_2 \cdot \text{PLR} + C_3 \cdot \text{PLR}^2 + C_4 \cdot \text{PLR}^3$$

where C_1 to C_4 are the coefficients described below and PLR is the Part Load Ratio.

Coefficient 1 of the part load performance curve

This numeric field contains the first coefficient, C_1 , in the part load ratio curve.

Coefficient 2 of the part load performance curve

This numeric field contains the second coefficient, C_2 , in the part load ratio curve.

Coefficient 3 of the part load performance curve

This numeric field contains the third coefficient, C_3 , in the part load ratio curve.

Coefficient 4 of the part load performance curve

This numeric field contains the fourth coefficient, C_4 , in the part load ratio curve.

Pump - Variable Speed Outputs

HVAC,Average,Pump Electric Power [W]
HVAC,Sum,Pump Electric Consumption [J]
HVAC,Average,Pump Shaft Power [W]
HVAC,Average,Pump Heat To Fluid [W]
HVAC,Sum,Pump Heat To Fluid Energy [J]
HVAC,Average,Pump Outlet Temp [C]
HVAC,Average,Pump Mass Flow Rate [kg/s]

Pump Electric Power [W] Pump Electric Consumption [J]

These outputs are the electric power input to the pump motor. Consumption is metered on Pumps:Electricity, Electricity:Plant, and Electricity:Facility.

Pump Shaft Power [W]

This is the shaft power delivered from the motor to the pump.

Pump Heat To Fluid [W]

Pump Heat To Fluid Energy [J]

These outputs are the energy added to the fluid as heat. For the current algorithm, this is equal to Pump Shaft Power, because the loops are closed and all energy added to the fluid will ultimately become heat due to friction.

Pump Outlet Temp [C]

Pump Mass Flow Rate [kg/s]

These outputs are the water outlet temperature and mass flow rate.

Ground Heat Exchanger

DesignBuilder provides 3 types of ground heat exchanger:

- [1-Vertical](#) (boreholes),
- [2-Pond](#),
- [3-Surface](#).

You can select the ground heat exchanger type from the Type drop list on the Ground heat exchanger dialog.

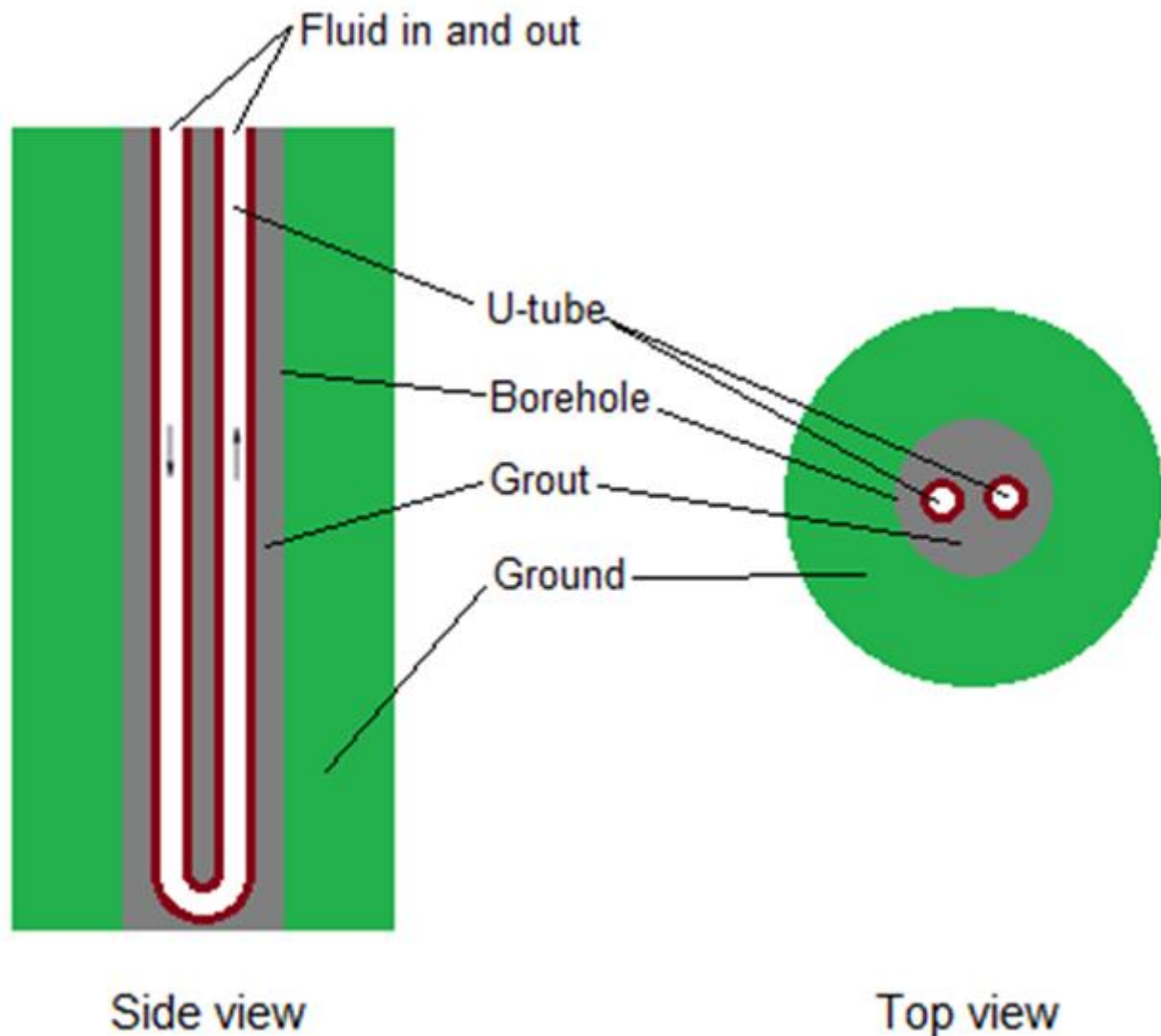
Autosizing

The Ground Heat Exchanger components do not offer an autosize option and so flow rates and other sizing information must be obtained using the method described in [Step 6 of the GSHP Case Study](#). An inaccurately sized heat exchanger component can cause GSHP systems to generate inaccurate results.

Tip: You are advised to work through the [GSHP Case Study](#) before attempting to model these systems for the first time.

Ground Heat Exchanger - Vertical

Vertical ground heat exchangers typically use boreholes containing U-tubes as shown in the figure below.



The EnergyPlus Ground Heat Exchanger (GHE) is a condenser component serving the condenser supply side alongside cooling towers and other condensing components.

Technical

The heat exchanger response is defined by a G-function. This is a non-dimensional function that is used to calculate the response to square heat pulses of different duration. (This function is not the same as 'G-factors' referred to in the ASHRAE Applications Handbook).

This continuous function is specified by a series of data pairs (LNTTSi, GFNCi) where:

- LNTTSi is the non-dimensional time: $\ln(T/T_s)$
- GFNCi is the G-function value

The G-function is different for each borehole field configuration (i.e. a 4x4 field has a different response than a 80x80 field) and the borehole thermal resistance. It is also dependant on the ratio of borehole spacing to depth. G-function values, for accurate simulation, have to be calculated for each specific heat exchanger design. This can be done using some commercial ground loop heat exchanger design tool and the like. A reference data set, containing examples input data for 1x2, 4x4 and 8x8 configurations and for both standard and thermally enhanced grout, have also been provided. These data are provided as examples only.

Further details on the implementation of this model can be found in:

General

Name

This alpha field contains an editable identifying name for the ground heat exchanger.

Ground Heat Exchanger Template

Template

You can use this control to load data to the dialog from a pre-defined Ground heat exchanger template as a starting point for your particular component.

Note: After loading a Ground heat exchanger template it is important to make sure to update the [Design](#) heat exchanger flow rates as the sum of connected heat pump rated flow rates.

Ground Heat Exchanger Type

Type

DesignBuilder provides 3 different ground heat exchanger types:

- [1-Vertical](#) (boreholes)
- [2-Pond](#)
- [3-Surface](#)

Note: The [Type](#) drop down list is not editable. To select a different type of heat exchanger, you must first select an appropriate template (above).

Ground Properties

Ground thermal conductivity

This numeric field contains the thermal conductivity of the ground (in W/m-K or Btu-in/h-ft²-F).

Ground thermal heat capacity

This numeric field contains the thermal heat capacity of the ground (in J/m³-K or Btu/ft³-F).

Ground temperature

This numeric field contains the far field temperature of the ground (in °C or °F).

Flow Rate

Design flow rate

This numeric field contains the design volume flow rate of the GHE (in m³/s or gal/min). The flow rate specified is the total flow rate for the entire borehole field. Flow will be assumed to be evenly distributed across all boreholes.

Borehole And Pipe Geometry

Number of boreholes

This numeric field contains the number of bore holes in the GHE installation.

Depth of top of borehole

This numeric field indicates the depth of the top of the borehole below the ground surface (in m or ft). The depth measured from the ground surface downward is positive.

Borehole length

This numeric field contains the active length of the borehole (in m or ft), referenced from the starting location (potentially below the ground surface), to the end of the borehole.

Borehole diameter

This numeric field contains the diameter of the borehole (in m or in).

Pipe outer diameter

This numeric field contains the outer diameter of the U-tube (pipe) (in m or in).

Pipe thickness

This numeric field contains the thickness of the pipe (in m or in).

U-Tube distance

This numeric field contains the distance between the two legs of the U-tube (in m or ft). Distance is measured from the U-tube pipe centre.

Grout And Pipe Properties

Grout thermal conductivity

This numeric field contains the thermal conductivity of the filler material (in W/m-K or Btu-in/h-ft²-F).

Grout thermal heat capacity

This numeric field contains the thermal heat capacity of the grout (in J/m³-K or Btu/ft³-F).

Pipe thermal conductivity

This numeric field contains the thermal conductivity of the pipe in (in W/mK or Btu-in/h-ft²-F).

Pipe thermal heat capacity

This numeric field contains the thermal heat capacity of the pipe (in J/m³-K or Btu/ft³-F).

Other

G-Function reference ratio

The G-Functions may be formulated slightly differently based on the program which generated them. The "raw" G-Functions are based on an borehole radius to active length ratio of 0.0005. If the physical ratio is different from this, a correction must be applied. EnergyPlus will apply the correction, based on the reference ratio entered in this field. There are therefore two possible input configurations:

- If the G-Functions have not had a correction applied, then the G-Functions are still based on a reference of 0.0005, so use a value of 0.0005 in this field. EnergyPlus will adjust the G-Functions internally to create the properly referenced G-Function.
- If the correction has already been applied, then the input G-Functions are based on a reference to the actual (physical) radius/length ratio, so enter the physical radius/length in this field. Entering the actual value will nullify any internal corrections, which will avoid re-basing the G-Function set.

The software [GLHEPRO](#) has been making this “pre-correction” to the data sets since version 3.1 of that software, so this input field should match the actual (physical) radius/length ratio.

G Function Data Pairs

The borehole response is defined by a non-dimensional ‘G-function’ and the second tab includes a large text box with a list of G-function data pairs. The list is specified as a series of data points giving values of non-dimensional time vs G-function value (LNTTS1, GFUNC1), (LNTTS2, GFUNC2), (LNTTS3, GFUNC3) (LNTTSn, GFUNCn)..

G-Function $\ln(T/T_s)$ value <x>

This numeric field contains the natural log of time/steady state time: $\ln(T/T_s)$.

G-Function 'G' value <x>

This numeric field contains the G-function value of the corresponding LNTTS.

Example data:

Ground heat exchanger Data	
General	G Function Data
G-Function Data	
-15.795833, -0.868005,	
-15.646939, -0.841532,	
-15.498044, -0.813862,	
-15.349150, -0.784824,	
-15.200255, -0.754311,	
-15.051361, -0.722260,	
-14.902466, -0.688635,	
-14.753571, -0.653424,	
-14.604677, -0.616633,	
-14.455782, -0.578288,	
-14.306888, -0.538426,	
-14.157993, -0.497093,	
-14.009099, -0.454339,	
-13.860204, -0.410206,	
-13.711310, -0.364726,	
-13.562415, -0.317904,	
-13.413520, -0.269714,	
-13.264626, -0.220096,	
-13.115731, -0.168953,	
-12.966837, -0.116163,	
-12.817942, -0.061585,	
-12.669048, -0.005079,	
-12.520153, 0.053480,	
-12.371259, 0.114187,	
-12.222364, 0.177098,	
-12.073469, 0.242221,	
-11.924575, 0.309516,	
-11.775680, 0.378895,	
-11.626786, 0.450230,	
-11.477891, 0.523360,	

Sizing Vertical Ground Heat Exchangers

You can learn more about practical aspects of Ground Heat Exchanger design and simulation for project work by watching our webinar: [Design and Modelling of Ground Source Heat Pump Systems](#)

Quick/Simple Sizing

DesignBuilder provides a quick, approximate way to size vertical ground loop heat exchangers assuming typical borehole layouts and ground properties. The procedure is to use DesignBuilder [Heating design](#) and [Cooling design](#) calculations to obtain peak heating and cooling loads in kW, then use the table below to look up the minimum number of boreholes for the model.

		Peak cooling load (kW)									
		10	20	30	40	50	60	70	80	90	100
Peak heating load (kW)	10	4	4	8	8	8	12	12	12	16	16
	20	8	8	8	12	12	12	12	12	16	16
	30	12	12	12	12	16	16	16	16	16	16
	40	16	16	16	16	16	24	24	24	24	24
	50	24	24	24	24	24	24	24	24	24	24
	60	24	24	24	24	24	24	24	24	24	40
	70	40	40	40	40	40	40	40	40	40	40
	80	40	40	40	40	40	40	40	40	40	40
	90	40	40	40	40	40	40	40	40	40	40
	100	40	40	40	40	40	40	40	40	40	40

Then load the "U-Tube 76m xx-boreholes" vertical ground heat exchanger [template](#) with the corresponding number of boreholes from the template library. These templates include parameters pre-calculated based on a typical rectangular geometry and property settings (see left red-box in the figure below). Each includes a corresponding set of g-function data. For very different a borehole layouts or when any of the ground or other parameter deviates significantly from these typical settings then you should use the [Detailed design sizing](#) method described below.

Edit Ground heat exchanger -

Ground heat exchanger Data

General

G Function Data

General

Name

Ground Heat Exchanger

Template

U-tube 76m 120-boreholes

Type

1-Vertical

Ground heat exchanger - vertical

Ground temperature (°C)

13.375

Flow rate

Design flow rate (m3/s)

0.00330

Maximum flow rate (m3/s)

0.00330

Borehole and pipe geometry

Number of boreholes

120

Borehole length (m)

76.0

Borehole radius (m)

0.063508

Pipe outer diameter (m)

0.0266667

U-tube distance (m)

0.0253977

Pipe thickness (m)

0.00241285

Ground, grout and pipe properties

Ground thermal conductivity (W/m-K)

0.692626

Ground thermal heat capacity (J/m3-K)

2347000.00

Grout thermal conductivity (W/m-K)

0.692626

Pipe thermal conductivity (W/m-K)

0.391312

Others

Maximum length of simulation

1

G-function reference ratio

0.000836

Help

Info

Data

Ground heat exchanger template

Surface ground heat exchanger

Vertical ground heat exchanger

U-tube 76m 120-boreholes

U-tube 76m 12-boreholes

U-tube 76m 16-boreholes

U-tube 76m 240-boreholes

U-tube 76m 24-boreholes

U-tube 76m 40-boreholes

U-tube 76m 4-boreholes

U-tube 76m 60-boreholes

U-tube 76m 80-boreholes

Data Report (Not Editable)

The records shown in above list are all based on some typical geometry and property settings indicated at the left. Please do not alter any of these figures after loading a record.

If other geometries or properties are required, the ground loop heat exchanger tools (not supplied by DesignBuilder) must be used to incorporate these changes and create a new record with updated g-function data.

Model data <admin>

Import

Help

Cancel

OK

Detailed design sizing

The net heating or cooling capacity of the ground over each season depends on the accumulated heat rejection/extraction, and therefore on the building loads throughout the whole year (not only the peak loads mentioned above), which might be available through simulation perhaps based on a simple HVAC setting in DesignBuilder.

Ground loop heat exchanger tools such as [GLHEPRO](#) and [GLD](#) (which are not supplied by DesignBuilder or EnergyPlus) are able to carry out a more detailed and accurate sizing calculation taking into account specific borehole layouts and ground properties. These tools can export calculated G-function data in the form of IDF files which can be loaded to the DesignBuilder Ground Heat Exchanger dialog as described below.

Import Vertical Ground Heat Exchanger Data From IDF

Data from a previously configured and sized vertical ground loop heat exchanger can be loaded by clicking on the **Import** button at the bottom of the dialog or the **Import vertical ground heat exchanger data** link from info panel (see figure below).

Edit Ground heat exchanger -

Ground heat exchanger Data

General | G Function Data

General

Name: Ground Heat Exchanger

Template: U-tube 76m 40-boreholes

Type: 1-Vertical

Ground heat exchanger - vertical

Ground temperature (°C): 13.375

Flow rate

Design flow rate (m3/s): 0.00450

Maximum flow rate (m3/s): 0.00450

Borehole and pipe geometry

Number of boreholes: 40

Borehole length (m): 76.0

Borehole radius (m): 0.063508

Pipe outer diameter (m): 0.0266667

U-tube distance (m): 0.0253977

Pipe thickness (m): 0.00241285

Ground, grout and pipe properties

Ground thermal conductivity (W/m-K): 0.692626

Ground thermal heat capacity (J/m3-K): 2347000.00

Grout thermal conductivity (W/m-K): 0.692626

Pipe thermal conductivity (W/m-K): 0.391312

Others

Maximum length of simulation: 1

G-function reference ratio: 0.0005

Help

Info | Data

Ground heat exchanger

Quick/Simple sizing

Generally, there is a quick/simple way in doing ground loop heat exchanger sizing. The procedure is to use DesignBuilder 'Heating Design' and 'Cooling Design' calculations to obtain peak heating and cooling loads in kW, then look at the table below to get a minimum borehole number for the model.

		Peak cooling load (kW)									
		10	20	30	40	50	60	70	80	90	100
Peak heating load (kW)	10	4	4	8	8	8	12	12	12	16	16
	20	8	8	8	12	12	12	12	16	16	16
	30	12	12	12	12	16	16	16	16	16	16
	40	16	16	16	16	16	24	24	24	24	24
	50	24	24	24	24	24	24	24	24	24	24
	60	24	24	24	24	24	24	24	24	24	40
	70	40	40	40	40	40	40	40	40	40	40
	80	40	40	40	40	40	40	40	40	40	40
	90	40	40	40	40	40	40	40	40	40	40
	100	40	40	40	40	40	40	40	40	40	40

Recommended flow rate for ground heat exchanger is the maximum of sum of rated flow rates from all connected heat pumps for heating or heat pumps for cooling.

Detailed/Precise sizing

The net heating or cooling of the ground over each season clearly depends on the accumulated heat rejection/extraction, and therefore on the building loads throughout the whole year (not only the peak loads mentioned above), which might be available through simulation based on a very simple HVAC setting in DesignBuilder.

The solution for above relies on using other ground loop heat exchanger tools (not supplied by DesignBuilder or EnergyPlus), which are capable of doing detailed/precise sizing.

Import vertical ground heat exchanger data

When using vertical ground heat exchanger, you can load settings and g-functions from idf files stored on your local drive.

[Import vertical ground heat exchanger data](#)

Model data <admin> | Import | Help | Cancel | OK

The corresponding g-function data will be loaded along with the rest of the data required on the dialog and can be viewed on the [G Function Data](#) tab.

Supported GHE IDF File Formats

DesignBuilder supports 4 different IDF GHE file formats which correspond to the versions of EnergyPlus targeted by the various versions of GLE and GLEHEPRO. The importer automatically identifies which of the formats below is being loaded and uses the appropriate [IDD file](#) for the import.

Tip: If the format of the GHE IDF file you are attempting to import is different from those illustrated below then you can manually adjust the format of the data in a text editor so it matches one of the supported formats.

Examples of the 4 supported GHE IDF formats are shown below.

EnergyPlus v6 and earlier format

This is the IDF format exported by GLHEPRO **version x**.

```
GroundHeatExchanger:Vertical,
Hybridlayout,           !- Ground Heat Exchanger Name
GHE Inlet Node,         !- GHE Inlet Node
GHE Outlet Node,        !- GHE Outlet Node
0.110517,               !- Max GHE FlowRate {m3/s}
190,                   !- Number of Boreholes
152.40,                !- Bore Hole Length {m}
0.07150,               !- Bore Hole Radius {m}
2.527,                 !- Thermal conductivity (k) of Ground {W/m-K}
2.1749E+06,            !- Density x Specific Heat (rhoCp) of Ground {J/m^3-K}
4.1840E+03,            !- Specific Heat of Fluid {J/kg-K}
12.8,                 !- Undisturbed temperature of Ground {C}
0.110517,             !- Design GHE Vol Flowrate {m3/s}
1.731,                 !- Thermal conductivity (k) of Grout {W/m-K}
0.389,                 !- Thermal conductivity (k) of Pipe {W/m-K}
0.609,                 !- Thermal conductivity (k) of Fluid {W/m-K}
999.552,               !- Density of Fluid (rho) {Kg/m^3}
0.001,                 !- Dynamic viscosity of Fluid (mu) {N-s/m/62}
0.04216,               !- Pipe Outside Diameter {m}
0.002,                 !- U-Tube Separation (Shank spacing) {m}
0.00940,               !- Pipe Wall Thickness {m}
40,                    !- Maximum length of simulation in years
41,                    !- Number of data pairs of the G function
! The G-function is defined by the following data pairs
-15.00, -0.528595,     !- LNTTS/GFNC Pair 1
-14.50, -0.278595,     !- LNTTS/GFNC Pair 2
-14.00, -0.028595,     !- LNTTS/GFNC Pair 3
-13.50, 0.221405,      !- LNTTS/GFNC Pair 4
-13.00, 0.471405,      !- LNTTS/GFNC Pair 5
-12.50, 0.721405,      !- LNTTS/GFNC Pair 6
-12.00, 0.971405,      !- LNTTS/GFNC Pair 7
-11.50, 1.221405,      !- LNTTS/GFNC Pair 8
-11.00, 1.471405,      !- LNTTS/GFNC Pair 9
-10.50, 1.721405,      !- LNTTS/GFNC Pair 10
-10.00, 1.971405,      !- LNTTS/GFNC Pair 11
-9.50, 2.221405,       !- LNTTS/GFNC Pair 12
-9.00, 2.471405,       !- LNTTS/GFNC Pair 13
-8.50, 2.721405,       !- LNTTS/GFNC Pair 14
-8.00, 2.971405,       !- LNTTS/GFNC Pair 15
-7.50, 3.221405,       !- LNTTS/GFNC Pair 16
-7.00, 3.471405,       !- LNTTS/GFNC Pair 17
-6.50, 3.721405,       !- LNTTS/GFNC Pair 18
-6.00, 3.971405,       !- LNTTS/GFNC Pair 19
-5.50, 4.221405,       !- LNTTS/GFNC Pair 20
-5.00, 4.471405,       !- LNTTS/GFNC Pair 21
-4.50, 4.721405,       !- LNTTS/GFNC Pair 22
-4.00, 6.919368,       !- LNTTS/GFNC Pair 23
-3.50, 9.572881,       !- LNTTS/GFNC Pair 24
-3.00, 11.609723,      !- LNTTS/GFNC Pair 25
-2.00, 25.852480,      !- LNTTS/GFNC Pair 27
-1.50, 38.110427,      !- LNTTS/GFNC Pair 28
-1.00, 58.801776,      !- LNTTS/GFNC Pair 29
-0.50, 100.224494,     !- LNTTS/GFNC Pair 30
0.00, 123.801539,      !- LNTTS/GFNC Pair 31
0.50, 144.480057,      !- LNTTS/GFNC Pair 32
1.00, 157.989392,      !- LNTTS/GFNC Pair 33
1.50, 160.843185,      !- LNTTS/GFNC Pair 34
2.00, 161.117193,      !- LNTTS/GFNC Pair 35
2.50, 161.234525,      !- LNTTS/GFNC Pair 36
3.00, 161.289948,      !- LNTTS/GFNC Pair 37
3.50, 161.316129,      !- LNTTS/GFNC Pair 38
4.00, 161.328495,      !- LNTTS/GFNC Pair 39
4.50, 161.334337,      !- LNTTS/GFNC Pair 40
5.00, 161.337096,      !- LNTTS/GFNC Pair 41
```


EnergyPlus v7 - v8.2 format

This is the IDF format exported by GLHEPRO **versions x-y and GLD versions a-b.**

```

GroundHeatExchanger:Vertical,
Vertical GHE,                !- Name
GHE Inlet Node,             !- Inlet Node Name
GHE Outlet Node,           !- Outlet Node Name
0.015140,                   !- Maximum Flow Rate {m3/s}
40,                         !- Number of Boreholes
151.05,                    !- Bore Hole Length {m}
0.07144,                   !- Bore Hole Radius {m}
2.423,                     !- Ground Thermal Conductivity {W/m-K}
2.343E+06,                 !- Ground Thermal Heat Capacity {J/m3-K}
13.6,                      !- Ground Temperature {C}
0.015140,                  !- Design Flow Rate {m3/s}
0.744,                    !- Grout Thermal Conductivity {W/m-K}
0.389,                    !- Pipe Thermal Conductivity {W/m-K}
0.04826,                  !- Pipe Out Diameter {m}
0.0154,                   !- U-Tube Distance {m}
0.00440,                  !- Pipe Thickness {m}
100,                      !- Maximum Length of Simulation {years}
0.00047,                  !- G-Function Reference Ratio
76,                       !- Number of Data Pairs of the G Function
-15.915942,               !- G-Function Ln(T/Ts) Value 1
-2.235646,               !- G-Function G Value 1
-15.764596,              !- G-Function Ln(T/Ts) Value 2
-2.190303,               !- G-Function G Value 2
-15.613251,              !- G-Function Ln(T/Ts) Value 3
-2.140816,               !- G-Function G Value 3
-15.461905,              !- G-Function Ln(T/Ts) Value 4
-2.086786,               !- G-Function G Value 4
-15.310559,              !- G-Function Ln(T/Ts) Value 5
-2.027848,               !- G-Function G Value 5
-15.159213,              !- G-Function Ln(T/Ts) Value 6
-1.963669,               !- G-Function G Value 6
-15.007868,              !- G-Function Ln(T/Ts) Value 7
-1.893939,               !- G-Function G Value 7
-14.856522,              !- G-Function Ln(T/Ts) Value 8
-1.818384,               !- G-Function G Value 8
-14.705176,              !- G-Function Ln(T/Ts) Value 9
-1.736763,               !- G-Function G Value 9
-14.553830,              !- G-Function Ln(T/Ts) Value 10
-1.648880,               !- G-Function G Value 10
-14.402485,              !- G-Function Ln(T/Ts) Value 11
-1.554591,               !- G-Function G Value 11
-14.251139,              !- G-Function Ln(T/Ts) Value 12
-1.453815,               !- G-Function G Value 12
-14.099793,              !- G-Function Ln(T/Ts) Value 13
-1.346547,               !- G-Function G Value 13
-13.948447,              !- G-Function Ln(T/Ts) Value 14
-1.232875,               !- G-Function G Value 14
-13.797102,              !- G-Function Ln(T/Ts) Value 15
-1.112999,               !- G-Function G Value 15
-13.645756,              !- G-Function Ln(T/Ts) Value 16
-0.987253,               !- G-Function G Value 16
-13.494410,              !- G-Function Ln(T/Ts) Value 17
-0.856123,               !- G-Function G Value 17
-13.343064,              !- G-Function Ln(T/Ts) Value 18
-0.720261,               !- G-Function G Value 18
-13.191718,              !- G-Function Ln(T/Ts) Value 19
-0.580488,               !- G-Function G Value 19
-13.040373,              !- G-Function Ln(T/Ts) Value 20
-0.437778,               !- G-Function G Value 20
-12.889027,              !- G-Function Ln(T/Ts) Value 21
-0.293237,               !- G-Function G Value 21
-12.737681,              !- G-Function Ln(T/Ts) Value 22
-0.148052,               !- G-Function G Value 22
-12.586335,              !- G-Function Ln(T/Ts) Value 23
-0.003435,               !- G-Function G Value 23
-12.434990,              !- G-Function Ln(T/Ts) Value 24
0.139441,                !- G-Function G Value 24
-12.283644,              !- G-Function Ln(T/Ts) Value 25
0.279510,                !- G-Function G Value 25
-12.132298,              !- G-Function Ln(T/Ts) Value 26
0.415870,                !- G-Function G Value 26
-11.980952,              !- G-Function Ln(T/Ts) Value 27
0.547830,                !- G-Function G Value 27
-11.829607,              !- G-Function Ln(T/Ts) Value 28

```

0.674932,	!- G-Function G Value 28
-11.678261,	!- G-Function Ln(T/Ts) Value 29
0.796954,	!- G-Function G Value 29
-11.526915,	!- G-Function Ln(T/Ts) Value 30
0.913888,	!- G-Function G Value 30
-11.375569,	!- G-Function Ln(T/Ts) Value 31
1.025913,	!- G-Function G Value 31
-11.224224,	!- G-Function Ln(T/Ts) Value 32
1.133338,	!- G-Function G Value 32
-11.072878,	!- G-Function Ln(T/Ts) Value 33
1.236565,	!- G-Function G Value 33
-10.921532,	!- G-Function Ln(T/Ts) Value 34
1.336037,	!- G-Function G Value 34
-10.770186,	!- G-Function Ln(T/Ts) Value 35
1.432206,	!- G-Function G Value 35
-10.618841,	!- G-Function Ln(T/Ts) Value 36
1.525504,	!- G-Function G Value 36
-10.467495,	!- G-Function Ln(T/Ts) Value 37
1.616327,	!- G-Function G Value 37
-10.316149,	!- G-Function Ln(T/Ts) Value 38
1.705029,	!- G-Function G Value 38
-10.164803,	!- G-Function Ln(T/Ts) Value 39
1.791917,	!- G-Function G Value 39
-10.013458,	!- G-Function Ln(T/Ts) Value 40
1.877256,	!- G-Function G Value 40
-9.862112,	!- G-Function Ln(T/Ts) Value 41
1.961270,	!- G-Function G Value 41
-9.710766,	!- G-Function Ln(T/Ts) Value 42
2.044152,	!- G-Function G Value 42
-9.559420,	!- G-Function Ln(T/Ts) Value 43
2.126062,	!- G-Function G Value 43
-9.408075,	!- G-Function Ln(T/Ts) Value 44
2.207138,	!- G-Function G Value 44
-9.256729,	!- G-Function Ln(T/Ts) Value 45
2.287496,	!- G-Function G Value 45
-9.105383,	!- G-Function Ln(T/Ts) Value 46
2.367236,	!- G-Function G Value 46
-8.954037,	!- G-Function Ln(T/Ts) Value 47
2.446441,	!- G-Function G Value 47
-8.802692,	!- G-Function Ln(T/Ts) Value 48
2.525183,	!- G-Function G Value 48
-8.651346,	!- G-Function Ln(T/Ts) Value 49
2.603526,	!- G-Function G Value 49
-8.500000,	!- G-Function Ln(T/Ts) Value 50
2.734584,	!- G-Function G Value 50
-7.800000,	!- G-Function Ln(T/Ts) Value 51
3.083496,	!- G-Function G Value 51
-7.200000,	!- G-Function Ln(T/Ts) Value 52
3.420775,	!- G-Function G Value 52
-6.500000,	!- G-Function Ln(T/Ts) Value 53
3.990871,	!- G-Function G Value 53
-5.900000,	!- G-Function Ln(T/Ts) Value 54
4.809220,	!- G-Function G Value 54
-5.200000,	!- G-Function Ln(T/Ts) Value 55
6.412973,	!- G-Function G Value 55
-4.500000,	!- G-Function Ln(T/Ts) Value 56
9.061057,	!- G-Function G Value 56
-3.963000,	!- G-Function Ln(T/Ts) Value 57
11.938320,	!- G-Function G Value 57
-3.270000,	!- G-Function Ln(T/Ts) Value 58
17.122104,	!- G-Function G Value 58
-2.864000,	!- G-Function Ln(T/Ts) Value 59
20.789758,	!- G-Function G Value 59
-2.577000,	!- G-Function Ln(T/Ts) Value 60
23.643189,	!- G-Function G Value 60
-2.171000,	!- G-Function Ln(T/Ts) Value 61
27.927561,	!- G-Function G Value 61
-1.884000,	!- G-Function Ln(T/Ts) Value 62
31.079091,	!- G-Function G Value 62
-1.191000,	!- G-Function Ln(T/Ts) Value 63
38.577931,	!- G-Function G Value 63
-0.497000,	!- G-Function Ln(T/Ts) Value 64
45.240111,	!- G-Function G Value 64
-0.274000,	!- G-Function Ln(T/Ts) Value 65
47.091768,	!- G-Function G Value 65
-0.051000,	!- G-Function Ln(T/Ts) Value 66
48.749472,	!- G-Function G Value 66
0.196000,	!- G-Function Ln(T/Ts) Value 67
50.383008,	!- G-Function G Value 67
0.419000,	!- G-Function Ln(T/Ts) Value 68

```

51.666063,      !- G-Function G Value 68
0.642000,      !- G-Function Ln(T/Ts) Value 69
52.757756,      !- G-Function G Value 69
0.873000,      !- G-Function Ln(T/Ts) Value 70
53.712552,      !- G-Function G Value 70
1.112000,      !- G-Function Ln(T/Ts) Value 71
54.520500,      !- G-Function G Value 71
1.335000,      !- G-Function Ln(T/Ts) Value 72
55.130582,      !- G-Function G Value 72
1.679000,      !- G-Function Ln(T/Ts) Value 73
55.828407,      !- G-Function G Value 73
2.028000,      !- G-Function Ln(T/Ts) Value 74
56.301658,      !- G-Function G Value 74
2.275000,      !- G-Function Ln(T/Ts) Value 75
56.531300,      !- G-Function G Value 75
3.003000,      !- G-Function Ln(T/Ts) Value 76
56.911700,      !- G-Function G Value 76

```

EnergyPlus v8.3 - v8.8 format

This is the IDF format exported by GLHEPRO **versions c-d and GLD versions e-f.**

```

GroundHeatExchanger:Vertical,
Vertical GHE,      !- Name
GHE Inlet Node,   !- Inlet Node Name
GHE Outlet Node,  !- Outlet Node Name
0.015140,         !- Design Flow Rate {m3/s}
40,              !- Number of Boreholes
122.44,          !- Bore Hole Length {m}
0.07144,         !- Bore Hole Radius {m}
2.423,           !- Ground Thermal Conductivity {W/m-K}
2.343E+06,       !- Ground Thermal Heat Capacity {J/m3-K}
13.6,            !- Ground Temperature {C}
1.731,           !- Grout Thermal Conductivity {W/m-K}
0.389,           !- Pipe Thermal Conductivity {W/m-K}
0.04216,         !- Pipe Out Diameter {m}
0.0154,          !- U-Tube Distance {m}
0.00384,         !- Pipe Thickness {m}
100,             !- Maximum Length of Simulation {years}
0.00058,         !- G-Function Reference Ratio
76,              !- Number of Data Pairs of the G Function
! The G-function is defined by the following data pairs
-15.495927,      !- G-Function Ln(T/Ts) Value 1
-1.389804,       !- G-Function G Value 1
-15.353153,      !- G-Function Ln(T/Ts) Value 2
-1.345340,       !- G-Function G Value 2
-15.210379,      !- G-Function Ln(T/Ts) Value 3
-1.298434,       !- G-Function G Value 3
-15.067605,      !- G-Function Ln(T/Ts) Value 4
-1.248831,       !- G-Function G Value 4
-14.924830,      !- G-Function Ln(T/Ts) Value 5
-1.196354,       !- G-Function G Value 5
-14.782056,      !- G-Function Ln(T/Ts) Value 6
-1.140874,       !- G-Function G Value 6
-14.639282,      !- G-Function Ln(T/Ts) Value 7
-1.082308,       !- G-Function G Value 7
-14.496508,      !- G-Function Ln(T/Ts) Value 8
-1.020601,       !- G-Function G Value 8
-14.353734,      !- G-Function Ln(T/Ts) Value 9
-0.955736,       !- G-Function G Value 9
-14.210960,      !- G-Function Ln(T/Ts) Value 10
-0.887722,       !- G-Function G Value 10
-14.068186,      !- G-Function Ln(T/Ts) Value 11
-0.816599,       !- G-Function G Value 11
-13.925412,      !- G-Function Ln(T/Ts) Value 12
-0.742438,       !- G-Function G Value 12
-13.782638,      !- G-Function Ln(T/Ts) Value 13
-0.665334,       !- G-Function G Value 13
-13.639864,      !- G-Function Ln(T/Ts) Value 14
-0.585412,       !- G-Function G Value 14
-13.497090,      !- G-Function Ln(T/Ts) Value 15
-0.502826,       !- G-Function G Value 15
-13.354316,      !- G-Function Ln(T/Ts) Value 16
-0.417760,       !- G-Function G Value 16
-13.211542,      !- G-Function Ln(T/Ts) Value 17
-0.330427,       !- G-Function G Value 17
-13.068768,      !- G-Function Ln(T/Ts) Value 18

```

-0.241071,	!- G-Function G Value 18
-12.925994,	!- G-Function Ln(T/Ts) Value 19
-0.149963,	!- G-Function G Value 19
-12.783220,	!- G-Function Ln(T/Ts) Value 20
-0.057403,	!- G-Function G Value 20
-12.640446,	!- G-Function Ln(T/Ts) Value 21
0.036293,	!- G-Function G Value 21
-12.497672,	!- G-Function Ln(T/Ts) Value 22
0.130792,	!- G-Function G Value 22
-12.354898,	!- G-Function Ln(T/Ts) Value 23
0.225761,	!- G-Function G Value 23
-12.212124,	!- G-Function Ln(T/Ts) Value 24
0.320868,	!- G-Function G Value 24
-12.069350,	!- G-Function Ln(T/Ts) Value 25
0.415803,	!- G-Function G Value 25
-11.926576,	!- G-Function Ln(T/Ts) Value 26
0.510277,	!- G-Function G Value 26
-11.783802,	!- G-Function Ln(T/Ts) Value 27
0.604039,	!- G-Function G Value 27
-11.641028,	!- G-Function Ln(T/Ts) Value 28
0.696876,	!- G-Function G Value 28
-11.498254,	!- G-Function Ln(T/Ts) Value 29
0.788621,	!- G-Function G Value 29
-11.355480,	!- G-Function Ln(T/Ts) Value 30
0.879151,	!- G-Function G Value 30
-11.212706,	!- G-Function Ln(T/Ts) Value 31
0.968386,	!- G-Function G Value 31
-11.069932,	!- G-Function Ln(T/Ts) Value 32
1.056288,	!- G-Function G Value 32
-10.927158,	!- G-Function Ln(T/Ts) Value 33
1.142852,	!- G-Function G Value 33
-10.784384,	!- G-Function Ln(T/Ts) Value 34
1.228104,	!- G-Function G Value 34
-10.641610,	!- G-Function Ln(T/Ts) Value 35
1.312091,	!- G-Function G Value 35
-10.498836,	!- G-Function Ln(T/Ts) Value 36
1.394878,	!- G-Function G Value 36
-10.356062,	!- G-Function Ln(T/Ts) Value 37
1.476539,	!- G-Function G Value 37
-10.213288,	!- G-Function Ln(T/Ts) Value 38
1.557157,	!- G-Function G Value 38
-10.070514,	!- G-Function Ln(T/Ts) Value 39
1.636815,	!- G-Function G Value 39
-9.927740,	!- G-Function Ln(T/Ts) Value 40
1.715598,	!- G-Function G Value 40
-9.784966,	!- G-Function Ln(T/Ts) Value 41
1.793585,	!- G-Function G Value 41
-9.642192,	!- G-Function Ln(T/Ts) Value 42
1.870854,	!- G-Function G Value 42
-9.499418,	!- G-Function Ln(T/Ts) Value 43
1.947477,	!- G-Function G Value 43
-9.356644,	!- G-Function Ln(T/Ts) Value 44
2.023520,	!- G-Function G Value 44
-9.213870,	!- G-Function Ln(T/Ts) Value 45
2.099043,	!- G-Function G Value 45
-9.071096,	!- G-Function Ln(T/Ts) Value 46
2.174103,	!- G-Function G Value 46
-8.928322,	!- G-Function Ln(T/Ts) Value 47
2.248748,	!- G-Function G Value 47
-8.785548,	!- G-Function Ln(T/Ts) Value 48
2.323025,	!- G-Function G Value 48
-8.642774,	!- G-Function Ln(T/Ts) Value 49
2.396974,	!- G-Function G Value 49
-8.500000,	!- G-Function Ln(T/Ts) Value 50
2.524576,	!- G-Function G Value 50
-7.800000,	!- G-Function Ln(T/Ts) Value 51
2.868617,	!- G-Function G Value 51
-7.200000,	!- G-Function Ln(T/Ts) Value 52
3.165953,	!- G-Function G Value 52
-6.500000,	!- G-Function Ln(T/Ts) Value 53
3.526658,	!- G-Function G Value 53
-5.900000,	!- G-Function Ln(T/Ts) Value 54
3.909966,	!- G-Function G Value 54
-5.200000,	!- G-Function Ln(T/Ts) Value 55
4.624119,	!- G-Function G Value 55
-4.500000,	!- G-Function Ln(T/Ts) Value 56
5.891947,	!- G-Function G Value 56
-3.963000,	!- G-Function Ln(T/Ts) Value 57
7.281048,	!- G-Function G Value 57
-3.270000,	!- G-Function Ln(T/Ts) Value 58

```

10.302878,      !- G-Function G Value 58
-2.864000,      !- G-Function Ln(T/Ts) Value 59
12.617891,      !- G-Function G Value 59
-2.577000,      !- G-Function Ln(T/Ts) Value 60
14.523443,      !- G-Function G Value 60
-2.171000,      !- G-Function Ln(T/Ts) Value 61
17.535574,      !- G-Function G Value 61
-1.884000,      !- G-Function Ln(T/Ts) Value 62
19.865723,      !- G-Function G Value 62
-1.191000,      !- G-Function Ln(T/Ts) Value 63
25.723645,      !- G-Function G Value 63
-0.497000,      !- G-Function Ln(T/Ts) Value 64
31.485939,      !- G-Function G Value 64
-0.274000,      !- G-Function Ln(T/Ts) Value 65
33.115027,      !- G-Function G Value 65
-0.051000,      !- G-Function Ln(T/Ts) Value 66
34.603705,      !- G-Function G Value 66
0.196000,      !- G-Function Ln(T/Ts) Value 67
36.065366,      !- G-Function G Value 67
0.419000,      !- G-Function Ln(T/Ts) Value 68
37.214650,      !- G-Function G Value 68
0.642000,      !- G-Function Ln(T/Ts) Value 69
38.207646,      !- G-Function G Value 69
0.873000,      !- G-Function Ln(T/Ts) Value 70
39.074520,      !- G-Function G Value 70
1.112000,      !- G-Function Ln(T/Ts) Value 71
39.807259,      !- G-Function G Value 71
1.335000,      !- G-Function Ln(T/Ts) Value 72
40.355842,      !- G-Function G Value 72
1.679000,      !- G-Function Ln(T/Ts) Value 73
40.978535,      !- G-Function G Value 73
2.028000,      !- G-Function Ln(T/Ts) Value 74
41.402990,      !- G-Function G Value 74
2.275000,      !- G-Function Ln(T/Ts) Value 75
41.609209,      !- G-Function G Value 75
3.003000,      !- G-Function Ln(T/Ts) Value 76
41.950573;      !- G-Function G Value 76

```

EnergyPlus v8.9 - v9.2 format

This is the latest IDF format used by versions of EnergyPlus since v8.9. It is not supported by current versions of GLD and GLHEPRO but it might be in future versions.

```

Site:GroundTemperature:Undisturbed:FiniteDifference,
TestGroundTemp,      !- Name
0.033333,            !- Soil Thermal Conductivity {W/m-K}
660,                 !- Soil Density {kg/m3}
0.9999,              !- Soil Specific Heat {J/kg-K}
30,                  !- Soil Moisture Content Volume Fraction {percent}
50,                  !- Soil Moisture Content Volume Fraction at Saturation {percent}
0.4;                 !- Evapotranspiration Ground Cover Parameter {dimensionless}

```

```

GroundHeatExchanger:System,
GLE New Format V92,    !- Name
GLE Inlet Node,       !- Inlet Node Name
GLE Outlet Node,      !- Outlet Node Name
0.05555,              !- Design Flow Rate {m3/s}

```

```

Site:GroundTemperature:Undisturbed:FiniteDifference,
TestGroundTemp,      !- Undisturbed Ground Temperature Model Name
0.05555,              !- Ground Thermal Conductivity {W/m-K}
5555000,              !- Ground Thermal Heat Capacity {J/m3-K}
VerticalGFunctions;    !- GHE:Vertical:ResponseFactors Object Name

```

```

GroundHeatExchanger:Vertical:Properties,
Vertical GHE 920,     !- Name
0.5,                  !- Depth of Top of Borehole {m}
121,                  !- Borehole Length {m}
0.02222,              !- Borehole Diameter {m}
0.06666,              !- Grout Thermal Conductivity {W/m-K}
6666000,              !- Grout Thermal Heat Capacity {J/m3-K}
0.07777,              !- Pipe Thermal Conductivity {W/m-K}
7777000,              !- Pipe Thermal Heat Capacity {J/m3-K}
0.33333,              !- Pipe Outer Diameter {m}
0.00444,              !- Pipe Thickness {m}
0.88888;              !- U-Tube Distance {m}

```

```

GroundHeatExchanger:ResponseFactors,

```

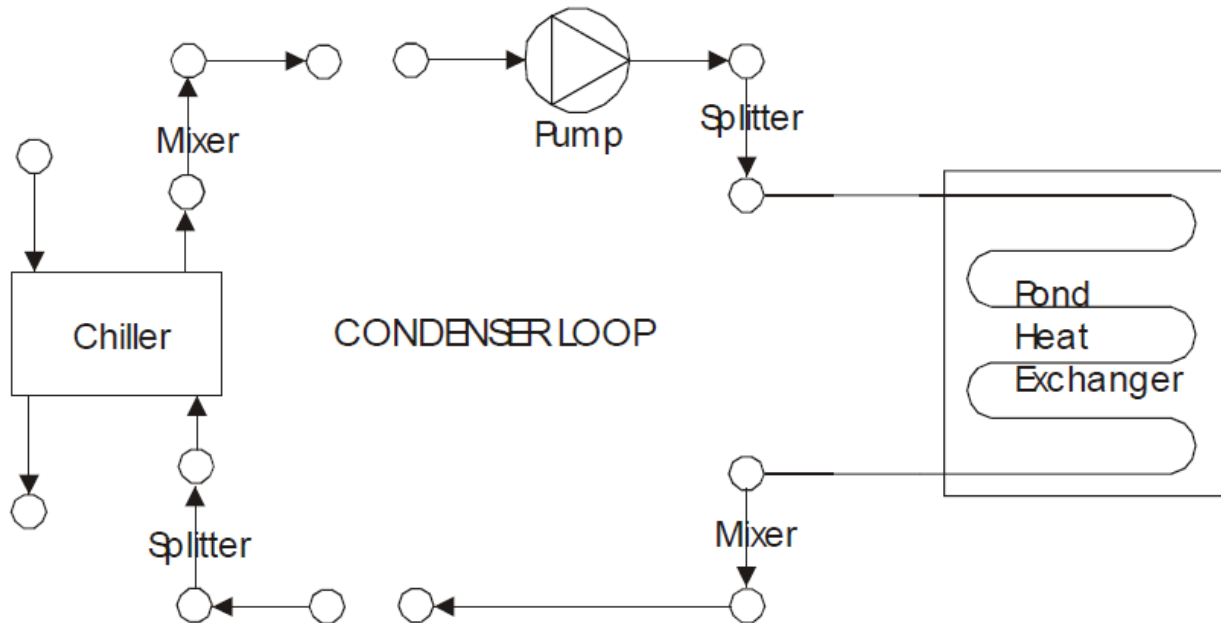
VerticalGFunctions,	!- Name
Vertical GHE 920,	!- GHE:Vertical:Properties Object Name
80,	!- Number of Boreholes
0.0005,	!- G-Function Reference Ratio {dimensionless}
-15.42,	!- G-Function Ln(T/Ts) Value 1
-2.26,	!- G-Function G Value 1
-15.96,	!- G-Function Ln(T/Ts) Value 2
-2.13,	!- G-Function G Value 2
-15.51,	!- G-Function Ln(T/Ts) Value 3
-2.16,	!- G-Function G Value 3
-15.05,	!- G-Function Ln(T/Ts) Value 4
-2.06,	!- G-Function G Value 4
-15.59,	!- G-Function Ln(T/Ts) Value 5
-2.08,	!- G-Function G Value 5
-15.13,	!- G-Function Ln(T/Ts) Value 6
-1.99,	!- G-Function G Value 6
-15.68,	!- G-Function Ln(T/Ts) Value 7
-1.89,	!- G-Function G Value 7
-14.22,	!- G-Function Ln(T/Ts) Value 8
-1.84,	!- G-Function G Value 8
-14.76,	!- G-Function Ln(T/Ts) Value 9
-1.73,	!- G-Function G Value 9
-14.30,	!- G-Function Ln(T/Ts) Value 10
-1.60,	!- G-Function G Value 10
-14.85,	!- G-Function Ln(T/Ts) Value 11
-1.51,	!- G-Function G Value 11
-14.39,	!- G-Function Ln(T/Ts) Value 12
-1.45,	!- G-Function G Value 12
-14.93,	!- G-Function Ln(T/Ts) Value 13
-1.37,	!- G-Function G Value 13
-13.47,	!- G-Function Ln(T/Ts) Value 14
-1.25,	!- G-Function G Value 14
-13.02,	!- G-Function Ln(T/Ts) Value 15
-1.19,	!- G-Function G Value 15
-13.56,	!- G-Function Ln(T/Ts) Value 16
-0.93,	!- G-Function G Value 16
-13.10,	!- G-Function Ln(T/Ts) Value 17
-0.83,	!- G-Function G Value 17
-13.64,	!- G-Function Ln(T/Ts) Value 18
-0.71,	!- G-Function G Value 18
-13.18,	!- G-Function Ln(T/Ts) Value 19
-0.58,	!- G-Function G Value 19
-13.73,	!- G-Function Ln(T/Ts) Value 20
-0.48,	!- G-Function G Value 20
-12.27,	!- G-Function Ln(T/Ts) Value 21
-0.27,	!- G-Function G Value 21
-12.81,	!- G-Function Ln(T/Ts) Value 22
-0.12,	!- G-Function G Value 22
-12.35,	!- G-Function Ln(T/Ts) Value 23
-0.05,	!- G-Function G Value 23
-12.90,	!- G-Function Ln(T/Ts) Value 24
0.13,	!- G-Function G Value 24
-12.44,	!- G-Function Ln(T/Ts) Value 25
0.27,	!- G-Function G Value 25
-12.98,	!- G-Function Ln(T/Ts) Value 26
0.41,	!- G-Function G Value 26
-11.52,	!- G-Function Ln(T/Ts) Value 27
0.54,	!- G-Function G Value 27
-11.07,	!- G-Function Ln(T/Ts) Value 28
0.67,	!- G-Function G Value 28
-11.61,	!- G-Function Ln(T/Ts) Value 29
0.79,	!- G-Function G Value 29
-11.15,	!- G-Function Ln(T/Ts) Value 30
0.91,	!- G-Function G Value 30
-11.69,	!- G-Function Ln(T/Ts) Value 31
1.02,	!- G-Function G Value 31
-11.24,	!- G-Function Ln(T/Ts) Value 32
1.13,	!- G-Function G Value 32
-11.78,	!- G-Function Ln(T/Ts) Value 33
1.23,	!- G-Function G Value 33
-10.32,	!- G-Function Ln(T/Ts) Value 34
1.33,	!- G-Function G Value 34
-10.86,	!- G-Function Ln(T/Ts) Value 35
1.43,	!- G-Function G Value 35
-10.41,	!- G-Function Ln(T/Ts) Value 36
1.52,	!- G-Function G Value 36
-10.95,	!- G-Function Ln(T/Ts) Value 37
1.61,	!- G-Function G Value 37
-10.49,	!- G-Function Ln(T/Ts) Value 38
1.70,	!- G-Function G Value 38

-10.03,	!- G-Function Ln(T/Ts) Value 39
1.79,	!- G-Function G Value 39
-10.58,	!- G-Function Ln(T/Ts) Value 40
1.87,	!- G-Function G Value 40
-9.82,	!- G-Function Ln(T/Ts) Value 41
1.96,	!- G-Function G Value 41
-9.76,	!- G-Function Ln(T/Ts) Value 42
2.04,	!- G-Function G Value 42
-9.50,	!- G-Function Ln(T/Ts) Value 43
2.12,	!- G-Function G Value 43
-9.45,	!- G-Function Ln(T/Ts) Value 44
2.20,	!- G-Function G Value 44
-9.29,	!- G-Function Ln(T/Ts) Value 45
2.28,	!- G-Function G Value 45
-9.13,	!- G-Function Ln(T/Ts) Value 46
2.36,	!- G-Function G Value 46
-8.97,	!- G-Function Ln(T/Ts) Value 47
2.44,	!- G-Function G Value 47
-8.82,	!- G-Function Ln(T/Ts) Value 48
2.52,	!- G-Function G Value 48
-8.66,	!- G-Function Ln(T/Ts) Value 49
2.60,	!- G-Function G Value 49
-8.50,	!- G-Function Ln(T/Ts) Value 50
2.73,	!- G-Function G Value 50
-7.80,	!- G-Function Ln(T/Ts) Value 51
3.08,	!- G-Function G Value 51
-7.20,	!- G-Function Ln(T/Ts) Value 52
3.42,	!- G-Function G Value 52
-6.50,	!- G-Function Ln(T/Ts) Value 53
3.99,	!- G-Function G Value 53
-5.90,	!- G-Function Ln(T/Ts) Value 54
4.80,	!- G-Function G Value 54
-5.20,	!- G-Function Ln(T/Ts) Value 55
6.41,	!- G-Function G Value 55
-4.50,	!- G-Function Ln(T/Ts) Value 56
9.06,	!- G-Function G Value 56
-3.90,	!- G-Function Ln(T/Ts) Value 57
11.90,	!- G-Function G Value 57
-3.20,	!- G-Function Ln(T/Ts) Value 58
17.14,	!- G-Function G Value 58
-2.80,	!- G-Function Ln(T/Ts) Value 59
20.78,	!- G-Function G Value 59
-2.50,	!- G-Function Ln(T/Ts) Value 60
23.69,	!- G-Function G Value 60
-2.10,	!- G-Function Ln(T/Ts) Value 61
27.91,	!- G-Function G Value 61
-1.80,	!- G-Function Ln(T/Ts) Value 62
31.01,	!- G-Function G Value 62
-1.10,	!- G-Function Ln(T/Ts) Value 63
38.51,	!- G-Function G Value 63
-0.40,	!- G-Function Ln(T/Ts) Value 64
45.21,	!- G-Function G Value 64
-0.20,	!- G-Function Ln(T/Ts) Value 65
47.08,	!- G-Function G Value 65
-0.00,	!- G-Function Ln(T/Ts) Value 66
48.72,	!- G-Function G Value 66
0.19,	!- G-Function Ln(T/Ts) Value 67
50.38,	!- G-Function G Value 67
0.41,	!- G-Function Ln(T/Ts) Value 68
51.63,	!- G-Function G Value 68
0.64,	!- G-Function Ln(T/Ts) Value 69
52.76,	!- G-Function G Value 69
0.87,	!- G-Function Ln(T/Ts) Value 70
53.72,	!- G-Function G Value 70
1.11,	!- G-Function Ln(T/Ts) Value 71
54.50,	!- G-Function G Value 71
1.33,	!- G-Function Ln(T/Ts) Value 72
55.12,	!- G-Function G Value 72
1.67,	!- G-Function Ln(T/Ts) Value 73
55.87,	!- G-Function G Value 73
2.02,	!- G-Function Ln(T/Ts) Value 74
56.38,	!- G-Function G Value 74
2.27,	!- G-Function Ln(T/Ts) Value 75
56.50,	!- G-Function G Value 75
3.00,	!- G-Function Ln(T/Ts) Value 76
56.90;	!- G-Function G Value 76

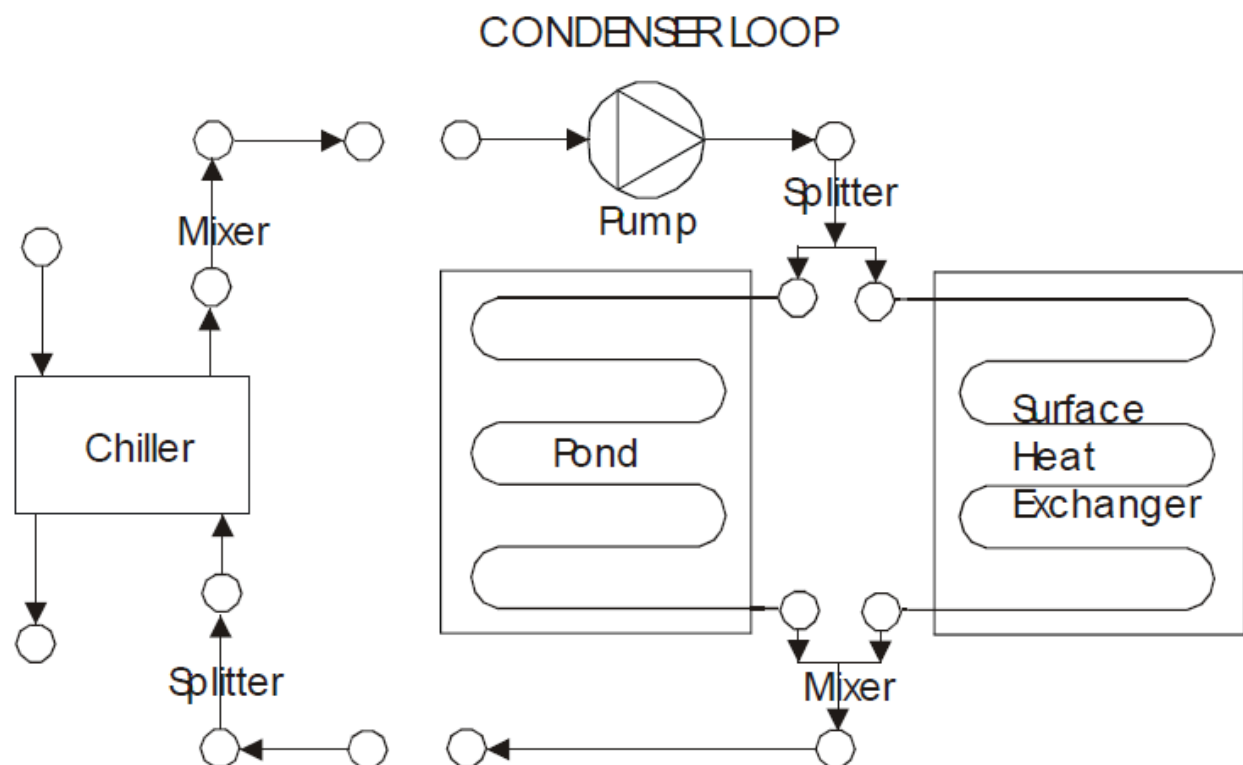
Ground Heat Exchanger - Pond

The pond heat exchanger model represents a shallow pond with submerged hydronic tubes through which the heat transfer fluid is circulated. The model represents a "shallow" pond in that no attempt is made to model any stratification effects that may be present in deeper ponds.

This type of heat exchanger is intended to be connected to the supply side of a condenser loop, and can be used with any type of plant loop. The pond may be specified as the only heat exchanger on the condenser loop (as shown in top figure below) or it may be connected in parallel with other condenser loop heat exchangers (such as cooling towers, ground surface heat exchangers) as shown in the second figure below.



Pond heat exchanger as only heat exchanger on condenser loop



Pond heat exchanger with other heat exchangers on condenser loop

Technical

The pond model uses a 'lumped parameter' method where the pond is represented by a single node with thermal mass. The pond surface temperature is the same as the temperature at this node, i.e. the surface temperature is the same as the bulk temperature. A first order differential equation is solved in the model to calculate the pond temperature at each time step. This type of heat rejecter is modelled as several circuits connected in parallel.

Heat transfer takes place by surface convection, long-wave radiation to the sky, absorption of solar energy, ground heat transfer and heat exchange with the fluid. A heat exchanger analogy is used to calculate the heat transfer between the heat transfer fluid and the pond.

General

Name

This alpha field contains the editable identifying name for the outside pond heat exchanger.

Ground Heat Exchanger Template

Template

You can use this control to load data to the dialog from a pre-defined Ground heat exchanger template as a starting point for your particular component.

Note: After loading a Ground heat exchanger template it is important to make sure to update the [Design](#) heat exchanger flow rates as the sum of connected heat pump rated flow rates.

Ground Heat Exchanger Type

Type

DesignBuilder provides 3 different ground heat exchanger types:

- [1-Vertical](#) (boreholes)
- [2-Pond](#)
- [3-Surface](#)

Note: The **Type** drop down list is not editable. To select a different type of heat exchanger, you must first select an appropriate template (above).

Ground Heat Exchanger - Pond

Ground thermal conductivity

This numeric field contains the ground thermal conductivity (in W/mK or Btu-in/h-ft²-F).

Pond depth

This numeric field contains the pond depth (in m or ft).

Pond area

This numeric field contains the pond area (in m² or ft²).

Hydronic tubing inside diameter

This numeric field contains the hydronic tubing inside diameter (in m or in).

Hydronic tubing outside diameter

This numeric field contains the hydronic tubing outside diameter (in m or in).

Hydronic tubing thermal conductivity

This numeric field contains the hydronic tubing thermal conductivity (in W/mK or Btu-in/h-ft²-F).

Number of tubing circuits

This numeric field contains the number of hydronic tubing circuits, total in parallel in this pond.

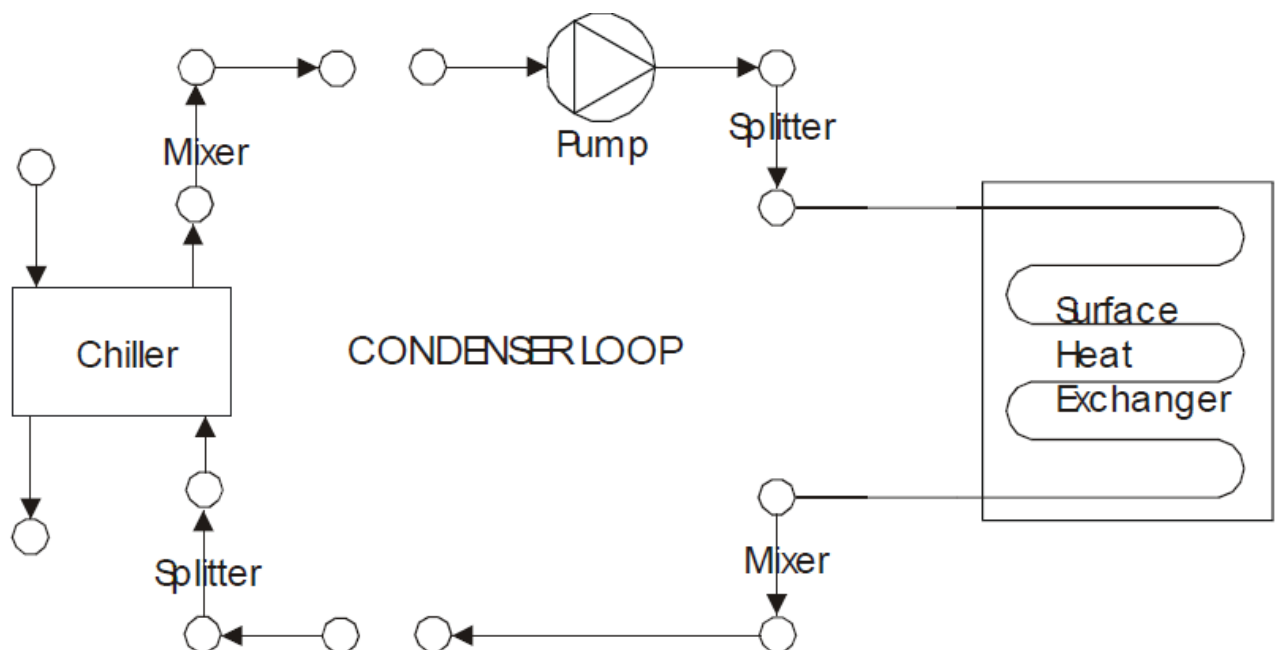
Length of each tubing circuit

This numeric field contains length (in m or ft) of each hydronic tubing circuit.

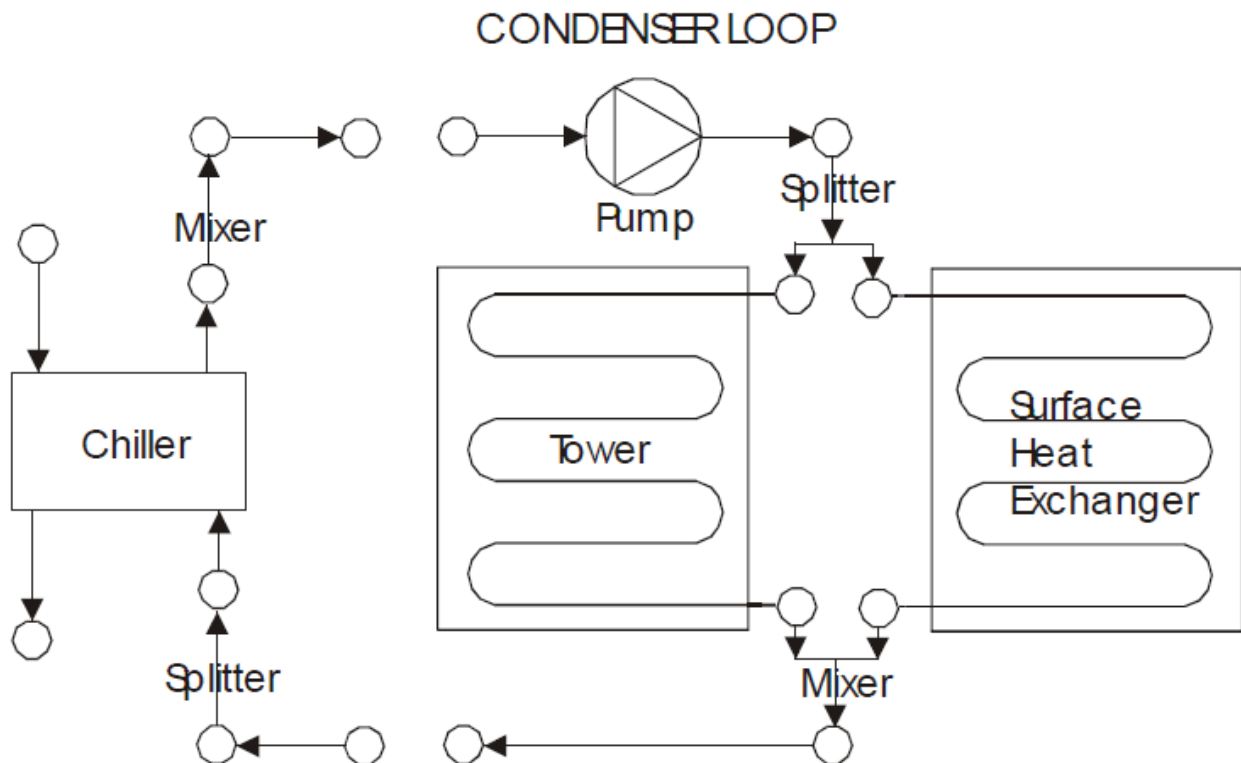
Ground Heat Exchanger - Surface

Surface heat exchangers include pavement surfaces with embedded pipes for snow-melting or heat rejection from hybrid ground source heat pump systems. The heat exchanger may be ground coupled or not and when not ground coupled the bottom surface is exposed to the wind but not solar gains.

This type of heat exchanger is intended to be connected to the supply side of a condenser loop, and can be used with any type of plant loop. The surface heat exchanger may be specified as the only heat exchanger on the condenser loop (as shown in the first figure below) or it may be connected in parallel with other condenser loop heat exchangers (such as cooling towers, ground surface heat exchangers) as shown in the second figure below.



Surface Ground Heat Exchanger as only heat exchanger on condenser loop



Surface Ground Heat Exchanger with other heat exchangers on condenser loop

Technical description

The Surface ground heat exchanger model is based on the QTF (Quadratic Transfer Function) formulation of heat transfer through building elements with embedded heat sources/sinks. The model uses a heat exchanger analogy to relate the inlet fluid temperature to the net heat transfer rate and consequently outlet temperature. The model is entirely passive, i.e. it does not set any flow rates or incorporate any controls. In order to deal with the non-linear boundary conditions at the top surface due to the presence of ice/snow fluxes have to be calculated by the QTF model and temperature calculated from the surface heat balance. This requires some iteration. Note, top surface variables correspond to 'outside' variables in standard CTF/QTF definition (CTF=Conduction Transfer Functions). Bottom surface variables correspond to 'inside' variables.

General

Name

This alpha field contains the editable identifying name for the outside panel heat exchanger.

Ground Heat Exchanger Template

Template

You can use this control to load data to the dialog from a pre-defined Ground heat exchanger template as a starting point for your particular component.

Note: After loading a Ground heat exchanger template it is important to make sure to update the [Design](#) heat exchanger flow rates as the sum of connected heat pump rated flow rates.

Ground Heat Exchanger Type

Type

DesignBuilder provides 3 different ground heat exchanger types:

- [1-Vertical](#) (boreholes)
- [2-Pond](#)
- [3-Surface](#)

Note: The **Type** drop down list is not editable. To select a different type of heat exchanger, you must first select an appropriate template (above).

Ground Heat Exchanger - Surface

Hydronic tubing inside diameter

This numeric field contains the hydronic tubing inside diameter (in m or in).

Number of tubing circuits

This numeric field contains the number of hydronic tubing circuits.

Hydronic tube spacing

This numeric field contains the hydronic tube spacing (in m or ft).

Surface length

This numeric field contains the surface length (in m or ft).

Surface width

This numeric field contains the surface width (in m or ft).

Lower Surface Environment

This alpha field expresses the lower surface exposure. Options are:

- [1-Ground](#) where the surface is in contact with the ground below.
- [2-Exposed](#) where the surface is exposed below to the ambient air, or

Construction

The [Construction](#) selected here defines the layers of materials used to make the heat exchange surface.

Note: The construction must include an [Internal Source](#) to accommodate the heat exchange pipes.

If you select a construction with large thickness to represent the ground you may need to use the [2-Finite differenceSolution algorithm](#) either for this construction only or for the whole model.

VRF Outdoor Unit

Used in:



AirConditioner:VariableRefrigerantFlow VRF Loop

This model simulates a variable-refrigerant-flow (or variable-refrigerant-volume) air-conditioning system. This system type models direct-expansion cooling and/or heating coils configured in a zone indoor unit.

There are two common types of variable refrigerant flow (VRF) heat pump systems:

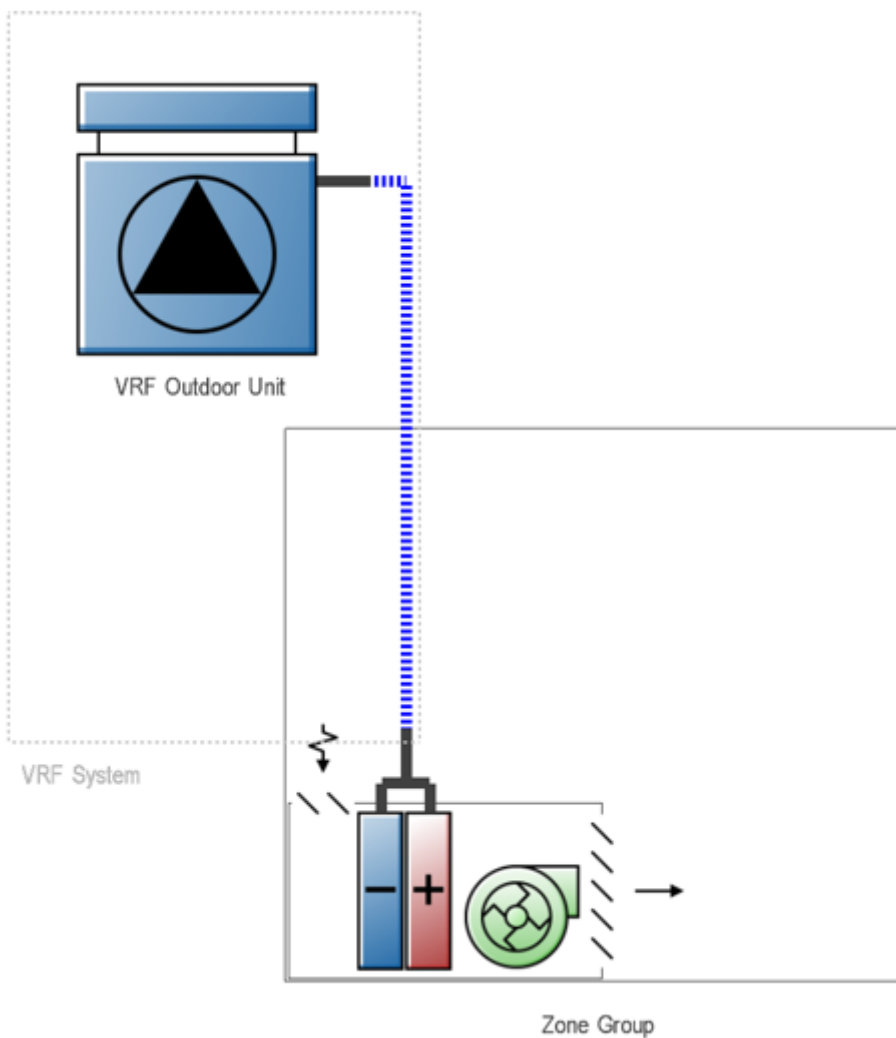
- Cooling only or heating only air-conditioning systems (a.k.a. heat pump), or
- Heat recovery systems that allow simultaneous cooling and heating

The VRF outdoor unit supports air-, evaporatively-, or water-cooled condenser equipment. Throughout this section, the term “condenser” refers to the outdoor unit where the compressor is located.

Enter data on one of the 4 tabs:

- [General](#)
- [Cooling](#)
- [Heating](#)
- [Heat Recovery](#)

The screenshot below shows a typical VRF layout diagram with outdoor and indoor units linked.



VRF Resources

You can learn more about practical aspects of VRF design and simulation for project work (in the context of GSHP systems) by watching our webinar: [Design and Modelling of Ground Source Heat Pump Systems](#)

- DesignBuilder curve generation spreadsheets can be downloaded from the [software downloads](#) area of the website.
- The EnergyPlus Engineering Guide includes a [Detailed technical description](#) of the way the VRF model works.
- Richard Raustad of Florida Solar Energy Center report on [Creating Performance Curves for Variable Refrigerant Flow Heat Pumps in EnergyPlus](#)
- [VRF System Design and Simulation Webinar](#) presented by DesignBuilder, Mitsubishi and Building Performance Team, recorded on 01.09.15.

VRF Outdoor Unit - General Tab

General

Note on terminology: The terms VRF "condenser", "outdoor unit" and "heat pump" are used interchangeably in the this section. Likewise the terms VRF indoor unit and terminal unit are used interchangeably.

Name

Enter the name of the VRF outdoor unit.

VRF outdoor unit template

To load a pre-configured VRF outdoor unit dataset, select a template from the list.

Fuel type

This field determines the type of fuel that this variable refrigerant flow system uses. This field has seven choices:

- **1-Natural gas,**
- **2-Electricity,**
- **3-Propane gas,**
- **4-FuelOil#1,**
- **5-FuelOil#2,**
- **6-Coal,**
- **7-Diesel,**
- **8-Gasoline,**
- **9-OtherFuel1,**
- **10-OtherFuel2.**

The default is **2-Electricity**. The use of alternate fuel types assumes an engine drives the variable speed compression system and also accounts for condenser air flow (i.e., a fan attached to the engine provides air flow through the outdoor condenser).

Minimum heat pump part-load ratio

This numeric field specifies the minimum operating part-load ratio (PLR) of the heat pump. When the heat pump operates at a PLR below this value, the heat pump's compressor will cycle to meet the cooling or heating demand. Above this value, the heat pump's compressor operates the entire time step to meet the cooling or heating demand. The minimum value for this field is 0. If this field is left blank, the default value is

0.15. When the heat pump compressor cycles, the heating part-load fraction correlation curve is used to determine cycling losses.

Availability schedule

Select the schedule that defines when the heat pump operates during a given time period. A schedule value equal to 0 denotes that the heat pump must be off for that time period. A value other than 0 denotes that the heat pump is available to operate during that time period. This schedule may be used to completely disable the heat pump (and all of its terminal units) as required.

Control

Master thermostat priority control type

This selection determines the logic used to simulate the “master” thermostat. Valid choices are:

- **1-Load priority** where the total zone load is used to choose the operating mode as either cooling or heating.
- **2-Zone priority** where the number of zones requiring cooling or heating determines the operating mode.
- **3-Thermostat offset priority**, in which the zone farthest from the thermostat set point determines the operating mode.
- **4-Master thermostat priority** which operates the system according the zone load where the master thermostat is located.
- **5-Scheduled** where the heat pump is scheduled to operate in either cooling or heating mode.

Zone for master thermostat location

When the master thermostat priority control type is set to **3-Thermostat offset priority**, or **4-Master thermostat priority** you must select the zone where the “master” thermostat is located. When the heat pump is connected to multiple zone terminal units, one terminal unit must be selected as the master thermostat. The remaining thermostats are slaves and can operate only in the same mode as the master thermostat.

Thermostat priority schedule

When the master thermostat priority control type is set to **5-Scheduled** the schedule must be selected.

Schedule values of 0 denote heating mode while values of 1 denote cooling mode. Any other values will force the system off.

Piping

Equivalent piping length used for piping correction factor in cooling mode

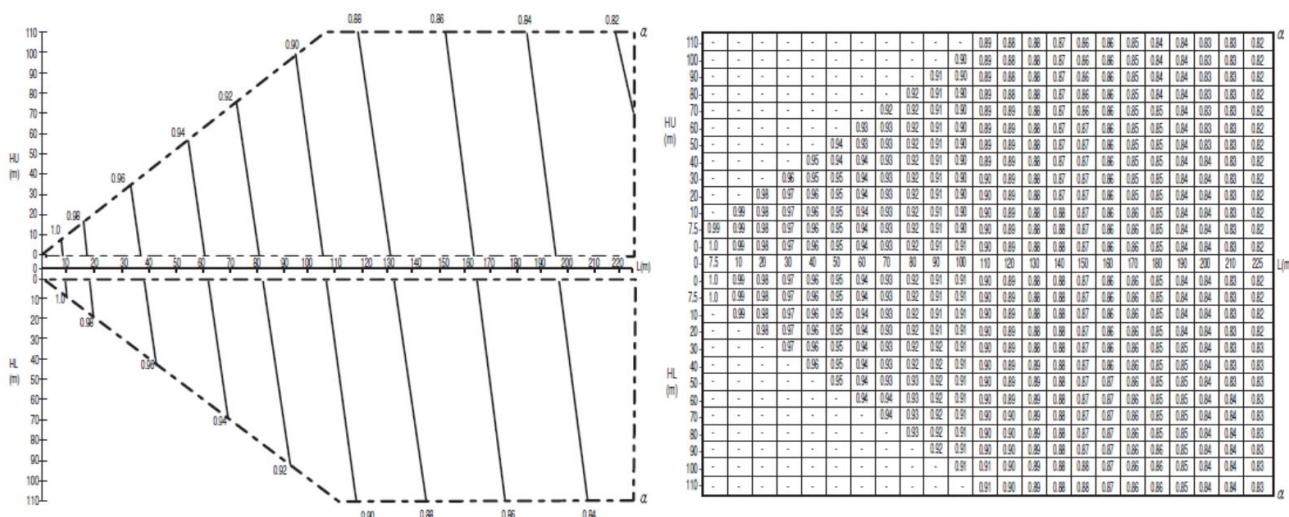
This field defines the equivalent pipe length (in m or ft) between the farthest terminal unit and the heat pump condenser. It includes the gas refrigerant line length (for both horizontal and vertical distances), fitting losses, pipe bends, and other connections that contribute to piping losses. This field is used to calculate the piping correction factor in cooling mode. It defines the head losses due to the pipe length between the farthest terminal unit and the heat pump condenser and impacts the maximum available capacity in cooling mode.

Vertical height used for piping correction factor

This field defines the vertical pipe height (in m or ft) between the highest or lowest terminal unit and the heat pump condenser. It defines the gravitational losses due to a change in height between the highest (positive value), or lowest (negative value) terminal unit and the heat pump condenser. The distance specified here is applied to the piping correction factor calculation for both cooling and heating. If the distance between the highest terminal unit above the heat pump condenser is greater than the distance between the lowest terminal unit below the condenser enter the difference between the highest and lowest terminal units as a positive distance, otherwise enter this difference as a negative distance.

Piping correction factor for height in cooling mode coefficient

The figure below from a manufacturer provides this information in the tilted vertical lines on the chart, which shows that the correction factor decreases by about 0.11 when the terminal unit is 110 m above the condenser. This means that the correction factor for height is $-0.11 / 110 \text{ m}$ or -0.001 .



Equivalent piping length used for piping correction factor in heating mode

Piping correction factor for height in heating mode coefficient

Piping correction factor for length in cooling mode curve

Piping correction factor for length in heating mode

- 1074 -

Compressor

Crankcase heater power per compressor

This numeric field defines the electrical power consumed by the crankcase heater (in W or Btu/h) for each compressor. This crankcase heater power is consumed when the outdoor temperature is below the maximum outdoor dry-bulb temperature for crankcase heater operation. The minimum value for this field is 0. Crankcase heater electrical consumption is applied only when the compressor is off or is applied during the off cycle when the compressor is cycling below the [Minimum heat pump part-load ratio](#). This field is only used to calculate crankcase heater power and has no impact on heat pump performance.

Number of compressors

This field defines the number of compressors in the heat pump condensing unit and is used exclusively to determine the operating characteristics of the crankcase heater. For example, if the number of compressors is 3, one crankcase heater will operate when the heat pump condensing unit's part-load ratio is less than or equal to 0.67 (when the ratio of compressor size to total compressor capacity input is 0.33) and the outdoor temperature is below the maximum outdoor temperature for crankcase heater operation. Similarly, two crankcase heaters will operate when the heat pump condensing unit's PLR is less than or equal to 0.33 and the outdoor temperature is below the maximum outdoor temperature for crankcase heater operation. If the heat pump condensing unit is off, all 3 crankcase heaters will operate if the outdoor temperature is below the maximum outdoor temperature for crankcase heater operation. The minimum value for this field is 1. This field is only used to calculate crankcase heater power and has no impact on heat pump performance.

Ratio of compressor size to total compressor capacity

This field defines the size of the first stage compressor to the total compressor capacity and is used exclusively for calculating crankcase heater energy. When the number of compressors is greater than 2, the 2nd stage compressor and all additional compressors are assumed to be equally sized. This field is only used to calculate crankcase heater power and has no impact on heat pump performance.

Maximum outdoor dry-bulb temperature for crankcase heater

This field defines the maximum outdoor temperature, (in °C or °F), below which the crankcase heater will operate. This field is only used to calculate crankcase heater power and has no impact on heat pump performance.

Defrost

Defrost strategy

This alpha field has two choices:

- **1-Resistive** where the frost is melted using an electric resistance heater.
- **2-Reverse-cycle** where the heating cycle is reversed periodically to provide heat to melt frost accumulated on the outdoor coil.

Defrost can be disabled by entering a resistive defrost strategy using a timed defrost control, a 0 defrost time period fraction and a 0 resistive defrost heater capacity in the following inputs fields. This method is used when the [Maximum outdoor dry-bulb temperature for defrost operation](#) value is greater than the expected minimum outdoor dry-bulb temperature simulated in the weather file.

Defrost control

This field has two choices:

- **1-Timed** where the defrost time period is calculated based on a fixed value of compressor runtime whether or not frost has actually accumulated. For timed defrost control, the fractional amount of time the unit is in defrost is entered in the input field [Defrost time period fraction](#) described below.
- **2-On-demand** where the defrost time period is calculated based on outdoor weather (humidity ratio) conditions.

Regardless of which defrost control is selected, defrost does not occur above the user specified outdoor temperature entered in the [Maximum outdoor dry-bulb temperature for defrost operation](#).

Defrost energy input ratio (EIR) modifier function of temperature curve

Available only when the [reverse-cycle](#) defrost strategy option is selected, this field defines the name of a bi-quadratic performance curve that parameterizes the variation of the energy input ratio (EIR) during reverse-cycle defrost periods as a function of the outdoor air dry-bulb temperature and the weighted average wet-bulb temperature of the air entering the indoor terminal units. The output of this curve is multiplied by the coil capacity, the fractional defrost time period and the runtime fraction of the heating coil to give the defrost power at the specific temperatures at which the indoor and outdoor coils are operating. The curve is normalized to a value of 1.0 at the rating point conditions.

Defrost time period fraction

This field defines the fraction of compressor runtime when the defrost cycle is active. For example, if the defrost cycle is active for 3.5 minutes for every 60 minutes of compressor runtime, then the user should enter $3.5/60 = 0.058333$. The value for this input field must be greater than or equal to 0. If this input field is left blank, the default value is 0.058333.

Resistive defrost heater capacity

This field defines the capacity of the resistive defrost heating element (in W or Btu/h). It is available only when the selected defrost strategy is [1-Resistive](#). The value for this input field must be greater than or equal to 0.

Maximum outdoor dry-bulb temperature for defrost operation

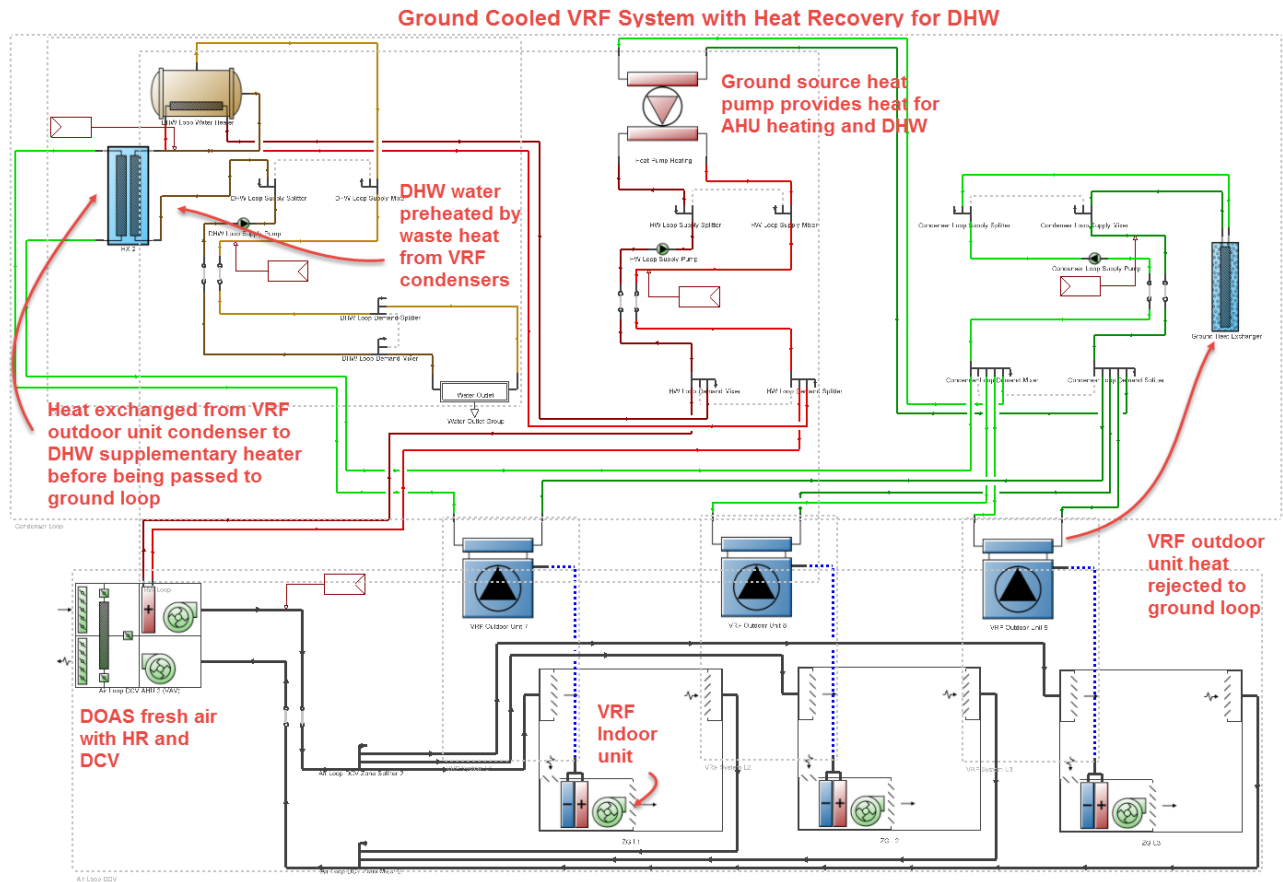
This field defines the outdoor air dry-bulb temperature (in °C or °F) above which outdoor coil defrosting is disabled. Defrost can be completely eliminated by selecting a temperature lower than the minimum expected outdoor temperature found in the weather file.

Condenser

Condenser type

This choice field defines the configuration of the heat pump condenser. Valid choices are:

- **1-Air cooled**,
- **2-Evaporatively cooled**, and
- **3-Water cooled**. In this case the outlet of the water cooled condenser unit is connected to the demand side of a condenser loop as shown in the example HVAC layout diagram below.



Example of VRF system where waste heat is recovered from water-cooled outdoor unit for preheating DHW

For details on evaporatively cooled systems see below under [Evaporative condenser effectiveness](#).

Water condenser volume flow rate

This autosizable numeric field defines the condenser water volume flow rate (in m3/s or gal/min) and is only available for water-cooled systems.

Evaporative condenser effectiveness

The effectiveness of the evaporative condenser, which is used to determine the temperature of the air entering the outdoor condenser coil as follows:

$$T_{\text{cond inlet}} = T_{\text{wbo}} + (1 - \text{EvapCondEffectiveness}) (T_{\text{dbo}} - T_{\text{wbo}})$$

where:

$T_{\text{cond inlet}}$ = the temperature of the air entering the condenser coil (°C)

T_{wbo} = the wet-bulb temperature of the outdoor air (°C)

T_{dbo} = the dry-bulb temperature of the outdoor air (°C)

The resulting condenser inlet air temperature is used by the [Cooling capacity ratio modifier function of temperature curve](#) and the [Cooling energy input ratio \(EIR\) modifier function of temperature curve](#). The default value for this field is 0.9, although valid entries can range from 0.0 to 1.0.

To model an air cooled condenser specify [Condenser type](#) = **1-Air cooled**. In this case the [Cooling capacity ratio modifier function of temperature curve](#) and the [Cooling energy input ratio \(EIR\) modifier function of temperature curve](#) should be defined as a function of outdoor dry-bulb temperature.

To model an evaporative-cooled condenser where you have performance curves that are a function of the wet-bulb temperature of air entering the condenser coil, then you should specify [Condenser type = 2-Evaporatively cooled](#). In this case the evaporative condenser effectiveness value should be entered as 1.0 and the [Cooling capacity ratio modifier function of temperature curve](#) and the [Cooling energy input ratio \(EIR\) modifier function of temperature curve](#) should both be defined as a function of the wet-bulb temperature of air entering the condenser coil.

To model an air-cooled condenser that has evaporative media placed in front of it to cool the air entering the condenser coil, specify [Condenser type = 2-Evaporatively cooled](#) and enter the appropriate evaporative effectiveness for the media. In this case, the [Cooling capacity ratio modifier function of temperature curve](#) and the [Cooling energy input ratio \(EIR\) modifier function of temperature curve](#) should both be defined as function of outdoor dry-bulb temperature. Be aware that the evaporative media will significantly reduce the dry-bulb temperature of the air entering the condenser coil, so the Cooling Capacity and Cooling EIR Modifier Curves must be valid for the expected range of dry-bulb temperatures that will be entering the condenser coil.

Evaporative condenser air flow rate

The air volume flow rate (in m³/s or ft³/min) entering the evaporative condenser. This value is used to calculate the amount of water used to evaporatively cool the condenser inlet air. The minimum value for this field must be greater than zero, and this input field is autosizable (equivalent to 0.000144 m³/s per watt of [Gross rated total cooling capacity](#) [850 cfm/ton]). This field is not used when [Condenser type = 1-Air cooled](#).

Evaporative condenser pump rated power consumption

The rated power of the evaporative condenser water pump (in W or Btu/h). This value is used to calculate the power required to pump the water used to evaporatively cool the condenser inlet air. The default value for this input field is zero, but it is autosizable (equivalent to 0.004266 W per watt [15 W/ton] of [Gross rated total cooling capacity](#)). This field is not used when [Condenser type = 1-Air cooled](#).

Basin heater capacity

This field contains the capacity of the heat pump's electric basin heater (in W/K or Btu/h). This field only applies for [Condenser type = 2-Evaporatively cooled](#). This field is used in conjunction with the Basin heater setpoint temperature described below. The basin heater electric power is equal to this field multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the heat pump compressor(s) is off, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero.

Basin heater setpoint temperature

This field contains the set point temperature (°C or °F) for the basin heater described above. This field only applies for [Condenser type = 2-Evaporatively cooled](#). The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the heat pump is off. This set point temperature must be greater than or equal to 2°C.

Basin heater operating schedule

Select the basin heater operating schedule. This field only applies for [Condenser type = 2-Evaporatively cooled](#). The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin heater operates when scheduled on and the outdoor air dry-bulb temperature is below the set point temperature described above. Regardless of this schedule, the basin heater may only operate when the heat pump is off.

VRF Outdoor Unit - Cooling Tab

General

Rated total cooling capacity

This field defines the total cooling capacity (sensible + latent) of the heat pump at rated conditions in W or Btu/h. The cooling capacity must be greater than 0 or set to autosize.

When the VRF outdoor unit is not autosized, the sum of the VRF terminal unit coils (autosized or not) and the VRF outdoor unit capacity are used to calculate the combination ratio for both cooling and heating. For a fully autosized system, the combination ratio = 1, but when the system is not fully autosized, the resulting sizes of the terminal unit coils and the outdoor unit are used to calculate a combination ratio.

Rated cooling COP

This field defines the cooling coefficient of performance at rated conditions. The cooling coefficient of performance includes compressor power and condenser fan power. This COP value does not include impacts due to the supply air fan.

Minimum outdoor temperature in cooling mode

This field defines the minimum outdoor dry-bulb temperature allowed for cooling operation (in °C or °F). Below this temperature, cooling is disabled.

Maximum outdoor temperature in cooling mode

This field defines the maximum outdoor dry-bulb temperature allowed for cooling operation (in °C or °F). Above this temperature, cooling is disabled.

Cooling Capacity Ratio Modifier Function Of Temperature Curves

Use single or multiple curves

The way that the cooling capacity ratio modifier varies with temperature can be defined using either a *single* curve (if the cooling capacity does not change dramatically with changes in outdoor conditions) or by using *multiple* (3) curves if the trend in cooling capacity changes dramatically as outdoor temperature changes. Select from:

- **1-Single** where the whole relation is defined in a single cooling capacity ratio modifier function of temperature curve, or,
- **2-Multiple.**

Cooling capacity ratio modifier function of temperature curve

Available only when using a [single](#) cooling capacity ratio modifier function of temperature curve, this field defines the cooling capacity ratio modifier at all outdoor dry-bulb temperatures. This curve is a bi-quadratic equation using weighted average indoor wet-bulb temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air dry-bulb temperature as the independent variables.

Cooling capacity ratio modifier function of low temperature curve

Available only when using a [multiple](#) cooling capacity ratio modifier function of temperature curves this field defines the cooling capacity ratio modifier at low outdoor dry-bulb temperatures. This curve is a bi-quadratic equation using weighted average indoor wet-bulb temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air dry-bulb temperature as the independent variables.

Cooling capacity ratio boundary curve

Available only when using a [multiple](#) cooling capacity ratio modifier function of temperature curves, this curve defines the cooling capacity ratio boundary. It is a linear, quadratic or cubic curve that defines a change in cooling capacity at a specific condenser entering air dry-bulb temperature as a function of indoor air wet-bulb temperature.

Cooling capacity ratio modifier function of high temperature curve

Available only when using a [multiple](#) cooling capacity ratio modifier function of temperature curves, this field defines the cooling capacity ratio modifier at high outdoor temperatures. This curve is a bi-quadratic equation using weighted average indoor wet-bulb temperature and condenser entering air dry-bulb temperature as the independent variables.

Cooling Energy Input Ratio (EIR) Modifier Function Of Temperature Curves

The Cooling energy input ratio (EIR) modifier is a factor used to adjust the rated EIR (reciprocal of [Rated cooling COP](#)) as outdoor temperature changes.

Use single or multiple curves

The way that the cooling EIR varies with temperature can be defined using either a *single* curve (if the cooling EIR does not change dramatically with changes in outdoor conditions) or by using *multiple* (3) curves if the trend in cooling EIR changes dramatically as outdoor temperature changes. Select from:

- **1-Single** where the whole relation is defined in a single cooling EIR function of temperature curve, or,
- **2-Multiple**.

Cooling energy input ratio (EIR) modifier function of temperature curve

Available only when using a [single](#) cooling capacity ratio modifier function of temperature curve, this field defines the cooling energy input ratio modifier at all outdoor temperatures. This curve is a bi-quadratic equation with a weighted average indoor wet-bulb temperature and condenser entering air dry-bulb temperature as the independent variables.

Cooling energy input ratio (EIR) modifier function of low temperature curve

Available only when using [multiple](#) cooling EIR curves, this field defines the cooling energy input ratio modifier at low outdoor temperatures. This curve is a bi-quadratic equation with a weighted average indoor wet-bulb temperature and condenser entering air dry-bulb temperature as the independent variables.

Cooling energy input ratio (EIR) boundary curve

Available only when using [multiple](#) cooling EIR curves, the cooling energy input ratio boundary curve is a linear, quadratic or cubic curve that defines a change in cooling energy at a specific condenser entering air dry-bulb temperature as a function of indoor air wet-bulb temperature.

Cooling energy input ratio (EIR) modifier function of high temperature curve

Available only when using [multiple](#) cooling EIR curves, this field defines the cooling energy input ratio modifier at high outdoor temperatures. This curve is a bi-quadratic equation with weighted average indoor wet-bulb temperature and condenser entering air dry-bulb temperature as the independent variables.

Cooling energy input ratio (EIR) modifier function of low part-load ratio curve

This curve defines how the cooling EIR modifier varies as a function of part-load ratio when PLR is less than or equal to 1. It is a linear, quadratic or cubic equation with cooling part-load ratio used as the independent variable. The cooling energy input ratio modifier curve is normalized to 1 at a part-load ratio of 1 and is used only when the operating part-load ratio is less than or equal to 1.

Cooling energy input ratio (EIR) modifier function of high part-load ratio curve

This curve defines how the cooling EIR modifier varies as a function of part-load ratio when PLR is greater than 1. It is a linear, quadratic or cubic equation with cooling part-load ratio used as the independent variable. The cooling energy input ratio modifier curve is normalized to 1 at a part-load ratio of 1 and is used only when the operating part-load ratio is greater than 1.

Other Curves

Cooling combination ratio correction factor curve

This field defines the cooling combination ratio (CR) correction factor curve for combination ratios greater than or equal to 1. The combination ratio is defined as the total rated indoor terminal unit cooling capacity divided by this heat pump's [Rated total cooling capacity](#). The curve is a linear, quadratic or cubic equation and uses the minimum value of x in the curve object to determine the maximum part-load ratio which is linearly proportional to capacity (i.e., the minimum value of x [CR] in the curve object must be ≥ 1). The output of this curve provides a multiplier (>1) which is applied to this heat pump's [Rated total cooling capacity](#). Between a combination ratio of 1 and the curve's minimum value of x, the multiplier is linearly interpolated. For combination ratio's less than 1 (i.e., the total indoor terminal unit capacity is less than this heat pump's rated total capacity), capacity is directly proportional to part-load ratio and this curve will not be used.

Cooling part-load fraction correlation curve

This field defines the cooling part-load fraction correlation curve. This curve is used to define the cycling losses when the condenser's compressors cycle on and off. The compressor cycles when the cooling part-load ratio is less than the [Minimum heat pump part-load ratio](#).

VRF Outdoor Unit - Heating Tab

General

Rated total heating capacity

This numeric field defines the total heat pump heating capacity at rated conditions in watts. The nominal heat pump heating capacity must be greater than 0 or set to autosize.

Rated total heating capacity sizing ratio

This field defines the ratio of heating to cooling capacity of the Outdoor unit. This value must be greater than or equal to 1. A [similar input](#) is available in the VRF indoor unit.

When the outdoor unit [Rated total heating capacity](#) (above) is autosized this ratio is used to scale the heating capacity of the outdoor unit to the [Rated total cooling capacity](#) of the outdoor unit regardless of the capacities of the DX heating coils in the indoor units.

The zone indoor units heating coils are also sized using this ratio unless they are hard-sized in the indoor unit.

When the VRF outdoor unit is not autosized, the sum of the VRF terminal unit coils (autosized or not) and the VRF outdoor unit capacity are used to calculate the combination ratio for both cooling and heating. For a fully autosized system, the combination ratio = 1, but when the system is not fully autosized, the resulting sizes of the terminal unit coils and the outdoor unit are used to calculate a combination ratio.

Rated heating COP

This field defines the heating coefficient of performance at rated conditions. The heating coefficient of performance includes compressor power and condenser fan power. This COP value does not include impacts due to the supply air fan. The nominal heat pump heating COP must be greater than 0.

Minimum outdoor temperature in heating mode

This field defines the minimum outdoor temperature allowed for heating operation. Below this temperature, heating is disabled. If this field is left blank, the default value is -20°C.

Maximum outdoor temperature in heating mode

This field defines the maximum outdoor temperature allowed for heating operation. Above this temperature, heating is disabled. If this field is left blank, the default value is 16°C.

Heating performance curve outdoor temperature type

This choice field defines the outdoor temperature type used for all performance curves. The valid choices are:

- **1-Wet-bulb temperature.**
- **2-Dry-bulb temperature** and

The default value is **1-Wet-bulb temperature**. Manufacturers will typically provide heating performance data as a function of outdoor air wet-bulb temperatures. This means that the performance (e.g., capacity and energy input ratio) curves will use outdoor wet-bulb temperature as one of the independent variables. At times, manufacturers will only provide performance data as a function of outdoor dry-bulb temperatures. In this case, all performance curves shall be developed using outdoor dry-bulb temperature and this field shall be selected as **2-Dry-bulb temperature**.

Heating Capacity Ratio Modifier Function Of Temperature Curves

Use single or multiple curves

The way that the heating capacity ratio modifier varies with temperature can be defined using either a *single* curve (if the heating capacity does not change dramatically with changes in outdoor conditions) or by using *multiple* (3) curves if the trend in heating capacity changes dramatically as outdoor temperature changes. Select from:

- **1-Single** where the whole relation is defined in a single heating capacity ratio modifier function of temperature curve, or,
- **2-Multiple.**

Heating capacity ratio modifier function of temperature curve

Available only when using [single](#) heating capacity ratio modifier function of temperature curve, this curve defines the heating capacity ratio modifier at all outdoor temperatures. It is a bi-quadratic equation with a weighted average indoor dry-bulb temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air dry-bulb or wet-bulb temperature as the independent variables. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified in [Heating performance curve outdoor temperature type](#).

Heating capacity ratio modifier function of low temperature curve

Available only when using [multiple](#) heating capacity ratio modifier function of temperature curves this field defines the heating capacity ratio modifier at low outdoor temperatures. This curve is a bi-quadratic equation with a weighted average indoor dry-bulb temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air dry-bulb or wet-bulb temperature as the independent variables. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified in [Heating performance curve outdoor temperature type](#).

Heating capacity ratio boundary curve

Available only when using [multiple](#) heating capacity ratio modifier function of temperature curves, this curve defines the heating capacity ratio boundary. It is a quadratic or cubic curve that defines a change in heating capacity at a specific condenser entering air dry-bulb or wet-bulb temperature as a function of indoor air dry-bulb temperature. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified in [Heating performance curve outdoor temperature type](#).

Heating capacity ratio modifier function of high temperature curve

Available only when using [multiple](#) heating capacity ratio modifier function of temperature curves, this curve defines the heating capacity ratio modifier at high outdoor temperatures. It is a bi-quadratic equation with a weighted average indoor dry-bulb temperature and condenser entering air dry-bulb or wet-bulb temperature as the independent variables. Since manufacturers may provide performance data using either outdoor dry-

bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified in [Heating performance curve outdoor temperature type](#).

Heating Energy Input Ratio (EIR) Modifier Function Of Temperature Curves

The heating energy input ratio (EIR) modifier is a factor used to adjust the rated EIR (reciprocal of [Rated heating COP](#)) as outdoor temperature changes.

Use single or multiple curves

The way that the heating EIR varies with temperature can be defined using either a *single* curve (if the heating EIR does not change dramatically with changes in outdoor conditions) or by using *multiple* (3) curves if the trend in heating EIR changes dramatically as outdoor temperature changes. Select from:

- **1-Single** where the whole relation is defined in a single heating EIR function of temperature curve, or,
- **2-Multiple.**

Heating energy input ratio (EIR) modifier function of temperature curve

Available only when using a [single](#) heating EIR curve, this curve defines the heating EIR modifier at *all* outdoor temperatures. It is a bi-quadratic equation with a weighted average indoor dry-bulb temperature and condenser entering air dry-bulb or wet-bulb temperature as the independent variables. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified by [Heating performance curve outdoor temperature type](#).

Heating energy input ratio (EIR) modifier function of low temperature curve

Available only when using [multiple](#) heating EIR curves, this curve defines the heating EIR modifier at low outdoor temperatures. It is a bi-quadratic equation with a weighted average indoor dry-bulb temperature and condenser entering air dry-bulb or wet-bulb temperature as the independent variables. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified by [Heating performance curve outdoor temperature type](#).

Heating energy input ratio (EIR) boundary curve

Available only when using [multiple](#) heating EIR curves, the heating EIR boundary curve is a quadratic or cubic equation that defines a change in heating energy at a specific condenser entering air dry-bulb or wet-bulb temperature as a function of indoor air wet-bulb temperature. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified by [Heating performance curve outdoor temperature type](#).

Heating energy input ratio (EIR) modifier function of high temperature curve

Available only when using [multiple](#) heating EIR curves, the heating EIR modifier at high temperature curve is a bi-quadratic equation with a weighted average indoor dry-bulb temperature and condenser entering air dry-bulb or wet-bulb temperature as the independent variables. Since manufacturers may provide performance data using either outdoor dry-bulb or wet-bulb temperatures, either of these temperature types may be used for heating performance curves as specified by [Heating performance curve outdoor temperature type](#).

Heating Energy Input Ratio Modifier Function Of Part-Load Ratio Curves

Heating energy input ratio (EIR) modifier function of low part-load ratio curve

This curve defines how the heating energy input ratio (EIR) modifier varies as a function of part-load ratio when PLR is less than or equal to 1. It is a linear, quadratic, or cubic equation with heating part-load ratio used as the independent variable. The heating energy input ratio modifier curve is normalized to 1 at a part-load ratio of 1 and is used only when the part-load ratio is less than or equal to 1.

Heating energy input ratio (EIR) modifier function of high part-load ratio curve

This curve defines how the heating energy input ratio (EIR) modifier varies as a function of part-load ratio when PLR is greater than 1. It is a linear, quadratic, or cubic equation with heating part-load ratio used as the independent variable. The heating energy input ratio modifier curve is normalized to 1 at a part-load ratio of 1 and is used only when the part-load ratio is greater than 1.

Other Curves

Heating combination ratio correction factor curve

This field defines the heating combination ratio (CR) correction factor curve for combination ratios greater than or equal to 1. The combination ratio is defined as the total rated indoor heating capacity divided by the rated heat pump heating capacity. The curve is either quadratic or cubic and uses the minimum value of x in the curve object to determine the maximum part-load ratio which is linearly proportional to capacity (i.e., the minimum value of x in the curve object must be ≥ 1). The output of this curve provides a multiplier (>1) which is applied to the Nominal Heat Pump Heating Capacity. Between a combination ratio of 1 and the curve's minimum value of x, the multiplier is linearly interpolated. For combination ratio's less than 1, capacity is directly proportional to part-load ratio and this curve will not be used. If this field is left blank, the Cooling Combination Ratio Correction factor will be used.

Heating part-load fraction correlation curve

This field defines the heating part-load fraction correlation curve. This curve is used to define the cycling losses when the condenser's compressors cycle on and off. The compressor cycles when the indoor to outdoor heating capacity ratio is less than the [Minimum heat pump part-load ratio](#).

VRF Outdoor Unit - Heat Recovery Tab

Refer to the [VRF heat pump model](#) section in the engineering reference for details on the transition from cooling only mode to heat recovery mode.

General

Heat pump waste heat recovery

Check this checkbox if heat recovery is to be enabled and the heat pump can independently cool and heat different zones. Otherwise the heat pump is only able to either cool or heat for any given time step.

All of the rest of the fields described below are only available when this checkbox is checked.

Tip: Refer to the [VRF heat pump model](#) section in the engineering reference for details on the transition from cooling only/heating only mode to heat recovery mode.

Minimum outdoor temperature in heat recovery mode

This field defines the minimum outdoor dry-bulb temperature allowed for heat recovery operation (in °C or °F). Below this temperature, heat recovery is disabled. This input must be greater than the larger of the minimum outdoor temperature in cooling or heating mode. This system may still operate in cooling or heating only mode when outdoor temperatures are below the minimum outdoor temperature in heat recovery mode.

Maximum outdoor temperature in heat recovery mode

This field defines the maximum outdoor dry-bulb temperature allowed for heat recovery operation (in °C or °F). Above this temperature, heat recovery is disabled. This input must be less than the smaller of the maximum outdoor temperature in cooling or heating mode. This system may still operate in cooling or heating only mode when outdoor temperatures are above the maximum outdoor temperature in heat recovery mode.

Cooling

Initial heat recovery cooling capacity fraction

This field defines the fraction of cooling capacity available when the system transitions from cooling only operation to simultaneous cooling and heating. It is common for the cooling capacity to decrease before the system recovers. The default value is 0.5 which means that there is a 50% reduction in cooling capacity at the start of heat recovery mode. The system will recover according to the time constant entered in [Heat recovery cooling capacity time constant](#). If the transition period is not to be modelled, this input field should be set to 1. This input is only used when the system changes from cooling only mode to heat recovery mode.

Heat recovery cooling capacity time constant

This field defines the cooling capacity time constant, in hours, used to model the time it takes for the system to change from cooling only operation to simultaneous cooling and heating. Total response time is defined as 5 time constants. If the transition period will not be modelled, the [Initial heat recovery cooling capacity fraction](#) field must be set to 1. This input is only used when the system changes from cooling only mode to heat recovery mode.

Initial heat recovery cooling energy fraction

This field defines the fraction of cooling energy consumed when the system transitions from cooling only operation to simultaneous cooling and heating. It is common for the cooling energy to drop before the system recovers. The default value is 1 i.e. no change in energy at the start of heat recovery mode. If the transition period is not to be modelled, this field must be set to 1. This input is only used when the system changes from cooling only mode to heat recovery mode.

Heat recovery cooling energy time constant

This field defines the cooling energy time constant, in hours, used to model the time it takes for the system to transition from cooling only operation to simultaneous cooling and heating. Total response time is defined as 5 time constants. The default value is 0.0. If the transition period is not to be modelled, the [Initial heat recovery cooling capacity fraction](#) field should be set to 1. This input is only used when the system changes from cooling only mode to heat recovery mode.

Heat recovery cooling capacity modifier curve

This curve defines the way cooling capacity modifier varies when heat recovery mode is active. This modifier is used as a multiplier for available cooling capacity, and when in heat recovery mode, it is usually less than 1. This curve is either a bi-quadratic equation using weighted average indoor temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air temperature as the independent variables or a cubic curve based on part-load ratio. This performance curve can be used to describe the cooling capacity modifier as either a constant (e.g., temperature or part-load ratio term coefficients are 0), a cooling modifier that varies with either indoor temperature and/or outdoor temperature, or part-load ratio (e.g., one or more temperature or part-load ratio term coefficients are 0), or varies with both indoor and outdoor temperatures or part-load ratio (e.g., all temperature or part-load ratio term coefficients are non-zero).

This modifier is applied only when heat recovery mode is active and when the system changes from cooling only mode to heat recovery mode.

Heat recovery cooling energy modifier curve

This curve defines the way cooling energy modifier varies when heat recovery mode is active. This modifier is used as a multiplier for operating cooling energy, and when in heat recovery mode, it is usually greater than 1. This curve is a bi-quadratic equation using weighted average indoor temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air temperature as the independent variables. This performance curve can be used to describe the cooling energy modifier as either a constant (e.g., temperature term coefficients are 0), a cooling energy modifier that varies with indoor and/or outdoor temperatures (e.g., one or more temperature term coefficients are 0), or varies with both indoor and outdoor temperatures (e.g., temperature term coefficients are non-zero). This modifier is applied only when heat recovery mode is active.

This input is only used when the system changes from cooling only mode to heat recovery mode.

Heating

Initial heat recovery heating capacity fraction

This field defines the fraction of heating capacity available when the system changes from heating only operation to simultaneous heating and cooling. It is common for the heating capacity to decrease before the system recovers. If this field is left blank, a default value of 0.5 is used (50% reduction in heating capacity at the start of heat recovery mode). The system will recover according to the time constant entered in [Heat recovery heating capacity time constant](#) below.

If the transition period will not be modelled, enter 1 here.

This input is only used when the system changes from heating only mode to heat recovery mode.

Heat recovery heating capacity time constant

This field defines the heating capacity time constant, in hours, used to model the time it takes for the system to transition from cooling only operation to simultaneous cooling and heating. Total response time is defined as 5 time constants. If this field is left blank, a default value of 0.083 is used. If the transition period will not be modeled, the Initial Heat Recovery Heating Capacity Fraction field must be set to 1.

This input is only used when the system changes from heating only mode to heat recovery mode.

Initial heat recovery heating energy fraction

This field defines the fraction of heating energy consumed when the system changes from heating only operation to simultaneous heating and cooling. It is common for the heating energy to decrease before the system recovers. The default setting is 0.5 which means that there is 50% reduction in heating energy at the start of heat recovery mode. The system will recover according to the time constant entered in [Heat recovery heating capacity time constant](#) (above).

If the transition period is not to be modelled, this input should be set to 1.

This input is only used when the system changes from heating only mode to heat recovery mode.

Heat recovery heating energy time constant

This field defines the heating energy time constant, in hours, used to model the time it takes for the system to change from cooling only operation to simultaneous cooling and heating. Total response time is defined as 5 time constants. The default value is 0.0.

If the transition period is not to be modelled, the [Initial heat recovery heating capacity fraction](#) field must be set to 1.

This input is only used when the system changes from heating only mode to heat recovery mode.

Heat recovery heating capacity modifier curve

This curve defines the heating capacity modifier when heat recovery mode is active. This modifier is used as a multiplier for available heating capacity, and when in heat recovery mode, this modifier is usually less than 1. This curve is a bi-quadratic equation using weighted average indoor temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air temperature as the independent variables. This performance curve can be used to describe the heating capacity modifier as either a constant (e.g., temperature term coefficients are 0), a heating modifier that varies with indoor and/or outdoor temperatures (e.g., one or more temperature term coefficients are 0), or varies with both indoor and outdoor temperatures (e.g., temperature term coefficients are non-zero).

This input is only used when the system changes from heating only mode to heat recovery mode.

Heat recovery heating energy modifier curve

This curve defines the heating energy modifier when heat recovery mode is active. This modifier is used as a multiplier for operating heating energy, and when in heat recovery mode, this modifier is usually greater than 1. This curve is a bi-quadratic equation using weighted average indoor temperature (i.e., the indoor terminal units weighted average inlet temperatures) and condenser entering air temperature as the independent variables. This performance curve can be used to describe the heating energy modifier as either a constant (e.g., temperature term coefficients are 0), a heating energy modifier that varies with indoor and/or outdoor temperatures (e.g., one or more temperature term coefficients are 0), or varies with both indoor and outdoor temperatures (e.g., temperature term coefficients are non-zero).

This modifier is applied only when heat recovery mode is active and when the system changes from heating only mode to heat recovery mode.

Fluid To Fluid Heat Exchanger

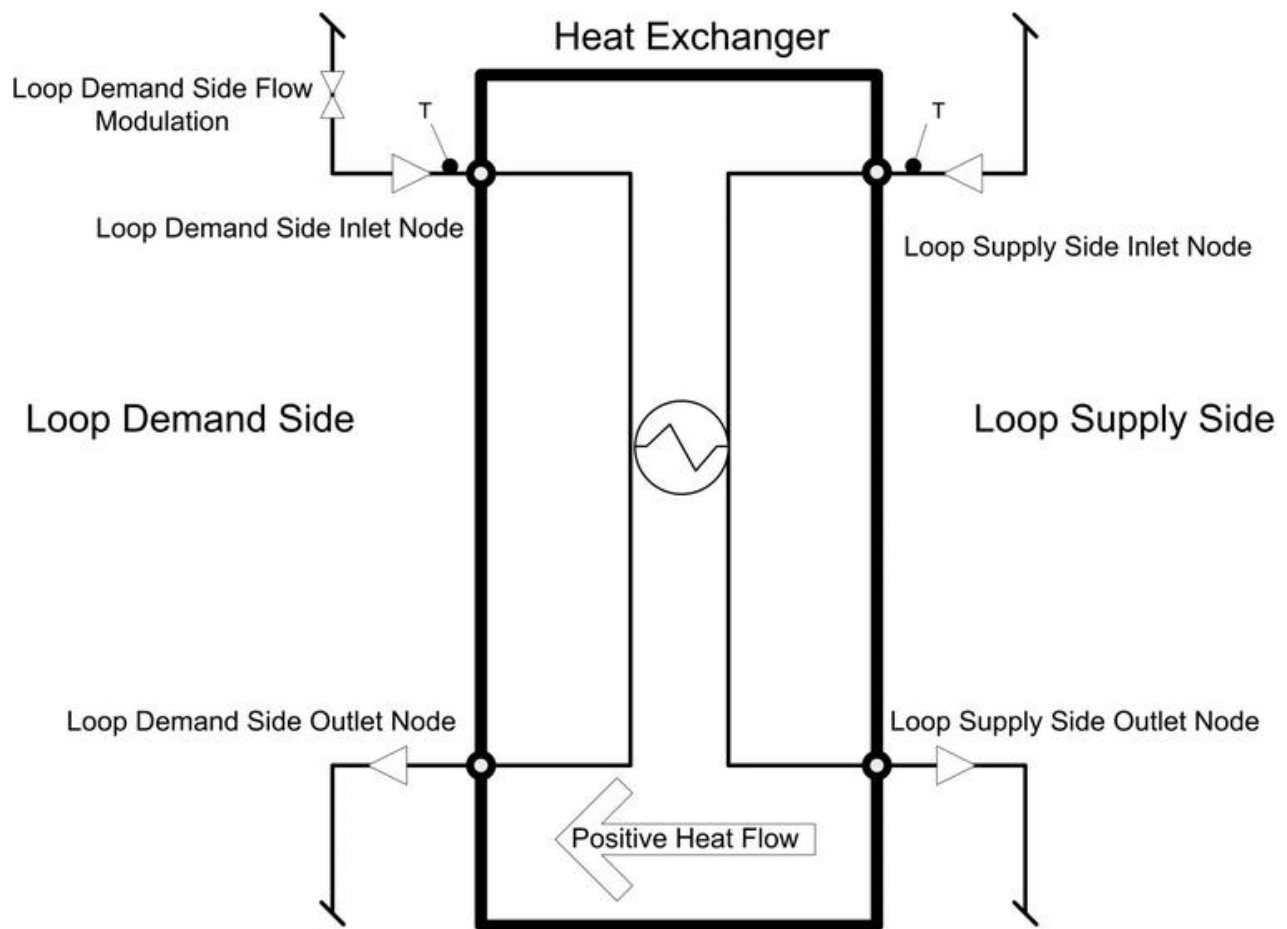


HeatExchanger:FluidToFluid

Used in:

- **Plant and condenser loops, supply and demand side**

The fluid-to-fluid heat exchanger component (aka heat exchanger or HX) is designed to couple the supply side of one plant or condenser loop to the demand side of another plant or condenser loop. The heat exchanger is a fairly general component and can be configured for use in a situation where any two loops need to be connected together. The only constraints are that one side must be connected to the supply side of one loop and the other side connected to the demand side of a different loop.



Plant Fluid to Fluid Heat Exchanger

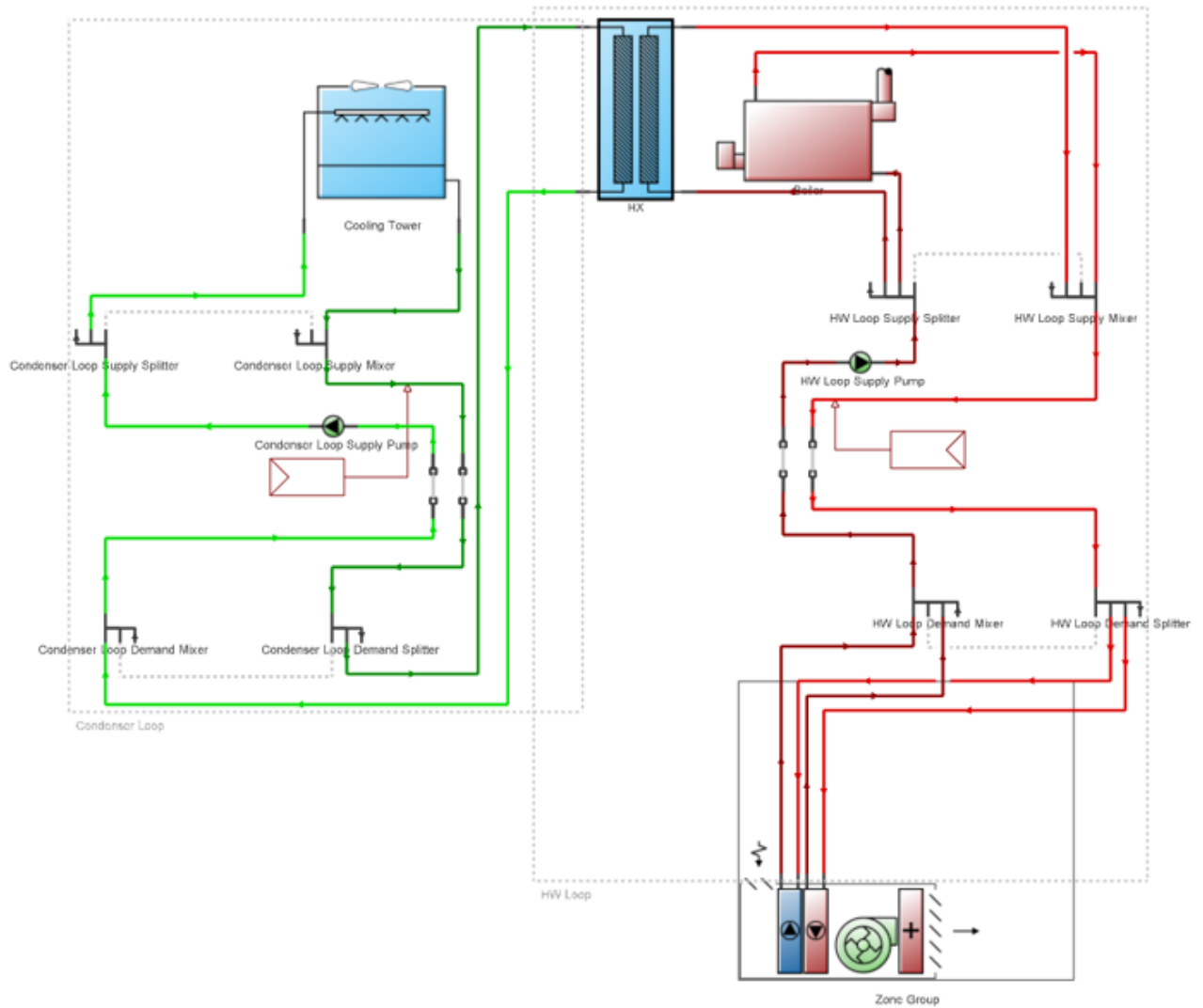
Note: The adopted convention is that the heat exchanger must always be placed in a supply sub-loop.

Because the heat exchanger is intended to be generic, its two sides are distinguished by the nature of the loop side being connected. One side is called the “Loop Supply Side” to indicate that side of the heat exchanger connected to the supply side of a loop. The other side is called the “Loop Demand Side” to indicate it is connected to the demand side of a loop. The heat exchanger is intended to act as a supply component for the loop connected to it as the “Loop Supply Side” and as a demand component for the loop connected to it as the “Loop Demand Side”. From the point of view of the heat exchanger model itself, the Loop Demand Side fluid serves as the source/sink to supply heating/cooling to the fluid in the Loop Supply Side. The “Loop Supply Side” or “Loop Demand Side” connections of the heat exchanger are automatically enabled and disabled depending on which sub-loop is currently active to facilitate correct connection, i.e. to connect the supply side of the heat exchanger you must first navigate to the supply sub-loop in which the heat exchanger has been located and similarly navigate to the demand sub-loop of the loop to which the demand side of the heat exchanger is to be connected.

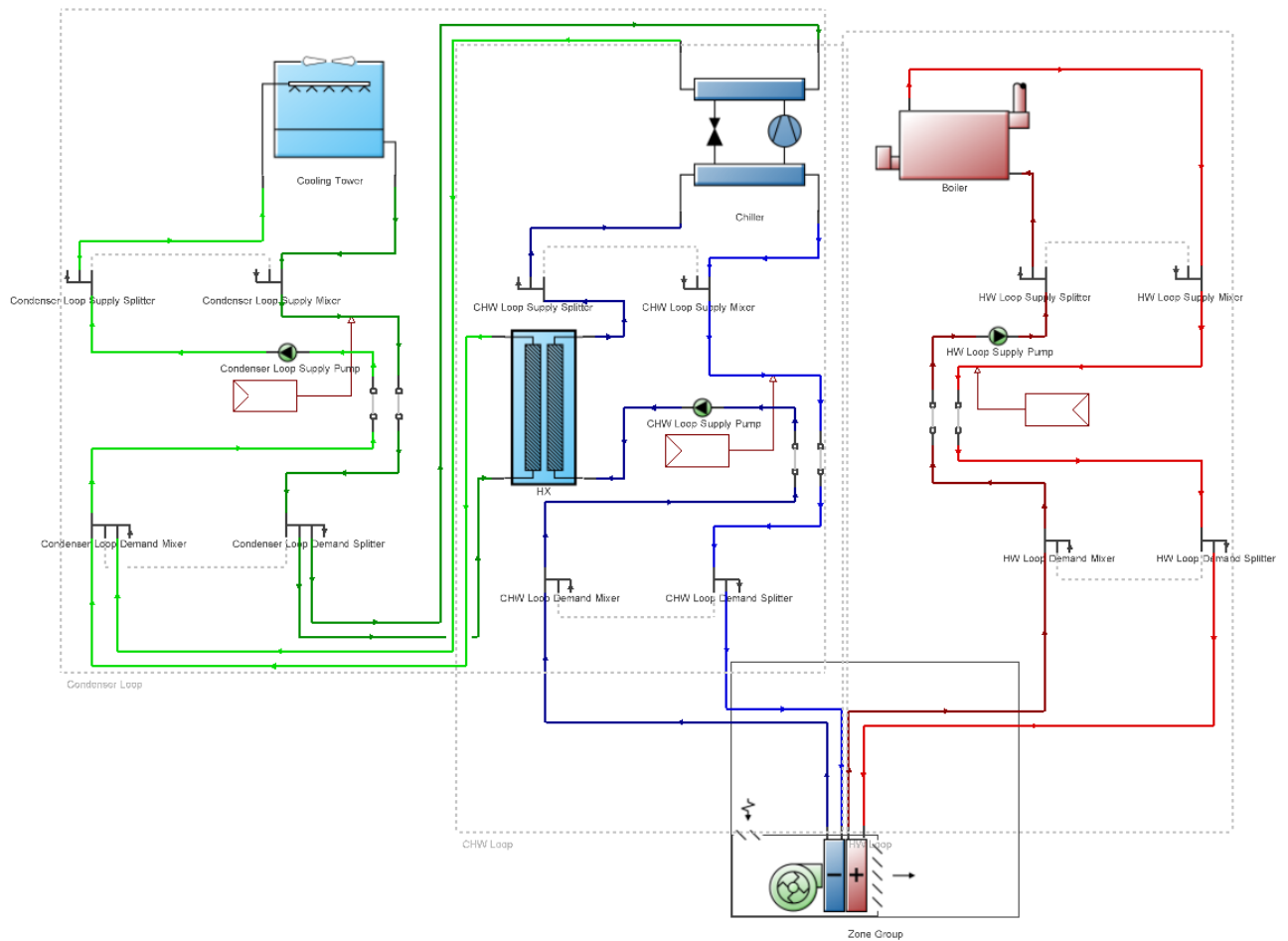
Various options are available for the heat exchanger component. The heat transfer between the two sides can be modelled using one of seven different models for different types of heat exchangers. Heat transfer is calculated and reported using a sign convention from chilled water applications where positive heat transfer indicates energy was extracted from the Loop Supply Side and added to the Loop Demand Side. A range of options are available for controlling the heat exchanger. One general type of control is “On/off” where the flow through the heat exchanger is either fully on or fully off. Another type of control is “Modulated” where the flow through the Loop Demand Side is controlled to try and meet a target setpoint or load on the Loop Supply Side.

Applications

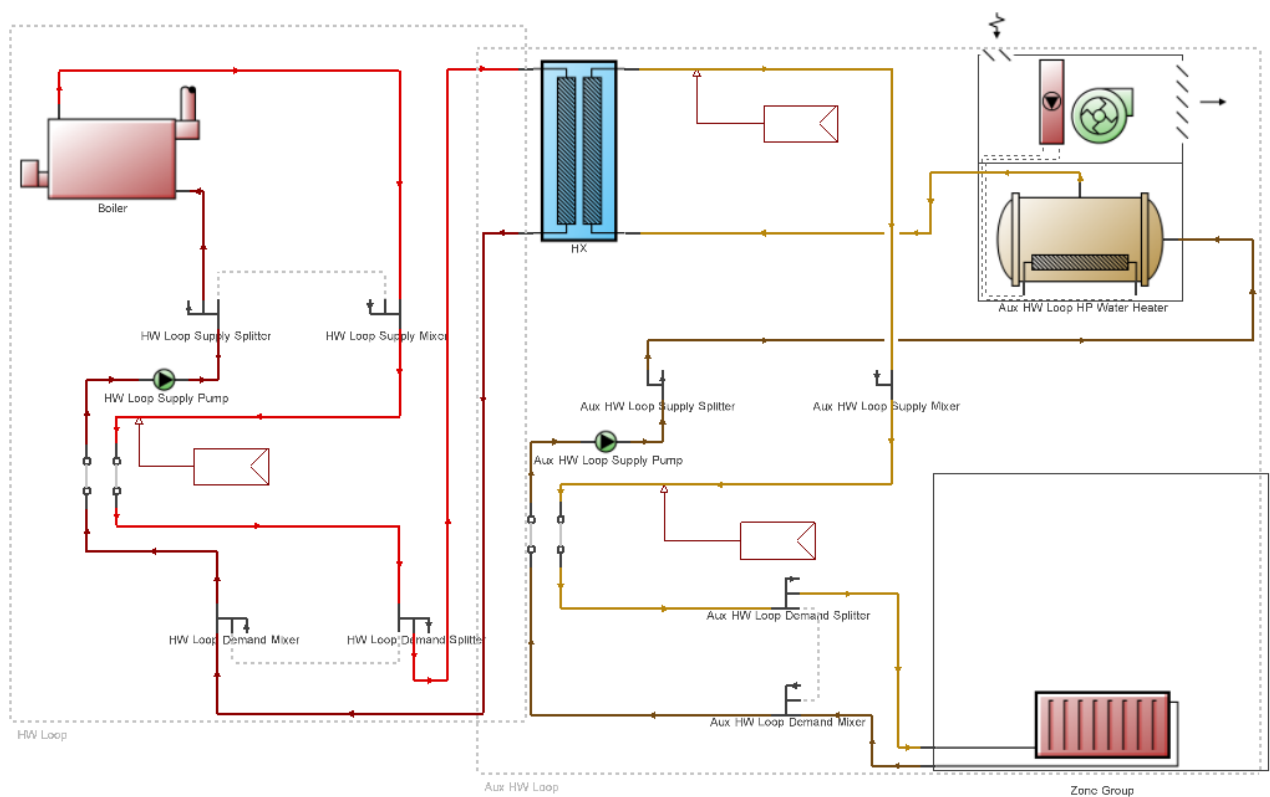
The fluid to fluid heat exchanger can be used for a wide variety of applications including chilled water, hot water, condenser, ground source, primary-secondary systems, etc.



HX Example Application: Zone Ground Source Heat Pumps



HX Example Application: Waterside Economiser



HX Example Application: Air Source Heat Pump with Supplementary Heating

The diagram illustrates a hydronic heating system with three main sections: HW Loop, Zone Group, and Aux HW Loop.

- HW Loop:** This loop starts with a **Boiler** at the top left. Water flows from the boiler through a **HW Loop Supply Splitter** to a **HW Loop Supply Pump**. From the pump, the water travels through a **HW Loop Demand Mixer** and a **HW Loop Demand Splitter** to the **Zone Group**.
- Zone Group:** This section contains a **Radiator** labeled **80°C**. It is connected to the HW Loop and the Aux HW Loop.
- Aux HW Loop:** This loop is connected to the Zone Group via a **HX** (Heat Exchanger). Water flows from the Zone Group through the HX to the **Aux HW Loop Supply Splitter**, then through a **Aux HW Loop Supply Pump** and a **Aux HW Loop Demand Mixer** to a **Water Outlet Group**. The Aux HW Loop is labeled **55°C**.

Additional components and flow paths include:

- Water Outlet Group:** The final destination for the water from the Aux HW Loop.
- Flow Direction:** Indicated by arrows on the pipes, showing a clockwise flow in each loop.
- Control Valves:** Various valves are shown at the mixers and splitters to regulate flow.

- 1091 -

General

Name

Enter a unique identifying name for this heat exchanger.

Operation

Availability schedule

Select an availability schedule to be used for supervisory control of the device. The heat exchanger is available for use whenever the schedule value is greater than zero. The heat exchanger is not available whenever the schedule value is zero or less.

Heat Exchanger Settings

Loop demand side design flow rate

This field specifies the design flow rate (in m³/s or gal/min) of the hydronic fluid passing through the heat exchanger on the Loop Demand Side. This field is autosizable. When autosized, this design flow rate is set to equal the design flow rate for the Loop Supply Side.

Loop supply side design flow rate

This field specifies the design flow rate (in m³/s or gal/min) of the fluid passing through the heat exchanger on the Loop Supply Side. This field is autosizable. When autosized, this design flow rate is set equal to the overall design flow rate of the loop that is connected. If a [sizing factor](#) is entered, then it is used to multiply this design flow rate.

Heat exchange model type

This alpha field identifies the nature of heat exchanger. Heat exchanger model type is specified by one of the following four key word choices:

- **1-Counter flow**. Specifies a counter-flow shell and tube heat exchanger. The effectiveness is calculated using a counter-flow shell and tube heat exchanger correlation.
- **2-Cross flow both unmixed**. Specifies a single-pass, cross-flow heat exchanger. The effectiveness is calculated using a cross-flow heat exchanger correlation for both streams unmixed.
- **3-Cross flow both mixed**. Specifies a single-pass, cross-flow heat exchanger. The effectiveness is calculated using a cross-flow heat exchanger correlation for both streams mixed.
- **4-Cross flow supply mixed demand unmixed**. Specifies a single-pass, cross-flow heat exchanger. The effectiveness is calculated using a cross-flow heat exchanger correlation for flow mixed on the Loop Supply side and flow unmixed on the Loop Demand Side.
- **5-Cross flow supply unmixed demand mixed**. Specifies a single-pass, cross-flow heat exchanger. The effectiveness is calculated using a cross-flow heat exchanger correlation for flow unmixed on the Loop Supply side and flow mixed on the Loop Demand Side.
- **6-Parallel flow**. Specifies a parallel-flow shell and tube heat exchanger. The effectiveness is calculated using a parallel-flow shell and tube heat exchanger correlation.
- **7-Ideal**. Specifies an ideal heat exchanger. The effectiveness is set to 1 and the specified UA is ignored. The heat transfer rate is calculated as the maximum possible heat transfer rate.

Heat exchanger U-Factor times area value

This autosizable numerical field is used to specify the overall U-Factor Times Area (UA) (in W/K or Btu/h-°F) for use in the calculation of the heat exchanger effectiveness using the appropriate D-NTU correlation. If **7-Ideal** is specified as the heat exchanger type, the effectiveness will be set to 1.0. When set to **Autosize** Heat Exchanger U-Factor Times Area Value is calculated based on an effectiveness of 1.0 where capacity is such that the temperatures in the plant sizing for the two loops can be maintained.

Control type

This option is used to specify how the heat exchanger is to be controlled during operation. Different applications for connecting two loops will require different control behavior and different control options are needed depending on the desired behaviour. There are a range of options to choose from:

- **1-Cooling differential on/off.** This control mode is applicable to situations where the Loop Demand Side can provide useful cooling to the Loop Supply Side. This mode is similar to **8-Cooling setpoint on/off** except that it ignores any cooling setpoint and its control is based only on the temperature difference between Loop Demand Side and the Loop Supply Side. The inlet temperatures must differ by more than the value set in the [Minimum temperature difference to activate heat exchanger](#) field for the heat exchanger to operate.
- **2-Uncontrolled on.** This control mode is applicable to situations where the heat exchanger is passively running all the time and always transfers as much heat as possible between the fluid streams. Note that the HX will only request flow on the Loop Demand Side when there is non-zero flow into the heat exchanger on the Loop Supply Side.
- **3-Operation scheme modulated.** This control mode is applicable to situations where the heat exchanger is controlled by a plant loop [operation scheme](#). When using this control mode the heat exchanger must be listed in the [list of equipment](#) on the Plant Equipment Operation tab of the Plant or Condenser loop and it serves as a supply component. The operation scheme will dispatch a load request to the heat exchanger which it will try meet by conditioning the fluid stream connected as the Loop Supply Side. If the heat exchanger could exceed the load request, then the flow through the fluid stream connected as the Loop Demand Side will be modulated to just meet the load request.
- **4-Operation scheme on/off.** This control mode is applicable to situations where the heat exchanger is controlled by a plant loop [operation scheme](#). When using this control mode the heat exchanger must be listed in the [list of equipment](#) on the Plant Equipment Operation tab of the Plant or Condenser loop and it serves as a supply component. The operation scheme will dispatch a load request to the heat exchanger which it will use as an on/off signal to decide if the heat exchange should run or not. If it runs, it will run at full capacity and may exceed the load request.
- **5-Heating setpoint modulated.** This control mode is applicable to situations where the Loop Demand Side can provide useful heating to the Loop Supply Side. For this control type, an SPM must be placed on the supply side outlet of the HX which is used to define the heating setpoint temperature for the HX. If the setpoint and inlet temperatures are such that heat exchanger could transfer heat from the Loop Demand Side to the Loop Supply Side to meet the heating setpoint, then the heat exchanger will run. The inlet temperatures must differ by more than the value set in [Minimum temperature difference to activate heat exchanger](#) for the heat exchanger to operate. If the heat exchanger could overshoot the setpoint, then the flow through the fluid stream connected as the Loop Demand Side will be modulated to just meet the setpoint.
- **6-Heating setpoint on/off.** This control mode is applicable to situations where the Loop Demand Side can provide useful heating to the Loop Supply Side. For this control type, an SPM must be placed on the supply side outlet of the HX which is used to define the heating setpoint temperature for the HX. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Demand Side to the Loop Supply Side to meet the heating setpoint, then the heat exchanger will run. The inlet temperatures must differ by more than the value set in [Minimum temperature difference to activate heat exchanger](#) for the heat exchanger to operate. If it runs, it will run at full capacity and may overshoot the setpoint.
- **7-Cooling setpoint modulated.** This control mode is applicable to situations where the Loop Demand Side can provide useful cooling to the Loop Supply Side. For this control type, an SPM must be placed on the supply side outlet of the HX which is used to define the cooling setpoint temperature for the HX. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Supply Side to the Loop Demand Side to meet the cooling setpoint, then the heat exchanger will run. The inlet temperatures must differ by more than the value set in [Minimum temperature difference to activate heat exchanger](#) for the heat exchanger to operate. If the heat exchanger could undershoot the setpoint, then the flow through the fluid stream connected as the Loop Demand Side will be modulated to just meet the setpoint.
- **8-Cooling setpoint on/off.** This control mode is applicable to situations where the Loop Demand Side can provide useful cooling to the Loop Supply Side. For this control type, an SPM must be placed on the supply side outlet of the HX which is used to define the cooling setpoint temperature for the HX. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Supply Side to the Loop Demand Side to meet the cooling setpoint, then the heat exchanger will run. The inlet temperatures must differ by more than the value set in [Minimum temperature difference to activate heat exchanger](#) for the heat exchanger to operate. If it runs, it will run at full capacity and may undershoot the setpoint. This control mode is suitable for use with waterside economisers.

- **9-Dual deadband setpoint modulated.** This control mode is applicable to situations where the Loop Demand Side can provide either useful cooling or heating to the Loop Supply Side. For this control type, an SPM must be placed on the supply side outlet of the HX which is used to define the dual deadband setpoint temperature for the HX. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Demand Side to the Loop Supply Side to meet the lower setpoint, then the heat exchanger will run. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Supply Side to the Loop Demand Side to meet the high setpoint, then the heat exchanger will run. The inlet temperatures must differ by more than the value set in [Minimum temperature difference to activate heat exchanger](#) for the heat exchanger to operate. If the heat exchanger could overshoot the lower setpoint, or undershoot the higher setpoint, then the flow through the fluid stream connected as the Loop Demand Side will be modulated to just meet the deadband setpoint.
- **10-Dual deadband setpoint on/off.** This control mode is applicable to situations where the Loop Demand Side can provide either useful cooling or heating to the Loop Supply Side. For this control type, an SPM must be placed on the supply side outlet of the HX which is used to define the dual deadband setpoint temperature for the HX. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Demand Side to the Loop Supply Side to meet the lower setpoint, then the heat exchanger will run. If the setpoints and inlet temperatures are such that heat exchanger could transfer heat from the Loop Supply Side to the Loop Demand Side to meet the high setpoint, then the heat exchanger will run. The inlet temperatures must differ by more than the value set in [Minimum temperature difference to activate heat exchanger](#) for the heat exchanger to operate. If the heat exchanger runs, it will run at full capacity and may overshoot the lower setpoint or undershoot the higher setpoint.

Minimum temperature difference to activate heat exchanger

This value specifies the temperature tolerance used in control decisions in (DeltaC or DeltaF). Whenever the control logic needs to compare two temperatures, the value entered in this field is used as a threshold for comparisons.

Heat transfer metering end use type

This option specifies how the metering for heat transfer will be accounted in summary reports with respect to end uses. Although the heat exchanger consumes no energy that needs to be metered, there are also meters for heat transfers that apply to this heat exchanger model. The nature of the end use may vary depending on the application. The available choices are:

- **1- Free cooling,**
- **2-Heat recovery,**
- **3-Heat rejection,**
- **4-Heat recovery for cooling,**
- **5-Heat recovery for heating**
- **6-Loop to loop.**

Temperature Limits

Operation minimum temperature limit

When the **Minimum temperature limit** checkbox is checked this field is used to provide supervisory control of the heat exchanger. If either of the inlet temperatures are below this limit (°C or °F), the heat exchanger will not operate.

Operation maximum temperature limit

When the **Maximum temperature limit** checkbox is checked this field is used to provide supervisory control of the heat exchanger. If either of the inlet temperatures are above this limit (°C or °F), the heat exchanger will not operate.

Sizing Factor

Sizing factor

When the Sizing factor checkbox is checked this field can be used to modify the results of autosize calculations. This sizing factor is used for this heat exchanger and modifies sizing results by multiplying them by the factor entered here. This factor is applied to the Loop Supply Side Design Flow rate and in turn affects the heat exchanger UA and the loop demand side flow rate which are derived from that flow rate. This allows fine control over the size of the heat exchanger when using autosize in those fields.

Chilled Water Storage



ThermalStorage:ChilledWater:Mixed
ThermalStorage:ChilledWater:Stratified

Used in:

Supply side of Chilled Water Loops

Chilled water storage tanks are typically placed on the supply side of a primary chilled water loop in parallel with one or more chillers. Operation is controlled through chiller and storage tank setpoints along with corresponding plant operation schemes. The way that the cooling load is shared and the timing of storage charging is typically controlled through a Scheduled SPM on the chiller outlet node.

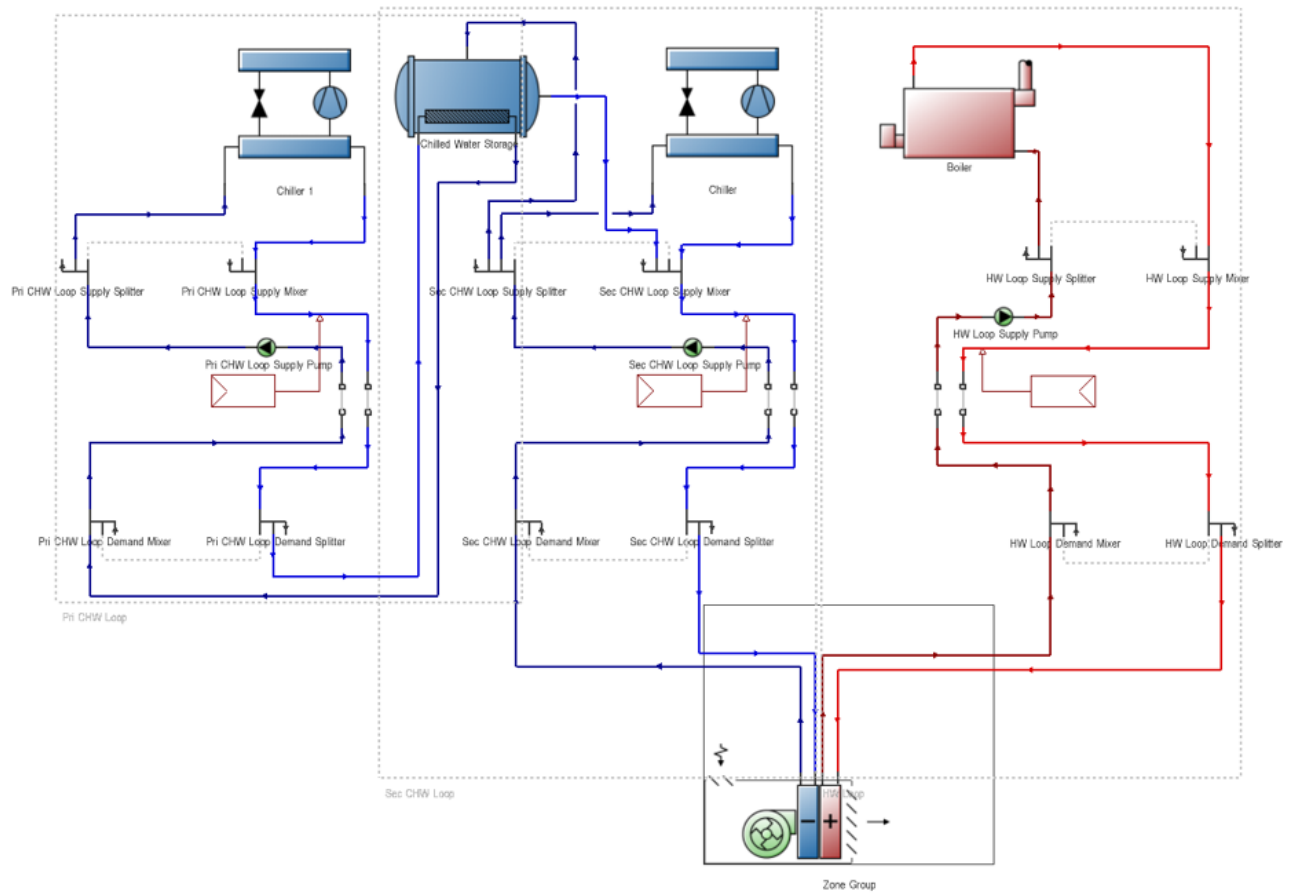
Benefits of Chilled Water Storage Systems

- Chilled water is normally generated using off-peak energy supply, stored in chilled water storage tanks then distributed for use during peak hours. The economic benefits of chilled water storage systems therefore generally rely on lower off-peak electrical rates.
- Chillers running at lower ambient temperatures which are typical of off-peak use tend to have higher efficiencies.
- It is usually possible to install smaller chillers, heat rejection equipment and pumps when stored chilled water is used to meet peak cooling loads.

Chilled Water Loop Configurations

Primary-Secondary CHW loops

A typical application of a chilled water storage tank would be on the supply side of a primary chilled water loop in parallel with one or more chillers as shown below.

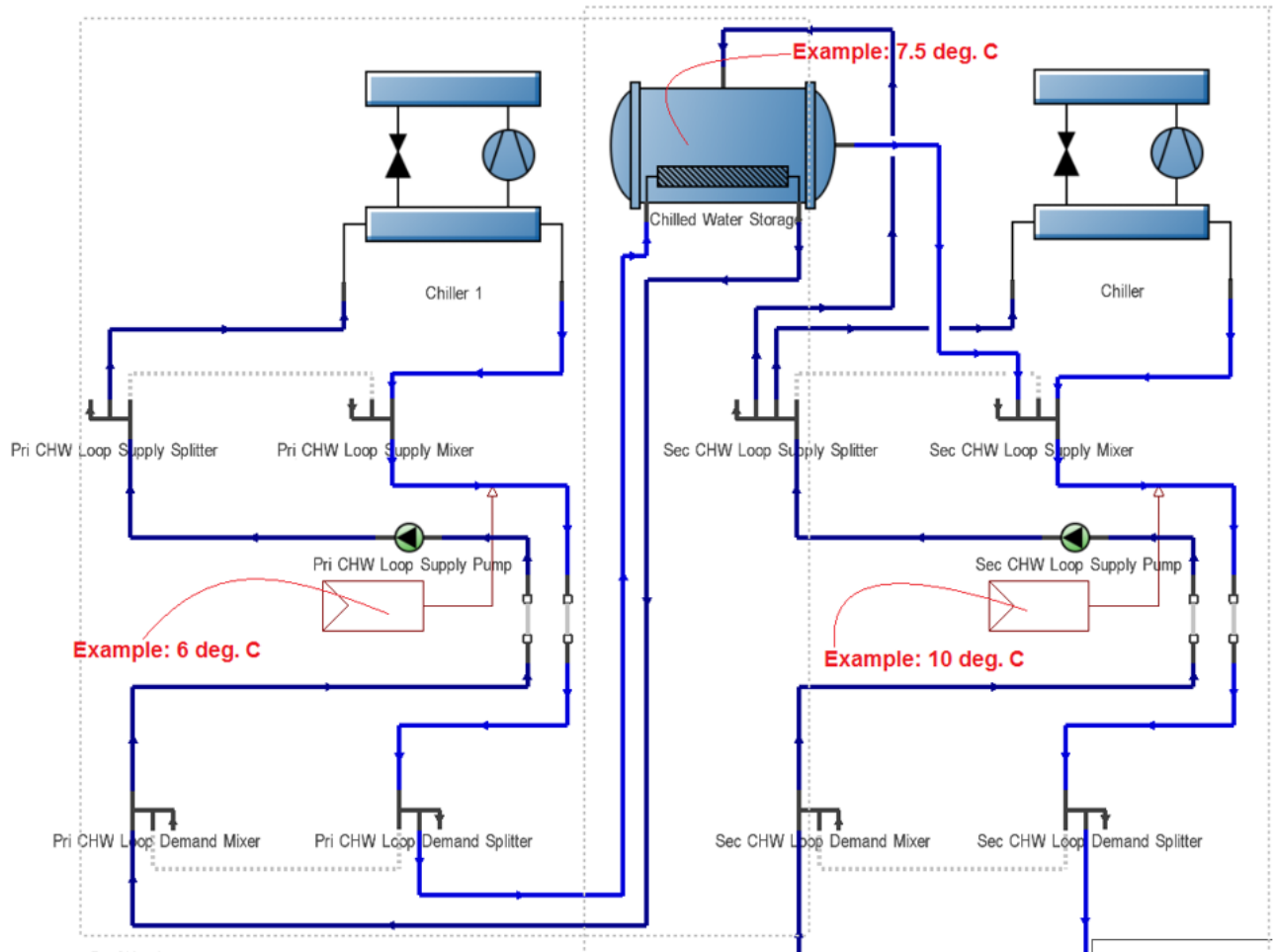


Four-pipe Fan-coil unit air-cooled chillers with chilled water storage

Chilled water storage requires a difference in setpoint temperature between the source (charge) side and use side:

- Source side (lower): Primary CHW loop setpoint temperature, e.g. 6°C (set in the loop SPM);
- Use side (higher): Chilled water storage setpoint temperature, e.g. 7.5°C (set on the chilled water storage dialog).

In the example below the secondary chilled water loop setpoint temperature is set to 10°C, but it could be set lower, down to 6°C.



Chilled water storage system temperature settings

The chiller provides 6°C chilled water while the chilled water storage tank setpoint is 7.5°C (cut-in temperature is $7.5^{\circ}\text{C} + 2.0^{\circ}\text{C} = 9.5^{\circ}\text{C}$). The secondary chiller SPM temperature should be set the temperature of the chilled water to be supplied to the equipment such as cooling coils etc.

Charging is normally done at night-time while tariffs are lower. This can be defined through the chilled water storage source side availability schedule (see screenshot below). Likewise, use side time can be set as required, typically for day-time only, through the use side availability schedule. The use side availability can be set using the On 24/7 schedule in many cases.

Edit Chilled water storage -

Chilled water storage

Chilled Water Storage

General

Name: CHW Loop 1 Chilled Water Storage

Type: 1-Mixed

Nominal cooling capacity (W): 50000.0

Tank recovery time (hr): 1.50

Tank volume (m3): 10.00

Temperature Settings

Setpoint temperature schedule: Chilled water storage cooling setpoint

Deadband temperature difference (deltaC): 2.00

Minimum temperature limit (°C): 1.00

Ambient Heat Transfer Settings

Ambient temperature indicator: 1-Schedule

Ambient temperature schedule: Water heater ambient temperature schedule

Heat gain coefficient from ambient temperature (W/K): 5.000

Use-Side Settings

Use side heat transfer effectiveness: 1.00

Use side availability schedule: On 24/7

Use side design flow rate (m3/s): Autosize

Source-Side Settings

Source side heat transfer effectiveness: 1.000

Source side availability schedule: Chilled water storage charge schedule

Source side design flow rate (m3/s): Autosize

Model data <admin>

Help

Schedules

Data Report (Not Editable)

General

Chilled water storage cooling setpoint

Source: DesignBuild

Category: <General>

Region: General

Schedule type: 2-Compact S

Profiles

Schedule: Compact

On, Any Number, Through: 12/31, For: AllDays, Until: 24:00, 7.5

Chilled water storage setpoint temperature

Chilled water storage setpoint temperature and source side availability schedule

Above settings provide tank cooling setpoint temp of 7.5degC, charging at night only and use side available anytime.

Plant equipment operation scheme should give a higher priority for chilled water storage, and a lower priority for chiller (screenshot below).

Plant loop

General **Plant Equipment Operation**

General

Number of schemes: 1

Scheme 1

Operation type: 2-Cooling load

Operation schedule: On 24/7

Number of ranges: 1

Range 1

Lower limit (W): 0.00

Upper limit (W): 1000000000000000.00

Equipment

☒ Chiller

Priority: 2

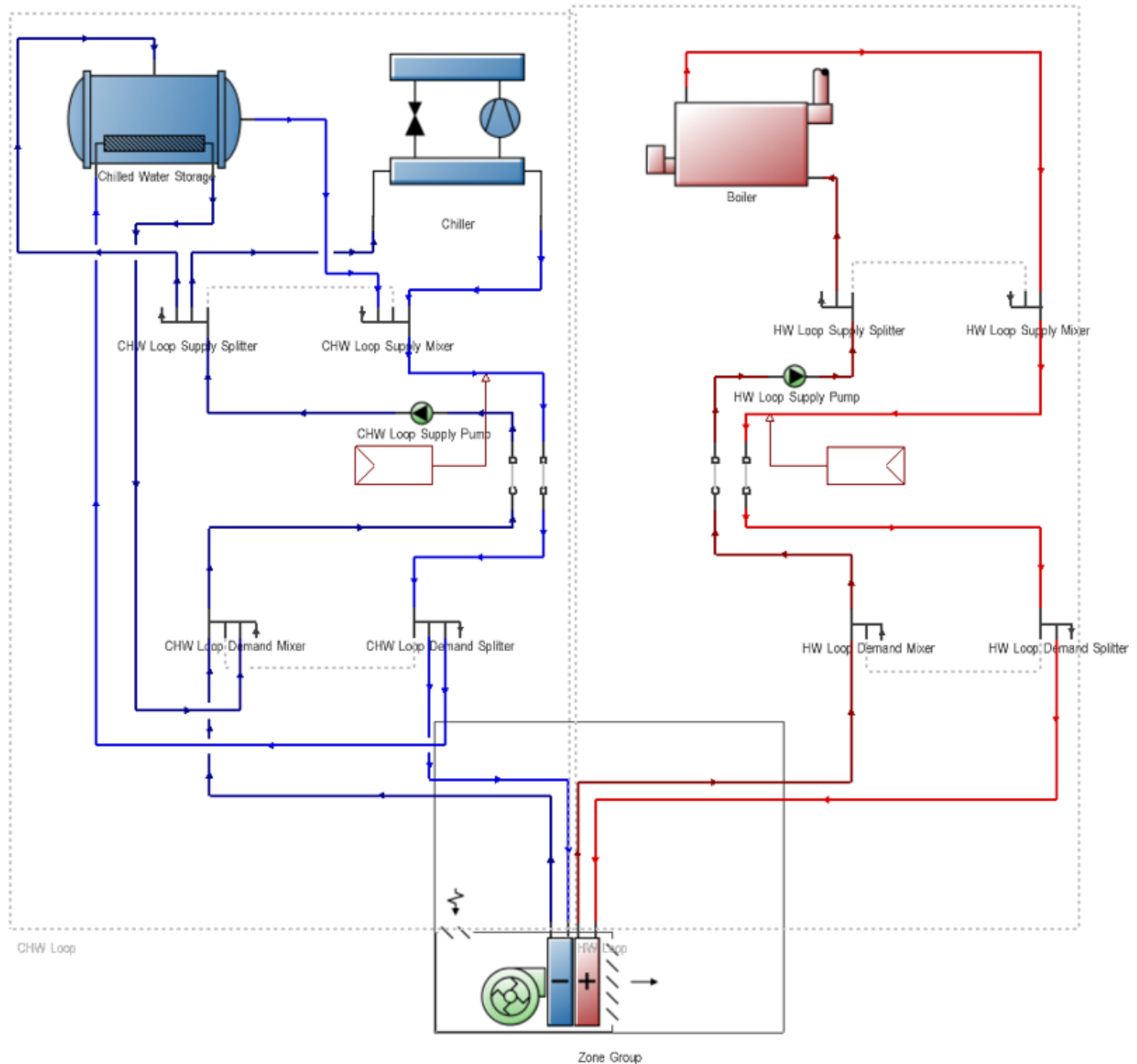
☒ Chilled Water Storage

Priority: 1

Secondary chilled water loop plant equipment operation settings

Single chilled water loop

This section covers use of chilled water storage tanks in a single parallel chiller arrangement as shown in the screenshot below.



Chilled water storage in parallel with the chiller

As with the primary-secondary loop configuration, the single chilled water loop needs a difference in setpoint temperatures between source (charge) side and use side:

- Source side (lower): CHW loop setpoint temperature, e.g. 6°C (through loop SPM);
- Use side (higher): Chilled water storage setpoint temperature, e.g. 7.5°C (set on chilled water storage dialog).

For a single chilled water loop configuration, the plant equipment operation needs to set up differently to the primary secondary loops described above. The day-time and night-time operation is defined separately using one item of equipment for each scheme (see screenshot below).

Plant loop	
General	Plant Equipment Operation
General	
Number of schemes	2
Scheme 1	
Operation type	2-Cooling load
Operation schedule	Night time 19:00 - 07:00
Number of ranges	1
Range 1	
Lower limit (W)	0.00
Upper limit (W)	1000000000000000.00
Equipment	
☒ Chiller	
Priority	1
☐ Chilled Water Storage	
Scheme 2	
Operation type	2-Cooling load
Operation schedule	8:00 - 18:00 Mon - Sat
Number of ranges	1
Range 1	
Lower limit (W)	0.00
Upper limit (W)	1000000000000000.00
Equipment	
☐ Chiller	
☒ Chilled Water Storage	
Priority	1

Plant equipment operation settings for single chilled water loop arrangement

Types of Chilled Water Storage Tanks

There are currently two types of chilled water storage tank:

- [1-Mixed](#),
- [2-Stratified](#).

The **1-Mixed** type simulates a well-mixed, single-node water tank. The **2-Stratified** type simulates a stratified, multi-node water tank. Both storage tanks share similar features, such as thermal losses to the zone, however, each type has its advantages which may make one more appropriate than the other depending on the application.

Advantages of **1-Mixed** type:

- requires less input than the stratified tank
- faster execution time than the stratified tank

Advantages of **2-Stratified** type:

- better modelling of thermal storage applications which rely on stratification to improve heat transfer performance.

Chilled Water Storage - Mixed



ThermalStorage:ChilledWater:Mixed

Used in:

Supply side of Chilled Water Loops

The Chilled water storage mixed object analytically solves the differential equation governing the energy balance of the water tank. The chilled water is “used” by drawing from the “Use Side” of the water tank. The tank is indirectly charged by circulating cold water through the “Source Side” of the water tank. Control is based on cycling flow through the source side. When the tank temperature rises above a “cut-in” temperature, source side flow is requested. Source side flow will continue until the tank is cooled to below the tank set point or “cut-out” temperature.

For heat gains from the ambient environment, the ambient air temperature can be taken from a schedule, a zone, or the exterior. When used with a zone, the skin gains are removed from the zone heat balance as negative internal heat gains.

See also:

- [Chilled Water Storage Overview](#)
- [Chilled Water Storage - Stratified](#)

General

Name

The name of the Chilled water storage tank.

Type

Select the type of Chilled water storage tank from the list:

- **1-Mixed,**
- [2-Stratified.](#)

For a discussion of the applications, advantages and disadvantages of the two water heater types see the [Chilled Water Storage overview](#) topic.

Nominal cooling capacity

The Nominal cooling capacity describes the typical cooling capacity that the chilled water tank will provide (in W or Btu/h). Since this is a passive device, the actual cooling capacity depends on water temperatures and flow rates. However, this field is used to describe the chilled water tank’s nominal cooling capacity for supervisory control where plant operation schemes require a cooling capacity to model how equipment is dispatched.

Tank recovery time

This field is used to autosize the Source Side Design Flow Rate. The field is used if the [Source side design flow rate](#) is set to Autosize and the water tank’s source side is on the demand side of a plant loop. This is the time, in hours, that the chilled water tank is to be indirectly cooled from 14.4°C to 9.0°C using the [Design loop exit temperature](#) of in the associated Plant loop.

Tank volume

The volume of the storage tank (in m3 or gal).

Temperature Settings

Setpoint temperature schedule

Select the schedule which specifies the chilled water temperature setpoint (in °C). This setting is also known as the “cut-out” temperature.

Deadband temperature difference

The delta temperature difference (in Delta °C or Delta °F) between the setpoint and the “cut-in” temperature at which the storage tank will request cooling. In other words, the “cut-in” temperature is Setpoint + Deadband.

Minimum temperature limit

The temperature (in °C or °F) at which the tank water becomes too cold. No source side flow is allowed when the tank temperature is below this limit. The minimum temperature must be lower than the setpoint temperature at all times.

Ambient Heat Transfer Settings

Ambient temperature indicator

The **Ambient temperature indicator** specifies how the ambient air temperature will be indicated. Select from:

- **1-Schedule** where the **Ambient temperature schedule** provides the values for ambient temperature.
- **2-Zone**, where the zone air temperature of the zone specified in the **Ambient temperature zone** provides the ambient temperature, or,
- **3-Outdoors**, where the outdoor dry-bulb air temperature provides the ambient temperature.

Ambient temperature schedule

Select schedule which specifies the ambient air temperature around the tank for skin gains. This field is only used if the **1-Schedule** Ambient temperature indicator is used.

Ambient temperature zone

Select the zone specifying the ambient air temperature around the tank for skin gains. This field is only used if the **2-Zone** Ambient temperature indicator is used.

Heat gain coefficient from ambient temperature

The gain coefficient from the ambient air temperature (in W/K or Btu/h-°F) . This coefficient is often referred to as the UA for the overall tank thermal performance with respect to heat gains from the tank's skin.

Use Side Settings

Use side heat transfer effectiveness

This field specifies the heat transfer effectiveness between the use side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the use side water and the tank water. If the effectiveness is lower, then the use side outlet water temperature will not be as cold as the tank water, simulating a heat exchanger.

Use side availability schedule

Select the availability schedule for the use side of the water tank. If the schedule's value is 0.0, then the use side is not available and flow will not be requested. If the schedule's value is not equal to 0.0 (usually 1 is used), the use side is available.

Use side design flow rate

This autosizable setting is used to specify the design flow rate through the use side of the chilled water tank (in m³/s or gal/min). For autosized tanks, sizing results are reported in the EIO file.

Source Side Settings

Source side heat transfer effectiveness

This field specifies the heat transfer effectiveness between the source side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the source side water and the tank water. If the effectiveness is lower, then the source side outlet water temperature will not be as cold as the tank water, simulating a heat exchanger.

Source side availability schedule

This field contains the name of an availability schedule for the source side of the water tank. If the schedule's value is 0.0, then the source side is not available and flow will not be requested. If the schedule's value is not equal to 0.0 (usually 1 is used), the source side is available.

Source side design flow rate

This autosizable setting is used to specify the design flow rate through the source side of the chilled water tank (in m³/s or gal/min). Sizing results are reported in the EIO file.

Chilled Water Storage - Stratified



ThermalStorage:ChilledWater:Stratified

Used in:

Supply side of Chilled Water Loops

The Stratified Chilled Water Storage component divides the water tank into multiple nodes of equal volume. The nodes are coupled by vertical conduction effects, internode fluid flow, and temperature inversion mixing. It simultaneously solves the differential equations governing the energy balances on the nodes using a numerical method. The system timestep is divided into many small substeps that allow the simulation to capture events that occur on a very short time scale. The chilled water is "used" by drawing from the "Use Side" of the water tank. The tank is indirectly charged by circulating cold water through the "Source Side" of the water tank. Control is based on cycling flow through the source side. When the tank temperature rises above a "cut-in" temperature, source side flow is requested. Source side flow will continue until the tank is cooled to below the tank set point or "cut-out" temperature.

For heat gains from the ambient environment, the ambient air temperature can be taken from a schedule, a zone, or the exterior. When used with a zone, the skin gains are removed from the zone heat balance as negative internal heat gains.

See also:

- [Chilled Water Storage Overview](#)
- "Chilled Water Storage - Mixed" on page 1101

General

Name

The name of the Chilled water storage tank.

Type

Select the type of Chilled water storage tank from the list:

- [1-Mixed](#).
- [2-Stratified](#).

For a discussion of the applications, advantages and disadvantages of the two water heater types see the [Chilled Water Storage overview](#) topic.

Nominal cooling capacity

The Nominal cooling capacity describes the typical cooling capacity that the chilled water tank will provide (in W or Btu/h). Since this is a passive device, the actual cooling capacity depends on water temperatures and flow rates. However, this field is used to describe the chilled water tank's nominal cooling capacity for supervisory control where plant operation schemes require a cooling capacity to model how equipment is dispatched.

Tank recovery time

This field is used to autosize the Source Side Design Flow Rate. The field is used if the [Source side design flow rate](#) is set to Autosize and the water tank's source side is on the demand side of a plant loop. This is the time, in hours, that the chilled water tank is to be indirectly cooled from 14.4°C to 9.0°C using the [Design loop exit temperature](#) of in the associated Plant loop.

Tank volume

The volume of the storage tank (in m3 or gal).

Tank height

The height of the tank (in m or in).

For the [2-Horizontal cylinder](#) tank shape (see below) the height of the tank is the measure in the axial direction, i.e., the height if you were to stand the cylinder up on its end. This field is autosizable.

Tank shape

The tank shape determines the size and skin losses of the stratified nodes. There are two options:

- [1-Vertical Cylinder](#), which describes most upright cylindrical tanks.
- [2-Horizontal Cylinder](#), which describes a few speciality water tanks and some large commercial storage tanks. This shape implies that the tank is divided into nodes of equal mass, but not equal height.

Temperature Settings

Setpoint temperature schedule

Select the schedule which specifies the chilled water temperature setpoint (in °C). This setting is also known as the "cut-out" temperature.

Deadband temperature difference

The delta temperature difference (in Delta °C or Delta °F) between the setpoint and the "cut-in" temperature at which the storage tank will request cooling. In other words, the "cut-in" temperature is Setpoint + Deadband.

Minimum temperature limit

The temperature (in °C or °F) at which the tank water becomes too cold. No source side flow is allowed when the tank temperature is below this limit. The minimum temperature must be lower than the setpoint temperature at all times.

Temperature sensor height

This setting is used to describe the location in the tank where the temperature is sensed for control decisions. The program will associate one of the nodes with this height and use that node's temperature for control decisions. The location is described from the bottom of the tank (in m or in).

Ambient Heat Transfer Settings

Ambient temperature indicator

The **Ambient temperature indicator** specifies how the ambient air temperature will be indicated. Select from:

- **1-Schedule** where the **Ambient temperature schedule** provides the values for ambient temperature.
- **2-Zone**, where the zone air temperature of the zone specified in the **Ambient temperature zone** provides the ambient temperature, or,
- **3-Outdoors**, where the outdoor dry-bulb air temperature provides the ambient temperature.

Ambient temperature schedule

Select schedule which specifies the ambient air temperature around the tank for skin gains. This field is only used if the **1-Schedule** Ambient temperature indicator is used.

Ambient temperature zone

Select the zone specifying the ambient air temperature around the tank for skin gains. This field is only used if the **2-Zone** Ambient temperature indicator is used.

Heat gain coefficient from ambient temperature

The gain coefficient from the ambient air temperature (in W/K or Btu/h-°F) . This coefficient is often referred to as the UA for the overall tank thermal performance with respect to heat gains from the tank's skin.

Use Side Settings

Use side heat transfer effectiveness

This field specifies the heat transfer effectiveness between the use side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the use side water and the tank water. If the effectiveness is lower, then the use side outlet water temperature will not be as cold as the tank water, simulating a heat exchanger.

Use side availability schedule

Select the availability schedule for the use side of the water tank. If the schedule's value is 0.0, then the use side is not available and flow will not be requested. If the schedule's value is not equal to 0.0 (usually 1 is used), the use side is available.

Use side design flow rate

This autosizable setting is used to specify the design flow rate through the use side of the chilled water tank (in m³/s or gal/min). For autosized tanks, sizing results are reported in the EIO file.

Use side inlet height

The height of the use side inlet to the tank (in m or in). Normally this value will be the same as or close to the height of the tank and cannot be higher than the tank height. A value of **autocalulate** sets the Use side inlet height to the height of the tank.

Use side outlet height

The height of the use side outlet from the tank (in m or in). The outlet height cannot be higher than the tank height.

Source Side Settings

Source side heat transfer effectiveness

This field specifies the heat transfer effectiveness between the source side water and the tank water. If the effectiveness is set to 1 then complete heat transfer occurs, simulating perfect mixing of the source side water and the tank water. If the effectiveness is lower, then the source side outlet water temperature will not be as cold as the tank water, simulating a heat exchanger.

Source side availability schedule

This field contains the name of an availability schedule for the source side of the water tank. If the schedule's value is 0.0, then the source side is not available and flow will not be requested. If the schedule's value is not equal to 0.0 (usually 1 is used), the source side is available.

Source side design flow rate

This autosizable setting is used to specify the design flow rate through the source side of the chilled water tank (in m³/s or gal/min). Sizing results are reported in the EIO file.

Source side inlet height

The height of the source side inlet to the tank (in m or in). The inlet height cannot be higher than the tank height.

Source side outlet height

The height of the source side outlet from the tank (in m or in). The outlet height cannot be higher than the tank height. A value of **autocalulate** sets the Source side outlet height to the height of the tank.

Inlet Mode

Inlet Mode

The inlet mode of entering fluid from the use and source sides. There are two options:

- **1-Fixed** where the fluid enters at the fixed heights specified above, or,
- **2-Seeking** where the fluid "seeks out" the stratified node that is closest to the inlet temperature and adds all flow to that node. This mode provides maximum stratification.

Stratification Settings

Additional destratification conductivity

An additional destratification conductivity (in W/m-K or Btu-in/h-ft²-°F) is added to the fluid conductivity of water (0.6 W/m-K) to account for vertical conduction effects along the inside of the tank wall, and perhaps other vertical components such as a dip tube pipe or anode rod.

Number of stratified nodes

The number of stratified nodes in the tank. There must be at least one node. The maximum number of nodes is 10, although this limit can be increased by editing the IDD.

Additional Loss Coefficient for node 1-10

An additional heat gain coefficient (in W/K or Btu/h-°F) added to the skin gains for each node to account for thermal shorting due to pipe penetrations, tank feet, and any other loss effects.

Ice Thermal Storage



ThermalStorage:Ice:Simple
ThermalStorage:Ice:Detailed

Used in:

- Chilled water loop, supply side

Ice thermal storage systems allow use of off-peak electricity to "charge" a storage tank of ice or other fluid and use that as a source of cooling to help reduce peak loads during periods of cooling demand. Two types of ice thermal storage systems can be modelled in DesignBuilder, simple and detailed, the latter option providing options to control the charge/discharge performance using curves.

General

Name

This alpha field contains the identifying name for the ice storage tank.

Type

Two types of ice thermal storage systems can be modelled in DesignBuilder:

- **1-Simple**, which is based on a simple simulation of an ice storage tank with a fixed capacity. The tank is charged or frozen in an ice-on-coil configuration where ice builds up on the outside of the tubes carrying the brine or glycol solution from the chiller. There are two discharge (melt) options, internal or external. Internal melt uses the same fluid tubes for charging and discharging. External melt uses a separate fluid path for discharge such that the outer layers of ice melt first. The ice storage model includes an implied 3-way valve to control the amount of charge/discharge based on the incoming water temperature and the outlet node setpoint temperature. The storage tank is assumed to be fully charged (full of ice) at the beginning of each environment. The tank is then allowed to charge and discharge during the warmup days of the environment. The tank is best controlled using the **4-Component Setpoint** plant operation scheme which requires that a setpoint be placed by a Setpoint manager on the ice storage Plant Outlet Node.
- **2-Detailed**, which can simulate ice storage tanks more closely based on specific manufacturers' published performance data. This is possible through the use of curve fits to simulate the performance of the ice storage unit during charging and discharging. In this implementation, both charging and discharging are a function of the fraction charged/discharged as well as the log mean temperature difference across the storage unit.

All parameters below apply to the **2-Detailed** option apart from Ice storage type below. The 1-Simple option only requires [Name](#), [Ice storage type](#) and [Capacity](#).

Ice storage type

This setting applies only to the **1-Simple** type of Ice thermal storage tank and specifies the type of ice storage tank to be modelled. There are two options:

- **1-Ice on coil internal** models ice-on-coil, internal melt.
- **2-Ice on coil external** models ice-on-coil, external melt.

Capacity

Enter the maximum amount of latent thermal storage available in the ice storage system in GJ (Giga is 10^9) or ton-hrs. The removal or addition of sensible energy from the tank is not modelled, so the tank is always assumed to be at the freezing temperature of the storage material.

Tank loss coefficient

This setting defines the loss of ice stored during a particular hour. This field is dimensionless (per hour). It is not multiplied by any temperature difference between the tank and the environment in which it might be located.

Freezing temperature of storage medium

This parameter defines the freezing/melting temperature of the ice storage medium (in °C or °F). For most tanks, this is simply 0.0°C (the default value). However, some tanks may use salts or other materials to change the freezing temperature. This can be changed using this parameter.

Thaw process indicator

This input assists in more accurate modelling of the charging process by defining how the thawing of ice takes place. There are two options for this input:

- **1-Outside melt** where the ice melts from the outside, leaving ice still on the charging surface when charging begins.
- **2-Inside melt** where the the charging process starts with a bare coil or no ice left on the charging surface even though there is still ice stored in the tank. An example of such a system is sometimes referred to as an ice-on-coil inside melt system.

For systems that have a charging process that does not vary significantly with fraction charged you can safely ignore this input by accepting the default value. The default value is **1-Outside melt**.

Operation

Availability schedule

Select the schedule that determines whether or not the ice storage system is available during a particular time period. This allows the system to be turned off during a particular season. A value of less than or equal to zero indicates that the ice storage system is not available. Any value greater than zero indicates that the system is available.

Curve Timestep

Timestep of the curve data

This field defines what timestep was used to produce the curve fits named in the previous inputs.

Note: This parameter is important because the curve fit is non-dimensional, so the data used to develop the curve fits were based on a specific length of time. In many cases, this is probably one hour or 1.0. The units for this parameter are hours.

Charging Settings

Charging curve object type

The detailed ice storage model uses the performance curves to model performance. Currently, the only two allowed curve fit types for the detailed ice storage model are:

- **1-Quadratic Linear** and
- **2-Cubic Linear** curves.

More information on this curve can be found in the section on [Curves](#).

Charging curve

Select the curve to be used to model the charging process of the detailed ice storage system.

Tip: EnergyPlus is provided with a spreadsheet-based utility, **IceStorageCurveFitTool.xlsm**, which is specifically developed to assist with generating performance curve coefficients for Ice Thermal Storage **charge** and **discharge** cycles. It allows performance data to be input for various operating conditions and automatically generates the polynomial coefficients needed for accurate equipment simulation. The tool can be found in the **PreProcess\HVACCurveFitTool** folder under the main EnergyPlus distribution folder.

Parasitic electric load during charging

The amount of parasitic electric consumption (for controls or other miscellaneous electric consumption associate with the ice storage unit itself) during the charge phase. This parameter is dimensionless and gets multiplied by the current load on the tank.

Discharging Settings

Discharging curve object type

The detailed ice storage model uses the performance curves to model performance. Currently, the only two allowed curve fit types for the detailed ice storage model are:

- **1-Quadratic Linear** and
- **2-Cubic Linear** curves.

More information on this curve can be found in the section on [Curves](#).

Discharging curve

Select the curve to be used to model the discharging process of the detailed ice storage system.

See above under [Charging curve](#) for information on the **Ice Storage Curve Fit Tool**.

Parasitic electric load during discharging

The amount of parasitic electric consumption (for controls or other miscellaneous electric consumption associate with the ice storage unit itself) during the discharge phase. This parameter is dimensionless and gets multiplied by the current load on the tank.

Parallel Or Series Configuration With Chiller

Ice thermal storage can be either in parallel or in series to the chiller depending on the modelling requirements.

The **parallel case** should normally include two chillers, with one being the main chiller to operate during peak cooling time and the other being the chiller used to charge the tank during specified off peak time as long as conditions are favourable. The main chiller works in lower priority to the ice thermal storage and tops up cooling demand once ice thermal storage reaches its maximum capacity during discharging. Generally the charging chiller can be thought of as a "dummy" component because it does not physically exist in the real building and the main and charging chillers actually represent one single chiller.

For the **series case**, the chiller can be either upstream or downstream of the ice thermal storage. In both cases the ice tank can help reduce peak chiller demand:

- Chiller upstream: chiller works first with ice thermal storage to top up when needed and chiller size can be reduced, and
- Chiller downstream: ice thermal storage will operate first with chiller to top up when required.

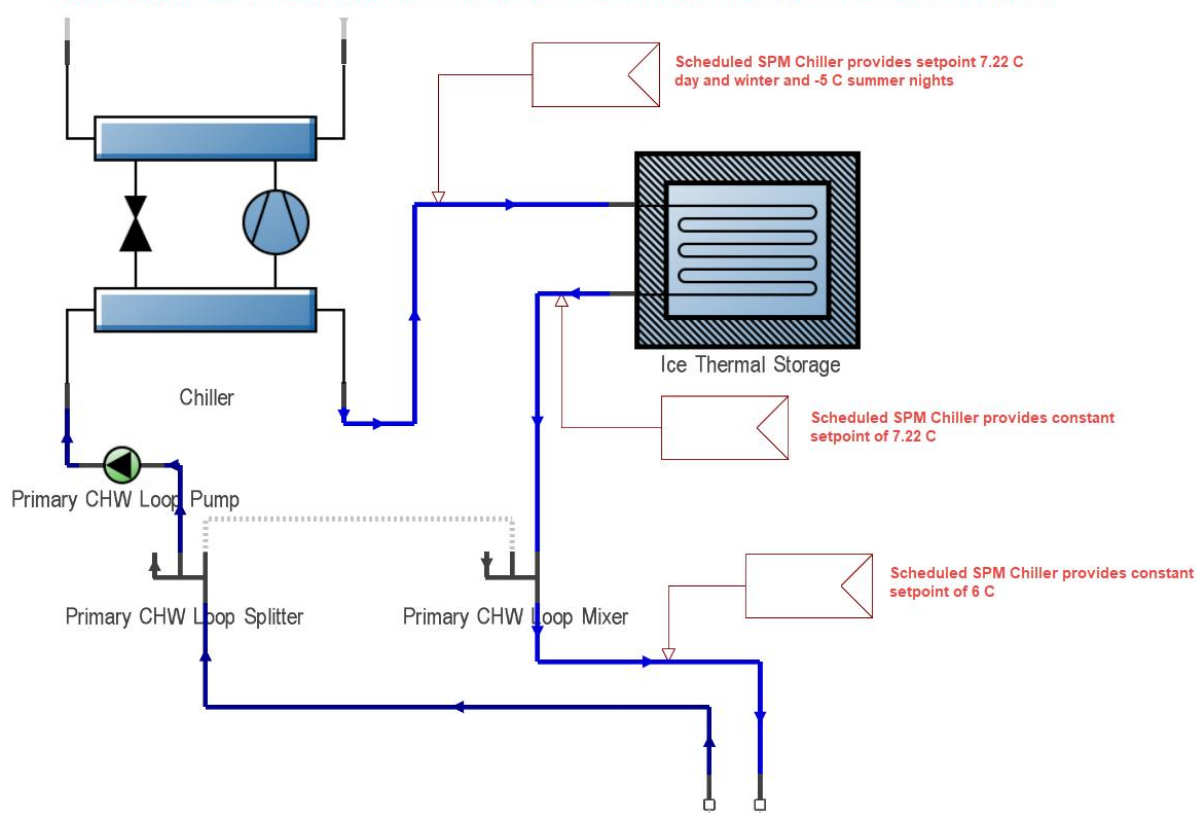
Note: for upstream chillers, the capacity of the chiller is an especially important setting. When the chiller has a high capacity, it can meet the full cooling demand and the ice thermal storage does not need to operate. In this case, to see the effect of ice thermal storage, the chiller capacity must be relatively low. However the chiller size must be high enough to allow the chiller to deliver low enough outlet CHW temperature to charge the ice thermal storage tank.

Controls And Chilled Water Plant Loop Operation Scheme Settings

The chilled water loop operation schemes include two [Scheme operation type](#) options that are suitable for modelling ice thermal storage objects:

- **4-Component Setpoint**, requires various setpoint managers, temperature schedules, etc. to allow the system to operate. As complex definitions are possible for this scheme, it allows the possibility of more accurate and flexible modelling, for instance, setting hourly variation of setpoint temperatures at the outlet of each piece of equipment. The screenshot below shows how to use Setpoint managers to control an upstream chiller with ice thermal storage series configuration with the scheme operation type set to **4-Component Setpoint**.

Series Configuration, Chiller Upstream, Plant Equipment Scheme Operation type = 4-Component setpoint



Plant loop

General Plant Equipment Operation

General

Number of schemes 1

Scheme 1

Operation type 4-Component setpoint

Operation schedule On 24/7

Equipment

☒ Chiller

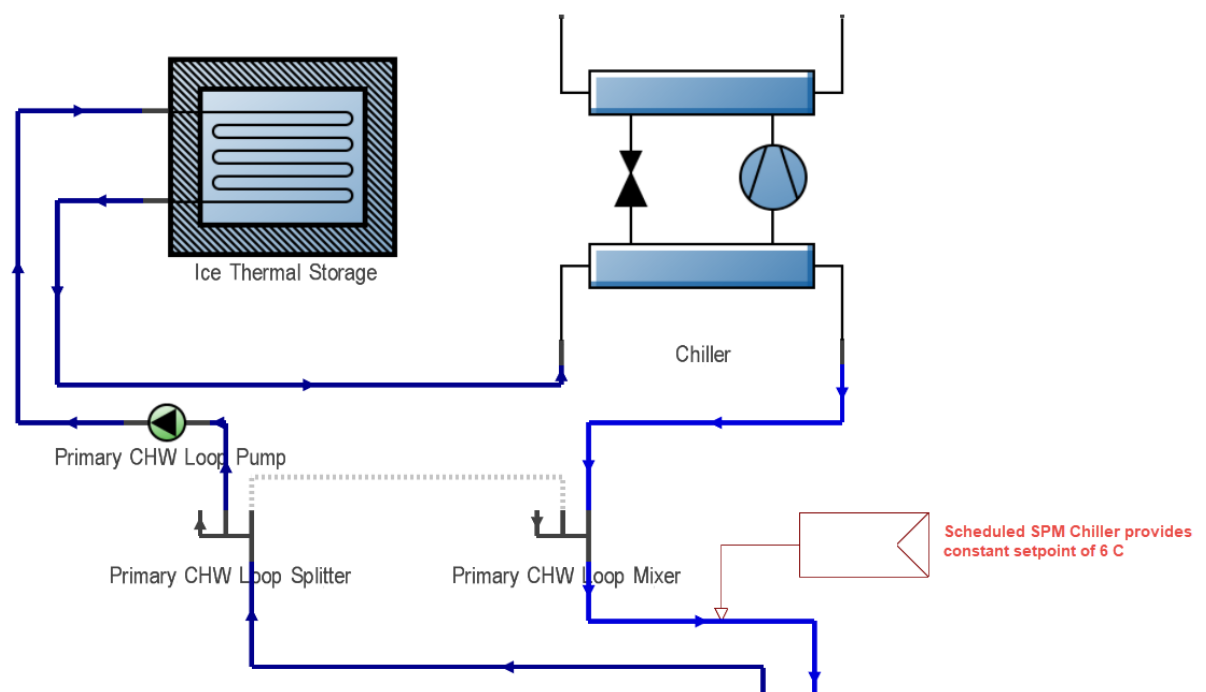
Priority 1

☒ Ice Thermal Storage

Priority 2

- 5-Energy Thermal Storage** this scheme operation type internally creates the setpoint managers required by the equipment and so allows a simple and convenient approach by eliminating the need for both setpoint manager and schedule for each piece of equipment. The only data required is a charging setpoint temperature and a discharging setpoint temperature. The screenshots below shows how to control a downstream chiller with ice thermal storage series configuration with the scheme operation type set to **5-Energy Thermal Storage**.

Series Configuration, Chiller Downstream, Plant Equipment Scheme Operation type = 5-Energy Thermal Storage



Plant loop	
General Plant Equipment Operation	
General	
Number of schemes	1
Scheme 1	
Operation type	5-Thermal energy storage
Operation schedule	On 24/7
Storage on-peak schedule	On Peak Ice Thermal Storage Schedule
Charging availability schedule	Ice Thermal Storage Charging Schedule
Non-charging chilled water temperature (°C)	7.00
Charging chilled water temperature (°C)	-5.00
Equipment	
☒ Chiller	
Priority	2
☒ Ice Thermal Storage	
Priority	1

On summer daytime

On summer nighttime

Zone Unit Data

The following categories of zone units can be added at [HVAC zone](#) level:

- "Air Distribution Units" on page 1112
- "Radiant Surface Data" on page 1135
- "Heated Floor" on page 1156
- "Chilled Ceiling" on page 1166
- "Radiative-Convective Heater data" on page 1178
- "Zone Forced Air Units" on page 1187
- "Air to Water Heat Pump / ASHP Data" on page 884

Air Distribution Units

To edit the data associated with an air distribution unit (ADU), you first need to select it by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the **Edit selected component** tool from the toolbar.

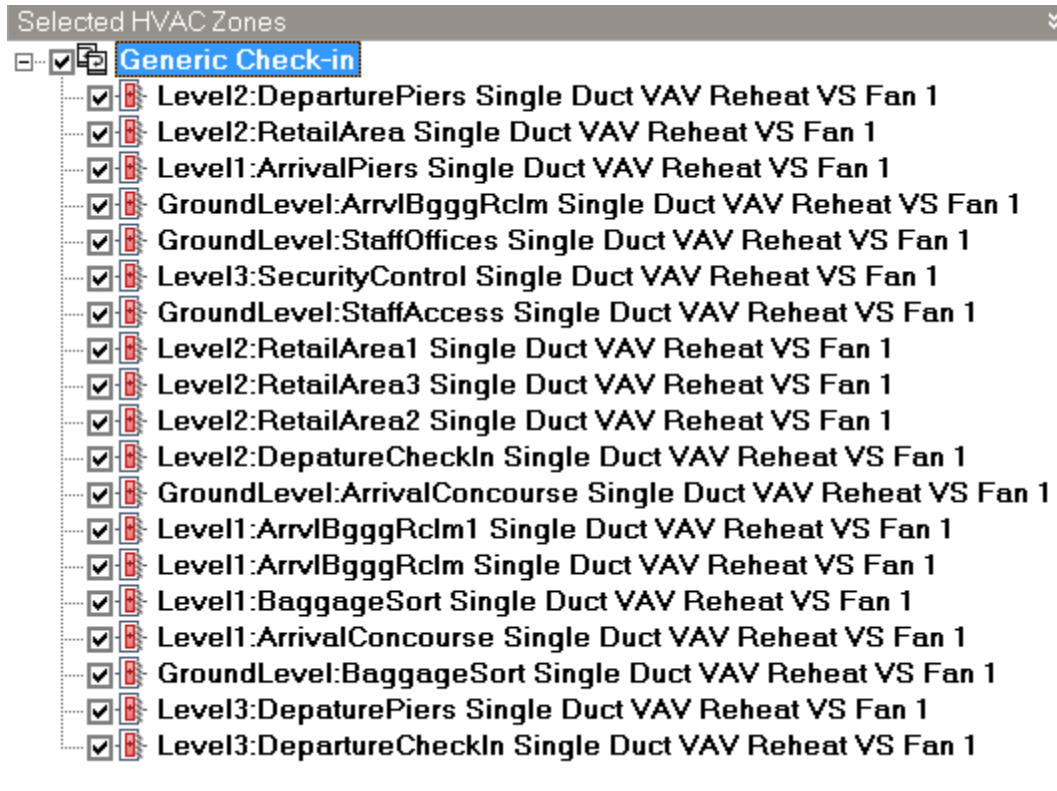
These ADU types are available:

- [Constant Air Volume with No Reheat ADU](#)
- [Constant Air Volume with Reheat ADU](#)
- [Variable Air Volume with Reheat ADU](#)
- [Variable Air Volume with Reheat and Variable Speed Fan ADU](#)
- [Variable Air Volume with No Reheat ADU](#)
- [Series Powered Induction Unit with Reheat ADU](#)
- [Parallel Powered Induction Unit with Reheat ADU](#)
- [Four Pipe Induction Unit ADU](#)
- [Dual Duct Constant Air Volume ADU](#)
- [Dual Duct Variable Air Volume ADU](#)
- [Cooled Beam](#)
- "Variable Air Volume Outdoor Air ADU" on page 1134

See the [Adding air loops and AHUs](#) tutorial

Target Tab

When editing the attributes associated with an ADU it is possible to apply the same changes to ADUs in other zones in the same HVAC Zone group. To do this select the ADUs on the **Target** tab of the ADU edit dialog as shown below.

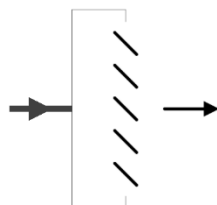


Constant Volume No Reheat ADU

Central system air is usually supplied to a zone through a terminal unit such as a single duct VAV reheat box. Sometimes, however, it is desirable to supply central system air directly to a zone without any zone level control or tempering, e.g. when used in conjunction with a unitary packaged system.

The CAV No Reheat ADU allows central system air to be supplied directly to a zone without any reheat. The data required for the unit consists of the unit name, availability (on/off) schedule, and a maximum flow rate. Settings to control outdoor air provision can also be made.

For the units to work correctly, it is important in any systems including CAV No Reheat ADUs for the sum of the maximum zone air flow rates to be equal to the maximum central system flow rate. The central air system flow rate is specified in the air loop AHU data.



Important Note: In spite of its name, the CAV No Reheat ADU can be used in HVAC zones connected to either constant or variable air volume air loops. When used in a VAV loop the supply flow provided by the unit will be fixed at the value of the **Maximum air flow rate** setting.

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum air flow rate

The design maximum volume flow rate (in m³/s or ft³/min) specified for the Direct Air ADU. This field may be auto-sized.

Outdoor Air

Control on outdoor air flow

If this option is selected, the terminal unit will increase flow as needed to meet the outdoor air requirement specified on the HVAC zone dialog. If **Outdoor air flow per person** is non-zero, then the outdoor air requirement will be calculated by the software based on the current number of occupants in the zone. At no time will the supply air flow rate exceed the value for **Maximum air flow rate**. If this option is not selected, then the terminal unit will not be controlled for outdoor air flow.

Per person ventilation rate mode

When the above **Control on outdoor air flow** checkbox is checked, this option is used to select how outdoor air ventilation rates are calculated when based on a rate per person. It can be either based on the current number of people as affected by time-varying occupancy schedules, or on the constant value for the maximum number of people. Choose one of:

- **1-Current occupancy** where outdoor air ventilation rates are based on the current number of people including the effect of time-varying occupancy schedules.
- **2-Design occupancy** where the constant value for the maximum number of people is used.

Note: this setting is only used when outdoor air ventilation rates are calculated when based on a per person rate.

Operation

Availability schedule

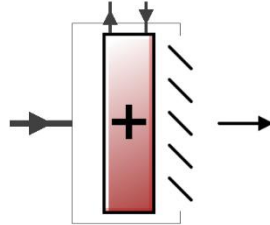
This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Constant Air Volume With Reheat ADU

Generally, a constant volume with reheat system provides cooling by means of a cooling coil located in the air supply sub-loop AHU that supplies cooling to the entire supply air volume. The cooling coil is controlled by a controller setpoint specified for the cooling coil. Zone control is accomplished by heating (reheating) the

airflow into each zone as determined by the zone thermostat. Reheat can be supplied by an electric, gas or hot water coil that tries to meet the zone demand.

The unit contains a single [heating coil](#) which can be edited to change the type to **1-Water**, **2-Electric** or **3-Gas**.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum air flow rate

The design maximum volume flow rate (in m³/sec) specified for the ADU. This field may be auto-sized.

Heating Coil

This section is visible only if the [heating coil](#) sub-component of this ADU has Type **1-Water**.

Maximum hot water flow rate

The maximum hot water volumetric flow rate (in m³/s) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Minimum hot water flow rate

The minimum hot water volumetric flow rate (in m³/s) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Advanced

Convergence tolerance

The coil is controlled by knowing the zone demand determined by the zone thermostat and setting the outlet conditions to meet this demand. For the electric and gas coils, this is set exactly since the coil model solution

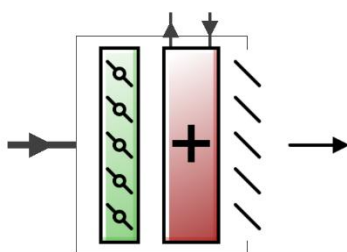
can be inverted. With the hot water coil that uses an effectiveness-NTU method, the solution cannot be inverted directly. Therefore, to determine the correct mass flow rate for the hot water the solution is solved for by iteration. The iterative solution uses an interval halving routine and needs a termination criterion that is set with the Convergence Tolerance parameter. This control offset is set to a decimal fraction of the zone demand as the criteria, i.e. 0.001. The default for the field is 0.001.

Variable Air Volume With Reheat ADU

Variable air volume (VAV) systems control the dry-bulb temperature inside a zone by varying the supply air volume instead of the air temperature. At full cooling the VAV damper is fully open supplying the specified maximum air flow rate. As the cooling load decreases, the damper closes until it reaches the minimum stop specified by the zone minimum air flow fraction.

VAV systems can be used for interior or perimeter zones with a common fan system, air temperature control, and reheating devices. The VAV concept may vary according to the VAV box locations, air temperature controls and types of heating elements. Heating can usually be provided by use of reheat coils or thermostatic baseboard.

The unit contains a single [heating coil](#) which can be edited to change the type to **1-Water**, **2-Electric** or **3-Gas**.



General

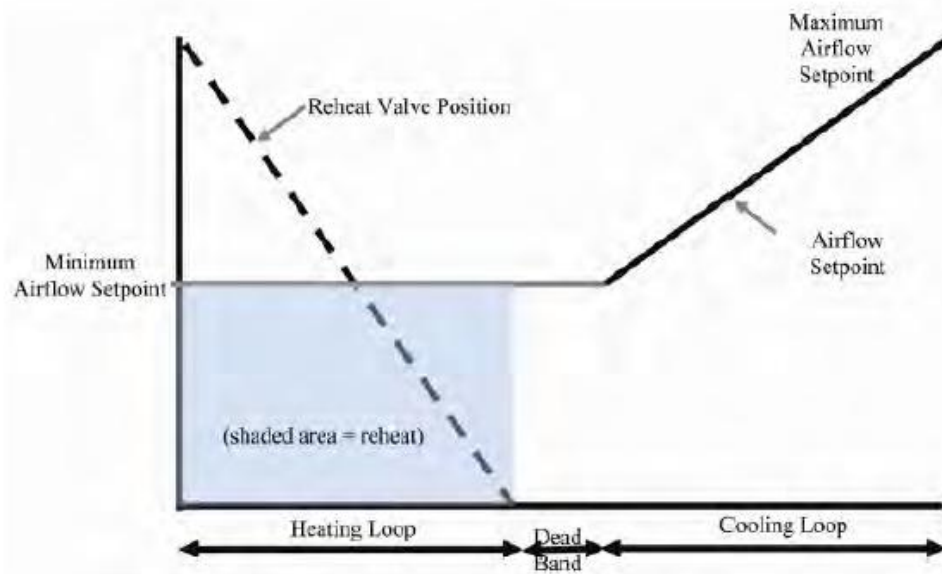
Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Damper heating action

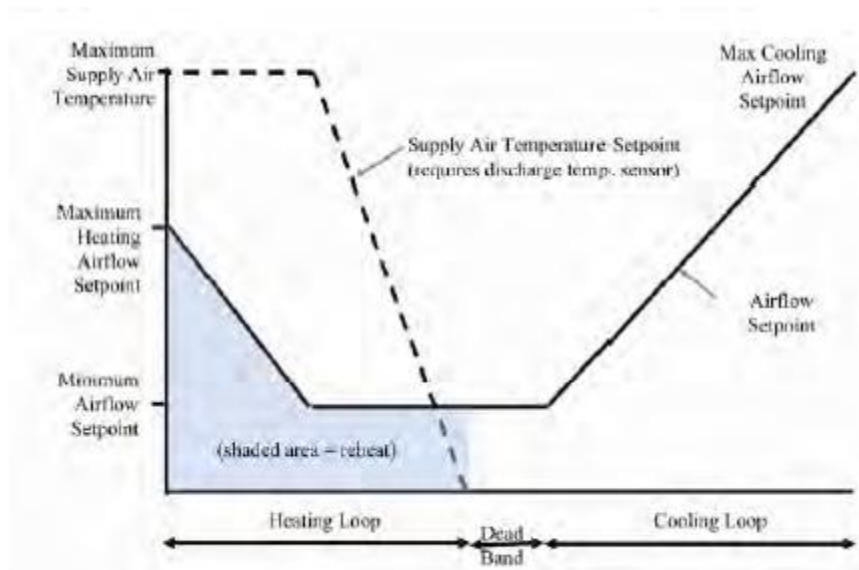
The damper heating action determines the damper action in the terminal unit as the zone moves above or below the zone setpoint. With both control options, the damper is at the minimum air flow rate whenever the zone temperature is between the cooling and heating setpoints (deadband condition).

With **1-Normal** (the default) action, the damper will remain at the minimum air flow rate during heating operation. As the heating load increases, the water flow rate in the reheat coil will be increased to maintain temperature in the zone until the maximum water flow rate is reached or the user-specified maximum reheat air temperature is reached.. This is sometimes called the single maximum control logic as illustrated below.



Single Maximum Control Logic

With **2-Reverse**, as the heating load increases, the unit starts at minimum air flow and minimum hot water flow. The hot water flow is increased until it reaches maximum flow or the user-specified maximum reheat air temperature is reached, then the air damper starts to open to meet the load. This option is used if the minimum air flow rate is not adequate to serve the peak heating load. This is sometimes called the dual maximum control logic as illustrated in the following figure. For an electric heating coil, the reverse action works the same as the normal action – always keeping the air flow at the minimum during heating.



Dual Maximum Control Logic

Maximum reheat temperature

This is the maximum supply air temperature (°C) leaving the reheat coil in the VAV terminal unit during heating operation.

Air Flow

Maximum air flow rate

The design maximum volume flow rate (m³/sec) specified for the ADU. This field may be auto-sized.

Zone minimum air flow method

This is used to select how the software will determine the minimum flow rate to the zone while the system is operating. There are three choices for selecting how the minimum flow rate is specified:

- **1-Constant** where the value entered for **Constant minimum air flow fraction** will be used.
- **2-Fixed flow rate**, the value entered for **Fixed minimum air flow rate** is used.
- **3-Scheduled** then the software will obtain the value for minimum flow fraction from the schedule selected from **Minimum air flow fraction schedule**.

Air flow control for coil sizing

This item will only appear if **3-Scheduled** is selected as the **Zone minimum air flow method**. It can be used to control the air flow rate used to size the unit heating coil if the coil is a water coil and the water flow rate for the coil is set to be auto-sized. The **Air flow control for coil sizing** drop-list contains three options:

- **1-None**,
- **2-Constant minimum air flow fraction** and
- **3-Fixed minimum air flow rate**.

If **1-None** is selected, then the air flow rates used for sizing normal-action reheat coils is the average of the minimum and maximum values in this schedule. The air flow rate used for reheat coil sizing is reported with other component sizing information as **Reheat coil sizing air volume flow rate**.

Constant minimum air flow fraction (Turndown ratio)

If **1-Constant** is selected for the **Zone minimum air flow method**, then the **turndown ratio** is used to define the minimum flow rate to the zone specified as a fraction of the maximum air flow rate while the system is operating. The minimum zone fraction is normally specified to meet the minimum ventilation requirement for the occupants. The reheat coil operates only when the damper is at this minimum flow rate when **Damper heating action** is set to **1-Normal** (the default).

Fixed minimum air flow rate

If **2-Fixed flow rate** is selected for the **Zone minimum air flow method**, then this setting is used to define the minimum flow rate (m³/s) to the zone specified as a fixed air flow rate while the system is operating. The minimum air flow rate is normally specified to meet the minimum ventilation requirement for the occupants. The reheat coil operates only when the damper is at this minimum flow rate when **Damper heating action** is set to **1-Normal** (the default). This field is used if the **Zone minimum air flow method** field is set to **2-Fixed flow rate**.

Minimum air flow fraction schedule

If **3-Scheduled** is selected for the **Zone minimum air flow method**, then this setting is used to define the schedule that determines the value of the minimum air flow fraction. The schedule should contain fractions from 0.0 to 1.0. These values will define the minimum flow rate to the zone while the system is operating, specified as a fraction of the maximum air flow rate. The reheat coil operates only when the damper is at this minimum flow rate when **Damper heating action** is set to **1-Normal** (the default).

Heating Coil

This section is visible only if the **heating coil** sub-component of this ADU has Type **1-Water**.

Maximum hot water flow rate

The maximum hot water volumetric flow rate (in m³/s or gal/min) through the unit's heating coil if [a water coil](#) has been selected. This value may be auto-sized.

Minimum hot water flow rate

The minimum hot water volumetric flow rate (in m³/s or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Maximum flow control during reheat

This item is only available if the reheat coil is a water coil and the [Damper heating action](#) is set to **2-Reverse**. There are two available options:

- **1-None**,
- **2-Maximum flow per zone floor area during reheat** and
- **3-Maximum flow fraction during reheat**.

These methods are used to calculate the maximum allowable air flow rate during reheat operation. If **1-None** is selected, the maximum flow will not be limited. If **Control on outdoor air flow** is selected, the limit established through a **2-Maximum flow control during reheat** option may be increased by the software to meet the outdoor air flow rate requirement. At no time will the maximum flow rate calculated here exceed the value for **Maximum air flow rate**.

This limit is active only when the zone thermostat requests heating and the VAV box damper is reverse acting.

Maximum flow per zone floor area during reheat

This factor (m³/s-m²) is multiplied by the zone area, to determine the maximum volume flow rate (m³/s) allowed during reheat operation (see detailed explanation above). This field may be auto-calculated by entering "autocalculate". If auto-calculated, the value is set to 0.002032 m³/s-m² (0.4 cfm/ft²).

Maximum flow fraction during reheat

This fraction is multiplied by the **Maximum air flow rate** to determine the maximum volume flow rate (m³/s) allowed during reheat operation (see detailed explanation above). This field may be auto-calculated by entering "autocalculate". If auto-calculated, the value is set to 0.002032 m³/s-m² (0.4 cfm/ft²) multiplied by the zone floor area divided by the **Maximum air flow rate**.

Outdoor Air

Control on outdoor air flow

If this option is selected, the terminal unit will increase flow as needed to meet the outdoor air requirement specified on the HVAC zone dialog. If **Outdoor air flow per person** is non-zero, then the outdoor air requirement will be calculated by the software based on the current number of occupants in the zone. At no time will the supply air flow rate exceed the value for **Maximum air flow rate**. If this option is not selected, then the terminal unit will not be controlled for outdoor air flow.

Operation

Availability schedule

This is the schedule that determines whether or not the unit is available for each hour of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the hour

Advanced

Convergence tolerance

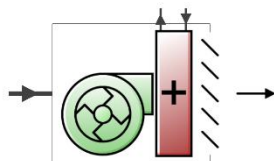
The coil is controlled by knowing the zone demand determined by the zone thermostat and setting the outlet conditions to meet this demand. For the electric and gas coils, this is set exactly since the coil model solution can be inverted. With the hot water coil that uses an effectiveness-NTU method, the solution cannot be inverted directly. Therefore, to determine the correct mass flow rate for the hot water the solution is solved for by iteration. The iterative solution uses an interval halving routine and needs a termination criterion that is set with the **Convergence tolerance** parameter. This control offset is set to a decimal fraction of the zone demand as the criteria, i.e. 0.001. The default for the field is 0.001.

Variable Air Volume With Reheat And Variable Speed Fan ADU

The VAV terminal unit with [variable-speed fan](#) and reheat coil is an air system terminal unit consisting of a variable speed fan in series with a [heating coil](#). These units are usually employed in under-floor air distribution (UFAD) systems where the air is supplied at low static pressure through an under-floor plenum. The fan is used to control the flow of conditioned air that enters the space. When the fan is off the plenum pressure drives the minimum air flow through the terminal unit. At maximum cooling the fan runs at its maximum speed. At full heating the fan runs at its heating maximum – usually less than the cooling maximum flow rate. Thus this unit has two separate maximum flow rates – one for heating and one for cooling.

For cooling, control is maintained simply by varying the fan speed. For heating, the unit first tries to meet the heating load by varying the heating coil output while keeping the air flow at minimum (fan off). If this is not adequate the fan turns on and operates in variable flow mode up to the heating maximum flow rate. This unit is modelled in EnergyPlus as a compound component – a [variable speed fan](#) and a heating coil in series in the air stream. The unit is blow through – the fan is upstream of the heating coil.

The [heating coil](#) can be edited to change the type to **1-Water**, **2-Electric** or **3-Gas**.



Note: When using the VAV with VS fan and reheat coil ADU, the associated AHU must have the extract fan option switched OFF.

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Air Flow

Maximum cooling air flow rate

This is the maximum volumetric air flow rate (in m³/s or ft³/min) through the unit when the thermostat is calling for cooling. Normally this is the same as the unit's fan maximum volumetric flow rate.

Maximum heating air flow rate

This is the maximum volumetric air flow rate (in m³/s or ft³/min) through the unit when the thermostat is calling for heating.

Minimum air flow fraction

This is the minimum flow rate to the zone while the system is operating, specified as a fraction of the maximum air flow rate. For this unit this is the flow rate when the fan is off.

Heating Coil

This section is visible only if the [heating coil](#) sub-component of this ADU has Type **1-Water**.

Maximum hot water flow rate

The maximum hot water volumetric flow rate (in m³/s or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Minimum hot water flow rate

The minimum hot water volumetric flow rate (in m³/s or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Advanced

Heating convergence tolerance

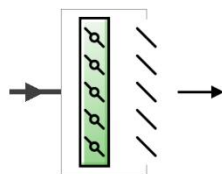
This is the control tolerance for the unit heating output. The unit is controlled by matching the unit output to the zone demand. The model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|Q_{unit,out} - Q_{zone\ load}|}{Q_{zone\ load}}$$

Variable Air Volume With No Reheat ADU

Variable air volume (VAV) systems control the dry-bulb temperature inside a zone by varying the supply air volume instead of the air temperature. At full cooling the VAV damper is fully open supplying the specified maximum air flow rate. As the cooling load decreases, the damper closes until it reaches the minimum stop specified by the zone minimum air flow fraction.

VAV systems can be used for interior or perimeter zones with a common fan system and air temperature control. The VAV concept may vary according to the VAV box locations and air temperature controls. Heating can be provided if necessary by use of baseboard.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum air flow rate

The design maximum volume flow rate (m³/s or ft³/min) specified for VAV ADU. This is auto-sizable.

Zone minimum air flow method

This is used to select how the software will determine the minimum flow rate to the zone while the system is operating. There are 5 choices for selecting how the minimum flow rate is specified:

- **1-Constant** where the value entered for **Constant minimum air flow fraction** will be used.
- **2-Fixed flow rate**, the value entered for **Fixed minimum air flow rate** is used.
- **3-Scheduled** then the software will obtain the value for minimum flow fraction from the schedule selected from **Minimum air flow fraction schedule**.
- **4-Sum** used to provide the summation of all entered inputs
- **5-Maximum** used to either provide the maximum of all entered inputs

Constant minimum air flow fraction

If **1-Constant** is selected for the **Zone minimum air flow method**, then this setting is used to define the minimum flow rate to the zone specified as a fraction of the maximum air flow rate while the system is operating. The minimum zone fraction is normally specified to meet the minimum ventilation requirement for the occupants.

Fixed minimum air flow rate

If **2-Fixed flow rate** is selected for the **Zone minimum air flow method**, then this setting is used to define the minimum flow rate (m³/s or ft³/min) to the zone specified as a fixed air flow rate while the system is operating. The minimum air flow rate is normally specified to meet the minimum ventilation requirement for the occupants.

Minimum air flow fraction schedule

If **3-Scheduled** is selected for the **Zone minimum air flow method**, then this setting is used to define the schedule that determines the value of the minimum air flow fraction. The schedule should contain fractions from 0.0 to 1.0. These values will define the minimum flow rate to the zone while the system is operating, specified as a fraction of the maximum air flow rate.

Outdoor Air

Control on outdoor air flow

If this option is selected, the terminal unit will increase flow as needed to meet the outdoor air requirement specified on the HVAC zone dialog. If **Outdoor air flow per person** is non-zero, then the outdoor air requirement will be calculated by the software based on the current number of occupants in the zone. At no time will the supply air flow rate exceed the value for **Maximum air flow rate**. If this option is not selected, then the terminal unit will not be controlled for outdoor air flow.

Operation

Availability schedule

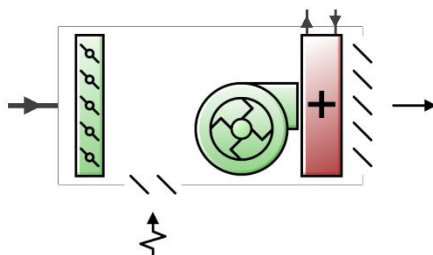
This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Series Powered Induction Unit With Reheat ADU

Powered Induction units (also known as fan-assisted terminal units) ensure that dumping of the supply air will not occur. They contain a small fan which mixes the supply air with re-circulated room air to provide a constant volume supply to the space. The mixing of the two streams is controlled to achieve an air temperature that satisfies the cooling load for the zone being served. Induction units generally have higher capital and maintenance costs and also the potential for increased noise levels.

The EnergyPlus **series** powered induction unit (PIU) is an air system terminal unit that mixes varying amounts of secondary (re-circulated) air and primary (conditioned supply) air to produce a fixed flow of air to a zone. The unit contains a small fan that acts to induce the secondary air and a heating coil for heating the mixed secondary and primary air. The fan runs at a constant volume flow rate whenever the unit is on. The fan is downstream of the primary and secondary air inlets. The variable mixing is accomplished by a damper in the unit's primary air supply inlet duct. This damper can move from fully open (100% primary air, 0% secondary air) to a minimum stop that is specified in the input description. At full cooling the damper will be fully open. At minimum cooling and for heating the damper will be at the minimum stop and the secondary air flow will be at its maximum.

The EnergyPlus model of the series PIU terminal unit is composed of three components: a zone mixer, a [constant volume fan](#), and a [heating coil](#) (water, electric, or gas).



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Air Flow

Maximum air flow rate

This is the maximum volumetric air flow rate through the unit (in m³/s or ft³/min). Since this is a constant air volume unit, this is also the design, rated air flow rate of the unit.

Maximum primary air flow rate

This is the maximum volumetric air flow rate of primary air through the unit (in m³/s or ft³/min). This is the primary air flow rate at full cooling load when the primary air damper is fully open. Usually this quantity is the same as the total unit flow rate, but it can be less.

Minimum primary air flow fraction

The minimum volumetric air flow rate of primary air through the unit expressed as a fraction of the maximum volumetric air flow rate of primary air. This input can be 0.0.

Heating Coil

This section is visible only if the [heating coil](#) sub-component of this ADU has Type **1-Water**.

Maximum hot water flow rate

The maximum hot water volumetric flow rate in (m³/s or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Minimum hot water flow rate

The minimum hot water volumetric flow rate in (m³/s or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Advanced

Convergence tolerance

This is the control tolerance for the unit heating output. The unit is controlled by matching the unit output to the zone demand. For units with water coils, the model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|Q_{unit,out} - Q_{zone\ load}|}{Q_{zone\ load}}$$

Parallel Powered Induction Unit With Reheat ADU

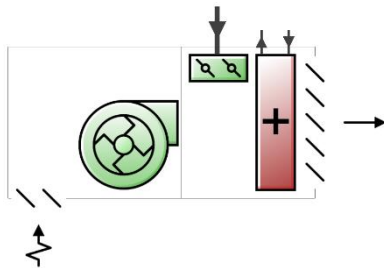
Powered Induction units (also known as fan-assisted terminal units) ensure that dumping of the supply air will not occur. They contain a small fan which mixes the supply air with re-circulated room air to provide a

constant volume supply to the space. The mixing of the two streams is controlled to achieve an air temperature that satisfies the cooling load for the zone being served. Induction units generally have higher capital and maintenance costs and also the potential for increased noise levels.

The EnergyPlus **parallel** powered induction unit (PIU) is an air system terminal unit that mixes varying amounts of secondary (re-circulated) air and primary (conditioned supply) air to produce a variable total flow of air to a zone. The unit contains a small fan that acts to induce the secondary air and a heating coil for heating the mixed secondary and primary air. The secondary and primary air streams enter the unit in parallel. The fan sits in the secondary air stream and runs only when secondary air is needed. The primary air inlet contains a damper that can move from fully open (maximum primary air) to a minimum stop (minimum primary air).

At full cooling load the primary air damper is fully open and the fan is off. The primary air flow is at maximum and there is little or no secondary air flow. As the cooling load decreases, the primary air damper gradually closes and the secondary air flow remains close to zero. At some point, usually when the primary air flow has reached the minimum, the fan switches on and secondary air is induced. The heating coil will switch on as needed to meet any heating demand.

The EnergyPlus model of the parallel PIU terminal unit is composed of three components: a [constant volume fan](#), a zone mixer, and a [heating coil](#) (water, electric or gas):



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Fan on flow fraction

This is the fraction of the primary air flow at which fan turns on. In the parallel PIU the fan operation is intermittent. If the primary air flow is above this fraction of the maximum, the fan is off. Otherwise it is on.

Supply Air Flow Rates

Maximum primary air flow rate

This is the maximum volumetric air flow rate of primary air through the unit (in m^3/s or ft^3/min). This is the primary air flow rate at full cooling load when the primary air damper is fully open.

Maximum secondary air flow rate

This is the maximum volumetric air flow rate of secondary air through the unit (in m^3/s or ft^3/min). This flow rate can be any amount but is commonly less than the maximum primary air flow rate.

Minimum primary air flow fraction

This is the minimum volumetric air flow rate of primary air through the unit expressed as a fraction of the maximum volumetric air flow rate of primary air. This input can be 0.0.

Heating Coil

This section is visible only if the [heating coil](#) sub-component of this ADU has Type **1-Water**.

Maximum hot water flow rate

The maximum hot water volumetric flow rate (in m³/sec or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Minimum hot water flow rate

The minimum hot water volumetric flow rate (in m³/sec or gal/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Advanced

Convergence tolerance

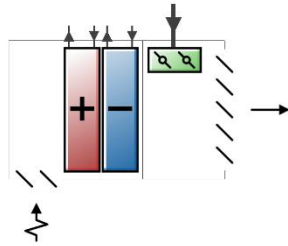
This is the control tolerance for the unit heating output. The unit is controlled by matching the unit output to the zone demand. For units with water coils, the model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|Q_{unit,out} - Q_{zone\ load}|}{Q_{zone\ load}}$$

Four Pipe Induction Unit ADU

The four pipe induction terminal unit provides local hot water heating or chilled water cooling of induced zone air which then mixes with centrally conditioned supply air. An air conditioning system consisting of these terminal units is effectively a mixed central air/local hydronic system. The centrally conditioned air supplied to the induction terminal units is **constant volume** at quite high pressure. The central air is discharged through a nozzle in the terminal unit, inducing a flow of room air over a hydronic heating/cooling coil. The coil is connected either to a single inlet and outlet pipe (2 pipe unit) or to 2 inlets and 2 outlets (4 pipe unit). The heated or cooled induced air mixes with the centrally conditioned air before being discharged into the zone. The terminal units are usually expected to do only sensible cooling – any dehumidification is done by the central air conditioning system.

The EnergyPlus model of the four pipe induction terminal unit is a compound component consisting of a hot water heating coil, a chilled water cooling coil, and an air mixer. The unit has two inlet air streams: the centrally conditioned supply air and the induced air from the zone. The induced air passes first through the heating coil, then through the cooling coil and finally through the mixer. The central supply air goes directly into the mixer. The water flow through the hot or cold water coil is varied to meet the zone air conditioning requirement. Note that EnergyPlus models the four pipe induction terminal unit as having separate heating and cooling coils whereas real units have only a single coil used for both heating and cooling. Note also that the four pipe induction unit model can be used to model a two pipe unit by simply adjusting the heating and cooling coil schedules so that the heating coil is off when the cooling coil is on and vice versa.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum total air flow rate

The maximum volumetric air flow rate discharged from the unit (in m³/s or ft³/min). Since this is a constant air volume unit, this is also the design, rated air flow rate of the unit. Note that this is the total discharge flow rate – including both central supply and induced air.

Induction ratio

The ratio of induced air flow rate to primary supply air flow rate. The default is 1.0 –the supply air induces an equal amount of zone air.

Heating Coil

This section is visible only if the [heating coil](#) sub-component of this ADU has Type **1-Water**.

Maximum hot water flow rate

The maximum hot water volumetric flow rate (in m³/s or ft³/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Minimum hot water flow rate

The minimum hot water volumetric flow rate (in m³/s or ft³/min) through the unit's heating coil if a [water coil](#) has been selected. This value may be auto-sized.

Cooling Coil

Maximum cold water flow rate

The maximum cold water volumetric flow rate (in m³/s or ft³/min) through the unit's cooling coil. This value may be auto-sized.

Minimum cold water flow rate

The minimum cold water volumetric flow rate (in m³/s or ft³/min) through the unit's cooling coil. This value may be auto-sized.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Advanced

Heating convergence tolerance

This is the control tolerance for the unit heating output. The unit is controlled by matching the unit output to the zone demand. For units with water coils, the model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|q_{unit,out} - q_{zone\ load}|}{q_{zone\ load}}$$

Cooling convergence tolerance

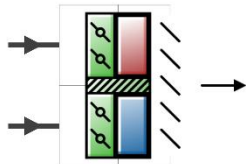
This is the control tolerance for the unit cooling output. The unit is controlled by matching the unit output to the zone demand. The model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|q_{unit,out} - q_{zone\ load}|}{q_{zone\ load}}$$

Dual Duct Constant Air Volume ADU

The dual duct constant air volume simulation or the typical multi-zone is described by this Air Distribution Unit (ADU). Multi-zone systems condition all the air in a central apparatus and distribute it to the conditioned zones through two parallel ducts. One duct carries cold air and the other warm air, providing air sources for both heating and cooling at all times. In each conditioned zone, a mixing valve responsive to a room thermostat mixes the warm and cold air in proper proportions to satisfy the prevailing heating or cooling load of the space.

The total airflow to each room is kept constant while the proportion of hot air to cold air is adjusted to maintain the temperature in each zone at the desired level.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum air flow rate

The design constant volume flow rate (in m³/s or ft³/min) specified for the multi-zone ADU.

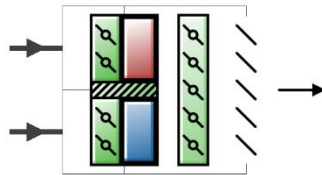
Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Dual Duct Variable Air Volume ADU

Dual duct variable air volume (DDVAV) systems are used to obtain zone temperature control by mixing the cold and warm air in various volume combinations. The fan is sized for the anticipated maximum coincident hot or cold volume, not the sum of the instantaneous peaks. This system has an advantage of a true single path VAV system, except for warm port leakage. When cold air is modulated for control before mixing, it operates similar to the VAV induction when mixing occurs without hot deck reheat. It is similar to a reheat system when mixing occurs while the hot deck is using the reheat coil. It uses more energy than a true VAV system, but less than a constant volume dual duct system.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum damper air flow rate

The design maximum volume flow rate (in m³/s or ft³/min) specified for the DDVAV ADU.

Zone minimum air flow fraction

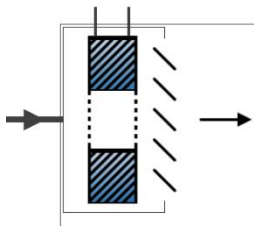
The minimum flow rate to the zone while the system is operating specified as a fraction of the Maximum Damper Air Flow Rate. The minimum zone fraction is normally specified to meet the minimum ventilation requirement for the occupants.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Cooled Beam



Cooled beams are long rectangular finned tubes mounted at high level at regular intervals within a space. They provide quiet draught-free cooling with capacities up to about 150 W/m². As the cooled beams cool the surrounding air, it falls into the occupied area below due its greater density. The beams are typically arranged at regular intervals above, or partly below, a false ceiling. A relatively small ceiling void depth is required.

It is important to ensure good air flow and so any ceiling tiles positioned below beams must have openings within the tiles equivalent to at least 50% of their area. In some cases beams are suspended below the ceiling.

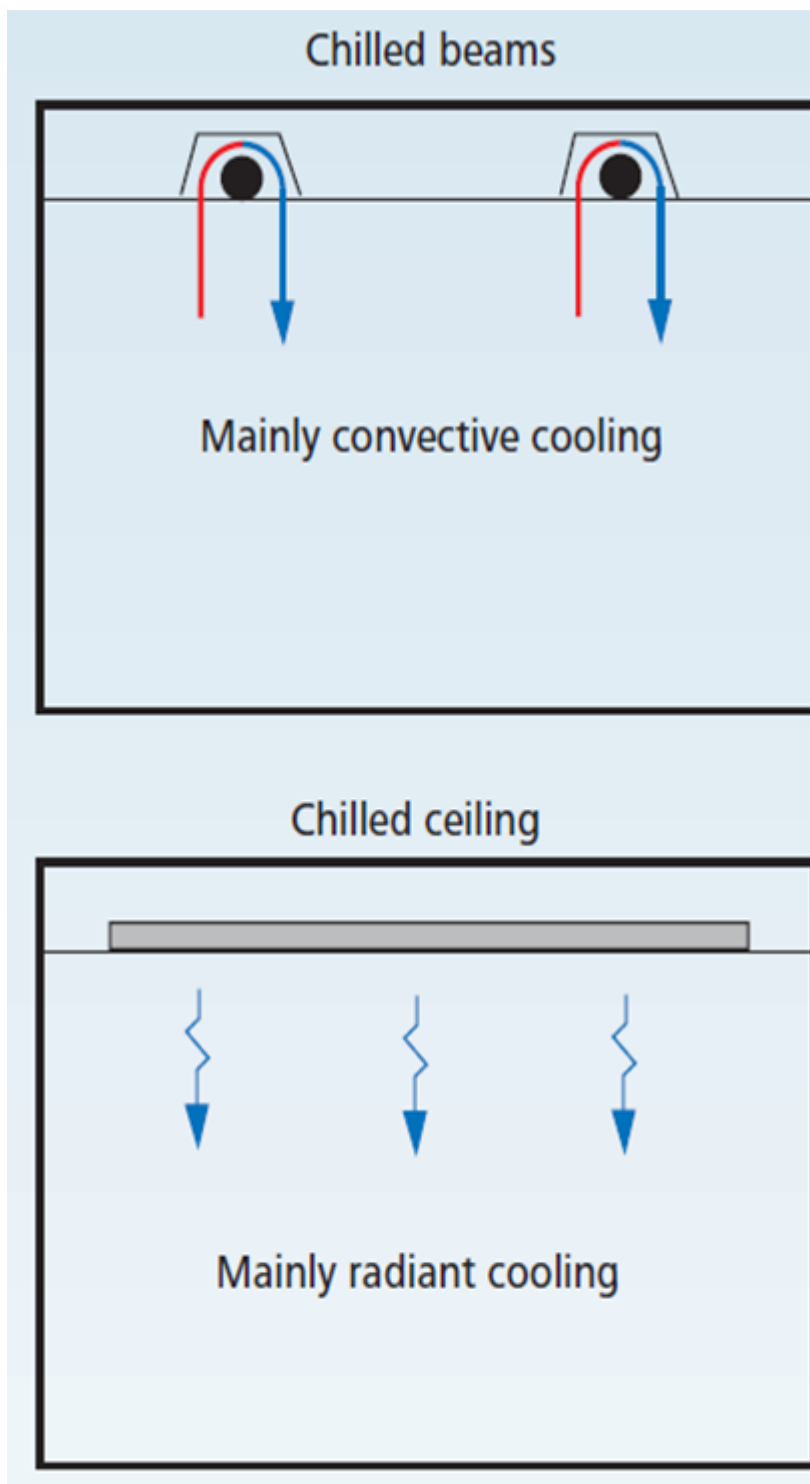
Cooled beam systems require a separate constant volume ventilation system to supply fresh air to the space.

The EnergyPlus Cooled Beam system is a mixed air-hydronic system.. A central constant volume air system supplies conditioned ventilation air to the zones. Chilled water circulates through ceiling mounted cooled beam units at a rate suitable for meeting the zone sensible cooling load. Any dehumidification is done by the central ventilation air system. Heating is usually accomplished with hot water radiators/baseboards. Thermodynamically, the cooled beam system resembles the four-pipe induction unit.

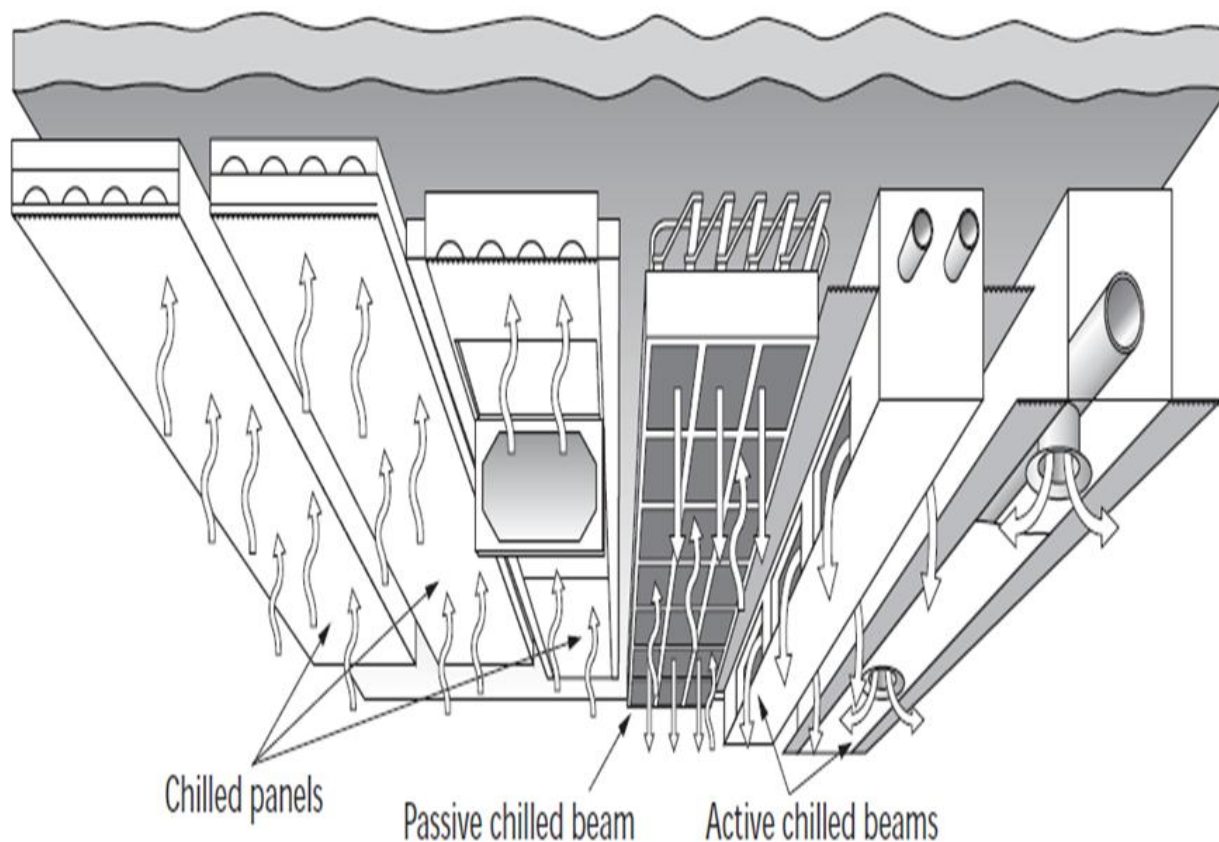
To model a typical cooled beam system a conventional central constant volume forced air system can be used. This system will normally be 100% outside air delivered at a fixed supply temperature (which could be reset by schedule or by outside air temperature). On the supply side of this air loop there will be the usual central AC equipment: outside air mixer, fan, heating and cooling coil.

The cooled beam equipment in a zone is treated by the program as a single terminal unit, however the actual installation will have multiple beams in each zone. EnergyPlus calculates how many beams of what length are needed to meet the zone design load during its sizing calculation for the system.

The images below (reproduced with permission from CIBSE) illustrate the difference between the cooled beams described here and [chilled ceilings](#).



CIBSE KS03 Figure 12



CIBSE Guide H Fig 5.79

Important Note: If an air loop is exclusively supplying cooled beams, the Air loop [Type of load to size on](#) setting should be set to **2-Ventilation requirement**, otherwise the system air flow rate will be sized on the zone sensible load and the resulting flow rate may be too high for the cooled beams to maintain the zone setpoint temperatures.

Tip: A useful resource on chilled ceilings and cooled beams can be found at [https://feta.co.uk/client/files/HEVAC/Chilled_Beams_Brochure_Final_7_\(web\).pdf](https://feta.co.uk/client/files/HEVAC/Chilled_Beams_Brochure_Final_7_(web).pdf)

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Cooled beam type

Two types of units are modelled:

- **1-Active** where primary air is supplied through the beam, inducing some secondary zone air into contact with the coil. This unit acts as an active convector.
- **2-Passive** in which is simply a passive, finned convector unit. Primary air is supplied through a normal diffuser.

Supply air volumetric flow rate

This is the air flow rate of the supply air entering the zone (in m³/s or ft³/min). This input would normally be autosized based on the [zone ventilation requirement](#).

Maximum total chilled water volumetric flow rate

The maximum chilled water flow rate (in m³/s or gal/min) for the unit. This input would normally be autosized based on the zone design load.

Number of beams

The number of individual cooled beam units in the zone. Normally this unit would be auto-sized by the program based upon the previous field and the nominal flow rate for a single beam unit (set by the program to 0.07 kg/s).

Beam length

This is the length of an individual beam in metres. Normally this will be auto-sized by the program based upon the number of beam units and the zone design sensible cooling load. 1 to 4 meters is a typical length range.

Design inlet water temperature

The nominal or design inlet water temperature (in °C or °F). The default is 15°C.

Design outlet water temperature

The nominal or design outlet water temperature (in °C or °F). The default is 17°C.

Performance Parameters

The following inputs are parameters used to characterize the performance of the chilled beam units. Values for a given unit can be obtained from the manufacturer. The parameters are used in the following equations.

$P_{beam} = A \times K \times \Delta T$	beam cooling output per unit length W/m
$K = \alpha \times \Delta T^{n1} \times V \times \rho^{n2} \times \omega^{n3}$	coil heat transfer coefficient W/(m ² K)
$V\rho = (q_{in}/\alpha_0) \times \rho_{air}$	room air mass flow rate across coil kg/(m ² s)
$q_{in} = K_1 \times \Delta T^n + K_{in} \times q_{pr}$	volumetric flow rate across coil per unit length m ³ /(s-m)

Where:

ΔT is the room air – water temperature difference (average water temperature is used) in °C.

ω is the water velocity in m/s.

q_{pr} is the supply air flow rate per unit length m³/ (s-m)

Coil surface area per coil length

This is the surface area on the air side of the beam per unit beam length (in m²/m or ft²/ft). The default is 5.422. This is A in the equations above.

Coefficient of induction (K_{in})

This is the coefficient of induction K_{in} in the equations above. The default is 2.0 for active beams and 0.0 for passive beams.

Pipe inside diameter

This is the water pipe inside diameter (in m or in). The default is 0.0145.

Model parameter α

This is α in the above equations. The default is 15.3

Model parameter n1

This is n_1 in the above equations. The default is 0.

Model parameter n2

This is n_2 in the above equations. The default is 0.84.

Model parameter n3

This is n_3 in the above equations. The default is 0.12.

Model parameter α_0

This is α_0 in the above equations. It is the free area of the coil in plan view (for the air flow) per unit beam length. The units are square meters per meter. The default is 0.171.

Model parameter K1

This is K_1 in the above equations. The default is 0.005.

Model parameter n

This is n in the above equations. The default is 0.4.

Operation

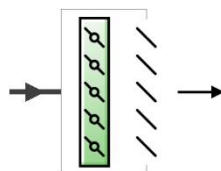
Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Variable Air Volume Outdoor Air ADU

The Variable Air Volume Outdoor Air terminal unit is intended for use in single duct systems for modelling centralized dedicated outdoor air (DOAS) distribution system when the required volume of outdoor air varies over time. The component is based on the EnergyPlus **AirTerminal:DualDuct:VAV:OutdoorAir** which provides dual duct air system in which one duct provides outdoor air ventilation and the other provides VAV cooling. However DesignBuilder uses only the outdoor air stream for modelling DOAS and the cooling duct is not used.

The terminal unit is meant for systems regulating outdoor air based only on demand control or code minimum settings and not on economizer operation. This air terminal does not have heating coils and is not configured for conditions zones that need heat. Additional ZoneHVAC equipment, such as baseboards or fan coil units, are needed to provide heating to the zone.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the ADU is located.

Maximum zone total air flow rate

Max total airflow rate (m³/s or ft³/min) at the terminal unit outlet including both the Outdoor Air stream and the Recirculated stream. Can be autosized based on the Air loop [Design outdoor airflow rate](#) and the zone thermal cooling load. The autosized flow rate will be the sum of the maximum outdoor air requirement and the flow for cooling. When autosizing, this maximum and the individual inlet flow rate sizes are reported to the eio file.

Per person ventilation rate mode

This option is used to select how outdoor air ventilation rates are calculated when based on a rate per person. It can be either based on the current number of people as affected by time-varying occupancy schedules, or on the constant value for the maximum number of people. Choose one of:

- **1-Current occupancy** where outdoor air ventilation rates are based on the current number of people including the effect of time-varying occupancy schedules.
- **2-Design occupancy** where the constant value for the maximum number of people is used.

Note: this setting is only used when outdoor air ventilation rates are calculated when based on a per person rate.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the unit is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Radiant Surface Data

Radiant surfaces allow you to model a wide range of radiant systems where water is passed through pipes in a panel or embedded in the construction, transferring energy to/from a building surface (wall, ceiling or floor). They can be used to model low-temperature heated only surfaces (e.g. heated floors), cooled only surfaces (e.g. chilled ceilings) or changeover systems which heated or chilled water is passed through pipes to provide heating or cooling depending on the zone load. They can also be used to simulate surfaces with buried electric heating elements.

See the [Radiant surfaces](#) tutorial

Three types of radiant surfaces are available in DesignBuilder HVAC:

- [Constant flow](#)
- [Variable flow](#)
- [Electric](#)

Note an important advantage of the EnergyPlus [variable flow](#) radiant surface over the [constant flow](#) equivalent is that it is fully autosizable, which makes it much quicker to set up, especially for large models.

When using the **2-Detailed HVAC** [Detailed HVAC Activity data](#) option, radiant surface systems will only control based on the radiant system controls defined for the component itself and not via the standard zone thermostat defined on the HVAC zone dialog. If the radiant system is serving a zone with forced air equipment, the radiant system will follow the priority order established by the zone thermostat but will still base its response on the controls defined by the user for the radiant system.

When using the **1-Simple HVAC** [Detailed HVAC Activity data](#) option, radiant surfaces are controlled using the usual HVAC zone heating setpoint temperature controls.

Note: In zones in which radiant systems are included, at least one surface must be assigned a construction with an internal source defined, which incorporates details of the embedded tubing system. See [Internal Source](#) under Constructions for further details.

Radiant surface HVAC components are placed, edited and deleted at [HVAC zone](#) level. To edit the data associated with a radiant surface, you first need to select it by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the Edit selected component tool from the toolbar.

Important Note 1: The [internal source](#) must be placed thermally closer to the inside surface than to the outside surface of the construction to allow it to be considered for inclusion in a zone's radiant surface system. In other words, if the thermal resistance between the inside surface and the internal source is less than the resistance to the outside surface then the surface is considered to be a valid radiant surface to be included with the zone next to the inside surface and will be simulated as such. This rule applies to surfaces adjacent to outside as well as internal surfaces.

Important Note 2: Internal radiant surfaces cannot control heating/cooling delivery to both adjacent zones. The radiant surface will be associated with the control system of the adjacent zone that is thermally closest to the internal source. The zone next to the outside surface (thermally furthest from the internal source) will of course still receive some heating/cooling from the source but the heat flow will generally be less and it will not be controlled.

Tip: To define which side of an internal surface is internal and which is external see under [Layer order](#) on the Constructions Layer data page.

Heated Floors

These systems typically use low flow temperatures of around 40-55°C compared with around 80°C for traditional low temperature hot water heating systems using radiators and convectors. Lower return temperatures enable more efficient operation using condensing boilers and heat pumps.

Tip: Heated floor radiant surfaces can be connected to [hot water loops](#) fed by [boiler](#), [GSHP](#) or [district heating](#) systems. Note that when connected to GSHP systems heated floors can offer a very efficient system as the low delivery temperatures they require (typically 35°C) are well matched to the optimal operating conditions for GSHP systems.

Troubleshooting Heated Floors

DesignBuilder EnergyPlus can realistically model heated floor systems and so many of the issues that apply to real systems such as slow response, floor mass, control, floor insulation, effect of different upper surface materials can be investigated using DesignBuilder. However heated floors can take a little bit of care to set up and to achieve good temperature control. If you are having trouble with this you may find the answer to the problem is the list below.

1. **Missing Internal Source Error report.** Each zone with a heated floor added to its HVAC zone must include at least one floor surface (or floor sub-surface) having a construction with an [internal source](#). If no such internal source surface is found, DesignBuilder will generate an error message to this effect before attempting to run the simulation.
2. **Unresponsive control.** Heated floor systems have relatively slow response times as the heating pipes provide indirect heating via the floor surface. This can result in zone temperatures deviating from set-point temperatures if the heating or cooling loads in the zone change quickly, through solar gain or natural ventilation cooling for example. The slow response is due to the high thermal mass of the floor causing heat to continue to be emitted even when the room thermostat has stopped calling for heating and the pump has stopped moving hot water through the embedded pipes. Overheating in such high mass heated floors can be a common problem on days where a high demand exists on a cold morning (for example) but then high solar (or other) gains in the day add to the uncontrolled heat continuing to be emitted from the floor.
3. **Throttling range.** Less of a factor than point 2 is variable flow heated floors work using a throttling range to control flow of water through the embedded floor water pipes. This acts like a deadband and means that even without the thermal mass lag issues, there will be a temperature control range rather than a fixed zone temperature.
4. **Underheating** can also be a common issue with heated floors. Likely causes are listed below.
5. **Large heat loss.** A useful rule of thumb is that it is not usually possible to supply more than about **100 W/m²** of heating with heated floors due to the thermal resistance of floor surfaces and maximum acceptable floor temperatures of around 29°C, though higher surface temperatures are possible in bathrooms. Typical maximum outputs are approximately 100 W/m² for concrete, reducing to 70 W/m² for timber floors and less for carpet and coverings with insulating properties. Zones having higher levels of heat loss will require supplementary heating to achieve comfortable conditions. This is true of real world systems and with DesignBuilder EnergyPlus heated floor models. The easiest way to check the heat loss against this rule is by using the [Normalise](#) display option which shows results per floor area.
6. **Intermittent heating.** Running heated floors on an intermittent basis requires a higher design sizing factor to be set for the HVAC zone. This is because of the thermal mass of the floor construction. Consider a building unoccupied over a cold weekend which then needs to be heated to operating temperatures on Monday morning. With the default sizing factor of 1.25 the heating system will have been sized to achieve the heating setpoint under steady-state winter outdoor design condition plus a margin of 25%. However this 25% margin will not be adequate if the heating is to be switched on at say 6am and expected to raise the zone temperature to comfortable levels in time for occupancy a few hours later. To deal with intermittent operation, either much higher design factors are required (e.g. 2) or the system should be run continuously (perhaps at a lower level during unoccupied periods) to avoid the need for a rapid warmup in the mornings. Of course if the heating is to be operated continuously the building fabric will need to be very well insulated to avoid waste.
7. **Incorrect position of the source.** A common error with setting up the heated floor definition is to select the wrong position for the heated floor source (i.e. the hot water pipes). In a well designed system the pipes will normally be buried just below the first construction layer. If you position the tubes at the wrong side of the construction, then the wrong zone (or the ground or exterior) will be heated! A [tool](#) is provided on the Constructions dialog to make it easy to position the source just below the innermost layer.
8. **Conductive upper layer.** Heated floors provide the most efficient and responsive control when the uppermost layer is low mass and conductive. While ceilings can be constructed using a thin metal layer between source and the room, this is rarely possible for heated floors which are required to provide a strong and comfortable surface to walk on as well as an efficient heat transfer path from heating pipes to the zone. Clearly, if the top layer is an (insulating) thick pile carpet then the heated floor will struggle to provide adequate heating to the room. More advice on [floor coverings suitable for heated floors](#) is provided below.
9. **Insulation.** Without good insulation below the heated floor source, much of the heat will not find its way into the intended zone. Instead the heat will be either lost to the zone below in an uncontrolled way, to outside or to the ground. There will consequently be less heat available to heat the intended zone and underheating will occur.
10. **Sizing error ([general HVAC modelling issue](#)).** It is important to consider heating and cooling sizing when setting up schedules. In particular, a common mistake is to include internal gains in heating

- sizing calculations. This has the effect of heating being undersized causing significant underheating (or no heating).
11. When used **in combination with a chilled ceiling in zone below**, heated floors will cause EnergyPlus to generate an error. This is because EnergyPlus only allows one "source" object per surface and so a heated floor cannot be located in the same surface as a chilled ceiling. There are 2 possible workarounds to this.
 - a. Export the DesignBuilder model to EnergyPlus and using the IDF Editor combine the heated floor and chilled ceiling into a single source component.
 - b. Insert a very thin "dummy zone" between the heated floor above and the chilled ceiling below. Include half of the true floor mass in the upper surface (heated floor in upper zone) and the other half in the lower surface (chilled ceiling in lower zone).
 12. You may need a separate high-grade heat generator for **DHW** if using a low-temperature heat source for heating.

Tip: To help diagnose some of the above problems and to check that maximum temperatures are compatible with the surface material (below) it can help to view hourly inside and outside surface temperatures and heat fluxes by selecting these output options: [Surface heat transfer incl. solar](#) (gives heat flux for floors) or [Inside surface temperature](#) (to check the surface temperature of the heated floor).

Coverings Suitable for Heated Floors

The advice in this section has been adapted from www.nu-heat.co.uk.

Stone and ceramic surfaces

From the thermal point of view, the best floor coverings for use with underfloor heating are usually hard surfaces such as stone and ceramic tile as they have the least thermal resistance allowing heat from the pipe array to transfer quickly to the surface of the stone/tile.

- **Ceramic tiles** - Due to their high thermal conductivity and slim profile, ceramic tiles are one of the best floor finishes for use with underfloor heating.
- **Polished screed** - Very conductive and good for use with underfloor heating.
- **Limestone** - Stone is the most thermally conductive of all floor coverings allowing energy from the UFH to transfer quickly to the room.
- **Slate** - An extremely conductive natural finish, ideal for use with underfloor heating.
- **Flagstones** - A thicker stone finish that exhibits very good thermal conductivity but beware of adding too much thermal inertia.
- **Marble** - An good heat conductor available in varying thicknesses.

Timber flooring and underfloor heating

Engineered timber is the best of the wooden floor options for use with underfloor heating as its structural stability allows it to perform well with fluctuating temperatures. Solid hardwoods and soft woods also transfer heat well but care should be taken when specifying board width and thickness.

- **Engineered timber** - The best timber floor finish for use with heated floors, offering good heat transfer and structural stability.
- **Solid hardwood** - Good heat transfer and best when specified in narrow board widths.
- **Softwood** - Suitable only in narrow board widths to reduce movement
- **Parquet** - Offers good heat transfer when securely glued to the sub-floor
- **Bamboo** - Usually similar in construction to engineered timber, making it very suitable for UFH

Laminates, vinyl and rubber

Laminates and vinyl floor coverings also perform well with underfloor heating. It is advisable to check the manufacturer's recommended maximum floor surface temperature to ensure the product is suitable for use with UFH. Most manufacturers state an upper limit of 27°C, which equates to approximately 75W/m² of heat output, which is adequate in most situations. These products have low thermal mass so heat up and cool down quickly in comparison to stone and timber.

- **Vinyl** - good heat transfer, often subject to a recommended maximum surface temperature
- **Amtico** - practical, high-quality finish that transfers heat well
- **Linoleum & Marmoleum** - thin natural products that are suited to use with underfloor heating
- **Laminate tiles** - Thin and thermally conductive making them perfect for use with UFH
- **Rubber** - A relatively thermally conductive material that can successfully be used with underfloor heating

Materials to avoid

While not ideal from the efficiency point of view, carpet and underlay with a combined Tog value of up to 2.5 can be used effectively with underfloor heating. Thicker carpets act as an insulator and stop sufficient heat reaching the room.

- **Carpet** - A maximum Tog value of 2.5 is recommended for carpet and underlay when used with underfloor heating
- **Cork** - A maximum thickness of 10mm is advised as cork is an efficient insulator
- **Coir** - A natural product similar to carpet that is subject to a maximum recommended Tog value of 2.5

You should bear in mind that some flooring manufacturers stipulate a maximum floor temperature for their product. Note also that EN 1264 states that underfloor heating should not operate at more than 29°C.

Chilled Ceilings

Chilled ceilings are mounted within the ceiling to provide quiet, draft free cooling of the space below. Each unit is made up of a small bore chilled water pipe arranged in an S-shape and attached to the upper surface of a ceiling panel. Ceiling panels are typically of thin metallic construction but can simply be plasterboard. In some systems the chilled water pipes are embedded within the ceiling panel.

During operation, the panel is cooled through contact with the chilled-water pipework allowing it to cool the space through a combination of convective and radiant output (up to 40% radiant). An insulating mat is often placed above the chilled water pipework and panel to minimise uncontrolled cooling of the area above. It is up to the user to include any such insulation in the construction definition.

One advantage of chilled ceilings is that they can be placed in a shallow ceiling void enabling them to be used in buildings with low floor to ceiling heights. However, their limited cooling output makes them unsuitable for areas with moderate to high heat gains. The maximum capacity of chilled ceiling systems is in the order of 70 W/m².

Chilled-ceiling systems require a separate ventilation system for fresh air supply.

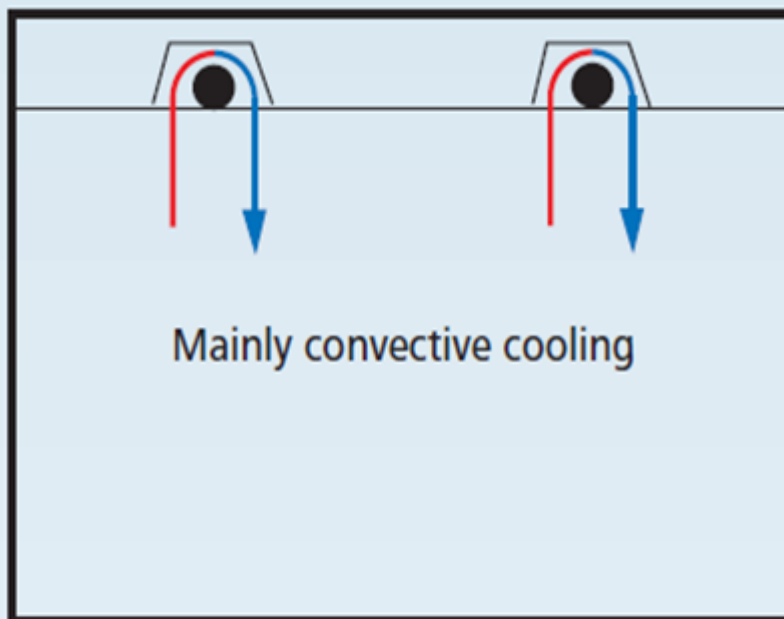
Condensation of room air on and within the chilled ceiling can be avoided by shutting of chilled water flow based on room dew point temperature.

Tip: Chilled ceilings can be connected to chilled water loops fed by [chillers](#) , [district cooling](#) or [GSHP](#) systems.

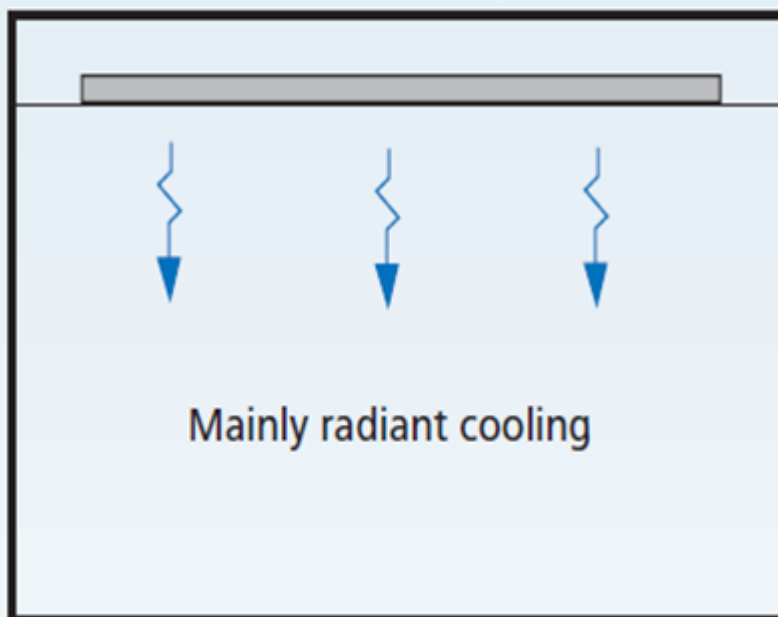
Chilled ceilings vs Cooled beams

The images below (reproduced with permission from CIBSE) illustrate the difference between the cooled beams described here and [chilled ceilings](#).

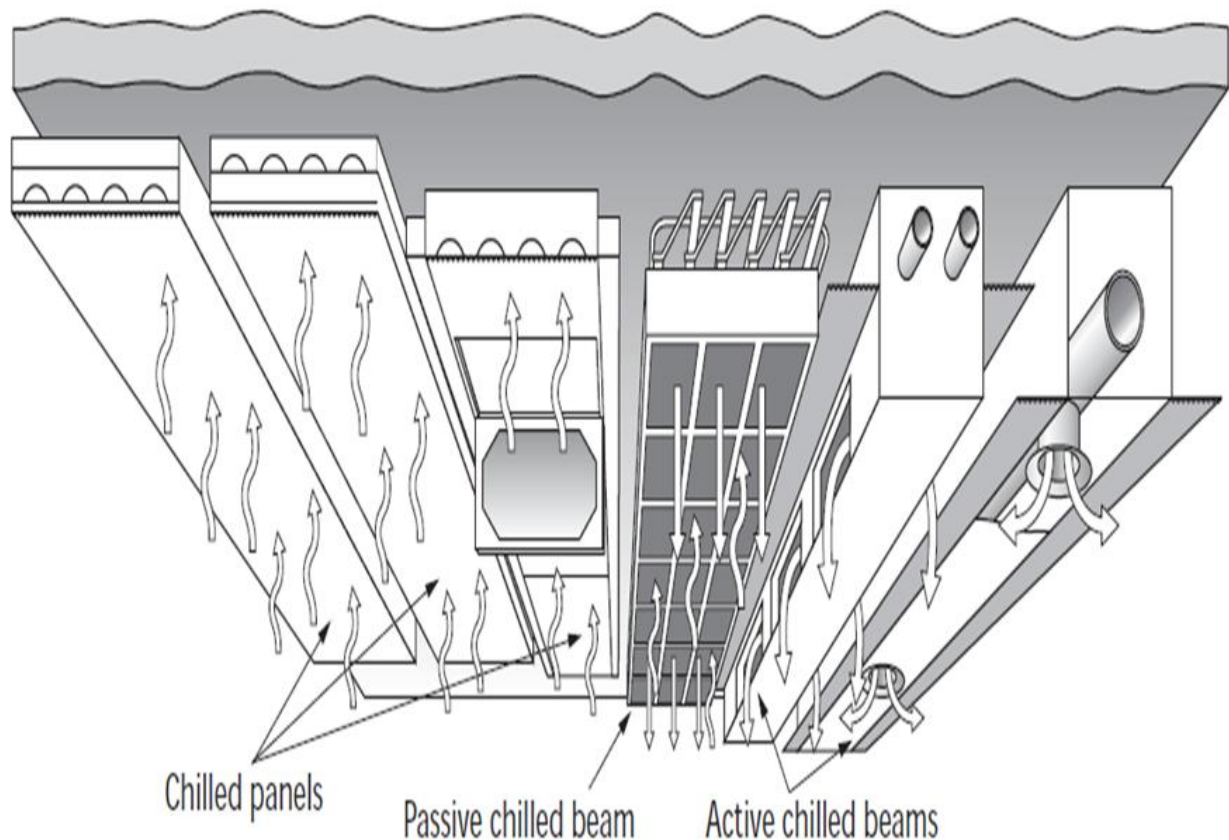
Chilled beams



Chilled ceiling



CIBSE KS03 Figure 12



CIBSE Guide H Fig 5.79

Tip: A useful resource on chilled ceilings and cooled beams can be found at https://www.feta.co.uk/uploaded_images/files/CBCA/Chilled%20Beams%20Brochure_Final%207%20%28web%29.pdf

Troubleshooting Chilled Ceilings

The DesignBuilder EnergyPlus ceiling model is able to realistically model the operation of chilled ceiling systems and many of the issues that apply to real systems such as intermittency, thermal mass, control, ceiling insulation, effect of different interior surface materials can be investigated using DesignBuilder. However chilled ceilings can take a little bit of care to set up and to achieve good temperature control. If you are having trouble with this you may find the answer to the problem is the list below.

1. **Missing Internal Source Error report.** Each zone with a chilled ceiling added to its HVAC zone must include at least one ceiling surface (or ceiling sub-surface) having a construction with an [internal source](#). If no internal source surface is found, DesignBuilder will provide an error message to this effect before attempting to run the simulation.
2. **Unresponsive control.** Chilled ceilings in real buildings tend to give less responsive control than air or other radiant systems. This is because the relatively high thermal mass of the ceiling causes zone heat to continue to be absorbed even when the room thermostat has stopped calling for cooling and the pump has stopped moving chilled water through the embedded pipes.
3. **Throttling range.** Less of a factor than point 2 is variable flow chilled ceilings work using a throttling range to control flow of water through the embedded floor water pipes. This acts like a deadband and means that even without the thermal mass lag issues, there will be a temperature control range rather than a fixed zone temperature.
4. **Inadequate cooling** can also a common issue with chilled ceilings. Likely causes are listed below.
5. **High levels of internal gains.** A useful rule of thumb is that it is not usually possible to supply more than about **70 W/m²** of cooling with chilled ceilings. Zones having higher levels of gains will require supplementary cooling to achieve comfortable conditions. This is true of real world systems and with DesignBuilder EnergyPlus chilled ceiling models. The easiest way to check the internal gains against this rule is by using the [Normalise](#) display option which shows results per floor area.

6. **Intermittent cooling.** Running chilled ceilings on an intermittent basis can require a higher design sizing factor to be set for the HVAC zone. This is because the thermal mass of the ceiling construction causes a lag in cooling provided to the coils and cooling provided to the room. Consider a building unoccupied over a warm weekend which then needs to be cooled to operating temperatures on Monday morning. With the default sizing factor of 1.15 the cooling system will have been sized to achieve the cooling setpoint under "periodic steady-state" summer design condition plus a margin of 15%. However this 15% margin will not be adequate if the cooling is to be switched on at say 6am and expected to lower the zone temperature to comfortable levels in time for occupancy a few hours later. To deal with intermittent operation, either much higher design factors are required (e.g. 2) or the system should be run continuously (perhaps at a lower level during unoccupied periods) to avoid the need for a rapid cooling down in the mornings. Of course if the cooling is to be operated continuously then the building fabric will need to be very well insulated to avoid waste.
7. **Incorrect position of the source.** A common error with setting up the chilled ceiling definition is to select the wrong position for the chilled ceiling source (i.e. the chilled water pipes). In a well designed system the pipes will normally be buried just below the first construction layer. If you position the tubes at the wrong side of the construction, then the wrong zone (or the ground or exterior) will be cooled! A [tool](#) is provided on the Constructions dialog to make it easy to position the source just above the innermost layer.
8. **Conductive lower layer.** Chilled ceilings provide the most efficient and responsive control when the uppermost layer is low mass and conductive. Chilled ceiling panels are often constructed using a thin metal layer between source and the room to provide an efficient heat transfer path from cooling pipes to the zone. Clearly, if the inside layer is an (insulating) ceiling tile for example then the chilled ceiling will struggle to provide adequate cooling to the room.
9. **Insulation.** Without good insulation above the chilled ceiling source, much of the cooling will not find its way into the intended zone. Instead the cooling will be either lost to the zone above in an uncontrolled way, to outside or to the ground. There will consequently be less cooling available to cool the intended zone and under-cooling will occur.
10. When used **in combination with a heated floor in zone above**, chilled ceilings will cause EnergyPlus to generate an error. This is because EnergyPlus only allows one "source" object per surface and so a chilled ceiling cannot be located in the same surface as a heated floor. There are 2 possible workarounds to this.
 - a. Export the DesignBuilder model to EnergyPlus and using the IDF Editor combine the heated floor and chilled ceiling into a single source component.
 - b. Insert a very thin "dummy zone" between the heated floor above and the chilled ceiling below. Include half of the true floor mass in the upper surface (heated floor in upper zone) and the other half in the lower surface (chilled ceiling in lower zone).

Tip: To help diagnose some of the above problems it can help to view hourly inside and outside surface temperatures and heat fluxes by selecting these output options: [Surface heat transfer incl. solar](#) (gives heat flux for floors) or [Inside surface temperature](#) (to check the surface temperature of the chilled ceiling).

Heated Floor above Chilled Ceiling

In some cases it may be necessary to model a heated floor heating a zone above with a chilled ceiling on the same surface cooling a different zone below. However, EnergyPlus does not allow more than one internal source per surface, so in this case it would be necessary to create a fictitious zone between the 2 surfaces allowing the heated floor and chilled ceiling to be created in their respective zones and to be controlled individually. and satisfying the requirement of no more than one internal source per surface. Note that the restrictions referred to above apply equally to radiant surfaces used to model heated floors and chilled ceiling as to the specific heated floor and chilled ceiling components.

Target Tab

When editing the attributes associated with a radiant surface component it is possible to apply the same changes to units in other zones in the same HVAC Zone group. To do this select the components on the **Target** tab of the edit dialog as required.

Radiant Surface - Constant Flow

The constant flow radiant surface keeps flow rate constant via a local circulation pump and varies the water temperature that is sent to the surface. This is accomplished with a mixing valve that is controlled by a sensor. The constant flow type has a built-in local secondary loop, which re-circulates flow coming out of the system and mixes this with flow from the loop to arrive at the desired inlet temperature to the surface (note that this radiant surface model has the temperature sensor **after** the local secondary pump to ensure proper inlet temperature to the surface). The local loop also contains a pump which is assumed to be upstream of the component and after the mixing valve. So, the local loop can have some recirculation. The flow from the main loop may also bypass the component if more than enough flow is available and the main loop is also a constant flow system.

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the radiant surface is located.

Type

There are two types of radiant surface available:

- **1-Constant flow**
- **2-Variable flow**
- **3-Electric**

Temperature control type

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

Along with setpoint (control) and water schedules, this setting allows you to specify how the radiant surface is to be controlled. The temperature denoted in the setpoint schedule can refer to one of five different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. The choice of temperature is controlled by the temperature control type. The user must select from the following options:

- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature**
- **4-Outdoor dry bulb temperature ,**
- **5-Outdoor wet bulb temperature**

Operative temperature for heated floor controls is the average of Mean Air Temperature and Mean Radiant Temperature. See the control temperature schedule settings below for more information.

Tube Settings

Hydronic tubing inside diameter

This is the inside diameter of the tubes through which water is circulated (in m or in). The inside diameter is used to determine the convective heat transfer from the water to the inside surface of the hydronic tubing.

Hydronic tubing length

This is the total length of embedded pipe (in m or ft). The length of the tube is used to determine the effectiveness of heat transfer from the fluid being circulated through the tubes and the tube/surface. Longer tubing lengths result in more heat that will be transferred to/from the radiant surface to the circulating fluid.

Note: With the constant flow radiant surface, this length is **not** autosizable.

Number of circuits

This input allows you to choose between modelling each surface in the radiant system as a single hydronic circuit or to allow the program to divide the surface into multiple parallel hydronic circuits based on the **Circuit length** (below). The corresponding options are:

- **1-One per surface** to model the system as a single circuit
- **2-Calculate from circuit length** to model the system as multiple circuits

It is recommended that **2-Calculate from circuit length** be chosen for new models. The default is **1-One per surface** for backward compatibility with older versions of DesignBuilder.

Circuit length

The length (in m or ft) of each parallel hydronic circuit in a surface. This data is only used when the **Number of circuits** (above) is set to **2-Calculate from circuit length**. The default is 106.7 meters (350 feet), which is the maximum circuit length allowed in Title 24.

Flow Settings

Rated flow rate

This is the maximum flow rate of water through the heated floor (in m³/sec or gal/min). This flow rate is held constant by the local component pump, but you have the option of varying this flow rate via a schedule (see Pump Flow Rate Schedule). The constant flow system will accept this flow rate and control the inlet temperature based on the control and water temperature schedules defined below.

Pump Settings

Pump flow rate schedule

This schedule modifies the maximum flow rate of water through the radiant surface (in m³/s only). Note that the values for this schedule must be between zero and one.

Rated pump head

This is the pump rated head (Pa or ftH₂O).

Rated power consumption

This is the pump rated power consumption (W).

Motor efficiency

This is the pump efficiency in decimal form (0 = 0%, 1 = 100%).

Fraction of motor inefficiencies to fluid stream

This is the fraction of the pump power lost to the fluid.

Operation

Availability schedule

This is the [schedule](#) that denotes whether the component can run during a given timestep. A schedule value greater than 0 (usually 1 is used) indicates that the unit is available and can be on during the timestep. A

value less than or equal to 0 (usually 0 is used) denotes that the unit is not available and must be off for the timestep.

Heating

Has heating element

If this radiant surface is connected to a hot water loop for heating then this checkbox should be checked.

Note: Either the **Has cooling element** or the **Has heating element** (or both) must be checked.

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

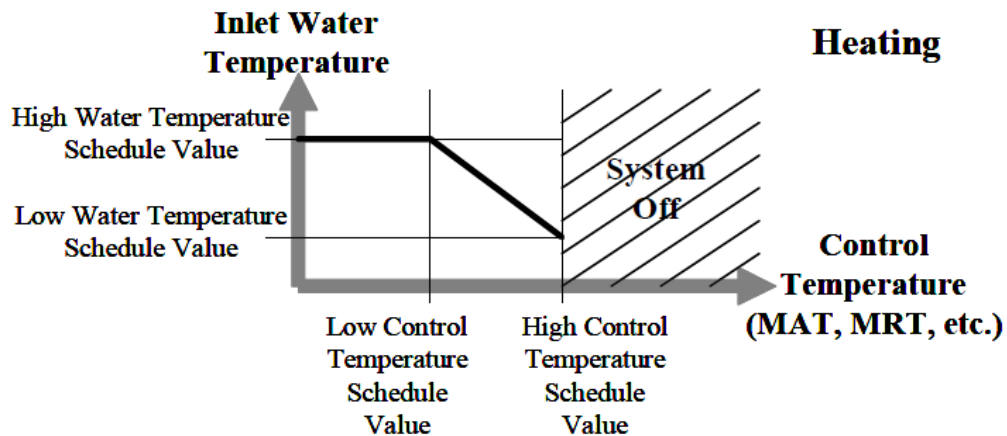
Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Control

Heating high water temperature schedule

This schedule specifies the high water temperature (in °C only) for the temperature control of the constant flow heated floor. Water and control temperatures for heating work together to provide a linear function that determines the water temperature sent to the heated floor. The current control temperature (see Temperature Control Type above) is compared to the high and low control temperatures at the current time. If the control temperature is above the high temperature, then the system will be turned off and the water mass flow rate will be zero. If the control temperature is below the low temperature, then the inlet water temperature is set to the high water temperature. If the control temperature is between the high and low value, then the inlet water temperature is linearly interpolated between the low and high water temperature values.



Heating low water temperature schedule

This schedule specifies the low water temperature (in °C only) for the temperature control of the constant flow heated floor. For more information on its interpretation, see [Heating high water temperature schedule](#) above.

Heating high control temperature schedule

This schedule specifies the high control temperature (in °C only) for the temperature control of a constant flow heated floor. For more information on its interpretation, see [Heating high water temperature schedule](#) above.

Heating low control temperature schedule

This schedule specifies the low control temperature (in °C only) for the temperature control of a constant flow heated floor. For more information on its interpretation, see [Heating high water temperature schedule](#) above.

Cooling

Has cooling element

If this radiant surface is connected to a chilled water loop for cooling then this checkbox should be checked.

Note: Either the **Has cooling element** or the **Has heating element** (or both) must be checked.

Cooling design capacity method

There are 3 ways to define the cooling capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) cooling capacity of the unit is entered, or the program calculates the maximum (or nominal) design cooling capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design cooling capacity from a user specified cooling capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design cooling capacity from user specified fraction and the auto-sized design cooling capacity.

Cooling design capacity

This autosizable field defines the convective electric nominal cooling capacity (in W or Btu/h). It is only available when the Cooling design capacity method is set to **1-Design capacity**.

Cooling design capacity per floor area

Enter the cooling capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Cooling design capacity method is **2-Capacity per floor area**.

The program calculates the cooling capacity from floor area of the zone served by the unit and the cooling capacity per unit floor area value specified here.

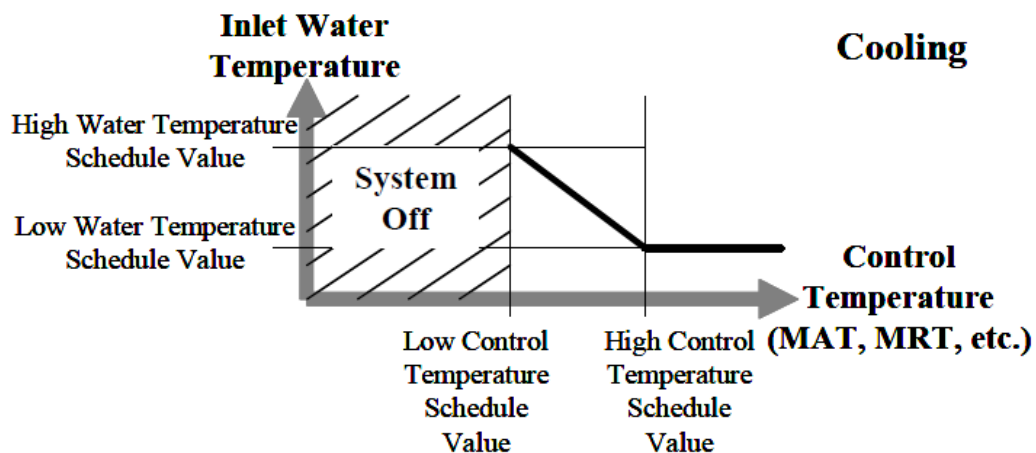
Fraction of autosized cooling design capacity

Enter the cooling capacity as a fraction of the autosized cooling capacity for convective electric baseboard unit. This data is only available when the Cooling design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the cooling capacity from the design autosized cooling capacity and this fraction. The default value is 1.0.

Control

Cooling high water temperature schedule

This [schedule](#) specifies the high water temperature (in °C only) for the temperature control of a constant flow chilled ceiling. Water and control temperatures for cooling work together to provide a linear function that determines the water temperature sent to the chilled ceiling. The current control temperature (see [Temperature control type](#) above) is compared to the high and low control temperatures at the current time. If the control temperature is above the high temperature, then the inlet water temperature is set to the low water temperature. If the control temperature is below the low temperature, then system will be turned off and the water mass flow rate will be zero. If the control temperature is between the high and low value, then the inlet water temperature is linearly interpolated between the low and high water temperature values.



Cooling low water temperature schedule

This [schedule](#) specifies the low water temperature (in °C only) for the temperature control of the constant flow chilled ceiling. For more information on its interpretation, see [Cooling high water temperature schedule](#) above.

Cooling high control temperature schedule

This [schedule](#) specifies the high control temperature (in °C only) for the temperature control of a constant flow chilled ceiling. For more information on its interpretation, see [Cooling high water temperature schedule](#) above.

Cooling low control temperature schedule

This [schedule](#) specifies the low control temperature (in °C only) for the temperature control of a constant flow chilled ceiling. For more information on its interpretation, see [Cooling high water temperature schedule](#) above.

Condensation Control

Condensation control type

With a chilled ceiling, there is the possibility that condensation will occur on the surface that is being cooled. This is due to the fact that the surface temperature may drop below the dew-point temperature of the space. When this occurs, condensation on the surface will occur. There are two options for handling this situation:

- **1-Simple Off** and
- **2-Off**.

If you choose the **2-Off** option, EnergyPlus will not do anything other than produce a warning message when condensation is predicted to occur. The simulation will simply proceed; no moisture will be removed from the zone air and there will be no adjustment of the surface temperature as a result of the condensation. When the **1-Simple off** option is selected, EnergyPlus will predict cases where condensation will occur and shut-off the chilled ceiling to avoid this situation. With this latter option, you also have the opportunity to adjust when the system will shut down. This is specified with the Condensation Control Dew-point Offset, below.

Condensation control dew-point offset

This offset temperature (in °C or °F) is only valid with the **1-Simple off** condensation handling algorithm (see Condensation control type above). It establishes the difference between the calculated dew-point temperature of the space and the allowed surface temperature to which the surface can drop before the chilled ceiling shuts down. This setting can be any positive, negative, or zero value. When this parameter is zero, the chilled ceiling will shut down when the surface temperature drops to the dew-point temperature or below.

When this parameter is **positive**, the radiant system will shut down when the surface is this value **above** the dew-point temperature. This allows some extra safety to avoid condensation. When this parameter is **negative**, the chilled ceiling will shut down when the surface temperature is this value **below** the dew-point temperature. While not recommended, this strategy allows the user to simulate a situation where small amounts of condensation are tolerable.

Note: This value is an offset not an absolute temperature. A typical value might be 0-2°C.

Radiant Surface - Variable Flow

Similar to the [constant flow](#) radiant surface but control of the variable flow radiant surface is accomplished by throttling the hot and/or chilled water flow to the unit.

Note An important advantage of the EnergyPlus variable flow radiant surface over the [constant flow](#) equivalent is that it is fully autosizable.

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the radiant surface is located.

Type

There are 3 types of radiant surface available:

- [1-Constant flow](#)
- [2-Variable flow](#)
- [3-Electric](#)

Temperature control type

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

Along with setpoint (control) and water schedules, this setting allows you to specify how the radiant surface is to be controlled. The temperature denoted in the setpoint schedule can refer to one of five different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. The choice of temperature is controlled by the temperature control type. The user must select from the following options:

- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature**
- **4-Outdoor dry bulb temperature ,**
- **5-Outdoor wet bulb temperature**

Operative temperature for radiant surface controls is the average of Mean Air Temperature and Mean Radiant Temperature. See the control temperature schedule settings below for more information.

Tube Settings

Hydronic tubing inside diameter

This is the inside diameter of the tubes through which water is circulated (in m or in). The inside diameter should be entered in metres and is used to determine the convective heat transfer from the water to the inside surface of the hydronic tubing.

Hydronic tubing length

This is the total length of embedded pipe (in m or ft). The length of the tube is used to determine the effectiveness of heat transfer from the fluid being circulated through the tubes and the tube/surface. Longer tubing lengths result in more heat that will be transferred to/from the radiant surface to the circulating fluid. This data is auto-sizable.

Number of circuits

This input allows you to choose between modelling each surface in the radiant system as a single hydronic circuit or to allow the program to divide the surface into multiple parallel hydronic circuits based on the **Circuit length** (below). The corresponding options are:

- **1-One per surface** to model the system as a single circuit
- **2-Calculate from circuit length** to model the system as multiple circuits

It is recommended that **2-Calculate from circuit length** be chosen for new models. The default is **1-One per surface** for backward compatibility with older versions of DesignBuilder.

Circuit length

The length (in m or ft) of each parallel hydronic circuit in a surface. This data is only used when the **Number of circuits** (above) is set to **2-Calculate from circuit length**. The default is 106.7 meters (350 feet), which is the maximum circuit length allowed in Title 24.

Heating

Has heating element

If this radiant surface is connected to a hot water loop for heating then this checkbox should be checked.

Note: Either the **Has cooling element** or the **Has heating element** (or both) must be checked.

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Maximum hot water flow rate

This is the maximum flow rate of hot water through the radiant surface (in m³/s or gal/min). The controls will vary the flow rate of hot water through the surface(s) using zero flow and the maximum flow rate specified in this field as the lower and upper bounds, respectively. This value is auto-sizable.

Tip: See [Checking Sizing Results for Design](#) below for advice on how to access the autosizing results for **Heating design capacity** and **Maximum hot water flow rate**.

Control

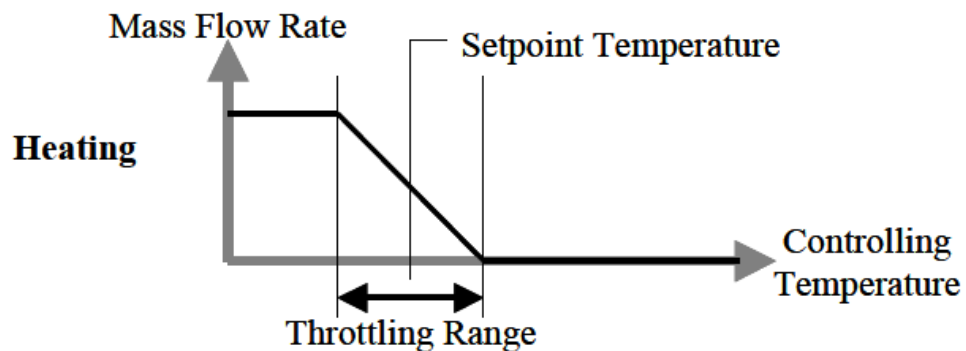
Heating control temperature schedule

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

The Heating control temperature [schedule](#) specifies the heating setpoint or control temperature for the radiant surface (in °C only). Used in conjunction with the [Throttling range](#), it defines whether or not the system is running and the current flow rate. Water flow rate to the system is varied linearly around the setpoint temperature based on the [Throttling range](#) and the [Maximum heating flow rate](#) parameters (see above). It should be noted that this control schedule will allow different setpoint temperatures throughout the year for heating. The control of the radiant surface is based solely on the values in this schedule, and the [Temperature control type](#) described above. The radiant surface will not use the [HVAC zone](#) thermostat that might be used by other systems serving the zone in which the component resides.

Throttling range

This is the range of temperature (°C) which the radiant surface throttles from zero flow rate up to the maximum defined by the [Maximum hot water flow rate](#) described above. The throttling range parameter is used in conjunction with the control temperature to define the response of the system to various zone conditions. The heating control temperature schedule specifies the setpoint temperature where the flow rate to the system is at half of the maximum flow rate. For example, if the heating control temperature setpoint is currently 15°C and the heating throttling range is 2°C, the water flow rate to the system will be zero when the controlling temperature (see Temperature Control Type below) is at or above 16°C and the maximum flow rate when the controlling temperature is at or below 14°C. This represents a throttling range of 2°C around the setpoint of 15°C. In between 14°C and 16°C, the flow rate to the radiant system is varied linearly.



The minimum throttling range is 0.5°C.

Cooling

Has cooling element

If this radiant surface is connected to a chilled water loop for cooling then this checkbox should be checked.

Note: Either the [Has cooling element](#) or the [Has heating element](#) (or both) must be checked.

Cooling design capacity method

There are 3 ways to define the cooling capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) cooling capacity of the unit is entered, or the program calculates the maximum (or nominal) design cooling capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design cooling capacity from a user specified cooling capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design cooling capacity from user specified fraction and the auto-sized design cooling capacity.

Cooling design capacity

This autosizable field defines the convective electric nominal cooling capacity (in W or Btu/h). It is only available when the Cooling design capacity method is set to **1-Design capacity**.

Cooling design capacity per floor area

Enter the cooling capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Cooling design capacity method is **2-Capacity per floor area**.

The program calculates the cooling capacity from floor area of the zone served by the unit and the cooling capacity per unit floor area value specified here.

Fraction of autosized cooling design capacity

Enter the cooling capacity as a fraction of the autosized cooling capacity for convective electric baseboard unit. This data is only available when the Cooling design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the cooling capacity from the design autosized cooling capacity and this fraction. The default value is 1.0.

Maximum cold water flow rate

This is the maximum flow rate of cold water through the radiant surface (in m³/s or gal/min). The controls will vary the flow rate of hot water through the surface(s) using zero flow and the maximum flow rate specified in this field as the lower and upper bounds, respectively. This value is auto-sizable.

Tip: See [Checking Sizing Results for Design](#) below for advice on how to access the autosizing results for **Cooling design capacity** and **Maximum cold water flow rate**.

Control

Cooling control temperature schedule

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

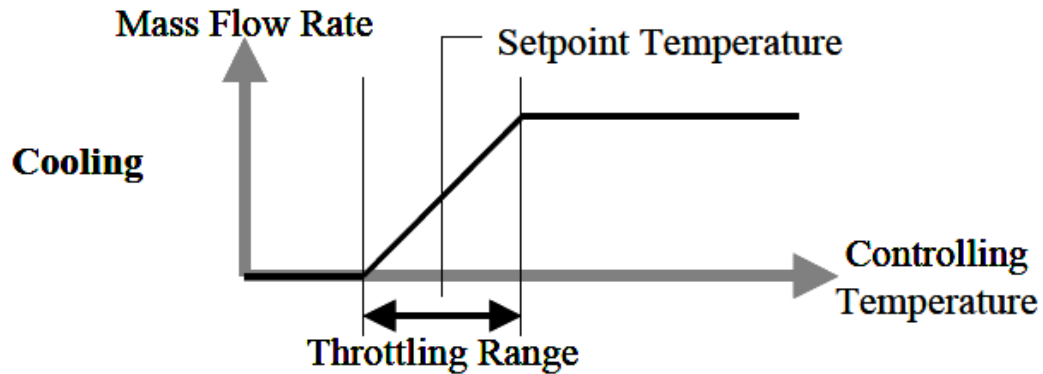
This [schedule](#) specifies the cooling setpoint or control temperature for the radiant surface (in °C only) when in cooling mode. Used in conjunction with the [Throttling range](#), it defines whether or not the system is running and the current flow rate. Water flow rate to the system is varied linearly around the setpoint temperature based on the [Throttling range](#) and the [Maximum cooling flow rate](#) parameters (see above). This control schedule will allow different setpoint temperatures throughout the year for cooling. The control of the radiant surface is based solely on the temperature values in this schedule, and the [Temperature control type](#) listed above. The radiant surface will not use any thermostats defined on the [HVAC zone](#) dialog that might be used by other systems serving the zone in which the component resides.

Tip: Some users have found it helpful to give the radiant surface priority by setting slightly lower cooling setpoint temperatures than the setpoints on the [HVAC zone](#) dialog used to control a supplementary air system.

Throttling range

This is the range of temperature (in °C or °F) over which the radiant surface throttles from zero flow rate up to the maximum defined by the Maximum Cold Water Flow Rate described above. The throttling range parameter is used in conjunction with the control temperature to define the response of the system to various zone conditions. The cooling control temperature schedule specifies the setpoint temperature where the flow rate to the system is at half of the maximum flow rate. For example, if the

cooling control temperature setpoint is currently 25°C and the cooling throttling range is 2°C, the water flow rate to the radiant system will be zero when the controlling temperature (see Temperature Control Type below) is at or below 24°C and the maximum flow rate when the controlling temperature is at or above 26°C. This represents a throttling range of 2°C around the setpoint of 25°C. In between 24°C and 26°C, the flow rate to the radiant system is varied linearly.



The minimum throttling range is 0.5°C.

Condensation Control

Condensation control type

With a chilled ceiling, there is the possibility that condensation will occur on the surface that is being cooled. This is due to the fact that the surface temperature may drop below the dew-point temperature of the space. When this occurs, condensation on the surface will occur. There are two options for handling this situation:

- **1-Simple Off** and
- **2-Off**.

If you choose the **2-Off** option, EnergyPlus will not do anything other than produce a warning message when condensation is predicted to occur. The simulation will simply proceed; no moisture will be removed from the zone air and there will be no adjustment of the surface temperature as a result of the condensation. When the **1-Simple off** option is selected, EnergyPlus will predict cases where condensation will occur and shut-off the radiant surface to avoid this situation. With this latter option, you also have the opportunity to adjust when the system will shut down. This is specified with the Condensation Control Dew-point Offset, below.

Condensation control dew-point offset

This offset temperature (in °C or °F) is only available with the **1-Simple off** condensation control type. It establishes the difference between the calculated dew-point temperature of the space and the allowed surface temperature to which the surface can drop before the radiant surface shuts down. This setting can be any positive, negative, or zero value. When this parameter is zero, the radiant surface will shut down when the surface temperature drops to the dew-point temperature or below.

When this parameter is **positive**, the radiant system will shut down when the surface is this value **above** the dew-point temperature. This allows some extra safety to avoid condensation. When this parameter is **negative**, the radiant surface will shut down when the surface temperature is this value **below** the dew-point temperature. While not recommended, this strategy allows the user to simulate a situation where small amounts of condensation are tolerable.

Note: This value is an offset not an absolute temperature. A typical value might be 0-2°C.

Operation

Availability Schedule

This is the [schedule](#) that determines whether or not the component is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Checking Sizing Results For Design

When Design heating/cooling capacity, Maximum hot/cold water flow rates and tubing lengths are to be autosized, you can find the Component Sizing Summary results on the [Summary](#) output generated by EnergyPlus. It is important to note that the "Design Size Heating Design Capacity", "Design Size Maximum Hot Water Flow", "Design Size Cooling Design Capacity", "Design Size Maximum Cold Water Flow" and "Design Size Hydronic Tubing Length" values reported are the sum for *all the active surfaces linked to the zone radiant surface object*. So these results cannot be used directly to calculate the fluid velocity. Fluid velocity, must be hand calculated by the user for the number of active surfaces when checking pressure drop and fluid turbulence. To help with that you can find the flow fraction for each EnergyPlus surface in the ZoneHVAC:LowTemperatureRadiant:SurfaceGroup data associated with each ZoneHVAC:LowTemperatureRadiant:VariableFlow object.

Radiant Surface - Electric

This low temperature electric radiant system is a component of zone equipment that is intended to model any "radiant system" where electric resistance heating is used to supply energy (heat) to a building surface (wall, ceiling, or floor).

In cases where no other systems are serving the zone in which this system resides it uses the heating equipment priority to determine which system will run first. If the radiant system is serving a zone with forced air equipment, the radiant system will follow the priority order established by the zone thermostat but will still base its response on the controls defined for the radiant system.

Control is accomplished by varying the electrical power supplied to the unit based on a setpoint and a deadband. This model covers either a radiant panel system or wires embedded in entire walls, floors, or ceilings. It is not intended to simulate high temperature electric or gas radiant heaters. Those devices are handled by the [High temperature radiant heaters](#).

General

Name

This field is a unique pre-defined user name for this radiant surface.

Type

Zone temperature control type

This option, along with the throttling range and heating setpoint, defines how the low temperature electric radiant system is to be controlled. The temperature defined in the setpoint schedule (whether defined here on this dialog or by using Simple HVAC activity settings) can refer to one of 5 different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. Select from the following options:

- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature** - average of Mean air temperature and Mean radiant temperature

- **4-Mean air temperature setpoint**
- **5-Mean radiant temperature setpoint**
- **6-Operative temperature setpoint**

See the [throttling range](#) and [control temperature schedule](#) fields below for more information.

Operation

Availability schedule

Select the schedule that denotes whether the electric low temperature radiant system can operate during a given time period. A schedule value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for that time period. A value greater than 0 (usually 1 is used) denotes that the unit is available to operate during that time period.

Heating

Heating design capacity method

Select the method to be used to determine the maximum power output of the unit. Allowed settings are:

- **1-Heating design capacity** - the maximum or nominal heating capacity is specified, or when autosize is entered the program calculates the maximum capacity.
- **2-Capacity per floor area** - the program calculates the design heating capacity from user specified heating capacity per floor area and floor area of the zone served by the low temperature radiant electric unit.
- **3-Fraction of autosized heating capacity** - the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field is available when the Heating design capacity method is set to **1-Heating design capacity**. It defines the maximum electric power converted into heat in the low temperature radiant electric unit (in W or Btu/h).

The controls for the radiant system vary the amount of power supplied to the surface between zero input and the maximum power specified in this field as the lower and upper bounds, respectively.

Heating design capacity per floor area

This field is available when the Heating design capacity method is set to **2-Capacity per floor area**. It defines the heating capacity per unit floor area (in W/m² or W/ft²) of the low temperature radiant electric system. The program calculates the heating capacity from the floor area of the zone served by the low temperature radiant electric unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

This field is available when the Heating design capacity method is set to **3-Fraction of autosized heating capacity**. It defines the heating capacity of the radiant surface as a fraction of the zone autosized heating capacity. The program calculates the heating capacity from the design autosized heating capacity and this fraction. Design day sizing must be specified. The default value is 1.0.

Heating setpoint temperature schedule

When [Detailed HVAC activity data](#) is set to **2-Detailed HVAC data** you can select a schedule to define the heating setpoint for the radiant system (in °C or °F). Used in conjunction with the heating throttling range, it defines whether or not the system is running and the power input to the radiant surface. Power input to the system is varied linearly around the setpoint temperature based on the throttling range and the maximum electrical power parameters (see above).

This schedule allows different heating setpoint temperatures to be defined for different periods through the year. In addition, it may be different that the thermostatic control schedule defined for overall operation of components serving the zone in which the radiant system is located. The thermostatic control determines whether or not there is a heating or cooling load in the space and thus whether the systems should be operating. This field simply controls the flow rate to the radiant system.

When [Detailed HVAC activity data](#) is set to **1-Simple HVAC data** the heating setpoint is defined by the [Heating setpoint temperature](#), [Heating setback temperature](#) and [Heating operation schedule](#) and this setting is not available, but otherwise it works as for when using **2-Detailed HVAC data**.

Heating throttling range

This field specifies the temperature range (in delta °C or °F) over which the radiant system varies heat input, from zero up to the maximum defined by the maximum electrical power field described above. The throttling range parameter is used in conjunction with the zone heating control setpoint temperature to define the response of the system to various zone conditions. The zone heating setpoint specifies the temperature where the power input to the system is at half of the Maximum power input. For example, if the setpoint is currently 15°C and the heating throttling range is 2°C, then the electrical power supplied to the radiant system will be zero when the controlling temperature (MAT, MRT, Operative Temperature, ODB, or OWB; see control type field above) is at or above 16°C and the maximum power input when the controlling temperature is at or below 14°C. This represents a throttling range of 2°C around the setpoint of 15°C. For zone temperatures between 14°C and 16°C, the power input to the radiant system is varied linearly. The minimum throttling range is 0.5°C.

Heated Floor



Heated floors are low temperature radiant systems with hot water pipes embedded in the floor construction. They use large surface area and relatively low surface temperature.

These systems typically use low flow temperatures of around 40-55°C compared with around 80°C for traditional low temperature hot water heating systems using radiators and convectors. Lower return temperatures enable more efficient operation using condensing boilers and heat pumps.

See the [Radiant surfaces](#) tutorial

Two types of heated floor are available in DesignBuilder HVAC:

- [Constant flow](#) and
- [Variable flow](#)

Note one advantage of the EnergyPlus [variable flow](#) heated floor over the [constant flow](#) heated floor is that it is fully autosizable.

When using the Detailed HVAC activity option, heated floors will only control based on the radiant system controls defined for the component itself and not via the standard zone thermostat defined on the HVAC zone dialog. If the radiant system is serving a zone with forced air equipment, the radiant system will follow the priority order established by the zone thermostat but will still base its response on the controls defined by the user for the radiant system.

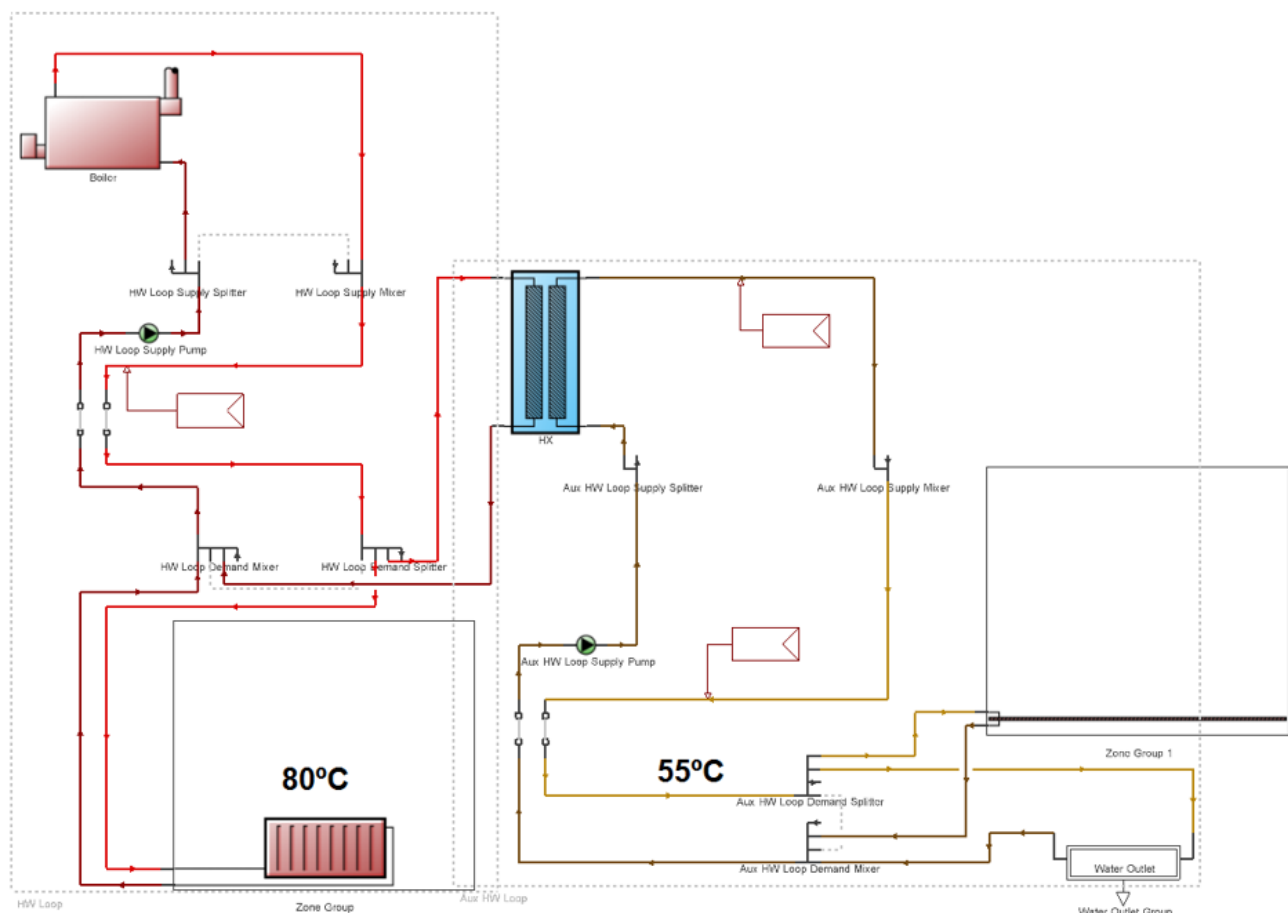
When using the Simple HVAC activity option, heated floors are controlled using the usual HVAC zone heating setpoint temperature controls.

Important Note: In zones in which heated floors are included, at least one floor construction must have a construction with internal source defined, which incorporates details of the embedded tubing system. See [Internal Source](#) under Constructions for further details.

Heated floor HVAC components are placed, edited and deleted at [HVAC zone](#) level. To edit the data associated with a heated floor, you first need to select it by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the Edit selected component tool from the toolbar.

Tip: Heated floors can be connected to [hot water loops](#) fed by either [boiler](#) or [GSHP](#) systems. Note that when connected to GSHP systems heated floors can offer a very efficient system as the low delivery temperatures they require (typically 35°C) are well matched to the optimal operating conditions for GSHP systems.

The screenshot below illustrates how a heated floor system may be connected to a boiler system which also supplies to radiators at a higher temperature.



Target Tab

When editing the attributes associated with a Heated floor component it is possible to apply the same changes to units in other zones in the same HVAC Zone group. To do this select the components on the **Target** tab of the edit dialog as required.

Troubleshooting Heated Floors

DesignBuilder EnergyPlus can realistically model heated floor systems and so many of the issues that apply to real systems such as slow response, floor mass, control, floor insulation, effect of different upper surface materials can be investigated using DesignBuilder. However heated floors can take a little bit of care to set up and to achieve good temperature control. If you are having trouble with this you may find the answer to the problem is the list below.

1. **Missing Internal Source Error report.** Each zone with a heated floor added to its HVAC zone must include at least one floor surface (or floor sub-surface) having a construction with an [internal source](#). If no such internal source surface is found, DesignBuilder will generate an error message to this effect before attempting to run the simulation.
2. **Unresponsive control.** Heated floor systems have relatively slow response times as the heating pipes provide indirect heating via the floor surface. This can result in zone temperatures deviating from set-point temperatures if the heating or cooling loads in the zone change quickly, through solar gain or natural ventilation cooling for example. The slow response is due to the high thermal mass of the floor causing heat to continue to be emitted even when the room thermostat has stopped calling for heating and the pump has stopped moving hot water through the embedded pipes. Overheating in such high mass heated floors can be a common problem on days where a high demand exists on a cold morning (for example) but then high solar (or other) gains in the day add to the uncontrolled heat continuing to be emitted from the floor.
3. **Throttling range.** Less of a factor than point 2 is variable flow heated floors work using a throttling range to control flow of water through the embedded floor water pipes. This acts like a deadband and means that even without the thermal mass lag issues, there will be a temperature control range rather than a fixed zone temperature.
4. **Underheating** can also be a common issue with heated floors. Likely causes are listed below.
5. **Large heat loss.** A useful rule of thumb is that it is not usually possible to supply much more than about **100 W/m²** of heating with heated floors due to the thermal resistance of floor surfaces and maximum acceptable floor temperatures of around 29°C, though higher surface temperatures are possible in bathrooms. Typical maximum outputs are approximately 100 W/m² for concrete, reducing to 70 W/m² for timber floors and less for carpet and coverings with insulating properties. Zones having higher levels of heat loss will require supplementary heating to achieve comfortable conditions. This is true of real world systems and with DesignBuilder EnergyPlus heated floor models. The easiest way to check the heat loss against this rule is by using the [Normalise](#) display option which shows results per floor area.
6. **Intermittent heating.** Running heated floors on an intermittent basis requires a higher design sizing factor to be set for the HVAC zone. This is because of the thermal mass of the floor construction. Consider a building unoccupied over a cold weekend which then needs to be heated to operating temperatures on Monday morning. With the default sizing factor of 1.25 the heating system will have been sized to achieve the heating setpoint under steady-state winter outdoor design condition plus a margin of 25%. However this 25% margin will not be adequate if the heating is to be switched on at say 6am and expected to raise the zone temperature to comfortable levels in time for occupancy a few hours later. To deal with intermittent operation, either much higher design factors are required (e.g. 2) or the system should be run continuously (perhaps at a lower level during unoccupied periods) to avoid the need for a rapid warmup in the mornings. Of course if the heating is to be operated continuously the building fabric will need to be very well insulated to avoid waste.
7. **Incorrect position of the source.** A common error with setting up the heated floor definition is to select the wrong position for the heated floor source (i.e. the hot water pipes). In a well designed system the pipes will normally be buried just below the first construction layer. If you position the tubes at the wrong side of the construction, then the wrong zone (or the ground or exterior) will be heated! A [tool](#) is provided on the Constructions dialog to make it easy to position the source just below the innermost layer.
8. **Conductive upper layer.** Heated floors provide the most efficient and responsive control when the uppermost layer is low mass and conductive. While ceilings can be constructed using a thin metal layer between source and the room, this is rarely possible for heated floors which are required to provide a strong and comfortable surface to walk on as well as an efficient heat transfer path from heating pipes to the zone. Clearly, if the top layer is an (insulating) thick pile carpet then the heated floor will struggle to provide adequate heating to the room. More advice on [floor coverings suitable for heated floors](#) is provided below.
9. **Insulation.** Without good insulation below the heated floor source, much of the heat will not find its way into the intended zone. Instead the heat will be either lost to the zone below in an uncontrolled way, to outside or to the ground. There will consequently be less heat available to heat the intended zone and underheating will occur.

10. **Sizing error** ([general HVAC modelling issue](#)). It is important to consider heating and cooling sizing when setting up schedules. In particular, a common mistake is to include internal gains in heating sizing calculations. This has the effect of heating being undersized causing significant underheating (or no heating).
11. When used **in combination with a chilled ceiling in zone below**, heated floors will cause EnergyPlus to generate an error. This is because EnergyPlus only allows one "source" object per surface and so a heated floor cannot be located in the same surface as a chilled ceiling. There are 2 possible workarounds to this.
 - a. Export the DesignBuilder model to EnergyPlus and using the IDF Editor combine the heated floor and chilled ceiling into a single source component.
 - b. Insert a very thin "dummy zone" between the heated floor above and the chilled ceiling below. Include half of the true floor mass in the upper surface (heated floor in upper zone) and the other half in the lower surface (chilled ceiling in lower zone).
12. You may need a separate high-grade heat generator for **DHW** if using a low-temperature heat source for heating.

Tip: To help diagnose some of the above problems and to check that maximum temperatures are compatible with the surface material (below) it can help to view hourly inside and outside surface temperatures and heat fluxes by selecting these output options: [Surface heat transfer incl. solar](#) (gives heat flux for floors) or [Inside surface temperature](#) (to check the surface temperature of the heated floor).

Coverings Suitable For Heated Floors

The advice in this section has been adapted from www.nu-heat.co.uk.

Stone and ceramic surfaces

From the thermal point of view, the best floor coverings for use with underfloor heating are usually hard surfaces such as stone and ceramic tile as they have the least thermal resistance allowing heat from the pipe array to transfer quickly to the surface of the stone/tile.

- **Ceramic tiles** - Due to their high thermal conductivity and slim profile, ceramic tiles are one of the best floor finishes for use with underfloor heating.
- **Polished screed** - Very conductive and good for use with underfloor heating.
- **Limestone** - Stone is the most thermally conductive of all floor coverings allowing energy from the UFH to transfer quickly to the room.
- **Slate** - An extremely conductive natural finish, ideal for use with underfloor heating.
- **Flagstones** - A thicker stone finish that exhibits very good thermal conductivity but beware of adding too much thermal inertia.
- **Marble** - An good heat conductor available in varying thicknesses.

Timber flooring and underfloor heating

Engineered timber is the best of the wooden floor options for use with underfloor heating as its structural stability allows it to perform well with fluctuating temperatures. Solid hardwoods and soft woods also transfer heat well but care should be taken when specifying board width and thickness.

- **Engineered timber** - The best timber floor finish for use with heated floors, offering good heat transfer and structural stability.
- **Solid hardwood** - Good heat transfer and best when specified in narrow board widths.
- **Softwood** - Suitable only in narrow board widths to reduce movement
- **Parquet** - Offers good heat transfer when securely glued to the sub-floor
- **Bamboo** - Usually similar in construction to engineered timber, making it very suitable for UFH

Laminates, vinyl and rubber

Laminates and vinyl floor coverings also perform well with underfloor heating. It is advisable to check the manufacturer's recommended maximum floor surface temperature to ensure the product is suitable for use

with UFH. Most manufacturers state an upper limit of 27°C, which equates to approximately 75W/m² of heat output, which is adequate in most situations. These products have low thermal mass so heat up and cool down quickly in comparison to stone and timber.

- **Vinyl** - good heat transfer, often subject to a recommended maximum surface temperature
- **Amtico** - practical, high-quality finish that transfers heat well
- **Linoleum & Marmoleum** - thin natural products that are suited to use with underfloor heating
- **Laminate tiles** - Thin and thermally conductive making them perfect for use with UFH
- **Rubber** - A relatively thermally conductive material that can successfully be used with underfloor heating

Materials to avoid

While not ideal from the efficiency point of view, carpet and underlay with a combined Tog value of up to 2.5 can be used effectively with underfloor heating. Thicker carpets act as an insulator and stop sufficient heat reaching the room.

- **Carpet** - A maximum Tog value of 2.5 is recommended for carpet and underlay when used with underfloor heating
- **Cork** - A maximum thickness of 10mm is advised as cork is an efficient insulator
- **Coir** - A natural product similar to carpet that is subject to a maximum recommended Tog value of 2.5

You should bear in mind that some flooring manufacturers stipulate a maximum floor temperature for their product. Note also that EN 1264 states that underfloor heating should not operate at more than 29°C.

Heated Floor - Constant Flow

The constant flow heated floor keeps flow rate constant via a local circulation pump and varies the water temperature that is sent to the floor. This is accomplished with a mixing valve that is controlled by a sensor. The constant flow heated floor type has a built-in local secondary loop, which re-circulates flow coming out of the system and mixes this with flow from the loop to arrive at the desired inlet temperature to the floor (note that this heated floor model has the temperature sensor **after** the local secondary pump to ensure proper inlet temperature to the floor). The local loop also contains a pump which is assumed to be upstream of the component and after the mixing valve. So, the local loop can have some recirculation. The flow from the main loop may also bypass the component if more than enough flow is available and the main loop is also a constant flow system.

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the heated floor is located.

Type

There are two types of heated floor available:

- **1-Constant flow** and
- [2-Variable flow](#)

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.

- **3-Fraction of autosized capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Tube Settings

Hydronic tubing inside diameter

This is the inside diameter of the tubes through which water is circulated (in m or in). The inside diameter is used to determine the convective heat transfer from the water to the inside surface of the hydronic tubing.

Hydronic tubing length

This is the total length of embedded pipe (in m or ft). The length of the tube is used to determine the effectiveness of heat transfer from the fluid being circulated through the tubes and the tube/surface. Longer tubing lengths result in more heat that will be transferred to/from the radiant surface to the circulating fluid.

Note: With the constant flow heated floor, this length is **not** autosizable.

Number of circuits

This input allows you to choose between modelling each surface in the radiant system as a single hydronic circuit or to allow the program to divide the surface into multiple parallel hydronic circuits based on the **Circuit length** (below). The corresponding options are:

- **1-One per surface** to model the system as a single circuit
- **2-Calculate from circuit length** to model the system as multiple circuits

It is recommended that **2-Calculate from circuit length** be chosen for new models. The default is **1-One per surface** for backward compatibility with older versions of DesignBuilder.

Circuit length

The length (in m or ft) of each parallel hydronic circuit in a surface. This data is only used when the **Number of circuits** (above) is set to **2-Calculate from circuit length**. The default is 106.7 meters (350 feet), which is the maximum circuit length allowed in Title 24.

Flow Settings

Rated flow rate

This is the maximum flow rate of water through the heated floor (in m³/sec or gal/min). This flow rate is held constant by the local component pump, but you have the option of varying this flow rate via a schedule (see Pump Flow Rate Schedule). The constant flow system will accept this flow rate and control the inlet temperature based on the control and water temperature schedules defined below.

Pump Settings

Pump flow rate schedule

This schedule modifies the maximum flow rate of water through the heated floor (in m³/s only). Note that the values for this schedule must be between zero and one.

Rated pump head

This is the pump rated head (Pa or ftH₂O).

Rated power consumption

This is the pump rated power consumption (W).

Motor efficiency

This is the pump efficiency in decimal form (0 = 0%, 1 = 100%).

Fraction of motor inefficiencies to fluid stream

This is the fraction of the pump power lost to the fluid.

Control

Temperature control type

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

Along with setpoint (control) and water schedules, this setting allows you to specify how the heated floor is to be controlled. The temperature denoted in the setpoint schedule can refer to one of five different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. The choice of temperature is controlled by the temperature control type. The user must select from the following options:

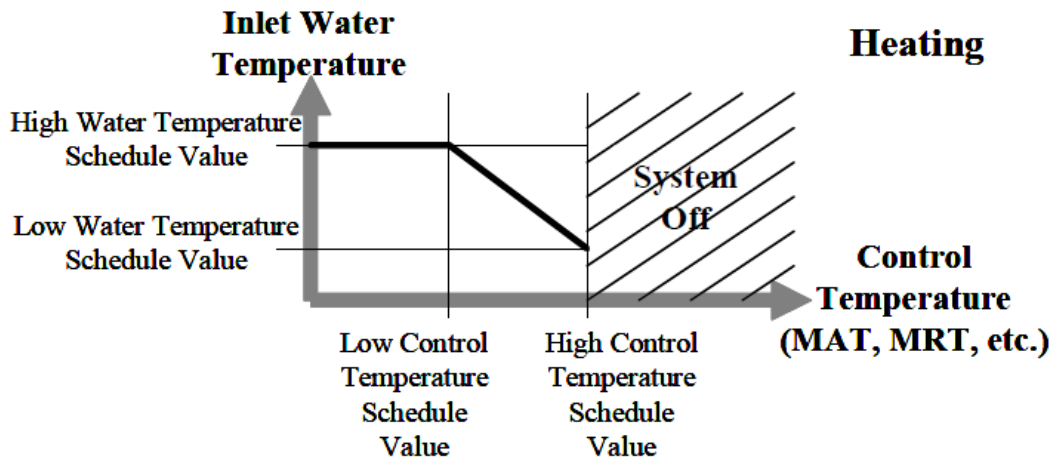
- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature**
- **4-Outdoor dry bulb temperature** ,
- **5-Outdoor wet bulb temperature**

Operative temperature for heated floor controls is the average of Mean Air Temperature and Mean Radiant Temperature. See the control temperature schedule settings below for more information.

Heating high water temperature schedule

This schedule specifies the high water temperature (in °C only) for the temperature control of the constant flow heated floor. Water and control temperatures for heating work together to provide a linear function that determines the water temperature sent to the heated floor. The current control temperature (see Temperature Control Type above) is compared to the high and low control temperatures at the current time. If the control temperature is above the high temperature, then the system will be turned off and the water mass flow rate

will be zero. If the control temperature is below the low temperature, then the inlet water temperature is set to the high water temperature. If the control temperature is between the high and low value, then the inlet water temperature is linearly interpolated between the low and high water temperature values.



Heating low water temperature schedule

This schedule specifies the low water temperature (in °C only) for the temperature control of the constant flow heated floor. For more information on its interpretation, see [Heating high water temperature schedule](#) above.

Heating high control temperature schedule

This schedule specifies the high control temperature (in °C only) for the temperature control of a constant flow heated floor. For more information on its interpretation, see [Heating high water temperature schedule](#) above.

Heating low control temperature schedule

This schedule specifies the low control temperature (in °C only) for the temperature control of a constant flow heated floor. For more information on its interpretation, see [Heating high water temperature schedule](#) above.

Operation

Availability schedule

This is the [schedule](#) that denotes whether the component can run during a given timestep. A schedule value greater than 0 (usually 1 is used) indicates that the unit is available and can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit is not available and must be off for the timestep.

Heated Floor - Variable Flow

Similar to the [constant flow](#) heated floor but control of the variable flow heated floor is accomplished by throttling the hot water flow to the unit.

Note An important advantage of the EnergyPlus variable flow heated floor over the [constant flow](#) heated floor is that it is fully autosizable.

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the heated floor is located.

Type

There are two types of heated floor available:

- [1-Constant flow](#) and
- [2-Variable flow](#).

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- [1-Design capacity](#) where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- [2-Capacity per floor area](#) where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- [3-Fraction of autosized capacity](#) which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to [1-Design capacity](#).

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is [2-Capacity per floor area](#).

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is [3-Fraction of autosized capacity](#). EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Tube Settings

Hydronic tubing inside diameter

This is the inside diameter of the tubes through which water is circulated (in m or in). The inside diameter should be entered in metres and is used to determine the convective heat transfer from the water to the inside surface of the hydronic tubing.

Hydronic tubing length

This is the total length of embedded pipe (in m or ft). The length of the tube is used to determine the effectiveness of heat transfer from the fluid being circulated through the tubes and the tube/surface. Longer tubing lengths result in more heat that will be transferred to/from the radiant surface to the circulating fluid. This data is auto-sizable.

Number of circuits

This input allows you to choose between modelling each surface in the radiant system as a single hydronic circuit or to allow the program to divide the surface into multiple parallel hydronic circuits based on the [Circuit length](#) (below). The corresponding options are:

- **1-One per surface** to model the system as a single circuit
- **2-Calculate from circuit length** to model the system as multiple circuits

It is recommended that **2-Calculate from circuit length** be chosen for new models. The default is **1-One per surface** for backward compatibility with older versions of DesignBuilder.

Circuit length

The length (in m or ft) of each parallel hydronic circuit in a surface. This data is only used when the **Number of circuits** (above) is set to **2-Calculate from circuit length**. The default is 106.7 meters (350 feet), which is the maximum circuit length allowed in Title 24.

Flow Settings

Maximum hot water flow rate

This is the maximum flow rate of hot water through the heated floor (in m³/s or gal/min). The controls for the floor will vary the flow rate of hot water through the surface(s) using zero flow and the maximum flow rate specified in this field as the lower and upper bounds, respectively.

Control

Temperature control type

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

Along with setpoint (control) and water schedules, this setting allows you to specify how the heated floor is to be controlled. The temperature denoted in the setpoint schedule can refer to one of five different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. The choice of temperature is controlled by the temperature control type. The user must select from the following options:

- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature**
- **4-Outdoor dry bulb temperature** ,
- **5-Outdoor wet bulb temperature**

Operative temperature for heated floor controls is the average of Mean Air Temperature and Mean Radiant Temperature. See the control temperature schedule settings below for more information.

Heating control temperature schedule

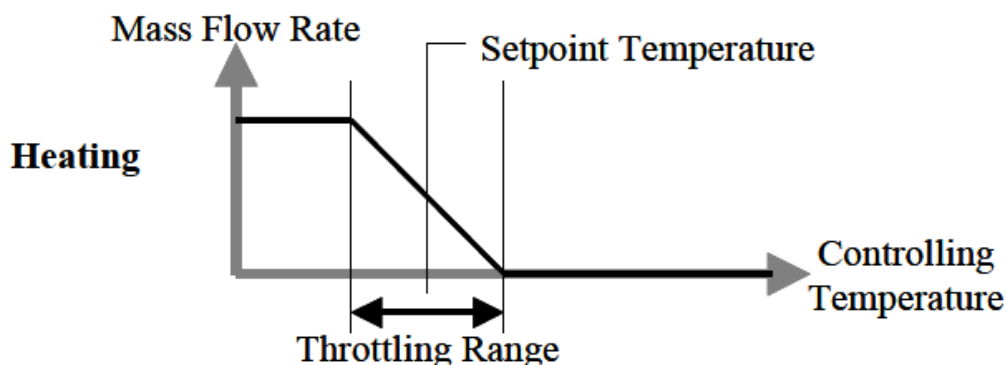
This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

The Heating control temperature [schedule](#) specifies the heating setpoint or control temperature for the heated floor (in °C only). Used in conjunction with the [Throttling range](#), it will define whether or not the system is running and the current flow rate. Water flow rate to the system is varied linearly around the setpoint temperature based on the [Throttling range](#) and the [Maximum heating flow rate](#) parameters (see above). It should be noted that this control schedule will allow different setpoint temperatures throughout the year for heating. The control of the heated floor is based solely on the values in this schedule, and the **Temperature control type** described above. The heated floor will not use the [HVAC zone](#) thermostat that might be used by other systems serving the zone in which the component resides.

Throttling range

This is the range of temperature (°C) which the heated floor throttles from zero flow rate up to the maximum defined by the [Maximum hot water flow rate](#) described above. The throttling range parameter is used in conjunction with the control temperature to define the response of the system to various zone conditions. The heating control temperature schedule specifies the setpoint temperature where the flow rate to the system is

at half of the maximum flow rate. For example, if the heating control temperature setpoint is currently 15°C and the heating throttling range is 2°C, the water flow rate to the system will be zero when the controlling temperature (see Temperature Control Type below) is at or above 16°C and the maximum flow rate when the controlling temperature is at or below 14°C. This represents a throttling range of 2°C around the setpoint of 15°C. In between 14°C and 16°C, the flow rate to the radiant system is varied linearly.



The minimum throttling range is 0.5°C.

Operation

Availability Schedule

This is the [schedule](#) that determines whether or not the component is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Chilled Ceiling



Chilled ceilings are mounted within the ceiling to provide quiet, draft free cooling of the space below. Each unit is made up of a small bore chilled water pipe arranged in an S-shape and attached to the upper surface of a ceiling panel. Ceiling panels are typically of thin metallic construction but can simply be plasterboard. In some systems the chilled water pipes are embedded within the ceiling panel.

During operation, the panel is cooled through contact with the chilled-water pipework allowing it to cool the space through a combination of convective and radiant output (up to 40% radiant). An insulating mat is often placed above the chilled water pipework and panel to minimise uncontrolled cooling of the area above. It is up to the user to include any such insulation in the construction definition.

One advantage of chilled ceilings is that they can be placed in a shallow ceiling void enabling them to be used in buildings with low floor to ceiling heights. However, their limited cooling output makes them unsuitable for areas with moderate to high heat gains. The maximum capacity of chilled ceiling systems is in the order of 70 W/m².

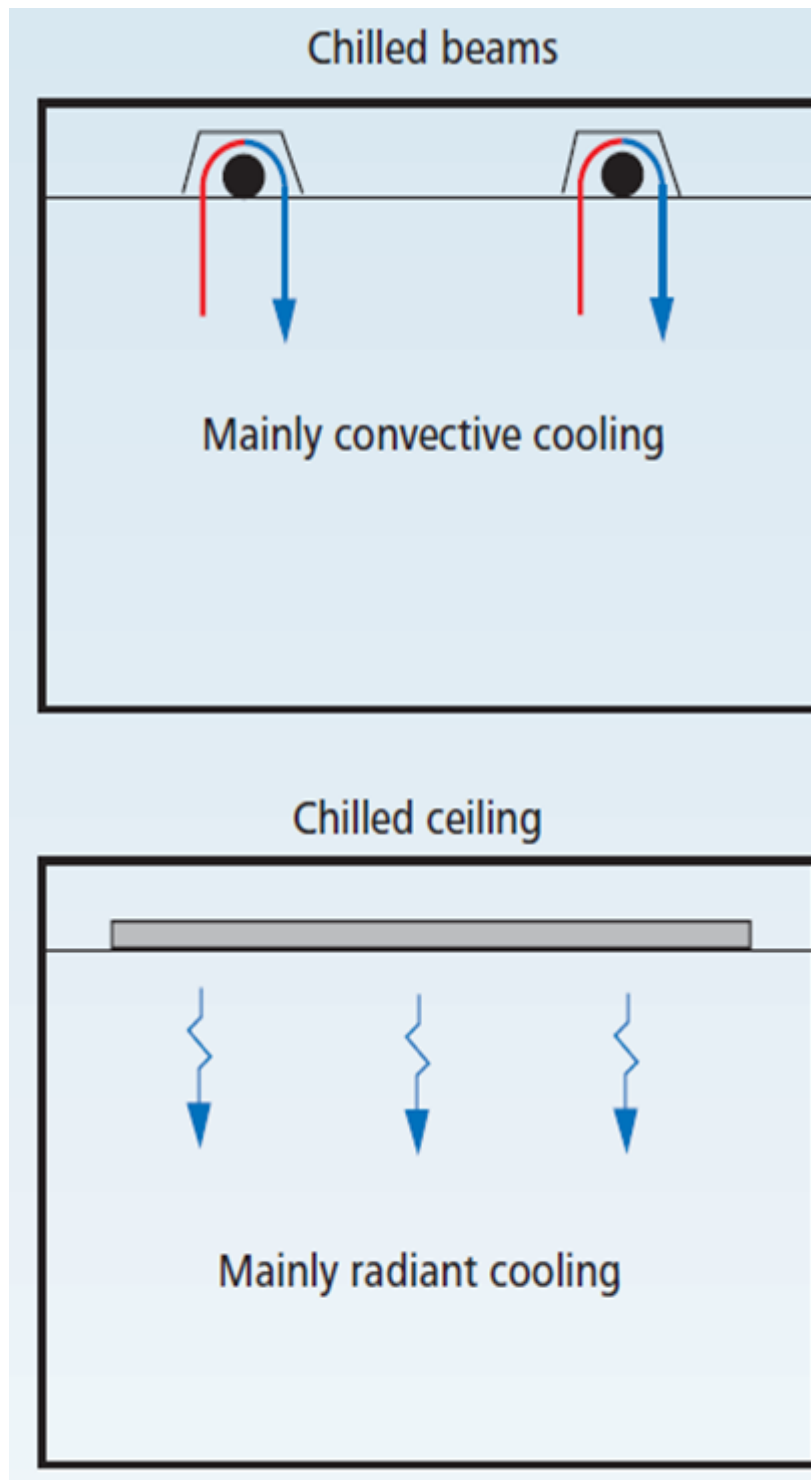
Chilled-ceiling systems require a separate ventilation system for fresh air supply.

Condensation of room air on and within the chilled ceiling can be avoided by shutting of chilled water flow based on room dew point temperature.

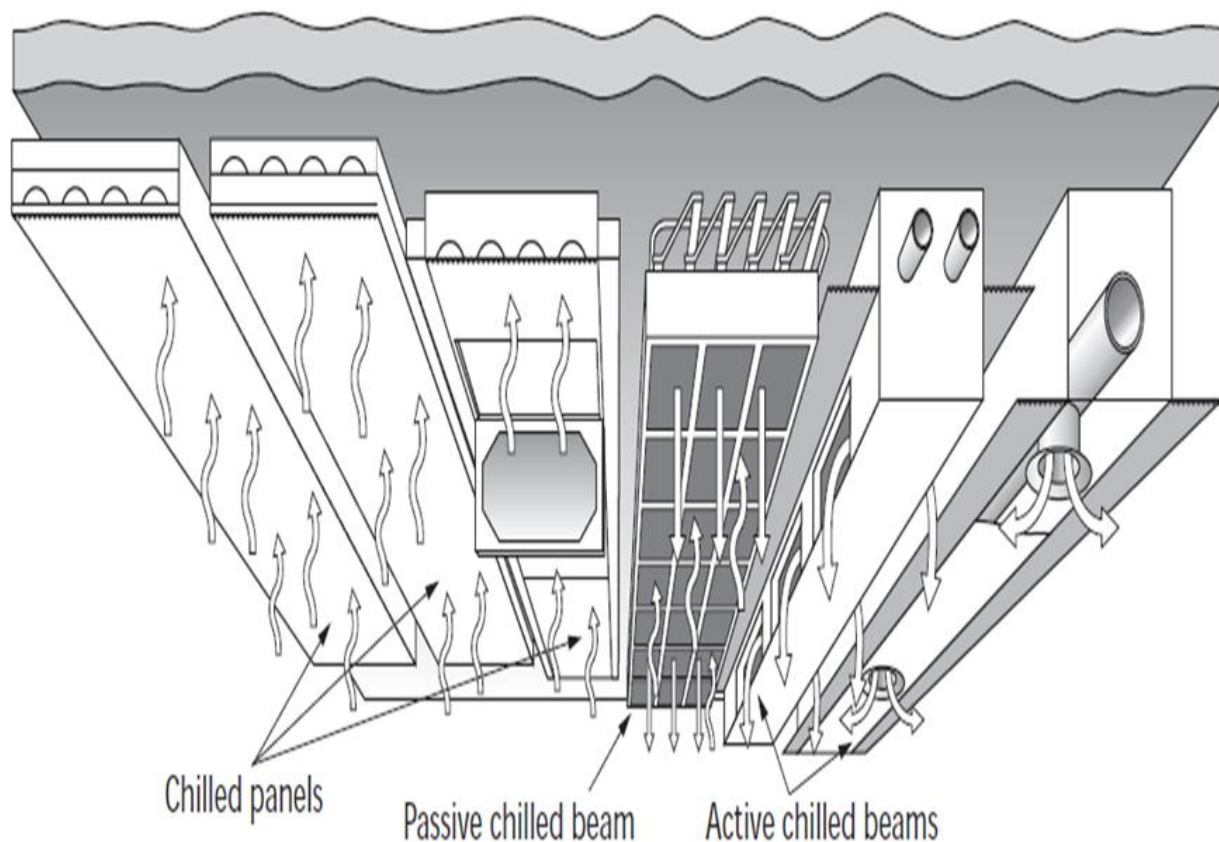
Tip: Chilled ceilings can be connected to chilled water loops fed by either [chillers](#) or [GSHP](#) systems.

Chilled ceilings vs Cooled beams

The images below (reproduced with permission from CIBSE) illustrate the difference between the chilled ceilings described here and [cooled beams](#).



CIBSE KS03 Figure 12



CIBSE Guide H Fig 5.79

Two types of chilled ceiling are available in DesignBuilder HVAC:

- [Constant flow](#) and,
- [Variable flow](#).

Note one advantage of the EnergyPlus [variable flow](#) chilled ceiling over the [constant flow](#) chilled ceiling is that it is fully autosizable.

When using the [Detailed HVAC activity option](#), chilled ceilings will only control based on the radiant system controls defined for the component and not via a zone thermostat such as is used for forced air systems. If the radiant system is serving a zone with forced air equipment, the radiant system will follow the priority order established by the zone thermostat but will still base its response on the controls defined by the user for the radiant system.

When using the [Simple HVAC activity option](#), chilled ceilings are controlled using the usual HVAC zone cooling setpoint temperature controls.

Important Note: At least one ceiling/roof construction in zones in which chilled ceilings are included must have a special internal source construction which incorporates details of the embedded tubing system. See [Internal Source](#) under Constructions for further details.

Chilled ceiling HVAC components are placed, edited and deleted at [HVAC zone](#) level. To edit the data associated with a chilled ceiling, you first need to select it by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the Edit selected component tool from the toolbar.

Tip: A useful resource on chilled ceilings and cooled beams can be found at https://www.feta.co.uk/uploaded_images/files/CBCA/Chilled%20Beams%20Brochure_Final%207%20%28web%29.pdf

Target Tab

When editing the attributes associated with a Chilled ceiling it is possible to apply the same changes to units in other zones in the same HVAC Zone group. To do this select the components on the **Target** tab of the edit dialog as required.

Troubleshooting Chilled Ceilings

The DesignBuilder EnergyPlus ceiling model is able to realistically model the operation of chilled ceiling systems and many of the issues that apply to real systems such as intermittency, thermal mass, control, ceiling insulation, effect of different interior surface materials can be investigated using DesignBuilder. However chilled ceilings can take a little bit of care to set up and to achieve good temperature control. If you are having trouble with this you may find the answer to the problem is the list below.

1. **Missing Internal Source Error report.** Each zone with a chilled ceiling added to its HVAC zone must include at least one ceiling surface (or ceiling sub-surface) having a construction with an [internal source](#). If no internal source surface is found, DesignBuilder will provide an error message to this effect before attempting to run the simulation.
2. **Unresponsive control.** Chilled ceilings in real buildings tend to give less responsive control than air or other radiant systems. This is because the relatively high thermal mass of the ceiling causes zone heat to continue to be absorbed even when the room thermostat has stopped calling for cooling and the pump has stopped moving chilled water through the embedded pipes.
3. **Throttling range.** Less of a factor than point 2 is variable flow chilled ceilings work using a throttling range to control flow of water through the embedded floor water pipes. This acts like a deadband and means that even without the thermal mass lag issues, there will be a temperature control range rather than a fixed zone temperature.
4. **Inadequate cooling** can also a common issue with chilled ceilings. Likely causes are listed below.
5. **High levels of internal gains.** A useful rule of thumb is that it is not usually possible to supply much more than about **70 W/m²** of cooling with chilled ceilings. Zones having higher levels of gains will require supplementary cooling to achieve comfortable conditions. This is true of real world systems and with DesignBuilder EnergyPlus chilled ceiling models. The easiest way to check the internal gains against this rule is by using the [Normalise](#) display option which shows results per floor area.
6. **Intermittent cooling.** Running chilled ceilings on an intermittent basis can require a higher design sizing factor to be set for the HVAC zone. This is because the thermal mass of the ceiling construction causes a lag in cooling provided to the coils and cooling provided to the room. Consider a building unoccupied over a warm weekend which then needs to be cooled to operating temperatures on Monday morning. With the default sizing factor of 1.15 the cooling system will have been sized to achieve the cooling setpoint under "periodic steady-state" summer design condition plus a margin of 15%. However this 15% margin will not be adequate if the cooling is to be switched on at say 6am and expected to lower the zone temperature to comfortable levels in time for occupancy a few hours later. To deal with intermittent operation, either much higher design factors are required (e.g. 2) or the system should be run continuously (perhaps at a lower level during unoccupied periods) to avoid the need for a rapid cooling down in the mornings. Of course if the cooling is to be operated continuously then the building fabric will need to be very well insulated to avoid waste.
7. **Incorrect position of the source.** A common error with setting up the chilled ceiling definition is to select the wrong position for the chilled ceiling source (i.e. the chilled water pipes). In a well designed system the pipes will normally be buried just below the first construction layer. If you position the tubes at the wrong side of the construction, then the wrong zone (or the ground or exterior) will be cooled! A [tool](#) is provided on the Constructions dialog to make it easy to position the source just above the innermost layer.
8. **Conductive lower layer.** Chilled ceilings provide the most efficient and responsive control when the uppermost layer is low mass and conductive. Chilled ceilings panels are often constructed using a thin metal layer between source and the room to provide an efficient heat transfer path from cooling pipes to the zone. Clearly, if the inside layer is an (insulating) ceiling tile for example then the chilled ceiling will struggle to provide adequate cooling to the room.

9. **Insulation.** Without good insulation above the chilled ceiling source, much of the cooling will not find its way into the intended zone. Instead the cooling will be either lost to the zone above in an uncontrolled way, to outside or to the ground. There will consequently be less cooling available to cool the intended zone and under-cooling will occur.
10. When used **in combination with a heated floor in zone above**, chilled ceilings will cause EnergyPlus to generate an error. This is because EnergyPlus only allows one "source" object per surface and so a chilled ceiling cannot be located in the same surface as a heated floor. There are 2 possible workarounds to this.
 - a. Export the DesignBuilder model to EnergyPlus and using the IDF Editor combine the heated floor and chilled ceiling into a single source component.
 - b. Insert a very thin "dummy zone" between the heated floor above and the chilled ceiling below. Include half of the true floor mass in the upper surface (heated floor in upper zone) and the other half in the lower surface (chilled ceiling in lower zone).

Tip: To help diagnose some of the above problems it can help to view hourly inside and outside surface temperatures and heat fluxes by selecting these output options: [Surface heat transfer incl. solar](#) (gives heat flux for floors) or [Inside surface temperature](#) (to check the surface temperature of the chilled ceiling).

Chilled Ceiling - Constant Flow



ZoneHVAC:LowTemperatureRadiant:VariableFlow

Used in:

- HVAC Zones

The **constant flow chilled ceiling** keeps flow rate constant via a local circulation pump and varies the water temperature that is sent to the ceiling. This is accomplished with a mixing valve that is controlled by a sensor. The constant flow chilled ceiling type has a built-in local secondary loop. It will re-circulate flow coming out of the system and mix this with flow from the loop to arrive at the desired inlet temperature to the ceiling (note that this chilled ceiling model has the temperature sensor **after** the pump to ensure proper inlet temperature to the ceiling). The local loop also contains a pump which is assumed to be upstream of the component and after the mixing valve. So, the local loop can have some recirculation. The flow from the main loop may also bypass the component if more than enough flow is available and the main loop is also a constant flow system.

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the chilled ceiling is located.

Type

There are two types of chilled ceiling available:

- **1-Constant flow** and
- **2-Variable flow**

Cooling design capacity method

There are 3 ways to define the cooling capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) cooling capacity of the unit is entered, or the program calculates the maximum (or nominal) design cooling capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design cooling capacity from a user specified cooling capacity per floor area and floor area of the zone served by the unit.

- **3-Fraction of autosized capacity** which means the program calculates the design cooling capacity from user specified fraction and the auto-sized design cooling capacity.

Cooling design capacity

This autosizable field defines the convective electric nominal cooling capacity (in W or Btu/h). It is only available when the Cooling design capacity method is set to **1-Design capacity**.

Cooling design capacity per floor area

Enter the cooling capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Cooling design capacity method is **2-Capacity per floor area**.

The program calculates the cooling capacity from floor area of the zone served by the unit and the cooling capacity per unit floor area value specified here.

Fraction of autosized cooling design capacity

Enter the cooling capacity as a fraction of the autosized cooling capacity for convective electric baseboard unit. This data is only available when the Cooling design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the cooling capacity from the design autosized cooling capacity and this fraction. The default value is 1.0.

Tube Settings

Hydronic tubing inside diameter

This is the inside diameter (in m or in) of the tubes through which water is circulated. The inside diameter is used to determine the convective heat transfer from the water to the inside surface of the hydronic tubing.

Hydronic tubing length

This is the total length of embedded pipe (in m or ft). The length of the tube is used to determine the effectiveness of heat transfer from the fluid being circulated through the tubes and the tube/surface. Longer tubing lengths result in more heat that will be transferred to/from the radiant surface to the circulating fluid.

Note: With the constant flow chilled ceiling, this length is **not** autosizable.

Number of circuits

This input allows you to choose between modelling each surface in the radiant system as a single hydronic circuit or to allow the program to divide the surface into multiple parallel hydronic circuits based on the **Circuit length** (below). The corresponding options are:

- **1-One per surface** to model the system as a single circuit
- **2-Calculate from circuit length** to model the system as multiple circuits

It is recommended that **2-Calculate from circuit length** be chosen for new models. The default is **1-One per surface** for backward compatibility with older versions of DesignBuilder.

Circuit length

The length (in m or ft) of each parallel hydronic circuit in a surface. This data is only used when the **Number of circuits** (above) is set to **2-Calculate from circuit length**. The default is 106.7 meters (350 feet), which is the maximum circuit length allowed in Title 24.

Flow Settings

Rated flow rate

This is the maximum flow rate of water through the chilled ceiling (in m³/s or gal/min). This flow rate is held constant by the local component pump, but you have the option of varying this flow rate via a schedule (see Pump flow rate schedule below). The constant flow system will accept this flow rate and control the inlet temperature based on the control and water temperature schedules defined below.

Pump Settings

Pump flow rate schedule

This [schedule](#) modifies the maximum flow rate of water through the chilled ceiling (in m³/s only). Note that the values for this schedule must be between zero and one.

Rated pump head

This is the pump rated head (Pa or ftH₂O).

Rated power consumption

This is the pump rated power consumption (W).

Motor efficiency

This is the pump efficiency in decimal form (0 = 0%, 1 = 100%).

Fraction of motor inefficiencies to fluid stream

This is the fraction of the pump power lost to the fluid.

Control

Temperature control type

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

Along with setpoint (control) and water schedules, this setting allows you to specify how the chilled ceiling is to be controlled. The temperature denoted in the setpoint schedule can refer to one of five different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. The choice of temperature is controlled by the temperature control type. Select from the following options:

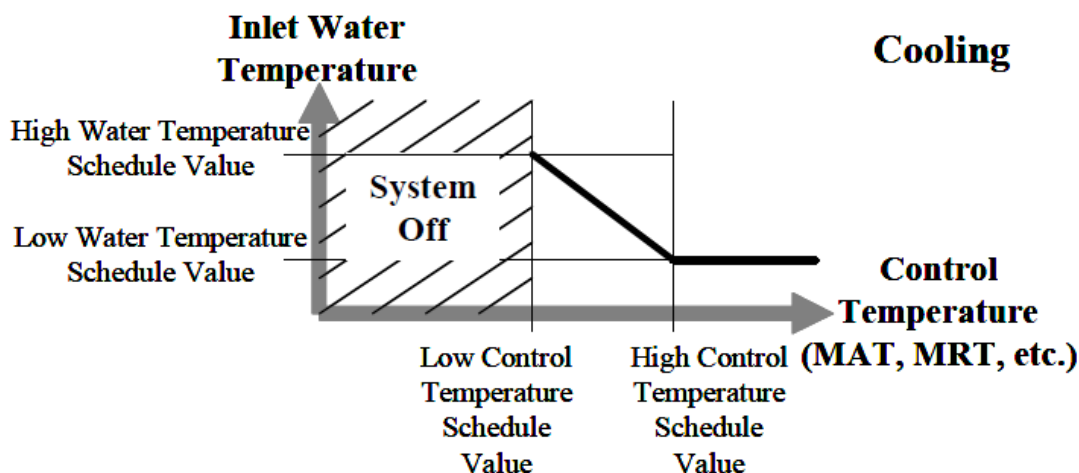
- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature**
- **4-Outdoor dry bulb temperature ,**
- **5-Outdoor wet bulb temperature**

Operative temperature for chilled ceiling controls is the average of Mean air temperature and Mean radiant temperature. See the control temperature schedule settings below for more information.

Cooling high water temperature schedule

This [schedule](#) specifies the high water temperature (in °C only) for the temperature control of a constant flow chilled ceiling. Water and control temperatures for cooling work together to provide a linear function that determines the water temperature sent to the chilled ceiling. The current control temperature (see [Temperature control type](#) above) is compared to the high and low control temperatures at the current time. If the control temperature is above the high temperature, then the inlet water temperature is set to the low water

temperature. If the control temperature is below the low temperature, then system will be turned off and the water mass flow rate will be zero. If the control temperature is between the high and low value, then the inlet water temperature is linearly interpolated between the low and high water temperature values.



Cooling low water temperature schedule

This [schedule](#) specifies the low water temperature (in °C only) for the temperature control of the constant flow chilled ceiling. For more information on its interpretation, see [Cooling high water temperature schedule](#) above.

Cooling high control temperature schedule

This [schedule](#) specifies the high control temperature (in °C only) for the temperature control of a constant flow chilled ceiling. For more information on its interpretation, see [Cooling high water temperature schedule](#) above.

Cooling low control temperature schedule

This [schedule](#) specifies the low control temperature (in °C only) for the temperature control of a constant flow chilled ceiling. For more information on its interpretation, see [Cooling high water temperature schedule](#) above.

Condensation Control

Condensation control type

With a chilled ceiling, there is the possibility that condensation will occur on the surface that is being cooled. This is due to the fact that the surface temperature may drop below the dew-point temperature of the space. When this occurs, condensation on the surface will occur. There are two options for handling this situation:

- **1-Simple Off** and
- **2-Off**.

If you choose the **2-Off** option, EnergyPlus will not do anything other than produce a warning message when condensation is predicted to occur. The simulation will simply proceed; no moisture will be removed from the zone air and there will be no adjustment of the surface temperature as a result of the condensation. When the **1-Simple off** option is selected, EnergyPlus will predict cases where condensation will occur and shut-off the chilled ceiling to avoid this situation. With this latter option, you also have the opportunity to adjust when the system will shut down. This is specified with the Condensation Control Dew-point Offset, below.

Condensation control dew-point offset

This offset temperature (in °C or °F) is only valid with the **1-Simple off** condensation handling algorithm (see Condensation control type above). It establishes the difference between the calculated dew-point temperature of the space and the allowed surface temperature to which the surface can drop before the chilled ceiling

shuts down. This setting can be any positive, negative, or zero value. When this parameter is zero, the chilled ceiling will shut down when the surface temperature drops to the dew-point temperature or below.

When this parameter is **positive**, the radiant system will shut down when the surface is this value **above** the dew-point temperature. This allows some extra safety to avoid condensation. When this parameter is **negative**, the chilled ceiling will shut down when the surface temperature is this value **below** the dew-point temperature. While not recommended, this strategy allows the user to simulate a situation where small amounts of condensation are tolerable.

Note: This value is an offset not an absolute temperature. A typical value might be 0-2°C.

Operation

Availability schedule

This [schedule](#) that denotes whether the component can run during a given hour. A schedule value greater than 0 (usually 1 is used) indicates that the unit is available and can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit is not available and must be off for the hour.

Chilled Ceiling - Variable Flow



ZoneHVAC:LowTemperatureRadiant:VariableFlow

Used in:

- HVAC Zones

Similar to the [constant flow](#) chilled ceiling but control of the variable flow [chilled ceiling](#) is accomplished by throttling the chilled water flow to the unit.

Note An important advantage of the EnergyPlus **variable flow** chilled ceiling over the [constant flow](#) chilled ceiling is that it is fully autosizable.

General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the chilled ceiling is located.

Type

There are two types of chilled ceiling available:

- [1-Constant flow](#) and
- [2-Variable flow](#).

Cooling design capacity method

There are 3 ways to define the cooling capacity of the unit as selected from the following list of options:

- [1-Design capacity](#) where the maximum (or nominal) cooling capacity of the unit is entered, or the program calculates the maximum (or nominal) design cooling capacity, if Autosize is specified.

- **2-Capacity per floor area** where EnergyPlus calculates the design cooling capacity from a user specified cooling capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design cooling capacity from user specified fraction and the auto-sized design cooling capacity.

Cooling design capacity

This autosizable field defines the convective electric nominal cooling capacity (in W or Btu/h). It is only available when the Cooling design capacity method is set to **1-Design capacity**.

Cooling design capacity per floor area

Enter the cooling capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Cooling design capacity method is **2-Capacity per floor area**.

The program calculates the cooling capacity from floor area of the zone served by the unit and the cooling capacity per unit floor area value specified here.

Fraction of autosized cooling design capacity

Enter the cooling capacity as a fraction of the autosized cooling capacity for convective electric baseboard unit. This data is only available when the Cooling design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the cooling capacity from the design autosized cooling capacity and this fraction. The default value is 1.0.

Tube Settings

Hydronic tubing inside diameter

This is the inside diameter of the tubes through which water is circulated (in m or in). The inside diameter is used to determine the convective heat transfer from the water to the inside surface of the hydronic tubing.

Hydronic tubing length

This is the total length of embedded pipe (in m or ft). The length of the tube is used to determine the effectiveness of heat transfer from the fluid being circulated through the tubes and the tube/surface. Longer tubing lengths result in more heat that will be transferred to/from the radiant surface to the circulating fluid. This value is auto-sizable.

Number of circuits

This input allows you to choose between modelling each surface in the radiant system as a single hydronic circuit or to allow the program to divide the surface into multiple parallel hydronic circuits based on the **Circuit length** (below). The corresponding options are:

- **1-One per surface** to model the system as a single circuit
- **2-Calculate from circuit length** to model the system as multiple circuits

It is recommended that **2-Calculate from circuit length** be chosen for new models. The default is **1-One per surface** for backward compatibility with older versions of DesignBuilder.

Circuit length

The length (in m or ft) of each parallel hydronic circuit in a surface. This data is only used when the **Number of circuits** (above) is set to **2-Calculate from circuit length**. The default is 106.7 meters (350 feet), which is the maximum circuit length allowed in Title 24.

Flow Settings

Maximum cold water flow rate

This is the maximum flow rate of cold water through the chilled ceiling (in m³/s or gal/min). The controls for the ceiling will vary the flow rate of hot water through the surface(s) using zero flow and the maximum flow rate specified in this field as the lower and upper bounds, respectively. This value is auto-sizable.

Control

Temperature control type

This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

Along with setpoint (control) and water schedules, this setting allows you to specify how the chilled ceiling is to be controlled. The temperature denoted in the setpoint schedule can refer to one of five different temperatures: the zone mean air temperature, the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. The choice of temperature is controlled by the temperature control type. Select from the following options:

- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature**
- **4-Outdoor dry bulb temperature** ,
- **5-Outdoor wet bulb temperature**

Operative temperature for chilled ceiling controls is the average of Mean air temperature and Mean radiant temperature. See the [Throttling range](#) and [Cooling control temperature schedule](#) settings for more information.

Cooling control temperature schedule

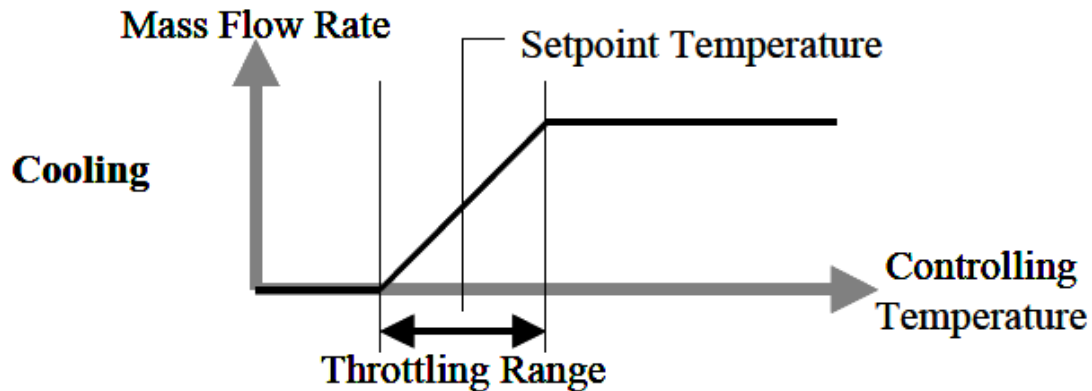
This option is only available when using the **2-Detailed HVAC** [Detailed HVAC Activity data](#)

This [schedule](#) specifies the cooling setpoint or control temperature for the chilled ceiling (in °C only). Used in conjunction with the [Throttling range](#), it will define whether or not the system is running and the current flow rate. Water flow rate to the system is varied linearly around the setpoint temperature based on the [Throttling range](#) and the [Maximum cooling flow rate](#) parameters (see above). This control schedule will allow different setpoint temperatures throughout the year for cooling. The control of the chilled ceiling is based solely on the temperature values in this schedule, and the [Temperature control type](#) listed above. The chilled ceiling will not use any thermostats defined on the [HVAC zone](#) dialog that might be used by other systems serving the zone in which the component resides.

Tip: Some users have found it helpful to give the chilled ceiling priority by setting slightly lower cooling setpoint temperatures than the setpoints on the [HVAC zone](#) dialog used to control a supplementary air system.

Throttling range

This is the range of temperature (in °C or °F) over which the chilled ceiling throttles from zero flow rate up to the maximum defined by the Maximum Cold Water Flow Rate described above. The throttling range parameter is used in conjunction with the control temperature to define the response of the system to various zone conditions. The cooling control temperature schedule specifies the setpoint temperature where the flow rate to the system is at half of the maximum flow rate. For example, if the cooling control temperature setpoint is currently 25°C and the cooling throttling range is 2°C, the water flow rate to the radiant system will be zero when the controlling temperature (see Temperature Control Type below) is at or below 24°C and the maximum flow rate when the controlling temperature is at or above 26°C. This represents a throttling range of 2°C around the setpoint of 25°C. In between 24°C and 26°C, the flow rate to the radiant system is varied linearly.



The minimum throttling range is 0.5°C.

Condensation Control

Condensation control type

With a chilled ceiling, there is the possibility that condensation will occur on the surface that is being cooled. This is due to the fact that the surface temperature may drop below the dew-point temperature of the space. When this occurs, condensation on the surface will occur. There are two options for handling this situation:

- **1-Simple Off** and
- **2-Off**.

If you choose the **2-Off** option, EnergyPlus will not do anything other than produce a warning message when condensation is predicted to occur. The simulation will simply proceed; no moisture will be removed from the zone air and there will be no adjustment of the surface temperature as a result of the condensation. When the **1-Simple off** option is selected, EnergyPlus will predict cases where condensation will occur and shut-off the chilled ceiling to avoid this situation. With this latter option, you also have the opportunity to adjust when the system will shut down. This is specified with the Condensation Control Dew-point Offset, below.

Condensation control dew-point offset

This offset temperature (in °C or °F) is only available with the **1-Simple off** condensation control type. It establishes the difference between the calculated dew-point temperature of the space and the allowed surface temperature to which the surface can drop before the chilled ceiling shuts down. This setting can be any positive, negative, or zero value. When this parameter is zero, the chilled ceiling will shut down when the surface temperature drops to the dew-point temperature or below.

When this parameter is **positive**, the radiant system will shut down when the surface is this value **above** the dew-point temperature. This allows some extra safety to avoid condensation. When this parameter is **negative**, the chilled ceiling will shut down when the surface temperature is this value **below** the dew-point temperature. While not recommended, this strategy allows the user to simulate a situation where small amounts of condensation are tolerable.

Note: This value is an offset not an absolute temperature. A typical value might be 0-2°C.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the component is available for each hour of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the hour.

Radiative-Convective Heater Data

To edit the data associated with a zone radiative-convective heater, you first need to select it by moving the mouse cursor over it and then clicking the mouse button to select it. You can then access the edit dialog by right-clicking the mouse and selecting the Edit selected component option or alternatively, select the Edit selected component tool from the toolbar.

4 types of radiant-convective units are available:

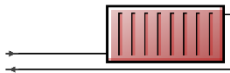
- [Hot Water Convector](#)
- [Electric Convector](#)
- [Hot Water Radiator](#)
- [Electric Radiator](#)
- "High Temperature Radiant Heater" on page 1184

Target Tab

When editing the attributes associated with a Radiative-convective unit it is possible to apply the same changes to units in other zones in the same HVAC Zone group. To do this select the components to be updated on the **Target** tab of the edit dialog.

Hot Water Convector

The hot water convector is controlled to meet any remaining zone load not met by any other item of equipment in the zone that has a higher heating priority. The control is accomplished by throttling the hot water flow.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the convector is located.

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

U-Factor times area value

This is the overall heat transfer coefficient for the convector (in W/K or Btu/h-F) at design conditions. The U-Factor Times Area Value (UA) is used in an NTU - effectiveness calculation. An estimate of the UA can be obtained from:

$$Q = UA \times (T_{\text{water, avg}} - T_{\text{air, avg}})$$

Where, q is the heat transferred from water to the air in watts, $T_{\text{water, avg}}$ is the average water temperature (°C) and $T_{\text{air, avg}}$ is the average air temperature (°C). Or the LMTD temperature difference can be used.

Maximum water flow rate

This is the maximum hot water volumetric flow rate in (m³/s or gal/min).

Convergence tolerance

This is the control tolerance for the convector output. The convector is controlled by matching the convector output to the zone demand. The model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|Q_{\text{unit, out}} - Q_{\text{zone load}}|}{Q_{\text{zone load}}}$$

The default is 0.001.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the component is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Electric Convector

The electric convector is controlled to meet any remaining zone load not met by any other item of equipment in the zone that has a higher heating priority. The control is accomplished by taking the remaining zone load and dividing by the efficiency of the baseboard.



The [HVAC Zone Equipment Tutorial](#) contains a description of how to use the Electric convector system

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the convector is located.

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Efficiency

This is the overall electrical efficiency of the convector. The zone load met by this unit is divided by the electrical efficiency to obtain the total electric energy used.

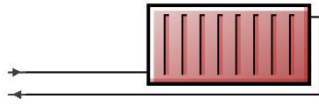
Operation

Availability schedule

This is the [schedule](#) that determines whether or not the component is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Hot Water Radiator

The zone hot water radiator model calculates the convective and radiant heat transfer from the device to the people and the surfaces within a zone so that surface heat balances can take into account the radiant heat transfer to the surfaces and thus enhance the accuracy of thermal comfort predictions within the space. The component is controlled to meet any remaining zone load not met by other zone equipment.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the radiator is located.

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized heating capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Maximum water flow rate

This is the maximum water volumetric flow rate in (m³/sec or gal/min). It can be auto-sized by EnergyPlus.

Rated average water temperature

This is the rated average water temperature for the radiator (in °C or °F). It typically ranges from 65°C to 115°C and the lowest allowable temperature is 32°C. The default value is 75°C. This information should be provided by manufacturer's literature.

Rated water mass flow rate

This is the rated standard water flow rate (in kg/s or lb/s) which should be provided by manufacturer's literature. It is used by the manufacturers when determining the rated capacity (see Rated capacity below). The default value is 0.063kg/s.

Note: If you are setting a **heating design capacity** (above) you must enter a corresponding **Rated water mass flow rate** here to ensure that you obtain the expected capacity in the simulation. Leaving this setting in default state could result in significant under or oversizing of the radiator. You can obtain maximum flow rates from the EnergyPlus Summary output as a guide.

Fraction radiant

This field specifies what fraction of the power input to the radiator is actually transferred to the space as radiant heat. The fraction should be between 0 and 1. This is the portion of the total power that is modelled as radiation. The portion that is radiant heat transfer from the radiator is distributed to people and specific surfaces using the remaining fields.

Fraction of radiant energy incident on people

This field specifies the fraction of radiant portion of heat transfer to the zone from the radiator that is incident directly on people within the space. This has an impact on the predicted thermal comfort of the zone occupants. Note that although this energy is "radiant" it is actually modeled in the zone heat balance as convective energy (like an internal gain). The basic assumption here is that most radiant energy falling on people will most likely be re-released to the zone air by convection. This is a simplification of reality, but it maintains the overall energy balance.

Note: The remaining fraction is divided across all of the surfaces of the zone in which the radiator is located.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the component is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

Advanced

Convergence tolerance

This is the control tolerance for the radiator output. The radiator is controlled by matching the radiator output to the zone demand. The model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|Q_{unit,out} - Q_{zone\ load}|}{Q_{zone\ load}}$$

The default is 0.001.

Electric Radiator

The zone electric radiator model calculates the convective and radiant heat transfer from the device to the people and the surfaces within a zone so that surface heat balances can take into account the radiant heat transfer to the surfaces and thus enhance the accuracy of thermal comfort predictions within the space. The component is controlled to meet any remaining zone load not met by other zone equipment and the total electric consumption of the radiator is calculated by dividing the load by the efficiency of the radiator.



General

Name

This is a read-only label that is automatically generated by the software and which incorporates the name of the zone in which the radiator is located.

Heating design capacity method

There are 3 ways to define the heating capacity of the unit as selected from the following list of options:

- **1-Design capacity** where the maximum (or nominal) heating capacity of the unit is entered, or the program calculates the maximum (or nominal) design heating capacity, if Autosize is specified.
- **2-Capacity per floor area** where EnergyPlus calculates the design heating capacity from a user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized capacity** which means the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field defines the convective electric nominal heating capacity (in W or Btu/h). It is only available when the Heating design capacity method is set to **1-Design capacity**.

Heating design capacity per floor area

Enter the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. This data is only available when the Heating design capacity method is **2-Capacity per floor area**.

The program calculates the heating capacity from floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

Enter the heating capacity as a fraction of the autosized heating capacity for convective electric baseboard unit. This data is only available when the Heating design capacity method is **3-Fraction of autosized heating capacity**. EnergyPlus calculates the heating capacity from the design autosized heating capacity and this fraction. The default value is 1.0.

Efficiency

This is the overall electrical efficiency of the radiator. The zone load met by this unit is divided by the electrical efficiency to obtain the total electric energy used.

Fraction radiant

This field specifies what fraction of the power input to the radiator is actually transferred to the space as radiant heat. The fraction should be between 0 and 1. This is the portion of the total power that is modelled as

radiation. The portion that is radiant heat transfer from the radiator is distributed to people and specific surfaces using the remaining fields.

Fraction of radiant energy incident on people

This field specifies the fraction of radiant portion of heat transfer to the zone from the radiator that is incident directly on people within the space. This has an impact on the predicted thermal comfort of the zone occupants. Note that although this energy is “radiant” it is actually modeled in the zone heat balance as convective energy (like an internal gain). The basic assumption here is that most radiant energy falling on people will most likely be re-released to the zone air by convection. This is a simplification of reality, but it maintains the overall energy balance.

Note: The remaining fraction is divided across all of the surfaces of the zone in which the radiator is located.

Operation

Availability schedule

This is the [schedule](#) that determines whether or not the component is available for each timestep of the simulation. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the timestep. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the timestep.

High Temperature Radiant Heater

The high temperature radiant system is a component of zone equipment that is intended to model any “high temperature” or “high intensity” radiant system where electric resistance or gas-fired combustion heating is used to supply heat directly to building occupants, as well as to the zone surfaces (wall, ceiling, or floor).

In cases where no other systems are serving the zone in which this system resides, it will use the heating equipment priority to determine which system will run first. If the radiant system is serving a zone with forced air equipment, the radiant system will follow the priority order established by the zone thermostat but will still base its response on the controls defined by the user for the radiant system.

The control is accomplished by varying the electrical power or gas consumed by the unit. It is not intended to simulate low temperature electric or hydronic radiant systems. Those devices are handled by [Radiant surfaces](#) and other types of “Radiative-Convective Heater data” on page 1178

General

Name

This field is a unique pre-defined user name for this high temperature radiant system.

Fuel type

Select the fuel from:

- **1-Natural gas** - the radiant surface is heated by burning gas with a combustion efficiency defined below.
- **2-Electricity** - the radiant surface is heated by electric elements

Temperature control type

This option, along with the throttling range and heating setpoint, defines how the radiant system is to be controlled. The temperature defined in the setpoint schedule (whether defined here on this dialog or by using Simple HVAC activity settings) can refer to one of 5 different temperatures: the zone mean air temperature,

the zone mean radiant temperature, the zone operative temperature, the outdoor dry-bulb temperature, or the outdoor wet-bulb temperature. Select from the following options:

- **1-Mean air temperature**
- **2-Mean radiant temperature**
- **3-Operative temperature** - average of Mean air temperature and Mean radiant temperature
- **4-Mean air temperature setpoint**
- **5-Mean radiant temperature setpoint**
- **6-Operative temperature setpoint**

For setpoint control strategies options 4-6, i.e. those ending in "setpoint" above, EnergyPlus calculates the correct heater output to meet the heating setpoint temperature through iteration and interpolation. This method will more closely match the operation of an actual thermostat, but it will require significantly longer execution time. For more information on the standard piecewise linear control algorithm used by the Mean Air Temperature, Mean Radiant Temperature, and Operative Temperature control types (the "non-Setpoint" controls),

See the throttling range and control temperature schedule fields below for more information.

Operation

Availability schedule

Select the schedule that denotes whether the high temperature radiant system can operate during a given time period. A schedule value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for that time period. A value greater than 0 (usually 1 is used) denotes that the unit is available to operate during that time period.

Heating

Heating design capacity method

Select the method to be used to determine the maximum power output of the unit. Allowed settings are:

- **1-Heating design capacity** - the maximum or nominal heating capacity is specified, or when autosize is entered the program calculates the maximum capacity.
- **2-Capacity per floor area** - the program calculates the design heating capacity from user specified heating capacity per floor area and floor area of the zone served by the unit.
- **3-Fraction of autosized heating capacity** - the program calculates the design heating capacity from user specified fraction and the auto-sized design heating capacity.

Heating design capacity

This autosizable field is available when the Heating design capacity method is set to **1-Heating design capacity**. It defines the maximum input power (electricity or gas) converted into heat in the unit (in W or Btu/h).

The controls for the radiant system vary the amount of power supplied to the unit between zero input and the maximum power specified in this field as the lower and upper bounds, respectively.

Heating design capacity per floor area

This field is available when the Heating design capacity method is set to **2-Capacity per floor area**. It defines the heating capacity per unit floor area (in W/m² or W/ft²) of the unit. The program calculates the heating capacity from the floor area of the zone served by the unit and the heating capacity per unit floor area value specified here.

Fraction of autosized heating design capacity

This field is available when the Heating design capacity method is set to **3-Fraction of autosized heating capacity**. It defines the heating capacity of the unit as a fraction of the zone autosized heating capacity. The program calculates the heating capacity from the design autosized heating capacity and this fraction. Design day sizing must be specified. The default value is 1.0.

Combustion efficiency

This is the combustion efficiency of the gas burner (when Fuel type is set to **1-Natural gas**). It should be greater than 0 and less than or equal to 1. The heater gas consumption is the heater output divided by the combustion efficiency.

Fraction of input converted to radiant energy

This field specifies the fraction of the power input to the heater is transferred to the zone as radiant heat. The fraction should be between 0 and 1. In conjunction with the next two parameters, it defines the breakdown of how the power input to the heater is distributed to the zone. The sum of these fractions must be less than or equal to 1. If the fractions are less than one, the remaining energy is added to the zone as convective heat transfer. The radiant heat transfer from the heat is distributed to people and surfaces using the distribution fractions listed below.

Fraction of input converted to latent energy

This field specifies the fraction of the power input to the heater that is converted to a latent heat gain within the space. This may be appropriate when a combustion process produces moisture that is transferred into the space. The latent heat addition from a high temperature radiant heater is handled as any other latent heat gain within the space, affecting the moisture balance on the zone air.

Fraction of input lost

This field specifies the fraction of power input to the high temperature radiant heater that is "lost". This energy is a loss term, and this fraction of the input power has no effect on the zone heat balances.

Heating setpoint temperature schedule

When [Detailed HVAC activity data](#) is set to **2-Detailed HVAC data** you can select a schedule to define the heating setpoint for the radiant system (in °C or °F). Used in conjunction with the heating throttling range, it defines whether or not the system is running and the power input to the radiant heater. Power input to the system is varied linearly around the setpoint temperature based on the throttling range and the maximum power input parameters (see above).

This schedule allows different heating setpoint temperatures to be defined for different periods through the year. In addition, it may be different from the setpoint defined for overall operation of components serving the zone in which the radiant system is located. The thermostatic control determines whether or not there is a heating or cooling load in the space and thus whether the systems should be operating. This field simply controls the heat input rate to the radiant system.

When [Detailed HVAC activity data](#) is set to **1-Simple HVAC data** the heating setpoint is defined by the [Heating setpoint temperature](#), [Heating setback temperature](#) and [Heating operation schedule](#) and this setting is not available, but otherwise it works as for when using **2-Detailed HVAC data**.

Heating throttling range

This field specifies the temperature range (in delta °C or °F) over which the radiant system throttles heat input (via the electric resistance wires or gas burner), from zero up to the maximum defined by the Maximum power input field described above. The throttling range parameter is used in conjunction with the zone setpoint temperature to define the response of the system to various zone conditions. The heating setpoint specifies the point where the power input to the system is at half of the maximum power input. For example, if the heating control temperature setpoint is currently 15°C and the heating throttling range is 2°C, the power supplied to the radiant system will be zero when the controlling temperature (Mean Air Temperature, Mean Radiant Temperature, or Operative Temperature; see control type field above) is at or above 16°C and the maximum power input when the controlling temperature is at or below 14°C. This represents a throttling range

of 2°C around the setpoint of 15°C. When the zone temperature is between 14°C and 16°C, the power input to the radiant system is varied linearly.

Fraction of radiant energy incident on people

This setting specifies the fraction of radiant heat output that is incident directly on the people within the space. This has an impact on the predicted thermal comfort of the zone occupants. Note that this energy is accounted for within the heat balance as convective energy, directly affecting the zone air heat balance. The basic assumption here is that most radiant energy falling on people will most likely be convected to the surroundings. This is a simplification of reality, but it maintains the overall energy balance.

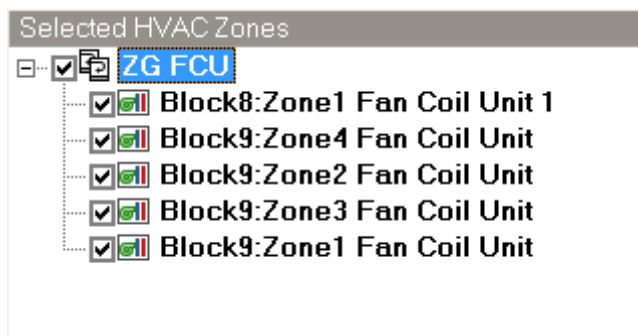
Zone Forced Air Units

These zone forced air units are available within Detailed HVAC:

- [Fan Coil Unit - Four Pipe](#)
- [Packaged Terminal Heat Pump](#)
- [Packaged Terminal Air Conditioner](#)
- "Zone Water To Air Heat Pump" on page 1203
- "VRF Indoor Unit" on page 1209

Target Tab

When editing the attributes associated with a Zone forced air unit it is possible to apply the same changes to units in other zones in the same HVAC Zone group. To do this select the components on the **Target** tab of the edit dialog as shown below.



Fan Coil Unit - Four Pipe



ZoneHVAC:FourPipeFanCoil

Used in:

HVAC Zones

The EnergyPlus Fan coil unit (FCU) is an in-room forced-convection hydronic unit. Typically these units are small (200 – 1200 cfm) and self-contained. They are mostly used in exterior zones, usually in hotels, apartments, or offices. They may be connected to ducted outside air, or have a direct outside air vent, but they do not have outside air economizers.

The heating or cooling output of the unit ventilator is controlled by varying the air flow rate, the water flow rate, or both. Air flow rate can be controlled by cycling the fan on/off or with a variable speed fan drive. The most common setup is a two or three speed fan with the speed selected by hand. The fan then cycles on/off to control heating / cooling output. The controls are often a wall mounted thermostat with hand selection of heating/cooling and fan speed (off/low/medium/high). These controls may also be mounted on the unit.

See the [Chilled water loops chillers and fan coil units](#)

Fan coil units are objects made up of other components. Each FCU consists all of these fixed components:

- Outdoor air mixer
- [Cooling coil - water](#)
- [Heating coil - water](#)
- [Constant volume supply air fan](#)

Default settings are assigned to these components at the point the FCU is created and these can be overridden by editing the individual component properties.

The fan coil unit is connected to a hot water loop (demand side) through its hot water coil and to a chilled water loop (demand side) through its cooling coil.

Outdoor air mixer

An outdoor air mixer is one of the components of the FCU. See above under [Outdoor Air Supply](#) for details of data entry.

Fan

A fan component is one of the components of a fan coil unit. See above under [Fan type](#) for details of fan types that are used.

Terminology

Like many HVAC terms, “fan coil unit” is used rather loosely. Sometimes it is used for terminal units that would be better described as powered induction units. Carrier and others use the term for the room side of refrigerant-based split systems.

Real world fan coil units can be 4-pipe or 2-pipe. For 4-pipe units there are 2 supply pipes and 2 return pipes. For 2-pipe units there is a single supply pipe and a single return pipe. Change-over systems enable you to switch the supply between hot and chilled water depending on the season. DesignBuilder provides only a 4-pipe unit, but the 4-pipe model can be used to model 2-pipe units simply by using the heating and cooling coil availability schedules to make sure that either hot or chilled water is exclusively available.

Units with outside air economizers are marketed (in the United States) as unit ventilators. Unit ventilators are typically bigger than fan coils and are widely used in classrooms or other applications where ventilation is a priority. If a zonal unit with an outside economizer is required, ZoneHVAC:UnitVentilator should be used (not yet implemented in DesignBuilder Detailed HVAC).

Name

A unique system-assigned name for an instance of a Fan Coil unit. Any reference to this Fan Coil unit by another object will use this name.

Availability schedule

[Schedule](#) that defines whether the fan coil unit can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the fan coil unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the fan coil unit is off.

Capacity control method

This input denotes how the unit's output is controlled in order to meet zone heating or cooling requirement. The choices are

- **1-Constant fan variable water flow** - the fan speed is held constant to produce a fixed air flow rate whenever the unit is scheduled on. The hot water or chilled flow rate is varied so that the unit output

matches the zone heating or cooling requirement. When this method is selected, the fan type can be defined, see below.

- **2-Cycling fan** - the fan speed is chosen so that the unit capacity is greater than or equal to the heating / cooling load and the fan is cycled to match unit output with the load. Cycling fans can have 3 speeds, Low, Medium and High speed and the ratio of supply air to the maximum supply rate is defaulted to 0.33, 0.66 and 1.00 for these speeds respectively. Custom values can be entered in [Low speed supply air flow ratio](#) and [Medium speed supply air flow ratio](#) described below.
- **3-Variable fan variable water flow** where both air and water flow rates are varied to match the load.
- **4-Variable fan constant water flow** where the water flow rate is at full flow and the fan speed varies to meet the load.

Low speed supply air flow ratio

This is the ratio of the low speed supply air flow rate to the maximum supply air flow rate. Its value should be less than [Medium speed supply air flow ratio](#) below. The default value is 0.33.

This data is only required if the capacity control method selected is **2-Cycling fan**.

Medium speed supply air flow ratio

This is the ratio of the medium speed supply air flow rate to the maximum supply air flow rate. Its value should be greater than the [Low speed supply air flow ratio](#) above, but less than 1. The default value is 0.66.

This data is only required if the capacity control method selected is **2-Cycling fan**.

Fan type

When the Capacity control method is set to **1-Constant fan variable flow** you can select the type of fan. Choose from:

- **1-On/off fan**
- **2-Constant volume fan**

For the other types of Capacity control method the fan type is fixed internally based on the method. For **2-Cycling fan**, an On/off fan is used and for **3-Variable fan variable flow** or **4-Variable fan constant flow** a Variable volume fan will be used.

Maximum supply air flow rate

The maximum volumetric airflow rate (m³/sec or ft³/min) through the fan coil unit. This is also the design, rated airflow rate of the unit.

Outdoor Air Supply

Outdoor air supply

Check this option if the fan coil unit includes an outdoor air supply.

Maximum outdoor air flow rate

If the fan coil unit uses outdoor air, this data specifies the outdoor air volumetric flow rate (m³/sec or ft³/min). This flow rate should be less than or equal to the maximum airflow rate.

Cooling Coil

A cooling coil is another component of the fan coil unit. Only the standard water cooling coil type is available.

Maximum cold water flow rate

The maximum cold water volumetric flow rate (m³/sec or gal/min) through the fan coil unit's cooling coil.

Minimum cold water flow rate

The minimum cold water volumetric flow rate (m3/sec or gal/min) through the fan coil unit's cooling coil.

Heating Coil

Heating coil component also make up part of the fan coil unit. Only the standard water heating coil type can be used.

Maximum hot water flow rate

The maximum hot water volumetric flow rate (m3/sec or gal/min) through the fan coil unit's heating coil.

Minimum hot water flow rate

The minimum hot water volumetric flow rate (m3/sec or gal/min) through the fan coil unit's heating coil.

Advanced

Cooling convergence tolerance

The convergence tolerance for the control of the unit cooling output can be defined. The unit is controlled by matching the unit output to the zone demand. For units with water coils, the model must be numerically inverted to obtain a specified output. The cooling convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$(Q_{\text{fan coil out}} - Q_{\text{zone load}}) / Q_{\text{zone load}} \leq \text{Cooling Convergence Tolerance}$$

Heating Convergence Tolerance

The convergence tolerance for the control of the unit heating output can be defined. The unit is controlled by matching the unit output to the zone demand. For units with water coils, the model must be numerically inverted to obtain a specified output. The heating convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$(Q_{\text{fan coil out}} - Q_{\text{zone load}}) / Q_{\text{zone load}} \leq \text{Heating Convergence Tolerance}$$

Fan Coil Unit - Four Pipe Outputs

```
HVAC,Average,Fan Coil Heating Rate[W]
HVAC,Sum,Fan Coil Heating Energy[J]
HVAC,Average,Fan Coil Total Cooling Rate[W]
HVAC,Sum,Fan Coil Total Cooling Energy[J]
HVAC,Average,Fan Coil Sensible Cooling Rate[W]
HVAC,Sum,Fan Coil Sensible Cooling Energy[J]
HVAC,Average,Fan Coil Electric Power[W]
HVAC,Sum,Fan Coil Electric Consumption[J]
HVAC,Average,Fraction Of Timestep Fancoil is ON []
HVAC,Average,Fan Speed []
```

Fan Coil Heating Rate[W]

This field reports the dry air heating addition rate of the fan coil unit to the zone it is serving in Watts. This is determined by outlet and zone air conditions and the mass flow rate through the unit.

Fan Coil Heating Energy[J]

This field is the dry air heat addition of the fan coil unit to the zone it is serving in Joules over the timestep being reported. This is determined by outlet and zone air conditions, the mass flow rate through the unit, and the timestep.

Fan Coil Total Cooling Rate[W]

This field is the total (sensible and latent) heat extraction rate of the fan coil unit from the zone it is serving in Watts. This is determined by the outlet and zone conditions and the mass flow rate through the unit.

Fan Coil Total Cooling Energy[J]

This field is the total (sensible and latent) heat extraction of the fan coil unit from the zone it is serving in Joules over the timestep being reported. This is determined by outlet and zone air conditions, the mass flow rate through the unit, and the timestep.

Fan Coil Sensible Cooling Rate[W]

This field reports the dry air sensible heat extraction rate of the fan coil unit from the zone it is serving in Watts. This is determined by the outlet and zone conditions and the mass flow rate through the unit.

Fan Coil Sensible Cooling Energy[J]

This field is the dry air sensible heat extraction of the fan coil unit from the zone it is serving in Joules over the timestep being reported. This is determined by the outlet and zone conditions, the mass flow rate through the unit, and the timestep.

Fan Coil Electric Power[W]

This field reports the electricity consumption rate of the fan coil unit in Watts.

Fan Coil Electric Consumption[J]

This field is the electricity consumption of the fan coil unit in Joules over the timestep being reported.

Fraction Of Timestep Fancoil is ON

This field is the fraction of the system timestep the fan coil unit is running for the CyclingFan capacity control method. This variable is defined only for the CyclingFan capacity control method.

Fan Speed

This field indicates the speed chosen for the fan in the CyclingFan capacity control method. A value of '0' means that the unit is off, '1' the fan is running at its low speed, '2' medium speed, and '3' high speed (maximum). This variable is defined only for the CyclingFan capacity control method.

Part Load Ratio

When the capacity control method is VariableFanVariableFlow or VariableFanConstantFlow, this report variable reports the unit part load ratio (ratio of unit heating / cooling output to the maximum heating / cooling output). This variable is defined only for VariableFanVariableFlow or VariableFanConstantFlow capacity control methods.

Packaged Terminal Heat Pump



ZoneHVAC:PackagedTerminalHeatPump

Used in:

HVAC Zones

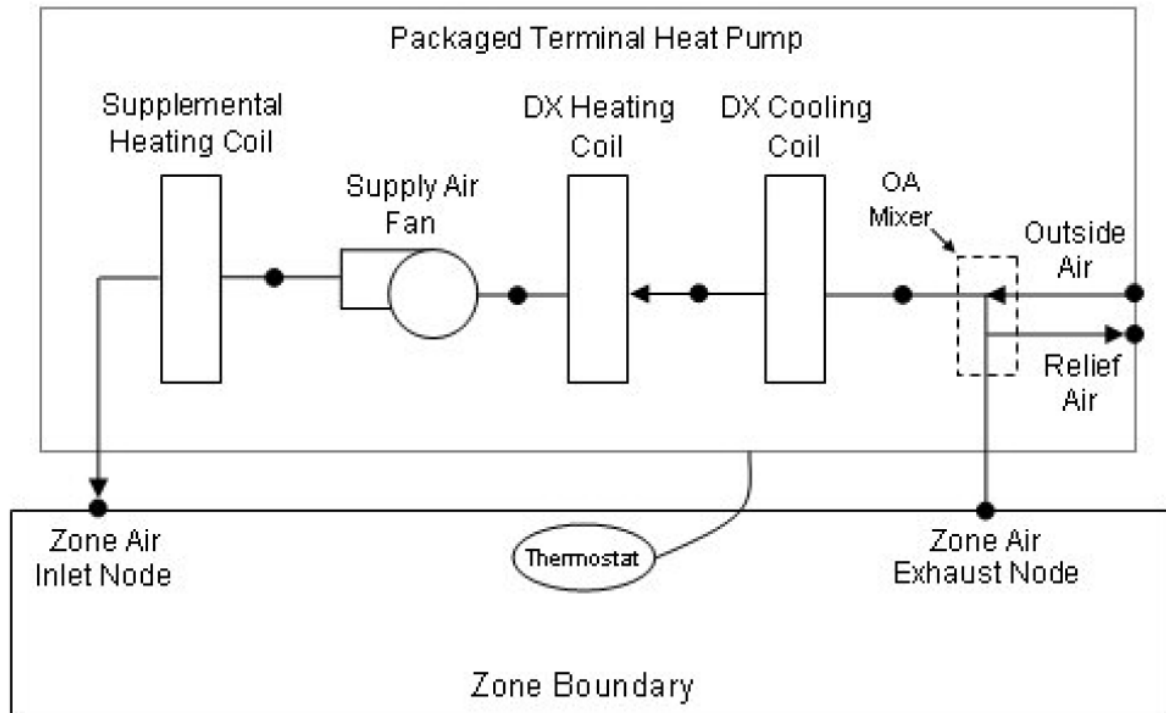
The packaged terminal heat pump (PTHP) is a compound object made up of other components. Each PTHP consists of these fixed components:

- Outdoor air mixer,
- [DX cooling coil - single speed](#),
- [DX heating coil](#)
- [Supplemental heating coil \(electric\)](#), and,

- [Constant volume supply air fan.](#)

Default settings are assigned to these components at the point the PTHP is created and these can be overridden by editing the individual component properties.

The layout is illustrated below.



Schematic of a packaged terminal heat pump (draw through fan placement)

The packaged terminal heat pump coordinates the operation of these components and is modelled as a type of zone equipment.

Note: The details of the DX cooling system including compressor, condenser, CoP, part-load performance etc are entered on the [DX cooling coil](#) dialog.

The data that must be entered for the PTHP is as follows.

Name

A unique system-assigned name for an instance of a packaged terminal heat pump. Any reference to this heat pump by another object will use this name.

Fan

A [constant volume fan component](#) is included as a component of the PTHP. Note that the fan's maximum flow rate should be greater than or equal to the maximum supply air flow rate for the PTHP.

Fan placement

There are 2 fan placement options:

- **1-Blow through** where the unit consists of an outdoor air mixer followed by a fan followed by the DX coils and supplemental heating coil. The fan "blows through" the cooling and heating coils.
- **2-Draw through** where the unit consists of an outdoor air mixer followed by the DX coils followed by a fan, with the supplemental heater located at the outlet of the fan. The fan "draws air through" the DX coils.

Fan operating mode

Two fan operation modes are available for selection in PTHP units:

- **1-Cycling** where the [Supply air fan operating mode schedule](#) (below) determines the flow regime.
- **2-Continuous** where the fan delivers a constant flow rate through the unit regardless of whether the unit is heating, cooling and/or providing fresh air. Continuous fans don't use the [Supply air fan operating mode schedule](#) (below).

Supply air fan operating mode schedule

This data specifies the supply air fan operating mode schedule. The supply air fan operating mode may vary during the simulation based on time-of-day or with a change of season. Schedule values of 0 denote that the supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (a.k.a. AUTO fan). Schedule values other than 0 denote that the supply fan runs continuously while the heating or cooling coil cycles to meet the load.

The default schedule is **Fan operation mode - Cycling** with a constant value of 0, i.e. that the supply fan runs with an AUTO configuration. To obtain the continuous fan configuration, select the **Fan operation mode - Continuous** schedule which has a constant value of 1.

To set up a time-varying fan operation mode, create and select a schedule with values of 0 and 1 to define the way the fan operation mode varies in time.

Supply Air Flow Rates

Supply air flow rate during cooling operation

This numeric field defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when the DX cooling coil is operating. Values must be greater than 0 or this field is autosizable.

Supply air flow rate during heating operation

This numeric field defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when the DX heating coil and/or supplemental heater are operating. Values must be greater than 0 or this field is autosizable.

Supply air flow rate when no cooling or heating is needed

This numeric field defines the supply air flow rate leaving the heat pump (in m³/s or ft³/min) when neither cooling or heating is required (i.e., DX coils and supplemental heater are off but the supply air fan operates). This field is only used when the heat pump's supply air fan operating mode schedule specifies continuous fan operation. Values must be greater than or equal to zero, or this field is autosizable.

If the heat pump's supply air fan operating mode schedule specifies continuous fan operation and this value is set to zero, then the model assumes that the supply air flow rate when no cooling/heating is needed is equal to the supply air flow rate when the cooling or heating coil was last operating (for cooling operation or heating operation).

Outdoor Air Supply

Outdoor air supply

If the PTAC provides outdoor air then check this option.

Outdoor air flow rate during cooling operation

This numeric field defines the outdoor air flow rate through the heat pump (in m³/s or ft³/min) when the DX cooling coil is operating. Values must be greater than or equal to 0, or this field is autosizable.

Note that the outdoor air flow rate during cooling operation is fixed; it cannot change during the simulation. In addition, the outdoor air flow rate during cooling operation cannot be greater than the heat pump's supply air volumetric flow rate during cooling operation.

Outdoor air flow rate during heating operation

This numeric field defines the outdoor air flow rate through the heat pump (in m³/s or ft³/min) when the DX heating coil and/or supplemental heater are operating. Values must be greater than or equal to 0, or this field is autosizable.

Note that the outdoor air flow rate during heating operation is fixed; it cannot change during the simulation. In addition, the outdoor air flow rate during heating operation cannot be greater than the heat pump's supply air volumetric flow rate during heating operation.

Outdoor air flow rate when no cooling or heating is needed

This numeric field defines the outdoor air flow rate through the heat pump (in m³/s or ft³/min) when neither cooling or heating is required (i.e., DX coils and supplemental heater are off but the supply air fan operates). Values must be greater than or equal to 0, or this field is autosizable.

Note that the outdoor air flow rate when no cooling/heating is needed is fixed; it cannot change during the simulation. In addition, the outdoor air flow rate when no cooling/heating is needed cannot be greater than the heat pump's supply air volumetric flow rate when no cooling/heating is needed. This field is only used when the heat pump's supply air fan operating mode schedule specifies continuous fan operation. If the heat pump's supply air fan operating mode schedule specifies continuous fan operation and the field 'Supply air volumetric flow rate when no cooling or heating is needed' is set to zero or is left blank, then the model assumes that the outdoor air flow rate when no cooling/heating is needed is equal to the outdoor air flow rate when the cooling or heating coil was last operating (for cooling operation [i.e., Outdoor air flow rate during cooling operation] or heating operation [i.e., Outdoor air flow rate during heating operation]) and this field is not used.

Compressor

Minimum outdoor dry-bulb temperature for compressor operation

This numeric field defines the minimum outdoor dry-bulb temperature (in °C or °F) for PTHP compressor operation. The compressor will not operate (for DX heating or DX cooling) when outdoor dry-bulb temperatures fall below this value. The minimum value must be greater than or equal to -20°C. The default value is -8°C. This temperature should match the minimum compressor operating temperature specified for the heat pump's DX heating coil (if they don't match, the highest of the two temperatures will be the cut-off temperature for compressor operation).

Supplemental Heater

The supplemental heating coil used by this PTHP is fixed as [Electric](#). The supplemental heating coil default settings can be overridden by editing the coil properties.

Maximum supply air temperature from supplemental heater

This numeric field defines the maximum supply air temperature (in °C or °F) exiting the heat pump supplemental heater coil. The supplemental heater will be controlled so that its supply air temperature does not exceed this value. This field is autosizable.

Maximum outdoor dry-bulb temperature for supplemental heater operation

This numeric field defines the maximum outdoor dry-bulb temperature (in °C or °F) for PTHP supplemental heater operation. The supplemental heater will not operate when the outdoor dry-bulb temperature is above this value. The maximum value must be less than or equal to 21°C. The default value is 21°C.

Operation

Availability schedule

Select the [Schedule](#) that defines whether the heat pump operates during a given time period. A schedule value equal to 0 denotes that the heat pump must be off for that time period. A value greater than 0 denotes that the heat pump is available to operate during that time period. This schedule may be used to completely disable the heat pump (all of its coils and the supply air fan) as required.

Advanced

Cooling convergence tolerance

This numeric field defines the convergence tolerance for the unit's cooling output. This field allows the user some control over how closely the heat pump will control the air-side conditions. The relative size of this parameter relates directly to the closeness of the control. A very small value in this field will result in tight control and will probably result in larger numbers of iterations. A large value in this field will result in looser controls and could result in unsatisfactory fluctuations in zone air temperature. Initial experience with this parameter lends to the recommendation of using 0.001 as the starting point.

The heat pump is controlled by matching its sensible (temperature) cooling output to the zone sensible load (demand). Because the performance of the DX cooling coil is frequently nonlinear, the heat pump model must call the DX cooling coil model several times (iterate) to determine the proper run time fraction to meet the zone load. The cooling convergence tolerance is the error tolerance used to terminate the iteration procedure when the following equation is satisfied:

$$(Q_{\text{zone load}} - Q_{\text{heat pump out}}) / Q_{\text{zone load}} \leq \text{Cooling Convergence Tolerance}$$

The maximum number of iterations is limited, with a warning message generated if the above equation is not satisfied within the maximum number of iterations.

Heating convergence tolerance

This numeric field defines the convergence tolerance for the unit's heating output. This field allows the user some control over how closely the heat pump will control the air-side conditions. The relative size of this parameter relates directly to the closeness of the control. A very small value in this field will result in tight control and will probably result in larger numbers of iterations. A large value in this field will result in looser controls and could result in unsatisfactory fluctuations in zone air temperature. Initial experience with this parameter lends to the recommendation of using 0.001 as the starting point.

The heat pump is controlled by matching its sensible (temperature) heating output to the zone sensible load (demand). Because the performance of the DX heating coil is frequently non-linear, the heat pump model must call the DX heating coil model several times (iterate) to determine the proper run time fraction to meet the zone load. The heating convergence tolerance is the error tolerance used to terminate the iteration procedure when the following equation is satisfied:

$$(Q_{\text{zone load}} - Q_{\text{heat pump out}}) / Q_{\text{zone load}} \leq \text{Heating Convergence Tolerance}$$

The maximum number of iterations is limited, with a warning message generated if the above equation is not satisfied within the maximum number of iterations.

Packaged Terminal Heat Pump Outputs

HVAC,Average,Packaged Terminal Heat Pump Total Zone Heating Rate[W]
 HVAC,Sum,Packaged Terminal Heat Pump Total Zone Heating Energy[J]
 HVAC,Average,Packaged Terminal Heat Pump Total Zone Cooling Rate[W]
 HVAC,Sum,Packaged Terminal Heat Pump Total Zone Cooling Energy[J]
 HVAC,Average,Packaged Terminal Heat Pump Sensible Zone Heating Rate[W]
 HVAC,Sum,Packaged Terminal Heat Pump Sensible Zone Heating Energy[J]
 HVAC,Average,Packaged Terminal Heat Pump Sensible Zone Cooling Rate[W]
 HVAC,Sum,Packaged Terminal Heat Pump Sensible Zone Cooling Energy[J]
 HVAC,Average,Packaged Terminal Heat Pump Latent Zone Heating Rate[W]
 HVAC,Sum,Packaged Terminal Heat Pump Latent Zone Heating Energy[J]

HVAC,Average,Packaged Terminal Heat Pump Latent Zone Cooling Rate[W]
HVAC,Sum,Packaged Terminal Heat Pump Latent Zone Cooling Energy[J]
HVAC,Average,Packaged Terminal Heat Pump Electric Power[W]
HVAC,Sum,Packaged Terminal Heat Pump Electric Consumption[J]
HVAC,Average,Packaged Terminal Heat Pump Fan Part-Load Ratio
HVAC,Average,Packaged Terminal Heat Pump Compressor Part-Load Ratio

Packaged Terminal Heat Pump Total Zone Heating Rate[W]

This output field is the total (enthalpy) heat addition rate of the packaged terminal heat pump to the zone it is serving in Watts. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams, and the air mass flow rate through the heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy addition only) are averaged for the timestep being reported.

Packaged Terminal Heat Pump Total Zone Heating Energy[J]

This output field is the total (enthalpy) heat addition of the packaged terminal heat pump to the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams, the air mass flow rate through the heat pump, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy addition only) are summed for the timestep being reported.

Packaged Terminal Heat Pump Total Zone Cooling Rate[W]

This output field is the total (enthalpy) heat extraction rate of the packaged terminal heat pump from the zone it is serving in Watts. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams, and the air mass flow rate through the heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy extraction only) are averaged for the timestep being reported.

Packaged Terminal Heat Pump Total Zone Cooling Energy[J]

This output field is the total (enthalpy) heat extraction of the packaged terminal heat pump from the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams, the air mass flow rate through the heat pump, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy extraction only) are summed for the timestep being reported.

Packaged Terminal Heat Pump Sensible Zone Heating Rate[W]

This output field is the sensible heat addition rate of the packaged terminal heat pump to the zone it is serving in Watts. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams at a constant humidity ratio, and the air mass flow rate through the heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (heating only) are averaged for the timestep being reported.

Packaged Terminal Heat Pump Sensible Zone Heating Energy[J]

This output field is the sensible heat addition of the packaged terminal heat pump to the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams at a constant humidity ratio, the air mass flow rate through the heat pump, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (heating only) are summed for the timestep being reported.

Packaged Terminal Heat Pump Sensible Zone Cooling Rate[W]

This output field reports the moist air sensible heat extraction rate of the packaged terminal heat pump from the zone it is serving in Watts. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams at a constant humidity ratio, and the air mass flow rate through the heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (cooling only) are averaged for the timestep being reported.

Packaged Terminal Heat Pump Sensible Zone Cooling Energy[J]

This output field reports the moist air sensible heat extraction of the packaged terminal heat pump from the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the heat pump outlet air and inlet air streams at a constant humidity ratio, the air mass flow rate through the heat pump, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (cooling only) are summed for the timestep being reported.

Packaged Terminal Heat Pump Latent Zone Heating Rate[W]

This output field is the latent heat addition (humidification) rate of the packaged terminal heat pump to the zone it is serving in Watts. This value is calculated as the difference between the total energy rate and the sensible energy rate provided by the packaged terminal heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat addition only) are averaged for the timestep being reported.

Packaged Terminal Heat Pump Latent Zone Heating Energy[J]

This output field is the latent heat addition (humidification) of the packaged terminal heat pump to the zone it is serving in Joules over the timestep being reported. This value is calculated as the difference between the total energy delivered to the zone and the sensible energy delivered to the zone by the packaged terminal heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat addition only) are summed for the timestep being reported.

Packaged Terminal Heat Pump Latent Zone Cooling Rate[W]

This output field is the latent heat extraction (dehumidification) rate of the packaged terminal heat pump from the zone it is serving in Watts. This value is calculated as the difference between the total energy rate and the sensible energy rate provided by the packaged terminal heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat extraction only) are averaged for the timestep being reported.

Packaged Terminal Heat Pump Latent Zone Cooling Energy[J]

This output field is the latent heat extraction (dehumidification) of the packaged terminal heat pump from the zone it is serving in Joules over the timestep being reported. This value is calculated as the difference between the total energy delivered to the zone and the sensible energy delivered to the zone by the packaged terminal heat pump. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat extraction only) are summed for the timestep being reported.

Packaged Terminal Heat Pump Electric Power[W]

This output field is the electricity consumption rate of the packaged terminal heat pump in Watts. The consumption includes electricity used by the compressor (including crankcase heater), fans (indoor supply air fan and the condenser fan), and the supplemental heating coil (if electric). This value is calculated for each HVAC system timestep being simulated, and the results are averaged for the timestep being reported.

Packaged Terminal Heat Pump Electric Consumption[J]

This output field is the electricity consumption of the packaged terminal heat pump in Joules for the time period being reported. The consumption includes electricity used by the compressor (including crankcase heater), fans (indoor supply air fan and the condenser fan), and the supplemental heating coil (if electric). This value is calculated for each HVAC system timestep being simulated, and the results are summed for the timestep being reported.

Packaged Terminal Heat Pump Fan Part-Load Ratio

This output field is the part-load ratio of the fan. The fan part-load ratio is defined as the average supply air mass flow rate divided by the maximum supply air mass flow rate. The maximum supply air mass flow rate depends on whether heating, cooling, or no heating or cooling is required during the timestep. This value is calculated for each HVAC system timestep being simulated, and the results are averaged for the timestep being reported.

Packaged Terminal Heat Pump Compressor Part-Load Ratio

This output field is the part-load ratio of the compressor used by the DX coils (cooling and heating). Compressor part-load ratio is defined as the total coil load divided by the coil steady-state capacity. This value is calculated for each HVAC system timestep being simulated, and the results are averaged for the timestep being reported.

Packaged Terminal Air Conditioner



ZoneHVAC:PackagedTerminalAirConditioner HVAC Zones

Used in:

The packaged terminal air conditioner (PTAC) is a compound object made up of other components. Each PTAC consists all of these fixed components:

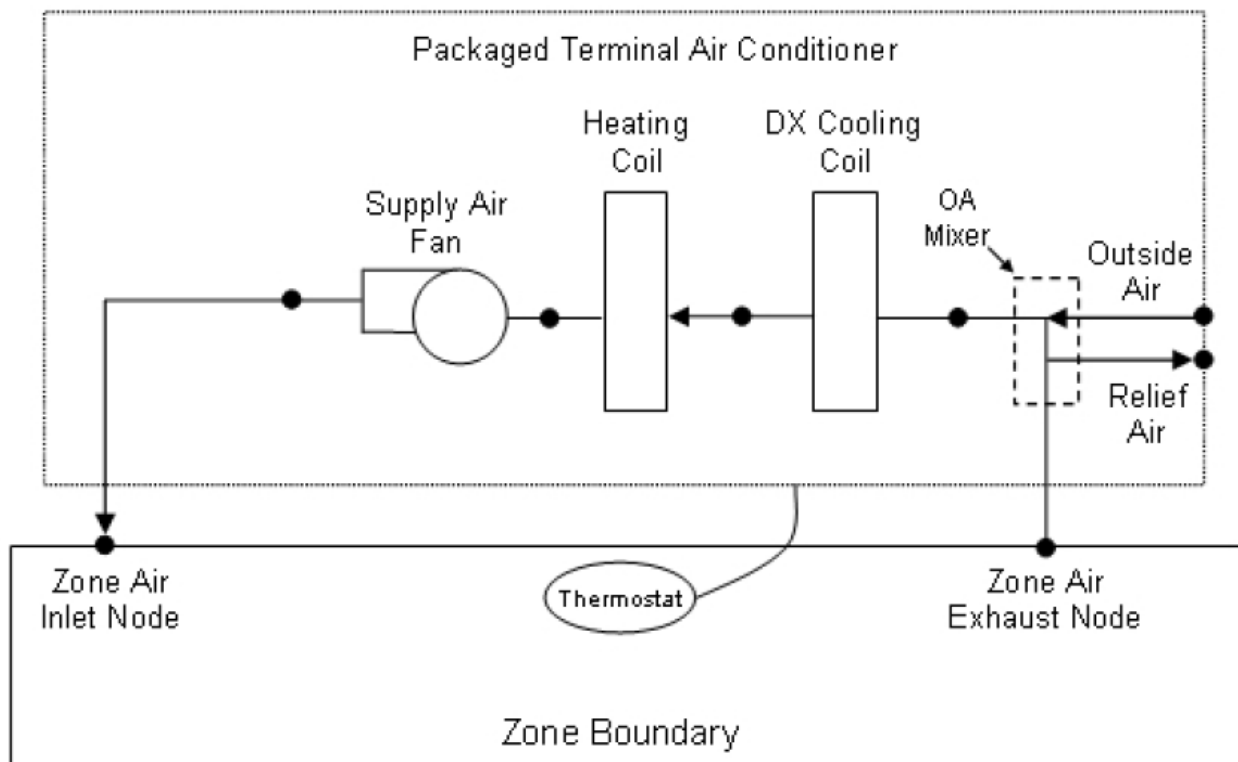
- Outdoor air mixer
- [DX cooling coil - single speed](#)
- [Electric, water or gas heating coil](#)
- [Constant volume supply air fan](#)

To change the type of heating coil you should edit the heating coil component and select the required coil type from the Heating coil dialog.

Note: The details of the DX cooling system including compressor, condenser, CoP, part-load performance etc are entered on the [DX cooling coil](#) dialog.

Default settings are assigned to these components at the point the PTAC is created and these can be overridden by editing the individual component properties.

While the figure below shows the PTAC with **draw through** fan placement, **blow through** fan placement can also be modelled by positioning the supply air fan between the outdoor air mixer and the DX cooling coil. The packaged terminal air conditioner coordinates the operation of these components and is modelled as a type of zone equipment.



Schematic of a Packaged terminal air conditioner with draw through fan placement

Name

A unique system-assigned name for an instance of a packaged terminal air conditioner. Any reference to this air conditioner by another object will use this name.

Fan

Fan placement

There are 2 fan placement options:

- **1-Blow through** where the unit consists of an outdoor air mixer followed by a fan followed by the DX cooling coil and heating coil. The fan “blows through” the DX cooling coil and the heating coil.
- **2-Draw through** where the unit consists of an outdoor air mixer followed by the DX cooling coil and heating coil with the fan located at the outlet of the PTAC. The fan “draws air through” the DX cooling coil and the heating coil.

Fan operating mode

Two fan operation modes are available for selection in PTAC units:

- **1-Cycling** where the [Supply air fan operating mode schedule](#) (below) determines the flow regime.
- **2-Continuous** where the fan delivers a constant flow rate through the unit regardless of whether the unit is heating, cooling and/or providing fresh air. Continuous fans don't use the [Supply air fan operating mode schedule](#) (below).

Supply air fan operating mode schedule

This data specifies the supply air fan operating mode schedule. The supply air fan operating mode may vary during the simulation based on time-of-day or with a change of season. Schedule values of 0 denote that the supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (aka AUTO fan). Schedule values other than 0 indicate that the supply fan runs continuously while the heating or cooling coil cycles to meet the load.

The default schedule is **Fan operation mode - Cycling** with a constant value of 0, i.e. that the supply fan runs with an AUTO configuration. To obtain the continuous fan configuration, select the **Fan operation mode - Continuous** schedule which has a constant value of 1.

To set up a time-varying fan operation mode, create and select a schedule with values of 0 and 1 to define the way the fan operation mode varies in time.

Supply Air Flow Rates

Supply air flow rate during cooling operation

This numeric field defines the supply air flow rate leaving the air conditioner (in m³/s or ft³/min) when the DX cooling coil is operating. Values must be greater than 0 or this field is autosizable.

Supply air flow rate during heating operation

This numeric field defines the supply air flow rate leaving the air conditioner (in m³/s or ft³/min) when the heating coil is operating. Values must be greater than 0 or this field is autosizable.

Supply air flow rate when no cooling or heating is needed

This numeric field defines the supply air flow rate leaving the air conditioner (in m³/s or ft³/min) when neither cooling nor heating is required (i.e., DX coil and heater are off but the supply air fan operates). This field is only used when the air conditioner's supply air fan operating mode schedule specifies continuous fan operation. Values must be greater than or equal to zero, or this field is autosizable. If the air conditioner's supply air fan operating mode schedule specifies continuous fan operation and this value is set to zero or this field is left blank, then the model assumes that the supply air flow rate when no cooling/heating is needed is equal to the supply air flow rate when the cooling or heating coil was last operating (for cooling operation or heating operation).

Outdoor Air

Outdoor air supply

If the PTAC provides outdoor air then check this option.

Outdoor air flow rate during cooling operation

This numeric field defines the outdoor air flow rate through the air conditioner (in m³/s or ft³/min) when the DX cooling coil is operating. Values must be greater than or equal to 0, or this field is autosizable. Note that the outdoor air flow rate during cooling operation is fixed; it cannot change during the simulation. In addition, the outdoor air flow rate during cooling operation cannot be greater than the air conditioner's supply air volumetric flow rate during cooling operation.

Outdoor air flow rate during heating operation

This numeric field defines the outdoor air flow rate through the air conditioner (in m³/s or ft³/min) when the heating coil is operating. Values must be greater than or equal to 0, or this field is autosizable. Note that the outdoor air flow rate during heating operation is fixed; it cannot change during the simulation. In addition, the outdoor air flow rate during heating operation cannot be greater than the air conditioner's supply air volumetric flow rate during heating operation.

Outdoor air flow rate when no cooling or heating is needed

This numeric field defines the outdoor air flow rate through the air conditioner (in m³/s or ft³/min) when neither cooling nor heating is required (i.e., cooling and heating coils are off but the supply air fan operates). Values must be greater than or equal to 0, or this field is autosizable.

Note that the outdoor air flow rate when no cooling/heating is needed is fixed; it cannot change during the simulation. In addition, the outdoor air flow rate when no cooling/heating is needed cannot be greater than the air conditioner's supply air volumetric flow rate when no cooling/heating is needed. This field is only used

when the air conditioner's supply air fan operating mode schedule specifies continuous fan operation. If the air conditioner's supply air fan operating mode schedule specifies continuous fan operation and the field 'Supply air volumetric flow rate when no cooling or heating is needed' is set to zero or is left blank, then the model assumes that the outdoor air flow rate when no cooling/heating is needed is equal to the outdoor air flow rate when the cooling or heating coil was last operating (for cooling operation [i.e., Outdoor air Flow rate during cooling operation] or heating operation [i.e., Outdoor air flow rate during heating operation]) and this field is not used.

Operation

Availability schedule

Select the [Schedule](#) that defines whether the air conditioner can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the air conditioner can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the air conditioner is off. This schedule may be used to completely disable the air conditioner (all of its coils and the supply air fan) as required.

Packaged Terminal Air Conditioner Outputs

HVAC,Average,Packaged Terminal Air Conditioner Total Zone Heating Rate[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Total Zone Heating Energy[J]
 HVAC,Average,Packaged Terminal Air Conditioner Total Zone Cooling Rate[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Total Zone Cooling Energy[J]
 HVAC,Average,Packaged Terminal Air Conditioner Sensible Zone Heating Rate[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Sensible Zone Heating Energy[J]
 HVAC,Average,Packaged Terminal Air Conditioner Sensible Zone Cooling Rate[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Sensible Zone Cooling Energy[J]
 HVAC,Average,Packaged Terminal Air Conditioner Latent Zone Heating Rate[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Latent Zone Heating Energy[J]
 HVAC,Average,Packaged Terminal Air Conditioner Latent Zone Cooling Rate[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Latent Zone Cooling Energy[J]
 HVAC,Average,Packaged Terminal Air Conditioner Electric Power[W]
 HVAC,Sum,Packaged Terminal Air Conditioner Electric Consumption[J]
 HVAC,Average,Packaged Terminal Air Conditioner Fan Part-Load Ratio
 HVAC,Average,Packaged Terminal Air Conditioner Part-Load Ratio

Packaged Terminal Air Conditioner Total Zone Heating Rate[W]

This output field is the total (enthalpy) heat addition rate of the packaged terminal air conditioner to the zone it is serving in Watts. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams, and the air mass flow rate through the air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy addition only) are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Total Zone Heating Energy[J]

This output field is the total (enthalpy) heat addition of the packaged terminal air conditioner to the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams, the air mass flow rate through the air conditioner, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy addition only) are summed for the timestep being reported.

Packaged Terminal Air Conditioner Total Zone Cooling Rate[W]

This output field is the total (enthalpy) heat extraction rate of the packaged terminal air conditioner from the zone it is serving in Watts. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams, and the air mass flow rate through the air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy extraction only) are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Total Zone Cooling Energy[J]

This output field is the total (enthalpy) heat extraction of the packaged terminal air conditioner from the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams, the air mass flow rate through the air conditioner, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (enthalpy extraction only) are summed for the timestep being reported.

Packaged Terminal Air Conditioner Sensible Zone Heating Rate[W]

This output field is the sensible heat addition rate of the packaged terminal air conditioner to the zone it is serving in Watts. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams at a constant humidity ratio, and the air mass flow rate through the air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (heating only) are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Sensible Zone Heating Energy[J]

This output field is the sensible heat addition of the packaged terminal air conditioner to the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams at a constant humidity ratio, the air mass flow rate through the air conditioner, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (heating only) are summed for the timestep being reported.

Packaged Terminal Air Conditioner Sensible Zone Cooling Rate[W]

This output field reports the moist air sensible heat extraction rate of the packaged terminal air conditioner from the zone it is serving in Watts. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams at a constant humidity ratio, and the air mass flow rate through the air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (cooling only) are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Sensible Zone Cooling Energy[J]

This output field reports the moist air sensible heat extraction of the packaged terminal air conditioner from the zone it is serving in Joules over the timestep being reported. This value is calculated using the enthalpy difference of the air conditioner outlet air and inlet air streams at a constant humidity ratio, the air mass flow rate through the air conditioner, and the HVAC simulation timestep. This value is calculated for each HVAC system timestep being simulated, and the results (cooling only) are summed for the timestep being reported.

Packaged Terminal Air Conditioner Latent Zone Heating Rate[W]

This output field is the latent heat addition (humidification) rate of the packaged terminal air conditioner to the zone it is serving in Watts. This value is calculated as the difference between the total energy rate and the sensible energy rate provided by the packaged terminal air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat addition only) are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Latent Zone Heating Energy[J]

This output field is the latent heat addition (humidification) of the packaged terminal air conditioner to the zone it is serving in Joules over the timestep being reported. This value is calculated as the difference between the total energy delivered to the zone and the sensible energy delivered to the zone by the packaged terminal air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat addition only) are summed for the timestep being reported.

Packaged Terminal Air Conditioner Latent Zone Cooling Rate[W]

This output field is the latent heat extraction (dehumidification) rate of the packaged terminal air conditioner from the zone it is serving in Watts. This value is calculated as the difference between the total energy rate and the sensible energy rate provided by the packaged terminal air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat extraction only) are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Latent Zone Cooling Energy[J]

This output field is the latent heat extraction (dehumidification) of the packaged terminal air conditioner from the zone it is serving in Joules over the timestep being reported. This value is calculated as the difference between the total energy delivered to the zone and the sensible energy delivered to the zone by the packaged terminal air conditioner. This value is calculated for each HVAC system timestep being simulated, and the results (latent heat extraction only) are summed for the timestep being reported.

Packaged Terminal Air Conditioner Electric Power[W]

This output field is the electricity consumption rate of the packaged terminal air conditioner in Watts. The consumption includes electricity used by the compressor (including crankcase heater), fans (indoor supply air fan and the condenser fan), and the heating coil (includes electricity consumption rate for electric heating coil or parasitic electricity consumption rate for non-electric coils). This value is calculated for each HVAC system timestep being simulated, and the results are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Electric Consumption[J]

This output field is the electricity consumption of the packaged terminal air conditioner in Joules for the time period being reported. The consumption includes electricity used by the compressor (including crankcase heater), fans (indoor supply air fan and the condenser fan), and the heating coil (includes electricity consumption for electric heating coil or parasitic electricity consumption for non-electric coils). This value is calculated for each HVAC system timestep being simulated, and the results are summed for the timestep being reported.

Packaged Terminal Air Conditioner Fan Part-Load Ratio

This output field is the part-load ratio of the fan. The fan part-load ratio is defined as the average supply air mass flow rate divided by the maximum supply air mass flow rate. The maximum supply air mass flow rate depends on whether heating, cooling, or no heating or cooling is required during the timestep. This value is calculated for each HVAC system timestep being simulated, and the results are averaged for the timestep being reported.

Packaged Terminal Air Conditioner Part-Load Ratio

This output field is the part-load ratio used by the coils (cooling and heating). Part-load ratio is defined as the total coil load divided by the coil steady-state capacity. This value is calculated for each HVAC system timestep being simulated, and the results are averaged for the timestep being reported.

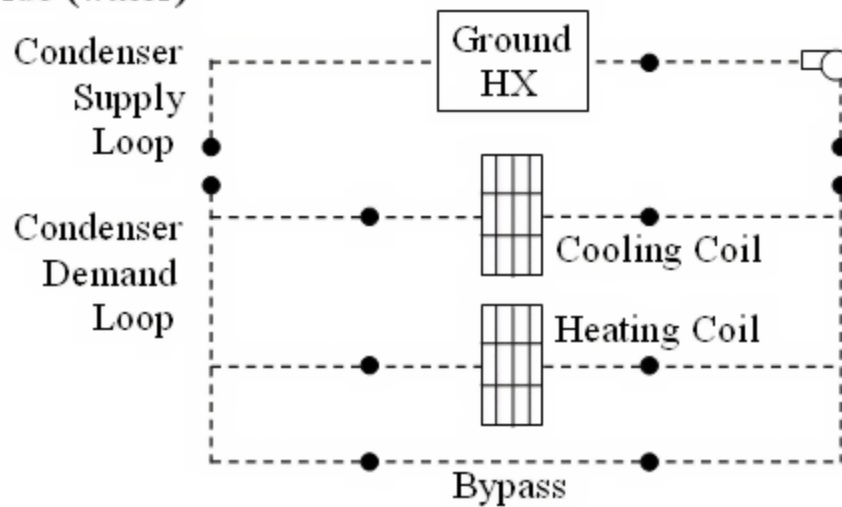
Zone Water To Air Heat Pump

The zone water-to-air heat pump is a compound component consisting of a fan, water-to-air cooling and heating coils, and a supplemental heating coil. The heat pump switches between cooling and heating depending on the zone's demand. The zone load (air) side of the zone water-to-air heat pump consists of an On/Off fan component, a [Water to air heat pump cooling coil](#) component, a [Water to air heat pump heating coil](#) component, and a Gas or Electric supplemental heating coil component. The source side (water) of the heat pump is connected to a condenser loop with a heat exchanger (ground heat exchanger or other type) or a plant loop with a heating source such as a boiler and a cooling source such as a chiller or cooling tower.

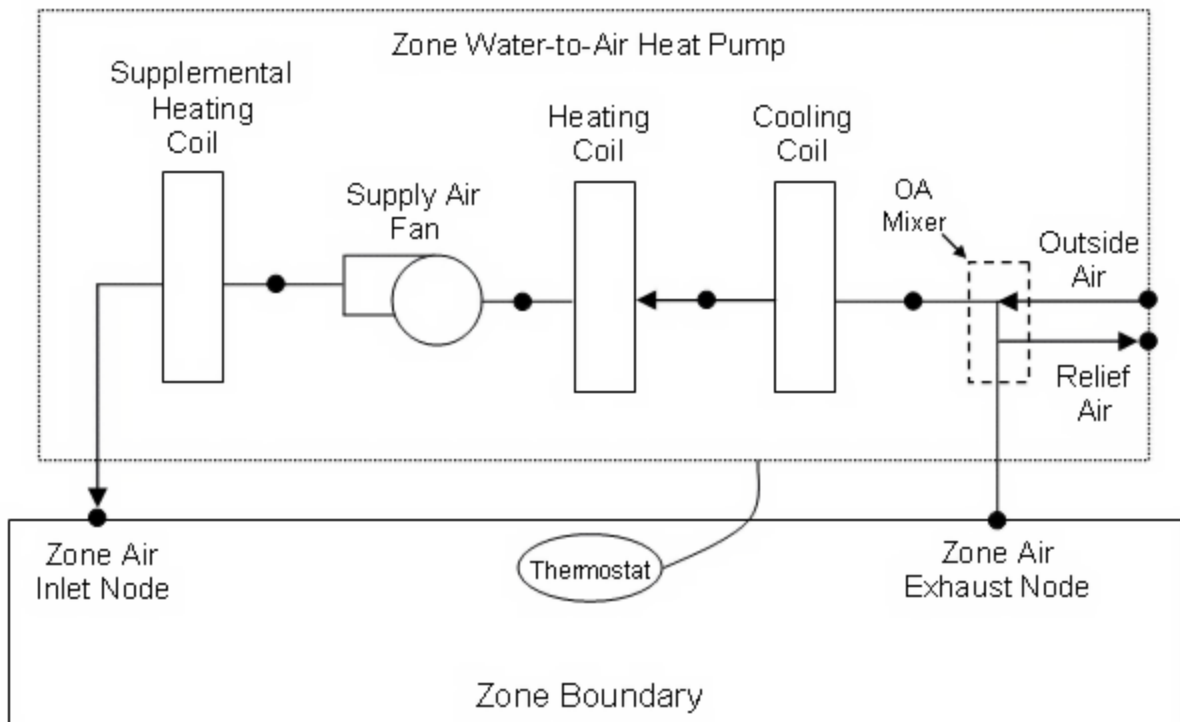
Application

The diagram below shows the setup and connection of the heat pump for the source side and load side for a ground heat exchanger configuration.

Source Side (water)

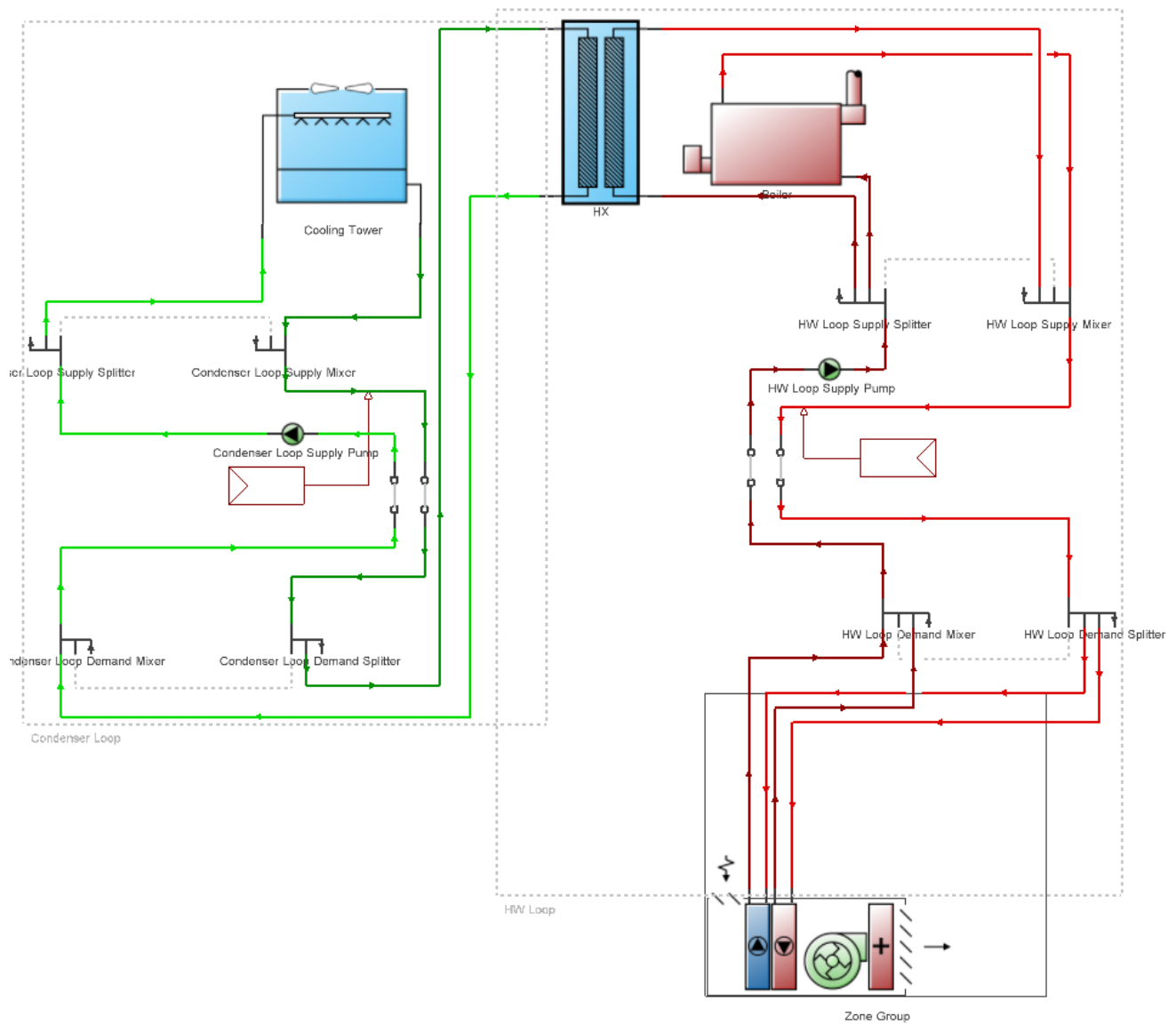


Load Side (air)



Note that on the load side, the cooling coil is always placed before the heating coil.

A typical whole system configuration within DesignBuilder is shown below:



Note the use of a HX component to provide free cooling from a cooling tower from a connected condenser loop to a mixed water loop which can provide either heating or cooling depending on the zone loads. The HW loop used to supply the Zone water to air heat pump was configured as follows:

- A [Dual setpoint SPM](#) added to the supply side to apply different setpoints depending on whether the plant loop is supporting a heating or cooling load.
- On the HW loop dialog:
 - a. Set the [Plant loop demand calculation scheme](#) to **2-DualSetPointDeadband**.
 - b. Set up Two [Plant equipment operation schemes](#) to switch in either the boiler or the cooling tower depending on the load on the loop:

Plant loop Data	
General	Plant Equipment Operation
General	
Number of schemes	2
Scheme 1	
Operation type	3-Heating load
Operation schedule	On 24/7
Number of ranges	1
Range 1	
Lower limit (W)	0.00
Upper limit (W)	1000000000000000.00
Equipment	
☒ Boiler	
Priority	1
☐ HX	
Scheme 2	
Operation type	2-Cooling load
Operation schedule	On 24/7
Number of ranges	1
Range 1	
Lower limit (W)	0.00
Upper limit (W)	1000000000000000.00
Equipment	
☐ Boiler	
☒ HX	
Priority	1

The HX component is controlled using the **1-Cooling differential on/off** Control type.

General

Name

A unique system-assigned name for this zone system heat pump. Any reference to this unit by another object will use this name.

Operation

Availability schedule

Select the schedule that defines when the heat pump is able to operate. A schedule value equal to 0 means that the heat pump is off at that time and a value greater than 0 denotes that it is available to operate. This schedule may be used to completely disable the heat pump (all of its coils and the supply air fan) as required.

Fan

Supply fan placement

There are 2 options:

- **1-Blow through** where the supply air fan is before both the heat pump cooling/heating coils and the supplementary heating coil.

- **2-Draw through** where the supply fan is between the heat pump cooling/heating coils and the supplementary heating coil.

Supply fan delay time

This numeric field contains the time delay in seconds for the heat pump supply air fan to shut off after compressor cycle off. This value can be obtained from the manufacturer or the heat pump catalog. Suggested value is 60 seconds.

Supply Air Flow Rate

Supply air flow rate during cooling operation

This numeric field defines the supply air flow rate leaving the heat pump (in m³/s or ft³/s) when the DX cooling coil is operating. Values must be greater than 0 or this field is autosizable.

Supply air flow rate during heating operation

This numeric field defines the supply air flow rate leaving the heat pump (in m³/s or ft³/s) when the DX heating coil and/or supplemental heater are operating. Values must be greater than 0 or this field is autosizable.

Supply air flow rate when no heating or cooling needed

This numeric field defines the supply air flow rate leaving the heat pump (in m³/s or ft³/s) when neither cooling nor heating is required (i.e., DX coils and supplemental heater are off but the supply air fan operates). This field is only used when the heat pump's supply air fan operating mode schedule specifies continuous fan operation. Values must be greater than or equal to zero, or this field is autosizable. If the heat pump's supply air fan operating mode schedule specifies continuous fan operation and this value is set to zero, then the model assumes that the supply air flow rate when no cooling/heating is needed is equal to the supply air flow rate when the cooling or heating coil was last operating (for cooling operation or heating operation).

Outdoor Air Supply

Outdoor air supply

Check this check box if the heat pump unit is to provide outdoor air to the zone.

Cooling outdoor air flow rate

This numeric field defines the outdoor air flow rate through the heat pump (in m³/s or ft³/s) when the DX cooling coil is operating. Values must be greater than or equal to 0, or this field is autosizable. Note that the outside air flow rate during cooling operation is fixed; it cannot change during the simulation. In addition, the outside air flow rate during cooling operation cannot be greater than the heat pump's supply air flow rate during cooling operation.

Heating outdoor air flow rate

This numeric field defines the outdoor air flow rate through the heat pump (in m³/s or ft³/s) when the DX heating coil and/or supplemental heater are operating. Values must be greater than or equal to 0, or this field is autosizable. Note that the outside air flow rate during heating operation is fixed; it cannot change during the simulation. In addition, the outside air flow rate during heating operation cannot be greater than the heat pump's supply air flow rate during heating operation.

Outdoor air flow rate when no cooling or heating is needed

This numeric field defines the outdoor air flow rate through the heat pump (in m³/s or ft³/s) when neither cooling or heating is required (i.e., DX coils and supplemental heater are off but the supply air fan operates). Values must be greater than or equal to 0, or this field is autosizable. Note that the no load outdoor air flow rate cannot change during the simulation. In addition, the no load outdoor air flow rate cannot be greater than the heat pump's no load supply air flow rate. This field is only used when the heat pump's supply air fan

operating mode schedule specifies continuous fan operation. If the heat pump's supply air fan operating mode schedule specifies continuous fan operation and the field [Supply air flow rate when no heating or cooling needed](#) set to zero, then the model assumes that the no load outdoor air flow rate is equal to the outdoor air flow rate when the cooling or heating coil was last operating (for cooling operation [i.e., Cooling outdoor air flow rate] or heating operation [i.e., Heating outdoor air flow rate]) and this field is not used.

On Cycle Power Use

Fraction of on-cycle power use

This numeric field contains the fraction of on-cycle power use to adjust the part load fraction based on the off-cycle power consumption due to crankcase heaters, controls, fans, and etc. Suggested value values are below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	0.01
Good	0.01
Poor	0.03

Cooling Coil

Cooling coil time constant

This numeric field contains the time constant for the cooling coil's capacity to reach steady state after startup. Suggested values are shown below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	60
Good	60
Poor	60

Compressor

Maximum cycling rate

This numeric field contains the maximum on-off cycling rate for the compressor, which occurs at 50% run time fraction. Suggested values are shown below (Henderson et al. 1999):

Heat Pump Condition	Recommended Values
Typical	2.5
Good	2.5
Poor	3.0

Supplemental Heater

Maximum supply air temperature from supplemental heater

This numeric field defines the maximum allowed supply air temperature exiting the heat pump supplemental heating coil (in °C or °F).

Maximum outdoor dry-bulb temperature for supplemental heater operation

This numeric field defines the outdoor air dry-bulb temperature (in °C or °F) above which the heat pump supplemental heating coil is disabled. The temperature for this input field must be less than or equal to 21°C.

Supply air fan operating mode

Select the name of the supply air fan operating mode schedule. The supply air fan operating mode may vary during the simulation based on time-of-day or with a change of season. Schedule values of 0 denote that the supply air fan and the heating or cooling coil cycle on and off together to meet the heating or cooling load (a.k.a. AUTO fan). Schedule values other than 0 denote that the supply air fan runs continuously while the heating or cooling coil cycles to meet the load.

Note: This setting is not available yet for selection. DesignBuilder provides a constant value of 1 in the simulation, i.e. the fan will run continuously while the coils cycle to meet the load.

Coil Water Flow Mode

Heat Pump Coil Water Flow Mode

This field specifies the way in which water flow through the heat pump coils will be modelled. There are 3 options:

- **1-Cycling** varies water flow through the coil based on the heat pump Part Load Ratio. This control method is appropriate for modeling heat pumps that are outfitted with a solenoid valve which allows water to flow through the coil only when the compressor is active. This is the default option.
- **2-Constant** provides a constant water flow regardless of heat pump operation. Remember that EnergyPlus has two coils (a heating coil and a cooling coil) to approximate the operation of one coil that can operate in either heating mode or cooling mode. Therefore, when the water flow mode is constant, there will be full flow through either the heating coil or the cooling coil, but not both at the same time.
- **3-Constant on demand** provides full flow through the coil whenever there is a load. When there is no load, there is zero flow through the coil.

Note: This field is only used when WatertoAirHeatPump:EquationFit coils are used.

VRF Indoor Unit



ZoneHVAC:TerminalUnit:VariableRefrigerantFlow

Used in:

HVAC Zones

VRF Indoor units with variable refrigerant flow DX coils are used exclusively with variable refrigerant flow (VRF) systems. Indoor units operate to satisfy a heating or cooling load in a zone based on a zone thermostat temperature set point. Direct-expansion (DX) cooling and DX heating coils are specified and used depending on the operating mode required. Although a DX cooling and a DX heating coil are included in the terminal unit, only one may be used at any one time.

Outdoor ventilation air is modelled with the use of an optional outside air mixer object. Outside air can be provided to the zone in one of two ways: either only when the coil is operating, or continuously even when the coil is not operating. A supply air fan is also included and it can be modelled as either draw through or blow through.

The zone terminal unit will typically be the first equipment operating for both cooling and heating (i.e., Sequence = 1 on the [Heating and Cooling Calculation Sequence](#) tab of the HVAC zone group dialog). Other Zone HVAC equipment may be used in the same zone and should be sequenced to operate after the zone terminal units (i.e., sequence = 2 or higher).

General

Name

The name of the VRF indoor unit is defined internally and cannot be changed.

Operation

Availability schedule

Select the schedule that defines when the terminal unit is able to operate. A schedule value equal to 0 means that the unit is off at that time and a value greater than 0 denotes that it is available to operate. This schedule may be used to completely disable the unit as required.

Supply Air Flow Rates

Supply air flow rate during cooling operation

This numeric field defines the terminal unit's operating volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit is operating in cooling mode.

Supply air flow rate during heating operation

This numeric field defines the terminal unit's operating volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit is operating in heating mode.

Supply air flow rate when no cooling is needed

This numeric field defines the terminal unit's operating volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit's cooling coil is not operating and the previous mode was cooling.

Supply air flow rate when no heating is needed

This numeric field defines the terminal unit's operating volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit's heating coil is not operating and the previous mode was heating.

Outdoor Air Supply

Outdoor air supply

Check this checkbox if the VRF indoor unit has its own supply of fresh air. Then enter the 3 flow rates described below.

Outdoor air flow rate during cooling operation

This autosizable field defines the outdoor air volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit is operating in cooling mode.

Outdoor air flow rate during heating operation

This autosizable field defines the outdoor air volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit is operating in heating mode.

Outdoor air flow rate when no cooling or heating is needed

This autosizable field defines the outdoor air volumetric air flow rate (in m³/s or ft³/min). This volumetric air flow rate is used when the terminal unit is not operating in cooling or heating mode.

Parasitic Electric Energy

Zone terminal unit on parasitic electric energy use

This numeric field defines the parasitic electrical energy use of the zone terminal unit when either terminal unit coil is operating. When in cooling mode, this electric energy use is reported in a zone terminal unit cooling electric consumption output variable. When in heating mode, this electric energy use is reported in a zone terminal unit heating electric consumption output variable. Units are W or Btu/hr.

Zone terminal unit off parasitic electric energy use

This numeric field defines the parasitic electrical energy use of the zone terminal unit when the terminal unit coil(s) is not operating. When the previous mode was cooling, this electric energy use is reported in a zone terminal unit cooling electric consumption output variable. When the previous mode was heating, this electric energy use is reported in a zone terminal unit heating electric consumption output variable. Units are W or Btu/hr.

Supply Fan

Fan type

This choice field contains the identifying type of supply air fan specified for the indoor unit. Fan type must be one of:

- **1-On/off fan** used when the fan cycles on and off with the cooling or heating coil (i.e. [Supply air fan operating mode schedule](#) values are at times 0).
- **2-Constant volume fan** used when the [Supply air fan operating mode schedule](#) values are never 0 and the fan operates continuously.

Supply air fan placement

This option has two choices:

- **1-Blow through** which means that the unit consists of a fan followed by the DX coils. The fan “blows through” the cooling and heating coils. If an outside air mixer is used, the fan inlet connects to the outside air mixer’s mixed air node. If an outside air mixer is not used, the fan inlet connects to the zone exhaust node. For this configuration, the fan outlet always connects to the DX cooling coil inlet node (or if a DX cooling coil is not used, the DX heating coil inlet node).
- **2-Draw through**, which means that the unit consists of the DX coil(s) followed by a fan. The fan “draws air through” the DX coil(s). In this case the fan inlet always connects to the DX heating coil outlet node (or if a DX heating coil is not use, the DX cooling coil outlet node) and the fan outlet node always connects to the zone inlet node.

Supply air fan operating mode schedule

Schedule values equal to 0 denote cycling fan/cycling coil operation. All other schedule values denote constant fan/cycling coil operation.

Heating Capacity

Override outdoor unit rated total heating capacity sizing ratio

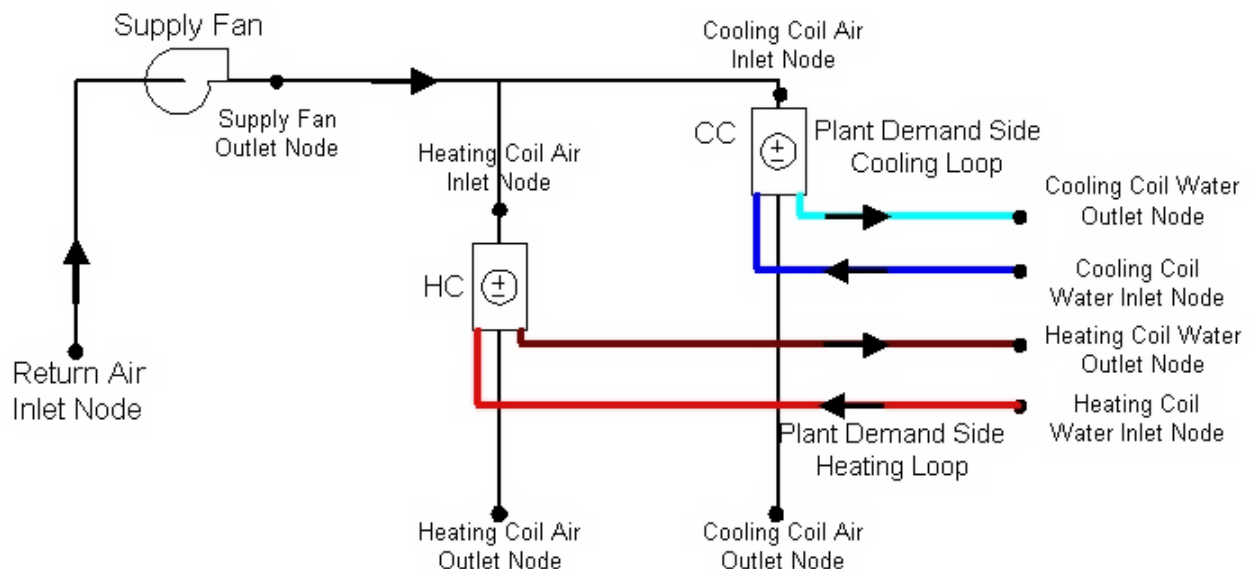
Check this option if you would like to override the [Rated total heating capacity sizing ratio](#) used for the Outdoor unit.

Rated total heating capacity sizing ratio

This field defines the ratio of the heating coil to cooling coil size when DX heating coil capacity is autosized. It must be greater than or equal to 1. The default value is 1.

Heating And Cooling Coil Data

Several different coils may be added to zone equipment and air loops. This includes simple heating (gas, electric, and hot water) and a simple water-cooling coil as well as a more detailed flat fin water-cooling coil model. In general, the heating coil description can be used for a heating coil, a reheat coil, or a preheat coil in the air loop simulation or for zone equipment. Figure is an example of a heating and cooling coil in the air loop simulation in a dual duct system. This does show the basic node structure that will be referenced in the input description. The figure does show water coils since they are the most complex to input in the simulation compared to the Electric and Gas coils which only have air connections.



- "Heating Coils " on page 1212
- "Cooling Coils" on page 1228

Heating Coils

These heating coil types are available within Detailed HVAC:

- [Heating Coil - Water](#)
- [Heating Coil - Electric](#)
- "Heating Coil - Fuel" on page 1217
- [Heating Coil - DX](#)
- "Heating Coil - VRF - DX" on page 1224
- "Heating Coil - Water to Air Heat Pump" on page 1226

Heating Coil - Water



Coil:Heating:Water

Used in:

- **Air Handling Units**
- **Demand loops of Dual duct systems**
- **Air Distribution Units**

The simple water heating coil model provides sensible heating of the air. The calculations use the Effectiveness-NTU algorithm and assume a cross-flow heat exchanger.

General

Name

An auto-generated name for the heating coil.

Type

The type of a hot water coil must be:

- **1-Water**

In all cases it will also be possible to select **2-Electric** in which case the [Electric heating coil](#) is used or **3-Fuel** where the [Fuel heating coil](#) is used instead.

Performance input method

The user can choose one of:

- **1-Nominal capacity**, or,
- **2-UA and Design water flow rate**.

If the **2-UA and Design water flow rate** option is selected, you must input values for **UA** of the Coil and **Max water flow rate** of the Coil. If **1-Nominal capacity** is chosen you must input a **Rated capacity** and **UA** of the Coil.

Rated capacity is defined as the heating capacity in watts of the coil at the rating points (i.e. the rated inlet and outlet water/air temperatures defined below). The rated capacity is used to calculate a water mass flow rate and a UA for the coil. The default is **1-Nominal capacity**.

To autosize the capacity, choose **2-UA and Design water flow rate** and use autosize as the inputs for UA, **Maximum water flow rate**, and **Rated capacity**. EnergyPlus will use the Sizing inputs to size the coil. The rated temperatures (see below) are not used in autosizing. These are used only when the user is specifying coil performance using the **1-Nominal capacity** input method.

Maximum water flow rate

The maximum possible water flow rate (m3/sec or gal/min) through the coil. This field is used when [Performance input method](#) = **2-UA and Design water flow rate**. This field is autosizable.

UA (U-Factor x Area)

Enter the UA value for the coil needed for the Effectiveness-NTU heat exchanger model. An estimate of the UA can be obtained from:

$$q = UA \times (T_{water,avg} - T_{air,avg})$$

where:

q is the heat transferred from water to the air in W;

T_{water, avg} is the average water temperature in °C; and

T_{air, avg} is the average air temperature in °C. Or the LMTD temperature difference can be used. This field is only used when [Performance input method](#) = **2-UA and Design water flow rate**

This field is autosizable.

Rated capacity

The heating capacity of the coil (in W or Btu/h) at the rated inlet and outlet air and water temperatures. This field is used when the [Performance input method](#) = **1-Nominal capacity**. This field is autosizable. The rating points are given in the four subsequent input fields.

Water Temperatures

Rated inlet water temperature

The inlet water temperature (°C or °F) corresponding to the rated heating capacity. The default is 82.2°C (180°F).

Rated outlet water temperature

The outlet water temperature (°C or °F) corresponding to the rated heating capacity. The default is 71.1°C (160°F).

Air Temperatures

Rated inlet air temperature

The inlet air temperature (°C or °F) corresponding to the rated heating capacity. The default is 16.6°C (60°F)..

Rated outlet air temperature

The outlet air temperature (°C or °F) corresponding to the nominal heating capacity. The default is 32.2°C (90°F).

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the coil can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

Controller

Control variable

The Control variable defines how the coil is controlled. Temperature control is currently the only option.

- **1-Temperature** control is used for air temperature control and is normally specified for the coil's outlet air node.

Action

The Action determines how the controlled variable (e.g. mass flow rate through a water coil) is changed based on the control signal. In a heating coil where water mass flow rate is to be controlled, the mass flow rate through the coil is increased when more heating is required, which increases the value of heat transfer from the water to the air stream. So this is considered a "normal" action controller and so the value of this field is fixed as **1-Normal**.

Actuator variable

This was again meant to be more generic but currently has only been used to control the water mass flow rate of a heating or cooling coil. This actuator variable is fixed as **1-Flow** to control the water mass flow rate.

Controller convergence tolerance

The coil is controlled by knowing the outlet temperature and/or humidity ratio specified by the setpoint managers, and setting the outlet conditions from the coil to meet these setpoints. The hot water coils use complex models that cannot be inverted directly. Therefore, to determine the correct mass flow rate the models are inverted numerically using an iterative procedure. The iterative solution uses an interval-halving routine and needs a termination criteria that is set here.

The convergence tolerance is the maximum difference between the actual temperature at the setpoint node and the setpoint temperature. This control offset is typically set to a small temperature difference, such as 0.01°C. The default is Autosize.

Maximum actuated flow

This is the maximum water flow (m3/sec or gal/min) through the coil. Set to the maximum design water flow for the coil.

Minimum actuated flow

Set to the minimum design water flow (m3/sec or gal/min) for the water coil, normally a shut off valve that is set to zero.

Rated ratio for air and water convection

This is the ratio of convective heat transfers between air side and water side of the heating coil at the rated operating conditions. The default is 0.5. This ratio describes the geometry and the design of the coil and is defined by:

$$ratio = \frac{\eta_f hA_{air}}{hA_{water}}$$

where η_f is the fin efficiency.

h is the surface convection heat transfer coefficient.

A is the surface area.

Heating Coil - Water Outputs

HVAC,Sum, Total Water Heating Coil Energy[J]

HVAC,Average,Total Water Heating Coil Rate[W]

HVAC,Average,Water Heating Coil U-factor Times Area [W]

Total Water Heating Coil Energy (J)

Total Water Heating Coil Energy is the total amount of heat transfer taking place in the coil at the operating conditions.

Total Water Heating Coil Rate (W)

Total Water Heating Coil Rate is the Rate of heat transfer taking place in the coil at the operating conditions. The units are (J/sec) or Watts.

Water Heating Coil U-factor Times Area (W/K)

This characterizes the overall heat transfer “UA” value, or U-factor times Area. The simple heating coil model adjusts UA value based on inlet temperatures and flow rates and this

Heating Coil - Electric



Coil:Heating:Electric

Used in:

- Air Handling Units
- Demand loops of Dual duct systems
- PTAC
- PTHP
- Unitary Heat Cool
- Unitary Heat Pump

The electric heating coil is a simple capacity model with a user-defined efficiency. In most cases, the efficiency for the electric coil will be 100%. Depending on where it is used this coil may be either temperature or capacity controlled. If used in the air loop simulation it will be controlled to a specified temperature scheduled from the Setpoint Manager. If it is used in zone equipment, it will be controlled from the zone thermostat by meeting the zone demand.

General

Name

A unique auto-generated name for the coil.

Type

The type is fixed as:

- **2-Electric.**

In [Generic AHUs](#) and [ADUs](#) it will also be possible to select **1-Hot water** or **3-Fuel** as the heating coil type, in which case a [Water heating coil](#) or [Fuel heating coil](#) is used instead.

Efficiency

This is user-inputted efficiency (fraction, not percent) and can account for any loss. In most cases for the electric coil, this will be 100% (enter a value of 1).

Nominal capacity

This is the maximum capacity of the coil (in W). This controlled coil will only provide the needed capacity to meet the control criteria whether it is temperature or capacity controlled. This field is autosizable.

Operation

Available schedule

The [schedule](#) that denotes whether the heating coil can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the coil can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the coil must be off.

Heating Coil - Electric Outputs

HVAC,Sum,Heating Coil Energy[J]
 HVAC,Average,Heating Coil Rate[W]
 HVAC,Sum,Heating Coil Electric Consumption [J]
 HVAC,Average,Heating Coil Electric Power [W]

Heating Coil Energy (J)

Heating Coil Energy is the total amount of heat transfer taking place in the coil at the operating conditions.

Heating Coil Rate[W]

Heating Coil Rate is the Rate of heat transfer taking place in the coil at the operating conditions. The units are (J/sec) or Watts.

Heating Coil Electric Consumption [J]

Heating Coil electric consumption after the efficiency of the coil has been taken into account in Joules for the timestep reported.

Heating Coil Electric Power [W]

This field is the average Heating Coil electric power after the efficiency of the coil has been taken into account in Watts for the timestep reported.

Heating Coil - Fuel



Coil:Heating:Fuel

Used in:

- **Air Handling Units**
- **Demand loops of Dual duct systems**
- **Air Distribution Units**

The fuel heating coil is a simple capacity model with a user inputted burner efficiency. The default for the burner efficiency is 80%. The coil can be used in air loops or in zone equipment as a reheat coil. Depending on where it is used determines whether this coil is temperature or capacity controlled. If used in the air loop simulation it will be controlled to a specified temperature scheduled from the [Setpoint manager](#). If it is used in zone equipment, it will be controlled from the zone thermostat by meeting the zone demand.

Name

A unique identifying name for each coil.

Type

The type of a fuel coil must be:

- **3-Fuel**

In all cases it will also be possible to select **2-Electric** in which case the [Electric heating coil](#) is used or **1-Water** where the [Water heating coil](#) is used instead.

Fuel

Select the fuel to be used for supplying heat to the coil from the list:

- **1-Gas,**
- **2-Natural gas,**
- **3-Propane,**

- **4-Diesel,**
- **5-Gasoline,**
- **6-Fuel oil 1,**
- **7-Fuel oil 2,**
- **8-Other fuel 1,**
- **9-Other fuel 2**

Efficiency

This is user inputted burner efficiency (decimal, not percent) and is defaulted to 0.8 (80%).

Parasitic electric load

This is the parasitic electric load associated with the coil operation, such as an inducer fan, etc.. This will be modified by the PLR (or coil runtime fraction if a part-load fraction correlation is provided in the next input field) to reflect the time of operation in a simulation timestep.

Parasitic fuel load

This numeric field is the parasitic fuel load associated with the coil's operation (in W), such as a standing pilot light. The model assumes that this parasitic load is consumed only for the portion of the simulation timestep where the heating coil is not operating.

Nominal capacity

This is the maximum capacity of the coil (in W). This controlled coil will only provide the needed capacity to meet

Part Load Performance

Part load fraction correlation

Check this option if a part load fraction correlation is to be applied.

Part load fraction correlation curve

The [Quadratic](#) or [Cubic](#) performance curve selected here parameterises the variation of fuel consumption rate by the heating coil as a function of the part load ratio (PLR, sensible heating load/nominal capacity of the heating coil). For any simulation timestep, the nominal fuel consumption rate (heating load/burner efficiency) is divided by the part-load fraction (PLF) if a part-load curve has been defined. The part-load curve accounts for efficiency losses due to transient coil operation.

The part-load fraction correlation should be normalized to a value of 1.0 when the part load ratio equals 1.0 (i.e., no efficiency losses when the heating coil runs continuously for the simulation timestep). For PLR values between 0 and 1 ($0 \leq \text{PLR} < 1$), the following rules apply:

$\text{PLF} \geq 0.7$ and $\text{PLF} \geq \text{PLR}$ If $\text{PLF} < 0.7$ a warning message is issued, the program resets the PLF value to 0.7, and the simulation proceeds. The runtime fraction of the heating coil is defined a PLR/PLF . If $\text{PLF} < \text{PLR}$, then a warning message is issues and the runtime fraction of the coil is limited to 1.0.

A typical part load fraction correlation for a conventional gas heating coil (e.g., residential furnace) would be:

$$\text{PLF} = 0.8 + 0.2(\text{PLR})$$

Operation

Available schedule

Schedule that defines when the coil is available.

Heating Coil - DX



Coil:Heating:DX:SingleSpeed

Used in:

- PTAC,
- PTHP,
- Generic AHU,
- Unitary Heat Pump AHU

The single speed heating DX coil model uses performance information at rated conditions along with curve fits for variations in total capacity, energy input ratio and part load fraction to determine performance at part-load conditions. The impacts of various defrost strategies (reverse cycle, resistive, timed or on-demand) are modelled based on a combination of user inputs and empirical models taken from the air-to-air heat pump algorithms in DOE-2.1E.

General

Name

This alpha field defines a unique user-assigned name for an instance of a DX heating coil. Any reference to this DX heating coil by another object will use this name.

Rated total heating capacity

This numeric field defines the total, full load heating capacity (in W or Btu/h) of the DX coil unit **at rated conditions** (outdoor air dry-bulb temperature of 8.33°C, outdoor air wet-bulb temperature of 6.11°C, heating coil entering air dry-bulb temperature of 21.11°C, heating coil entering air wet-bulb temperature of 15.55°C, and a heating coil air flow rate defined by field “rated air flow volume” below).

The value entered here must be greater than 0. Capacity should not include supply air fan heat. **Rated total heating capacity** should be within 20% of the **Rated total cooling capacity**, otherwise a warning message is issued.

Note: Manufacturers may calculate coil performance at different rating conditions to those used by EnergyPlus. For example, in Europe heating coil performance data is provided at the following conditions, in accordance with EN14511:

Indoor ambient temperature: 20°CDB / 15°CWB
Outdoor ambient temperature: 7°CDB / 6°CWB

Although the EN14511 Indoor Ambient Temp rated conditions of 20°CDB doesn't match exactly with the EnergyPlus rated condition of 21.11°CDB (for example), the difference in capacity should be minor and the manufacturer data can be used with confidence.

Gross rated COP

This numeric field defines the coefficient of performance (COP=heating power output in watts divided by electrical power input in watts) of the DX heating coil unit at rated conditions (outdoor air dry-bulb temperature of 8.33°C, outdoor air wet-bulb temperature of 6.11°C, coil entering air dry-bulb temperature of 21.11°C, coil entering air wet-bulb temperature of 15.55°C, and a heating coil air flow rate defined by field “rated air flow volume rate” below). The value entered here must be greater than 0. The input power includes power for the compressor(s) and outdoor fan(s) but does not include the power consumption of the indoor supply air fan. The heating power output is the value entered above in the field **Rated total heating capacity**.

Rated air flow

This numeric field defines the volume air flow rate (in m³/s or ft³/min), across the DX heating coil at rated conditions. The value entered here must be greater than 0. The rated air volume flow rate should be between 0.00004027 m³/s and 0.00006041 m³/s per watt of rated total heating capacity. The **Rated total heating capacity** and **Rated COP** should be performance information for the unit with outdoor air dry-bulb temperature of 8.33°C, outdoor air wet-bulb temperature of 6.11°C, heating coil entering air dry-bulb temperature of 21.11°C, heating coil entering air wet-bulb temperature of 15.55°C, and the rated air volume flow rate defined here.

2017 Rated supply fan power per volume flow rate

This field is the electric power for the heating coil (condenser) fan per air volume flow rate through the coil at the rated conditions (in W/(m³/s) or W/(ft³/min)). The default value is 773.3 W/(m³/s) (365 W/1000 cfm). Values must be ≥ 0.0 and ≤ 1250 W/(m³/s).

Note: This value is only used to calculate the following metrics according to the 2017 version of the ANSI/AHRI 210-240 standard: High Temperature Heating Standard (Net) Rating Capacity, Low Temperature Heating Standard (Net) Rating Capacity, and Heating Seasonal Performance Factor (HSPF). These values will be outputs in the EnergyPlus eio file (see [EnergyPlus Engineering Reference](#), Single Speed DX Heating Coil, Standard Ratings) and in the predefined tabular output reports (Output:Table:SummaryReports object, Equipment Summary). This value is not used for modelling the heating coil (condenser) fan during simulations; instead, it is used for calculating the above standard ratings to assist the user in verifying their inputs for modelling this type of equipment.

2023 Rated supply fan power per volume flow rate

This field is the electric power for the heating coil (condenser) fan per air volume flow rate through the coil at the rated conditions (in W/(m³/s) or W/(ft³/min)). The default value is 934.4 W/(m³/s) (441 W/1000 cfm). Values must be ≥ 0.0 and ≤ 1505 W/(m³/s).

Note: This value is only used to calculate the following metrics according to the 2023 version of the ANSI/AHRI 210-240 standard: High Temperature Heating Standard (Net) Rating Capacity, Low Temperature Heating Standard (Net) Rating Capacity, and Heating Seasonal Performance Factor (HSPF2). These values will be outputs in the EnergyPlus eio file (see [EnergyPlus Engineering Reference](#), Single Speed DX Heating Coil, Standard Ratings) and in the predefined tabular output reports (Output:Table:SummaryReports object, Equipment Summary). This value is not used for modelling the heating coil (condenser) fan during simulations; instead, it is used for calculating the above standard ratings to assist the user in verifying their inputs for modelling this type of equipment.

Compressor

Minimum outdoor dry-bulb temperature for compressor operation

This numeric field defines the minimum outdoor air dry-bulb temperature (in °C or °F) where the heating coil compressor turns off. The temperature for this input field must be greater than or equal to -20°C. The default value is -8°C.

Outdoor dry-bulb temperature to turn on compressor

This numeric field defines the outdoor air dry-bulb temperature (in °C or °F) when the compressor is automatically turned back on following an automatic shut off because of low outdoor temperature. This field is only used in calculating the Heating Seasonal Performance Factor (HSPF (2017 ANSI/AHRI standard) and HSPF2 (2023 ANSI/AHRI standard)) of heating coils.

Defrost

Defrost strategy

Choose from two options:

- **1-Reverse-cycle** or
- **2-Resistive.**

If the reverse-cycle strategy is selected, the heating cycle is reversed periodically to provide heat to melt frost accumulated on the outdoor coil. If a resistive defrost strategy is selected, the frost is melted using an electric resistance heater. The default defrost strategy is reverse-cycle.

Defrost control

Choose from two options:

- **1-Timed** or
- **2-On-demand.**

If timed control is selected, the defrost time period is calculated based on a fixed value or compressor runtime whether or not frost has actually accumulated. For timed defrost control, the fractional amount of time the unit is in defrost is entered in the input field [Defrost time period fraction](#) described below. If on-demand defrost control is selected, the defrost time period is calculated based on outdoor weather (humidity ratio) conditions. Regardless of which defrost control is selected, defrost does not occur above the user specified outdoor temperature entered in the input field [Maximum outdoor dry-bulb temperature for defrost operation](#) described above.

Defrost time period fraction

This numeric field defines the fraction of compressor runtime when the defrost cycle is active, and only applies to “timed” defrost (see Defrost Control input field above). For example, if the defrost cycle is active for 3.5 minutes for every 60 minutes of compressor runtime, then the user should enter $3.5/60 = 0.058333$. The value for this input field must be greater than or equal to 0. The default value is 0.058333.

Resistive defrost heater capacity

This numeric field defines the capacity of the resistive defrost heating element (in W). This input field is used only when the selected defrost strategy is ‘resistive’ (see input field [Defrost strategy](#) above). The value for this input field must be greater than or equal to 0.

Maximum outdoor dry-bulb temperature for defrost operation

This numeric field defines the outdoor air dry-bulb temperature (in °C or °F) above which outdoor coil defrosting is disabled. The temperature for this input field must be greater than or equal to 0 °C and less than or equal to 7.22°C. The default value is 5°C.

Performance Curves

The DX Cooling coil requires up to 6 curves be selected from the [Curves database](#) to define coil performance depending on the defrost strategy selected.

Total heating capacity function of temperature curve

The [Bi-quadratic](#), [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of the total heating capacity as a function of the both the indoor and outdoor air dry-bulb temperature or just the outdoor air dry-bulb temperature depending on the type of curve selected. The bi-quadratic curve is recommended if sufficient manufacturer data is available as it provides sensitivity to the indoor air dry-bulb temperature and a more realistic output. The output of this curve is multiplied by the rated total heating capacity to give the total heating capacity at specific temperature operating conditions (i.e., at an indoor air dry-bulb temperature or outdoor air dry-bulb temperature different from the rating point temperature). The curve is normalised to have the value of 1.0 at the rating point.

Total heating capacity function of flow fraction curve

The [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of total heating capacity as a function of the ratio of actual air flow rate across the heating coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the rated total heating capacity and the total heating capacity modifier curve (function of temperature) to give the total heating capacity at the specific temperature and air flow conditions at which the coil is operating. The curve is normalised to have the value of 1.0 when the actual air flow rate equals the rated air flow rate.

Energy input ratio function of temperature curve

The [Bi-quadratic](#), [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of the energy input ratio (EIR) as a function of the both the indoor and outdoor air dry-bulb temperature or just the outdoor air dry-bulb temperature depending on the type of curve selected. The bi-quadratic curve is recommended if sufficient manufacturer data is available as it provides sensitivity to the indoor air dry-bulb temperature and a more realistic output. The EIR is the inverse of the COP. The output of this curve is multiplied by the rated EIR (inverse of rated COP) to give the EIR at specific temperature operating conditions (i.e., at an indoor air dry-bulb temperature or outdoor air dry-bulb temperature different from the rating point temperature). The curve is normalised to have the value of 1.0 at the rating point.

Energy input ratio function of flow fraction curve

The [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of the energy input ratio (EIR) as a function of the ratio of actual air flow rate across the heating coil to the rated air flow rate (i.e., fraction of full load flow). The EIR is the inverse of the COP. The output of this curve is multiplied by the rated EIR and the EIR modifier curve (function of temperature) to give the EIR at the specific temperature and air flow conditions at which the coil is operating. This curve is normalised to a value of 1.0 when the actual air flow rate equals the rated air flow rate.

Part load fraction correlation curve

The [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of electrical power input to the DX unit as a function of the part load ratio (PLR, sensible heating load/steady-state sensible heating capacity). The product of the rated EIR and EIR modifier curves is divided by the output of this curve to give the “effective” EIR for a given simulation timestep. The part load fraction (PLF) correlation accounts for efficiency losses due to compressor cycling.

The 2 important rules to be followed when setting up this curve are:

1. The part load fraction correlation curve coefficients need to be defined so that the output of the curve equals 1.0 when the PLR is 1.0, e.g.

$$PLF = 0.85 + 0.15(PLR)$$

So the associated quadratic would have Coefficient 1 = 0.85 and Coefficient 2 = 0.15 with a maximum x = 1.0.

2. When $0.0 \leq PLR < 1.0$, the PLF must be ≥ 0.7 which is guaranteed as long as Coefficient 1 of the PLF curve is > 0.7 . Also, in this PLR range, the PLF must be less than the PLR which is also guaranteed as long as the coil capacity is always greater than the demand. If the demand is greater than the coil capacity and PLF becomes greater than PLR, the software automatically limits the runtime fraction (PLR/PLF) to 1.0.

Defrost energy input ratio function of temperature curve

The [Bi-quadratic](#) performance curve that parameterises the variation of the energy input ratio (EIR) during reverse-cycle defrost periods as a function of the outdoor air dry-bulb temperature and the wet-bulb temperature of the air entering the indoor coil. The EIR is the inverse of the COP. The output of this curve is multiplied by the coil capacity, the fractional defrost time period and the runtime fraction of the heating coil to give the defrost power at the specific temperatures at which the indoor and outdoor coils are operating. This curve is only required when a reverse-cycle defrost strategy is selected. The curve is normalised to a value of 1.0 at the rating point conditions.

Crankcase Heater

Crankcase heater capacity

This numeric field defines the crankcase heater capacity (in W). When the outdoor air dry-bulb temperature is below the value specified in the input field [Maximum outdoor dry-bulb temperature for crankcase heater operation](#) (described below), the crankcase heater is enabled during the time that the compressor is not running. If this heating coil is used as part of an air-to-air heat pump (Ref. UnitarySystem:HeatPump:AirToAir or PackageTerminal:HeatPump:AirToAir), the crankcase heater defined for this DX heating coil is enabled during the time that the compressor is not running for either heating or cooling (and the crankcase heater power defined in the DX cooling coil object is disregarded in this case). The value for this input field must be greater than or equal to 0. The default value is 0. To simulate a unit without a crankcase heater, enter a value of 0.

Maximum outdoor dry-bulb temperature for crankcase heater operation

This numeric field defines the outdoor air dry-bulb temperature (in °C or °F) above which the compressor's crankcase heater is disabled. The value for this input field must be greater than or equal to 0.0 C.

Operation

Availability schedule name

The [schedule](#) that denotes whether the DX heating coil can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

Heating Coil - DX Outputs

HVAC,Average,DX Coil Total Heating Rate[W]
 HVAC,Sum,DX Coil Total Heating Energy[J]
 HVAC,Average,DX Heating Coil Electric Power[W]
 HVAC,Sum,DX Heating Coil Electric Consumption[J]
 HVAC,Average,DX Heating Coil Electric Defrost Power[W]
 HVAC,Sum,DX Heating Coil Electric Defrost Consumption[J]
 HVAC,Average,DX Heating Coil Crankcase Heater Power[W]
 HVAC,Sum,DX Heating Coil Crankcase Heater Consumption[J]
 HVAC,Average,DX Heating Coil Runtime Fraction

DX Coil Total Heating Rate [W]

This field is the total heating rate output of the DX coil in Watts. This is determined by the coil inlet and outlet air conditions and the air mass flow rate through the coil.

DX Coil Total Heating Energy[J]

This is the total heating output of the DX coil in Joules over the timestep being reported. This is determined by the coil inlet and outlet air conditions and the air mass flow rate through the coil. This output is also added to a report meter with Resource Type = EnergyTransfer, End Use Key = CoolingCoils, Group Key = System (ref. Report Meter).

DX Heating Coil Electric Power[W]

This output is the electricity consumption rate of the DX coil compressor and outdoor fan(s) in Watts. This rate is applicable when the unit is providing heating to the conditioned zone(s), and excludes periods when the unit is in reverse-cycle defrost mode.

DX Heating Coil Electric Consumption[J]

This is the electricity consumption of the DX coil compressor and condenser fan(s) in Joules for the timestep being reported. This consumption is applicable when the unit is providing heating to the conditioned zone(s),

and excludes periods when the unit is in reverse-cycle defrost mode. This output is also added to a report meter with Resource Type = Electricity, End Use Key = Cooling, Group Key = System (ref. Report Meter).

DX Heating Coil Electric Defrost Power[W]

This is the electricity consumption rate of the DX coil unit in Watts when the unit is in defrost mode (reverse-cycle or resistive).

DX Heating Coil Electric Defrost Consumption[J]

This is the electricity consumption of the DX coil unit in Joules for the timestep being reported. This consumption is applicable when the unit is in defrost mode (reverse-cycle or resistive).

DX Heating Coil Crankcase Heater Power[W]

This is the average electricity consumption rate of the DX coil compressor's crankcase heater in Watts for the timestep being reported.

DX Heating Coil Crankcase Heater Consumption[J]

This is the electricity consumption of the DX coil compressor's crankcase heater in Joules for the timestep being reported. This output is also added to a report meter with Resource Type = Electricity, End Use Key = Miscellaneous, Group Key = System (ref. Report Meter).

DX Heating Coil Runtime Fraction

This is the runtime fraction of the DX heating coil compressor and outdoor fan(s) for the timestep being reported.

Heating Coil - VRF - DX

The variable refrigerant flow (VRF) DX heating coil model uses performance information at rated conditions along with performance curves for variations in total capacity, energy input ratio and part load fraction to determine performance at part-load conditions. The impacts of defrost operation is modeled based a combination of user inputs and empirical models taken from the air-to-air heat pump algorithms in DOE-2.1E.

The VRF DX heating coil input requires an availability schedule, the gross rated heating capacity and the rated air volume flow rate. The rated air volume flow rate should be between 0.00008056 m³/s and 0.00002684 m³/s per W of gross rated heating capacity.

Two performance curves are required. The first performance curve defines the heating capacity as a function of indoor air dry-bulb and outdoor condenser entering air wet-bulb temperature. This curve is specified in the variable refrigerant flow air-to-air heat pump object. The second performance curve is specific to each VRF DX heating coil and defines the change in heating capacity as a function of air flow fraction. These performance curves are further discussed below.

1. **Heating capacity modifier curve (function of temperature, specified in Heat Pump)**. The heating capacity modifier curve (function of temperature) can be a function of both the outdoor wet-bulb temperature and indoor air dry-bulb temperature. Users have the choice of a bi-quadratic curve with two independent variables or a tri-quadratic curve with three independent variable. The tri-quadratic curve is recommended if sufficient manufacturer data is available as it provides sensitivity to the combined total capacity of all indoor units connected to the heat pump condenser and a more realistic output. The output of this curve is multiplied by the gross rated heating capacity to give the gross heating capacity at specific temperature operating conditions (i.e. at an outdoor or indoor air temperature different from the rating point temperature) and the combination ratio of the installed system.
2. **Heating capacity modifier curve (function of flow fraction, specified in DX coil)**. The heating capacity modifier curve (function of flow fraction) is a quadratic or cubic curve with the independent variable being the ratio of the actual air flow rate across the heating coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the gross rated heating capacity and

the heating capacity modifier curve (function of temperature) to give the gross heating capacity at the specific temperature and air flow conditions at which the coil is operating.

General

Name

A unique auto-assigned name for an instance of a VRF DX heating coil. Any reference to this DX coil by another object will use this name.

Operation

Availability schedule

Select the DX heating coil availability schedule. Schedule values of 0 mean that the DX heating coil is off. A schedule value greater than 0 indicates that the coil can operate during the time period.

Airflow

Rated air flow rate

This numeric field defines the volume air flow rate (in m³/s or ft³/s) across the DX heating coil at rated conditions. The value entered here must be greater than 0. The rated air volume flow rate should be between 0.00004027 m³/s and 0.00006041 m³/s per W of gross rated heating capacity. The gross rated heating capacity and the gross rated COP should be performance information for the unit with outside air dry-bulb temperature of 8.33 C, outside air wet-bulb temperature of 6.11 C, heating coil entering air dry-bulb temperature of 21.11 C, heating coil entering air wet-bulb temperature of 15.55 C, and the rated air volume flow rate defined here.

Capacity

Gross rated heating capacity

This numeric field defines the total, full load gross heating capacity (in W or btu/hr) of the DX coil unit at rated conditions (outside air dry-bulb temperature of 8.33 C, outside air wet-bulb temperature of 6.11 C, heating coil entering air dry-bulb temperature of 21.11 C, heating coil entering air wet-bulb temperature of 15.55 C, and a heating coil air flow rate defined by field rated air flow volume below).

The value entered here must be greater than 0. The gross total heating capacity should not account for the effect of supply air fan heat.

Note: Manufacturers may calculate coil performance at different rating conditions to those used by EnergyPlus. For example, in Europe heating coil performance data is provided at the following conditions, in accordance with EN14511:

Indoor ambient temperature: 20°CDB / 15°CWB

Outdoor ambient temperature: 7°CDB / 6°CWB

Although the EN14511 Indoor Ambient Temp rated conditions of 20°CDB doesn't match exactly with the EnergyPlus rated condition of 21.11°CDB (for example), the difference in capacity should be minor and the manufacturer data can be used with confidence.

Heating capacity ratio modifier function of temperature curve

Select the curve that defines the heating capacity ratio modifier as a function of indoor dry-bulb temperature or indoor dry-bulb and outdoor wet-bulb temperatures. This is a [linear](#), [quadratic](#) or [cubic](#) performance curve if

the heating capacity is solely a function of indoor dry-bulb temperature (i.e., the indoor terminal units weighted average inlet air dry-bulb temperatures). Without specific manufacturers data indicating otherwise, the use of a single independent variable is recommended for this coil type. If, however, the user has reason to believe the heating capacity is both a function of indoor dry-bulb temperature and outdoor wet-bulb temperature (and has manufacturers data to create the performance curve), a [bi-quadratic](#) equation using weighted average indoor dry-bulb temperature and condenser entering air wet-bulb temperature as the independent variables may be used. See the [Engineering Reference](#) for more discussion on using this input field.

Note: The choice of using either outdoor dry-bulb temperature or outdoor wet-bulb temperature as the independent variable in this performance curve is set in the parent object [VRF Outdoor Unit](#).

Heating capacity ratio modifier function of flow fraction curve

Select the curve that defines that parameterises the variation of heating capacity as a function of the ratio of actual air flow rate across the heating coil to the rated air flow rate (i.e., fraction of full load flow). The curve must be a [linear](#), [quadratic](#) or [cubic](#) performance curve. The output of this curve is multiplied by the gross rated heating capacity and the heating capacity modifier curve (function of temperature) to give the gross heating capacity at the specific temperature and air flow conditions at which the coil is operating. The curve is normalized to have the value of 1.0 when the actual air flow rate equals the rated air flow rate.

Heating Coil - Water To Air Heat Pump



Coil:Heating:WaterToAirHeatPump:EquationFit

Coil:Heating:WaterToAirHeatPump:ParameterEstimation

Used in:

- Zone Water To Air Heat Pump
- Unitary Water To Air Heat Pump AHU

The Water to Air Heat Pump Heating Coil provides access to the EnergyPlus Equation fit (Curve fit) and Parameter Estimation models.

- **Equation Fit:** The Equation Fit Water to Air heat pump heating coil is a simple curve-fit model that requires coefficients generated from heat pump catalog data to model performance. The model resembles a black box with no use of heat transfer equations. A detailed description of the equation-fit model is available in the following reference: [Tang, C. C. 2005. Modeling Packaged Heat Pumps in Quasi-Steady State Energy Simulation Program. M.S. Thesis. Department of Mechanical and Aerospace Engineering, Oklahoma State University.](#) It is also downloadable from www.hvac.okstate.edu.
- **DesignBuilder v2026.1 and later only: Parameter Estimation:** The Parameter Estimation Water to Air heat pump Heating coil is a deterministic model that requires parameters to describe the operating conditions of the heat pump's components. The parameters are generated from the manufacturer catalog data using multi-variable optimisation method. The parameters required for WatertoAirHP heating coil are similar to WatertoAirHP cooling except the source side outside heat transfer coefficient parameter is not required for heating.

General

Name

A unique auto-assigned name identifying name for the coil. Any reference to this coil by another object (typically a [Zone Water To Air Heat Pump](#) or a [Unitary Water To Air Heat Pump AHU](#)) will use this name.

Coil type

Two types of coil are available:

- **1-Equation Fit** - a simple curve-fit model that requires coefficients generated from heat pump catalog data to model performance.
- **2-Parameter Estimation** - a deterministic model that requires parameters to describe the operating conditions of the heat pump's components. The parameters are generated from the manufacturer catalog data using multi-variable optimisation method.

Note: The **Coil type** selected here must match the setting on the paired [Water-to-Air Heat Pump Cooling Coil](#).

Equation Fit Coil Type Settings

The following settings are required when the Coil type is set to **1-Equation Fit**.

General

Rated air flow rate

This numeric field contains the rated volumetric air flow rate on the load side of the heat pump (in m³/s or ft³). This field is autosizable.

Rated water flow rate

This numeric field contains the rated volumetric water flow rate on the source side of the heat pump (in m³/s or gal/min). This field is autosizable.

Gross rated heating capacity

This numeric field contains the gross heating capacity of the heat pump at the rated condition (in W or Btu/hr). The gross heating capacity should not account for the effect of supply air fan heat. This field is autosizable.

Rated heating coefficient of performance (COP)

This numeric field contains the gross rated heating coefficient of performance of the heat pump. The gross rated COP of the heat pump should not account for the effect of supply air fan heat.

Ratio of rated heating capacity to rated cooling capacity

This numeric field contains the ratio of rated heating capacity to the rated cooling capacity. It is used to determine the autosized rated heating capacity from the rated cooling capacity. It is also used to determine the autosized rated cooling capacity when peak heating loads are dominating during sizing calculations. If both this object's and its associated cooling coil's capacity are provided by the user, this field not used.

Typical values for this field depend on the application in which the coils are used. Some suggested values are 1.23 for water loop applications, 0.89 for ground water applications, and 0.76 for ground loop applications (source: 2021 AHRI directory).

Temperatures

Rated entering water temperature

This setting contains the entering water temperature at which the coil capacity, COP and power are rated at. As such, the coil's COP should be entered at this temperature. The default value is 20°C (rating temperature for water loop water-to-air heat pumps in ISO-13256-1-1998).

Rated entering air dry-bulb temperature

This setting contains the entering air dry-bulb temperature at which the coil capacity, COP and power are rated at. As such, the coil's COP should be entered at this temperature. The default value is 20°C (rated temperature for water loop water-to-air heat pumps in ISO-13256-1-1998).

Performance Curves

Heating capacity curve

Select a [Quad Linear](#) Performance Curve for the heat pump capacity. The Single Speed Equation-Fit Model for Unitary Water-To-Air Heat Pump under the Air System Compound Component Groups in the Engineering Reference document contains more explanation regarding these coefficients.

Heating power consumption curve

Select a [Quad Linear](#) Performance Curve for the heat pump power consumption. The Single Speed Equation-Fit Model for Unitary Water-To-Air Heat Pump under the Air System Compound Component Groups in the Engineering Reference document contains more explanation regarding this.

Parameter Estimation Coil Type Settings

The following settings are required when the Coil type is set to **1-Parameter Estimation**. **DesignBuilder v2026.1 and later only**.

Cooling Coils

These cooling coil types are available within Detailed HVAC:

- [Cooling Coil - Water](#)
- [Cooling Coil - Single Speed DX](#)
- [Cooling Coil - Two Stage with Humidity Control DX](#)
- "Cooling Coil - VRF - DX" on page 1243
- "Cooling Coil - Water to Air Heat Pump" on page 1244

Cooling Coil - Water



Coil:Cooling:Water

Used in:

- **Air Handling Units,**
- **Demand loops of Dual duct systems**
- **Four-Pipe Air Distribution Units**

The water cooling coil provides detailed output with simplified inputs without the need for complex coil geometry definition. Instead the coil is sized in terms of auto-sizeable thermodynamic inputs such as temperatures, mass flow rates and humidity ratios.

Coil heat transfer (UA) values are calculated from the design conditions. A rough estimate of the coil area is provided along with percentage of surface wet and/or dry. This model uses the NTU-effectiveness approach to model heat transfer and has two types of flow arrangements cross-flow or counter-flow.

See the [Chilled Water Loop and FCU Tutorial](#)

See section “Cooling Coil Model” in the [EnergyPlus Engineering Document](#) for further details on this model.

General

Name

A unique name for an instance of a cooling coil which is predetermined by DesignBuilder.

Design water flow rate

The maximum possible water volume flow rate (m³/sec or gal/min) through the coil. This value is autosizable and the default is Autosize.

Design air flow rate

The maximum possible air volume flow rate (m³/sec or ft³/min) through the coil. This value is autosizable and the default is Autosize.

Design inlet water temperature

The inlet water temperature for the design flow (°C or °F). This value is autosizable and the default is Autosize.

Design inlet air temperature

The inlet air temperature for the design flow (°C or °F). This value is autosizable and the default is Autosize.

Design outlet air temperature

The outlet air condition desired for design flow (°C or °F). This value is autosizable and the default is Autosize.

Design inlet air humidity ratio

The highest value of humidity ratio possible for the Design inlet air stream. This value is autosizable and the default is Autosize.

Design outlet air humidity ratio

The value of humidity ratio for the Design outlet air stream. This value is autosizable and the default is Autosize.

Type of analysis

The coil has two modes of operation:

- **1-Simple analysis**
- **2-Detailed analysis**

The difference between the two modes being, the simple mode reports the value of surface area fraction wet of the coil as dry or wet. The detailed mode give the exact value, however the execution time in detailed mode is noticeably higher.

Heat exchanger configuration

The coil is operable in two modes:

- **1-Cross flow** for general A/C applications and

- **2- Counter flow** mode.

Air-conditioning systems generally use cross flow heat exchangers, hence the default is set to **1-Cross flow**.

Operation

Availability schedule

[Schedule](#) that defines when the coil is available, i.e. whether the coil can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit is off.

Controller

The controller allows the flow of water through the coil to be controlled based on temperature and/or humidity ratio typically downstream of the coil.

Control variable

The Control variable defines how the coil is controlled. Select one of:

- **1-Temperature** control is used for air temperature control and is normally specified for the coil's outlet air node
- **2-Humidity ratio** control is used for humidity control and would normally be specified for a dehumidifier outlet node
- **3-Temperature and humidity ratio** control is used for controlling both air temperature and high humidity levels, and is normally specified for a cooling coil's outlet air node.

Options 2 and 3 (the control variables involving humidity ratio) both have these requirements:

- Humidistat must be selected on the HVAC zone dialog and
- One of the "Maximum humidity" [setpoint managers](#) must be associated with the cooling coil.

More information is available in the [Humidity control](#) section.

Action

The Action determines how the controlled variable (mass flow rate through a water coil) is changed based on the control signal. In a coil where water mass flow rate is to be controlled, the mass flow rate through the coil is increased when more cooling is requested. In a cooling coil, an increase in water mass flow rate through the coil decreases the value of heat transfer from the water to the air stream (absolute value increases, but since cooling is traditionally described as a negative number, an increase in absolute value results in a decrease in the actual heat transfer value). Thus the cooling coil controller has **Reverse** action since an increase in flow rate results in a decrease in heat transfer.

Actuator variable

The actuator variable must currently be use the keyword **1-Flow** to control the water mass flow rate.

Controller convergence tolerance

The coil is controlled by knowing the outlet temperature and/or humidity ratio specified by the setpoint managers, and setting the outlet conditions from the coil to meet these setpoints. The chilled water coils use complex models that cannot be inverted directly. Therefore, to determine the correct mass flow rate for the cold water the models are inverted numerically using an iterative procedure. The iterative solution uses an interval-halving routine and needs a termination criteria that is set here.

The convergence tolerance is the maximum difference between the actual temperature at the setpoint node and the setpoint temperature. This control offset is typically set to a small temperature difference, such as 0.01°C.

This value is autosizable and the default is Autosize.

Maximum actuated flow

This is the maximum water flow (m3/sec or gal/min) through the coil. Set to the maximum design water flow for the coil.

This value is autosizable and the default is Autosize.

Minimum actuated flow

Set to the minimum design water flow (m3/sec or gal/min) for the water coil, normally a shut off valve that is set to zero.

Cooling Coil - Water Outputs

Following are the list of possible output report variables from this coil model:

```
HVAC,Sum,Total Water Cooling Coil Energy[J]
HVAC,Sum,Sensible Water Cooling Coil Energy[J]
HVAC,Average,Total Water Cooling Coil Rate[W]
HVAC,Average,Sensible Water Cooling Coil Rate[W]
HVAC,Average,Cooling Coil Area Wet Fraction
HVAC,Average,Cooling Coil Condensate Volumetric Flow Rate [m3/s]
Zone,Meter,OnSiteWater:Facility [m3]
Zone,Meter,OnSiteWater:HVAC [m3]
Zone,Meter,Condensate:OnSiteWater [m3]
HVAC,Sum,Cooling Coil Condensate Volume [m3]
```

Total Water Cooling Coil Energy (J)

Total Water Cooling Coil Energy is the total amount of heat transfer taking place in the coil at the operating conditions.

Sensible Water Cooling Coil Energy (J)

Sensible Water Cooling Coil Energy is the total amount of Sensible heat transfer taking place in the coil at the operating conditions. It only takes into account temperature difference in the inlet and outlet air streams at operating conditions.

Total Water Cooling Coil Rate (W)

Total Water Cooling Coil Rate is the Rate of heat transfer taking place in the coil at the operating conditions. The units are (J/sec) or Watts.

Sensible Water Cooling Coil Rate (W)

Sensible Water Cooling Coil Rate is the Rate of Sensible heat transfer taking place in the coil at the operating conditions.

Cooling Coil Area Wet Fraction

It defines the fraction of total surface area of coil which is wet due to moisture condensation on the surface of the coil. Value varies between 0.0 and 1.0. I

In addition, if a Water Storage Tank is used to collect coil condensate, then the following outputs will be available.

Cooling Coil Condensate Volumetric Flow Rate [m3/s]

Cooling Coil Condensate Volume [m3]

These reports provide the rate and amount of condensate from the coil. Condensate is water condensed out of the air as a result of cooling. The condensate volume is also reported on the meter for "OnSiteWater."

Single Speed DX Cooling Coil



Coil:Cooling:DX:SingleSpeed

Used in:

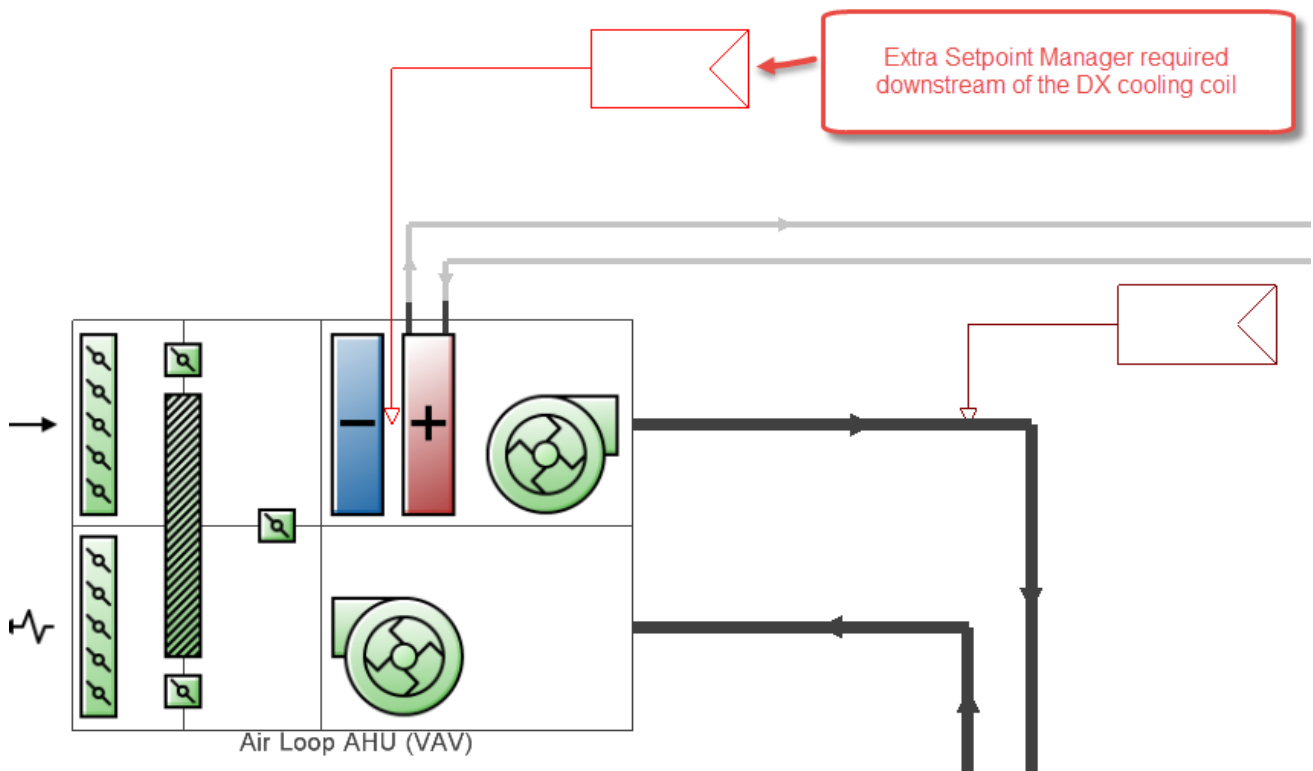
- PTAC,
- PTHP,
- Unitary Heat Cool AHU,
- Unitary Heat Pump AHU,
- Generic AHU

The DX single speed cooling coil model and data are quite different from that for the heating and cooling water coils. The simple water coils use an NTU-effectiveness heat exchanger model. The single speed DX coil model uses performance information at rated conditions along with curve fits for variations in total capacity, energy input ratio and part-load fraction to determine performance at part-load conditions. Sensible/latent capacity splits are determined by the rated sensible heat ratio (SHR) and the apparatus dewpoint/bypass factor (ADP/BF) approach. This approach is analogous to the NTU-effectiveness calculations used for sensible-only heat exchanger calculations, extended to a cooling and dehumidifying coil.

This DX cooling coil input requires an availability schedule, the rated total cooling capacity, the rated SHR, the rated COP, and the rated air volume flow rate. The latter 4 inputs determine the coil performance at the rating point (air entering the cooling coil at 26.7°C dry-bulb/19.4°C wet-bulb and air entering the outdoor condenser coil at 35°C dry-bulb/23.9°C wet-bulb). The rated air volume flow rate should be between .00004027 m³/s and .00006041 m³/s per watt of rated total cooling capacity (300 to 450 cfm/ton).

See section "DX Cooling Coil Model" in the [EnergyPlus Engineering Document](#) for further details regarding this model.

Important Note: When a Single speed DX cooling coil is placed in an AHU upstream from a heating coil, it is important to include an extra SPM immediately downstream of the cooling coil. The extra SPM should include all of the same settings as the SPM downstream of the AHU. See screenshot below. This use of a duplicate SPM in this way is the workaround recommended by the EnergyPlus support team for an issue which is expected to be fixed in versions of E+ after v8.5.



General

Name

A unique auto-assigned name for an instance of a DX cooling coil. Any reference to this DX coil by another object will use this name.

Gross rated total cooling capacity

The total, full load cooling capacity (sensible plus latent) (in W or Btu/h) of the DX coil unit at rated conditions. These are: air entering the cooling coil at 26.7°C dry-bulb/19.4°C wet-bulb, air entering the outdoor condenser coil at 35°C dry-bulb/23.9°C wet-bulb (The 23.9°C wet-bulb temperature condition is not applicable for air-cooled condensers which do not evaporate condensate), and a cooling coil air flow rate fan heat is NOT included. When used in a heat pump, the rated total cooling capacity should be within 20% of the rated total heating capacity, otherwise a warning message is issued.

Note: Manufacturers may calculate coil performance at different rating conditions to those used by EnergyPlus. For example, in Europe cooling coil performance data is provided at the following conditions, in accordance with EN14511:

Indoor ambient temperature: 27°CDB / 19°CWB
Outdoor ambient temperature: 35°CDB / 24°CWB

Although the EN14511 indoor ambient temperature rated conditions don't match exactly with the EnergyPlus rated conditions, the difference in capacity should be minor and the manufacturer data can be used with confidence.

Note: The **Gross rated total cooling capacity** is not the same as the net "rated capacity" value shown in manufacturer literature because it includes the fan heat per the AHRI standard. To calculate the gross value from the net manufacturer value use the formula:

Gross = Net + 773.3 * Airflow (m3/s) (SI units)

Gross rated sensible heat ratio

The sensible heat ratio (SHR=sensible capacity divided by total cooling capacity) of the DX cooling coil at rated conditions (air entering the cooling coil at 26.7°C dry-bulb/19.4°C wet-bulb, air entering the outdoor condenser coil at 35°C dry-bulb/23.9°C wet-bulb, and a cooling coil air flow rate defined by field "rated air volume flow rate" below). Both the sensible and total cooling capacities used to define the Rated SHR should be "gross" (i.e., supply air fan heat is NOT included).

Gross rated COP

The coefficient of performance (cooling power output in watts divided by electrical power input in watts) of the DX cooling coil unit at rated conditions (air entering the cooling coil at 26.7°C dry-bulb/19.4°C wet-bulb, air entering the outdoor condenser coil at 35°C dry-bulb/ 23.9°C wet-bulb, and a cooling coil air flow rate defined by field "rated air volume flow rate" below). The input power includes electric power for the compressor(s) and condenser fan(s) but does not include the power consumption of the supply air fan. The cooling power output is the value entered above in the field "Rated Total Cooling Capacity (gross)".

Rated air flow rate

The air volume flow rate (in m3/s or ft3/min), across the DX cooling coil at rated conditions. The rated air volume flow rate should be between 0.00004027 m3/s and 0.00006041 m3/s per watt of rated total cooling capacity (300 to 450 cfm/ton). The rated total cooling capacity, rated SHR and rated COP should be performance information for the unit with air entering the cooling coil at 26.7°C dry-bulb/19.4°C wet-bulb, air entering the outdoor condenser coil at 35°C dry-bulb/23.9°C wet-bulb, and the rated air volume flow rate defined here.

2017 Rated evaporator fan power per volume flow rate

This field is the electric power for the evaporator (cooling coil) fan per air volume flow rate through the coil at the rated conditions (in $W/(m^3/s)$ or $W/(ft^3/min)$). The default value is $773.3 W/(m^3/s)$ ($365 W/1000 cfm$). Values must be ≥ 0.0 and $\leq 1250 W/(m^3/s)$.

Note: This value is only used to calculate the following metrics according to the 2017 version of ANSI/AHRI 210-240 standard: Seasonal Energy Efficiency Ratio (SEER), Energy Efficiency Ratio (EER), Integrated Energy Efficiency Ratio (IEER) and the Standard Rating (Net) Cooling Capacity which will be outputs in the EnergyPlus eio file (see [EnergyPlus Engineering Reference](#), Single Speed DX Cooling Coil, Standard Ratings). This value is not used for modelling the evaporator (cooling coil) fan during simulations; instead, it is used for calculating SEER, EER, IEER and Standard Rating Cooling Capacity to assist the user in verifying their inputs for modelling this type of equipment. Note that two values are calculated for SEER. 'SEER User' is calculated using user-input PLF curve and cooling coefficient of degradation. 'SEER Standard' is calculated using AHRI Std 210/240-2017 default PLF curve and cooling coefficient of degradation.

2023 Rated evaporator fan power per volume flow rate

This field is the electric power for the evaporator (cooling coil) fan per air volume flow rate through the coil at the rated conditions in $W/(m^3/s)$. The default value is $934.4 W/(m^3/s)$ ($441 W/1000 cfm$). Values must be ≥ 0.0 and $\leq 1505 W/(m^3/s)$.

Note: This value is only used to calculate the following metrics according to the 2023 version of ANSI/AHRI 210-240 standard: Seasonal Energy Efficiency Ratio (SEER2), Energy Efficiency Ratio (EER2), Integrated Energy Efficiency Ratio (IEER2) and the Standard Rating (Net) Cooling Capacity which will be outputs in the EnergyPlus eio file (see [EnergyPlus Engineering Reference](#), Single Speed DX Cooling Coil, Standard Ratings). This value is not used for modelling the evaporator (cooling coil) fan during simulations; instead, it is used for calculating SEER2, EER2, IEER2 and Standard Rating Cooling Capacity to assist the user in verifying their inputs for modelling this type of equipment. Note that two values are calculated for SEER2. 'SEER2 User' is calculated using user-input PLF curve and cooling coefficient of degradation. 'SEER2 Standard' is calculated using AHRI Std 210/240-2017 default PLF curve and cooling coefficient of degradation.

Operation

Availability schedule

The name of the [schedule](#) that denotes whether the DX cooling coil can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off.

Performance Curves

The DX Cooling coil requires 5 curves be selected from the [Curves database](#) to define coil performance:

Total cooling capacity function of temperature curve

The [Bi-quadratic performance curve](#) that parameterises the variation of the total cooling capacity as a function of the wet-bulb temperature of the air entering the cooling coil, and the dry-bulb temperature of the air entering the air-cooled condenser coil (wet-bulb temperature if modelling an evaporative-cooled condenser). The output of this curve is multiplied by the rated total cooling capacity to give the total cooling capacity at specific temperature operating conditions (i.e., at temperatures different from the rating point temperatures). The curve is normalized to have the value of 1.0 at the rating point.

Total cooling capacity function of flow fraction curve

The [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of total cooling capacity as a function of the ratio of actual air flow rate across the cooling coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the rated total cooling capacity and the total cooling capacity modifier curve (function of temperature) to give the total cooling capacity at the specific temperature and air flow conditions at which the coil is operating. The curve is normalized to have the value of 1.0 when the actual air flow rate equals the rated air flow rate.

Energy input ratio function of temperature curve

The [Bi-quadratic performance curve](#) that parameterises the variation of the energy input ratio (EIR) as a function of the wet-bulb temperature of the air entering the cooling coil, and the dry-bulb temperature of the air entering the air-cooled condenser coil (wet-bulb temperature if modelling an evaporative-cooled condenser). The EIR is the inverse of the COP. The output of this curve is multiplied by the rated EIR (inverse of rated COP) to give the EIR at specific temperature operating conditions (i.e., at temperatures different from the rating point temperatures). The curve is normalized to a value of 1.0 at the rating point.

Energy input ratio function of flow fraction curve

The [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of the energy input ratio (EIR) as a function of the ratio of actual air flow rate across the cooling coil to the rated air flow rate (i.e., fraction of full load flow). The EIR is the inverse of the COP. The output of this curve is multiplied by the rated EIR and the EIR modifier curve (function of temperature) to give the EIR at the specific temperature and air flow conditions at which the cooling coil is operating. This curve is normalized to a value of 1.0 when the actual air flow rate equals the rated air flow rate.

Part load fraction correlation curve

The [Quadratic](#) or [Cubic](#) performance curve that parameterises the variation of electrical power input to the DX unit as a function of the part load ratio (PLR, sensible cooling load/steady-state sensible cooling capacity). The product of the rated EIR and EIR modifier curves is divided by the output of this curve to give the “effective” EIR for a given simulation timestep. The part load fraction (PLF) correlation accounts for efficiency losses due to compressor cycling.

The 2 important rules to be followed when setting up this curve are:

1. The part load fraction correlation curve coefficients need to be defined so that the output of the curve equals 1.0 when the PLR is 1.0, e.g.

$$PLF = 0.85 + 0.15(PLR)$$

So the associated quadratic would have Coefficient 1 = 0.85 and the Coefficient 2 = 0.15 with a maximum $x = 1.0$.

2. When $0.0 \leq PLR < 1.0$, the PLF must be ≥ 0.7 which is guaranteed as long as the Coefficient 1 of the PLF curve is > 0.7 . Also, in this PLR range, the PLF must be less than the PLR which is also guaranteed as long as the coil capacity is always greater than the demand. If the demand is greater than the coil capacity and PLF becomes greater than PLR, the software automatically limits the runtime fraction (PLR/PLF) to 1.0.

Latent Capacity Degradation

Latent capacity degradation

Check this option if the degradation of latent cooling capacity is to be modelled. In this case the following four input fields are shown allowing you to enter data relating to the degradation of latent cooling capacity when the supply air fan operates continuously while the cooling coil/compressor cycle on and off to meet the cooling load.

The fan operating mode is determined in the parent object and is considered to either be constant (e.g. AirLoopHVAC:UnitaryCoolOnly) or can be scheduled (e.g. AirLoopHVAC:UnitaryHeatCool). When scheduled,

the schedule value must be greater than 0 to calculate degradation of latent cooling capacity. At times when the parent object's supply air fan operating mode schedule is 0, latent degradation will be ignored. When modelling latent capacity degradation, these next four input fields must all have positive values.

Nominal time for condensate removal to begin

The nominal time (in seconds) after startup for condensate to begin leaving the coil's condensate drain line at the coil's rated airflow and temperature conditions, starting with a dry coil. Nominal time is equal to the ratio of the energy of the coil's maximum condensate holding capacity (J) to the coil's steady-state latent capacity (W). Suggested value is 1000; zero value means the latent degradation model is disabled. The default value for this field is zero. The supply air fan operating mode must be continuous (i.e., the supply air fan operating mode may be specified in other "parent" objects and is assumed continuous in some objects (e.g., AirloopHVAC:UnitaryCoolOnly) or can be scheduled in other objects [e.g., AirloopHVAC:UnitaryHeatCool]), and this field as well as the next three input fields for this object must have positive values in order to model latent capacity degradation.

Ratio of initial moisture evaporation rate and steady state latent capacity

The ratio of the initial moisture evaporation rate from the cooling coil (when the compressor first turns off (in W) and the coil's steady-state latent capacity (W) at rated airflow and temperature conditions. Suggested value is 1.5; zero value means the latent degradation model is disabled. The default value for this field is zero. The supply air fan operating mode must be continuous (i.e., the supply air fan operating mode may be specified in other "parent" objects and is assumed continuous in some objects (e.g., AirloopHVAC:UnitaryCoolOnly) or can be scheduled in other objects [e.g., AirloopHVAC:UnitaryHeatCool]); and this field, the previous field and the next two fields must have positive values in order to model latent capacity degradation.

Maximum cycling rate

The maximum on-off cycling rate for the compressor (cycles per hour), which occurs at 50% run time fraction. Suggested value is 3; zero value means latent degradation model is disabled. The default value for this field is zero. The supply air fan operating mode must be continuous (i.e., the supply air fan operating mode may be specified in other "parent" objects and is assumed continuous in some objects (e.g., AirloopHVAC:UnitaryCoolOnly) or can be scheduled in other objects [e.g., AirloopHVAC:UnitaryHeatCool]); and this field, the previous two fields and the next field must have positive values in order to model latent capacity degradation.

Latent capacity time constant

Time constant (in seconds) for the cooling coil's latent capacity to reach steady state after startup. Suggested value is 45. The supply air fan operating mode must be continuous (i.e., the supply air fan operating mode may be specified in other "parent" objects and is assumed continuous in some objects (e.g., AirloopHVAC:UnitaryCoolOnly) or can be scheduled in other objects [e.g., AirloopHVAC:UnitaryHeatCool]), and this field as well as the previous three input fields for this object must have positive values in order to model latent capacity degradation.

Condenser

Condenser type

The type of condenser used by the DX cooling coil. Valid choices for this input field are:

- **1-Air cooled**, or
- **2-Evaporatively cooled**.

The default Condenser type is 1-Air cooled.

Evaporative Condenser

The data below is required only for [Condenser type](#) = **Evaporatively cooled**.

Evaporative condenser effectiveness

The effectiveness of the evaporative condenser, which is used to determine the temperature of the air entering the outdoor condenser coil as follows:

$$T_{\text{cond inlet}} = T_{\text{wbo}} + (1 - \text{EvapCondEffectiveness}) (T_{\text{dbo}} - T_{\text{wbo}})$$

where:

$T_{\text{cond inlet}}$ = the temperature of the air entering the condenser coil (°C)

T_{wbo} = the wet-bulb temperature of the outdoor air (°C)

T_{dbo} = the dry-bulb temperature of the outdoor air (°C)

The resulting condenser inlet air temperature is used by the **Total Cooling Capacity Modifier Curve (function of temperature)** and the **Energy Input Ratio Modifier Curve (function of temperature)**. The default value for this field is 0.9, although valid entries can range from 0.0 to 1.0. This field is not used when **Condenser Type** = **Air Cooled**.

If the user wants to model an air-cooled condenser, they should simply specify an air cooled Condenser Type. In this case, the **Total Cooling Capacity Modifier Curve (function of temperature)** and the **Energy Input Ratio Modifier Curve (function of temperature)** input fields for this object should reference performance curves that are a function of outdoor dry-bulb temperature.

If you wish to model an evaporative-cooled condenser AND have performance curves that are a function of the wet-bulb temperature of air entering the condenser coil, then you should specify **Condenser Type** = **Evaporatively cooled** and the evaporative condenser effectiveness value should be entered as 1.0. In this case, the **Total Cooling Capacity Modifier Curve (function of temperature)** and the **Energy Input Ratio Modifier Curve (function of temperature)** input fields for this object should reference performance curves that are a function of the wet-bulb temperature of air entering the condenser coil.

If you wish to model an air-cooled condenser that has evaporative media placed in front of it to cool the air entering the condenser coil, then you should specify **Condenser Type** = **Evaporatively cooled**. You must also enter the appropriate evaporative effectiveness for the media. In this case, the **Total Cooling Capacity Modifier Curve (function of temperature)** and the **Energy Input Ratio Modifier Curve (function of temperature)** input fields for this object should reference performance curves that are a function of outdoor dry-bulb temperature. Be aware that the evaporative media will significantly reduce the dry-bulb temperature of the air entering the condenser coil, so the Total Cooling Capacity and EIR Modifier Curves must be valid for the expected range of dry-bulb temperatures that will be entering the condenser coil.

Evaporative condenser air flow rate

The air volume flow rate, (in m3/s or ft3/min), entering the evaporative condenser. This value is used to calculate the amount of water used to evaporatively cool the condenser inlet air. The minimum value for this field must be greater than zero, and this input field is autosizable (equivalent to 0.000144 m3/s per watt of rated total cooling capacity [850 cfm/ton]). This field is not shown for Air cooled condensers.

Evaporative condenser pump rated power consumption

The rated power of the evaporative condenser water pump (in W or Btu/h). This value is used to calculate the power required to pump the water used to evaporatively cool the condenser inlet air. The default value for this input field is zero, but it is autosizable (equivalent to 0.004266 W per watt [15 W/ton] of rated total cooling capacity). This field is not shown for Air cooled condensers.

Crankcase Heater

Crankcase heater capacity

This numeric field defines the crankcase heater capacity (in W). When the outdoor air dry-bulb temperature is below the value specified in the input field "Maximum Outdoor Dry-bulb Temperature for Crankcase Heater Operation" (described below), the crankcase heater is enabled during the time that the compressor is not running. If this cooling coil is used as part of an air-to-air heat pump (Ref. UnitarySystem:HeatPump:AirToAir or PackageTerminal:HeatPump:AirToAir), the crankcase heater defined for this DX cooling coil is ignored and

the crankcase heater power defined for the DX heating coil (Ref. Coil:Heating:DX:SingleSpeed) is enabled during the time that the compressor is not running for either heating or cooling. The value for this input field must be greater than or equal to 0, and the default value is 0. To simulate a DX cooling coil without a crankcase heater, enter a value of 0.

Maximum outdoor dry-bulb temperature for crankcase heater operation

This numeric field defines the outdoor air dry-bulb temperature (in °C or °F) above which the compressor's crankcase heater is disabled. The value for this input field must be greater than or equal to 0.0°C, and the default value is 10°C.

Basin Heater

Basin heater capacity

This numeric field contains the capacity of the DX coil's electric evaporative cooler basin heater in (W/K or Btu/h-F). This field only applies for [Condenser type](#) = **Evaporatively cooled**. This field is used in conjunction with the Basin Heater Setpoint Temperature described in the following field. The basin heater electric power is equal to this field multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the DX coil is off, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero.

Basin heater setpoint temperature

This numeric field contains the set point temperature (in °C or °F) for the basin heater described in the previous field. This field only applies for [Condenser type](#) = **Evaporatively cooled**. The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the DX coil is off. This set point temperature must be greater than or equal to 2°C.

Basin heater operating schedule

This alpha field contains the name of the basin heater operating schedule. This field only applies for [Condenser type](#) = **Evaporatively cooled**. The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin heater operates when scheduled on and the outdoor air dry-bulb temperature is below the set point temperature described in the previous field. Regardless of this schedule, the basin heater can only operate when the DX coil is off.

Run Settings

For DX cooling coils located in a unitary system the run settings can be controlled.

Run on sensible load only

Check this option if the coil will operate to meet a sensible load as determined by the inlet node dry-bulb temperature and the dry-bulb temperature setpoint on the control node. When this option is checked the unit will run if there is a sensible load, otherwise the unit will not run if there is only a sensible load.

Run on latent load only

Check this option if the coil will operate to meet a latent load as determined by the inlet node humidity ratio and the max humidity setpoint on the control node. When this option is checked the coil will operate if there is a latent load. If both sensible and latent loads exist, the system will operate to maintain the temperature set point. When only a latent load exists, the system will operate to meet the humidity ratio set point and requires the use of a heating coil and heating coil outlet air temperature set point manager downstream of this cooling coil to maintain the temperature set point. If this option is not checked then this coil will not run if there is only a latent load.

DOAS Settings

Use DX cooling coil for DOAS

This checkbox should be checked if the DX coil is to be placed in an AHU used for DOAS. In this case the [Rated air flow rate](#) to [Rated cooling capacity](#) ratio for 100% dedicated outdoor air (DOAS) application DX cooling coils should be between 0.00001677 (m³/s)/W (125 cfm/ton) and 0.00003355 (m³/s)/W (250 cfm/ton).

DX Cooling Coil Outputs

HVAC,Average,DX Coil Total Cooling Rate[W]
 HVAC,Sum,DX Coil Total Cooling Energy[J]
 HVAC,Average,DX Coil Sensible Cooling Rate[W]
 HVAC,Sum,DX Coil Sensible Cooling Energy[J]
 HVAC,Average,DX Coil Latent Cooling Rate[W]
 HVAC,Sum,DX Coil Latent Cooling Energy[J]
 HVAC,Average,DX Cooling Coil Electric Power[W]
 HVAC,Sum,DX Cooling Coil Electric Consumption[J]
 HVAC,Average,DX Cooling Coil Runtime Fraction
 If not part of UnitarySystem:HeatPump:AirToAir:
 HVAC,Average,DX Cooling Coil Crankcase Heater Power[W]
 HVAC,Sum,DX Cooling Coil Crankcase Heater Consumption[J]

Evaporative-cooled condenser:

HVAC,Average,DX Cooling Coil Condenser Inlet Temp[C]
 HVAC,Sum,DX Cooling Coil Evap Condenser Water Consumption[m3]
 HVAC,Average,DX Cooling Coil Evap Condenser Pump Electric Power[W]
 HVAC,Sum,DX Cooling Coil Evap Condenser Pump Electric Consumption[J]
 HVAC,Average,DX Cooling Coil Basin Heater Electric Power[W]
 HVAC,Sum,DX Cooling Coil Basin Heater Electric Consumption[J]

Additional variables for Coil:Cooling:DX:TwoStageWithHumidityControlMode only:

HVAC,Average,DX Cooling Coil Stage 2 Runtime Fraction
 HVAC,Average,DX Cooling Coil Dehumidification Mode
 HVAC,Average,DX Cooling Coil Condensate Volumetric Flow Rate [m3/s]
 Zone,Meter,Condensate:OnSiteWater [m3]

DX Coil Total Cooling Rate [W]

This field is the total (sensible and latent) cooling rate output of the DX coil in Watts. This is determined by the coil inlet and outlet air conditions and the air mass flow rate through the coil.

DX Coil Total Cooling Energy [J]

This is the total (sensible plus latent) cooling output of the DX coil in Joules over the timestep being reported. This is determined by the coil inlet and outlet air conditions and the air mass flow rate through the coil. This output is also added to a report meter with Resource Type = EnergyTransfer, End Use Key = CoolingCoils, Group Key = System (Ref. Report Meter).

DX Coil Sensible Cooling Rate [W]

This output is the moist air sensible cooling rate output of the DX coil in Watts. This is determined by the inlet and outlet air conditions and the air mass flow rate through the coil.

DX Coil Sensible Cooling Energy [J]

This is the moist air sensible cooling output of the DX coil in Joules for the timestep being reported. This is determined by the inlet and outlet air conditions and the air mass flow rate through the coil.

DX Coil Latent Cooling Rate [W]

This is the latent cooling rate output of the DX coil in Watts. This is determined by the inlet and outlet air conditions and the air mass flow rate through the coil.

DX Coil Latent Cooling Energy [J]

This is the latent cooling output of the DX coil in Joules for the timestep being reported. This is determined by the inlet and outlet air conditions and the air mass flow rate through the coil.

DX Cooling Coil Electric Power [W]

This output is the electricity consumption rate of the DX coil compressor and condenser fan(s) in Watts. This value is calculated for each HVAC system timestep, and the results are averaged for the timestep being reported.

DX Cooling Coil Electric Consumption [J]

This is the electricity consumption of the DX coil compressor and condenser fan(s) in Joules for the timestep being reported. This output is also added to a report meter with Resource Type = Electricity, End Use Key = Cooling, Group Key = System (Ref. Report Meter).

DX Cooling Coil Runtime Fraction

This is the runtime fraction of the DX coil compressor and condenser fan(s) for the timestep being reported.

DX Cooling Coil Crankcase Heater Power[W]

This is the average electricity consumption rate of the DX coil compressor's crankcase heater in Watts for the timestep being reported.

DX Cooling Coil Crankcase Heater Consumption[J]

This is the electricity consumption of the DX coil compressor's crankcase heater in Joules for the timestep being reported. This output is also added to a report meter with Resource Type = Electricity, End Use Key = Cooling, Group Key = System (ref. Report Meter).

Cooling Coil - Two Stage With Humidity Control DX



Coil:Cooling:DX:TwoStageWithHumidityControlMode
CoilPerformance:DX:Cooling

Used in:

- DX cooling coils in Air handling units when **Two stage with dehumidification** option used

The multimode DX coil is functionally equivalent to the [single-speed DX cooling coil](#) but with multiple performance modes available. It is capable of modelling two-stage DX units and units with an enhanced dehumidification mode such as coil bypass or subcool reheat. It can have up to 4 performance modes to accommodate a 2-stage 2-mode unit.

General

Name

A unique user-assigned name for an instance of a DX cooling coil. Any reference to this DX coil by another object will use this name. This name is fixed and cannot be changed by the user.

Number of capacity stages

The value for this input defines the number of capacity stages and must be 1 or 2. The default value is 1. Larger DX units often have two capacity stages, which are often two completely independent compressor/coil circuits with the evaporator coils arranged in parallel in the supply air stream. Two-stage operation affects cycling losses and latent degradation due to re-evaporation of moisture with continuous fan operation.

Enhanced dehumidification mode

Check this option if the DX coil provides enhanced dehumidification to increase dehumidification based on a humidistat signal.

Dehumidification control type

Defines the type of dehumidification control. The following options are valid for this field:

- **1-None** - meet sensible load only, no active dehumidification control
- **2-Multi mode** - activate enhanced dehumidification mode as needed and meet sensible load. This option is used to model DX equipment with a switchable option such as subcool reheat.
- **3-Cool reheat** - cool beyond the dry-bulb setpoint as required to meet the humidity setpoint.

The default is **1-None**. For all dehumidification controls, the max humidity setpoint on the control node is used. This must be set using a Zone humidistat and a downstream setpoint manager of type:

- **5-Multi zone maximum humidity average**, or,
- **7-Multi zone humidity maximum**, or,
- **9-Single zone humidity maximum**

When extra dehumidification is required, the equipment may not be able to meet the humidity setpoint if its full capacity is not adequate.

Operation

Availability schedule

The name of the [schedule](#) that denotes whether the DX cooling coil can run during a given time period. A schedule value greater than 0 (usually 1 is used) indicates that the coil can be on during a given time period. A value less than or equal to 0 (usually 0 is used) denotes that the coil must be off.

Normal Mode Stage 1 Coil Performance

The data under this header defines the DX coil performance for stage 1 operation without enhanced dehumidification (normal mode).

The data items for **Normal Mode Stage 1 Coil Performance** are described under [Cooling Coil - Single Speed DX](#) coils.

Normal Mode Stage 1+2 Coil Performance

The data under this header defines the DX coil performance for stage 1+2 operation (both stages active) without enhanced dehumidification (normal mode).

The data items for **Normal Mode Stage 1+2 Coil Performance** are described under [Cooling Coil - Single Speed DX](#) coils.

Dehumidification Mode 1 Stage 1 Coil Performance

The data under this header defines the DX coil performance for stage 1 operation with enhanced dehumidification active.

The data items for **Dehumidification Mode 1 Stage 1 Coil Performance** are described under [Cooling Coil - Single Speed DX](#) coils.

Dehumidification Mode 1 Stage 1+2 Coil Performance

The data under this header defines the DX coil performance for stage 1+2 operation (both stages active) with enhanced dehumidification active. An EnergyPlus **CoilPerformance:DX:Cooling** object is generated based on these inputs.

The data items for **Dehumidification Mode 1 Stage 1+2 Coil Performance** are described under [Cooling Coil - Single Speed DX](#) coils.

Crankcase Heater

Crankcase heater capacity

This numeric field defines the crankcase heater capacity (in W). When the outdoor air dry-bulb temperature is below the value specified in the input field "Maximum Outdoor Dry-bulb Temperature for Crankcase Heater Operation" (described below), the crankcase heater is enabled during the time that the compressor is not running. The value for this input field must be greater than or equal to 0, and the default value is 0. To simulate a DX cooling coil without a crankcase heater, enter a value of 0.

Maximum outdoor dry-bulb temperature for crankcase heater operation

This numeric field defines the outdoor air dry-bulb temperature (in °C or °F) above which the compressor's crankcase heater is disabled. The value for this input field must be greater than or equal to 0°C, and the default value is 10°C.

Basin Heater

Basin heater capacity

This numeric field contains the capacity of the DX coil's electric evaporative cooler basin heater (in W/K or Btu/h-°F). This field only applies for [Condenser type](#) = **Evaporatively cooled**. This field is used in conjunction with the Basin Heater Setpoint Temperature described in the following field. The basin heater electric power is equal to this field multiplied by the difference between the basin heater set point temperature and the outdoor dry-bulb temperature. The basin heater only operates when the DX coil is off, regardless of the basin heater schedule described below. The basin heater capacity must be greater than or equal to zero.

Basin heater setpoint temperature

This numeric field contains the set point temperature (in °C or °F) for the basin heater described in the previous field. This field only applies for [Condenser type](#) = **Evaporatively cooled**. The basin heater is active when the outdoor air dry-bulb temperature falls below this setpoint temperature, as long as the DX coil is off. This set point temperature must be greater than or equal to 2°C.

Basin heater operating schedule

This alpha field contains the name of the basin heater operating schedule. This field only applies for [Condenser type](#) = **Evaporatively cooled**. The basin heater operating schedule is assumed to be an on/off schedule and the heater is available to operate any time the schedule value is greater than 0. The basin heater operates when scheduled on and the outdoor air dry-bulb temperature is below the set point temperature described in the previous field. Regardless of this schedule, the basin heater can only operate when the DX coil is off.

Run Settings

For DX cooling coils located in a unitary system the run settings can be controlled.

Run on sensible load only

Check this option if the coil will operate to meet a sensible load as determined by the inlet node dry-bulb temperature and the dry-bulb temperature setpoint on the control node. When this option is checked the unit will run if there is a sensible load, otherwise if this option is unchecked then the unit will not run if there is only a sensible load.

Run on latent load only

Check this option if the coil will operate to meet a latent load as determined by the inlet node humidity ratio and the max humidity setpoint on the control node. When this option is checked the coil will operate if there is a latent load. If both sensible and latent loads exist, the system will operate to maintain the temperature set point. When only a latent load exists, the system will operate to meet the humidity ratio set point and requires the use of a heating coil and heating coil outlet air temperature set point manager downstream of this cooling coil to maintain the temperature set point. If this option is not checked then this coil will not run if there is only a latent load.

DX Cooling Coil Outputs

See Outputs section under [Cooling Coil - Single Speed DX](#) coils.

Cooling Coil - VRF - DX

The variable refrigerant flow (VRF) direct expansion (DX) cooling coil model is nearly identical to the [single-speed DX cooling coil](#) model. For this reason, an adaptation of the single-speed DX cooling coil model is used to model the variable-speed compression system used in VRF AC systems. The model inputs are quite similar to the input requirements for the single-speed DX cooling coil model, however, the location of a majority of the inputs have been moved to the [variable refrigerant flow air conditioner](#) object since multiple DX cooling coils will use the same DX compression system.

General

Name

A unique auto-assigned name for an instance of the VRF DX cooling coil. Any reference to this DX coil by another object will use this name.

Operation

Availability schedule

Select the DX cooling coil availability schedule. Schedule values of 0 mean that the DX cooling coil is off. A schedule value greater than 0 indicates that the coil can operate during the time period.

Airflow

Rated air flow rate

The air volume flow rate (in m³/s or ft³/s) across the DX cooling coil at rated conditions. The rated air volume flow rate should be between 0.00004027 m³/s and 0.00006041 m³/s per watt of rated total cooling capacity (300 to 450 cfm/ton). The gross rated total cooling capacity and gross rated SHR should be performance information for the unit with at this rated air volume flow rate.

Capacity

Gross rated total cooling capacity

This numeric field defines the gross rated total cooling capacity of the DX cooling coil (in W or btu/hr). The total cooling capacity should be a gross, i.e., the effect of supply air fan heat NOT accounted for.

Gross ratio sensible heat ratio

This numeric field defines the gross sensible heat ratio (sensible capacity divided by total cooling capacity) of the DX cooling coil at rated conditions. Both the sensible and total cooling capacities used to define the Rated SHR should be gross (i.e., the effect of supply air fan heat is NOT accounted for)

Cooling capacity ratio modifier function of temperature curve

Select the curve that defines the cooling capacity ratio modifier as a function of indoor wet-bulb temperature or indoor wet-bulb and outdoor dry-bulb temperatures. This is a [linear](#), [quadratic](#) or [cubic](#) performance curve if the cooling capacity is solely a function of indoor wet-bulb temperature (i.e., the indoor terminal units weighted average inlet air wet-bulb temperatures). Without specific manufacturers data indicating otherwise, the use of a single independent variable is recommended for this coil type. If, however, the user has reason to believe the cooling capacity is both a function of indoor wet-bulb temperature and outdoor dry-bulb temperature (and has manufacturers data to create the performance curve), a [bi-quadratic](#) equation using weighted average indoor wet-bulb temperature and condenser entering air dry-bulb temperature as the independent variables may be used. See the [Engineering Reference](#) for more discussion on using this input field.

Cooling capacity modifier curve function of flow fraction

Select the curve that defines that parameterises the variation of total cooling capacity as a function of the ratio of actual air flow rate across the cooling coil to the rated air flow rate (i.e., fraction of full load flow). The curve must be a [linear](#), [quadratic](#) or [cubic](#) performance curve. The output of this curve is multiplied by the gross rated total cooling capacity and the total cooling capacity modifier curve (function of temperature) to give the gross total cooling capacity at the specific temperature and air flow conditions at which the coil is operating. The curve is normalized to have the value of 1.0 when the actual air flow rate equals the rated air flow rate.

Cooling Coil - Water To Air Heat Pump



Used in:

Coil:Cooling:WaterToAirHeat
Pump:EquationFit

Coil:Cooling:WaterToAirHeatPump:ParameterEstimation

- Zone Water To Air Heat Pump
- Unitary Water To Air Heat Pump AHU

The Water to Air Heat Pump Cooling Coil provides access to the EnergyPlus Equation fit and Parameter Estimation models.

- **Equation Fit:** The Equation Fit Water to Air heat pump Cooling coil is a simple curve-fit model that requires coefficients generated from heat pump catalog data to model performance. The model resembles a black box with no use of heat transfer equations. A detailed description of the equation-fit model is available in the following reference: [Tang, C. C. 2005. Modeling Packaged Heat Pumps in Quasi-Steady State Energy Simulation Program. M.S. Thesis. Department of Mechanical and Aerospace Engineering, Oklahoma State University](#). It is also downloadable from www.hvac.okstate.edu.
- **DesignBuilder v2026.1 and later only: Parameter Estimation:** The Parameter Estimation Water to Air heat pump Cooling coil is a deterministic model that requires parameters to describe the operating conditions of the heat pump's components. The parameters are generated from the manufacturer catalog data using multi-variable optimisation method. The parameters required for WatertoAirHP

cooling coil are similar to WatertoAirHP heating except the source side outside heat transfer coefficient parameter is not required for heating.

General

Name

A unique auto-assigned name identifying name for the coil. Any reference to this coil by another object (typically a [Zone Water To Air Heat Pump](#) or a [Unitary Water To Air Heat Pump AHU](#)) will use this name.

Coil type

Two types of coil are available:

- **1-Equation Fit** - a simple curve-fit model that requires coefficients generated from heat pump catalog data to model performance.
- **2-Parameter Estimation** - a deterministic model that requires parameters to describe the operating conditions of the heat pump's components. The parameters are generated from the manufacturer catalog data using multi-variable optimisation method.

Note: The **Coil type** selected here must match the setting on the paired [Water-to-Air Heat Pump Heating Coil](#).

Rated air flow rate

This numeric field contains the rated volumetric air flow rate on the load side of the heat pump (in m3/s or ft3). This field is autosizable.

Rated water flow rate

This numeric field contains the rated volumetric water flow rate on the source side of the heat pump (in m3/s or gal/min). This field is autosizable.

Gross rated total cooling capacity

This numeric field contains the gross rated total cooling capacity of the heat pump (in W or Btu/hr). This field is autosizable. The gross rated total cooling capacity should be within 20% of the gross rated heating capacity, otherwise a warning message is issued. The gross rated total cooling capacity should not account for the effect of supply air fan heat.

Gross rated sensible cooling capacity

This numeric field contains the gross rated sensible capacity of the heat pump (in W or Btu/hr). This field is autosizable. The gross rated sensible cooling capacity should not account for the effect of supply air fan heat.

Rated cooling coefficient of performance

This numeric field contains the rated cooling coefficient of performance of the heat pump.

Temperatures

Rated entering water temperature

This numeric field contains the entering water temperature (in °C or °F) at which the coil capacity, COP and power are rated at. As such, the coil's COP should be entered at this temperature. The default value is 30°C (rating temperature for water loop water-to-air heat pumps in ISO-13256-1-1998).

Rated entering air dry-bulb temperature

This numeric field contains the entering air dry-bulb temperature (in °C or °F) at which the coil capacity, COP and power are rated at. As such, the coil's COP should be entered at this temperature. The default value is 27°C (rated temperature for water loop water-to-air heat pumps in ISO-13256-1-1998).

Rated entering air wet-bulb temperature

This numeric field contains the entering air wet-bulb temperature (in °C or °F) at which the coil capacity, COP and power are rated at. As such, the coil's COP should be entered at this temperature. The default value is 19°C (rated temperature for water loop water-to-air heat pumps in ISO-13256-1-1998).

Performance Curves

Cooling capacity curve

Select a [Quad Linear](#) Performance Curve for the heat pump total cooling capacity. The Single Speed Equation-Fit Model for Unitary Water-To-Air Heat Pump under the Air System Compound Component Groups in the Engineering Reference document includes an explanation of the curve coefficients.

Sensible cooling capacity curve

Select a [Quint Linear](#) Performance Curve for the heat pump power consumption. The Single Speed Equation-Fit Model for Unitary Water-To-Air Heat Pump under the Air System Compound Component Groups in the Engineering Reference document includes an explanation of the curve coefficients.

Cooling power consumption curve

Select a [Quad Linear](#) Performance Curve for the heat pump power consumption. The Single Speed Equation-Fit Model for Unitary Water-To-Air Heat Pump under the Air System Compound Component Groups in the Engineering Reference document includes an explanation of the curve coefficients.

Others

Nominal time for condensate removal to begin

This numeric field defines the nominal time (in seconds) after startup for condensate to begin leaving the coil's condensate drain line at the coil's rated airflow and temperature conditions, starting with a dry coil. Nominal time is equal to the ratio of the energy of the coil's maximum condensate holding capacity (J) to the coil's steady-state latent capacity (W). Suggested value is 1000. A zero value means the latent degradation model is disabled. The default value for this field is zero.

Ratio of initial moisture evaporation rate and steady state latent capacity

This numeric field defines ratio of the initial moisture evaporation rate from the cooling coil (when the compressor first turns off, in Watts) and the coil's steady-state latent capacity (Watts) at rated airflow and temperature conditions. Suggested value is 1.5. A zero value means the latent degradation model is disabled. The default value for this field is zero.

Parameter Estimation Coil Type Settings

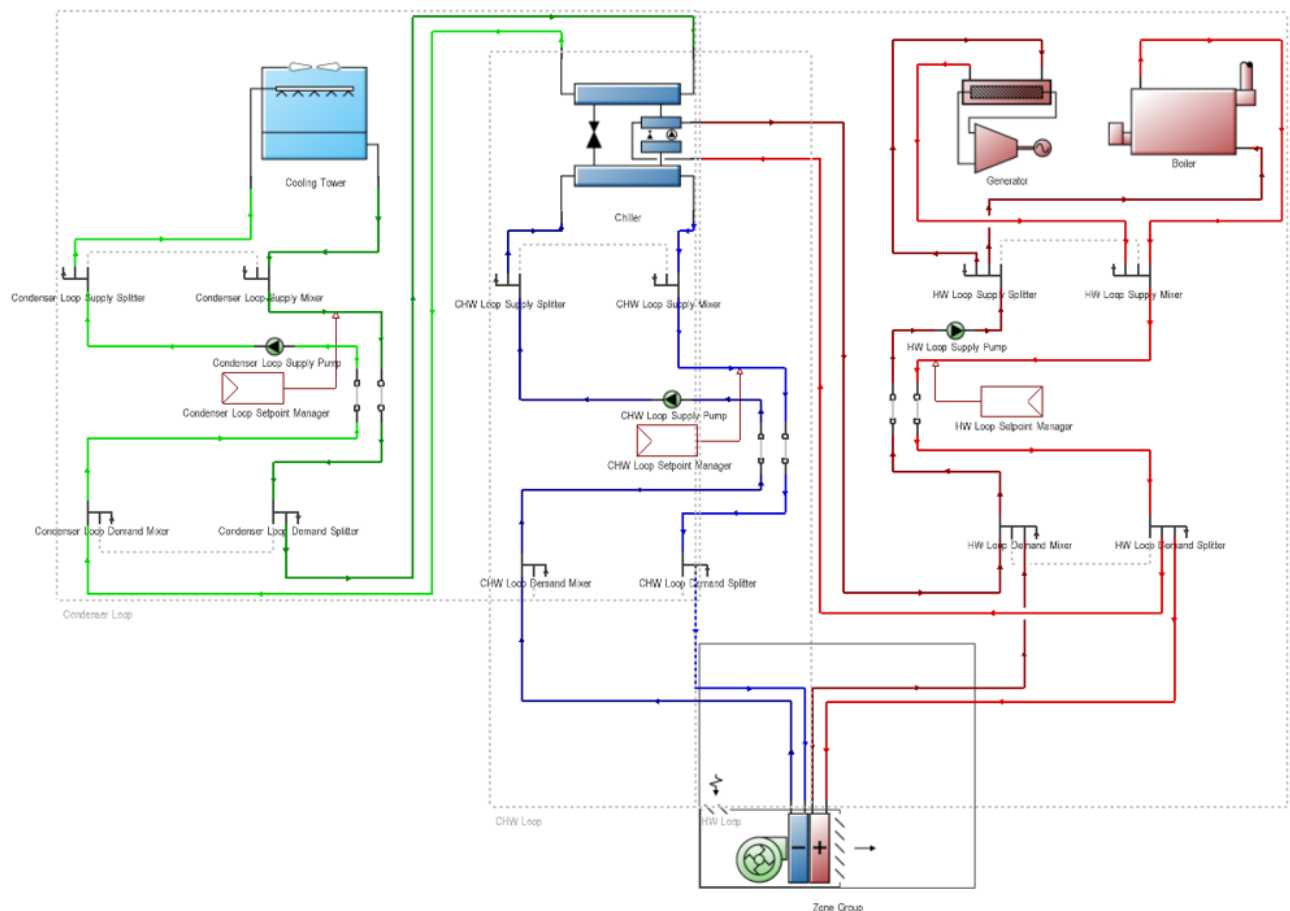
The following settings are required when the Coil type is set to **1-Parameter Estimation**. **DesignBuilder v2026.1 and later only**.

Generators

DesignBuilder generators enable the simulation of cogeneration systems, aka Combined Heat and Power (CHP), and also trigeneration systems, aka Combined Cooling, Heat and Power (CCHP).

Generators consume a fuel such as natural gas or diesel to turn an engine or turbine which drives an electric generator to produce electricity. Water is used to cool the generator and is heated to provide a useful source of hot water. In terms of energy conversion, chemical energy in the fuel is converted to mechanical energy in the engine or turbine, which is converted to electrical energy in the electric generator. The use of the generated electricity is coordinated during the simulation through use of one or more [Electric load centres](#). The system needs to be configured for cogeneration through connections to hot water or domestic hot water (DHW) plant loops in the detailed HVAC system. DesignBuilder currently supports the [Internal combustion engine](#) and [Micro turbine](#) generators.

Tip: to model a trigeneration system, where hot and chilled water and electricity are all generated, the generator hot water loop should be connected to an [absorption chiller](#). A Detailed HVAC template is included in the software to help get you started with modelling this system and the screenshot below illustrates the component layout.



- "Generator - Internal Combustion Engine" on page 1247
- "Generator - Micro Turbine" on page 1251
- "Generators - Modelling Guide" on page 1257

Generator - Internal Combustion Engine



Generator: Internal Combustion Engine

Used in:

- Hot water loop, supply side
- DHW loop, supply side

The Internal Combustion (IC) Engine generator uses a modified Otto cycle. This generator model uses the electrical load and rated engine generator size to compute part-load ratios (PLR). Fuel energy input and recoverable jacket and lube oil heat are then computed. Finally, the recoverable exhaust heat is calculated.

Manufacturer's curves must be obtained for IC engine generators to derive the equipment performance parameters that are specified in the quadratic curve fits. Note that simple transformation of the form of the manufacturer's curves may be required. Electric energy output/fuel energy input is related to the part-load ratio (i.e., electric/load generator capacity) with a polynomial. Recoverable jacket heat/fuel energy input equipment performance, recoverable lube oil heat/fuel energy input equipment performance, total exhaust heat/fuel energy input equipment performance, exhaust gas temperature/fuel energy input equipment performance parameters are all specified with a quadratic curve fit.

When the heat recovery curve fits = 0.0, this means that no heat recovery is done on the IC engine generator exhaust gas; and no heat recovery is specified.

General

Name

This alpha field contains the identifying name for the IC engine generator.

Type

Select the type of generator from the list:

- **1-Internal combustion engine** - as described on this page.
- [2-Micro turbine](#)

Operation

Availability schedule

Select the schedule that defines the times that the generator can operate.

Note: The way that this **Availability schedule** interacts with the Plant equipment operation, [Scheme schedule](#) for the attached Plant loop, depends on the Electric load centre, [Generator operation scheme type](#) setting as follows:

For the **1-Base load** scheme type the generator availability schedule *overrides* the Plant equipment operation scheme schedule, so the generator can run whenever this schedule has a value > 0 regardless of the scheme schedule value.

For the **5-Follow thermal** or **6-Follow thermal limit electrical** scheme types, both this **Availability schedule** and the Plant equipment operation scheme schedule must have a value > 0 for the generator to operate.

Power Output

Rated thermal to electric power ratio

This setting allows you to define the ratio of the rated thermal output to the rated electric output. This nominal ratio is only used for control and dispatch decisions and should be generally consistent with the more detailed

performance input in the rest of this dialog, but it is not used in the component model. It is only used if the operation scheme is set on the [Electric Load centre](#) dialog is set to **5-Follow thermal** or **6-Follow thermal limit electrical**.

Rated power output

This setting is used to define:

- The design nominal capacity of the generator (in W or Btu/hr) for the component simulation and,
- Nominal electric power output to be requested from the generator for supervisory control and generator dispatch for electric load centres. More specifically, it defines how much a generator is loaded (i.e. requested electric power output) and can also impact the operation of an [electric storage unit](#) if one is selected on the associated [Electric load centre](#). Search for "Electrical Storage" in the [EnergyPlus Engineering Reference](#) for more details on this input and its relationship with electric storage.

Shaft power curve

Select the shaft power [quadratic curve](#) which is used to calculate the electric energy output divided by the fuel energy consumption as a function of part-load ratio. The defining equation is:

$$ShaftPowerToFuelRatio = C_1 + C_2RL + C_3RL^2$$

where RL is the Ratio of load to rated power output.

Fuel

Fuel type

This alpha field determines the type of fuel that the generator uses. The default is **3-Diesel**. Valid values are:

- **1-Natural gas,**
- **2-Propane gas,**
- **3- Diesel,**
- **4-Gasoline,**
- **5-Fuel oil 1,**
- **6-Fuel oil 2,**
- **7-Other fuel 1,**
- **8-Other fuel 2.**

Fuel higher heating value

This numeric field contains the higher heating value of the fuel used (in kJ/kg or Btu/lb).

Part Load Ratios

Minimum part load ratio

This setting contains the minimum allowed operating fraction of full load. The input value must be ≥ 0.0 and ≤ 1.0 .

Maximum part load ratio

This setting contains the maximum allowed operating fraction of full load. The input value must be >0.0 and ≤ 1.0 .

Optimum part load ratio

This setting contains the optimal operating fraction of full load. This is the part load ratio that gives the optimal production of electric power with the least amount of fuel usage.

Exhaust

Design minimum exhaust temperature

This is the design engine stack saturated steam temperature (in °C or °F).

Maximum exhaust flow per unit of power output

Maximum exhaust flow per unit capacity for the generator. The parameter sets an upper limit on exhaust gas flow and exhaust gas heat recovery for the generator. Units are (kg/s)/W or (lbm/sec)/(Btu/hr).

Total exhaust energy curve

Select the Total exhaust energy [quadratic curve](#) which is a quadratic equation used to calculate the total exhaust heat as a function of part-load ratio. The defining equation is:

$$TotalExhaustToFuelRatio = C_1 + C_2RL + C_3RL^2$$

where RL is the Ratio of load to rated power output.

Exhaust temperature curve

Select the Exhaust temperature [quadratic curve](#) which is used to determine the exhaust gas temperature as a function of part-load ratio. The defining equation is:

$$AbsoluteExhaustTemperature = C_1 + C_2RL + C_3RL^2$$

where RL is the Ratio of load to rated power output.

Exhaust U-Factor Times Area Curve

The UA curve applies to the exhaust gas heat exchanger. The curve is an equation that determines the overall heat transfer coefficient for the exhaust gases with the stack. The heat transfer coefficient ultimately helps determine the exhaust stack temperature. The defining equation is:

$$UAToCapacityRatio = C_1GeneratorRatedPowerOutput^{C_2}$$

The following two fields contain the coefficients for the equation.

Coefficient 1 of U-Factor times area curve

This numeric field contains the first coefficient for the overall heat transfer coefficient curve.

Coefficient 2 of U-Factor times area curve

This numeric field contains the second (exponential) coefficient for the overall heat transfer coefficient curve.

Heat Recovery

Design heat recovery water flow rate

Design water volumetric flow rate through heat recovery loop (in m3/s or gal/min).

Heat recovery maximum temperature

This field sets the maximum temperature (in °C or °F) that this piece of equipment can produce for heat recovery, putting an upper on the recovered heat and limiting the maximum temperatures leaving the component. As temperatures in the loop approach the maximum temperature, the temperature difference between the entering water and the surfaces in the piece of equipment becomes smaller. For the given heat recovery flow rate and that temperature difference the amount of heat recovered will be reduced, and eventually there will be no heat recovered when the entering water temperature is equal to the maximum temperature specified by the user in this field. The reduced amount of heat recovered will diminish if the temperature of the loop approach is the maximum temperature, and this will show up in the reporting. This allows the user to set the availability or the quality of the heat recovered for usage in other parts of the system or to heat domestic hot water supply.

Jacket heat recovery curve

Select the Recovery jacket heat [quadratic curve](#) which is a quadratic equation that determines the ratio of recovery jacket heat to fuel energy. It is used to calculate the recoverable jacket heat as a function of part-load ratio. The defining equation is:

$$RecoveryJacketHeatToFuelRatio = C_1 + C_2RL + C_3RL^2$$

where RL is the Ratio of load to rated power output.

Lube heat recovery curve

Select the Recovery lube heat [quadratic curve](#) which is a quadratic equation that determines the ratio of recovery lube heat to fuel energy. It is used to calculate the recoverable lube oil heat as a function of part-load ratio. The defining equation is:

$$RecoveryLubeHeatToFuelRatio = C_1 + C_2RL + C_3RL^2$$

where RL is the Ratio of load to rated power output.

Generator - Micro Turbine



Generator:MicroTurbine

Used in:

- Hot water loop, supply side
- DHW loop, supply side

Micro Turbine generators are small combustion turbines that produce electricity on a relatively small scale (e.g., 25kW to 500kW). This model uses nominal performance at reference conditions along with several modifier curves to determine electrical power output and fuel use at non-reference conditions. Standby and ancillary power can also be taken into account. Energy recovery from exhaust air can be used to heat water. Similar to electrical power output, thermal power (heat recovery) output is calculated using nominal performance at reference conditions with modifier curves to account for variations at non-reference conditions.

General

Name

This alpha field specifies a unique user-defined name to identify this generator. This is a required input.

Type

Select the type of generator from the list:

- [1-Internal combustion engine](#)
- [2-Micro turbine](#) as described on this page

Reference elevation

This numeric field specifies the reference elevation relative to sea level (in m or ft). The value entered must be greater than or equal to -300.0m. The default value is 0.

Operation

Availability schedule

Select the schedule that defines the times that the generator can operate.

Note: The way that this **Availability schedule** interacts with the Plant equipment operation, [Scheme schedule](#) for the attached Plant loop, depends on the Electric load centre, [Generator operation scheme type](#) setting as follows:

For the **1-Base load** scheme type the generator availability schedule *overrides* the Plant equipment operation scheme schedule, so the generator can run whenever this schedule has a value > 0 regardless of the scheme schedule value.

For the **5-Follow thermal** or **6-Follow thermal limit electrical** scheme types, both this **Availability schedule** and the Plant equipment operation scheme schedule must have a value > 0 for the generator to operate.

Power Output

Rated power output

This setting is used to define:

- The design nominal capacity of the generator (in W or Btu/hr) for the component simulation and,
- Nominal electric power output to be requested from the generator for supervisory control and generator dispatch for electric load centres. More specifically, it defines how much a generator is loaded (i.e. requested electric power output) and can also impact the operation of an [electric storage unit](#) if one is selected on the associated [Electric load centre](#). Search for "Electrical Storage" in the [EnergyPlus Engineering Reference](#) for more details on this input and its relationship with electric storage.

Reference electrical power output

This numeric field specifies the full-load electrical power output of the generator (in W or Btu/h) at reference conditions. The reference conditions are defined via additional input fields for this object (see below). This is a required input, and the value entered in this field must be greater than zero.

Rated thermal to electric power ratio

This setting allows you to define the ratio of the rated thermal output to the rated electric output. This nominal ratio is only used for control and dispatch decisions and should be generally consistent with the more detailed performance input in the rest of this dialog, but it is not used in the component model. It is only used if the operation scheme is set on the [Electric Load centre](#) dialog is set to **5-Follow thermal** or **6-Follow thermal limit electrical**.

Minimum full load electrical power output

This numeric field specifies the minimum electrical power output (in W or Btu/h) at full-load conditions. The electrical power output of the generator is determined by multiplying the [Reference electrical power output](#) by the [Electrical power function of temperature and elevation modifier curve](#). If the result is less than the numeric

value specified in this input field, then the generator's electrical power output is reset to the minimum full-load value specified in this input field. The value entered in this field must be greater than or equal to zero.

Maximum full load electrical power output

This numeric field specifies the maximum electrical power output (in W or Btu/h) at full-load conditions. The electrical power output of the generator is determined by multiplying the [Reference electrical power output](#) by the [Electrical power function of temperature and elevation modifier curve](#). If the result is greater than the numeric value specified in this input field, then the generator's electrical power output is reset to the maximum full-load value specified in this input field. The value entered in this field must be greater than zero.

Electrical power function of temperature and elevation curve

Select a [bi-quadratic](#) performance curve that parameterises the variation of electrical power output as a function of the combustion air inlet temperature and elevation. The output of this curve is multiplied by the [Reference electrical power output](#) to give the full-load power output at a specific combustion air inlet temperature and elevation (i.e., at values different from the reference conditions). This curve should be normalized to have a value of 1.0 at the reference conditions, and the curve should be valid for the range of inlet air temperatures anticipated for the simulation period and for the actual elevation of the generator.

Power Consumption

Standby power

This setting specifies the standby electric power consumed by the generator (in W or Btu/h). The standby power is the electrical power consumed by the generator (e.g., air fans and controls) when the generator is available to operate but the generator electrical power output is zero (power output is not being requested by the electric load centre). The value specified for this field must be greater than or equal to zero. The default value is 0.

Ancillary power

This numeric field specifies the ancillary electric power consumed by the generator (in W or Btu/h). The ancillary power is the electrical power consumed by other associated equipment (e.g., external fuel pressurization pumps) when the generator is operating. Specify this input as 0.0 if the [Reference electrical power output](#) and the [Reference electrical efficiency using lower heating value](#) input fields and associated modifier curves reflect the net electrical power output from the generator (i.e., ancillary power already deducted from the generator's gross electrical power output). A value greater than zero indicates that this electrical power is consumed whenever the generator is operating and will be deducted from the generator's overall electrical power output (Generator Produced Electric Power). The value specified for this field must be greater than or equal to zero. The default value is 0.

Ancillary power function of fuel input curve

Select a [quadratic](#) performance curve that parameterises the variation of ancillary power as a function of the generator's input fuel mass flow rate (kg/s). The output of this curve is multiplied by the ancillary power to give the ancillary power at a specific fuel mass flow rate.

Fuel

Fuel type

Select the type of fuel used by the generator from the list:

- **1-Natural gas** (default)
- **2-Propane gas**

Fuel lower heating value

This numeric field specifies the lower heating value of the fuel used (in Btu/lb). The value specified for this field must be greater than zero but less than the specified **Fuel higher heating value** (below). The default value is 45,450 kJ/kg.

Fuel higher heating value

This numeric field specifies the higher heating value of the fuel used (in Btu/lb). The value specified for this field must be greater than zero and greater than the specified **Fuel lower heating value** (above). The default value is 50,000 kJ/kg.

Electrical Efficiency

Reference electrical efficiency using lower heating value

This numeric field contains the electrical efficiency of the generator at reference conditions, based on the lower heating value of the fuel. The electrical efficiency is the electric power output divided by the fuel energy consumption rate (LHV basis). The reference conditions are defined via additional input fields for this object (see below). This is a required input, and the value entered in this field must be greater than zero and less than or equal to 1.0.

Electrical efficiency function of temperature curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of electrical efficiency as a function of the combustion air inlet temperature. The output of this curve is multiplied by the [Reference electrical efficiency using lower heating value](#) to give the full-load electrical efficiency at specific combustion air inlet temperatures (i.e., at inlet air temperatures different from the [Reference combustion air inlet temperature](#)). This curve should be normalized to have a value of 1.0 at the [Reference combustion air inlet temperature](#), and the curve should be valid for the range of inlet air temperatures anticipated for the simulation period.

Electrical efficiency function of part load ratio curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of electrical efficiency as a function of the generator's part-load ratio (part-load ratio is the actual electrical power output divided by the full-load electrical power output at the current operating conditions). The output of this curve is multiplied by the [Reference electrical efficiency using lower heating value](#) and the output of the [Electrical Efficiency Function of Temperature curve](#) to give the electrical efficiency at specific part-load and combustion air inlet (temperature) conditions. This curve should be normalized to have a value of 1.0 when the generator's part-load ratio is 1.0, and the curve should be valid for the range of part-load ratios anticipated for the simulation period.

Combustion Air

Reference combustion air inlet temperature

This numeric field specifies the reference temperature for the combustion inlet air (in °C or °F). The default value is 15°C.

Reference combustion air inlet humidity ratio

This numeric field specifies the reference humidity ratio for the combustion inlet air in kgWater/kgDryAir. The value specified for this field must be greater than zero. The default value is 0.00638 (kgWater/kgDryAir).

Exhaust

Reference exhaust air mass flow rate

This numeric field is the reference exhaust air mass flow rate (in kg/s or lb/s). Entered values must be greater than zero.

Nominal exhaust air outlet temperature

This numeric field is the exhaust air outlet temperature at nominal (reference) conditions (in °C or °F).

Exhaust air flow rate function of temperature curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of exhaust air flow rate as a function of the combustion air inlet temperature. The output of this curve is multiplied by the [Reference exhaust air mass flow rate](#) to give the exhaust air mass flow rate at non-reference combustion air inlet temperatures. This curve should be normalized to have a value of 1.0 at the [Reference combustion air inlet temperature](#), and the curve should be valid for the range of inlet air temperatures anticipated for the simulation period.

Exhaust air flow rate function of part load ratio curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of exhaust air flow rate as a function of the generator's part-load ratio (part-load ratio is the actual electrical power output divided by the full-load electrical power output at the current operating conditions). The output of this curve is multiplied by the [Reference exhaust air mass flow rate](#) to give the exhaust air mass flow rate at specific part-load operating conditions. This curve should be normalized to have a value of 1.0 when the generator's part-load ratio is 1.0, and the curve should be valid for the range of part-load ratios anticipated for the simulation period.

Exhaust air temperature function of temperature curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of exhaust air outlet temperature as a function of the combustion air inlet temperature. The output of this curve is multiplied by the [Nominal exhaust air outlet temperature](#) to give the exhaust air temperature at non-reference combustion air inlet temperatures. This curve should be normalized to have a value of 1.0 at the [Reference combustion air inlet temperature](#), and the curve should be valid for the range of inlet air temperatures anticipated for the simulation period.

Exhaust air temperature function of part load ratio curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of exhaust air outlet temperature as a function of the generator's part-load ratio (part-load ratio is the actual electrical power output divided by the full-load electrical power output at the current operating conditions). The output of this curve is multiplied by the [Nominal exhaust air outlet temperature](#) to give the exhaust air temperature at specific part-load operating conditions. This curve should be normalized to have a value of 1.0 when the generator's part-load ratio is 1.0, and the curve should be valid for the range of part-load ratios anticipated for the simulation period.

Heat Recovery

Heat recovery water flow operating mode

This field is used to choose between different modes of controlling the mass flow rate of water being heated by energy recovered from exhaust air. There are two options available:

- **1-Plant control** which indicates that the heat recovery water flow rate through the generator is determined externally (by the wider balance of plant). In this case, the generator will request the Reference Heat Recovery Water Flow Rate whenever it operates but the actual flow rate may be limited by other plant components (e.g., pump).

- **2-Internal control** which indicates that the flow of water is controlled inside the generator based on current operating conditions. In this case the generator should generally include a bypass branch when connecting to the plant loop.

Reference heat recovery water flow rate

This numeric field is the reference heat recovery (volumetric) water flow rate (in m³/s or gal/min). Entered values must be greater than zero.

Reference inlet water temperature

This numeric field specifies the reference temperature for the inlet water to the heat recovery heat exchanger (in °C or °F).

Reference thermal efficiency using lower heat value

This numeric field specifies the thermal efficiency (heat recovery to water) at reference conditions, based on the lower heating value of the fuel. The thermal efficiency is the thermal power output (to water) divided by the fuel energy consumption rate (LHV basis). The reference conditions are defined via additional input fields for this object. This value must be from 0.0 to 1.0. The default value is 0.

Minimum heat recovery water flow rate

This numeric field specifies the minimum (volumetric) water flow rate through the heat recovery heat exchanger (in m³/s or gal/min). The minimum input value is 0.0. The default value is 0.

Maximum heat recovery water flow rate

This numeric field specifies the maximum (volumetric) water flow rate through the heat recovery heat exchanger (in m³/s or gal/min). The minimum and default input values for this field are 0.0. The maximum heat recovery water flow rate must be greater than or equal to the minimum heat recovery water flow rate.

Maximum heat recovery water temperature

This field sets the maximum water temperature (in °C or °F) that this generator can produce via heat recovery. This temperature limit puts an upper bound on the recovered heat and limits the max temperatures leaving the component. As temperatures in the water loop approach this maximum temperature, the temperature difference between the entering water and the surfaces in generator's heat recovery heat exchanger becomes smaller. For the given heat recovery flow rate and that temperature difference the amount of heat recovered will be reduced, and eventually there will be no heat recovered when the entering water temperature is equal to the maximum temperature specified by the user in this field. The amount of heat recovered will diminish if the inlet water temperature approaches the maximum temperature, and this will show up in the reporting.

Thermal efficiency function of temperature and elevation curve

Select a [bi-quadratic](#) performance curve that parameterises the variation of thermal efficiency as a function of the combustion air inlet temperature and elevation. The output of this curve is multiplied by the [Reference thermal efficiency using lower heating value](#) to give the full-load thermal efficiency at a specific combustion air inlet temperature and elevation (i.e., at values different from the reference conditions). This curve should be normalized to have a value of 1.0 at the reference conditions, and the curve should be valid for the range of inlet air temperatures anticipated for the simulation period and for the actual elevation of the generator.

Heat recovery rate function of part load ratio curve

Select a [quadratic](#) or [cubic](#) performance curve that parameterises the variation of heat recovery to water (thermal power output) as a function of the generator's part-load ratio (part-load ratio is the actual electrical power output divided by the full-load electrical power output at the current operating conditions). The output of this curve is multiplied by the steady-state heat recovery at the current combustion inlet air temperature and elevation to give the heat recovery rate (thermal power output) at specific part-load operating conditions. This curve should be normalized to have a value of 1.0 when the generator's part-load ratio is 1.0, and the curve should be valid for the range of part-load ratios anticipated for the simulation period.

Heat recovery rate function of inlet water temperature curve

Select a [quadratic](#) performance curve that parameterises the variation of heat recovery to water (thermal power output) as a function of the inlet water temperature. The output of this curve is multiplied by the steady-state heat recovery at the current combustion inlet air temperature and elevation to give the heat recovery rate (thermal power output) at non-reference inlet water conditions. This curve should be normalized to have a value of 1.0 at the [Reference inlet water temperature](#), and the curve should be valid for the range of inlet water temperatures anticipated for the simulation period.

Heat recovery rate function of water flow rate curve

Select a [quadratic](#) performance curve that parameterises the variation of heat recovery to water (thermal power output) as a function of the heat recovery water flow rate. The output of this curve is multiplied by the steady-state heat recovery at the current combustion inlet air temperature and elevation to give the heat recovery rate (thermal power output) at non-reference heat recovery water flow rates. This curve should be normalized to have a value of 1.0 at the [Reference heat recovery water flow rate](#), and the curve should be valid for the range of water flow rates anticipated for the simulation period.

Heat recovery water flow rate function of temperature and power curve

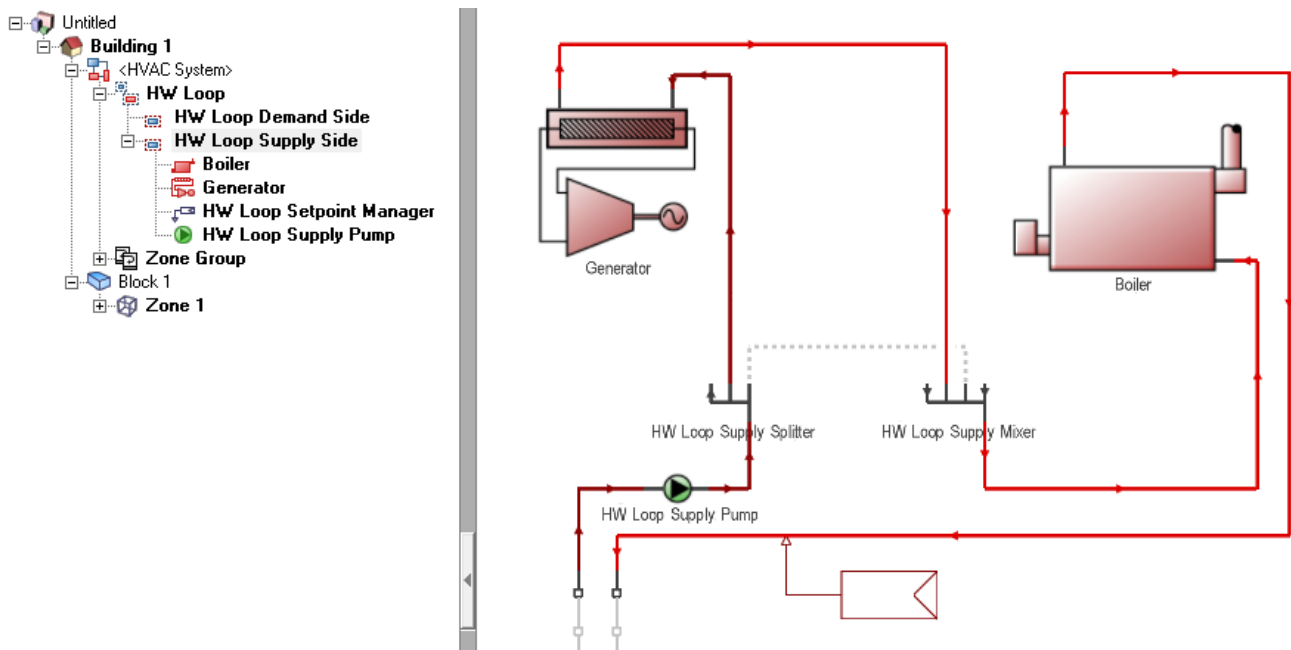
Select a [bi-quadratic](#) performance curve that parameterises the variation of heat recovery water flow rate as a function of the inlet water temperature and net electrical power output. This data is only used if the [Heat recovery water flow operating mode](#) is **2-Internal control**. The output of this curve is multiplied by the [Reference heat recovery water flow rate](#) to give the water flow rate at the specific inlet water temperature and net power operating conditions. This curve should be normalized to have a value of 1.0 at the reference conditions, and the curve should be valid for the range of inlet water temperatures and net electrical power output anticipated for the simulation period.

Generators - Modelling Guide

This modelling guide summarises the main steps involved in setting up [generators](#) as part of a co-generation (electricity + heat generation) system. You can read more details about the individual components on other pages.

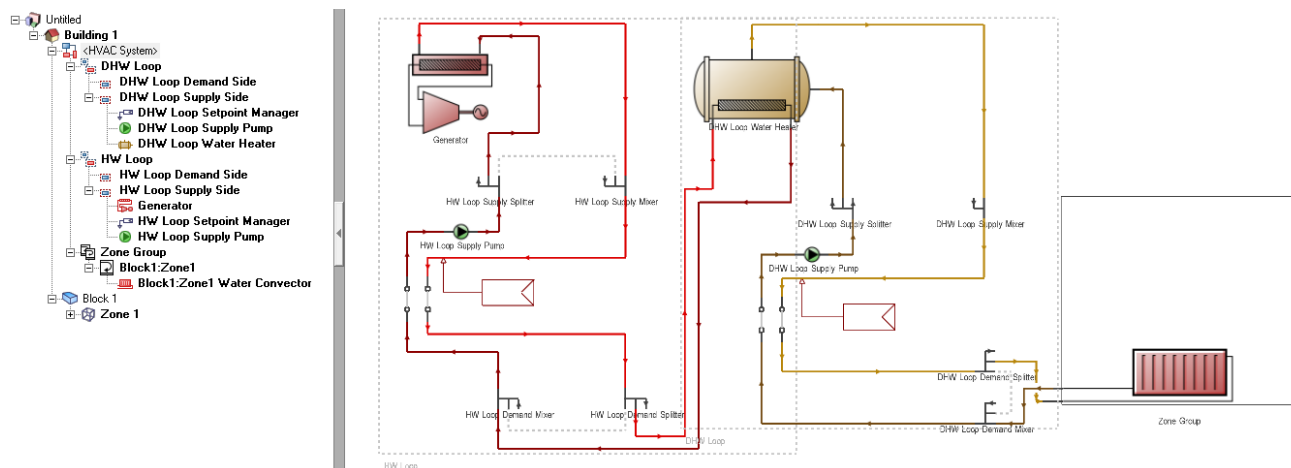
1. Add and configure generators in a detailed HVAC model, typically in hot water or DHW supply sub-loops or condenser demand sub loops (see screenshots below).

Note: To model a generator that works alongside a boiler as a backup heating source you should use the series configuration where the boiler is placed on the supply sub-loop outlet branch instead of the usual branch that connects between the splitter and the mixer (see screenshot below). **The parallel arrangement does not work as expected in EnergyPlus v8.9 and is not recommended.**



Generator in a HW loop with a backup boiler

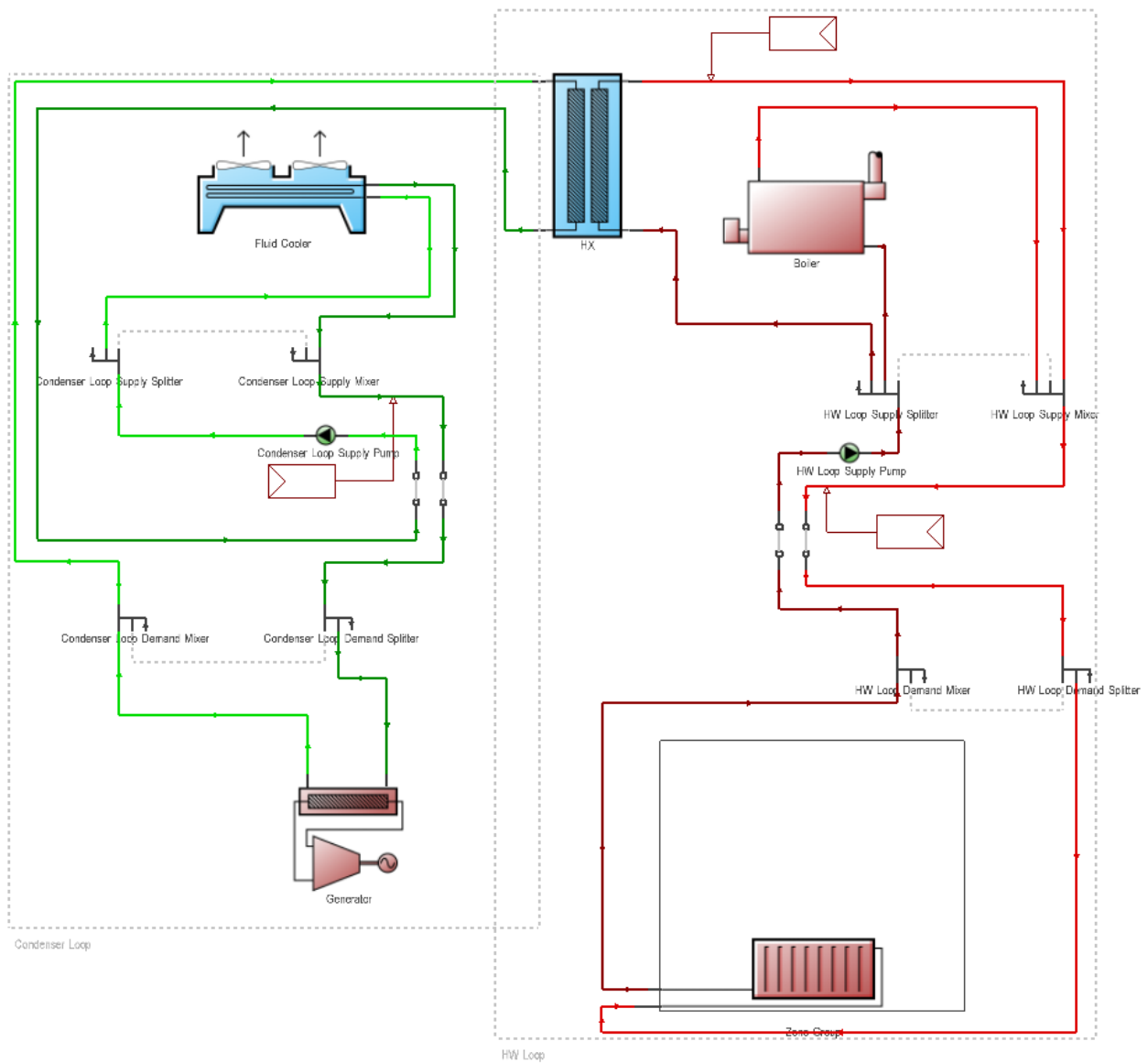
The arrangement of A DHW loop hot water tank heated by a hot water loop with generator(s) integrated and acting as a heating source is another tried and tested system (see below). The hot water tank has an internal built-in gas (or electric) boiler that provides backup heating to top up the building heat demand if the heat delivered from the generator(s) is not sufficient to meet the heating load.



Generator acts as a heating source to a hot water tank

Tip: Both of the systems described above can be loaded from DesignBuilder Detailed HVAC templates.

An alternative arrangement is where the generator is placed on the demand sub-loop of a condenser loop and the condenser loop uses a fluid cooler for heat rejection. The recovered heat from the generator is supplied to a hot water loop through a fluid-to-fluid heat exchanger (HX) that is in parallel to a boiler. The HX is set at higher priority than boiler in hot water loop equipment operation scheme.



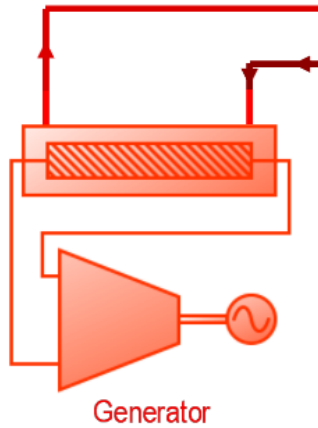
Note: In the above case, the operation scheme for the electric load centre that contains this generator should be set to **1-Base load** to allow the generator to operate and for waste heat to be recovered. The requirement to use the **1-Base load** scheme applies because the generator located in the demand sub-loop cannot be included in the plant equipment definition, and therefore cannot obtain the heat demand signal from the heating zone. If the heat exchanger reaches operating conditions, i.e., there is available heat that can be transferred from the demand side to the supply side, and also the heating zone has a heat demand, it will operate.

2. After adding generators into HVAC loops, make sure to enable/include them by checking the corresponding checkbox on the [Plant equipment operation](#) tab of the plant loop dialog (screenshot below).

General	
Number of schemes	1
Scheme 1	
Operation type	3-Heating load
Operation schedule	On 24/7
Number of ranges	1
Range 1	
Lower limit (W)	0.00
Upper limit (W)	1000000000000000.00
Equipment	
☒ Generator	
Priority	1
☒ Boiler	
Priority	2

Enable generators in plant equipment operation equipment list

3. The generator type, one of [1-Internal combustion engine](#) or [2-Micro turbine](#), can be selected from the generator dialog (screenshot below) along with the other generator attributes.

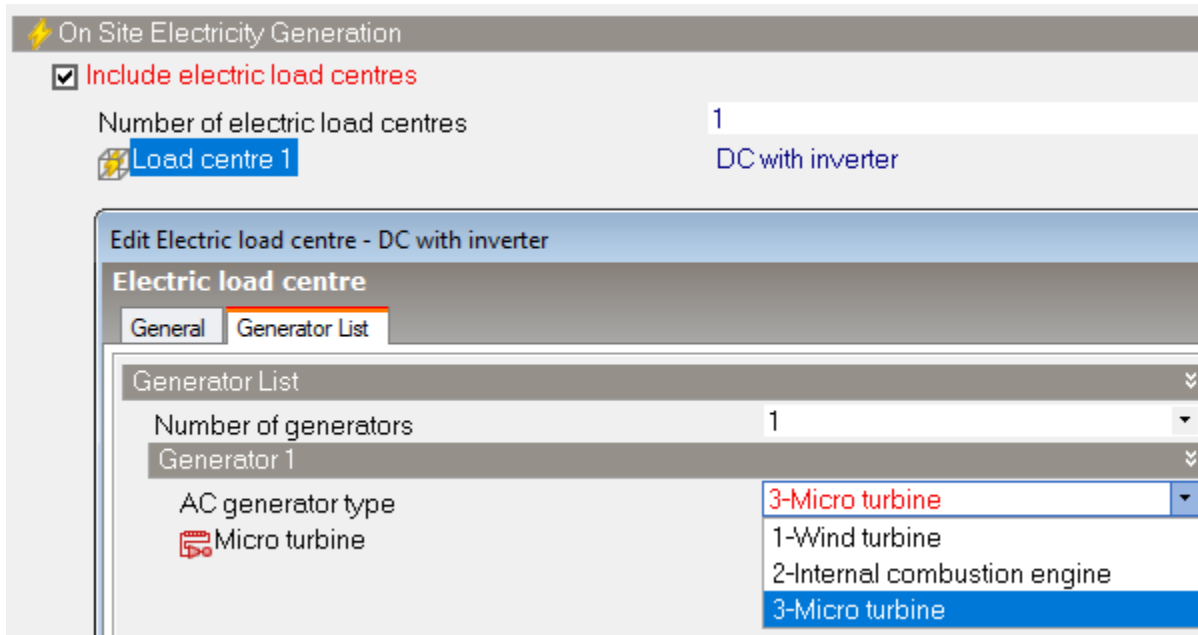


Generator

Generator	
General	
Name	Generator
Type	2-Micro turbine
Reference elevation (ft)	1-Internal combustion engine
Power Output	2-Micro turbine
Rated power output (Btu/h)	17060.71
Reference electrical power output (Btu/h)	17401.92
Rated thermal to electric power ratio	0.500
Minimum full load electrical power output (Btu/h)	0.00
Maximum full load electrical power output (Btu/h)	17401.92
☒ Electrical power function of temperature and elevation curve	Power_vs_Temp_Elev
Power Consumption	
Standby power (Btu/h)	0.00
Ancillary power (Btu/h)	0.00
☒ Ancillary power function of fuel input curve	AncillaryPowerFuelInput

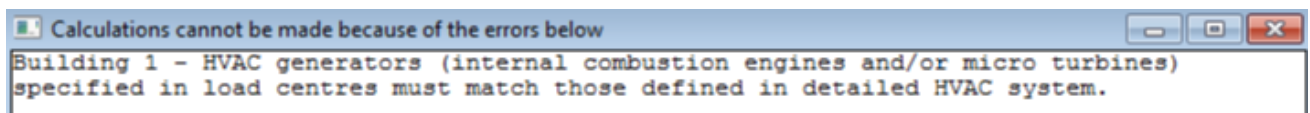
Generator type selection

4. Ensure that the **Include electric load centres** checkbox is checked at building level (on Generation tab) and enter the number of electric load centres for the model, which *must* contain all the HVAC generators modelled in the HVAC system. The selection can be done on the [Generator List](#) tab on the Electric load centre dialog when one of the alternating current [Electrical buss type](#) is selected (screenshot below). Both type and number of generators defined in electric load centres should match those modelled in the detailed HVAC system, and there should not be any duplicated entries. In other words each generator defined in the Detailed HVAC system must be selected exactly once on a load centre generator list.



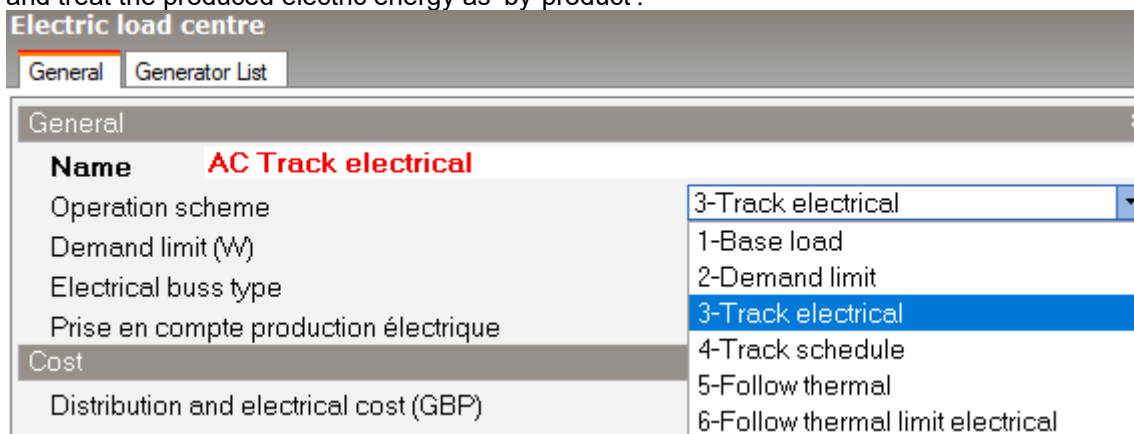
Define generators list on electric load centres

5. If the generators list defined in electric load centres does not match the number and type of generators modelled in the Detailed HVAC system, then the following error message will be displayed when attempting to run a simulation:



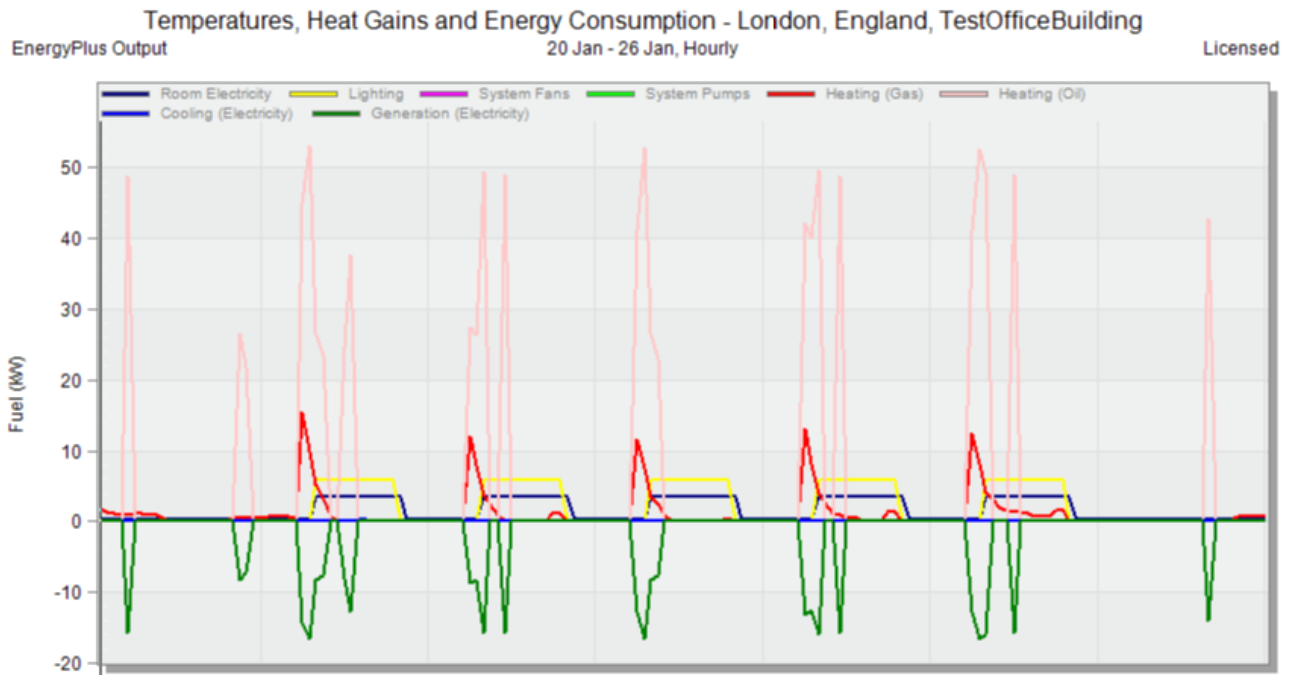
Error message indicating the mismatched generators list

6. [Operation schemes](#) are selected on the General tab of the electric load centre dialog. The commonly used operation schemes for generators are: **1-Base load**, **5-Follow thermal** or **6-Follow thermal limit electrical**. The **1-base load** scheme operates the generators at their rated electric power output whenever the generator is scheduled on, and treats the produced thermal energy as 'by-product'. In contrast, the **5-Follow thermal** and **6-Follow thermal limit electrical** schemes operate the generators to meet building's thermal demand, and treat the produced electric energy as 'by-product'.



Electric load centre operation schemes

7. After the simulation, the fuel consumption breakdown and electricity generation can be displayed at building level (screenshot below).



Fuel consumption and electricity generation reporting

Model Options



All Model Options are stored in the DesignBuilder dsb file and so any changes you make to the model configuration will not affect other models.

You can open the Model Options dialog by:

- Clicking on the spanner icon on the toolbar (above).
- From the [Edit](#) menu.
- Pressing F11.

All Model Options are stored in a single dialog but the options you can access depend on where you are in the [model hierarchy](#).

Some model options are stored at the **site level**:

- Options on the [Display](#) and [Drawing tools](#) tabs

Most model options are stored at the **building level**:

- Options on the [Data](#), [Advanced](#), [Heating design](#), [Cooling design](#), [Simulation](#), [Project details](#) and [Carbon](#) tabs

and some are stored at **block level**:

- Options on the [Block](#) tab.

So if you open the Model Options dialog when you are at building level you will not be able to access the Block Model Options. Likewise if you are at site level you will only be able to access the Display and Drawing tools tabs.

See also the [Program and Model Options](#) tutorial

Data - Model Options (Building)

Data tab on [Model Options](#) dialog

Use this dialog to customise the Model Data Options for the current building.

Model Options - Building and Block

Model Options

Data | Advanced | Heating Design | Cooling Design | Simulation | Display | Drawing tools | Block | Project details | Carbon

Construction and Glazing Data

Construction and glazing data

Pre-design | General

Multi-state electrochromic control method | 1-Sensor groups

Gains Data

Gains data

Lumped | Early | Detailed

Gain definition level | 1-Globally in model options

Occupancy method | 1-Occupancy density

Occupancy latent gains | 1-Dynamic calculation

Equipment gain units | 1-Power density

Lighting gain units | 2-Normalised power density

Early gains

Internal gains are separated into various categories (e.g. occupancy, lighting, computing etc.)

Timing

Timing

Typical workday | Schedules

☐ Internal gains operate with occupancy

Schedules

Timing is defined using the schedules and profiles mechanism which allows each day of the week to have a different profile.

HVAC

HVAC

Simple | Detailed

HVAC sizing | 3-Autosize

Simple HVAC autosize method | 1-EnergyPlus

☒ Specify Simple/Design HVAC details

Auxiliary energy calculations | 2-Separate fans and pumps

Mechanical ventilation method | 2-Ideal loads

Simple HVAC

HVAC systems are modelled using Ideal Loads, fuel consumption is calculated from loads using seasonal efficiencies

Natural Ventilation and Infiltration

Natural ventilation

Scheduled | Calculated

Infiltration units | 1-ac/h

Scheduled ventilation

Ventilation is defined as an air-change rate modified by an operation schedule and controlled using a set-point temperature.

CFD

Imported Surfaces

☒ Shade surfaces

You can edit the amount of detail in various aspects of the model:

- [Construction and Glazing](#)
- [Gains Data](#)
- [Timing](#)
- [HVAC](#)

- [Natural ventilation and Infiltration](#)
- [Ref974684535](#)"Imported Surfaces" on page 1274

The descriptions to the right of the sliders, together with the various drop list selections, provide a summary of the current model options.

Construction And Glazing Model Option

[Data tab on Model Options dialog.](#)

Construction and glazing

The **Construction and glazing** model option controls the way [Construction templates](#) are loaded on the [Construction](#) model data tab. There are 2 options:

- **Pre-design** which allows you to load a limited set of "Early Design" construction templates using the simple [Insulation and Thermal mass slider controls](#) based on 4 x insulation options and 3 x thermal mass options.
- **General** where construction templates are loaded by [selection](#) from a Construction templates browse list in the usual way. **General** is the default setting.

Tip: You can read a summary of how Construction templates are set up based on the [Region](#) and [Energy code](#) on the "Legislative Region Templates - Standard Insulation" on page 1510 help page.

Multi-state electrochromic control method

There are 2 ways to set up sensors to control the tinting of Multi-state electrochromic systems. These 2 methods are accessed through the options:

- **1-Sensor groups** the default option where a single sensor sensing exterior incident solar radiation can be used to control multiple windows. In this case the signal from a single sensor window is used to control all windows with the associated SageGlass® window shading system applied.
- **2-Individual sensors** where each window has its own built in sensor and there is no need to define control connections between windows and sensors.

Gains Data Model Option

[Data tab on Model Options dialog.](#)

Gains data

There are three levels of gains model detail:

- **Lumped** - all internal gains including occupancy, computer, office equipment, miscellaneous, catering process and lighting gains are lumped together into a single value. Solar gains are still calculated in the usual way. Note that with **Lumped** gains you cannot generate [Temperature distribution](#) and [Comfort](#) simulation output data.

Note: When using the **Lumped** Gains model option, [Comfort](#) data is not generated because in EnergyPlus the request for comfort calculations is made in the **People** statement and this is not generated by DesignBuilder when **Lumped** gains is set.

- **Early** - gains can be defined separately under various categories. This is the default and is normally the preferred option for design and energy assessment work.
- **Detailed** - gains are specified by defining each individual item of equipment in each zone on the [Equipment tab](#). If you select this option then DesignBuilder will automatically set the [Timing model option](#) to **Schedules**. Note that this option was developed for creating very detailed models of existing buildings based on survey data. Entering this data for a large building is very time-consuming and in most cases you will be better served by the **Early** gains option above.

Important Note: The **Detailed** gains option is included for legacy models only and may be discontinued in future versions of the software.

Gain definition level

This option allows you to choose whether the gain definition methods are defined globally here on the model options dialog or zone by zone in model data. Select from:

- **1-Globally in model options** where a single set of settings is applied globally across the model. In this case the settings below for Occupancy method, Occupancy latent gains, Equipment gain units and Lighting gain units can be selected on the model options dialog as described below.
- **2-Zone by zone in model data** where the definition methods for Occupancy method, Occupancy latent gains, Equipment gain units and Lighting gain units are set in model data and can be different for each zone.

Occupancy method

When the [Gain definition level](#) option is set to **1-Globally in model options**, select the occupancy definition method from the following options:

- **1-People/area** - the default option where occupancy is defined as occupancy density (in people / m2 or people / ft2).
- **2-Area per person** - is sometimes preferred as an inverse occupancy density. Note that when switching between this option and occupancy density the definitions on the Activity tab zone level are translated accordingly. This is not the case when the number of people option below is selected.
- **3-Number of people** - used when detailed zone by zone survey or design data is available. Unless you need to enter specific known occupancy gains zone by zone, this option is best avoided as it increases the amount of work involved

Note: When using the **3-Number of people** option, the number of people in each room is entered zone by zone as values inherited from building and block levels will typically not be correct.

Occupancy latent gains

When the [Gain definition level](#) option is set to **1-Globally in model options**, select the Occupancy latent gains method. There are 2 ways for the sensible-latent split of occupancy gains to be defined:

- **1-Dynamic calculation** - where the latent fraction of occupancy gains is calculated within EnergyPlus at runtime based on internal temperatures and metabolic rate. In this case the ratio of sensible to total occupancy gain will reduce as the internal temperature in the zone increases and at temperatures much above 30°C all of the occupancy gain will be latent (occupants sweating).
- **2-Fixed fraction** - where the [latent fraction](#) of occupancy gains is constant regardless of internal conditions and defined on the Activity tab under the [Occupancy header](#).

Note: The total occupancy gain (sensible + latent) is not affected by this option, only the split of sensible vs latent gain.

Equipment gain units

When the [Gain definition level](#) option is set to **1-Globally in model options**, set the Equipment gain units. This option provides control over the method used to enter Internal gains such as [computers](#), [office equipment](#), [miscellaneous](#), [catering](#), [process](#).

- **1-Power density** where zone equipment gains are entered as a power density, i.e. as a power per zone floor area in W/m² or W/ft². This option is usually the most convenient method for most general purpose modelling as the data is normalised by area and so can be inherited from levels above in the model hierarchy and can be effectively loaded from Activity template.
- **2-Absolute zone power** where zone equipment gains are entered as a power value in W. This option is usually used when detailed zone by zone survey or design data is available. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Note: When using the **2-Absolute zone power** option, the equipment gains data is normally entered individually for each zone because values inherited from building and block levels will typically not be correct.

Lighting gain units

When the [Gain definition level](#) option is set to **1-Globally in model options**, select the Light gain units. There are 3 ways to define [General lighting gains](#) in DesignBuilder:

- **1-Power density** - where the [General lighting gains](#) are defined per floor area. This method has the advantage that the level of gains can clearly be understood by looking at W/m² values and floor area. However it can cause problems when loading data from lighting templates in that if the activity is changed subsequently with a different illuminance requirement, the lighting power density is not updated. This issue has led to the introduction of the second option.
- **2-Normalised power density** - Although the units for Normalised power density (W/m²-100 lux or W/ft²-fc) may seem quite obscure, this method of defining lighting gains has advantages over using simple Power density. Namely that the actual lighting level is associated with the lighting system type without the need to refer to the zone activity and hence illuminance requirement. So if a particular lighting system type is installed throughout a building, but the building has a range of activities each with different illuminance requirements, it will usually be possible to enter the lighting **Normalised power density** value once only at building level, set the activity for each zone and obtain realistic lighting gains. See [General lighting gains](#) for details on the calculation of lighting gains from Normalised power density, floor area and illuminance requirements.
- **3-Absolute zone power** - where the lighting gains are defined as W without reference to the zone floor area. This option is used when the actual absolute lighting gains levels are known in W. Unless you need to enter specific known lighting equipment gains zone by zone, this option is best avoided as it increases the amount of work involved due to the inheritance mechanism not working effectively for gains that are not normalised by floor area.

Note: When using the **3-Absolute zone power** option, the lighting gains data is normally entered individually for each zone because values inherited from building and block levels will typically not be correct.

Timing Model Option

[Data tab on Model Options dialog.](#)

You can specify schedules in DesignBuilder using two approaches:

- **Typical workday** - where the timing is defined by a start time, end time, working days per week and seasonal variation. This approach is intended for easy early stage modelling of non-domestic buildings. This option is not available when using [Detailed HVAC](#) or [Detailed gains](#).
- **Schedules** - where [Schedules](#) can either be defined for each day of the week and each month of the year using daily [Profiles](#) or by using [Compact Schedule](#) format. Schedules are more flexible and can be used in other projects but they take longer to set up.

Generally if you start to find the Typical workday timing mode too restrictive you should switch to Schedules mode.

See also the [Timings, schedules and profiles](#) tutorial

You can also request:

Timing HVAC + natural ventilation operates with occupancy

Select this option if all heating, cooling, natural ventilation and DHW operation follows occupancy.

*This option is only available if Timing is set to **Typical workday**.*

Note that HVAC operation data can modify the HVAC operation even when **Timing HVAC + natural ventilation operates with occupancy** is set. For example:

- **Simple HVAC** - Mechanical ventilation, Heating and Cooling operation have additional Days / week and seasonal operation data. Heating operation also has Preheat hours and Cooling operation also has Precool hours.

These differences are applied as 'modifiers' to the basic occupancy schedule because HVAC systems generally switch on and off based on the occupancy time but with differences. e.g. Heating systems are often switched off in summer, they may operate during weekends when the building is unoccupied and they will often have a preheat period to get the building up to temperature before occupancy.

Internal gains operate with occupancy

Select this option if lighting and other internal gains operation follows occupancy. If you wish to enter specific schedules for lighting and/or other internal gains you should uncheck this option. For example there may be low levels of emergency lighting and some IT equipment usage while the building is unoccupied.

HVAC Model Options

[Data tab on Model Options dialog.](#)

HVAC model detail

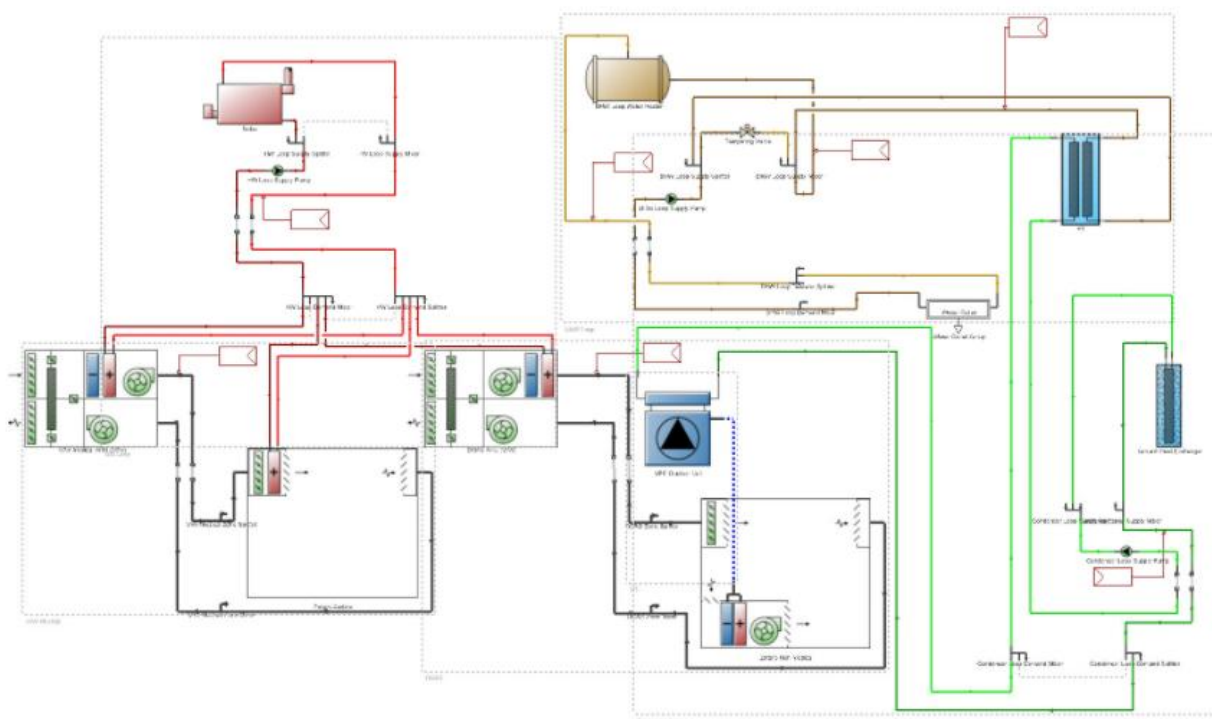
HVAC systems can be defined in DesignBuilder using a choice of 2 different levels of detail:

- **Simple HVAC** where the heating/cooling system is simulated using the basic EnergyPlus *ZoneHVAC:IdealLoadsAirSystem* method. This supplies hot/cold air to meet heating and cooling loads.

See topics on [Heating](#) and [Cooling](#) for more on defining the condition of the supply air. Mechanical ventilation loads are calculated locally for each zone. With Simple HVAC, fuel energy consumption for heat generation (typically a boiler or heat pump) and chiller is calculated from zone heating and cooling loads as a [post-process](#) using simple seasonal efficiency factors.

See also the [Simple HVAC Data](#) tutorial

- **Detailed HVAC** where the HVAC system is modelled in full detail using EnergyPlus air and water-side components linked together on a schematic layout drawing (e.g. below). In this case HVAC data is accessed by clicking on the [<HVAC System> navigator node](#). This method requires more effort in setting up the HVAC model description and a greater knowledge of HVAC systems than the Simple HVAC option.



Detailed HVAC Activity data

When using Detailed HVAC, this option controls how zone temperature and humidity setpoints as well as ventilation requirement and hot water consumption data is defined. Two options are available:

- **1-Simple HVAC Data** where zone temperature and humidity setpoints, ventilation requirements and hot water consumption rates for simulations are taken from the Activity and HVAC model data tabs. The advantage of this option is that the Activity-related HVAC data will typically already be set up based on the activity selection and so a) there is less work involved in setting up the setpoints etc and b) the simulation outputs will be more comparable with any previous Simple and Detailed HVAC simulations carried out. The downside is some loss of flexibility in setpoints (only 2 temperature setpoints per zone can be used - Setpoint and Setback temperatures).
- **2-Detailed HVAC Data** where the activity-related HVAC data is entered on Detailed HVAC dialogs. This option provides increased flexibility but at the expense of extra work in setting up the setpoints etc, especially for large models. For example it is possible to have a range of different heating and cooling setpoint values set up using schedules for different times of the day or year. It is also possible to apply different setpoints for [Chilled ceilings](#) and other cooling equipment and likewise for [Heated floors](#).

The tables below show specifically where each item of Activity-related HVAC data is entered for the 2 options.

1-Simple HVAC Data

Activity Tab - Setpoints, Fresh Air Rates, DHW Consumption

[Heating setpoint and setback temperatures](#)
[Cooling setpoint and setback temperatures](#)
[Humidification setpoint](#)
[Dehumidification setpoint](#)
[Minimum fresh air per person](#)
[Minimum fresh air per floor area](#)
[DHW consumption rate per floor area](#)

HVAC Tab - Setpoint timing / Operation

[Heating operation schedule](#)
[Cooling operation schedule](#)
[DHW operation schedule](#)

2-Detailed HVAC Data

HVAC Zone Dialog

[Heating setpoint schedules](#)
[Cooling setpoint schedules](#)
[Humidifying setpoint schedules](#)
[Dehumidifying setpoint schedules](#)
[Minimum fresh air per person](#)
[Minimum fresh air per floor area](#)

Water Outlet Dialog

[Flow rate fraction schedule](#)
[Peak flow rate](#)

Heated Floor Dialog

[Heating setpoint schedules](#)

Chilled Ceiling Dialog

[Cooling setpoint schedules](#)

Note: The tables above summarise the location of the various setpoints, fresh air, schedules etc for the 2 Detailed HVAC activity data options. For **heating and cooling design** calculations the humidity and temperature setpoints and ventilation requirement data comes from the Activity and HVAC tabs as described in the **1-Simple HVAC** table, regardless of the **Detailed HVAC Activity Data** setting.

Run free (disable detailed HVAC system)

This option is provided as a quick easy way to run simulations without the HVAC system operating without the need to remove the system from the model.

Specify Simple/Design HVAC details

When using **Simple HVAC**, you can choose whether to specify some basic details on the system. If you activate this option then you will be able to specify:

- System type (convective or convective/radiant).
- Supply air temperatures for simulation.
- HVAC auxiliary data (below).
- Mechanical ventilation method.
- Humidity control, heat recovery and economiser (below).

This option might be switched off for early design stage architectural use to simplify the HVAC data entry. If you switch this option off then any heating or cooling is assumed to be entirely convective.

This option is additionally available when using **Detailed HVAC**, in which case it has a similar effect to that for Simple HVAC, but in this case it applies to data on the HVAC tab that is only used for Heating and Cooling design calculations. When using Detailed HVAC you should only uncheck this option if you do not plan to use Heating and Cooling design calculations.

Auxiliary energy calculations - Simple HVAC

When using Simple HVAC and you choose to specify Simple HVAC details (above), there are 3 options for modelling fans, pumps and other [auxiliary energy](#) associated with the HVAC system:

- **0-None** - Auxiliary energy is not calculated. This option is typically used at early design stages where HVAC modelling may not be required.
- **1-NCM** - This option models auxiliary energy as a fixed annual energy consumption per floor area, with default values stored in the HVAC templates coming from the UK NCM.
- **2-Separate fans and pumps** - This option allows you to use the EnergyPlus fan energy consumption calculation and a basic W/m² + schedule approach to modelling pumps and control equipment.

Mechanical ventilation method - Simple HVAC

This setting defines how mechanical ventilation is treated for Simple HVAC Simulation and Cooling design calculations. The options are:

- **1-Room ventilation** - Mechanical ventilation is modelled using EnergyPlus **ZoneVentilation:DesignFlowRate** data separate from the main HVAC system. With this option the fan energy consumption is included and the effect of heat pick up from fans is included in the zone heat balance. No humidity control, economiser or heat recovery is possible with this option. [Radiant/convective heating](#) is possible with this option. Mechanical ventilation outdoor air loads are displayed separately from HVAC heating and cooling loads in heat balance results.
- **2-Ideal loads** - Mechanical ventilation is modelled using the EnergyPlus **ZoneHVAC:IdealLoadsAirSystem** data in combination with the heating/cooling system. In this case there are options to include the effects of heat recovery, economiser, humidification and dehumidification. Fan energy and pickup is not included in the simulation. Select this option if humidity control is required in Cooling design calculations, or if Calculated natural ventilation simulation is required. [Radiant/convective heating](#) is not possible with this option. When this option is selected, Mechanical ventilation outdoor air loads are displayed combined with HVAC heating and cooling loads in heat balance results.

Note: Simple HVAC combined with the **1-Room ventilation** method and Calculated natural ventilation means you cannot model Mechanical ventilation. If you wish to combine Calculated natural ventilation and Mechanical ventilation in a Simple HVAC simulation, you should use the **2-Ideal loads** method.

HVAC Sizing (Autosizing)

When using Simple HVAC, DesignBuilder can automatically calculate heating and cooling capacity in each zone based on the output from the Heating and Cooling design calculations. The HVAC sizing model option provides control over how [heating](#) and [cooling](#) capacities are defined. Select from:

- **1-Adequate**
- **2-Manual**
- **3-Autosize**
- **4-Autosize when not set**

These HVAC sizing options are described below.

Note: In Heating and Cooling design calculations, heating and cooling loads are always met and so this option does not apply - it only applies to Simulations.

Adequate

When this option is set, HVAC equipment is modelled as having unlimited capacity allowing it to meet any heating or cooling demand ensuring that setpoint temperatures are always maintained. This option can cause extremely large peaks in heating and cooling delivery to zones and will tend to significantly over-predict heating and cooling energy and under predict discomfort hours.

Manual

Heating and cooling capacity Model data must be entered manually on the HVAC Model data tab. Choose this option if you have specific heating and cooling capacity data you wish to use and do not want it to be overwritten when model data or model options change.

Autosize

Autosize is the default option where heating and cooling capacities are always calculated prior to each simulation and the heating and cooling capacity model data is not displayed. When using Simple HVAC the heating and cooling design calculations will be run as required before the simulation (within the same EnergyPlus run) to calculate the required sizes.

The autosizing calculation method is defined through the [Simple HVAC autosizing method](#) model option described below.

Autosize when not set

In this case heating and cooling capacities are autosized and loaded into model data following DesignBuilder Heating and Cooling design simulations. If heating/cooling capacity model data is not present before a simulation, it is generated by automatically by carrying out heating and cooling design calculations and is written into the HVAC model data.

You should be aware that when using the **4-Autosize when not set** option, the heating/cooling zone capacity model data is not treated the same way as other calculated results. In general, calculated results are deleted when any edit is made to the building model, but this is not the case for zone capacity model data. It is only updated when:

1. There is no heating/cooling capacity data (the value is either blank or zero when heating/cooling is switched on) and a simulation is requested. In this case the appropriate design calculations are carried out to fill in the capacity data, before the simulation.
2. The appropriate design calculation is carried out by clicking on the Heating/Cooling design tab.
3. Model Options change or a geometric edit is made, such as deleting, adding or editing the geometry of an object. *In this case all zone HVAC capacities are reset to zero.*

If you always want the heating capacity to be kept up-to-date with your model and do not have any specific capacity sizes you wish to use, you should normally use the default **3-Autosize** option. This will cause EnergyPlus autosizing to be carried out prior to every simulation.

Tip: When using **4-Autosize when not set**, you can manually change heating and cooling capacity settings on the HVAC tab if you wish, but you should bear in mind that, if Model options or model geometry change, all heating and cooling capacities are reset to zero. To avoid this happening, once you have established the required heating and cooling capacity settings, set **HVAC Sizing** to **2-Manual**.

Simple HVAC autosize method

When using Simple HVAC with the **3-Autosize** option, there are 2 ways to carry out HVAC autosizing calculations:

- **1-EnergyPlus** where EnergyPlus autosizing pre-simulations are carried out as part of the main simulation using the EnergyPlus built-in autosizing function, in the same way as autosizing for Detailed HVAC. This is the default option.

- **2-DesignBuilder**, where [Heating design](#) and [Cooling design](#) calculations are run first to calculate the size of heating and cooling plant to use in the main simulation. This option involves 3 separate simulations of the building: 2 for the autosizing design calculations plus 1 for the main simulation.

The **1-EnergyPlus** option is recommended as the fastest way to carry out autosizing calculations. With this option, simulation times tend to be much shorter, particularly for short simulation periods. However, one advantage of the **2-DesignBuilder** alternative option is that it provides the results of the 2 autosizing simulations on the [Heating design](#) and [Cooling design](#) screens. These results will be consistent with the plant sizes used in the simulation.

With the **1-EnergyPlus** option the autosizing simulation outputs are generated in the EnergyPlus autosizing spreadsheet output files:

- **epluszsz.csv** for zone outputs and
- **eplusssz.csv** for system outputs.

Some minor differences in zone sizing results between the 2 methods are likely as explained below.

Differences between 1-EnergyPlus and 2-DesignBuilder Autosizing methods

In general the autosized heating and cooling sizes between the 2 methods should be close, however because the simulation methods are different some differences may be observed due to one or more of:

1. The Cooling design calculations used in the **2-DesignBuilder** method are run using 2 timesteps per hour whereas the cooling autosizing simulations are run using the number of timesteps selected for the simulation. The timestep difference doesn't arise for heating design where the inside and outside conditions are static.
2. The **WinterDesignDay** settings control the operation for the heating autosizing simulations, whereas for Heating design calculations, fixed values are always used and there is less scope for obtaining "surprising" results. E.g. there is no way to include internal gains from occupants and equipment, whereas it is possible to include these gains (deliberately or inadvertently) in heating autosizing simulations through compact schedule settings.
3. When the Calculated natural ventilation model option is selected this is used in autosizing simulations (if the relevant schedules allow it) whereas for Cooling design calculations the scheduled natural ventilation settings are used (if natural ventilation is enabled).
4. Heat recovery can be activated or not for Cooling design calculations whereas it is *not* included when sizing zone loads in autosizing simulations.

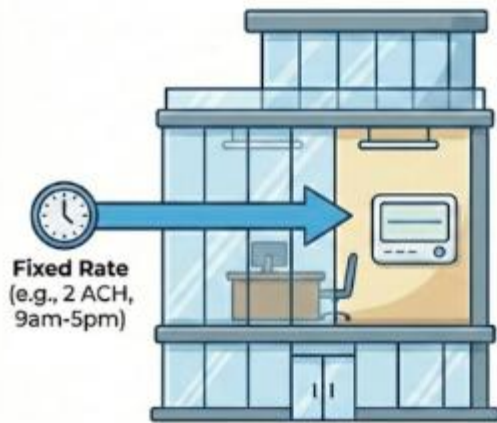
Natural Ventilation And Infiltration Model Options

[Data tab on Model Options dialog.](#)

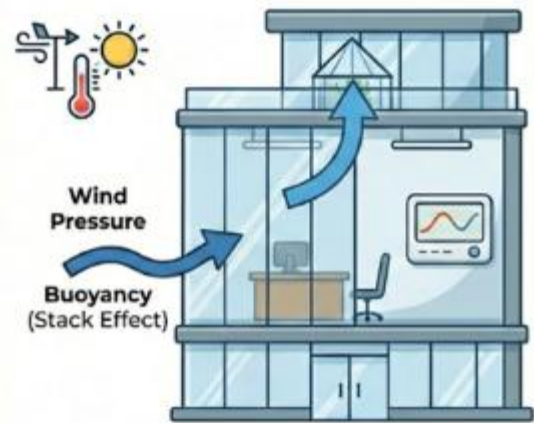
Natural ventilation

Natural ventilation (and infiltration) can either be 'Scheduled' or 'Calculated'.

SCHEDULED NATVENT (Simplified and Early Design)



CALCULATED NATVENT (Dynamic & Accurate)



- **Scheduled** - the ventilation rates are predefined using a [maximum air change rate](#) modified by [operation schedules](#).
- **Calculated** - the ventilation rates are calculated using wind and buoyancy-driven pressure, opening sizes and [operation](#), crack sizes etc. using the EnergyPlus Airflow Network.

You should generally use the **Scheduled** natural ventilation model option because it is quicker and easier to enter the related model data and because the simulations run more quickly. However if you need to calculate natural ventilation rates based on wind and buoyancy-driven pressure, opening sizes and [operation](#), crack sizes etc then you should use the **Calculated** option.

Note: This natural ventilation setting also affects the way that infiltration is treated. It isn't possible to calculate infiltration as a constant (or scheduled) value while using the EnergyPlus Airflow Network for Calculated natural ventilation.

[More on Natural Ventilation modelling.](#)

Infiltration units

There are 4 different ways to specify infiltration in calculations not using the AirflowNetwork (Heating and Cooling design and Simulation with Scheduled natural ventilation):

- **1-ac/h** - the default option where infiltration is defined in air changes per hour (ac/h), i.e. zone volume per hour under normal operating pressures.
- **2-m3/h-m2 at 50 Pa** infiltration is defined as flow rate per unit exposed surface area, when the pressure difference between inside and outside is 50 Pa. Unit commonly used in the UK.
- **3-m3/h-m2 at 4 Pa** infiltration is defined as flow rate per unit exposed surface area, when the pressure difference between inside and outside is 4 Pa. Units commonly used in France and Belgium.
- **4-n50 (ac/h at 50 Pa)**- infiltration is defined in air changes per hour (ac/h), i.e. zone volume per hour, when the pressure difference between inside and outside is 50 Pa.

The selection made here will affect the units of the infiltration data prompted for on the Constructions tab under [Airtightness](#).

The infiltration defined in the model is converted to m/s for input to EnergyPlus. The conversion from these options to m/s uses the methods described in EN 12831.

Airtightness method

When the Calculated natural ventilation method is selected there are 2 ways to define the airtightness of the building:

- **1-Template slider**, where 5 predefined airtightness settings are available. This method is most suitable for more early stage analysis.
- **2-Crack template**, where custom crack templates can be defined and selected. This method provides more control over the airtightness settings used.

Imported Surfaces

Shade surfaces

This checkbox defines whether any [shading plane surface](#) objects imported from a model via gbXML or IDF are to be applied to the DesignBuilder model. Enabled shade surfaces will be listed in the Navigator, displayed on the Edit screen as planes of various colours, rendered on the Visualisation screen and included in Radiance daylighting and EnergyPlus simulations as shading surfaces.

The only type of surface currently supported are shading plane surfaces which are displayed in green.

Shade surfaces that are not selected are not actually deleted from the model - they can be displayed again by changing the selection on this dialog.

Note: When shade surface planes are being shown (**Shade surfaces** is ticked), the [Component block shades and reflects](#) checkbox on the Constructions tab can be unchecked to prevent all or just particular planes from being included in EnergyPlus simulations and cooling design calculations.

Note: Shading plane surfaces do not have any impact on external CFD analyses.

Advanced - Model Options (Building)

Advanced tab on [Model Options](#) dialog.

- [Simplification](#) - options to allow zones to be merged and for controlling lumping of similar windows, openings and cracks and eliminating unnecessary zones and elements from the model.
- [Adjacency settings](#) - options to control the way that DesignBuilder calculates the adjacency for neighbouring surfaces.
- [Natural ventilation](#) - options to control the operation of natural ventilation.
- [Lighting](#) - options for lighting.
- [Filters](#) - excluding elements from the model.
- [Component Blocks](#) - options related to component blocks

This data applies to the current building.

Simplification

Advanced tab on [Model Options](#) dialog.

You can simplify and speed up simulations (sometimes quite significantly) by lumping zones, walls and windows and/or eliminating unnecessary zones and elements from the model. The following options are available:

Merge zones of same activity

This option reduces the number of zones by [merging zones](#) having the same activity.

Merge zones connected by holes

This option [merges zones](#) which are connected by [holes drawn at surface level](#) (*not* the holes used to represent the airflow path in [virtual partitions](#)). You can merge one or more zones within the same block or across multiple blocks.

See also the [Drawing holes and merging zones](#) tutorial

Tip: The recommended way to merge zones is through the **Merge zones by selection** option below as it gives a greater degree of control over which zone is used as the parent.

Merge zones by selection

Check this option if zones are to be merged by selection using the [Merge zones dialog](#).

Lump similar windows on surface

When selected, this option causes all windows on a zone surface having the same area, construction, frame/dividers, shading etc to be "lumped" into a single representative opening using the window multiplier concept. In this case just one of the repeated windows is simulated and the window multiplier is used within EnergyPlus to multiply up the window heat flows so that, overall, the simulation behaves as if all of the window had been included. For models having surfaces with many similar windows this option can provide a worthwhile reduction in simulation times.

Technical Note

When this model option is selected, for surfaces with multiple similar windows, the IDF file will contain less window objects than are defined in the DesignBuilder model. The one window in the IDF used to represent all equivalent windows on the surface will use the window multiplier to ensure equivalence with the DesignBuilder model.

Important Note: With Calculated natural ventilation you should not use this option if you have repeated windows having same size, Glazing type and [Window shading](#) on a surface but different [Window Operation and Free Aperture](#) data such as [% Glazing opens](#), [Opening position](#), [Discharge coefficient](#) and/or [Operation schedule](#), because any differences in these natural ventilation settings are not accounted for when checking for similar windows on a surface.

Similarly, you should not use this option if you have repeated windows having same size, Glazing type and [Window shading](#) on a surface but different [Local shading](#) settings.

Also, you should also switch this option off if you are running simulations to generate boundary conditions for CFD or for [graphical data visualisation](#).

DesignBuilder does not allow a multiplier for windows to be used for zones with [daylight lighting control](#) or if the **3-Full interior and exterior Solar distribution** option is set. This is because for daylight control or **Full interior and exterior Solar distribution**, EnergyPlus must know where each of the individual windows is located to properly calculate the distribution of solar and visible light transmitted.

This is currently the only 'Simplification' option switched on by default for new models.

Lump similar cracks

This option causes all cracks in a zone which are facing the same direction and have similar height to be lumped into a single crack. This option applies only when the natural ventilation model option is **Calculated**.

Adjacency Settings

[Advanced tab](#) on [Model Options](#) dialog.

Adjacency separation tolerance

This is the maximum gap between neighbouring parallel (see [Adjacency angular tolerance](#) below) surfaces that will be interpreted as being an adjacency between 2 objects. It is used to determine which elements are adjacent to other spaces and which are adjacent to the exterior. It is also used to decide whether a component block is adjacent to a block surface or not.

Note: By default in cases where a component block is positioned closer to an external wall than this tolerance, windows on the external wall will be removed. If you would like to allow component blocks to be positioned closer to the window than the default tolerance of 0.5m then enter a smaller value here.

This setting applies to all model geometry but is particularly important when importing BIM geometry which often includes gaps between zones. You might enter a value higher than the default 0.5m if the adjacent spaces are separated by very large gaps due to thick partitions or ceiling/floor voids being excluded from any gbXML data imported. For example if the largest gap between spaces that are to be modelled as adjacent is 0.6m then enter 0.6m here.

The Adjacency tolerance can also be set from the [Import BIM/gbXML Model](#) dialog before finalising the import.

Adjacency angular tolerance

In order for 2 neighbouring surfaces to create an internal adjacency they should be closer to each other than the limit defined for the [Adjacency separation tolerance](#) (above) and the angle between them should be no more than this tolerance. The default tolerance is 5°, the minimum is 1° and the maximum is 10°.

Standard component block adjacencies

[Ground and Adiabatic](#) component blocks always have adjacencies with any neighbouring building blocks (when closer than the [Adjacency separation tolerance](#)) and so will cause any openings to be lost on shared adjacency surfaces when placed next to walls. By default [Standard](#) component blocks do not do this and they can be placed close to (or touching) walls without windows etc being lost. However it is possible to cause Standard component blocks to behave like other component blocks in this respect by checking this option.

Natural Ventilation Options

[Advanced tab](#) on [Model Options](#) dialog.

Model holes (including holes in virtual partitions)

You can optionally switch off all [holes](#) in the model including those holes generated by defining partitions as [virtual partitions](#). In this case the building is modelled as if no holes were defined.

Tip: Unchecking this data can be a quick way to model all virtual partitions in the model as standard partitions.

Calculated Natural Ventilation

Discharge coefficient for open doors and holes

Enter the discharge coefficient for all open doors and holes. The default of 0.65 is a good general purpose setting.

The [discharge coefficient for windows](#) is defined in model data on the Openings tab.

The discharge coefficient for vents is defined on the Vent dialog.

Scheduled Natural Ventilation

Airflow through internal openings

When using Scheduled natural ventilation, windows, vents, doors and holes in internal partitions can be modelled by mixing a predefined amount of air from one zone with the adjacent zone by checking the **Airflow through internal openings** checkbox and entering the **Airflow rate per opening area**. This inter-zone airflow is transferred equally in both directions and is modelled using the EnergyPlus **ZoneMixing** option.

The mixing flow rate for internal openings is calculated as:

Airflow through internal openings x Opening area x *Factor*

The value of the *Factor* used in the above calculation depends on the type of opening as follows:

- Windows: % Glazing area opens / 100
- Vents: 0.9 (the rationale is that the vent is 90% free area with 10% obstruction by the vent slats)
- Doors: (% Time open / 100) * (% Area door opens) / 100
- Holes: 1

See also:

- "Interzone Airflow" on page 1349

Note: An internal airflow path (IDF **ZoneMixing** object) will only be added if both zones are have the **1-Standard Zone type** (**1-Conditioned** for ASHRAE 90.1 models) set.

Lighting

[Advanced tab](#) on [Model Options](#) dialog.

Daylighting method

The options for calculating daylighting in EnergyPlus are:

- **1-Detailed** where the zone daylighting illuminance levels are calculated and then used to determine how much the electric lighting can be reduced. The daylight illuminance level in a zone depends on many factors, including sky condition; sun position; calculation point; location, size, and glass transmittance of windows; window shading devices; and reflectance of interior surfaces. Reduction of electric lighting depends on daylight illuminance level, illuminance set point, fraction of zone controlled and type of lighting control. This is the default method implementing the EnergyPlus **Daylighting:Controls** object.
- **2-DElight**. The DElight method of analysing daylighting in buildings is very similar to that used in the Detailed method. For each point in time, DElight calculates the interior daylighting illuminance at specified reference points and then determines how much the electric lighting can be reduced while still

achieving a combined daylighting and electric lighting illuminance target. The daylight illuminance level in a zone depends on many factors, including exterior light sources; location, size, and visible light transmittance of simple and complex fenestration systems; reflectance of interior surfaces; and location of calculation reference points. The subsequent reduction of electric lighting depends on daylight illuminance level, illuminance set point, fraction of zone controlled, and type of lighting control. There are two primary differences between the Detailed and DELight methods of calculating interior illuminance levels. The first is that DELight includes the capability of analysing complex fenestration systems that include geometrically complicated shading systems (e.g., roof monitors) and/or optically complicated glazings (e.g., prismatic or holographic glass). The second key difference is that DELight uses a radiosity method to calculate the effects of light reflection inside a zone. These methods are discussed in more detail in the engineering documentation. There are other important differences between the two methods. One is the inability of DELight to perform the type of dynamic shading controls possible using the Detailed method at each point in time during the thermal simulation (e.g., changes in electrochromic glazing transmittances and blind slat angles). Another is the DELight ability to include more than two reference points in its interior illuminance and electric lighting reduction calculations. A third is the current lack of visual quality (e.g., glare) calculations performed by DELight. Fourth, the modeling of interior obstructions is different in the two methods. In the DELight method interior obstructions block inter-reflections but do not block the initial direct illuminance. In the Detailed method, interior obstructions block the initial direct illuminance but do not block inter-reflections. Input for invoking the DELight method involves three object types: **Daylighting:DELIGHT:Controls**, **Daylighting:DELIGHT:ReferencePoint**, and **Daylighting:DELIGHT:ComplexFenestration**. DesignBuilder uses the first 2 of these.

Warning: Because of the above-mentioned limitations and because generally we believe that the **1-Detailed** method is much more widely used and better tested, we strongly recommend that you avoid using the DELight option unless you have very good reason to do otherwise.

Filters

[Advanced tab](#) on [Model Options](#) dialog.

Exclude surface elements smaller than

You can also filter out very small surfaces from the simulations (and from the Navigator panel) by defining a minimum area.

Experience suggests that a value of 0.05 is a good default. If you set the limit much lower than this you will run the risk of including very small surface slivers in the calculations. These can cause problems with EnergyPlus.

Component Blocks - Model Options

[Advanced tab](#) on [Model Options](#) dialog.

Maximum flat surface thickness

Component blocks that are considered to be flat (i.e. consist of 2 large surfaces and several smaller 'edge' surfaces) are normally represented in the simulations by a single shading surface. The Maximum flat surface thickness entered here defines the thickness of component blocks below which they are considered to be "flat" and therefore to be modelled using a single shading surface. This same threshold value is used for all component blocks in the model. The default threshold value is 0.1m.

The method used to define which component block surface(s) are used to represent flat component blocks can be entered on the Construction tab under the Component Block header using the [Flat surface position](#) option.

Heating Design Calculation Options - Model Options (Building)

Heating Design tab on [Model Options](#) dialog.

Data on this tab allows you to control the Heating Design calculations.

- [Winter Design Day](#)
- "Calculation Options - Heating Design" on page 1550
- "Output Options - Heating Design" on page 1281
- "Advanced Calculation Options - Heating Design" on page 1553

All of the above data apart from **Winter Design Day** is also available on the Heating Design Calculation Options dialog.

See also the [Heating Design Calculations](#) tutorial

Winter Design Day

[Heating Design](#) tab on [Model Options](#) dialog.

Operation For 7/12 And Typical Workday Schedules

This series of options allows you control which 7/12 Schedules are switched on during Detailed HVAC heating autosizing. [7/12 Schedules](#) are categorised using the following categories when using the **End-use defaults** option:

- **General** - include general purpose schedules in Detailed HVAC heating [autosizing](#) calculations - on by default.
- **Occupancy** - include occupancy schedules - off by default
- **Lighting** - include lighting schedules - off by default.
- **Equipment** - include equipment schedules - off by default.
- **Heating demand** - include heating demand schedules - on by default. Unchecking this will prevent Heating design calculations from working.
- **Cooling demand** - include cooling demand schedules - on by default.
- **HVAC** - HVAC schedules - on by default.
- **Natural ventilation demand** - natural ventilation schedules - off by default.
- **DHW** - DHW schedules - on by default.

For example, occupancy gains should normally be excluded when sizing heating plant to ensure that the design heating plant meets all heating loads itself. So leave the Occupancy check box unchecked.

These options are also used for [Typical workday](#) schedules.

Important Note: Heating and cooling demand schedules must be included to ensure the system operates, so the **Heating demand** and **Cooling demand** options should normally left selected.

The same principle applies to **Heating design calculations** (since they use Typical workday schedules behind the scenes), so if the **Heating demand** setting is unchecked, the Heating design calculations will run as if no heating has been selected.

Important Note: This system of categories can be confusing and so generally you are advised to use either the [2-Profiles](#) 7/12 Schedules option where the schedule operation during heating and cooling are defined explicitly using a profile or use [compact schedules](#) where again heating and cooling design operation is defined explicitly.

Calculation Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog and [General](#) tab on [Heating Design](#) dialog.

Calculation Options

Calculation description

Enter some text to identify the calculation. This will be used in reports and in the filename for [automatically generated ESO files](#).

Temperature control

Heating and cooling systems control internal temperatures to meet the setpoint temperatures specified on the [Activity tab](#). These setpoint temperatures can be interpreted as air, operative or some other radiant fraction and DesignBuilder provides corresponding options to allow HVAC systems to be controlled by:

- **1-Air temperature** - control the zone mean air temperatures to the [heating and cooling setpoint temperatures](#) specified on the [Activity tab](#).
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this option.
- **3-Other** - you can enter the radiant temperature control fraction.

When using the **3-Other** option the radiant fraction should be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Radiative Fraction vs Air Velocity

Air Velocity (m/s)	<0.2	0.2 - 0.6	0.6 - 1.0
Radiant fraction	0.5	0.4	0.3

Note: The temperature control settings used in heating design and cooling design will also apply in simulation autosizing calculations for the winter and summer design days respectively.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

You can think of the Temperature control option as:

- The radiant fraction sensed by the room thermostat, or,
- Given a room thermostat that mainly senses air temperature, the extent of automatic modification of the setpoint temperature to ensure occupant comfort.

Note: This option does not affect natural and mechanical [ventilation setpoints](#) - these always use **air temperature** set points.

Tip: See the equivalent topic for Simulation for an [advanced discussion on Operative vs Air temperature control](#).

System Sizing

Design margin

The Heating Design margin is used to multiply calculated steady state heating loads in each zone to give a recommended heating system size. It accounts for the additional heat required to bring the building up to temperature in a reasonably short preheat period and allows you to be confident that comfort conditions will be maintained in all but the most extreme winter conditions.

The default heating design margin is 1.25 which results in heating systems being oversized by 25% relative to the steady state load calculated by EnergyPlus. This value is based on ASHRAE Standard 90.1. For projects not subject to an energy code or standard that specifies a mandatory safety factor or design margin, the selection of an appropriate value is ultimately at the discretion of the designer.

Output Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog, [General](#) tab on [Heating Design Calculation Options](#) dialog and [Miscellaneous](#) model data tab

Heating design output options apply to the whole building and are only visible at building level when accessed from the [Outputs](#) model data tab.

Include unoccupied zones in block and building totals and averages

By default data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Store surface output

Causes the following output to be stored for all walls, floors, roofs, ceilings, partitions etc:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Loss
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Store opening output

Only visible when the **Store surface output** option is selected, this option causes the following data to be stored for windows vents, doors, holes, sub-surfaces:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Loss
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Warning: Use the surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be stored.

Advanced Calculation Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog and General tab on the [Heating Design Calculation](#) options dialog.

General Solution

Temperature and loads convergence

The temperature and loads convergence values represent the maximum allowable difference in zone temperature and loads between successive daily iterations before convergence is considered to have been reached during warmup.

Convergence of the simultaneous heat balance/HVAC solution is reached when both the loads and temperature criteria are satisfied. Both tolerances work the same way, one looks at temperatures and one looks at heating and cooling loads. After the second warm-up day, the program compares the maximum temperature experienced in a space with the maximum temperature from the previous day. If those two temperatures are within the tolerance, then it has passed the first warm-up check. It does a similar comparison with lowest temperatures experience within all the zones. If the current simulation day and the previous day values are within the tolerance, then it has passed the second warm-up check. A similar comparison is carried out with the loads tolerance and the maximum heating and cooling loads that are experienced within the spaces. Those are compared individually to the values for the previous day. If they are both in tolerance, then the simulation has passed the third and fourth warm-up check. The simulation stays in the warm-up period until ALL FOUR checks have been passed.

Inside and Outside convection algorithms

You can select from a range of EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. More details on this and external convection can be found under [Constructions](#) Model Data > [Surface Convection](#) header.

Other

Surfaces within zone treated as adiabatic

EnergyPlus support recommend modelling surfaces wholly contained within a zone as adiabatic and this option allows you to follow this advice. We have found that in practice this option does not make much difference in results or in simulation speed so for most cases you can leave it in its default state.

See also [EnergyPlus Errors and Warnings](#)

Note: 'surfaces contained within a zone' are frequently generated when one of the [zone merging](#) options is used where a partition or floor which would have separated two zones actually sits within the merged zone. These surfaces *do not* refer to [hanging partitions](#) which are modelled using [Internal thermal mass](#).

Cooling Design Calculation Options - Model Options (Building)

Cooling Design tab on [Model Options](#) dialog.

Data on this tab allows you to control the Cooling Design calculations:

- [Calculation Options](#)
- [Summer Design Day](#)
- [Cooling system sizing](#)
- [Solar Options](#)
- [Output Options](#)
- [Advanced Calculation Options](#)

This data can also be accessed before Cooling Design Calculations.

See also the [Cooling Design Calculations](#) tutorial

Calculation Options - Cooling Design

[Ref641103961](#) [Cooling Design](#) tab on [Model Options](#) dialog and General tab on [Cooling Design Calculation Options](#) dialog.

Calculation Description

Enter some text to identify the calculation. This will be used in reports and in the filename for [automatically generated ESO files](#). The Calculation description is only available on the Cooling design calculation options dialog.

Calculation Options

Temperature control

Heating and cooling systems control internal temperatures to meet the setpoint temperatures specified on the [Activity tab](#). These setpoint temperatures can be interpreted as air, operative or some other radiant fraction and DesignBuilder provides corresponding options to allow HVAC systems to be controlled by:

- **1-Air temperature** - control the zone mean air temperatures to the [heating and cooling setpoint temperatures](#) specified on the [Activity tab](#).
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this option.
- **3-Other** - you can enter the radiant temperature control fraction.

When using the **3-Other** option the radiant fraction should be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Radiative Fraction vs Air Velocity

Air Velocity (m/s)	<0.2	0.2 - 0.6	0.6 - 1.0
Radiant fraction	0.5	0.4	0.3

Note: The temperature control settings used in heating design and cooling design will also apply in simulation autosizing calculations for the winter and summer design days respectively.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

You can think of the Temperature control option as:

- The radiant fraction sensed by the room thermostat, or,
- Given a room thermostat that mainly senses air temperature, the extent of automatic modification of the setpoint temperature to ensure occupant comfort.

Note: This option does not affect natural and mechanical [ventilation setpoints](#) - these always use **air temperature** set points.

Tip: See the equivalent topic for Simulation for an [advanced discussion on Operative vs Air temperature control](#).

Exclude all zone natural ventilation (infiltration is always included)

This option should be selected when natural ventilation is to be excluded from the cooling calculations in all zones. Infiltration is always included regardless of this setting.

Exclude all zone mechanical ventilation

Select this option if you wish to exclude the zone mechanical ventilation load from the cooling loads in all zones. The option is useful in cases where the AHU is to meet mechanical ventilation loads and these loads are to be calculated separately when sizing the AHU coil capacity.

Exclude heat recovery

Check this option if you would like to exclude the savings introduced by heat recovery in any of the zones in the model that have heat recovery selected on the HVAC tab.

Summer Design Day

[Cooling Design](#) tab on [Model Options](#) dialog.

Day

Select the day of the month for the design day. The default day is 15, i.e. the middle of the month.

This data is only used to define the solar path over the design day and does not affect [temperatures](#) etc which are defined at site level.

Note: This same day of the month will be used for all months selected below when using the **2-Multiple design months** Design temperature period.

Month (or Start month and End Month)

Depending on the selection at site level for [Design temperature period](#), you can either select a single design month (when it is set to **1-Single design month**) or a month range (when it is set to **2-Multiple design months**).

When a single month is being used, the default design month will be 1 month after the month of mid-summer by default (July in the Northern Hemisphere, January in the Southern Hemisphere). When multiple months are being used, the default start month is the mid-summer month and the end month is 2 months later. This covers most design cases where peak conditions may occur significantly later after mid summer, especially when there are large amounts of South-facing glazing (in the Northern Hemisphere).

Important Note: If you are using the default **1-Single design month** [Design temperature period](#) option the design temperature and humidity values used in the calculations will be the “Yearly” design temperatures set at site level under Summer Design Weather Data. This is true regardless of the month and day of month selected on the Cooling design calculation options dialog. You will therefore find that the outdoor air temperatures won’t change when changing the month with this option set. You must use the **2-Multiple design months** option to see seasonal variations of design outside air conditions.

Tip: Use the **2-Multiple design months** [Design temperature period](#) option for a thorough analysis. DesignBuilder will sequentially test various monthly temperature and solar radiation profiles to automatically identify the peak cooling month on your project.

Day of week

This selection is used to identify the appropriate daily profile within the schedules to use for the Cooling design calculations. The day type should be the day when the most extreme conditions and highest cooling loads are expected. You should normally keep the **9-SummerDesignDay** default option, especially when also running simulations. Select the day of week from the options:

- **1-Sunday** - not recommended if the cooling system does not operate on Sundays.
- **2-Monday,**
- **3-Tuesday,**
- **4-Wednesday,**

- **5-Thursday**,
- **6-Friday**,
- **7-Saturday** - not recommended if the cooling system does not operate on Saturdays.
- **8-Holiday** - not recommended if the cooling system does not operate on holidays.
- **9-SummerDesignDay** - the default recommended setting.
- **10-WinterDesignDay** - not recommended.
- **11-CustomDay1**,
- **12-CustomDay2**

Cooling System Sizing

[Cooling Design](#) tab on [Model Options](#) dialog.

Design margin

The Cooling Design margin is a 'safety factor' used to multiply calculated cooling loads in each zone to give a recommended maximum cooling equipment capacity. It accounts for the additional cooling which might be required to cool the building down in a reasonably short pre-cool period and allows you to be confident that comfort conditions will be maintained in all but the most extreme summer conditions. The risk depends on the [Summer Design Weather Data](#) options selected at site level.

The default design margin is 1.15 which means that the cooling system will be oversized by 15%. This value is based on ASHRAE Standard 90.1. For projects not subject to an energy code or standard that specifies a mandatory safety factor or design margin, the selection of an appropriate value is ultimately at the discretion of the designer.

Sizing method

Sizing methods available are:

- **1-ASHRAE** - standard 'Heat Balance' method implemented in EnergyPlus.
- **2-Unitary DX** - calculations are made through a detailed EnergyPlus simulation using Unitary DX equipment.

Airflow calculation method

When calculating the design airflow rate from the cooling load DesignBuilder can use one of 2 methods:

- **1-Sensible only** the default option where the airflow rate is simply calculated from the cooling load, the supply air temperature and the zone air temperature setpoint. This method is widely used in the industry to calculate the airflow rate in cases where the latent load is a relatively low proportion of the total.

$$\text{Design flow rate} = \text{SizingFactor} \cdot \text{DesignCoolingLoad} / (T_{\text{zoneairdb}} - T_{\text{supplyair}}) \cdot \text{CP}_{\text{air}} \cdot \text{Density}_{\text{air}}$$

where:

CP_{air} is the specific heat capacity of air

$T_{\text{zoneairdb}}$ is the zone dry bulb temperature

$T_{\text{supplyair}}$ is the supply air dry-bulb temperature

- **2-Sensible + latent** where a similar enthalpy calculation is used. This method may be adopted in some cases where latent loads predominate:

$$\text{Design flow rate} = \text{SizingFactor} \cdot \text{DesignCoolingLoad} / (E_{\text{zoneair}} - E_{\text{supplyair}}) \cdot \text{Density}_{\text{air}}$$

where:

E_{zoneair} is the zone air enthalpy

$E_{\text{supplyair}}$ is the supply air enthalpy

ASHRAE Cooling Sizing

In the ASHRAE cooling sizing method the cooling design load calculations are carried out using the standard ASHRAE Heat Balance method implemented in EnergyPlus using the 'ZoneHVAC:IdealLoadsAirSystem' system type. You can define the conditions of the supply air delivered to cooled zones using Design supply temperature and Supply air humidity ratio. The total (sensible + latent) load is calculated as the enthalpy change between the zone return air and the specified supply air condition for the current load.

The ASHRAE system is modelled assuming:

- No economiser,
- No heat recovery,
- No heating,
- Zero pressure rise fan,
- No humidification/dehumidification,
- Zone plenums will not be used if specified,
- Outside airflow rate and operation schedule defined by the data under the [Mechanical Ventilation](#) and [Natural ventilation](#) headers on the [HVAC](#) tab at the zone level,
- Cooling coil availability defined by the schedule under the [Cooling](#) header on the [HVAC](#) tab at the zone level,
- Zone design cooling temperature setpoints are defined by the data under [Environmental Control](#) on the [Activity](#) tab.

Supply Air Conditions

In DesignBuilder v3.1 and later the supply air conditions used in Cooling design come from the [HVAC tab](#) of model data.

Unitary DX Cooling Sizing

When using the Unitary DX cooling sizing method the Cooling design calculation is made through a detailed EnergyPlus simulation using the EnergyPlus HVAC Template Single zone Unitary DX equipment.

Each zone has it's own Unitary system with a fan (generating no pickup), a DX cooling coil, an outside air controller and a damper to reduce supply air flow. More specifically:

- No economiser,
- No heat recovery,
- No heating coil,
- Zero pressure rise fan,
- No humidification/dehumidification,
- Zone plenums will be used if specified,
- Outside airflow rate and operation schedule defined by the data under the [Mechanical Ventilation](#) and [Natural ventilation](#) headers on the [HVAC](#) tab at the zone level,
- Cooling coil availability defined by the schedule under the [Cooling](#) header on the [HVAC](#) tab at the zone level,
- Zone design cooling temperature setpoints are defined by the data under [Environmental Control](#) on the [Activity](#) tab.

Note: Unitary DX will tend to calculate higher latent loads than the other 3 cooling sizing methods and the zone air humidity will generally be lower. This is because DX coils run colder than water-cooled coils and will condense out more water from the supply air. Unitary DX can be used to provide a conservative calculation for the latent cooling loads.

Solar Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog and General tab on [Cooling Design Calculation Options](#) dialog.

Solar Options allow you to control aspects of the model related to solar gains.

Include all buildings in shading calcs

The standard wall, floor, roof surfaces defining the current building provide shading but by default the other buildings on the site are excluded from the analysis. Use this option to include other buildings on the site as shading obstructions.

Note: there is no way to simulate more than 1 building at a time. All external surfaces in buildings other than the current one are modelled as shading surfaces. If you need to model the adjacency of another building in contact with the current one, you can use the [adiabatic](#) adjacency option to model the touching surfaces.

Note: if one or more 'other buildings' are large/complex then you could generate a large number of shading elements and the [simulation could be slow](#) (even if the current building being simulated is simple).

Include shading from excluded zones in simulation

When zones are excluded from simulations, by unchecking the "Include Zone " on page 330 checkbox, it is often still necessary for the excluded the zone's surfaces to be used to shade other parts of the building. Checking this checkbox ensures that all building surfaces are used in shading calculations, even those that have been excluded.

Model all external reflections and shading of ground reflected solar

See [help](#) for equivalent topic for simulation.

Solar distribution

See [help](#) for equivalent topic for simulation.

Output Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog, General tab on [Cooling Design Calculation Options](#) dialog and [Miscellaneous](#) model data tab

Cooling design output options apply to the whole building and are only visible at building level when accessed from the [Outputs](#) model data tab.

Include unoccupied zones in block and building averages and totals

By default, data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Store surface output

Causes the following output to be stored for all walls, floors, roofs, ceilings, partitions etc:

- Inside Surface Temperature
- Outside Surface Temperature
- Surface Heat Gain
- Solar Incident
- External Sunlit Fraction
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Store opening output

Only visible when the **Store surface output** option is selected, this option causes the following data to be stored for windows vents, doors, holes, sub-surfaces:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Gain
- Solar Incident
- Solar Transmitted
- External Sunlit Fraction
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Warning: Use the surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be stored.

Advanced Calculation Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog and General tab on the [Cooling Design Calculation](#) options dialog.

General Solution

Inside face surface temperature convergence criteria

The surface heat balance model at the inside face has a numerical solver that uses a convergence parameter for a maximum allowable differences in surface temperature. This field can optionally be used to modify this convergence criteria. The default value is 0.002 and was selected for stability. Lower values may further increase stability at the expense of longer runtimes, while higher values may decrease runtimes but lead to possible instabilities. The units are °C or °F.

Temperature and loads convergence

The temperature and loads convergence values represent the maximum allowable difference in zone temperature and loads between successive daily iterations before convergence is considered to have been reached during warmup.

Convergence of the simultaneous heat balance/HVAC solution is reached when both the loads and temperature criteria are satisfied. Both tolerances work the same way, one looks at temperatures and one looks at heating and cooling loads. After the second warm-up day, the program compares the maximum temperature experienced in a space with the maximum temperature from the previous day. If those two temperatures are within the tolerance, then it has passed the first warm-up check. It does a similar comparison with lowest temperatures experience within all the zones. If the current simulation day and the previous day values are within the tolerance, then it has passed the second warm-up check. A similar comparison is carried out with the loads tolerance and the maximum heating and cooling loads that are experienced within the spaces. Those are compared individually to the values for the previous day. If they are both in tolerance, then the simulation has passed the third and fourth warm-up check. The simulation stays in the warm-up period until ALL FOUR checks have been passed.

Convection

Inside and Outside convection algorithms

You can select from a range of EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. More details on this and external convection can be found under [Constructions](#) Model Data > [Surface Convection](#) header.

Shading

Maximum number of shadow overlaps

Enter the maximum number of figures per shadow overlap. The shadow overlaps is a measure of the amount of complexity in the shading calculation and this maximum value allows you to limit the amount of time spent in the solar initialisation calculations.

Note: entering a small value here can speed up simulations in complex buildings. If you plan to use **Maximum shadow overlaps** below the default value of 15000 you should check accuracy of solar gains relative to results using default value

Polygon clipping algorithm

This is an advanced feature. Prior to V7, the internal polygon clipping method was a special case of the Weiler-Atherton method. Now, two options are available:

- **1-Sutherland Hodgman** (default)
- **2-Convex Weiler Atherton**

Theoretically, Sutherland-Hodgman is a simpler algorithm but it works well in cases where receiving surfaces (of shadows) are non-convex. The Weiler-Atherton implementation is only accurate where both casting and receiving surfaces are convex. Warnings/severe errors are displayed when necessary. More details on polygon clipping are contained in the [Engineering Reference](#).

Other

Surfaces within zone treated as adiabatic

EnergyPlus support recommend modelling surfaces wholly contained within a zone as adiabatic and this option allows you to follow this advice. We have found that in practice this option does not make much difference in results or in simulation speed so for most cases you can leave it in its default state.

See also [EnergyPlus Errors and Warnings](#)

Note: 'surfaces contained within a zone' are frequently generated when one of the [zone merging](#) options is used where a partition or floor which would have separated two zones actually sits within the merged zone. These surfaces *do not* refer to [hanging partitions](#) which are modelled using [Internal thermal mass](#).

Simulation Calculation Options - Model Options (Building)

Simulation tab on [Model Options](#) dialog.

Data on this tab allows you to control the simulations:

- [Simulation Period](#)
- [Calculation options](#)
- [Solar options](#)
- [Advanced Calculation Options](#)
- [Output options](#)

This data is also shown before simulations in the [Simulation Calculation Options](#) dialog.

See also the [Setting up simulations](#) tutorial

Simulation Calculation Options - Simulation Options

[Simulation tab](#) on [Model Options](#) dialog and [General tab](#) on [Simulation Options](#) dialog.

Calculation Description

Enter some text to identify the calculation. This will be used in reports, on Simulation Manager dialog and in the filename for [automatically generated ESO files](#).

Tip: If you forget to update the Description before running a simulation using the Simulation Manager you can [rename](#) it later in the Simulation Manager dialog.

Simulation Period

Start day, Start month, End day, End month

Select the start and end days for the simulation by selecting the start day and month and the end day and month.

Simulation Period	
From	
Start day	17
Start month	Aug
☒ Specify year	
Start year	2024
To	
End day	23
End month	Aug
☐ Run simulation for multiple years	

Alternatively select a typical period using the links on the Info panel:

- **Annual simulation** - all 365 days of the year.
- **Summer design week** - a week identified by the weather data translator as being the hottest of the year.
- **Summer typical week** - a week identified by the weather data translator as being typical of the summer.
- **All summer** - simulate the whole summer (as identified by the weather data translator).
- **Winter design week** - a week identified by the weather data translator as being the coldest of the year.
- **Winter typical week** - a week identified by the weather data translator as being typical of the winter.
- **All winter** - simulate the whole winter (as identified by the weather data translator).

Note: The above typical simulation periods are derived from the [Statistics data](#) from the currently selected [Hourly weather data](#). If the hourly weather data was loaded from a custom weather file that did not include statistics data (the .stat file) then data may not be available to allow these links to typical periods to work. In this case you must select the dates manually.

Specify year

For some modelling work it is necessary to simulate a specific year. For example when carrying out operational analysis on an occupied building and comparing simulation results with site measurements for a specific period. In such cases a "Typical Year" weather file cannot be used and the hourly weather file should represent a specific year. In these cases, you should check this checkbox and enter the year in the **Start year** textbox below.

Start year

When the **Specify year** checkbox has been checked, you must enter the year represented by the currently selected [Hourly weather file](#).

Note: When the **Start year** is specified here it overrides the [Day of week for start day](#) site level setting, and the year set here defines the day of the week of the start day.

Run simulation for multiple years

If you would like to run the simulation for more than one year then check this box. This can be useful for models where there is very heavy thermal mass involved anywhere in the model. A typical example is when modelling ground domains where the thermal mass of the ground itself is simulated in detail. Similarly, when simulating a ground heat exchanger it can take several years for the ground temperature fields to stabilise to periodic steady-state.

Number of years

When the Run simulation for multiple years checkbox is checked you must enter the number of years to be simulated. Enter a number 2 or greater.

Note: When simulating multiple years it is only possible to run for a whole number of years, each of which start in January and end in December. So it isn't possible to simulate from May one year to June 3 years later for example.

Output Intervals For Reporting

Select one or more intervals for reporting from the available list:

- Monthly and Run period
- Daily
- Hourly
- Sub-hourly

Warning: selecting output at hourly or sub-hourly intervals can produce large amounts of data which slows processing, results in large file sizes and in extreme cases can cause DesignBuilder to fail when attempting to load results back into the model for reporting. If you want to generate hourly or sub-hourly data you may wish to consider switching off some of the [Output options](#) to ensure you are only generating the data you require.

Simulation Calculation Options - Calculation Options

[Ref641103961](#) [Simulation](#) tab on [Model Options](#) dialog and Options tab on [Simulation Options](#) dialog.

Time steps per hour

The Time steps per hour setting specifies the overall timestep for the simulation. The value entered here is also known as the Zone Timestep. This is used in the Zone Heat Balance Model calculation as the driving timestep for heat transfer and load calculations. The value entered here is the number of timesteps to use within an hour. Longer length timesteps have lower values for Number of timesteps per hour. For example a value of 6 entered here directs the program to use a zone timestep of 10 minutes and a value of 60 means a 1 minute timestep.

The user's choice for Number of timesteps per hour must be evenly divisible into 60 and the allowable choices are 1, 2, 4, 6, 10, 12, 30, and 60.

The choice made for this field has important implications for modelling accuracy and the overall time it takes to run a simulation. Here are some considerations when choosing a value:

- The solution technique used in EnergyPlus has been designed to be stable with zone timesteps of up to sixty minutes (Number Timesteps in Hour = 1). However, 60 minutes is considered a "long" timestep and it should only be used in rare occasions where there is no HVAC system, accuracy is not a concern, and short run times are critical. Such long timesteps are not recommended to use because simulation results are more accurate for shorter timesteps, of say 10 minutes or less (Number of timesteps per hour of 6 or more). Shorter zone timesteps improve the numerical solution of the Zone Heat Balance Model because they improve how models for surface temperature and zone air temperature are coupled together. Longer timesteps introduce more lag and lead to more a dampened dynamic response.
- Simulation run time increases with shorter timesteps or larger values for Number of timesteps per hour. The effect varies with the nature of the model. The user can test out different values on their particular

model to understand the implications for his or her particular case. Sometimes large models with multizone HVAC and Plant systems execute nearly as fast with 15 minute timesteps as with 60 minute timesteps because fewer iterations are required in the system modelling since the prior timestep's results are close to the final outcome of next timestep.

- The weather data files usually have 60-minute (or hourly) data. However, it does not follow that this should be used as the basis for choosing the zone timestep because:
 - o EnergyPlus carefully interpolates the weather data between data points for use at shorter timesteps.
 - o Many aspects of a model have time scales that differ from the that of the weather data. A goal of the modelling is to predict how the building will respond to the weather. However, the building's response is not governed by the time scale that the weather data are available at, but rather the time scales of the dynamic performance of the thermal envelope as well as schedules for internal gains, thermostats, and equipment availability.
- If the model will also be used to calculate the cost of electricity then you should be aware that many electric utility tariffs base charges on demand windows of a specified length of time. If the choice of Number of timesteps per hour is not consistent with the demand window, then unexpected results may be obtained. For reasonable prediction of the maximum rates for electricity use for in calculating demand charges, the length of the zone timestep needs to be consistent with the tariff's demand window. The following table lists what values are consistent with various demand windows.

Demand Window Applicable	Number of timesteps per hour
Quarter Hour	4, 12, or 60
Half Hour	2, 4, 6, 10, 12, 30, or 60
Full Hour, Day, Week	Any

There is also second type of timestep inside EnergyPlus that is known as the System or HVAC timestep. This is a variable-length timestep that governs the driving timestep for HVAC and Plant system modelling. The user cannot directly control the system timestep (except by use of the Convergence limits data). When the HVAC portion of the simulation begins its solution for the current zone timestep, it uses the zone timestep as its maximum length but then can reduce the timestep, as necessary, to improve the solution. The technical details of the approach are explained in the Engineering Documentation under "Integrated Solution Manager".

Advanced EnergyPlus users can obtain and view data at intervals of the HVAC time step used if they select the 'detailed' option on an HVAC report variable when working directly with IDF data.

Though many buildings can be successfully simulated with 1 or 2 time steps per hour, EnergyPlus suggest a minimum of 4 for non-HVAC simulations and 6 for simulations with HVAC.

20 Timesteps per hour is the minimum when using the Finite difference solution method.
[Green roof](#) simulations may also require more timesteps.

Note 1: In general, increasing the number of time steps improves accuracy but slows the simulation (and generates more data if output is requested at the 'sub-hourly' interval).

Note 2: When using **1 time steps per hour** you will not be able to access [Temperature distribution](#) results

Temperature control

Heating and cooling systems control internal temperatures to meet the setpoint temperatures specified on the [Activity tab](#). These setpoint temperatures can be interpreted as air, operative or some other radiant fraction and DesignBuilder provides corresponding options to allow HVAC systems to be controlled by:

- **1-Air temperature** - control the zone mean air temperatures to the [heating and cooling setpoint temperatures](#) specified on the [Activity tab](#).
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this option.
- **3-Other**- you can enter the radiant temperature control fraction.
- **4-Fanger comfort** - this option is only available when using Detailed HVAC. It allows you to control zone comfort conditions using [Fanger PMV heating and cooling comfort setpoints](#) instead of the usual

temperature setpoints. The PMV setpoint values are defined under the [Environmental control](#) header on the Activity tab when using the default **1-Simple HVAC Detailed HVAC Activity data** option. Alternatively, when using the **2-Detailed HVAC Detailed HVAC Activity data** option, time-varying PMV setpoints for heating and cooling are defined by Fanger PMV comfort schedules selected on the HVAC zone dialog .

When using the **3-Other** option the radiant fraction should be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Radiative Fraction vs Air Velocity

Air Velocity (m/s)	<0.2	0.2 - 0.6	0.6 - 1.0
Radiant fraction	0.5	0.4	0.3

Note: When the **2-DesignBuilder Simple HVAC autosize method** is selected, the temperature control settings used in [heating design](#) and cooling design will also apply in simulation autosizing calculations for the winter and summer design days respectively.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

You can think of the Temperature control option as:

- The radiant fraction sensed by the room thermostat, or,
- Given a room thermostat that mainly senses air temperature, the extent of automatic modification of the setpoint temperature to ensure occupant comfort.

Note: This option is overridden when using radiant heating systems by the [radiant heating system control setting](#).

Note: This option does not affect natural and mechanical [ventilation setpoints](#) - these always use **air temperature** set points.

Operative vs Air temperature control - Advanced discussion

There has been much debate amongst simulation experts over the years on the extent to which radiant effects should be included on the simulated thermostat. Most real-world thermostats probably don't actually sense more than about 20% radiant heat transfer - the temperature sensor will mainly be sensitive to the temperature of the nearby room air. So you might think that air temperature (or 20% radiant) control is the best choice. But Operative control (radiant fraction = 0.5) can be useful for calculating realistic heating and cooling energy based on published summer and winter temperature requirements for the activities in each zone. This is because HVAC systems controlled using the operative temperature continue to condition the building until comfort conditions are met (just like they are in real buildings where occupants may adjust thermostats until they are comfortable). Also, the default [temperature set points](#) from the [Activity templates](#) are generally derived from sources quoting operative temperatures. With Air temperature control the room *air* temperature is controlled to the set point temperature, which (depending on internal radiant temperatures) may not necessarily be comfortable.

The disadvantage of using **Operative** temperature control is that start up loads can be unrealistically high due to the lag in thermal response of the walls, floor, ceilings. The slow temperature response of the building fabric part governs the output of the operative thermostat and hence the operation of the heating/cooling equipment. If this effect dominates it can lead to an overestimate of the required design loads. You should be familiar with this issue before using operative temperature control to size heating and cooling equipment. In our experience, using Operative temperature control usually leads to significantly higher peak loads in Heating and Cooling design calculations and higher heating and cooling energy consumption in Simulations. DesignBuilder therefore uses air temperature control for heating and cooling autosizing simulations, even when the Operative or the Other control option is selected to avoid using

unrealistically high capacities in the simulations. You should be aware that this can lead to undersizing and unmet hours when running ASHRAE 90.1 simulations. An alternative solution to avoid undersizing when using the Operative option is to make appropriate increases sizing factors or HVAC sizing settings. Some trials and error may be needed with this.

Caution: Operative temperature control can cause [EnergyPlus Error 3](#) when using Simple HVAC, or Cooling design calculations with Operative control in zones with strong radiant heat gains causing high radiant temperatures (e.g. uninsulated roof or zone is highly glazed). The error is caused by the fixed supply air temperature being higher than the zone air temperature required to give the operative setpoint (e.g. 24°C). The solution may be to use Air temperature control and to manage the high radiant temperatures using solar shading/insulation as appropriate.

Air temperature control is easier to use as none of the aforementioned problems apply, but it can lead to inadequate equipment sizes peak loads in Heating and Cooling design calculations when not used with a design safety factor. This is especially true when radiant temperatures are very different from air temperatures for example in poorly insulated buildings, buildings with large unshaded glazing areas or high ventilation rates. Generally using air temperature control in Simulations of such buildings will underestimate energy consumption.

Autosizing simple HVAC convective systems with operative temperature control

Another issue to bear in mind with operative temperature control is that in simple HVAC convective systems, autosized cooling systems use a different algorithm to calculate the maximum supply airflow rate used in the EnergyPlus **Ideal Loads** system. In convective systems the equation used is:

$\Delta T = \text{ZoneCoolingSetPointTemperature} - \text{HVACCoolingCoilSetpointTemp}$ (*difference in temperature between zone air and supply air*)

$\text{DesignSupplyAirFlowM3PerS} = \text{DesignMaxCoolingLoad} / (\text{Cp} * \Delta T * \text{AirDensity})$

This equation works because the temperature of the air in the zone can safely be assumed to be the zone cooling set point and so there is fixed difference in temperature between zone air and supply air. Calculating the design supply cooling airflow rate in this way does not work for operative temperature control because the air temperatures in the space are often much lower than the zone cooling setpoint temperature and sometimes in zones having very high radiant temperatures, the air temperature in the zone approaches the supply air temperature. In other words the difference in temperature between zone air and supply air in the simulated system becomes very low and therefore very large airflow rates are required to meet cooling loads.

Tip: As a general rule you should prioritise checking building **comfort** levels when using Air temperature control and **realistic plant operation** (oversized equipment, supply temperatures very low) when using Operative temperature control.

Simulation Calculation Options - Solar

[Cooling Design](#) and [Simulation](#) tabs on [Model Options](#) dialog and Options tab on [Simulation Options](#) dialog.

Solar Options allow you to control aspects of the model related to solar gains.

Include all buildings in shading calcs

The standard wall, floor, roof surfaces defining the current building provide shading but by default the other buildings on the site are excluded from the analysis. Use this option to include other buildings on the site as shading obstructions.

Note: there is no way to simulate more than 1 building in the same simulation. All external surfaces in buildings other than the current one are modelled as shading surfaces. If you need to model the adjacency of another building in contact with the current one, you can use the [adiabatic](#) adjacency option to model the touching surfaces.

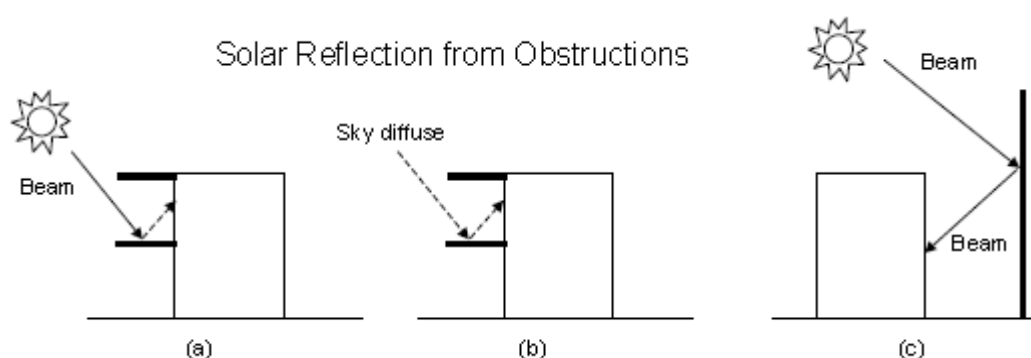
Note: if one or more of the 'other buildings' are large and/or complex then using this option could result in a large number of shading elements and the [simulation could run slowly](#) (even if the current building being simulated is simple).

Include shading from excluded zones in simulation

When zones are excluded from simulations, by unchecking the "Include Zone " on page 330 checkbox, it is often still necessary for the excluded the zone's surfaces to be used to shade other parts of the building. Checking this checkbox ensures that all building surfaces are used in shading calculations, even those that have been excluded.

Model all external reflections and shading of ground reflected solar

When this option is selected, the program calculates the beam and sky solar radiation that is reflected from exterior surfaces and then strikes the building. Also, this option enables the shadowing of ground-reflected radiation by obstructions such as the building itself and component blocks.

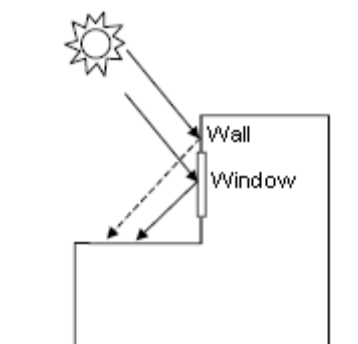


The above schematic shows solar reflection from various surface types. Solid arrows are beam solar radiation; dashed arrows are diffuse solar radiation. (a) Diffuse reflection of beam solar radiation from the top of an overhang (point 1 below). (b) Diffuse reflection of sky solar radiation from the top of an overhang (point 1 below). (c) Beam-to-beam (specular) reflection from the façade of an adjacent highly-glazed building represented by a vertical shadowing surface (point 2 below).

Reflecting surfaces fall into 3 categories:

1. Shadowing surfaces. These are surfaces of objects like overhangs, component blocks, assemblies and shading planes. These surfaces have diffuse reflectance derived from the associated material properties.

2. Exterior building surfaces. In this case one section of the building reflects solar radiation onto another section (and vice-versa). See schematic below. The building surfaces are assumed to be diffusely reflecting if they are opaque (walls, for example) and specularly reflecting if they are windows (or glass doors defined as windows). The reflectance values for opaque surfaces are calculated by the program from the [Solar Absorptance](#) and [Visible Absorptance](#) of the outer material layer of the surface's construction. The reflectance values for windows and glass doors are calculated by the program from the reflectance properties of the [individual glass layers](#) that make up surface's construction assuming no shading device is present and taking into account inter-reflections among the layers.

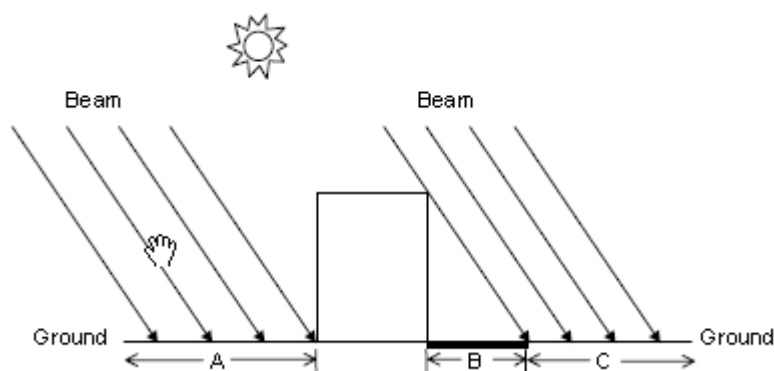


Solar reflection from building surfaces onto other building surfaces.

In this example beam solar reflects from a vertical section of the building onto a roof section. The reflection from the window is specular. The reflection from the wall is diffuse.

3. The ground surface. Reflection from the ground is calculated even if the **Model all external reflections and shading of ground reflected solar** option is not used. In this case the shadowing of the ground by the building itself or by obstructions such as neighbouring buildings is ignored. However if this option is applied then:

- Shadowing of ground reflected solar radiation by louvres, component blocks etc is taken into account, and,



- Shadowing from the building affects beam solar reflection from the ground. Beam-to-diffuse reflection from the ground onto the building occurs only for sunlit areas, A and C, not from shaded area, B.

Solar distribution

This option determines how EnergyPlus treats beam solar radiation and reflectance from exterior surfaces that strike the building and, ultimately, enter the zone. There are 3 choices:

1-Minimal shadowing - In this case, there is no exterior shadowing except from window and door reveals. All beam solar radiation entering the zone is assumed to fall on the floor, where it is absorbed according to the floor's solar absorptance. Any reflected by the floor is added to the transmitted diffuse radiation, which is assumed to be uniformly distributed on all interior surfaces. If no floor is present in the zone, the incident beam solar radiation is absorbed on all interior surfaces according to their absorptances. The zone heat balance is then applied at each surface and on the zone's air with the absorbed radiation being treated as a flux on the surface.

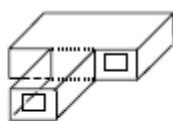
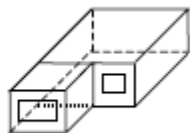
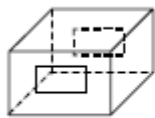
2-Full Exterior - in this case, shadow patterns on exterior surfaces caused by detached shading, wings, overhangs, and exterior surfaces of all zones are computed. As for Minimal shadowing, shadowing by window and door reveals is also calculated. Beam solar radiation entering the zone is treated as for 'Minimal shadowing' - all beam solar radiation entering the zone is assumed to fall on the floor, where it is absorbed according to the floor's solar absorptance. Any reflected by the floor is added to the transmitted diffuse radiation, which is distributed among interior surfaces according to view factors. If no floor is present in the zone, the incident beam solar radiation is absorbed on all interior surfaces according to their absorptance.

Note: If you use **2-Full Exterior** you should make sure that all of your zones have a floor. If your building model has any zones of unusual shape without floors then the EnergyPlus simulation may become unstable because it will not have a surface to apportion solar gains to.

3-Full interior and exterior - this is the same as **Full exterior** except that instead of assuming all transmitted beam solar falls on the floor the program calculates the amount of beam radiation falling on each surface in the zone, including floor, walls and windows, by projecting the sun's rays through the exterior windows, taking into account the effect of exterior shadowing surfaces and window shading devices. If this option is used, you should be sure that the surfaces of the zone totally enclose a space, i.e. that all blocks are [closed manifold](#).

Also, unless you are using the **2-Pixel counting Shading calculation method**, you should ensure that all zones in the model are convex when using this option. Examples of convex and non-convex zones are shown below. A common non-convex zone is an L-shaped zone. A formal definition of convex is that any straight line passing through the zone intercepts at most two surfaces. Another definition is that a zone is convex if every surface in a zone can 'see' every other surface - no surfaces 'around the corner' from each other.

If the above requirements are not met you should use **Full exterior** instead.



Convex zones

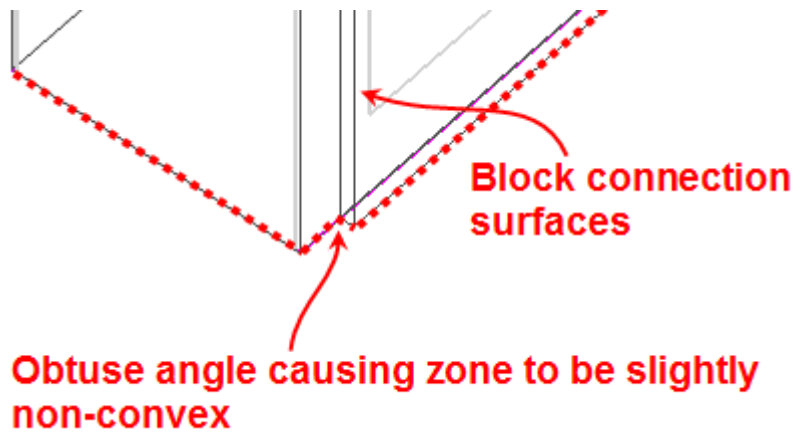
Non-Convex zones

If you use **3-Full Interior and exterior** the program will also calculate how much beam radiation falling on the inside of an exterior window (from other windows in the zone) is absorbed by the window, how much is reflected back into the zone, and how much is transmitted to the outside. In this calculation the effect of a shading device, if present, is accounted for.

Note: You must use **3-Full Interior and exterior** to account for direct solar and light transmission through internal windows.

Check for non-convex zones

When the solar distribution selection is **3-Full interior and exterior** (above) you can choose whether DesignBuilder is to check for non-convex zones. The DesignBuilder check for non-convex zones is very strict whereas the EnergyPlus check is less so. Generally EnergyPlus will tolerate a small degree of non-convex geometry. By default this option is switched on and the strict internal DesignBuilder convex check is carried out, but if the zones are only slightly non-convex you can switch off this check and allow EnergyPlus to continue calculations. You will probably receive [error message 2](#) if you switch this option off and have non-convex zones.



One common reason for the strict DesignBuilder check to flag a zone as being non-convex even when at first sight it is convex is the use of block connection geometry. In the above example of a double skin facade (DSF) cavity zone, you can see when zooming right in on the corner of a set of [block connection surfaces](#) that these surfaces cause obtuse angles to the zone geometry resulting in very slight non-convex geometry. These are exactly the sort of small deviations from convex that the DesignBuilder checker will fail and the EnergyPlus checks will pass. In this case it is perfectly safe to switch off this check to allow EnergyPlus calculations to proceed.

Shadow Calculations

The shadow calculation options are used to control some details of EnergyPlus' solar, shadowing and daylighting models. There are two basic methods available for the calculations. In order to speed up the calculations, shadowing calculations (sun position, etc.) for the default **1-Average over days in frequency** method are performed over a period of days defined by the [Shadowing interval](#).

Shadow calculation method (Simulation only)

This setting is important for determining the amount of sun entering the building and by inference the amount of cooling or heating load needed for maintaining the building. Though termed "shadowing" calculations it, in effect, determines the sun position for a particular day in a simulation (each design day will use the date of the design day object). Even though weather file data contains the amount of solar radiation, the internal calculation of sun position will govern how that affects various parts of the building. By default, the calculations are done for every 20 days (depending on the setting in [Shadowing interval](#)) throughout a weather run period; an average solar position is chosen and the solar factors (such as sunlit areas of surfaces) remain the same for that number of days. When more integrated calculations are needed for controlling dynamic windows or shades (e.g. SageGlass), a second method is available where solar calculations are performed at each zone timestep.

Choose from:

- **1-Average over days in frequency**, the default and fastest method.
- **2-Timestep frequency**, a more detailed and slower method that must be used for modelling dynamic fenestration like SageGlass and shading surfaces

Shadowing interval (Simulation only)

This numeric field defines the interval between shadowing calculations in days. It only applies if **1-Average over days in frequency** [Shadow calculation method](#) is used. Using this field will allow you to synchronize the shadowing calculations with changes in shading devices. Using the default of 20 days in each period is the average number of days between significant changes in solar position angles. For these shadowing calculations, an "average" (over the time period) of solar angles, position, equation of time are also used.

Enter the number of days (1-365) covered by each shadowing calculation.

Enter 1 if you want shadowing to be calculated every day (for greater detail but slower calculations).

Shading calculation method

Select between CPU-based polygon clipping method or the GPU-based pixel counting method. Options are:

- **1-Polygon clipping** is the default option where shading is calculated on the CPU.
- **2-Pixel counting** where the shading is calculated using the GPU hardware if it is available. If GPU hardware (or GPU emulation) is not available, a warning will be displayed and EnergyPlus will revert to Polygon clipping. Unlike Polygon clipping, Pixel counting has no limitations related to zone concavity when used with the **3-Full Interior and exterior Solar distribution** option (i.e., it can accommodate both concave and convex zones equally). Use of the Pixel counting method requires some overhead in passing instructions between the CPU and the GPU. For low numbers of shading surfaces (less than about 200 for most hardware), Polygon clipping requires less runtime than Pixel counting. However, Pixel counting runtime scales significantly better at higher numbers of shading surfaces.

Important Note: When using the [Simulation Manager](#) to run your simulations, it is not straightforward to assign a specific graphics adapter to EnergyPlus because the program that executes the simulation is set up dynamically and so cannot be selected from the Windows Graphics Settings dialog. You should consider this when attempting to use the **2-Pixel counting** method and check the EnergyPlus error file for a possible report that the Polygon clipping method was used instead.

Note: Because the [JESS](#) simulation server doesn't include the necessary GPU hardware, the **2-Pixel counting** option will not work when simulations are run on this service, and the **1-Polygon clipping** option will be used instead.

Sky diffuse modelling algorithm (Simulation only)

Two choices are available:

- **1-Simple sky diffuse modelling**, the default option performs a one-time calculation for sky diffuse properties.
- **2-Detailed sky diffuse modelling**, the required setting if you have shadowing surfaces with changing transmittance (i.e. not all opaque or not all transparent) during the year. The program checks to see if this might be the case and automatically selects this option if the shading transmittance varies. When this option is selected, a warning is generated if the [Shadowing interval](#) (above) is > 1. In general (and you should also read the previous field description), if shadowing surfaces are used with the transmittance property, you should be careful to synchronize this calculation with the scheduled occurrence of the transmittance (if any), or use 1, which will be the most accurate but will cause more time in the calculations.

This field only applies to the **1-Average over days in frequency** [Shadow calculation method](#). When the **2-Timestep frequency** method is used, the **2-Detailed sky diffuse modelling** option is always used.

Simulation Calculation Options - Advanced

[Simulation](#) tab on [Model Options](#) dialog and Options tab on [Simulation Options](#) dialog

General Solution

Solution algorithm

This option controls the overall algorithm used for all the surfaces in the simulation apart from any that may be overridden through use of the construction [Simulation solution algorithm](#) selection for particular surfaces. Select from:

- **1-CTF** - The default method used in EnergyPlus for CTF calculations is known as the state space method (Ceylan and Myers 1980; Seem 1987; Ouyang and Haghighat 1991). CTF is a sensible heat only solution not taking into account moisture storage or diffusion in the construction elements.

- **2-Finite difference** - This solution technique uses a 1-D finite difference solution in the construction elements. It is a sensible heat only solution and does not take into account moisture storage or diffusion in the construction elements. This option is required for [PCM](#) simulations and may improve accuracy for sheet metal material layers in constructions and for chilled ceilings.
- **4-Combined Heat And Moisture Finite Element (HAMT)** - A coupled heat and moisture transfer and storage solution uses a one dimensional finite difference solution in the construction elements. It requires further moisture transfer property data inputs which are defined in the "Moisture Transfer" on page 1393 component.
- **5-Moisture Penetration Depth Conduction Transfer Function (EMPD)** - A sensible heat diffusion and an inside surface moisture storage algorithm that also needs additional moisture material property data inputs which are defined in the "Moisture Transfer" on page 1393 component.

Allow individual constructions to override solution method

Check this option to allow [Simulation solution algorithm](#) settings made for Constructions to override the general setting above.

Finite Difference Settings

The settings below are required when the general [Solution algorithm](#) is set to **2-Finite Difference** or if any constructions used in the simulation [override](#) the general setting to use the finite difference algorithm.

Difference Scheme

This field determines the solution scheme used by the Conduction Finite Difference model. There are two options:

- **1-Fully implicit first order** scheme which is first order in time and is more stable over time. But it may be slower than option 2.
- **2-Crank Nicholson 2nd order** which is second order in time and may be faster than option 1 but it can be unstable over time when boundary conditions change abruptly and severely.

Space discretisation constant

The Space discretisation constant controls the how the model determines the number of nodes used to represent each material layer in the construction. The model calculates the nominal distance associated with a node, Δx , using:

$$\Delta x = (C\alpha\Delta t)^{0.5}$$

Where:

- α is the thermal diffusivity of the material layer, in m²/s
- Δt is the length of the timestep in seconds.
- C is a constant set by this field.

The default is 3. Typical values are from 1 to 3. Lower values for this constant lead to more nodes and finer-grained space discretisation.

Relaxation factor

The finite difference solver includes under-relaxation for improved stability for interactions with the other surfaces. This input field can optionally be used to modify the starting value for the relaxation factor. Larger numbers may solve faster, while smaller numbers may be more stable. The default is 1.0. If the program detects numerical instability, it may reduce the value entered here to something lower and more stable.

Inside face surface temperature convergence criteria

The surface heat balance model at the inside face has a numerical solver that uses a convergence parameter for a maximum allowable differences in surface temperature. This field can optionally be used to modify this convergence criteria. The default value is 0.002 and was selected for stability. Lower values may further

increase stability at the expense of longer runtimes, while higher values may decrease runtimes but lead to possible instabilities. The units are °C or °F.

Airflow Network

Maximum iterations

Enter the maximum number of iterations allowed in finding an Airflow Network solution. If the number of iterations at each simulation timestep is above the maximum number of iterations defined by this field, the program could not find the solution and a Severe error is issued and the program is aborted. The default value is 1000.

Relative airflow convergence tolerance

The [calculated natural ventilation](#) solution is assumed to have converged when the absolute value of the sum of the mass flow rates divided by the sum of the absolute value of the mass flow rates is less than the tolerance value entered here. The mass flow rates refer to the flow at all nodes in the airflow network model. The solution converges when both this tolerance and the tolerance in the **Absolute airflow convergence tolerance** are satisfied. The default value is 1×10^{-4} . Units are kg/s or lb/s.

Note: In cases where a large opening exists (in particular horizontal openings) this value and the absolute airflow convergence below may need to be increased by a factor of 10 or more to allow convergence to take place.

Absolute airflow convergence tolerance

The solution is assumed to have converged when the summation of the absolute value of all network airflows is less than the value specified for this input field. The default value is 1×10^{-6} .

Convection

Inside and Outside convection algorithms

You can select from a range of EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. More details on this and external convection can be found under [Constructions](#) Model Data > [Surface Convection](#) header.

Warmup

Warmup or pre-conditioning is the process of repeatedly simulating the first day before the simulation proper starts to ensure that the temperatures in the building fabric are realistic. Warmup continues until temperatures and heat flows in each zone have converged. If convergence does not occur then simulation continues for the maximum number of days as specified in the calculation option below.

Warmup and starting simulations on a weekend

Preconditioning of the building by repeated simulation of the first day until convergence is an important part of the simulation but it can cause issues if the first day in the simulation is extreme in any way. For example if the first day in the simulation is a Sunday and the building is not heated or cooled at all on that day and extreme overheating occurs then the fabric of the will be "charged" up with heat as if those conditions had been occurring repeatedly for many days (or even weeks in some heavyweight buildings). This can mean that on the next day, the Monday the building model is far more liable to overheat than would have been the case if the simulation had started on a different day. In cases where this may be an issue, it may be worth starting simulations on the first occupied day rather than on a weekend day.

Minimum number of warmup days

This data specifies the minimum number of "warmup" days before EnergyPlus will check if it has achieved convergence and can thus start running the simulation proper.

Value to use

Research into the minimum number of warmup days indicates that 6 warmup days is generally enough on the minimum end of the spectrum to avoid false predictions of convergence and thus to produce enough temperature and flux history to start EnergyPlus simulation. This was based on a study that used a set of benchmark reference buildings. It also was observed that convergence performance improved when the number of warmup days increased. As a result, the default value for the minimum warmup days has been set to 6. You should decrease this number only if you have knowledge that a specific model converges more quickly than 6 days. You may wish to increase the value in certain situations when, based on review of simulation outputs, it is determined that EnergyPlus has not converged. While this parameter should be less than the previous parameter, a value greater than the value entered in the field **Maximum number of warmup days** above may be used when users wish to increase warmup days more than the previous maximum number of days. In this particular case, the maximum value will be automatically reset to the value entered here and EnergyPlus will run exactly the number of warmup days specified here. See also Temperature and loads convergence below.

Maximum number of warmup days

The maximum number of "warmup" or pre-conditioning days that can be used in the simulation before the simulation itself starts. A warning message will occur when the warmup simulation does not converge in the allowed maximum number of warmup days:

```
Loads Initialization did not Converge (CheckWarmupConvergence)
```

This error is usually caused by using very thick constructions (e.g. ground floors). You may need to increase the **Maximum number of warmup days** to get convergence, but some anomalous buildings may still not converge.

See also Temperature and loads convergence below.

Temperature and loads convergence

The temperature and loads convergence values represent the maximum allowable difference in zone temperature and loads between successive daily iterations before convergence is considered to have been reached during warmup.

Convergence of the simultaneous heat balance/HVAC solution is reached when both the loads and temperature criteria are satisfied. Both tolerances work the same way, one looks at temperatures and one looks at heating and cooling loads. After the second warm-up day, the program compares the maximum temperature experienced in a space with the maximum temperature from the previous day. If those two temperatures are within the tolerance, then it has passed the first warm-up check. It does a similar comparison with lowest temperatures experience within all the zones. If the current simulation day and the previous day values are within the tolerance, then it has passed the second warm-up check. A similar comparison is carried out with the loads tolerance and the maximum heating and cooling loads that are experienced within the spaces. Those are compared individually to the values for the previous day. If they are both in tolerance, then the simulation has passed the third and fourth warm-up check. The simulation stays in the warm-up period until ALL FOUR checks have been passed.

Note: The [maximum number of warmup days](#) will override the above convergence criteria, i.e. the simulation proper will start even if convergence has not occurred after the maximum number of warmup days.

Shading

Maximum number of shadow overlaps

Enter the maximum number of figures per shadow overlap. The shadow overlaps is a measure of the amount of complexity in the shading calculation and this maximum value allows you to limit the amount of time spent in the solar initialisation calculations.

Note: entering a small value here can speed up simulations in complex buildings. If you plan to use **Maximum shadow overlaps** below the default value of 15000 you should check accuracy of solar gains relative to results using default value

Polygon clipping algorithm

This is an advanced feature. Prior to V7, the internal polygon clipping method was a special case of the Weiler-Atherton method. Now, two options are available:

- **1-Sutherland Hodgman** (default)
- **2-Convex Weiler Atherton**

Theoretically, Sutherland-Hodgman is a simpler algorithm but it works well in cases where receiving surfaces (of shadows) are non-convex. The Weiler-Atherton implementation is only accurate where both casting and receiving surfaces are convex. Warnings/severe errors are displayed when necessary. More details on polygon clipping are contained in the [Engineering Reference](#).

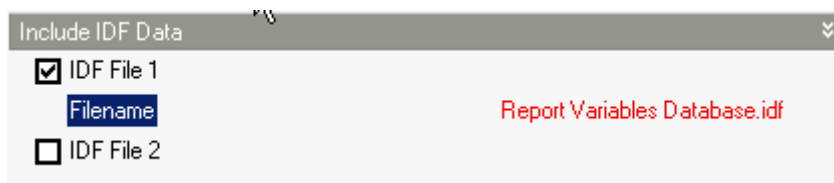
Include IDF Data

You can include up to 2 custom IDF data in your EnergyPlus simulations by checking the **IDF File 1** or **IDF File 2** check boxes and entering the name of the IDF data source file. The contents of any files specified are copied to the end of the IDF data generated by DesignBuilder before simulation. This can be a useful way to add in extra output reports, simulation objects etc, that do not interfere with the main IDF data.

For example if you needed to access output reports data not normally provided by DesignBuilder such as "Time Heating Setpoint Not Met While Occupied" then you might save a small IDF file in the DesignBuilder [EnergyPlus folder](#) called **Reports Variables Database.idf** containing this data:

```
Output:Variable, *, Time Heating Setpoint Not Met While Occupied, monthly;
Output:Variable, *, Time Cooling Setpoint Not Met While Occupied, monthly;
```

The file must be selected on this dialog as shown below.



After the simulation is finished the above setpoint not met monthly data would be included in the eso file which can be viewed outside DesignBuilder to obtain the extra outputs.

Important Note: Included IDF files must be located in the [EnergyPlus folder](#).

Tip: A more flexible way to include IDF is available through [EMS Scripting](#). Scripts are saved with the dsb file so they are completely portable. Also you can store and manage a list of various IDF datasets with EMS which makes it easier to switch various models and/or outputs in and out very easily. See "How to Generate and View Extra EnergyPlus Outputs" on page 2205 for more details on the recommended method.

Other

Surfaces within zone treated as adiabatic

EnergyPlus support recommend modelling surfaces wholly contained within a zone as adiabatic and this option allows you to follow this advice. We have found that in practice this option does not make much difference in results or in simulation speed so for most cases you can leave it in its default state.

See also [EnergyPlus Errors and Warnings](#)

Note: 'surfaces contained within a zone' are frequently generated when one of the [zone merging](#) options is used where a partition or floor which would have separated two zones actually sits within the merged zone. These surfaces *do not* refer to [hanging partitions](#) which are modelled using [Internal thermal mass](#).

Simulation Calculation Options - Output

[Simulation](#) tab on [Model Options](#) dialog and Output tab on [Simulation Options](#) dialog and [Outputs](#) model data tab.

Output Data

You can control the data generated from simulations by making selections on the Simulation tab of the Model options dialog, on the Output tab of the Simulation calculation options dialog or on the [Miscellaneous](#) model data tab.

Building and block output of zone data

This option causes average and total zone data to be generated for building and blocks. For example average temperatures, comfort conditions, total solar radiation, internal gains, fabric transmission losses etc. It does not affect the generation of energy consumption and CO2 data at the building level.

Note: It is not possible for DesignBuilder to display block level results when any type of [zone merging](#) has been enabled regardless of this setting.

Include unoccupied zones in block and building totals and averages

By default, data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

The way building and block averages are calculated is described in the [Comfort Analysis](#) section.

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Allow custom outputs

By default this option is checked and it is possible to select specific outputs for building, zones, surfaces and openings on the [Outputs](#) model data tab. The disadvantage of specifying specific output selections for each EnergyPlus object is that it can slow the simulations down, especially the initialisation period before the main simulation starts. For large models this can cause extremely long delays and in this case it is advisable to switch this option off.

When this option is switched off the "*" character is used in the Output:Variable IDF data to mean "all objects", instead of specifying outputs for each object one by one. Because of the way that EnergyPlus is structured this results in much more efficient processing.

The following limitations apply when **Allow custom outputs** is deselected:

- The Simulation output options on the [Outputs](#) model data tab can only be selected at building level and building level options apply to all zones.
- Glazing results aren't displayed on the Heat Balance output graph on the Simulation Screen.

Tip: For very large models it is usually best to uncheck this option to reduce simulation times.

Graphable Outputs - Energy

Surface heat transfer

Select this option to generate heat flow at surface boundaries. Note that selecting this option results in large amounts of data being generated as EnergyPlus must generate output for every surface and window in the zone and you should only select it if you need to look at the zone heat balance. The following EnergyPlus reports are generated:

- Opaque Surface Inside Face Conduction
- Window Heat Gain
- Window Heat Loss
- Window Transmitted Solar

For more information, see under the [Fabric and Ventilation](#) header of the "Simulation Detailed Results" on page 1622 page.

Internal gains including solar

Select this option to generate internal gains-related output and any associated energy consumption. The following EnergyPlus reports are generated:

- Zone Lights Total Heat Gain
- Zone Task Lights Total Heat Gain
- Zone People Sensible Heat Gain
- Zone Electric Equipment Cat02 Electric Consumption
- Zone Electric Equipment Cat03 Electric Consumption
- Zone Electric Equipment Cat04 Electric Consumption
- Zone Electric Equipment Cat05 Electric Consumption
- Zone Transmitted Solar
- Zone Infiltration Sensible Heat Loss
- Zone Infiltration Sensible Heat Gain
- Zone Ventilation Sensible Heat Loss
- Zone Ventilation Sensible Heat Gain

For more information, see under the [Internal Gains](#) header of the "Simulation Detailed Results" on page 1622 page.

Energy, HVAC etc

Select this option to generate energy consumption data for the HVAC system.

For more information, see under the [System Heat Flows](#) and [Fuel breakdown](#) headers of the "Simulation Detailed Results" on page 1622 page.

Latent loads

Select this option to include output of latent gains from occupants and other internals gains.

For more information, see under the [Internal Gains](#) header of the "Simulation Detailed Results" on page 1622 page.

Graphable Outputs - Comfort And Environmental

Reporting period

You can restrict the simulation *comfort and environmental* outputs using the options:

- **1-All periods** - the default option where data is generated for all periods including times when the zone is unoccupied.
- **2-Just occupied periods** - data is only output during times when the zone is occupied so daily and monthly comfort statistics become more meaningful.
- **3-User-defined schedule** - data is only output during times when the value of the **Reporting schedule** selected below has a value greater than zero.

Note that temperature distribution outputs (below) are unaffected by this setting as they have their own specific reporting period and schedule options.

Reporting schedule

Select the schedule to be used to restrict outputs when the **Reporting period** (above) is set to **3-User-defined schedule**.

Environmental

This option generates temperatures and humidity data. The following EnergyPlus reports are generated:

- Zone Mean Air Temperature
- Zone Mean Radiant Temperature
- Zone Air Relative Humidity
- Outdoor Dry Bulb

For more information, see under the [Environmental/Comfort Output](#) header of the "Simulation Detailed Results" on page 1622 page.

Fresh air supply

Select this option to include total infiltration, natural ventilation plus mechanical fresh air supply outputs.

For more information, see under the [Fabric and Ventilation](#) header of the "Simulation Detailed Results" on page 1622 page.

Simple ASHRAE Standard 55

- Time Not Comfortable Summer Clothes
- Time Not Comfortable Winter Clothes
- Time Not Comfortable Summer or Winter Clothes

For more information, see under the [Environmental/Comfort Output](#) header of the "Simulation Detailed Results" on page 1622 page.

Adaptive ASHRAE Standard 55

Checking this option causes these outputs to be generated:

- **Zone Thermal Comfort ASHRAE 55 Adaptive Model 80% Acceptability Status** - Reports whether the operative temperature falls into the 80% acceptability limits of the adaptive comfort in ASHRAE 55-2010. A value of 1 means within (inclusive) the limits, i.e. the zone operative temperature is no more than 2.5°C from than the ASHRAE 55 Adaptive model temperature. A value of 0 means outside the limits (i.e. uncomfortable), and a value of -1 means not applicable, i.e. the running average outdoor air temperature is below 10°C.
- **Zone Thermal Comfort ASHRAE 55 Adaptive Model 90% Acceptability Status** - Reports whether the operative temperature falls into the 90% acceptability limits of the adaptive comfort in ASHRAE 55-2010. A value of 1 means within (inclusive) the limits, i.e. the zone operative temperature is no more than 3.5°C from than the ASHRAE 55 Adaptive model temperature. A value of 0 means outside the limits (i.e. uncomfortable), and a value of -1 means not applicable, i.e. the running average outdoor air temperature is below 10°C.
- **Zone Thermal Comfort ASHRAE 55 Adaptive Model Running Average Outdoor Air Temperature** - the running average of the daily average outdoor dry-bulb temperatures of the previous 30 days.
- **Zone Thermal Comfort ASHRAE 55 Adaptive Model Temperature** - the ideal indoor operative temperature, or comfort temperature, as determined by the ASHRAE-55 adaptive comfort model. The 80% acceptability limits for indoor operative temperature are defined as no greater than 2.5°C from the adaptive comfort temperature. The 90% acceptability limits are defined as no greater than 3.5°C from the adaptive comfort temperature.

Note: ASHRAE 55 Adaptive outputs must be viewed in the Results Viewer - they are not available on DesignBuilder Simulation screen.

Note: Since valid ASHRAE 55 Adaptive outputs are only generated when the running mean outside air temperature is higher than 10°C, they apply mainly to summer overheating comfort analyses and so do not pick up winter underheating.

Adaptive CEN Standard 15251

Checking this option causes the following outputs to be generated:

- **Zone Thermal Comfort CEN 15251 Adaptive Model Category I Status** - reports whether the operative temperature falls into the Category I (90% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits, and a value of -1 means not applicable.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Category II Status** - reports whether the operative temperature falls into the Category II (80% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits, and a value of -1 means not applicable.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Category III Status** - reports whether the operative temperature falls into the Category III (65% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits, and a value of -1 means not applicable.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Running Average Outdoor Air Temperature** - reports the weighted average of the outdoor air temperature of the previous five days, an input parameter for the CEN-15251 adaptive comfort model.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Temperature** - reports the ideal indoor operative temperature, or comfort temperature, as determined by the CEN-15251 adaptive comfort model. Category I, II, and III limits for indoor operative temperature are defined as no greater than 2, 3, and 4°C from this value respectively.

Note: CEN Standard 15251 Adaptive outputs must be viewed in the Results Viewer - they are not available on DesignBuilder Simulation screen.

CIBSE TM52

Select this option to generate [TM52 outputs](#).

Building category

Select the building category. See "Comfort Analysis" on page 1628 for more details.

Note: [CIBSE TM52 outputs](#) must be viewed in the Results Viewer - they are currently not available on DesignBuilder Simulation screen.

CIBSE TM59

Select this option to generate [TM59 overheating analysis outputs](#). When this option is checked the following further options can be selected.

Note: TM59 results are not available on DesignBuilder Simulation screen. CIBSE [TM59:2017 outputs](#) must be viewed in the Results Viewer, and [TM59:2026 outputs](#) must be reviewed through the dedicated plugin.

Vulnerable occupants

Selecting this output will change the building category for predominantly naturally ventilated buildings according to TM59 chapter 4.2 as described in the [TM59 section](#).

Natural ventilation rules

There are 4 options to choose from:

- **1-Standard EnergyPlus controls** - applies standard natural ventilation controls as explained in the main help sections.
- **2-TM59 rules** - overrides default EnergyPlus natural ventilation behaviour in favour of the implied TM59 rule to open windows when the inside temperature is greater than 22°C, regardless of outdoor air temperature.
- **3-Approved Document O rules** - applies TM59 rules but with the [limits imposed by Approved Document O](#) automatically applied.
- **4-TM49 Scotland** - For Scotland-specific TM59 calculations.

Note: When choosing options 2 or 3, please be sure to read the [detailed discussion on this option](#) in the TM59 temperature control section, especially if you plan to use the model for energy analysis as well as for TM59 overheating analysis.

Corridors

Selecting this output will produce results according to TM59 chapter 4.5 as described in the [TM59 section](#).

Restrict venting factor by schedule

This option allows you to vary an opening's free aperture through the day, typically to hold windows more closed overnight than during the day for security and/or noise reasons. When this option is selected, the [Opening operation schedule](#) defines a fraction that multiplies the window's free aperture, so the effective opening at any hour is % Glazing area opens × schedule fraction.

Note: The [Restrict venting factor by schedule](#) option is only available for the **2-TM59 Rules** and **3-Approved Document O Rules** natural ventilation rules outputs. It is not available for the other TM59

rules options. Also, this functionality does not apply to non-TM59 natural ventilation simulations, where the natural ventilation operations schedule only defines availability, in which case, schedule values of 0 mean the window is closed, and values > 0 mean that it is fully open.

Fanger

Select this option to generate Fanger PPD and PMV outputs.

Note: PMV outputs should only be viewed on a zone by zone basis at hourly or sub-hourly intervals. When viewing averages over longer intervals the meaning of this output can easily become distorted. For example consider a simple case where there is extreme overheating in the summer (e.g. average PMV = +3) and similarly extreme underheating in the winter (e.g. average PMV = -3). The average over the year might be close to zero, but that does not mean that the building was comfortable throughout the year! For similar reasons the building average PMV should be used with extreme caution.

Pierce 2-Node

Select this option to generate the following Pierce 2-Node outputs:

- Pierce Model Effective Temperature PMV
- Pierce Model Standard Effective Temperature PMV
- Pierce Model Discomfort Index
- Pierce Model Thermal Sensation Index

Kansas State University two-node

Select this option to generate KSU Model Thermal Sensation Vote outputs using the KSU two-node thermal comfort model.

Temperature distribution

Select this option to generate [Temperature distribution](#) data for each zone.

Reporting period

You can restrict the outputs for simulation *temperature distribution* only in 2 ways using the options:

- **1-All periods** - data is included for all periods including times when the zone is unoccupied.
- **2-Just occupied periods** - the default option where data is only included for times when the zone is occupied.
- **3-User-defined schedule** - data is only included for times when the value of the **Reporting schedule** selected below has a value greater than zero.

Reporting schedule

Select the schedule to be used to restrict outputs when the **Reporting period** (above) is set to **3-User-defined schedule**.

Detailed Daylight Outputs

Daylight map output

Generates the EnergyPlus daylight map outputs.

Details of this report are shown in the [EnergyPlus Output Details document](#).

Daylight factor, illuminance and glare data

When this option is selected additional daylighting data is generated as described below.

Daylight factor data is written to the .eio file and the daylight illuminance and glare data is written to the eso file for viewing in the [Results Viewer](#).

Daylight factors

Daylight factors are ratios of interior illuminance at a specific location to exterior horizontal illuminance. In EnergyPlus, they are pre-calculated hourly for every shadow calculation day for every exterior window of a daylight zone. The pre-calculated daylight factors are interpolated between hours and for the actual sky conditions in the time-step calculation of the daylighting performance of daylight zones.

EnergyPlus reports the daylight factors for four sky types (clear, turbid clear, intermediate, and overcast) at noon time (12:00pm) for each exterior windows (base window without shading) in the eio file when the daylight factors are first calculated. This reports all hourly pre-calculated daylight factors.

Details of this report are shown in the [EnergyPlus Output Details document](#).

Daylighting Reference Point <x> Illuminance

The total daylight illuminance at the first and second reference points from all of the windows in a daylit zone.

Daylighting Reference Point <x> Glare Index

The daylight glare index at the first and second reference points in a daylit zone.

Summary Outputs

Summary output units

EnergyPlus internally calculates energy in the form of J, but since the magnitude of those numbers for most buildings is very large, conversion options are provided for the summary report tables.

Select the units for energy to be reported in the EnergyPlus summary table. Options are:

- 1-kWh - J converted into kWh (1 / 3,600,000).
- 2-None - no conversions performed, i.e. energy data is provided in J.
- 3-MJ - J converted into MJ (1 / 1,000,000)
- 4-GJ - J converted into GJ (1 / 1,000,000,000)
- 5-IP - convert all annual, monthly, economics and timebins tabular values to common Inch-Pound equivalents.

The **1-kWh**, **3-MJ** and **4-GJ** unit conversion input options apply only to the Output:Table:Monthly reports and partially to the ABUPS report. For ABUPS, the **1-kWh** option changes the report but the **3-MJ** and **4-GJ** options do not since they it is already in MJ/m² and GJ. In addition, the **1-kWh** option also changes results in the LEED Summary report, changing GJ to kWh and MJ/m² to kWh/m².

Summary Annual Reports

The easiest option is to select the **All Summary** report option which will produce all of the annual summary reports, except the Component Load Summary reports.

Including the **Component Load Summary** report will increase the simulation run time.

Also note that the **Envelope Summary**, **Surface Shadowing Summary** and **Shading Summary** reports can be very long for large models since they include entries for every surface / opening. The screenshot below shows the summary annual reports available for selection.

Summary Annual Reports

- ☐ All Summary
- ☐ LEED Summary
- ☒ Annual Building Utility Performance Summary (ABUPS)
- ☒ Demand End Use Components Summary
- ☒ Sensible heat gain summary
- ☒ Input Verification and Results Summary
- ☐ Source Energy End Use Components Summary
- ☒ Adaptive Comfort Summary
- ☐ Zone Component Load Summary
- ☒ Standard 62.1 Summary
- ☐ Energy Meters
- ☒ Climatic Data Summary
- ☒ Equipment Summary
- ☒ Envelope Summary
- ☐ Initialization Summary
- ☐ Surface Shadowing Summary
- ☐ Shading Summary
- ☒ Lighting Summary
- ☒ HVAC Sizing Summary
- ☒ System Summary
- ☒ Component Sizing Summary
- ☒ Outdoor Air Summary
- ☒ Object Count Summary
- ☐ Component Cost Economics Summary

Tip: DesignBuilder v7 doesn't provide specific options for the more recent EnergyPlus summary tables listed below, but these can still be generated by selecting the **All Summary** report option:

- Tariff Report
- Life Cycle Cost Report
- Heat Emissions Report
- Thermal Resilience Summary
- CO2 Resilience Summary
- Visual Resilience Summary

Details on the Summary Annual Reports

Annual Building Utility Performance Summary (sometimes called ABUPS)

The **Annual Building Utility Performance Summary** report often called ABUPS produces a report with an overall summary of the utility consumption of the building. It contains a number of sub tables that are each described below.

- **Site and Source Energy** – Indicates the total site and source energy use. For electricity the net electricity from the utility is used for the electric contribution. The site to source conversion factors are based on those entered by the user. These are entered in the Environmental Impact Factors and Fuel Factors data.
- **Building Area** – Shows the total floor area of the building and the conditioned floor area.
- **End Uses** – This shows the total use of electricity, natural gas, other fuels, purchased cooling, purchased heating and water for each major end-use category. The end-use categories are Heating,

Cooling, Interior Lighting, Exterior Lighting, Interior Equipment, Exterior Equipment, Fans, Pumps, Heat Rejection, Humidification, Heat Recovery, Hot Water, Refrigeration, and Generators. Not all fuels have corresponding end uses. The values in this sub-table are from output meters. To determine which components are attached to each end-use meter, consult the meter details output file (*.mtd). The source of the resource does not affect this table – the amount of electricity used for lights does not change if the electricity is from the utility or from an on-site generator. The district heating column also includes steam.

- **End Uses By Subcategory** – Shows a breakdown of the major end uses by user-defined category. If an end-use subcategory was not input for an object, it is automatically added to the General subcategory for that major end-use category.
- **Utility Use Per Floor Area** – These two sub-tables show the results from the end-uses table divided by the total floor area defined for the building and for the total conditioned floor area. Only three categories for end-uses are used for these sub-tables, lighting, HVAC and other. HVAC includes fans, pumps, heating, cooling, heat rejection, humidification, and domestic hot water heating. The district heating column also includes steam.
- **Electric Loads Satisfied** – Shows the different methods that electric loads are satisfied in the building. The values shown for on site power generation are: Fuel-Fired Power Generation, High Temperature Geothermal, Photovoltaic Power, and Wind Power. The flows to and from the electric utility are shown next and finally the total electricity used at the site is compared to the total generated on site plus the net amount from the electric utility. The percentages shown are based on the total electricity used by the end-uses.
- **On-Site Thermal Sources** – Shows the on-site thermal sources of energy such as Water-Side Heat Recovery, Air to Air Heat Recovery for Cooling, Air to Air Heat Recovery for Heating, High-Temperature Geothermal, Solar Water Thermal, Solar Air Thermal.
- **Water Loads Summary** – Shows the different methods the water loads were satisfied.

Input Verification and Results Summary (or IVRS)

The [Input Verification and Results Summary](#) report produces a report with several tables including:

- **General** which includes general information like the Program Version and Build, Weather, Latitude, Longitude, Elevation, Time Zone, North Axis Angle, and Hours Simulated.
- **Window-Wall Ratio** table for envelope which includes the wall area, the window area and the ratio of the two. These are computed for all walls and for walls that are oriented generally north, south, east and west. All walls are categorized into one of these four cardinal directions. This is computed for walls that have a tilt of 60 to 120 degrees.
- **Skylight-Roof Ratio** table for envelope which includes the roof area and the skylight area and the ratio of the two. This includes all surfaces with a tilt of less than 60 degrees.
- **Zone Summary** includes internal load summary for each zone including area, if conditioned, volume, multipliers, above ground gross wall area, underground gross wall area, window area, design lighting, design people, and design plug and process.
- **Hybrid Model: Internal Thermal Mass** table is only shown when hybrid model: internal thermal mass simulation. This includes Hybrid Modeling (Y/N) for internal mass and Temperature Capacitance Multiplier values for each zone.

Source Energy End Use Components Summary

The [Source Energy End Use Components Summary](#) report produces a report that includes three tables. These tables display source energy by fuel type that is calculated based on site to source energy factors specified by the user in the EnvironmentalImpactFactors and FuelFactors objects. The last two tables display the source energy in terms of area normalized metrics. Following is a description of each table:

- **Source Energy End Use Components** – This shows the total use of source electricity, source natural gas, source values of other fuels, source purchased cooling and purchased heating for each major end-use category. The end-use categories are Heating, Cooling, Interior Lighting, Exterior Lighting, Interior Equipment, Exterior Equipment, Fans, Pumps, Heat Rejection, Humidification, Heat Recovery, Hot Water, Refrigeration, and Generators. Not all fuels have corresponding end uses. The values in this sub-table are from output meters. To determine which components are attached to each end-use meter, consult the meter details output file (*.mtd). The source of the resource will affect this table – the amount of electricity used for lights will change if the electricity is from the utility or from an on-site generator. The district heating column also includes steam.

- **Source Energy End Use Components normalized by Conditioned Floor Area** – This table shows the total end uses in source energy normalized by conditioned floor area.
- **Source Energy End Use Components normalized by Total Floor Area** – This table shows the total end uses in source energy normalized by total floor area.

Climatic Data Summary

The **Climatic Data Summary** includes the main details on each of the design days including: maximum dry-bulb, daily temperature range, humidity value, humidity type, wind speed, and wind direction.

Envelope Summary

The **Envelope Summary** includes the following tables:

- **Opaque** which includes all opaque surfaces and includes the name of the construction, reflectance, U-Factor, gross area, azimuth, tilt, cardinal direction.
- **Fenestration** which includes all non-opaque surfaces and includes the name of the construction, areas (glass, frame, divider, single opening, multiplied openings), U-Factor, SHGC (the solar heat gain coefficient based on summer conditions), visible transmittance, conductance (frame, divider), indication of shade control, the name of the parent surface, azimuth, tilt, cardinal direction.

Surface Shadowing Summary

The **Surface Shadowing Summary** includes two tables. Note that surfaces starting with “Mir-” are automatically generated by EnergyPlus and are the mirror images of user entered surfaces.

- **Surfaces** (Walls, Roofs, etc) that may be Shadowed by Other Surfaces and includes the name of the surface and a list of surfaces that may possibly cast shadows on that named surface. The list of possible shadow casters does not necessarily mean that they do cast shadows during the simulation, only that their relative position makes it possible that shadows from a surface in the list may fall on the named surface.
- **Subsurfaces** (Windows and Doors) that may be Shadowed by Surfaces, includes the name of the subsurface such as a window or a door and a corresponding list of surfaces that may be casting shadows on the windows and doors.

Shading Summary

The **Shading Summary** includes the following tables:

- **Sunlit Fraction** which shows a list of windows and the fraction of the window that is sunlit for nine specific times of the year. The nine specific times include 9am, noon and 3pm on March 21, June 21, and December 21. These nine times were chosen to represent the range of sun angles. The simulation must include those times for the value to be included in the report.
- **Window control** includes the names of all windows that have a window shading control (see WindowShadingControl) and includes the name of the control, the type of shading, the shaded construction, the kind of control, and if glare control is used.

Lighting Summary

The **Lighting Summary** includes the following tables:

- **Interior Lighting** which includes the name of the lights object, the zone that it is used in, the lighting power density, zone area, total power, end use subcategory, schedule name, average hours per week, return air fraction, and whether the zone is conditioned.
- **Daylighting** which includes the names of the daylighting objects, the zone they are used in, the type of daylighting being used, the control type, the fraction of the lighting controlled, the total power of the lighting installed in the zone and the lighting power that is controlled by the daylighting object.
- **Exterior Lighting** which includes the name of the ExteriorLights object, the total watts described by the object, if the ExteriorLights uses an astronomical clock or just a schedule, the name of the

schedule used, and the average hours per week for the named schedule for the year. The effect of the astronomical clock does not get included in the averaged hours per week shown.

Equipment Summary

The **Equipment Summary** includes some details on the major HVAC equipment present. The report has seven parts.

- **Central Plant** includes details on chillers, boilers, and cooling towers including the capacity and efficiency. For Chiller:Electric:EIR and Chiller:Electric:ReformulatedEIR, IPLV at AHRI standard test conditions is reported.
- **Cooling Coils** includes the nominal total, sensible and latent capacities, the nominal sensible heat ratio, the nominal efficiency, nominal UA value, and nominal surface area for each cooling coil. These values are calculated by calling the cooling coil simulation routine with the rated inlet conditions: inlet air dry bulb temperature = 26.67C, inlet air wet bulb temperature = 19.44C, inlet chilled water temperature = 6.67C.
- **DX Cooling Coils** summarizes the Standard Rating (Net) Cooling Capacity, SEER, EER and IEER values at AHRI standard test. Currently, these values are only reported for coil type = Coil:Cooling:DX:SingleSpeed and Coil:Cooling:DX:MultiSpeed with condenser type = AirCooled. However, the EER value is not reported for the multi-speed DX cooling coil. There are two SEER values reported: SEER and SEER Default. The SEER value is calculated using user specified Part Load Factor (PLF) curve used for energy performance calculation and the SEER Default value is calculated using the AHRI Standard 210/240-2008 default PLF curve and cooling coefficient of degradation value of 0.25.
- **DX Heating Coils** summarizes the High Temperature Heating Standard (Net) Rating Capacity, Low Temperature Heating Standard (Net) Rating Capacity and Heating Seasonal Performance Factor (HSPF) values at AHRI standard test. Currently, these values are only reported for coil type = Coil:Heating:DX:SingleSpeed.
- **Heating Coils** includes the nominal capacity and efficiency for each heating coil. The capacity is calculated by calling the heating coil simulation routine at the rated inlet conditions: inlet air dry bulb temperature = 16.6C, inlet relative humidity = 50%, inlet hot water temperature = 82.2C.
- **Fan** includes the type of fan, the total efficiency, delta pressure, max flow rate, motor heat in air fraction, and end use.
- **Pumps** includes the type of pump, control type, head pressure, electric power, and motor efficiency for each pump.
- **Service Water Heating** includes the type of water heater, the storage volume, input, thermal efficiency, recovery efficiency, and energy factor.

HVAC Sizing Summary

The **HVAC Sizing Summary** includes the following tables:

- **Zone Cooling** which includes the following columns for each zone: the calculated design load, the user specified design load, the calculated design air flow, the user specified design air flow, the name of the sizing period, the time of the peak load during the sizing period, the temperature at the time of the peak load during the sizing period, and the humidity ratio at the time of the peak load during the sizing period used.
- **Zone Heating** which includes the following columns for each zone: the calculated design load, the user specified design load, the calculated design air flow, the user specified design air flow, the name of the sizing period, the time of the peak load during the sizing period, the temperature at the time of the peak load during the sizing period, and the humidity ratio at the time of the peak load during the sizing period used.
- **System Design Air Flow Rates** which includes the following columns for each air loop: the calculated cooling air flow rate, the user specified air flow rate for cooling, the calculated heating air flow rate, the user specified air flow rate for heating.
- **Coil Sizing Summary** which includes the following types of information for each coil: coil type, HVAC location, capacity and flow rates at the zone peak, system peak, and final values, design day name and time for the zone peak and system peak, zone conditions at the system peak, etc. This table is a subset of the Coil Sizing Details table. See the Output Details and Examples document for a complete description of each column.

Note: values listed as “calculated” are the unaltered result of the zone or system sizing calculations, using the design sizing period weather and schedules specified in the input. Values listed as “user specified” are either the calculated values modified by global or zone sizing factors or values specified with the flow/zone or flow/system design air flow method.

Component Sizing Summary

The **Component Sizing Summary** includes different tables depending on the kinds of HVAC components that exist in the input file. A table is shown for each type of HVAC component that is sized. The table lists the objects of that type of component that appear in the input file and one or more parameters related to that component. For example, the AirTerminal:SingleDuct:VAV:Reheat component creates a table showing the maximum air flow rate from the sizing calculations and the maximum reheat water flow rate. Another example is the Fan:VariableVolume object which shows a table with both the maximum and minimum flow rates for each fan based on the results from the sizing calculations.

Coil Sizing Details

The **Coil Sizing Details** table includes the following information for each coil: coil type, HVAC location, capacity and flow rates at the zone peak, system peak, and final values, design day name and time for the zone peak and system peak, zone conditions at the system peak, coil entering and leaving conditions, supply fan information, related plant sizing information, etc. See the Output Details and Examples document for a complete description of each column.

Outdoor Air Summary

The **Outdoor Air Summary** report includes the following tables:

- **Average Outside Air During Occupied Hours** table shows for each zone the average and nominal number of occupants, the zone volume, the average air change rate based on mechanical ventilation, infiltration and simple ventilation during occupied hours.
- **Minimum Outside Air During Occupied Hours** table shows for each zone the average and nominal number of occupants, the zone volume, the minimum air change rate based on mechanical ventilation, infiltration and simple ventilation during occupied hours.

System Summary

The **System Summary Report** includes the following tables:

- **Economizer** which includes the following columns for each Controller:OutdoorAir object: the high limit shutoff control, the minimum outdoor air flow, the maximum outdoor air flow, if the return air temperature has a control limit, if the return air has an enthalpy limit, the outdoor air temperature limit, and the outdoor air enthalpy limit.
- **Demand Controlled Ventilation** table is for each Controller:MechanicalVentilation object and shows the name, the nominal outdoor air per person and the nominal outdoor air per zone area.
- **Time Not Comfortable Based on Simple ASHRAE 55-2004** table shows how many hours that the space is not comfortable for each zone under the criteria of assuming winter clothes, summer clothes or both summer and winter clothes. See the People object for more information about this thermal comfort calculation.
- **Time Setpoint is Not Met** table shows how many hours the space is more than 0.2C from the setpoint during heating and during cooling. The last two columns indicate those hours that the setpoint is not met while the space is occupied.

Adaptive Comfort Summary

The **Adaptive Comfort Summary** report tabulates the sum of occupied hours not meeting adaptive comfort acceptability limits for each relevant People object (People objects for which adaptive comfort calculations are requested). These acceptability limits include ASHRAE Std. 55 80%, ASHRAE Std. 55 90%, CEN-15251 Category I, CEN-15251 Category II, and CEN-15251 Category III.

Sensible Heat Gain Summary

The **Sensible Heat Gain Summary** provides results for each zone and the overall building for some of the major heat gain components. The first four columns show the loads satisfied by sensible air heating and cooling as well as radiant heating and cooling surfaces in the zone. The heat gains from people, lighting, equipment, windows, interzone air flow, and infiltration are shown when adding heat to the zone and separately when removing heat from the zone (for applicable components). Finally the balance is shown as “Opaque Surface Conduction and Other Heat Addition” and “Opaque Surface Conduction and Other Heat Removal” which is a term indicating the affect of the walls, floors and ceilings/roof to the zone as well as the impact of the delay between heat gains/losses and loads on the HVAC equipment serving the zone. The following shows each output variable that is used for each column. For each timestep in the simulation, positive values are shown as additions and negative values are shown as removal for most variables.

HVAC Input Sensible Air Heating

- Zone Air Heat Balance System Air Transfer Rate
- Zone Air Heat Balance System Convective Heat Gain Rate

HVAC Input Sensible Air Cooling

- Zone Air Heat Balance System Air Transfer Rate
- Zone Air Heat Balance System Convective Heat Gain Rate

HVAC Input Heated Surface Heating

- Zone Radiant HVAC Heating Energy
- Zone Ventilated Slab Radiant Heating Energy

HVAC Input Cooled Surface Cooling

- Zone Radiant HVAC Cooling Energy
- Zone Ventilated Slab Radiant Cooling Energy

People Sensible Heat Addition

- Zone People Sensible Heating Energy

Lights Sensible Heat Addition

- Zone Lights Total Heating Energy

Equipment Sensible Heat Addition & Equipment Sensible Heat Removal

- Zone Electric Equipment Radiant Heating Energy
- Zone Gas Equipment Radiant Heating Energy
- Zone Steam Equipment Radiant Heating Energy
- Zone Hot Water Equipment Radiant Heating Energy
- Zone Other Equipment Radiant Heating Energy
- Zone Electric Equipment Convective Heating Energy
- Zone Gas Equipment Convective Heating Energy
- Zone Steam Equipment Convective Heating Energy
- Zone Hot Water Equipment Convective Heating Energy
- Zone Other Equipment Convective Heating Energy

Window Heat Addition & Window Heat Removal

- Zone Windows Total Heat Gain Energy

Interzone Air Transfer Heat Addition & Interzone Air Transfer Heat Removal

- Zone Air Heat Balance Interzone Air Transfer Rate

Infiltration Heat Addition & Infiltration Heat Removal

- Zone Air Heat Balance Outdoor Air Transfer Rate

The Opaque Surface Conduction and Other Heat Addition and Opaque Surface Conduction and Other Heat Removal columns are also calculated on a timestep basis as the negative value of the other removal and gain columns so that the total for the timestep sums to zero. These columns are derived strictly from the other columns.

Component Load Summary

Three different component load summary reports can be generated:

- Zone Component Loads Summary
- AirLoop Component Loads Summary
- Facility Component Loads Summary

The AirLoop and Facility level reports are generally aggregations of the results reported in the Zone Component Loads Summary report. The Component Loads Summary reports provide an estimate of the heating and cooling peak loads for each zone, airloop or the entire facility broken down into various components. These reports may help determine which components of the load have the largest impact for the heating and cooling peak design conditions. When specified, the Zone Component Loads Summary report is created for each zone that is conditioned. Similarly, the AirLoop Component Load Summary is generated for each AirLoop and a single Facility Component Loads Summary is generated for the entire facility. The difference between the peak design sensible load and the estimated instant + delayed sensible load (as shown in the Peak Conditions subtable) is an indication of how consistent the overall total estimate may be to the computed total peak loads for the zone. When the report is called the zone sizing calculations are repeated twice so this may result in longer simulation times. The keys used to obtain these reports are ZoneComponentLoadSummary, AirLoopComponentLoadSummary, and FacilityComponentLoadSummary. Since including this report may increase the simulation time, new key options have been added that will display all reports but the Zone Component Load Summary those keys used are AllSummaryButZoneComponentLoad and AllSummaryAndMonthlyButZoneComponentLoad.

The report has six parts:

1. Estimated Cooling Peak Load Components

Contains the sensible-instant, sensible-delay, sensible-return air, latent, total and %grand total for people, lights, equipment, refrigeration, water use equipment, HVAC equipment loads, power generation equipment, infiltration, zone ventilation, interzone mixing, roof, interzone ceiling, other roof, exterior wall, interzone wall, ground contact wall, other wall, exterior floor, interzone floor, ground contact floor, other floor, fenestration conduction, fenestration solar, opaque door. The values in the sensible-delay column are estimated using a procedure shown in the Engineering Reference. Also shown are the related areas for each type of component. For People and Lights the floor area is shown but for walls, fenestration. and other surfaces, the area of that surface is used.

2. Cooling Peak Conditions

Contains the time of the peak load and the outside dry bulb and wet bulb temperatures as well as the outside humidity ratio for that time. It also shows the zone temperature and relative humidity and humidity ratio for that time. The airflow and outside airflow as well as the supply air temperature are shown. Also the sensible peak load accounting for the sizing factor is shown along with the difference due to the sizing factor.

3. Engineering Checks for Cooling

This table shows some ratios that may be handy in evaluating the results. It contains the percentage of outside air, the airflow per floor area, the airflow per capacity, the floor area per capacity and the capacity per floor area. It also shows the number of people.

4. Estimated Heating Peak Load Components

Contains the sensible-instant, sensible-delay, sensible-return air, latent, total and %grand total for people, lights, equipment, refrigeration, water use equipment, HVAC equipment loads, power generation equipment, infiltration, zone ventilation, interzone mixing, roof, interzone ceiling, other roof, exterior wall, interzone wall, ground contact wall, other wall, exterior floor, interzone floor, ground contact floor, other floor, fenestration conduction, fenestration solar, opaque door. The values in the sensible-day column are estimated using a procedure shown in the Engineering Reference. Also shown are the related areas for each type of component. For People and Lights the floor area is shown but for walls, fenestration. and other surfaces, the area of that surface is used.

5. Heating Peak Conditions

Contains the time of the peak load and the outside dry bulb and wet bulb temperatures as well as the outside humidity ratio for that time. It also shows the zone temperature and the relative humidity and humidity ratio for that time. The airflow and outside airflow as well as the supply air temperature are shown. Also the sensible peak load accounting for the sizing factor is shown along with the difference due to the sizing factor

6. Engineering Checks for Heating

This table shows some ratios that may be handy in evaluating the results. It contains the percentage of outside air, the airflow per floor area, the airflow per capacity, the floor area per capacity and the capacity per floor area. It also shows the number of people.

The "Air Loop Component Summary" tables also include tables that indicate the zones included in the aggregated results for heating and cooling.

If the time of the peak load for each Zone for cooling exactly matches the time of the peak load for the AirLoop or Facility than the Estimated Cooling Peak Load Components will represent a sum of the values from the corresponding zones. Likewise the Estimated Heating Peak Load Components will add up for the AirLoop or Facility if the times of the heating peaks exactly match. This is not necessarily the case for Peak Conditions or the Engineering Checks tables. Since the sizing of Air loops is based on the system sizing they usually will be different than the sum of the corresponding zones.

Note: generating the **Component Load Summary** report will cause simulations to take longer.

Standard 62.1 Summary

The **Standard 62.1 Summary** produces a report that is consistent with many of the outputs needed when doing calculations consistent with ASHRAE Standard 62.1-2010. The report is generated when sizing calculations are specified. The abbreviations used in the report are consistent with the abbreviations used in Appendix A4 of the Standard. The following tables are part of the report:

- **System Ventilation Requirements for Cooling** containing: Sum of Zone Primary Air Flow - Vpz-sum, System Population - Ps, Sum of Zone Population - Pz-sum, Occupant Diversity - D, Uncorrected Outdoor Air Intake Airflow - Vou, System Primary Airflow - Vps, Average Outdoor Air Fraction - Xs, System Ventilation Efficiency - Ev, Outdoor Air Intake Flow - Vot, Percent Outdoor Air - %OA.
- **System Ventilation Requirements for Heating** containing: Sum of Zone Primary Air Flow - Vpz-sum, System Population - Ps, Sum of Zone Population - Pz-sum, Occupant Diversity - D, Uncorrected Outdoor Air Intake Airflow - Vou, System Primary Airflow - Vps, Average Outdoor Air Fraction - Xs, System Ventilation Efficiency - Ev, Outdoor Air Intake Flow - Vot, Percent Outdoor Air - %OA.
- **Zone Ventilation Parameters** containing: AirLoop Name, People Outdoor Air Rate - Rp, Zone Population - Pz, Area Outdoor Air Rate - Ra, Zone Floor Area - Az, Breathing Zone Outdoor Airflow - Vbz, Cooling Zone Air Distribution Effectiveness - Ez-clg, Cooling Zone Outdoor Airflow - Voz-clg, Heating Zone Air Distribution Effectiveness - Ez-htg, Heating Zone Outdoor Airflow - Voz-htg.
- **System Ventilation Parameters** containing: People Outdoor Air Rate - Rp, Sum of Zone Population - Pz-sum, Area Outdoor Air Rate - Ra, Sum of Zone Floor Area - Az-sum, Breathing Zone Outdoor Airflow - Vbz, Cooling Zone Outdoor Airflow - Voz-clg, Heating Zone Outdoor Airflow - Voz-htg.
- **Zone Ventilation Calculations** for Cooling Design containing: AirLoop Name, Box Type, Zone Primary Airflow - Vpz, Zone Discharge Airflow - Vdz, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Primary Outdoor Air Fraction - Zpz, Primary Air Fraction - Ep, Secondary Recirculation Fraction- Er, Supply Air Fraction- Fa, Mixed Air Fraction - Fb, Outdoor Air Fraction - Fc, Zone Ventilation Efficiency - Evz.
- **System Ventilation Calculations** for Cooling Design containing: Sum of Zone Primary Airflow - Vpz-sum, System Primary Airflow - Vps, Sum of Zone Discharge Airflow - Vdz-sum, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Zone Ventilation Efficiency - Evz-min.
- **Zone Ventilation Calculations for Heating Design** containing: AirLoop Name, Box Type, Zone Primary Airflow - Vpz, Zone Discharge Airflow - Vdz, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Primary Outdoor Air Fraction - Zpz, Primary Air Fraction - Ep,

Secondary Recirculation Fraction- Er, Supply Air Fraction- Fa, Mixed Air Fraction – Fb, Outdoor Air Fraction – Fc, Zone Ventilation Efficiency – Evz.

- **System Ventilation Calculations for Heating Design** containing: Sum of Zone Primary Airflow - Vpz-sum, System Primary Airflow – Vps, Sum of Zone Discharge Airflow - Vdz-sum, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Zone Ventilation Efficiency - Evz-min.

Note: The **Standard ASHRAE 62.1 Summary** report is available for generation when using Detailed HVAC or Simple HVAC with the default **1-EnergyPlus** [Simple HVAC autosizing method](#).

Energy Meters Summary

The **Energy Meters Summary** (which is a slight misnomer as some meters may not be strictly energy) provides the annual period (runperiod) results for each meter (reference the meter data dictionary file (.mdd) and/or the meter details file (.mtd). The results are broken out by fuel type (resource type) in this report.

Initialization Summary

The **Initialization Summary** provides the information shown in the initialization output file (.eio) but in a tabular format. Because of this, some of the outputs may be difficult to understand without referencing the documentation on the .eio file that is located in the Output Details and Examples document.

LEED Summary

The **LEED Summary** report provides many of the simulation results required for certification of Energy and Atmosphere Credit 1 Optimized Energy Performance according to the LEED Green Building Rating System™.

Tariff Report

The **Tariff Report** provides the results of the [Tariff Analysis](#) simulation. The report consists of a summary table, categories, charges, ratchets, qualifies, native variables, other variables and computation.

Economic Result Summary

The **Economic Result Summary** provides a summary of the Tariff Reports. The report consists of annual costs and costs for each tariff considered and why it was or was not selected.

Component Cost Economics Summary

The **Component Cost Economics Summary** provides the construction cost estimate summary for the project. The costs are broken into eight categories and the reference building costs are provided as a comparison. A second table is also produced that provides line item details with one line for every line item object.

Life Cycle Cost Report

The **Life Cycle Cost Report** provides a summary of the information from the Life Cycle Cost calculations. This report shows the costs and the timing of costs, often called “cash flows,” along with the present value in several different tables. The tabular results show the present value of all current and future costs.

Heat Emissions Report

The **Heat Emissions Report** provides a summary of the building heat emission to the ambient air. This report shows the heat emissions from buildings by surface heat transfer, zone exhaust, zone relief, HVAC relief, HVAC heat rejection, as well as the total emissions.

Thermal Resilience Summary

The **Thermal Resilience Summary** provides the indoor thermal resilience metrics unmet hours and occupant-hours summary report. This includes the Heat Index Hours and OccupantHours, Humidex Hours and OccupantHours, Heating SET Hours and Cooling SET Hours reports. The Heating SET Hours and Cooling SET Hours are reported only when the Pierce model are chosen for thermal comfort calculations. To activate the Pierce SET calculation in EnergyPlus, users need to define the People and chose Pierce as the thermal comfort method.

CO2 Resilience Summary

The **CO2 Resilience Summary** provides the indoor CO2 level unmet hours and occupant-hours summary report for resilience. This includes the CO2 Level Hours and OccupantHours reports. To activate the CO2 concentration calculation in EnergyPlus, the [Simulate CO2 concentration](#) site level option must be selected and a schedule of outdoor air CO2 levels selected.

Visual Resilience Summary

The **Visual Resilience Summary** provides the indoor illuminance level unmet hours and occupant-hours summary report for resilience. This includes the Illuminance Level Hours and OccupantHours reports. To activate the indoor illuminance calculation in EnergyPlus, it is necessary to select the "Lighting control" on page 461 option.

Summary Monthly Reports

Select from a range of summary monthly reports:

Summary Monthly Reports

- ☐ All Monthly
- ☐ Zone Cooling Summary Monthly
- ☐ Zone Heating Summary Monthly
- ☐ Zone Electric Summary Monthly
- ☐ Space Gains Monthly
- ☐ Peak Space Gains Monthly
- ☐ Space Gain Components At Cooling Peak Monthly
- ☐ Energy Consumption Electricity Natural Gas Monthly
- ☐ Energy Consumption Electricity Generated Propane Monthly
- ☐ Energy Consumption Diesel Fuel Oil Monthly
- ☐ Energy Consumption District Heating Cooling Monthly
- ☐ Energy Consumption Coal Gasoline Monthly
- ☐ End Use Energy Consumption Electricity Monthly
- ☐ End Use Energy Consumption Natural Gas Monthly
- ☐ End Use Energy Consumption Diesel Monthly
- ☐ End Use Energy Consumption FuelOil Monthly
- ☐ End Use Energy Consumption Coal Monthly
- ☐ End Use Energy Consumption Propane Monthly
- ☐ End Use Energy Consumption Gasoline Monthly
- ☒ End Use Energy Consumption Other Fuels Monthly
- ☐ Peak Energy End Use Electricity Part1 Monthly
- ☐ Peak Energy End Use Electricity Part2 Monthly
- ☐ Electric Components Of Peak Demand Monthly
- ☐ Peak Energy End Use Natural Gas Monthly
- ☐ Peak Energy End Use Diesel Monthly
- ☐ Peak Energy End Use Fuel Oil Monthly
- ☐ Peak Energy End Use Coal Monthly
- ☐ Peak Energy End Use Propane Monthly
- ☐ Peak Energy End Use Gasoline Monthly
- ☒ Peak Energy End Use Other Fuels Monthly
- ☐ Setpoints Not Met With Temperatures Monthly
- ☐ Comfort Report Simple 55 Monthly
- ☐ Unglazed Transpired Solar Collector Summary Monthly
- ☐ Occupant Comfort Data Summary Monthly
- ☐ Chiller Report Monthly
- ☐ Tower Report Monthly
- ☐ Boiler Report Monthly
- ☐ DX Report Monthly
- ☐ Window Report Monthly
- ☐ Window Energy Report Monthly
- ☐ Window Zone Summary Monthly
- ☐ Window Energy Zone Summary Monthly
- ☐ Average Outdoor Conditions Monthly
- ☐ Outdoor Conditions Maximum Dry Bulb Monthly
- ☐ Outdoor Conditions Minimum Dry Bulb Monthly
- ☐ Outdoor Conditions Maximum Wet Bulb Monthly
- ☐ Outdoor Conditions Maximum Dew Point Monthly
- ☐ Outdoor Ground Conditions Monthly
- ☐ Window AC Report Monthly
- ☐ Water Heater Report Monthly
- ☐ Generator Report Monthly
- ☐ Daylighting Report Monthly
- ☐ Coil Report Monthly
- ☐ Plant Loop Demand Report Monthly
- ☐ Fan Report Monthly
- ☐ Pump Report Monthly
- ☐ Cond Loop Demand Report Monthly
- ☐ Zone Temperature Oscillation Report Monthly
- ☐ Air Loop System Energy And Water Use Monthly
- ☐ Air Loop System Component Loads Monthly
- ☐ Air Loop System Component Energy Use Monthly
- ☐ Mechanical Ventilation Loads Monthly

Information about the various monthly summary reporting options can be found in the [EnergyPlus Output Details And Examples](#) documentation.

Miscellaneous Outputs

HVAC System temperatures

Check this option if you would like DesignBuilder to generate temperature output for each air and water node in the HVAC system. This output data cannot be displayed directly in DesignBuilder but can be viewed in the separate [Results Viewer](#) application.

HVAC System mass flow rates

Check this option if you would like DesignBuilder to generate mass flow output for each air and water node in the HVAC system. This output data cannot be displayed directly in DesignBuilder but can be viewed in the separate [Results Viewer](#) application.

HVAC System humidity ratios

Check this option if you would like DesignBuilder to generate humidity ratio output for each air node in the HVAC system. This output data cannot be displayed directly in DesignBuilder but can be viewed in the separate [Results Viewer](#) application.

SQLite output

Output from EnergyPlus can be written to SQLite files for manipulation outside of DesignBuilder. When this option is selected all of the predefined database tables as well as time series related data is included in the SQLite output file [eplusout.sql](#). The description for SQLite outputs is described fully in the EnergyPlus Output Details document.

Tip: SQLite outputs can be processed by the DesignBuilder [db_eplusout_reader](#) Python library for generating custom reports and optimisation KPIs.

DXF model output

This option causes EnergyPlus to generate a DXF file summarising the building model. The file is called [eplusout.dxf](#) and it is stored in the [EnergyPlus folder](#). This option can be useful for checking that the correct geometry has been generated.

Construction and surface details

This option causes EnergyPlus to generate a comma separated value file summarising the building model. The file is called [eplusout.eio](#) and it is stored in the [EnergyPlus folder](#).

RDD/EDD File

Select this option if you would like RDD and EDD files to be generated by the simulation for use in future simulations. The RDD file provides a list of all the EnergyPlus output reports that could be selected for the current model, while the EDD files contains a list of all the available actuators.

Time Setpoints Not Met Tolerances

This section of input allows more control over the reporting of “Time setpoint not met” hours.

Tolerance for time heating setpoint not met

This data defines the maximum difference from the heating setpoint when reporting the time heating setpoints are not met. If the zone temperature is below the heating setpoint by more than this value, the following report variables will increment as appropriate:

- Time Heating Setpoint Not Met
- Time Heating Setpoint Not Met While Occupied

It also impacts table report "Annual Building Utility Performance Summary" sub-table "Comfort and Setpoint Not Met Summary".

Tolerance for time cooling setpoint not met

This data defines the maximum difference from the cooling setpoint when reporting the time cooling setpoints are not met. If the zone temperature is above the cooling setpoint by more than this value, the following report variables will increment as appropriate:

- Time Cooling Setpoint Not Met
- Time Cooling Setpoint Not Met While Occupied

It also impacts table report "Annual Building Utility Performance Summary" sub-table "Comfort and Setpoint Not Met Summary".

Display - Model Options (Site)

Display tab on [Model Options](#) dialog

Options related to the display of data:

- [Show Data for Calculations](#)
- [Edit Screen](#)
- [Visualisation Display](#)

See also the [Program and Model Options](#) tutorial

Display Model Options - Show Data For Calculations

[Display tab](#) on [Model Options](#) dialog.

Analysis type

DesignBuilder user interface adapts to the currently selected Analysis type as set under Show Data for Calculations. This ensures that the Model data and other UI elements are relevant to the calculation results. You can change the Analysis type for the model at any time to carry out different calculations.

The list of Analysis type options available will depend on the [Location](#) of the site. For example when the site is located in the UK, it possible to set the Analysis type to **2-SBEM** or **10-DSM** for simplified or more detailed EPC and Part-L compliance checking modes.

The full list of options currently available is as follows:

- **1-EnergyPlus** - the analysis type for general purpose simulation work.
- **2-SBEM** - for SBEM EPC and Building Regulations compliance calculations in the UK and Ireland at NOS levels 3 and 4. This option is only available when the site is located in the UK or Ireland.

- **7-RT2012** - for Building Regulations compliance calculations in France. This option is only available when the site is located in France.
- **10-DSM** - for DSM EPC and Building Regulations compliance calculations in the UK and Ireland at NOS level 5. This option is only available when the site is located in the UK or Ireland.

Display Model Options - Edit Screen

[Display tab](#) on [Model Options](#) dialog.

Dynamics

By default the full rendered image is displayed during dynamic view operations such as orbiting, zooming and panning. However on systems with slower graphics display you may prefer to switch to using the wire frame option for a more "snappy" response. Options are:

- **1-Rendered** - the default setting where the whole building image is retained during the operation. For a large models/slow graphic adapters, this can result in a jerky motion in which case you should use...
- **2-Wire-frame** - where the building is represented using a wire frame which moves smoothly even for large models/slow graphic adapters.

Show zone labels

Select this option to display the name of each zone in the Edit screen while at block level. Zone labels are shown by default. See also [Zone label size](#) and [Zone label multiplier](#) below.

Zone label size

Select the size of the zone labels, if they are being displayed (see above). Options are:

- **1-Small** - may be useful when viewing smaller blocks.
- **2-Medium** - the default setting.
- **3-Large** - may be useful when viewing larger blocks.
- **4-Custom** - define your own text size for the current block using a custom multiplier (below).

Note: This option is different from the others on the Display tab in that **any setting you make while at block level or below is stored independently for that block**. This allows you to make different settings on different blocks depending on the size of the zones in that block. **Any setting made while at building level will be applied as a default for all blocks in the building** that have not had their own label size setting made. This allows you to set a label size while at building based on the size of most zones in the model, but override that setting for blocks that may have larger or smaller zones than most in the building.

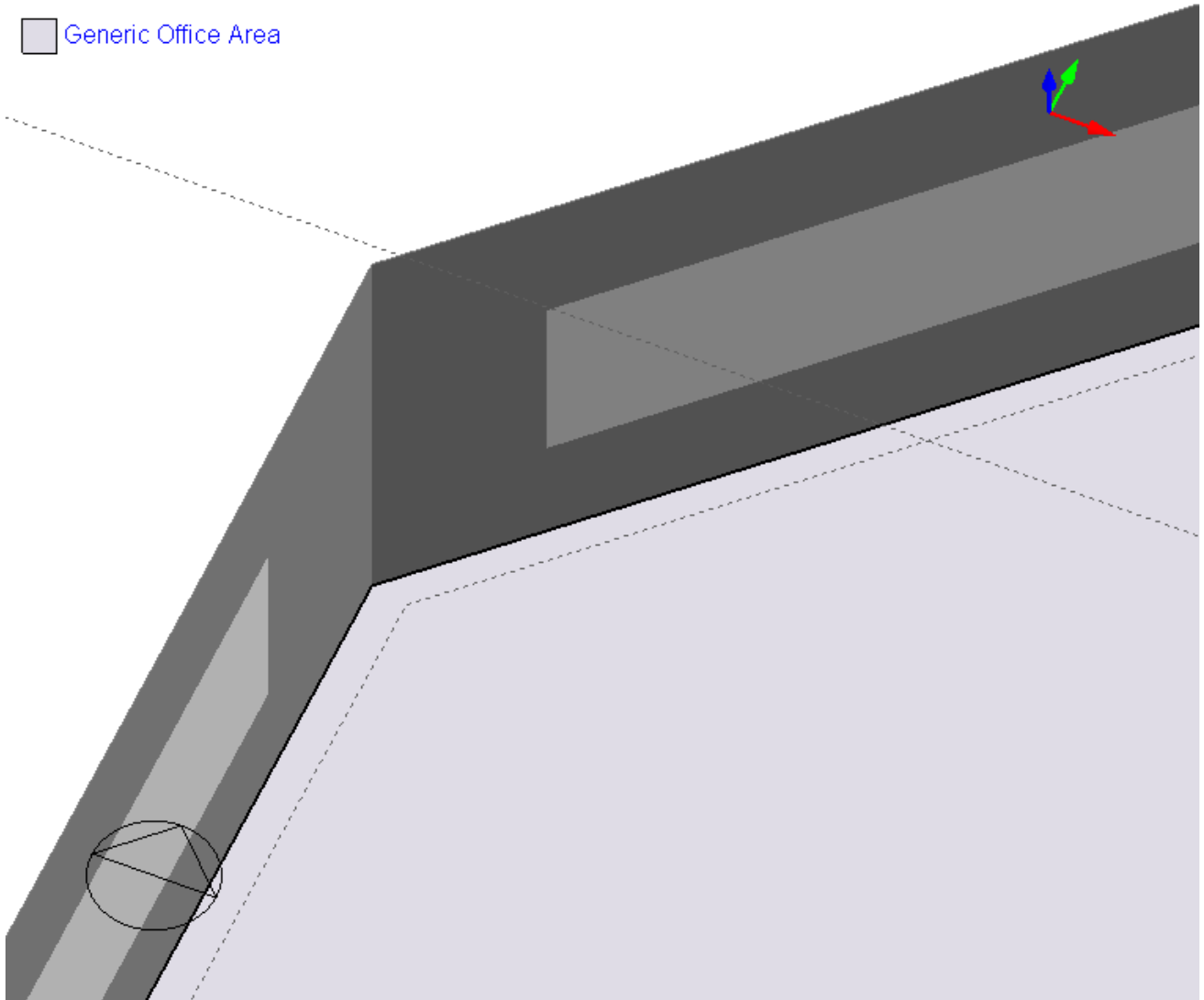
Zone label multiplier

When the Zone label size (above) is set to **4-Custom** you can enter a custom multiplier to define the size of the text. A value of 1 results in the same text size as when the **2-Medium** Zone label size option is selected.

Display zone inner surfaces

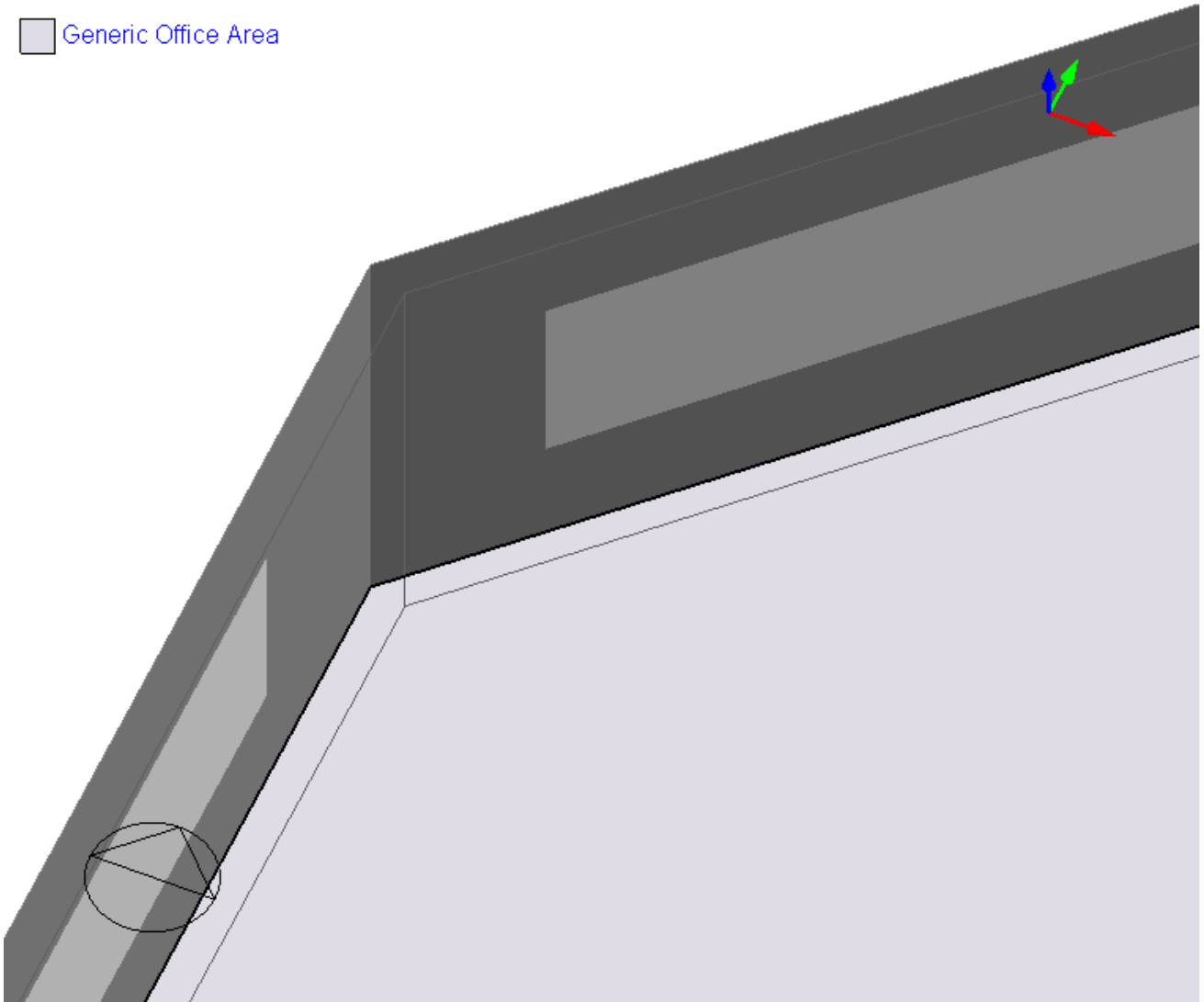
When checked this option ensures that the zone inner surfaces are displayed in addition to the block perimeter surfaces when at block level.

 Generic Office Area



Display zone inner surfaces OFF

 Generic Office Area



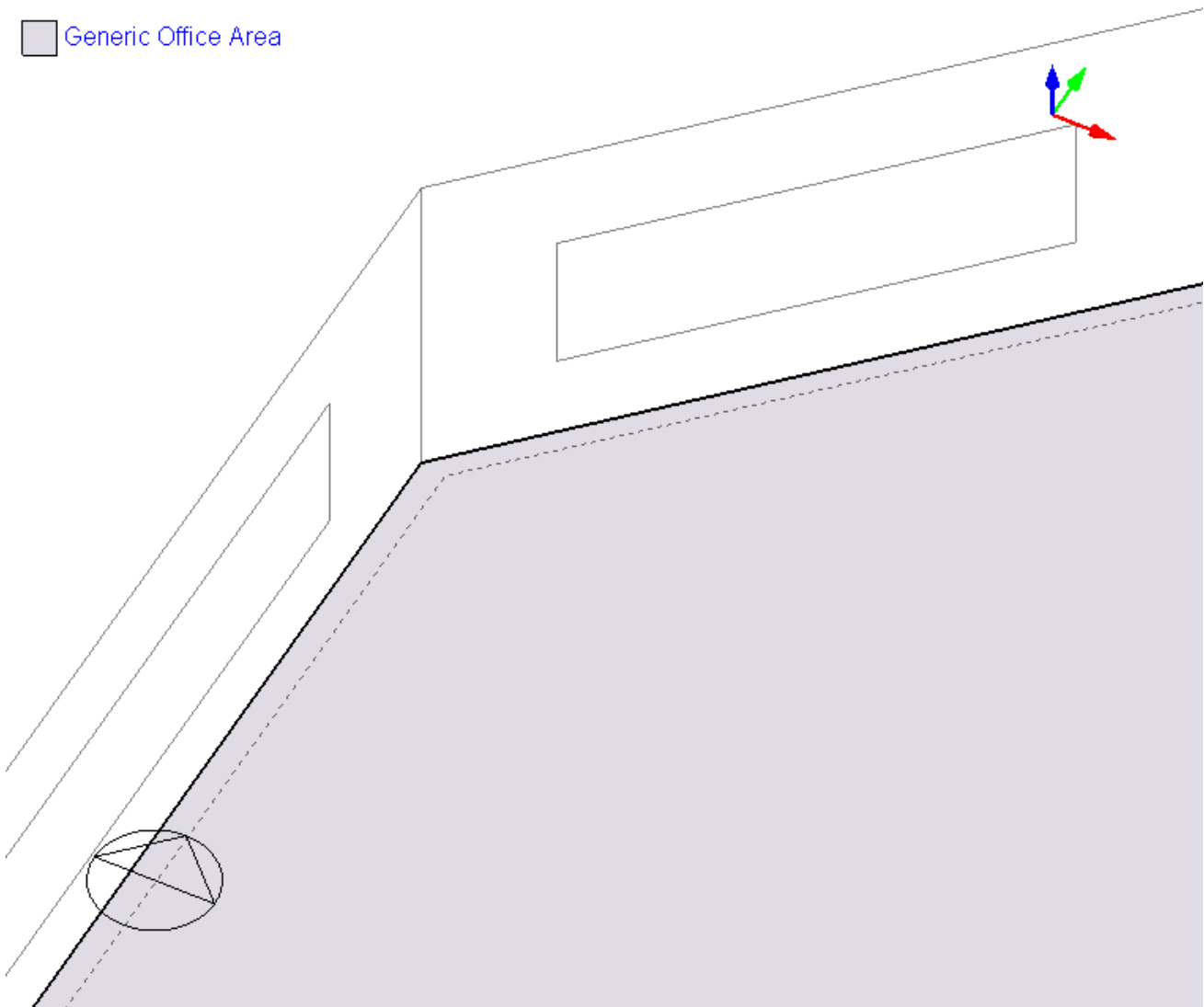
Display zone inner surfaces ON

By default the option is switched off and the zone inner surfaces are only represented by a dotted line on the floor plane when at block level. To get a clearer picture of where all of the zone inner surfaces are, you can switch this option on.

Display filled external walls

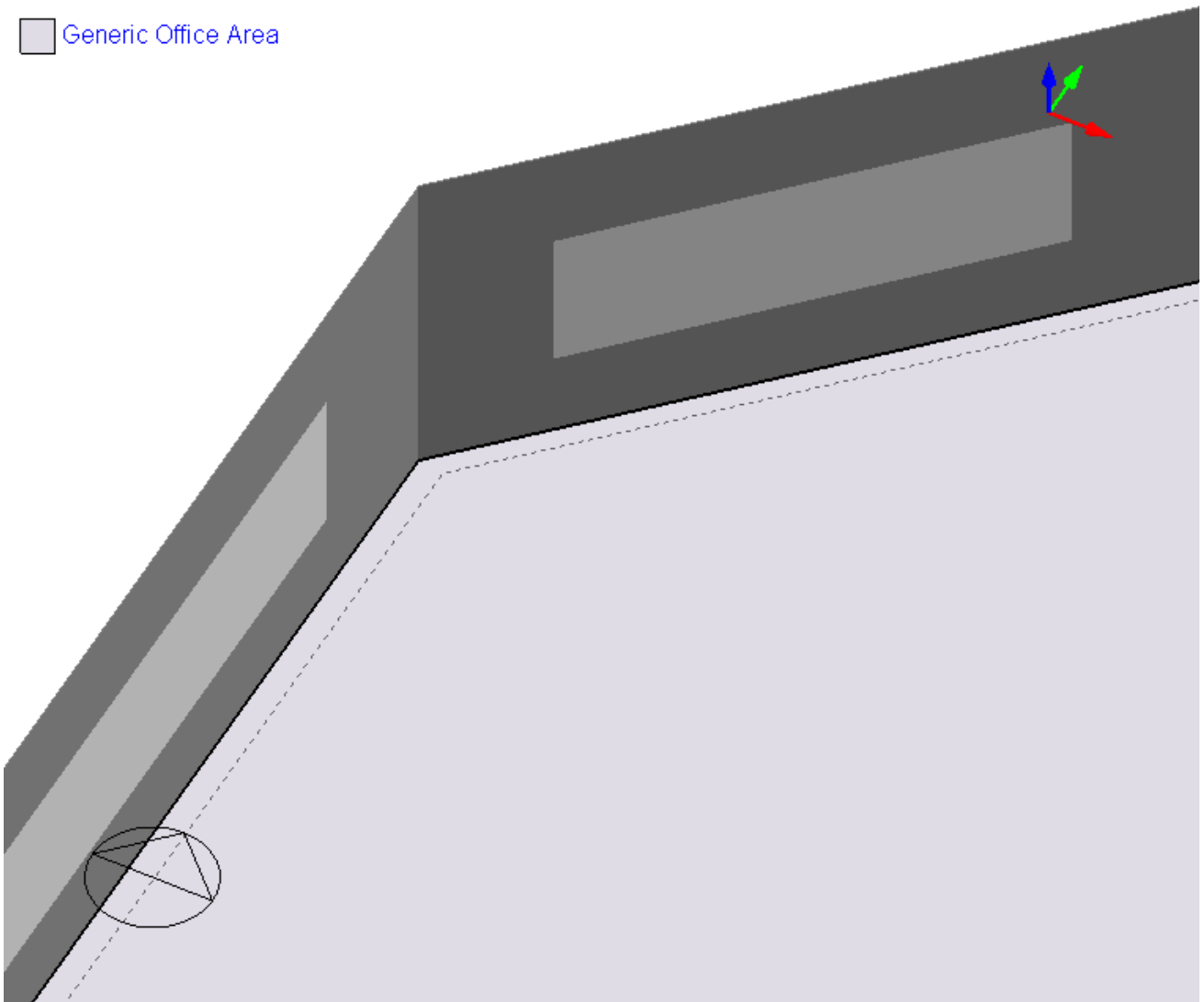
You can choose to switch on/off display of external walls for each block. Check the option to show external walls filled in dark grey when at block level. This option is on by default.

 Generic Office Area



Display filled external walls OFF

 Generic Office Area



Display filled external walls ON

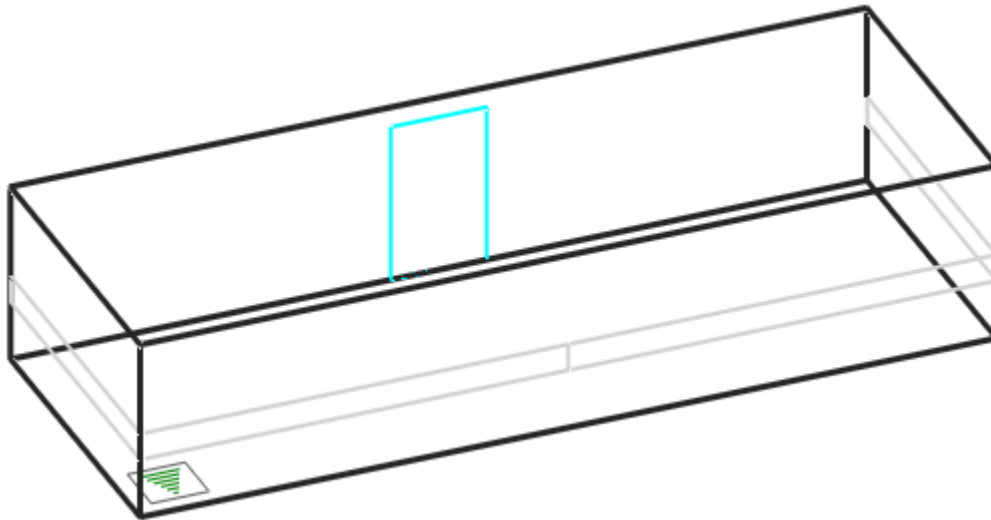
Generally for standard extruded blocks it is best to show external walls to help reference locations within the block with respect to existing openings etc.

For [Pitched roof blocks](#) and [General block forms](#) the external walls can obscure the internal space and so the default for these block types is to hide the external walls when at block level in the modeller. You can override this default behaviour using this option.

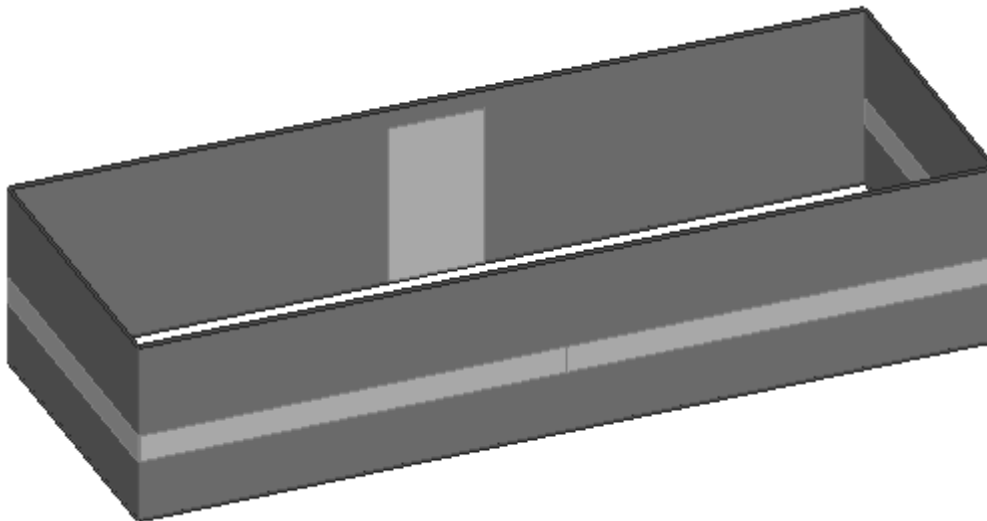
Note: It can be useful to switch off external wall display when drawing partitions in roof blocks.

Display zone filled walls

By default zone surfaces are displayed in wire frame which allows the opening type and surface adjacency symbols to be displayed. For example in the screenshot below the ground adjacency of the floor and the door opening (cyan colour) can clearly be seen.

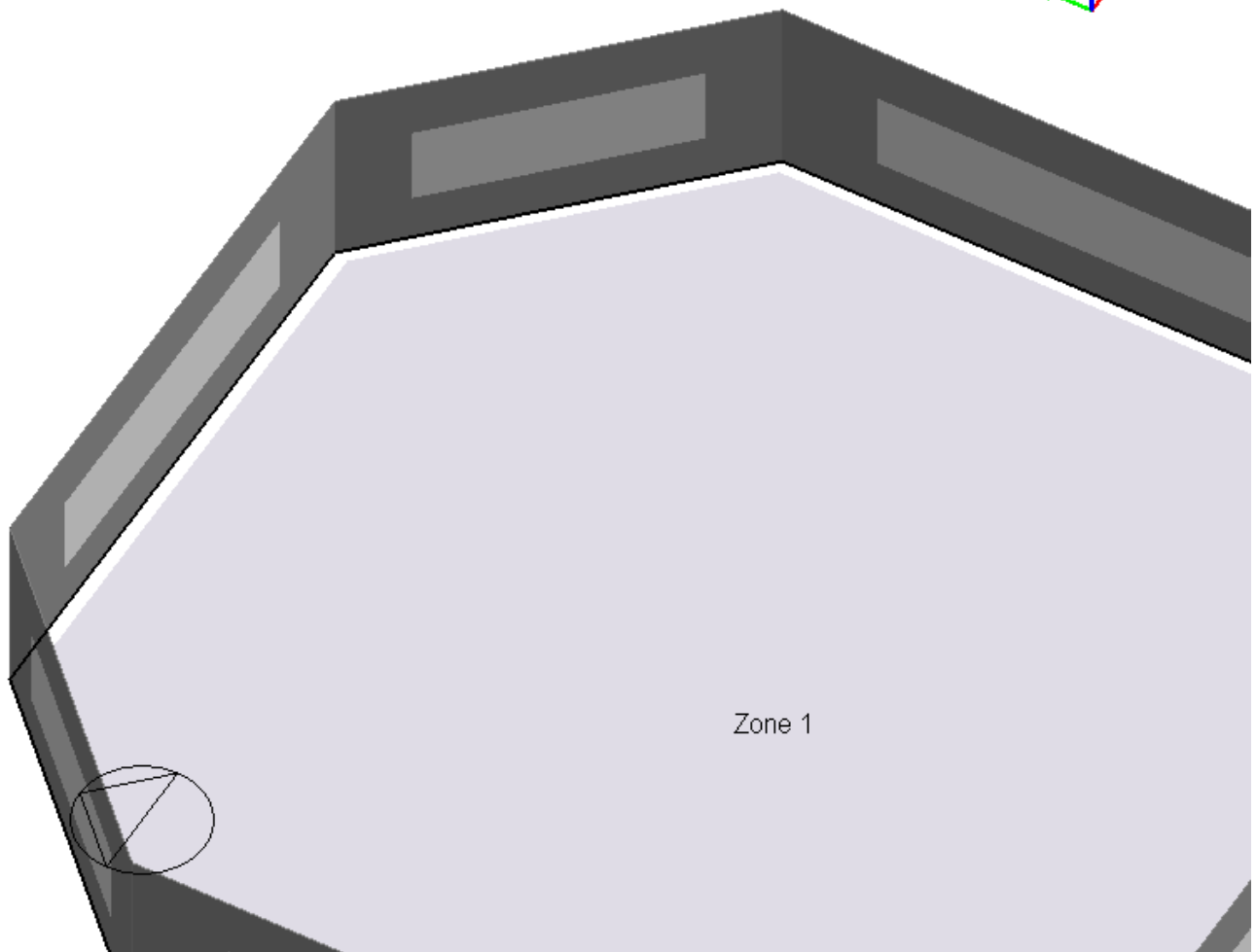
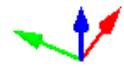
**Display zone filled external walls OFF**

When the **Display zone filled walls** option is switch on the display of zones looks similar to blocks but the information referred to above is lost. See screenshot below.

**Display zone filled external walls ON****Display zone inner volume floors**

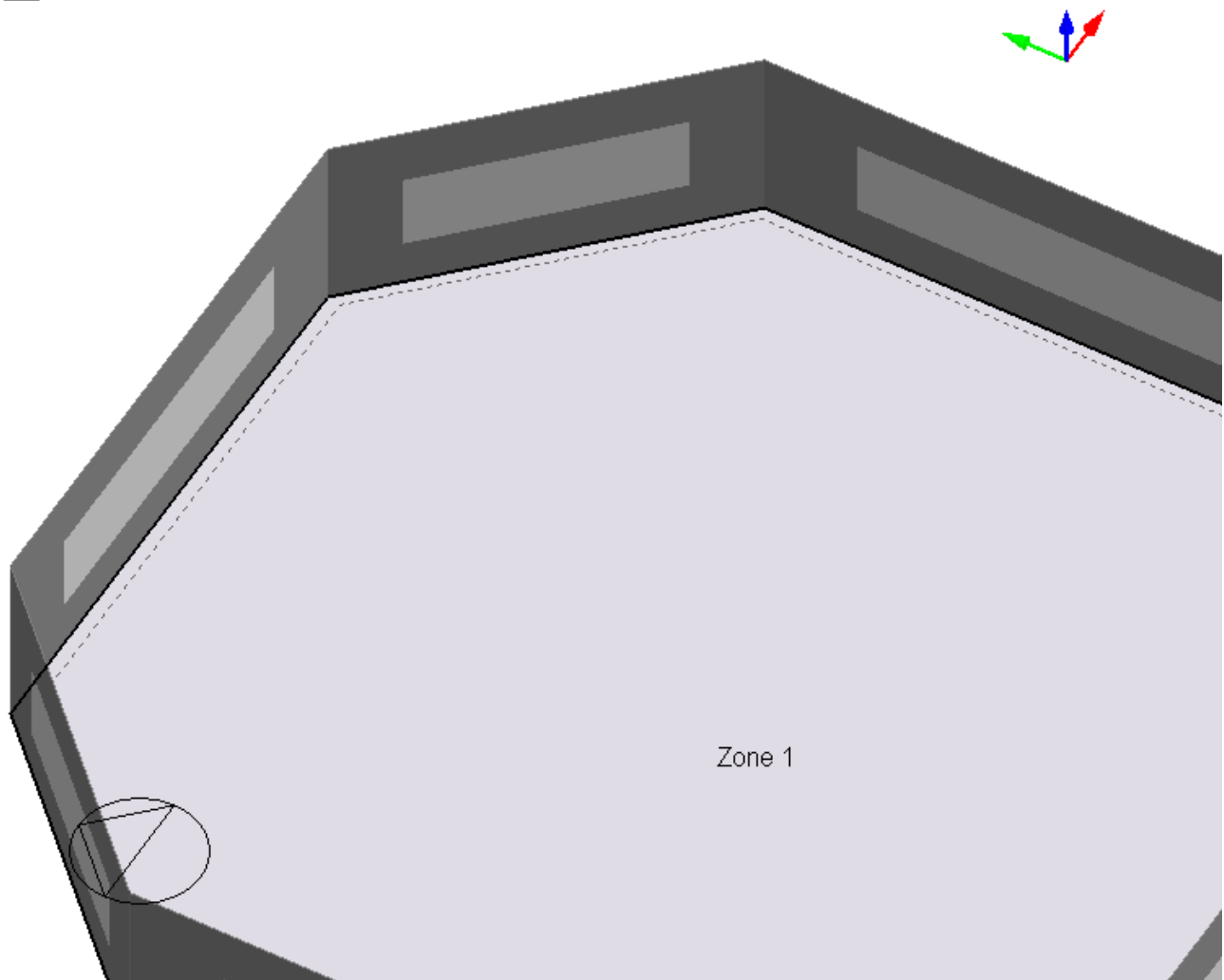
Use this option to define whether the inner volume is marked on the floor using a dotted line. This option is off by default.

Generic Office Area



Display zone inner volume floors OFF

 Generic Office Area

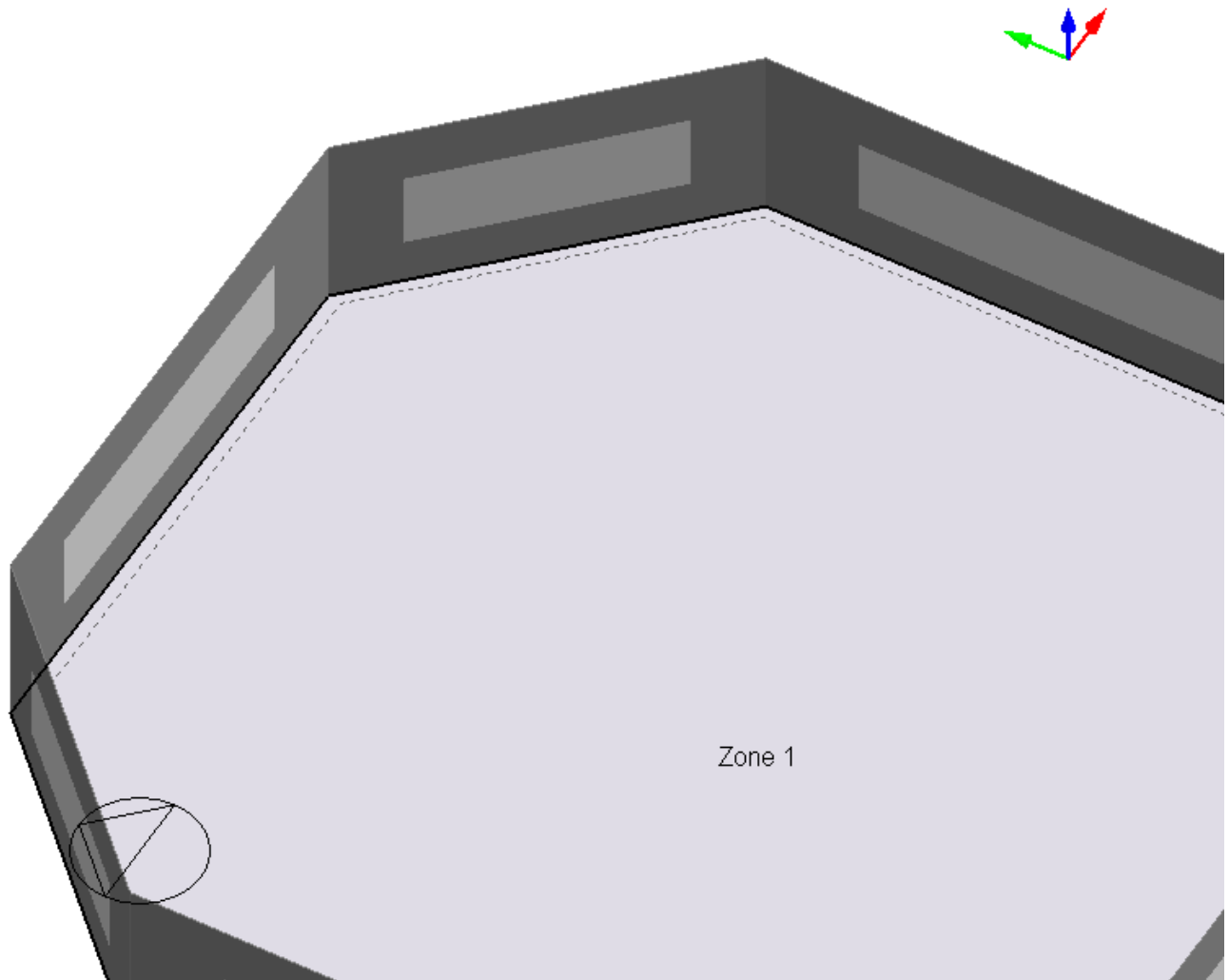


Display zone inner volume floors ON

Display zone inner floor perimeters

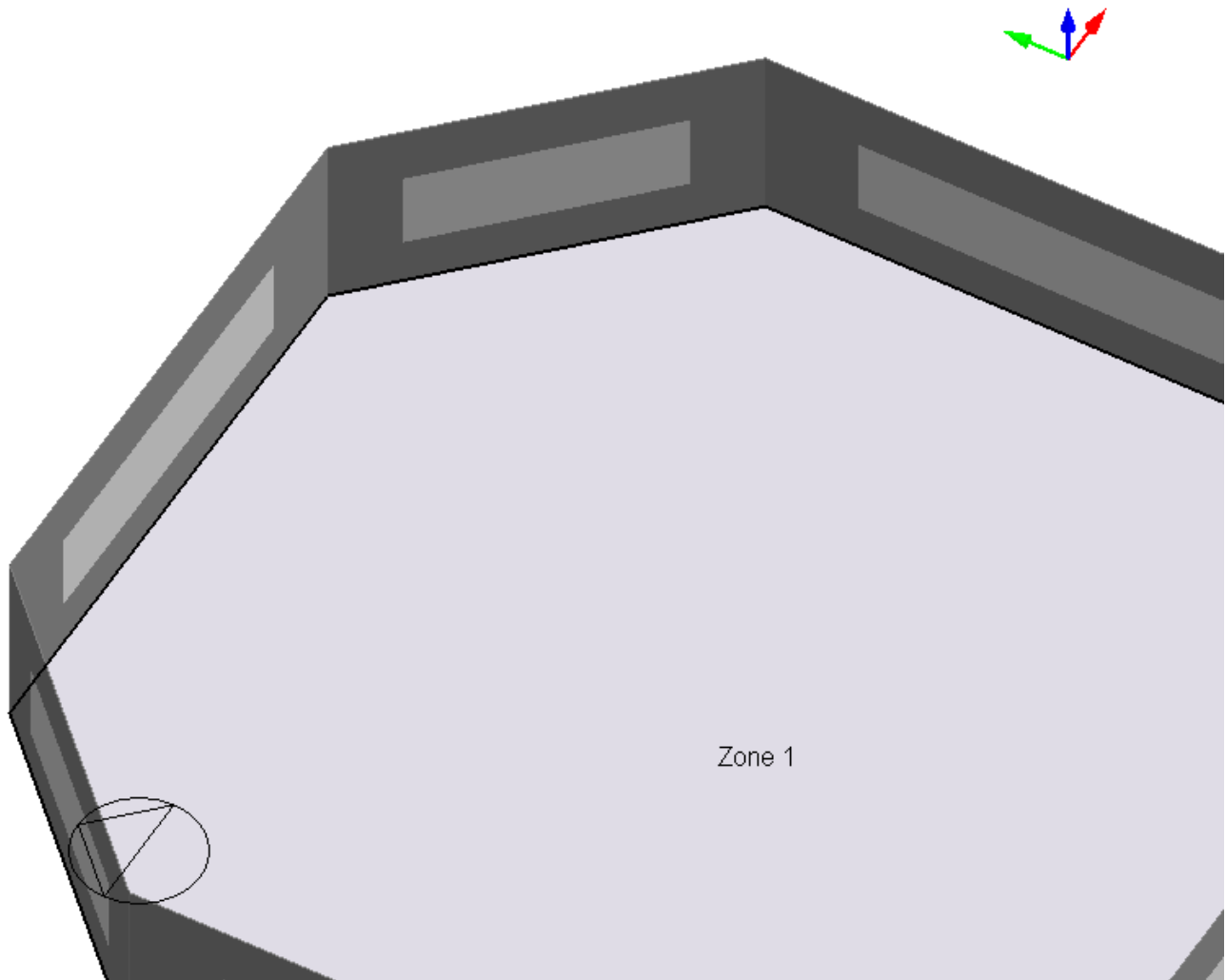
Check this option to obtain a dotted line representation of the zone inner floor perimeter. This option only has an effect when both [Display zone inner surfaces](#) and [Display zone inner volume floors](#) are both off. This option is on by default.

Generic Office Area



Display zone inner floor perimeters ON

 Generic Office Area



Display zone inner floor perimeters OFF

Fit includes imported drawing

Select this option to get the imported drawing included in the calculation of the model extent at site and building levels when the [Fit view](#) command is run and when the screen is refreshed internally within DesignBuilder.

Show imported 2-D drawing at zone and surface levels

Select this option to show the imported 2-D floor plan / elevation drawing at zone and surface levels. The option is switched off by default for the sake of clarity but you may wish to switch it on when drawing openings at surface level to help position windows, doors etc.

Tip: This option must be switched on when using an [imported elevation](#) to show the position of windows, doors etc.

Show shading devices

Select this option to display shading devices while in the Edit screen. Shading devices are shown by default.

Show windows

You can speed up display in the edit screen by switching off window display. Windows are shown by default.

Surrounding block display

Check this option to show surrounding blocks in wire frame when at block level. This option is off by default.

Tip: A more convenient way to control the visibility of other blocks in the current building, when at block level, is to through the **Surrounding block/zone display** toolbar icon.

Surrounding block zone display

Check this option to show all of the surrounding zones in the current block in wire frame when at zone level. This option is off by default.

Tip: A more convenient way to control the visibility of other zones in the block, when at zone level, is to through the **Surrounding block/zone display** toolbar icon.

Surrounding building zone display

Check this option to show all of the surrounding zones in the current building in wire frame when at zone level. This option is off by default. This option is only available when the **Surrounding block zone display** option is checked.

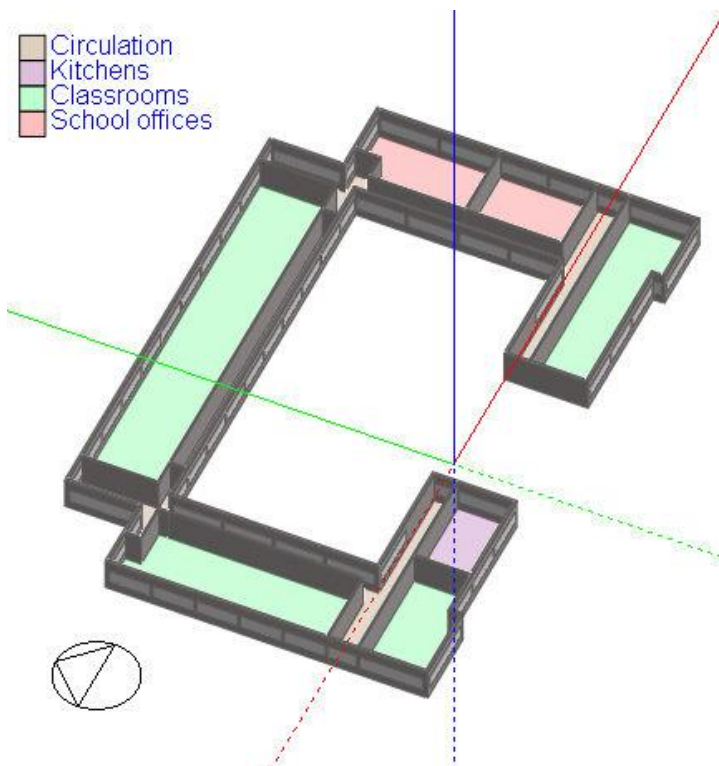
Zone floor shading at block level

You can easily display zones with different colours according to the selected Activity template or HVAC template when at block level. This provides a visual quality assurance check and a useful summary of how the model was set up.

Zone floors can be shaded by:

- **1-Activity Template**, the default setting where zone floors are shaded according to the Activity template selection allowing different areas of the block to be easily identified.
- **2-HVAC Template**, where zone floors are shaded according to the HVAC template selection to show the distribution of HVAC systems across the block. See the [step by step](#) guide below for details.
- **3-None**, where zones floors aren't shaded.

The image below illustrates the block view on the Layout tab when shading by Activity template is selected.



Step by Step Guide

Display zone colours according to HVAC system type

This step-by-step guide explains how to display zone floor colours according to the selected HVAC template. This is done by assigning a different colour to each HVAC template used in the model.

1. Start by opening the [Model options](#) dialog, clicking on the Display tab and under the Edit screen header setting the [Zone floor shading at block level](#) option to **2-HVAC**.

Model Options - Building

Model Options

Data Advanced Heating Design Cooling Design Simulation **Display** Drawing tools Block Project details Carbon

Show Data for Calculations

Analysis type 1-EnergyPlus

Edit Screen

Dynamics 1-Rendered

☒ Show zone labels

Zone label size 2-Medium

☐ Display zone inner surfaces

☒ Display filled external walls

☐ Display zone filled walls

☐ Display zone inner volume floors

☒ Display zone inner floor perimeters

☐ Fit includes imported floor plan

☒ Show imported 2-D drawing at zone and surface levels

☒ Show shading devices

☒ Show openings and sub-surfaces at building level

☐ Surrounding block display

☒ Surrounding block zone display

☐ Surrounding building zone display

Zone floor shading at block level 2-HVAC

☒ Automatic HVAC network breaks

2. Navigate to Building level and click on the [HVAC](#) Model data tab.
3. Clone a template of the most-used HVAC system type. For example, if most of the building is supplied by a VAV system with a water-cooled chiller, select the library "VAV, Water-cooled Chiller, Hot Water" item in the browse list and click on the **Create a copy of highlighted item** icon. This creates an editable copy of the library template, allowing you to modify template settings, including the [Floor shade colour](#).
4. Click on the **Edit /View highlighted item** icon to edit the copy. Enter a suitable template name and assign a colour. In our example we will make these settings:
 - a. Name: "System 1: VAV, Water-cooled Chiller"
 - b. Floor shade colour: [Blue](#).

Edit HVAC template - System 1: VAV, Water-cooled Chiller

HVAC templates

General Ventilation Heating and Cooling Humidity Control Air temperature distribution Cost

General

Name System 1: VAV, Water-cooled Chiller

Description

Source DB

Category Generic

Region General

Simple

Auxiliary energy (kWh/m2) 35.00

Colour Shading in Model

 Floor shade colour

5. Select the newly created "System 1: VAV, Water-cooled Chiller" HVAC template at Building level as the default setting for all zones in the building.
6. Repeat the previous step to create further custom HVAC System Templates, one for each additional system type in your building model. In this example, create a new HVAC template based on the "VRF (Air-Cooled), Heat Recovery, DOAS" library HVAC template and edit the copy as follows:
 - a. Name: "System 2: VRF with DOAS".
 - b. Floor shade colour: [Green](#).

Edit HVAC template - System 2: VRF with DOAS

HVAC templates

General Ventilation Heating and Cooling Humidity Control Air temperature distribution Cost

General

Name System 2: VRF with DOAS

Description

Source DB

Category Generic

Region General

Simple

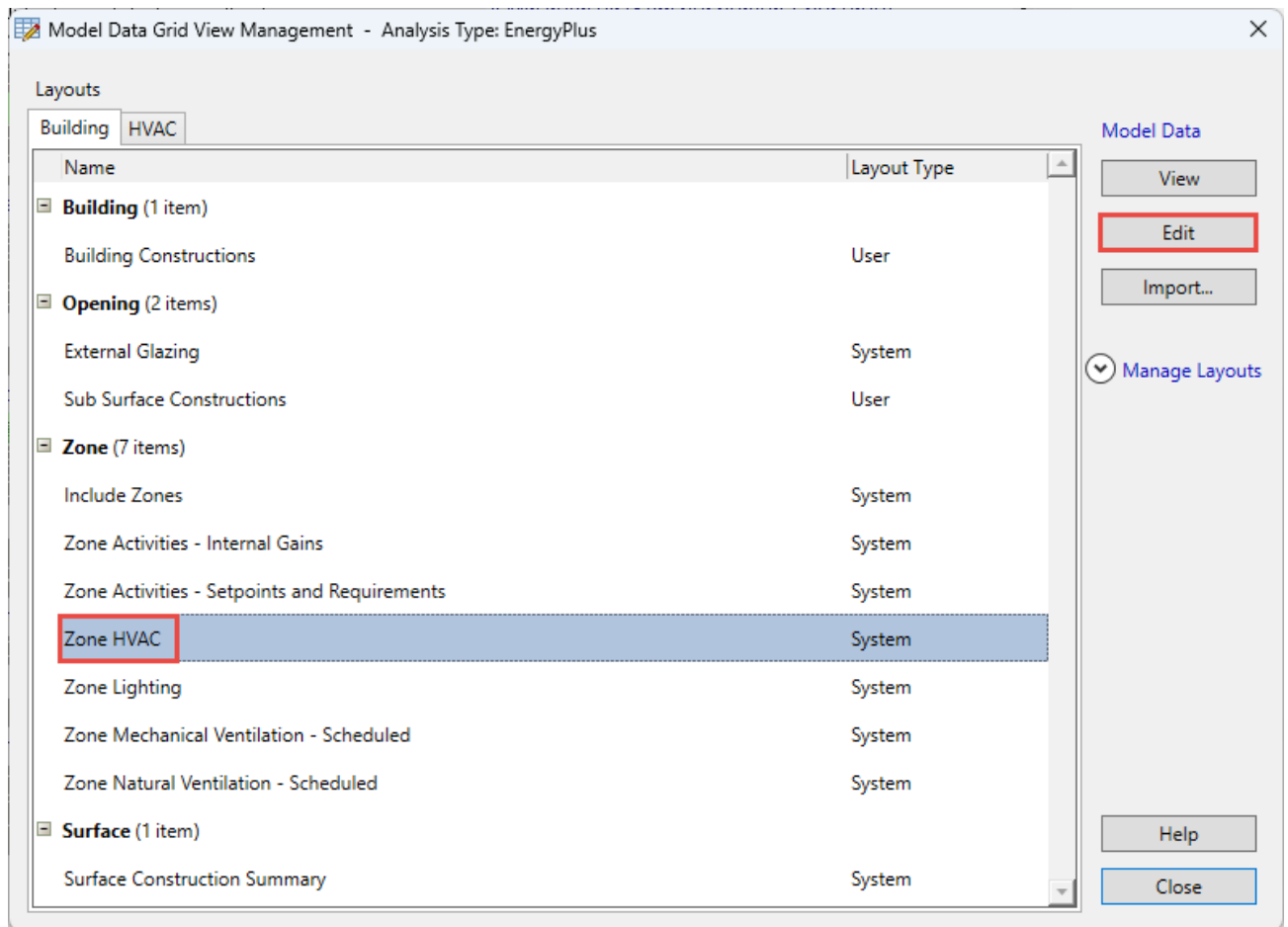
Auxiliary energy (kWh/m2) 25.00

Colour Shading in Model

 Floor shade colour

Press OK to confirm the changes.

7. Open the [Model Data Grid View](#) (MDGV) dialog to assign the secondary ("System 2: VRF with DOAS") HVAC systems to their relevant zones. Select 'Zone HVAC' and choose 'Edit'.



8. All zones should initially show "System 1: VAV, Water-cooled Chiller". If not, you can use the bulk edit feature in MDGV to change all zones to use this template.

Zone HVAC [Bulk Edit]

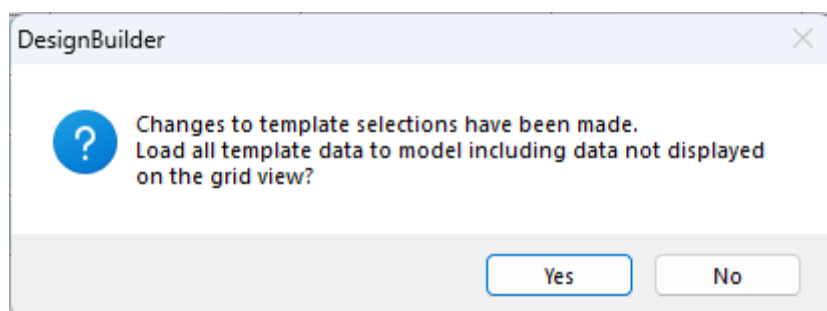
☒ Edit ☒ Bulk Edit

Search (Ctrl+F) ^ v x

Block

	Identity	Activity	
	Identity	Floor Areas and Volum	HVAC Template
Zone	Name	Floor area (m2)	Template
3 First Floor (11 items)			
301_Open_Office_C	301_Open_Office_C	640.1	System 1: VAV, Water-cooled Chiller
304_Toilet_F	304_Toilet_F	30.7	System 1: VAV, Water-cooled Chiller
305_Toilet_M	305_Toilet_M	25.3	System 1: VAV, Water-cooled Chiller
301_Open_Office_W	301_Open_Office_W	115.6	System 1: VAV, Water-cooled Chiller
301_Open_Office_N1	301_Open_Office_N1	72.5	System 1: VAV, Water-cooled Chiller
302_Corridor	302_Corridor	76.8	System 1: VAV, Water-cooled Chiller
301_Open_Office_N2	301_Open_Office_N2	55.5	System 1: VAV, Water-cooled Chiller
303_Lift	303_Lift	4.5	System 1: VAV, Water-cooled Chiller
306_Stairs_Main	306_Stairs_Main	16.8	System 1: VAV, Water-cooled Chiller
301_Open_Office_E	301_Open_Office_E	115.6	System 1: VAV, Water-cooled Chiller
301_Open_Office_S	301_Open_Office_S	242.6	System 1: VAV, Water-cooled Chiller

Select "Yes" when asked whether to load template data:



9. Now use the filter option in MDGV to select the zones that are served by the secondary System 2. To do this move the mouse pointer to the right-hand side of the "Name" column to display a drop down arrow and click this to display the filter control. In this example, we will select all areas that are not offices spaces by checking the appropriate checkboxes:

Zone HVAC [Bulk Edit]

☒ Edit ☒ Bulk Edit

Search (Ctrl+F) ^ v x

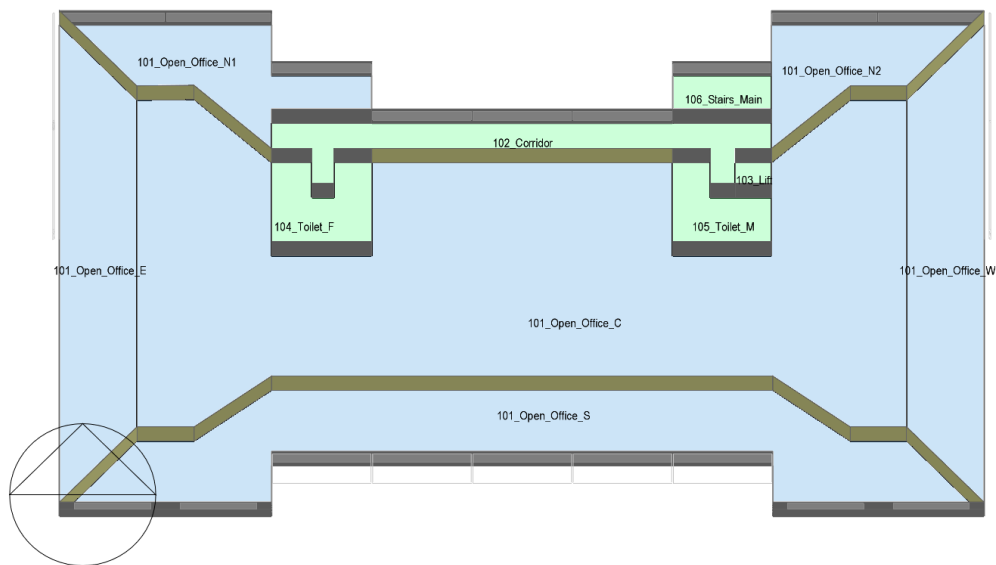
Block

Identity		Activity	
Identity		Floor Areas and Volum	HVAC Template
Zone	Name	Floor area (m2)	Template
3 First Floor (5 items)		(Clear Filter)	
		(Select All)	
304_Toilet_F	304_Toilet_F	☒ 203_Lift	/AV, Water-cooled Chiller
305_Toilet_M	305_Toilet_M	☒ 204_Toilet_F	/AV, Water-cooled Chiller
302_Corridor	302_Corridor	☒ 205_Toilet_M	/AV, Water-cooled Chiller
303_Lift	303_Lift	☒ 206_Stairs_Main	/AV, Water-cooled Chiller
306_Stairs_Main	306_Stairs_Main	☐ 301_Open_Office_C	/AV, Water-cooled Chiller
		☐ 301_Open_Office_E	
		☐ 301_Open_Office_N1	
		☐ 301_Open_Office_N2	/AV, Water-cooled Chiller
		☐ 301_Open_Office_S	/AV, Water-cooled Chiller
		☐ 301_Open_Office_W	/AV, Water-cooled Chiller
		☒ 302_Corridor	/AV, Water-cooled Chiller
		☒ 303_Lift	/AV, Water-cooled Chiller
		☒ 304_Toilet_F	/AV, Water-cooled Chiller
		☒ 305_Toilet_M	/AV, Water-cooled Chiller
		☒ 306_Stairs_Main	
2 Upper Ground Floor (5 items)			
204_Toilet_F	204_Toilet_F		
205_Toilet_M	205_Toilet_M		
202_Corridor	202_Corridor		
203_Lift	203_Lift		
206_Stairs_Main	206_Stairs_Main		
1 Lower Ground Floor (5 items)			
104_Toilet_F	104_Toilet_F	30.7	System 1: VAV, Water-cooled Chiller
105_Toilet_M	105_Toilet_M	25.3	System 1: VAV, Water-cooled Chiller
102_Corridor	102_Corridor	76.8	System 1: VAV, Water-cooled Chiller
103_Lift	103_Lift	4.5	System 1: VAV, Water-cooled Chiller
106_Stairs_Main	106_Stairs_Main	16.8	System 1: VAV, Water-cooled Chiller

10. Use the [Bulk Edit](#) tool to assign System 2 to all the selected zones.

11. Navigate to a Block and open the Layout tab. Zone floors are shaded in **Blue** for System 1 and **Green** for System 2. This makes it easy to check and report the HVAC model settings.

■ System 1: VAV, Water-cooled Chiller
■ System 2: VRF with DOAS



Tip: Although it is based on reviewing Simple HVAC templates, this technique works equally well for Simple and Detailed HVAC system types. This is because each zone in a Detailed HVAC system will also have a corresponding Simple HVAC template set which is based on the Detailed HVAC system it is connected to.

Display zone colours according to Activity

To assign colours using Activity templates, follow a similar process described above for HVAC templates but with the following changes:

1. Select **1-Activity** as the **Zone floor shading at block level** setting.
2. Work within Activity templates to define custom project activity names and floor colours.

Automatic HVAC network break

When using Detailed HVAC, the branches on the layout diagram will often need to cross each other without making any actual connections. When this option is on, DesignBuilder helps to clarify complex layouts by inserting "breaks" where pipes and ducts cross each other so that they don't appear connected. For very large complicated HVAC layouts, the calculations that determine where to make breaks can be very time consuming. This option allows you to switch off the automatic breaks display feature to help speed up program operation when editing complex HVAC layouts. The option only affects the HVAC layout display and does not impact at all on calculations.

Display Model Options - Visualisation

[Display tab](#) on [Model Options](#) dialog.

Dynamics

This option affects the display of the model when orbiting, zooming, panning etc in the Visualisation Screen:

- **1-Rendered** - the default setting where the whole building image is retained during the operation. For a large models/slow graphic adapters, this can result in a jerky motion in which case you should use...

- **2-Wire-frame** - where the building is represented using a wire frame which moves smoothly even for large models/slow graphic adapters.

Show shadows

Use this option to display shadows in the Visualisation screen.

Show North arrow

You can use this control to display/hide the North arrow in the Visualisation screen. The North arrow is always displayed on the Layout tab of the Edit screen.

Show ground plane

You can choose to show/hide the plane used to represent the ground in the Visualisation screen. You may wish to hide the ground plane if you have used component blocks to represent the ground.

Note: when displayed, the ground plane extends just beyond the x-y size of the model and hides any other building object situated at or below z=0.

View

Enter the angular field of view. 50° is the default. For a wide-angle view use a larger value. This can be useful inside a building where you need to capture more of the space in the view. For a "telephoto" effect, with reduced perspective effects, use a smaller value.

Antialiasing

Antialiasing is the process used to remove 'jaggy' edges of lines which are not horizontal or vertical. Most modern medium-high-end graphics adapters allow effective antialiasing to be automatically applied to OpenGL screens using hardware acceleration. If you have one of these cards you should be getting sharp lines already and you can switch the DesignBuilder software antialiasing off. See [hardware](#) requirements for more on this.

If you switch on software antialiasing you can select a quality setting from 1-4 where 4 gives the smoothest edges and takes longest time to generate.

Note: some low-end graphics adapters are not suited to software antialiasing because they lack hardware acceleration for this feature.

Drawing Tools - Model Options (Site)

[Drawing Tools tab](#) on [Model Options](#) dialog.

Options for:

- [Draw Block](#)
- [Direction Snaps](#)
- [Point Snaps](#)
- [Drawing Guides](#)
- [Object Rotation](#)
- [Protractor](#)
- [Miscellaneous Edit](#)

To learn more about Drawing Tools go to the [Block Creation Options](#) and [Snaps, Drawing Guides & Protractor](#) sections.

See also the [Program and Model Options](#) tutorial

Drawing Tools - Draw Block

[Drawing Tools tab](#) on [Model Options](#) dialog.

Auto complete block

See [Auto-complete block](#) entry under [Block Creation Options](#).

Drawing Tools - Direction Snaps

[Drawing Tools tab](#) on [Model Options](#) dialog.

See [Direction Snaps](#) entry under [Snaps, Drawing Guides & Protractor](#)

Drawing Tools - Point Snaps

[Drawing Tools tab](#) on [Model Options](#) dialog.

See [Point Snaps](#) entry under [Snaps, Drawing Guides & Protractor](#).

Drawing Tools - Drawing Guides

[Drawing Tools tab](#) on [Model Options](#) dialog.

See [Drawing Guides](#) entry under [Snaps, Drawing Guides & Protractor](#).

Drawing Tools - Object Rotation

[Drawing Tools tab](#) on [Model Options](#) dialog.

You can change the rotation increment to control the angular increments for the object rotation process.

See [Rotation](#) entry under [General Operations](#).

Drawing Tools - Protractor Tool

[Drawing Tools tab](#) on [Model Options](#) dialog.

See [Protractor](#) entry under [Snaps, Drawing Guides & Protractor](#).

Drawing Tools - Miscellaneous Edit

[Drawing tools tab](#) on [Model Options](#) dialog.

Automatic block zoning

By default DesignBuilder automatically generates zones as [partitions](#) are drawn. You can change this behaviour by unchecking the **Automatic block zoning** option. In this case partitions are shown in blue as they are drawn and are referred to as [outline partitions](#). Outline partitions do not create new zones and are modelled in the same way in simulations as [hanging partitions](#). To convert any outline partitions to standard partitions and to rezone the block use the [Rezone block](#) command accessed from the [Tools](#) menu. Note that outline partitions that do not connect at both ends (known in DesignBuilder as [hanging partitions](#)) will remain in blue even after the Rezone block command.

Alternatively, to convert **all** outline partitions in the **building** to standard partitions and hence rezone the whole building you can use the [Rebuild](#) command. This will take longer to process than the [Rezone block](#) command as it recalculates the surface and adjacency geometry as well rezoning.

Note: switching **Automatic block zoning** off also prevents rezoning following other partition edit commands such as delete, copy and undo/redo.

Automatically include voids from lower blocks

Select this option if you would like DesignBuilder to automatically punch voids through new blocks drawn on top of existing blocks having voids. If you leave this option switched off (default state) you can still [draw voids](#) at the block level using the ghosted outline of adjacent voids in a block below as a guide.

Block - Model Options (Block)

[Block tab](#) on [Model Options](#) dialog.



You can edit certain aspects of a building block geometry by clicking on the Model options icon while at the block level. This opens the [Model Options](#) dialog box with the Block tab open.

Model Options	
Data Advanced Heating Design Cooling Design Simulation Display Drawing tools Block Project details Carbon	
Block Geometry	
☒ Active	
Form	1-Extruded
Height (m)	2.40
Transition Angles	
☒ Allow pitched roof construction	
Flat to pitched roof transition angle (°)	0.0
Roof-wall transition Angle (°)	70.0

Note: This tab is only enabled when the Model options dialog is opened at Block level in the model

See also the [Program and Model Options](#) tutorial

Block Geometry

Active

Keep this checkbox checked if the block is to be accessible for editing etc, i.e. it is not "ghosted". However, there may be times when it is helpful to ghost the block so that it becomes semi-transparent allowing blocks behind to be accessed, selected and edited. In this case this checkbox can be unchecked.

Tip: It is usually more convenient to control the ghosting of blocks through the "Ghost Blocks" on page 116 toolbar icon while at building level.

Form

Once a block has been created its [Form](#) can be changed retrospectively by making a selection from the drop list. Options are typically:

- [1-Extruded](#), where the block geometry is generated by [extruding](#) the floor plan by a user-defined height setting.
- [2-Sloped walls](#) where the block geometry is generated by applying uniformly [sloping walls](#) to a block perimeter drawn on a horizontal plane, or
- [3-Pitched roof](#) in which a pitched roof form is automatically generated from a block perimeter drawn on a horizontal plane, or

If the block has **5-General** block form due to a previous block edit operation, then only these block form options are available:

- [2-Sloped walls](#) or
- [3-Pitched roof](#) or
- [5-General](#) - the form will be automatically set to this form when it has been edited in some way, for example by using the cutting or boolean geometry tools so that it takes on a general form.

Tip: When you change the form of a block from **1-Extruded** to **3-Pitched Roof** or **2-Sloping Walls** the original form will be remembered so that even if you have edited the block by cutting it or dragging a face, when you convert the form back to **1-Extruded**, the original modified form will be restored.

Slope

For blocks of **2-Sloping wall** form only, enter the slope angle between horizontal and the sloping walls. A slope of 10° means a shallow slope and 80° means a steep slope.

Height

The height setting is used for the **1-Extruded** and **2-Sloping wall** block forms to create the block geometry. It can be retrospectively changed here.

Gable roof

For blocks of **3-Pitched roof** form only, check this option if the pitched roof should be generated using a gable, where possible.

Roof overhang

For blocks of **3-Pitched roof** form only, enter the extent of the pitched roof overhang (in m or ft).

Roof slope

For blocks of **3-Pitched roof** form only, enter the slope angle between horizontal and the sloping pitched roof elements. A slope of 10° means a shallow slope and 80° means a steep slope.

Transition Angles

Floor-wall transition angle

This block level setting allows sloping floors to be defined. Any surface having a slope of less than or equal to the value entered here is considered by DesignBuilder to be a floor. Surfaces with slope angles greater than this value are considered to be walls. The default **Floor-wall transition angle** is 5° which means that any surfaces with a slope of 5° or less will be treated as floors and the appropriate constructions will be used. Also the surface will contribute to the overall zone floor area which is used along with various other inputs to define internal gains, ventilation rates etc.

Allow pitched roof construction

If pitched roof constructions are to be differentiated from external wall and flat roof constructions for the current block then this checkbox should be checked. This is the default setting and enables two further settings (below) to control how DesignBuilder classifies block surfaces as being external wall, pitched roof or flat roof. Alternatively, you can force all non-horizontal surfaces to be walls by unchecking this option.

Flat to pitched roof transition angle

By default only roofs that are horizontal are treated as flat roofs, but this block level setting can be used when the **Allow pitched roof construction** option is checked to allow non-horizontal roofs to be treated as flat as well. For example if 10° is entered here then any roof surfaces of slope $\leq 10^\circ$ in the block will be treated as flat roof surfaces and the appropriate construction will be used in simulations. Also the appropriate thermal bridging settings will be used.

Roof-wall transition angle

When the **Allow pitched roof construction** option is checked you can enter the angle which defines the surface slope below which a wall becomes a roof. For example a setting of 70° means that any block surfaces steeper than 70° will be treated as external walls (using external wall constructions, glazing layout etc) and those less steep than 70° will be treated as pitched roof surfaces (using pitched roof constructions, roof glazing layout etc).

Project Details - Model Options (Building)

Project details tab on [Model Options](#) dialog

Use the Notes tab to record General information about the building model:

- [Project Details](#)
- [Owner Details](#)
- [Analyst Details](#)
- [Notes](#)

The Project, Owner and Analyst Details data is used as a header in UK Part-L2 Compliance Check reports.

See also the [Program and Model Options](#) tutorial

Project Details

Enter the building Sector, Project name and address.

Owner Details

Enter the name and contact details for the building owner/client.

Analyst Details

Enter your own name and contact details.

Notes

You can enter any information about the building model in free text format.

Note: If you want to add hard carriage returns in the text you should hold the <Ctrl> key down while pressing <Enter>. Pressing <Enter> on its own will save and close the dialog.

Carbon - Model Options (Building)

Carbon tab on [Model Options](#) dialog

LCA

LCA time of study

Enter the number of years over which the optimisation [LCA \(Simple\)](#) optimisation KPI should be calculated. Note that this setting does not affect [One Click LCA](#) outputs or any of the [embodied carbon outputs](#) on the Cost and Carbon screen.

Glazing Carbon Model

Glazing CO2 embodied carbon emissions

Enter the carbon emissions for glazing as kgCO₂ per kg of glass. This value will be used for all glazing in the model in the [embodied carbon emissions calculations](#).

Note: When calculating the embodied CO₂ per area of glazing, DesignBuilder uses the standard glass density value of 2,500 kg/m³ along with the total thickness of the glass layers selected for the Glazing component.

Glazing coating embodied CO2

Enter the carbon emissions for glazing coatings (kgCO₂/m² or lb/m²) of coated glass. This value will be used for all glazing in the model in the [embodied carbon emissions calculations](#).

Note: This extra coating factor is currently only applied to low emissivity coatings. Assessment is based on assumptions built into DesignBuilder which compare the glazing U-value with that of a typical unit without Lo-e coating.

Renewables

PV Panels

The embodied carbon of all PV panels in the model can be defined here in kgCO₂/m² or lb/ft².

One Click LCA

Correlate materials in report by construction

Check this check box if you wish the source construction to be included in the "Comments" column of the One Click LCA report, as shown below.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
CLASS	MATERIAL	QUANTITY	QTY_TYPE	THICKNESS_MM	THICKNESS_IN	LENGTH_M	LENGTH_FT	AREA_M2	AREA_SQ_FT	VOLUME_M3	VOLUME_CU_FT	VOLUME_CU_YD	UNIT_PCS	MASS_KG	COMMENT
1	SLAB	Urea Formaldehyde Foam	600 M2	132.7	5.224409483	0	0	600	6458.34625	79.62	2812.165425	104.1390268	3		Project ground floor
2	SLAB	Cast Concrete	600 M2	100	3.9370079	0	0	600	6458.34625	60	2119.190222	78.477036	3		Project ground floor
3	SLAB	Floor/Roof Screed	600 M2	70	2.75590553	0	0	600	6458.34625	42	1483.433156	54.9339252	3		Project ground floor
4	SLAB	Timber Flooring	600 M2	30	1.18110237	0	0	600	6458.34625	18	635.7570667	23.5431108	3		Project ground floor
5	SLAB	Cast Concrete (Dense)	600 M2	100	3.9370079	0	0	600	6458.34625	60	2119.190222	78.477036	3		Project internal floor
6	WALL	Gypsum Plasterboard	420 M2	25	0.984251975	0	0	420	4520.842375	10.5	370.8582889	13.7334813	8		Project partition
7	WALL	Air gap 10mm	210 M2	100	3.9370079	0	0	210	2260.421188	21	741.7165779	27.4669626	4		Project partition
8	WALL	Brickwork Outer	615.2801191 M2	102.5	4.035433098	0	0	615.2801191	6622.820083	63.06621221	2227.488338	82.48749009	17		Project wall
9	WALL	XPS Extruded Polystyrene - CO2 Blowing	615.2801191 M2	100	3.9370079	0	0	615.2801191	6622.820083	61.52801191	2173.159354	80.47560009	17		Project wall
10	WALL	Concrete Block (Medium)	615.2801191 M2	100	3.9370079	0	0	615.2801191	6622.820083	61.52801191	2173.159354	80.47560009	17		Project wall
11	WALL	Gypsum Plastering	615.2801191 M2	13	0.511811027	0	0	615.2801191	6622.820083	7.998641548	282.510716	10.46182801	17		Project wall
12	WINDOW	Generic LOW IRON 4MM	240.93765 M2	4	0.157480316	160.6251	526.9852362	240.93765	2593.431281	0.9637506	34.03951414	1.26058176	32		Project external glazing
13	WINDOW	Generic CLEAR 12MM	240.93765 M2	12	0.472440948	160.6251	526.9852362	240.93765	2593.431281	2.8912518	102.1185424	3.781614527	32		Project external glazing
14	WINDOW	Painted Oak	20.86251 M2	20	0.78740158	321.2502	1053.970472	20.86251	224.5621887	0.4172502	14.73720907	0.545742649	64		Painted Wooden window frame
15	SLAB	Plywood (Heavyweight)	600 M2	10	0.39370079	0	0	600	6458.34625	6	211.9190222	7.8477036	3		Project semi-exposed ceiling
16	SLAB	MW Glass Wool (rolls)	600 M2	131.9	5.19291342	0	0	600	6458.34625	79.14	2795.211903	103.5112105	3		Project semi-exposed ceiling
17	SLAB	Cast Concrete (Lightweight)	600 M2	100	3.9370079	0	0	600	6458.34625	60	2119.190222	78.477036	3		Project semi-exposed ceiling
18	SLAB	Air gap >=25mm	600 M2	200	7.8740158	0	0	600	6458.34625	120	4238.380445	156.954072	3		Project semi-exposed ceiling
19	SLAB	Plasterboard	600 M2	13	0.511811027	0	0	600	6458.34625	7.8	275.4947289	10.20201468	3		Project semi-exposed ceiling
20	ROOF	Clay Tile (roofing)	731.1332869 M2	25	0.984251975	0	0	731.1332869	7869.853203	18.27833217	645.5877137	23.90715553	5		Project unoccupied pitched roof
21	ROOF	Air gap 10mm	731.1332869 M2	20	0.78740158	0	0	731.1332869	7869.853203	14.62266574	516.470171	19.12572443	5		Project unoccupied pitched roof
22	ROOF	Roofing Felt	731.1332869 M2	5	0.196850395	0	0	731.1332869	7869.853203	3.655666434	129.1175427	4.781431106	5		Project unoccupied pitched roof
23	SLAB	External Rendering	33.18 M2	25	0.984251975	0	0	33.18	357.1465476	0.8295	29.29780483	1.084945023	1		Project external floor
24	SLAB	MW Stone Wool (rolls)	33.18 M2	148.2	5.834645708	0	0	33.18	357.1465476	4.917276	173.677387	6.431554095	1		Project external floor
25	SLAB	Timber Flooring	33.18 M2	5	0.196850395	0	0	33.18	357.1465476	0.1659	5.859560905	0.216989005	1		Project external floor

Note: When checking this option, the One-click LCA spreadsheet report will generally be much longer because it records each use of a material rather than simply summing the total. Although One Click LCA is able to import large data sets, you may wish to consider this when working with very large models.

See also the [Program and Model Options](#) tutorial

Interzone Airflow

The **Interzone Airflow** dialog can be accessed from the [Tools menu](#)  when the [Scheduled](#) natural ventilation model option is set.

Interzone Airflow provides simplified treatment of air exchange between zones when working with the [Scheduled](#) natural ventilation model option. For example, when extracting air from a corridor zone it is may be necessary to model the flow of "make-up" air from various adjacent occupied spaces such as offices due to the reduced pressure in the corridor. The **Interzone Airflow** dialog allows time-varying flows between the various zones to be defined so that EnergyPlus can calculate the flow rate at each time step in the simulation.

Flows on the **Interzone Airflow** dialog are applied to Simulations as well as Heating and Cooling design calculations.

Note: Interzone airflows only affect the energy balance of the **receiving** zone. There is no effect on the **source** zone. Interzone airflows can be complementary and include multiple zones, but **the balancing of flows between zones is left to the user's discretion**.

Interzone airflows are modelled in EnergyPlus using **ZoneMixing** statements.

Tip: More advanced interzone airflow calculations are possible using [Calculated](#) natural ventilation for multi-zone airflow with or without HVAC system operation.

Units

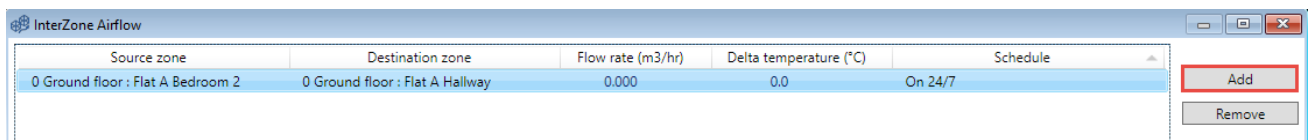
If you are using [SI units](#) then you can select the airflow units using the drop list on the right side of the dialog. Choose from the following options:

- m3/hr
- l/s
- m3/s

If you are using [IP units](#) then the airflow rate units are ft3/min.

Add An Airflow Path

To add an airflow path, click on the **Add** button to the right of the **Interzone Airflow** dialog, below.



The **Add New Airflow Path** dialog, allows you to define each airflow path as it is added by selecting the source and destination zones and entering the flow rate and a schedule to define how the flow varies with time. A delta temperature can also be entered to provide a simple temperature-based control option.

Source zone

Select the “source” zone that exhausts the Interzone airflow. To select the zone, click on the ellipsis on the right, highlighted in the screenshot below.

Source zone



Destination zone

Select the “destination” zone that receives the Interzone airflow. To select the zone, click on the ellipsis on the right, highlighted in the screenshot below.

Destination zone



Flow rate

Enter the full design volume flow rate using the [units](#) selected on the main Interzone Airflow dialog. The design volume flow rate is the maximum amount of mixing air expected. The design value is modified by the schedule fraction (below).

Delta temperature

This setting controls when air from the source zone moves into the receiving zone. It is a delta temperature (in delta C or delta F). If the value is positive, the temperature of the zone from which the air is being drawn (source zone) must be **Delta temperature** warmer than the receiving zone air or else no mixing occurs. If this value is negative, the temperature of the source zone must be **Delta temperature** cooler than the receiving zone air or else no mixing occurs. If this parameter is zero, mixing occurs regardless of the relative zone temperatures.

Schedule

Select the schedule that modifies the maximum design flow rate parameter. Values in the schedule should be between 0.0 and 1.0.

Editing Existing Airflow Paths

Each of the above data items can be edited retrospectively after an Airflow path has been added by clicking on the data on the main Interzone Airflow dialog. For example clicking on the **Source/Destination zone** data opens the zone selection Browse dialog allowing the currently set zone to be changed. Likewise clicking on the **Schedule** data opens the Schedule Browse dialog allowing you to select a different Schedule. The **Flow rate** and **Delta temperature** values can also be edited in situ, again by clicking on the data in the grid.

To delete an Airflow path, select its line in the grid and click on the **Remove** button on the right of the **Interzone Airflow** dialog.

Outputs

The thermal impact of interzone airflows can be viewed on the DesignBuilder results screen through the **Internal Natural Ventilation** output.

Components And Templates

- "Components" on page 1353
- "Templates" on page 1491

- "Import Components/Templates" on page 1514
- "Export Components/Templates" on page 1515
- "Library Management" on page 1517

Components

Components represent individual building elements and can be referenced from other components, [templates](#) or [model data](#). The following components are available:

- [Constructions](#)
- [Materials](#)
- [Glazing](#)
- [Panels](#)
- "Pane Group Components" on page 1408
- [Schedules](#)
- "Simulation Hourly Weather Data" on page 1418
- [Profiles](#)
- [Textures](#)
- [Window gas](#)
- "Metabolic Rates" on page 1441
- [Local shading](#)
- [Window blinds](#)
- [Vents](#)
- [Holidays](#)
- "Ground Domain" on page 1456
- "Kiva Foundation" on page 1463
- "Kiva Foundation Settings" on page 1475
- "Moisture Transfer Data" on page 1477
- "Fuel Emission Factors" on page 1483
- "Horizon Component" on page 1489
- "Currency" on page 1490

Library components are loaded into the model at the time the file is created and from then on they are associated specifically with the model and so are known as **Model components**. Any later change made to the Library components does not affect existing building models because they reference their own set of Model components.

Managing component data

You can edit and move component data around in various ways:

1. Add, Edit, Delete, [Import](#) and [Export](#) Component library data from the [Opening Screen - Component Library](#) tab.
2. If a model file is loaded you can edit Model components from the Components tab in the Navigator panel ([Learning mode](#) switched off) or by using the Data management panel on the right of the screen ([Learning mode](#) switched on).
3. In either case, **to add a component** open up the component type you wish to define by clicking on the + to the left of the component icon, then click on the category of component you wish to define. Now click on the Green '+' to add a component of the selected category.
4. Import/export Model component using the **File > Export > Library data** menu command to create a .ddf file with the .dsb model file open.
5. To import the previously exported Model component data to the Component library close the .dsb file and [import](#) the .ddf file to the library.
6. Load the latest library components into your model using the **Import library components and templates to model** command from the Tools menu when a model is loaded. Any new data that has been added to the library since the model file was created will be imported to the model. Also if there have been any changes to library component data, this new data will overwrite the original data.

Important Note: any component you add while a model is open will be added to Model components and not to the Component library. It will only be added to the library and hence to new models if you export the new component model data in .ddf form and import the .ddf file to the library before creating the new model.

Note: As well as importing DDF files created directly by DesignBuilder more advanced users can also create their own component and template data sets (outside DesignBuilder) for importing. See the [Importing Custom Templates and Components](#) topic for more information on the process involved.

Colour codes

Components are coded by colour in all selection lists as follows:

- **Red** shows user data, i.e. data that was not supplied by DesignBuilder.
- **Green** indicates that the data is derived from a reputable national or international source (e.g. ASHRAE, CIBSE).
- **Blue** indicates that the data is generated internally by DesignBuilder.
- **Black** is used for all other data.

See also [Model Data Colour](#).

Construction Components

The thermal properties of the construction are calculated based on [layers](#), each layer having a [material](#) reference and thickness. The combination of thickness and material gives the thermal characteristics of the layer. Layers can include repeating thermal bridges.

Construction components also store [derived](#) data on the thermal properties of the overall layered construction for example U-value, convective heat transfer coefficients etc.

See also the [Create and edit constructions](#) tutorial

There are 6 tabs on the Constructions dialog:

- [Layers](#)
- [Surface Properties](#)
- [Image](#)
- [Calculated](#)
- [Internal Source](#)
- [Condensation Analysis](#)

Construction component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the construction icon in the model data,
- by clicking on the Edit button when selecting constructions from a list.

Construction Layer Data

Layers tab on [Constructions](#) Dialog.

General

See also the [Create and edit constructions](#) tutorial

Name

Enter a unique name for this construction.

Source

A text field where you can enter the source of the construction data. For example if it came from a national energy code or certification scheme then it might be useful to say something about that here.

Category

Choose a category for the construction.

Region

If the construction is only relevant in a particular region then select that region here.

Colour

The colour selected here is used to shade any surfaces that use this construction on the Constructions and Glazing [Model Data Visualisation](#) screen.

Definition

You can select the way you would like to define the construction. The options depend on whether you are using EnergyPlus or SBEM Analysis type and whether it is a construction from the **Floor (ground)** or **Below grade wall** categories.

For EnergyPlus the allowable definition methods are:

- **1-Layers** where the construction is defined by specifying layers each having a material and thickness.
- **5-C-factor** when the category is **Below grade wall**
- **6-F-factor** when the category is **Floor (ground)**

Calculation Settings

Simulation solution algorithm

Some constructions may need a particular solution algorithm to obtain most realistic results. For example PCMs require a finite difference solution. However simulations may run slowly if all surfaces are simulated using the finite difference solution method. This option allows you to override the overall simulation [solution algorithm](#) for any surfaces using this construction.

Options are:

- **1-Default** where surfaces using this construction receives no special treatment and is simulated using the default solution algorithm.
- **2-CTF** where surfaces using this construction are simulated using the CTF algorithm regardless of the general solution method selected.
- **3-Finite difference** where surfaces using this construction are simulated using the finite difference algorithm regardless of the general solution method selected.
- **4-HAMT** where surfaces using this construction are simulated using the **Combined Heat And Moisture Finite Element (HAMT)** solution method. This coupled heat and moisture transfer and storage solution uses a one dimensional finite difference solution in the construction elements. It

requires further moisture transfer properties which are defined in the "Moisture Transfer" on page 1393 component.

- **5-EMPD** where surfaces using this construction are simulated using the **Moisture Penetration Depth Conduction Transfer Function (EMPD)** solution method. This sensible heat diffusion and an inside surface moisture storage algorithm also needs additional moisture material property information defined in the "Moisture Transfer" on page 1393 component.

You may wish to use this option to select the **2-Finite difference** option for constructions which have one or more of:

- [Phase change materials](#) (PCMs) - these can be simulated with the CTF option but the phase change characteristics will be ignored so any construction using PCMs would normally have the **3-Finite difference** option set.
- We understand that Detailed HVAC [Chilled ceilings](#) may benefit from use of the finite difference algorithm.
- Constructions using thin sheets of metal may not be treated accurately by the default CTF algorithm and for most accurate treatment you may wish to use finite difference for such constructions.

Bear in mind that the simulation times will increase when using the finite difference option even for just a few surfaces. If you are in doubt, you may find it worthwhile experimenting with the 2 solution methods, comparing hourly results, to see the effect of changes. Depending on the case, it may or may not be worth the extra simulation time required by the finite difference option.

Note 1: Any settings made here will only take effect if the [Allow individual constructions to override solution method](#) model option is checked.

Note 2: This setting only applies to simulations. Heating and Cooling design calculations use the CTF algorithm for all surfaces.

Includes metal cladding

If the construction can be defined as a metal cladding system then you should check this option. This option affects [linear thermal bridging at junctions](#) involving surfaces made up of this construction.

Definition of Metal cladding

Constructions involving metal cladding are roof or wall systems where metal forms an integral part of the construction, such as metal twin skin systems where the insulation is located between the metal skins and where the metal skins are typically in the range 0.4 mm to 1.2 mm. Cladding with z-spacers would come into this category as would composite metal panel systems.

If the metal is simply used as an external shield against weather, such as a rainscreen, this is not classed, for the purposes of calculations as "metal cladding".

Metal cladding systems are divided into two broad categories, these being:

- a. built up metal cladding systems involving rail and bracket or z-spacer systems with insulation within the panels
- b. composite panel metal cladding systems with insulation inside the panels

Layers

Set the number of layers first, then select the material and thickness for each layer. Use the Controls in the Info Panel to insert and delete layers. For example to delete a layer, first click on the material for the layer to identify which layer you want to delete, then click on Delete layer.

You can create constructions with up to 10 layers.

Note: You should not include surface resistance (film coefficient) layers to represent the resistance of the air films adjacent to the inner and outer surfaces. These are [included](#) automatically by DesignBuilder.

Layer order

You should define the layers in the same order they appear in the actual construction starting with the outermost layer and finishing with the innermost. The outermost layer will be positioned adjacent to outside (or in the case of [Semi-exposed walls](#)) adjacent to the [Semi-exterior unconditioned](#) zone.

For interior partition surfaces the order of the layers is not defined directly from the usage of the construction in a surface, so in this case the layer order is determined through a series of checks to see which zone is relatively 'more external'. For example, if one zone is unconditioned and the other not then the partition layers are ordered so that the outermost layer is next to the unconditioned zone. When both zones are occupied, a similar check is made using heating and cooling setpoint temperatures. If after all these checks the zones are still both equally 'internal' then the direction of the layers will be set up in an arbitrary way.

Tip: When modelling an asymmetric partition between 2 zones that have identical zone type and heating and cooling setpoints and you need to specify the exact order of the partition layers, you can make a small difference in heating or cooling setpoint temperature between the 2 zones. For example, consider the case of an occupied atrium zone adjacent to an office zone and the partition between the 2 has a thermal mass layer that must be adjacent to the atrium. This situation could be modelled for example by setting the heating setpoint of the atrium to be 0.1°C lower than the office (or if the heating setpoints are the same then the cooling setpoint could be set 0.1°C higher) and by making the thermal mass layer to be the outermost layer in the construction layer definition.

Detailed description of asymmetric partition layer order algorithm

DesignBuilder sets up the layer order for asymmetric partitions such that the outermost layer is next to the zone which it considers to be the more "external" of the two zone. The method used to decide which of the zones is relatively more external is as follows:

Step 1. Check whether zones are heated or not (**Heating** checkbox on the HVAC tab).

- a. If only one zone has heating switched on then the outermost layer is set to be next to the unheated zone.
- b. If both zones are heated then the [Heating setpoint temperatures](#) are checked for both zones. The zone with the lower heating temperature setpoint is considered to be the most external and the outermost layer is set next to it.
- c. If both zones are heated and have the same heating setpoint temperature then go to Step 2.
- d. If both zones are unheated then go to Step 2.

Step 2. Check whether cooling is active in the zones (**Cooling** checkbox on the HVAC tab).

- a. If only one zone has cooling switched on then the outermost layer is set to be next to the uncooled zone.
- b. If both zones are cooled then the [Cooling setpoint temperatures](#) are checked for both zones. The zone with the higher cooling setpoint temperature is considered to be the most external and the outermost layer is set next to it.
- c. If both zones are cooled and have the same cooling setpoint temperature then go to Step 3.
- d. If both zones don't have cooling then go to Step 3.

Step 3. Both zones are considered to be "equally external" and the direction of the layers are set up in an arbitrary way.

Note: The above checks are carried out before the simulation starts and scheduled setpoint values are not used. The setpoints referred to above are the "headline" setpoint values entered under the [Environmental control](#) header on the Activity tab.

Thickness

This field characterizes the thickness of the material layer. This is the dimension of the layer in the direction perpendicular to the main path of heat conduction.

Note: Modelling layers thinner (less) than 0.003 m is not normally recommended; rather, add those properties to one of the adjacent layers.

If the material referenced is defined as a [simple R-value](#) then the layer thickness does not affect the thermal performance of the construction.

If the material is defined as having a [fixed thickness](#) then the layer thickness is loaded automatically when the material is selected and the thickness does not have to be entered and cannot be edited.

Photovoltaic Panel

When the **Building Integrated Photovoltaics** Construction category is selected, you can define the properties of a BIPV panel below the data for the outermost layer.

Tip: See "How to Model Photovoltaic (PV) Systems" on page 523 for a description of the process involved in modelling BIPV systems.

Performance type

Select from:

- **1-Simple**, and
- **2-Equivalent One-Diode**.

Depending on the selection you will be able to select from either a simple or equivalent one-diode definition of the panel.

Tip: BIPV panels are included in the electrical generation side of the model by specifying them on the [Generator list](#) tab of the Electric load centre dialog when one of the d.c. [Buss types](#) is selected.

Performance model

Choose the component to be used to define the performance of the collector. When the **1-Simple** performance type is selected, select from the list of previously defined [Simple Photovoltaic components](#), or when **2-Equivalent One-Diode** is selected then choose from one of the [Equivalent One-Diode components](#).

Heat transfer integration mode

The PV model allows for different ways of integrating with other EnergyPlus heat transfer surfaces and models and calculating cell temperature. Choose from:

- **1-Decoupled**, where the cell temperature of modules in the array is computed based on an energy balance relative to NOCT conditions.
- **2-Decoupled Ulleberg Dynamic**, where the cell temperature is calculated based on a dynamic model developed by Ulleberg12.
- **3-Integrated Surface Outside Face** where the cell temperature is obtained from the outside face of the surface. Energy exported by the module is removed from the heat transfer surface.
- **4-Integrated Exterior Vented Cavity** The cell temperature is obtained from the [exterior baffle](#) temperature in the naturally ventilated exterior cavity model. Energy exported by the module is removed from the exterior baffle surface.

To model PV panels that are integrated in some way with the underlying construction, choose one of the integrated heat transfer mode options. Otherwise, the decoupled options will have a similar effect as drawing a separate PV panel on top of the surface at building level.

Modules in parallel

This field is the number of series-wired strings of PV modules that are in parallel to form the PV array. The product of this field and the next field should equal the total number of modules in the array.

The **Modules in parallel** and **Modules in series** data are only used for **2-Equivalent One-Diode** PV arrays.

Modules in series

This field is the number of modules wired in series (on each string) to form the PV array. The product of this field and the previous field should equal the total number of modules in the array.

Bridging

You can add repeating thermal bridging to any layer to model the effect of a relatively more conductive material bridging a less conductive material. For example wooden joists bridging an insulation layer.

Note: bridging effects are not yet used directly in EnergyPlus, but are used in energy code compliance checks requiring U-values to be calculated according to BS EN ISO 6946. However see below for a workaround to this limitation.

Method for approximating repeating bridging effects in simulations

Although EnergyPlus cannot allow repeating bridging directly in simulations it is possible to approximate the effect of bridging by adjusting the thickness of the insulation to a value that gives the same U-value as the bridged construction calculated using BS EN ISO 6946. This is easily achieved by following these steps:

1. Define the construction including bridging data as in the screenshot below.

Constructions Data

Layers Surface properties Image Calculated

General

Name Example Wall

Source

Category Walls

Region INDIA

Layers

Number of layers 4

Outermost layer

Material Brick - Brickwork, Outer Leaf

Thickness (m) 0.1000

☐ Bridged?

Layer 2

Material XPS Extruded Polystyrene -

Thickness (m) 0.2

☒ Bridged?

Material Metals - steel

Percent bridging 4

Layer 3

Material Concrete Block (Medium)

Thickness (m) 0.1000

☐ Bridged?

Innermost layer

Material Gypsum Plastering

Thickness (m) 0.0130

☐ Bridged?

- Note the construction bridged U-value on the Calculated tab, see below.

Constructions Data

Layers Surface properties Image Calculated

Inner surface

Convective heat transfer coefficient (W/m²-K) 2.152

Radiative heat transfer coefficient (W/m²-K) 5.540

Surface resistance (m²-K/W) 0.130

Outer surface

Convective heat transfer coefficient (W/m²-K) 19.870

Radiative heat transfer coefficient (W/m²-K) 5.130

Surface resistance (m²-K/W) 0.040

No Bridging

U-Value surface to surface (W/m²-K) 0.161

R-Value (m²-K/W) 6.400

U-Value (W/m²-K) 0.156

With Bridging (BS EN ISO 6946)

Km - Internal heat capacity (KJ/m²-K) 134.8000

Upper resistance limit (m²-K/W) 4.413

Lower resistance limit (m²-K/W) 0.627

U-Value surface to surface (W/m²-K) 0.477

R-Value (m²-K/W) 2.520

U-Value (W/m²-K) 0.397

Record the bridged U-value

3. Going back to the layers tab, switch off all bridging and set the U-value to the bridged U-value by clicking on the **Set U-value** link in the Info panel.

Layers Surface properties Image Calculated

General

Name **Example Wall**

Source

Category Walls

Region INDIA

Layers

Number of layers 4

Outermost layer

Material Brick - Brickwork, Outer Leaf

Thickness (m) 0.1000

☐ Bridged?

Layer 2

Material XPS Extruded Polystyrene - CO

Thickness (m) 0.2

☐ Bridged? **1. Switch off bridging**

Layer 3

Material Concrete Block (Medium)

Thickness (m) 0.1000

☐ Bridged?

Innermost layer

Material Gypsum Plastering

Thickness (m) 0.0130

☐ Bridged?

2. Set the U-value to bridged value

Info Data

Construction Layers

Set the number of layers first, then select the material and thickness for each layer.

[+ Insert layer](#)

[X Delete layer](#)

Bridging

You can also add bridging to any layer to model the effect of a relatively more conductive material bridging a less conductive material. For example wooden joists bridging an insulation layer.

Note that bridging effects are NOT used in EnergyPlus, but are used in energy code compliance checks requiring U-values to be calculated according to BS EN ISO 6946.

Energy Code Compliance

You can calculate the thickness of insulation required to meet the mandatory energy code U-value as set on the Energy Code tab at site level.

This calculation identifies the 'insulation layer' as the layer having the highest r-value and requires that no bridging is used in the construction.

[Set U-Value](#)

4. Enter the recorded bridged U-value (0.397 W/m²-K in this case) on the dialog and press OK.

5. Confirm the reduced insulation thickness when prompted.
6. Check the new insulation thickness and U-value of the construction. The insulation thickness should be reduced as per the pop up message and the U-value should be the same as the bridged value you recorded earlier. The construction should now behave in a very similar way to the bridged construction in heating and cooling design calculations and in simulations.

C-Factor And F-Factor Constructions

Building energy code and standards, such as ASHRAE 90.1 used for LEED in DesignBuilder, require underground wall constructions (aka below-grade walls) and slab-on-grade floors not to exceed certain maximum values of C-factor and F-factor.

EnergyPlus provides a simplified approach to create equivalent constructions and model the ground heat transfer through below-grade walls and slab-on-grade floors for the building energy code compliance calculations. The approach taken behind the scenes in EnergyPlus is to create constructions based on the user-defined C-factor or F-factor with two layers: one concrete layer (0.15 m thick) with thermal mass, and one fictitious insulation layer with no thermal mass. So it is only necessary to set up C-factor and F-factor construction input settings and allow EnergyPlus to take care of the details.

ASHRAE 90.1 defines below-grade wall, which needs to use C-factor construction, as the portion of a wall in the building envelope that is entirely below the finish grade and in contact with the ground. C-factor constructions differ from the usual wall constructions in that they describe an entire construction rather than individual layers. This object is used when only the wall height (depth to the ground) and the C-factor are available.

ASHRAE 90.1 defines slab-on-grade floor, which needs to use F-factor construction, as the portion of a slab floor of the building envelope that is in contact with the ground and that is either above grade or is slightly below the grade (less than or equal to 24 inch, or 600 mm, below the final elevation of the nearest exterior grade). The F-factor represents the heat transfer through the floor, induced by a unit temperature difference between the outside and inside air temperature, on per linear length of the exposed perimeter of the floor.

Both C-factor and F-factor constructions exchange heat on their outer boundary with the ground temperature which is defined by the site level monthly Ground > [FCfactorMethod ground temperatures](#) and which can be accessed from the site details section at site level in DesignBuilder. During the simulation these C-factor and F-factor surfaces exchange heat with the FCfactor ground temperatures only. These FCfactorMethod ground temperatures can be either read from the header of the hourly weather file used or taken from user's manual input.

Note: [Basement ground floors](#) were introduced into DesignBuilder for ASHRAE 90.1 models only in order to distinguish floors from slab-on-grade floors and are normally outside the scope of slab-on-grade floor definition described in ASHRAE 90.1 and are not regulated by the 90.1 Standard. The baseline model takes the same construction for basement ground floor as that used in the proposed model, where a standard layered floor construction is normally used.

C-Factor (Underground Wall)

C Factor walls differ from usual walls in that they describe an entire construction rather than individual layers. C Factor walls are used when only the wall height (depth to the ground) and the C-factor are available. They access a model that creates an equivalent layer-by-layer construction for the underground wall to approximate the heat transfer through the wall considering the thermal mass of the earth soil.

C-Factor

C-Factor is the time rate of steady-state heat flow through unit area of the construction, induced by a unit temperature difference between the body surfaces. The C-Factor unit is $W/m^2 \cdot K$. The C-factor does not include soil or air films. ASHRAE Standard 90.1 and California Title 24 specify maximum C-factors for underground walls depending on space types and climate zones.

(Auto generated) Height

This field describes the height of the underground wall, i.e. the depth to the ground surface (in m or ft).

F-Factor (Ground Floor)

F Factor floors differ from the usual ground floor constructions in that they describe an entire construction rather than individual layers. F Factor floors are used when only the floor area, exposed perimeter, and the F-factor are available. They access a model internal to EnergyPlus that creates an equivalent layer-by-layer construction for the slab-on-grade or underground floor to approximate the heat transfer through the floor considering the thermal mass of the earth soil.

Note: F-Factor constructions should only be selected for Ground adjacent surfaces with exposed perimeter greater than zero.

The area of the slab-on-grade or underground floor and the exposed perimeter length (direct contact with ambient air) are both calculated automatically by DesignBuilder behind the scenes.

F-Factor

F-Factor represents the heat transfer through the floor, induced by a unit temperature difference between the outside and inside air temperature, on the per linear length of the exposed perimeter of the floor. The unit for this input is W/mK . ASHRAE Standard 90.1 and California Title 24 specify maximum F-factors for slab-on-grade or underground floors depending on space types and climate zones.

Note: F-Factor constructions cannot be used in ground surfaces which are adjacent to a [Kiva Foundation](#). Generally F-Factor and Kiva are separate and non-compatible methods to account for ground 2-D/3-D conduction and should not be used together.

Surface Properties

Surface properties tab on [Constructions](#) Dialog.

Data on this tab allows the inside and outside surface convection properties of the construction to be set to fixed values. This may be useful where specific known convection properties are to be used or for validation exercises where DesignBuilder outputs are to be compared with those for other software in which particular convection coefficients have been used.

In most cases you should leave the check boxes on this tab unchecked.

Note: The values entered here do not affect the heat transfer coefficients reported on the [Calculated](#) tab.

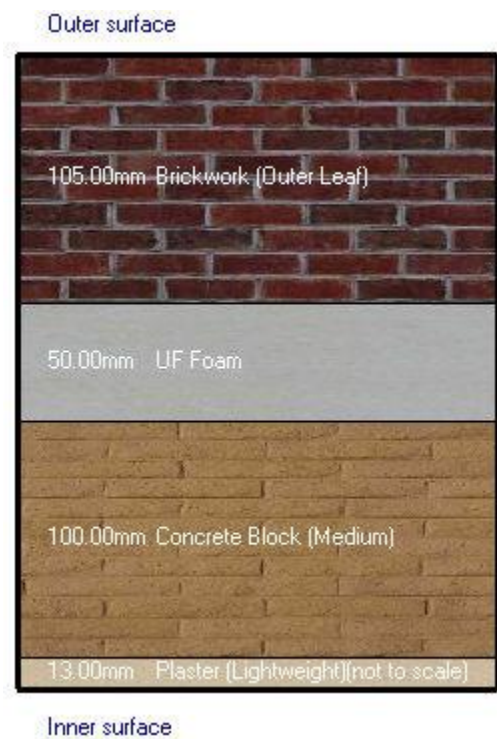
Tip: To fix the outside convection heat transfer coefficient for all surfaces using this construction, check the **Fix convective heat transfer coefficient** check box under the Outside Surface header and enter the coefficient. Likewise for inside properties.

Note: If the **SimpleCombined** Outside convection option has been selected **at building level on the Constructions tab under [Surface Convection](#) or on the Calculation options or Model options dialogs** the outside convective heat transfer coefficients set will actually be used as a combined convective plus radiation coefficient.

Construction Image

Image tab on [Constructions](#) Dialog.

This tab shows the construction to scale (except for very thin layers) using material textures as specified.



Note: Bridging elements are not displayed on this image.

Construction Calculated Data

Calculated tab on [Constructions](#) Dialog.

This tab provides further information on the heat transmission properties of the construction.

Inner surface	
Convective heat transfer coefficient (W/m ² -K)	2.152
Radiative heat transfer coefficient (W/m ² -K)	5.540
Surface resistance (m ² -K/W)	0.130
Outer surface	
Convective heat transfer coefficient (W/m ² -K)	19.870
Radiative heat transfer coefficient (W/m ² -K)	5.130
Surface resistance (m ² -K/W)	0.040
No Bridging	
U-Value surface to surface (W/m ² -K)	0.372
R-Value (m ² -K/W)	2.856
U-Value (W/m²-K)	0.350
With Bridging (BS EN ISO 6946)	
Thickness (m)	0.2925
K _m - Internal heat capacity (kJ/m ² -K)	134.8000
Upper resistance limit (m ² -K/W)	2.856
Lower resistance limit (m ² -K/W)	2.856
U-Value surface to surface (W/m ² -K)	0.372
R-Value (m ² -K/W)	2.856
U-Value (W/m²-K)	0.350

U-values and R-Values are shown including and excluding the effect of surface resistance and are calculated with and without bridging effects.

Note that when working with **library constructions**, the outer surface resistance is calculated assuming a 'typical' exposure to wind. When working with **model constructions** the outer surface resistance is calculated using the wind exposure as defined on the Location tab at site level.

Convective heat transfer coefficients

The convective heat transfer coefficients displayed on this tab are only used in EnergyPlus when the 'CIBSE' Inside/Outside convection algorithm is selected. Otherwise EnergyPlus uses its' own convection algorithm as set in the simulation options and the transmission data displayed here is not used.

K_m - Internal heat capacity (thermal mass)

The K_m value is the effective heat capacity per unit area and is a measure of the thermal mass storage characteristics of the construction. The higher the number, the more heat will be stored for release later.

The heat capacity or kappa value per unit area, kJ/m²K, for a construction element is calculated following the UK NCM requirements as:

$$k_m = \sum (t_i d_i c_i) \times 10^{-6}$$

where:

The calculation is over all layers (i) in the element, starting at the inside surface and stopping at whichever of the following conditions is encountered first, possibly part way through a layer:

- The total thickness of the layers exceeds 100mm.
- The mid point of the construction is reached.
- An insulation layer is reached (defined as thermal conductivity ≤ 0.08 W/mK).

t_i is the thickness of layer (mm)

d_i is density of layer (kg/m³)

c_i is specific heat capacity of layer (J/kg·K)

Tip: In simple terms, K_m can be thought of as the thermal heat capacity per area of the innermost 100mm of material in a building element before the insulation layer(s). Or, in elements that are less than 200mm thick, of the innermost half of the construction layers that occur before the insulation layer.

Construction Cost Data

Cost tab on [Constructions Dialog](#)

Cost type

There are 2 ways to define the cost per area of a construction:

- **1-Auto calculate** where the cost per surface area of the construction is calculated based of the cost of the layers used in the construction, or,
- **2-Manual entry** where the cost per surface area of the construction is entered manually.

Cost per surface area

The cost per surface area is enabled the **2-Manual entry** cost type is selected. In this case you should enter the overall cost of the construction per area. Make sure to include labour and any other costs directly relating to this cost. When the **1-Auto calculate** cost type is used this cell is automatically loaded with the sum of the cost of the individual layers per area and greyed out.

Internal Source

Internal Source tab on [Constructions Dialog](#).

The internal source tab is visible only when the model is set to use [Detailed HVAC](#). It allows data on the position and nature of any heating or cooling pipes included within the construction. Internal sources are associated with one of these components:

- [Heated floors](#) or [Chilled ceiling](#) components by virtue of the construction being selected as the floor/ceiling in a zone with heated floor or chilled ceiling defined. For example in the case of a heated floor the heated floor component must be added to the appropriate HVAC zones, but also a floor construction with an internal source must be associated with at least one floor surface in that building zone. If you set up a model with a heated floor added to an HVAC zone whose equivalent building zone has any floor surfaces with a floor construction selected not having an internal source then an error will be generated.
- [Ground loop heat exchangers of the type 3-Surface](#) to define the location of the heat exchanger pipes.

Internal source

Check this option if the construction includes a heating or cooling source.

Layer after which the internal source is positioned

Enter a number between 1 and 1-the number of layers. Entering 1 means that the source is positioned between the outermost layer and the next layer. If you are defining a heated floor construction then you will normally enter 1-the number of layers. The easiest way to do this is to use the **Set source position below innermost layer** tool described below. When defining a chilled ceiling as a floor construction you should enter a value of 1 here.

Important Note: The internal source must be placed thermally closer to the inside surface than to the outside surface of the construction to allow it to be considered for inclusion in a zone's [radiant surface](#), [heated floor](#) or [chilled ceiling](#) system. In other words, if the thermal resistance between the inside surface

and the internal source is less than the resistance to the outside surface then the surface is considered to be a valid radiant surface to be included with the zone next to the inside surface and will be simulated as such. This rule applies to surfaces adjacent to outside as well as internal surfaces.

Dimensions

This field refers to the detail level of the calculation. Select from the 2 options below for a 1-D or 2-D solution:

- **1-D** is the default choice for a one-dimensional calculation. This is appropriate for hydronic heating/cooling when water heater/cooler performance is relatively unaffected by return and supply water temperatures.
- **2-D** triggers a two-dimensional solution. With 2-D solutions, the calculation of the conduction transfer functions (CTF) is fairly intensive and requires extra simulation time. Note that with the 2-D option the solution regime is two-dimensional internally but it has a one-dimensional boundary condition imposed at the inside and outside surface (i.e., surface temperatures are isothermal).

Note: Because the **1-D** solution method does not allow the return water temperature to be accurately calculated it is not recommended for when accurate plant energy consumption outputs are required. In this case choose the **2-D** option.

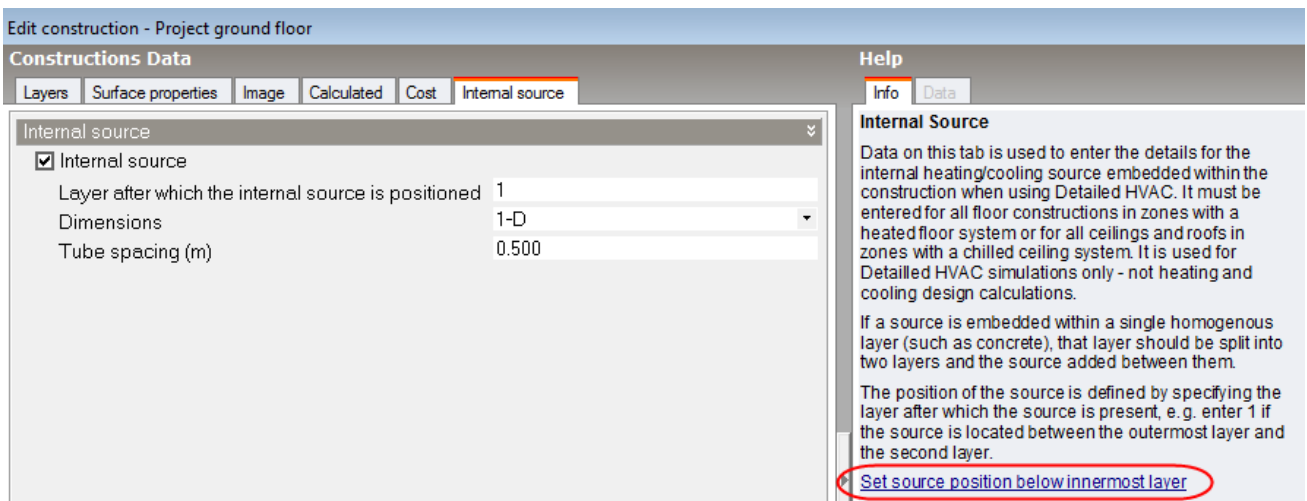
The **2-D** solution method generally takes about 20% longer to run a simulation relative to the **1-D** option.

Tube spacing

This field defines how far apart (in m or in) the hydronic tubing are spaced in the direction perpendicular to the main direction of heat transfer. This parameter is used when either the **1-D** or **2-D** Dimension setting is selected (above).

Set source position below innermost layer

A tool is provided in the info panel when using learning mode which calculates which is the innermost layer and adds the source between this and the next layer. For example in a chilled ceiling construction you might have 0.5 mm steel ceiling panel as the innermost layer with 3 other layers. In this case when clicking on the **Set source position below innermost layer** link, the innermost layer value would be set to 3.



Note: Constructions with an internal source cannot be used in ground surfaces which are adjacent to a [Kiva Foundation](#). You can use surfaces with an internal source in any other surface in Kiva simulations though.

Condensation Analysis

Condensation analysis tab on [Constructions Dialog](#).

DesignBuilder Condensation Analysis allows you to generate reports on the occurrence of interstitial and surface condensation based on the ISO 13788 standard.

The analysis is accessed from the **Condensation analysis** tab on the Constructions dialog. Controls are provided under the [Options](#) header to control the outputs generated and there are also tools to allow you to set internal and external boundary conditions for the condensation calculations.

Note: The moisture [vapour resistance](#) properties for each material used in the construction layers must be entered on the materials dialog for accurate condensation analysis results. The use of manufacturers data is recommended, particularly for materials with high vapour resistance.

Options

Data under the Options header allows you to set various calculation and display options for the condensation analysis.

Show Glaser diagram

Check this option if you would like to view a Glaser diagram showing a graphical section through the construction with temperature, partial vapour pressure and saturated vapour pressure data plotted.

Month

When viewing a Glaser diagram select the month from the drop list. The default option is **13-Worst case month** where DesignBuilder displays results for the month with the worst case of moisture build up.

Show interstitial condensation details

If you would like to view details of the monthly and cumulative condensation build up for each interface with condensation then check this option. In this case, if condensation is present you will see output similar to:

Condensation Interfaces

Number of condensation interfaces: 2

Distance of condensation interface 1 from the external surface: 100mm

Distance of condensation interface 2 from the external surface: 107,95mm

Detailed Results: Interstitial Condensation

For condensation interface 1

	Monthly Condensation (g/m2)	Cumulative Condensation (g/m2)
Oct	-302,38	0,00
Nov	-213,07	0,00
Dec	-186,61	0,00
Jan	-1,09	0,00
Feb	-192,27	0,00
Mar	-272,62	0,00
Apr	-365,28	0,00
May	-479,81	0,00
Jun	-509,66	0,00
Jul	-554,60	0,00
Aug	-512,93	0,00
Sep	-394,15	0,00

Interface

If you requested interstitial condensation details then you should enter the interface number here. The definition of each interface with condensation is given under the **Condensation Interfaces** heading in the report.

Surface condensation calculation

To see surface condensation details check this box. In this case you should enter further details to define the boundary conditions as explained below.

Environment source

Select from:

- **1-Maintained internal environment**, where you can enter fixed monthly values for internal air temperature and RH
- **2-Maintained internal air temperature**, where you can enter fixed monthly values for internal air temperature only.
- **3-Boundary data**, Where boundary data is used.

In all cases monthly external conditions are defined in the Boundary data.

Fixed internal air temperature

Enter the constant internal air temperature for the Surface condensation calculation. This setting is available when **Environment Source** is set to **1-Maintained internal environment** or **2-Maintained internal air temperature**.

Fixed internal RH

Enter the constant internal relative humidity for the Surface condensation calculation. This setting is available when **Environment Source** is set to **1-Maintained internal environment**.

Humidity class

Enter the humidity class for the Surface condensation calculation. This setting is available when **Environment Source** is set to **1-Maintained internal environment**. The allowed classes are as follows:

Humidity Class	Building Type
1	Storage areas
2	Offices, shops
3	Dwellings with low occupancy
4	Dwellings with high occupancy, sport halls, kitchens, canteens; buildings heated with un-flued gas heaters
5	Special buildings

Show surface condensation details

Check the Show surface condensation details checkbox to display a table similar to the one below in the report.

	Ext Temp	Ext RH (%)	Int Temp	Int RH (%)	Min Temp Factor	Min Tsi	Tsi
Jan	3.9	85	20.0	50	0.633	14.1	19.3
Feb	3.9	82	20.0	50	0.633	14.1	19.3
Mar	5.7	79	20.0	50	0.586	14.1	19.3
Apr	8.0	75	20.0	50	0.507	14.1	19.5
May	11.3	73	20.0	50	0.320	14.1	19.6
Jun	14.2	75	20.0	50	-0.020	14.1	19.7
Jul	15.8	75	20.0	50	-0.408	14.1	19.8
Aug	15.7	77	20.0	50	-0.375	14.1	19.8
Sep	13.5	80	20.0	50	0.090	14.1	19.7
Oct	10.6	83	20.0	50	0.371	14.1	19.6
Nov	6.4	85	20.0	50	0.565	14.1	19.4
Dec	4.6	86	20.0	50	0.616	14.1	19.3

Definitions:

Ext Temp (°C or °F) is the average monthly external dry bulb air temperature.

Ext RH (%) is the average monthly relative humidity of the outside air.

Int Temp (°C or °F) is the average inside temperature over the month.

Int RH is the average inside relative humidity over the month.

Min Tsi (°C or °F) is the minimum internal surface temperature over the month.

Tsi (°C or °F) is the average monthly internal surface temperature.

Min Temp Factor = (Min TSi – Ext Temp) / (TSi – Ext Temp).

Note: Internal and external boundary conditions can be edited by clicking on the **Edit boundary data** Info panel link.

Example Output

An example report generated by the Condensation analysis tool is shown below.

Condensation Analysis Report

Interstitial Condensation

Interstitial condensation occurs at one or more interfaces but all condensate is predicted to evaporate during the summer months.

The risk of degradation of building materials and deterioration of thermal performance as a consequence of the calculated maximum amount of moisture shall be considered according to regulatory requirements and other guidance in product standards.

Condensation Interfaces

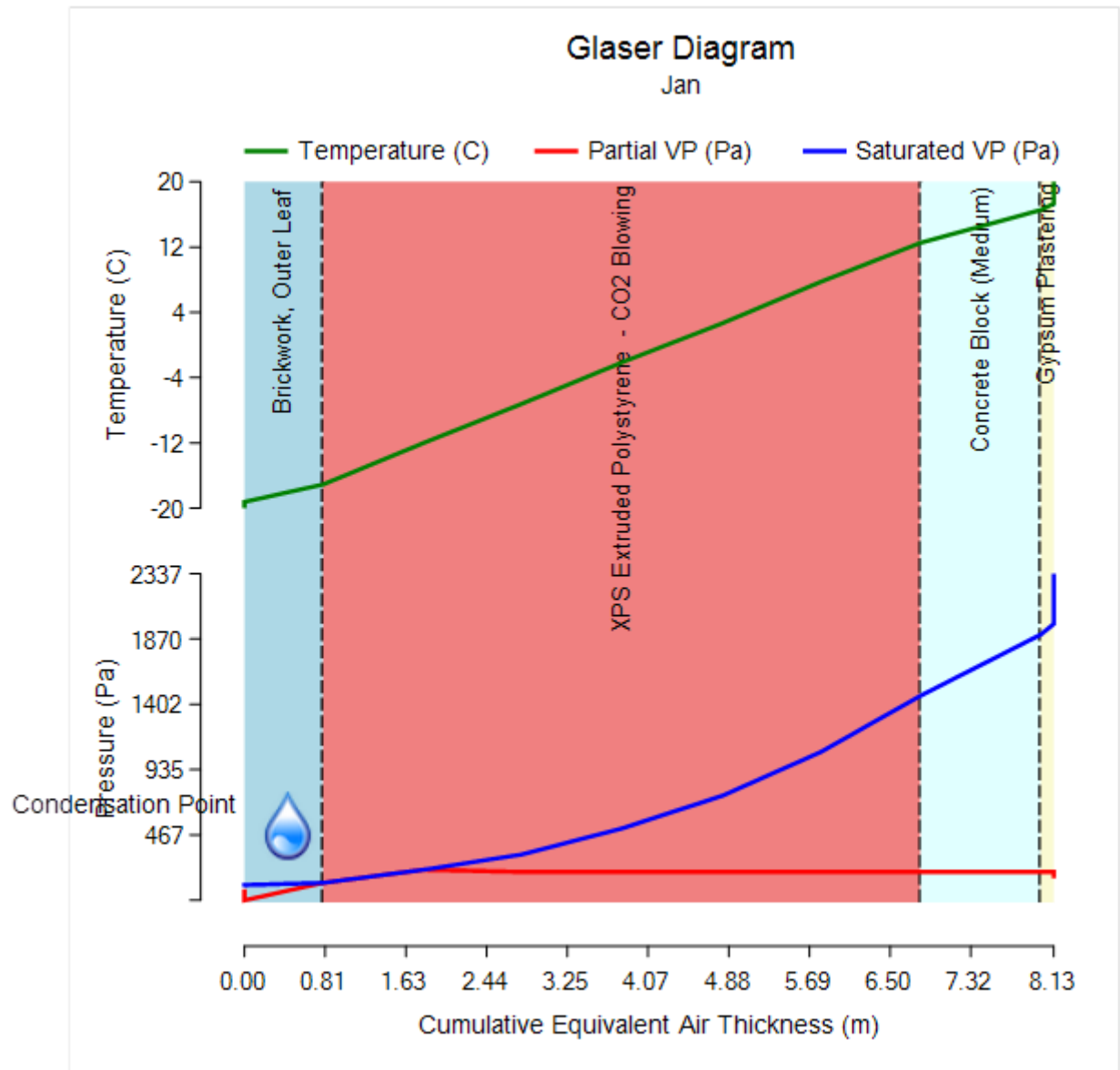
Number of condensation interfaces: 2

Distance of condensation interface 1 from the external surface: 100mm

Distance of condensation interface 2 from the external surface: 108.333333333333mm

Surface Condensation

Thermal quality Good. Mould growth unlikely.



Note: In some cases the constructions layers are displayed in the Glaser diagram as being compressed together. To understand this consider that Glaser diagrams use the concept of "equivalent thickness of air" on the X-axis. This defines the amount of resistance to moisture diffusion relative to the equivalent thickness of air. When layers are compressed together this means that they offer little resistance to moisture diffusion. For example if a layer has an equivalent thickness of 5m that means that it offers the same diffusion resistance as 5m of air. It is usual for materials with high diffusion resistance to be used on the warm side and with lower diffusion resistance on the cold side to minimise the risk of interstitial condensation occurring. It is also important to note that the correct [moisture resistance data](#) has been entered for each of the material layers included in the construction before using the Glaser diagram and report for project work. The use of manufacturers data is recommended, particularly for materials with high vapour resistance.

Boundary Data

The internal and external boundary conditions can be edited by clicking on the [Edit boundary data](#) Info panel link. This opens the **Condensation Analysis Boundary Data** dialog which allows the average monthly internal and external air temperature and relative humidity (RH) to be entered through a grid user interface as shown below.

Month	Int Temp (°C)	Int RH (%)	Ext Temp (°C)	Ext RH (%)
Jan	20	70	2	85
Feb	20	57	3	82
Mar	20	56	5	79
Apr	20	56	8	75
May	20	58	11	73
Jun	20	64	14	75
Jul	20	67	15	75
Aug	20	68	15	77
Sep	20	65	13	80
Oct	20	62	10	83
Nov	20	59	6	85
Dec	20	58	4	86

Buttons: Load Simulation Weather Data, Help, Cancel, OK

The default external conditions come from CIBSE Guide A7, but they should be updated to reflect the actual project local conditions.

Tip: You can load monthly external air temperature and RH values from the current site simulation weather data by clicking on the [Load Simulation Weather Data](#) button on the right of the dialog. Note that this is only enabled when site data has been loaded on the Simulation screen. The easiest way to achieve this if a simulation hasn't been run is to go to the Simulation screen, navigate to site level and press the Update toolbar icon.

The same condensation boundary data is stored for later use with this or other constructions.

Materials

Material data contains data accessed from these tabs:

- [General](#) thermo-physical and "Moisture Transfer" on page 1393 data on the building material.
- [Surface Properties](#) - data on the surface characteristics of the material.
- [Green Roof](#) - if the material is a green roof enter the data on the corresponding here.
- [Embodied Carbon](#) - data on embodied carbon contained the material.
- [Phase Change](#) - dynamic thermo-physical properties for Phase Change Materials (PCM).

Material component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the material icon in the model data,
- by clicking on the Edit button when selecting a material from a list.

Material Thermophysical Properties

General tab on [Materials](#) Dialog.

Materials are used to define the properties of construction layers. There are 2 types of material:

1. **Detailed properties** including the thermophysical properties, surface properties and visual appearance for the material.
2. **Simple resistive material** with no thermal mass. This option will typically be used to model air gaps.

Extremely thin and/or highly conductive layers - technical note on applicability

When a Material is used for the Construction of a building surface, care should be taken to not attempt to model assemblies that were not included in the intended scope of applicability for the underlying heat transfer models. The building surface models are for normal applications to building energy efficiency where the main focus is on assemblies with some thermal resistance. Extremely thin and/or highly conductive material layers should be neglected from the Construction rather than included because they will not contribute to the assembly's overall thermal resistance or heat capacity. For some cases, thin and/or highly conductive materials are a serious problem for the heat transfer modelling and the values for thickness, conductivity, density and specific heat are checked for appropriateness. This check calculates the Material's thermal diffusivity from the inputs for conductivity, density, and specific heat and compares it to a maximum threshold of 1.E-5 (m²/s). If the diffusivity is above this threshold, then the program checks if the layer is sufficiently thick and may issue a warning if it is too thin and highly conductive.

Material Thickness

Force thickness

If the material should only be used in layers of a particular thickness then check this box and enter the material thickness that applies below. When materials having this option set are selected on the [Constructions](#) dialog the [layer thickness](#) will be fixed with the material thickness described below.

This option may be useful for materials such as bricks, tiles, insulation and PCMs which are usually supplied with a particular thickness.

Note: you must check this option for materials to be used to define insulation properties within a [Kiva Foundation](#). In this case the **Default thickness** value below is used to define the insulation thickness.

Default thickness

The fixed layer thickness of the material (in m or in) when selected.

This should be the dimension of the layer in the direction perpendicular to the main path of heat conduction. Modelling layers thinner (less) than 0.003 m is not normally recommended; rather, add those properties to one of the adjacent layers.

Detailed Properties

Thermal bulk properties

Thermal conductivity

This field is used to enter the thermal conductivity of the material layer. Thermal conductivity must be greater than zero.

This field is used to enter the thermal conductivity of the material layer (in W/m-K or Btu-in/h-ft²-F). Thermal conductivity must be greater than zero. Modelling layers with conductivity higher than 5.0 W/(m-K) is not normally recommended.

Specific heat

This field represents the specific heat of the material layer (in J/kg-K or Btu/lb-F).

Note: EnergyPlus uses scientific SI units but data reported in textbooks and references will often use kJ/kg-K or J/g-K instead. Take care!

Only values of specific heat of 100 or larger are allowed. Typical ranges are from 800 to 2000 J/kg-K.

Density

This field is used to enter the density of the material layer in (kg/m³ or lb/ft³). Density must be a positive quantity.

Note: Some textbooks and references may use units of g/m³. Take care!

Simple Resistance

Thermal resistance

This field is used to enter the thermal resistance (R-value) of the material layer (in m²-K/W or ft²-F-hr/Btu).

Note: Materials defined using an R-value do not contribute to construction (and hence building) thermal mass in the calculations and so should be used with care, especially if you plan to use the model for dynamic thermal simulations. You are especially advised not to use R-value materials on the innermost layer of constructions and you should not generally use them for representing materials that have mass. They are best used for modelling air gaps and insulation layers.

Vapour Resistance

You can enter vapour diffusion properties here if the material is to be used within a construction which is have a [Condensation analysis](#) carried out. This data is not used for general simulations. To include moisture diffusion in simulations you should use the [Moisture transfer](#) settings on each of the materials used in each of the constructions in the project.

Note: Vapour resistance/moisture diffusion data provided in the literature tends to be very approximate, much more so than for thermal properties. This is due to large variations in these properties that have been measured in lab tests.

Vapour resistance definition

There are 3 ways to define the vapour diffusion properties of a material:

- **1-Factor**, in which case the [Vapour factor](#) is entered below.
- **2-Resistivity**, in which case the [Vapour resistivity](#) is entered below.
- **3-Non-permeable**, for cases where the material is not permeable to moisture vapour and no further data is required. Use this option for vapour barriers and plastic materials which do not transmit water vapour.

Vapour factor

Enter the vapour factor for the material. There are no units.

Vapour resistivity

Enter the vapour resistivity, a property of the bulk material (in MNs/g.m).

Surface Properties

Surface properties tab on [Materials Dialog](#)

Surface properties are only used in simulations when the material is located on an inner or outer surface of the construction.

Note: The absorptance values in this object impart surface properties to the construction and should be applied to the thermally significant inner and outer layers in the overall assembly. Attempting to trick the program by modelling thin “paint” layers to apply surface properties is not a good idea as the thermal models were not intended to support such strategies.

Thermal absorptance (emissivity)

The thermal absorptance represents the fraction of incident long wavelength radiation that is absorbed by the material. This parameter is used when calculating the long wavelength radiant exchange between various surfaces and affects the surface heat balances (both inside and outside as appropriate). Values for this field must be between 0.0 and 1.0 (with 1.0 representing “black body” conditions).

Solar absorptance

The solar absorptance field in the Material input syntax represents the fraction of incident solar radiation that is absorbed by the material. Solar radiation includes the visible spectrum as well as infrared and ultraviolet wavelengths. This parameter is used when calculating the amount of incident solar radiation absorbed by various surfaces and affects the surface heat balances (both inside and outside as appropriate). Values for this field must be between 0.0 and 1.0.

Visible absorptance

The visible absorptance field in the Material input syntax represents the fraction of incident visible wavelength radiation that is absorbed by the material. Visible wavelength radiation is slightly different than solar radiation in that the visible band of wavelengths is much more narrow while solar radiation includes the visible spectrum as well as infrared and ultraviolet wavelengths.

This parameter is used when calculating the amount of incident visible radiation absorbed by various surfaces and affects the surface heat balances (both inside and outside as appropriate) as well as the daylighting calculations.

It is very important to check the surface reflectance values used in EnergyPlus daylighting calculations because any library values used will be generic and the actual surface reflectance will typically depend more on any surface finish such as paint than on the material itself.

Tip: You can calculate Visible absorptance from a surface Light Reflectance Value (LRV) by dividing the LRV by 100 and subtracting from 1.

Values for this field must be between 0.0 and 1.0.

Roughness

This field is a character string that defines the relative roughness of a particular material layer. This parameter only influences the the calculated exterior convection coefficient. A special keyword is expected in this field with the options being “VeryRough”, “MediumRough”, “Rough”, “Smooth”, “MediumSmooth”, and “VerySmooth” in order of roughest to smoothest options.

Colour

The colour data is used for display purposes when the texture is not available for any reason. It is not used in any of the calculations.

Texture

The texture is used to represent this material when it is specified as either the inside or outside material of a construction which is selected in the building model. It is also used to represent the material in diagrams of any construction using this material.

Radiance Daylighting

The following data items are used to define the surface properties of the material when it is used in a Radiance simulation. They allow materials to be defined as transparent or semi-transparent.

Note: settings in this section do not affect EnergyPlus or any other simulations.

Material class

The material class defines how the material is defined in Radiance daylighting simulations. The options are:

- **1-Plastic**, the default option uses the Radiance "Plastic" material option
- **2-Metal**, uses the Radiance "Metal" material option. **2-Metal** is similar to **1-Plastic**, but specular highlights are modified by the material colour.
- **3-Translucent**, uses the Radiance "Trans" material option which is a translucent material similar to plastic. The transmissivity is the fraction of penetrating light that travels all the way through the material. The transmitted specular component is the fraction of transmitted light that is not diffusely scattered. Transmitted and diffusely reflected light is modified by the material colour. Translucent objects are infinitely thin.
- **4-Translucent (simple)**, Is similar to **3-Translucent**, and uses the Radiance "Trans" material option. When this option is selected the only data that needs to be entered is the Transmissivity. Other settings take on default values.

To account for non-zero component block transmissivity, the material class should be set to **3-Translucent** or **4-Translucent (simple)**.

Absorptance

The fraction of light that is absorbed within the material. This option is only available when the material class is set to **3-Translucent**.

Reflectance + Transmissivity + Absorptance = 1

Reflectance

Reflectance is the proportion of incident light that is reflected away from the surface.

Specularity

Enter a specularity value for the material surface to define the way that light reflects from it. The value entered must be between 0 and 1. A value of 0 provides purely diffuse reflection, while a value of 1 gives no diffusion, representing mirror-like behaviour. Specularity fractions greater than 0.1 are generally not very realistic for typical building materials. The specularity of metals is usually 0.9 or greater.

Note: the [Visible absorptance](#) must also be adjusted to complete the surface property behaviour for Radiance calculations.

Specular reflection is the mirror-like reflection of light from a surface where each incident ray is reflected, with the reflected ray having the same angle to the surface normal as the incident ray. Some devices used to enhance daylighting performance such as light shelves and blinds with directional reflection properties require a non-zero value of specularity.

Roughness

Roughness is specified as the rmf slope of surface facets. A value of 0 corresponds to a perfectly smooth surface, and a value of one would be a very rough surface. Roughness values greater than 0.2 are not very realistic.

Transmissivity

The transmissivity is the fraction of penetrating light that travels all the way through the material at normal incidence, i.e. the fraction that is not absorbed in one traversal of the material.

A transmissivity value of 0 means fully opaque and 1 means fully clear.

Transmittance, the value usually measured, is the total light transmitted through the pane, including multiple reflections. To compute Transmissivity (tn) from Transmittance (Tn) use:

$$tn = (\text{sqrt}(.8402528435 + .0072522239 * Tn * Tn) - .9166530661) / .0036261119 / Tn$$

This option is only visible when one of the **3-Translucent** or **4-Translucent (simple)** Material class options are selected.

Translucent objects are infinitely thin.

Fraction diffused

The fraction of light that is diffusely scattered as it is transmitted. This option is only visible when the **3-Translucent** Material class option is selected.

Green Roof

Green roof tab [Materials](#) Dialog

To create a green roof in DesignBuilder, follow these steps:

1. Create a green roof material using data on the Green roof tab of the material dialog.
2. Create a roof construction using the green roof material as the outermost layer.
3. Specify the green roof construction on roof surfaces as appropriate.

The green roof can receive water during the simulation from an [irrigation system](#) and/or from [site precipitation](#) (defined separately from the hourly weather data). The initial properties of the soil layer (day 0 of the simulation) are defined on the Green roof tab of the Materials dialog.

EnergyPlus documentation states: *"This model was developed for low-sloped exterior surfaces (roofs). It is not recommended for high-sloped exterior surfaces (e.g., walls)."* However it may be possible to approximately model green walls, though we are not yet aware of any validation of the underlying EnergyPlus model and certainly the irrigation would have to be treated differently to roofs as walls will not naturally trap much precipitation.

Note: Green roof simulations must be run using the **1-CTF** [Solution algorithm](#). If any other option is selected, you will receive severe errors.

Tip: EnergyPlus documentation suggests that green roofs may require "more timesteps" than normal simulations.

Note: Specifying a green roof material as the material for a [component block](#) will not work - these only use materials for their reflective properties.

Background

From **A green roof model for building energy simulation programs, D.J. Sailor, Portland State University:**

A green roof (aka EcoRoof or vegetated roof) is a roof that contains a soil (growing media) and vegetation layer as its outermost surface. The construction between the growing media and the roof structure varies, but typically includes a drainage layer, a root barrier, and a waterproof membrane. Green roof growing media depth is typically between 10 and 30 cm, although some implementations (referred to as intensive green roofs) have deeper soils capable of sustaining large shrubs and even trees. The thinner implementations (typically < 20 cm), known as extensive green roofs, are more common, but can only sustain smaller plants and ground cover.

While green roofs have been in use for centuries, there recently has been a surge in interest in installing green roofs in both retrofit and new construction applications. Potential benefits of green roofs include aesthetic appeal, habitat, storm water reduction, and energy savings."

There is widespread recognition and a growing literature of measured data that suggest green roofs can reduce building energy consumption. The EnergyPlus Green Roof capability can assist developers and architects in assessing the likely magnitude of energy savings associated with various implementation options (e.g., soil type/depth, irrigation options, plant type). It provides a quantitative and physically-based building energy simulation tool that represents the effects of green roof constructions and facilitates more rapid spread of green roof technologies and make it possible to account for green roof benefits in state energy codes and related energy efficiency standards such as LEED.

The green roof model accounts for:

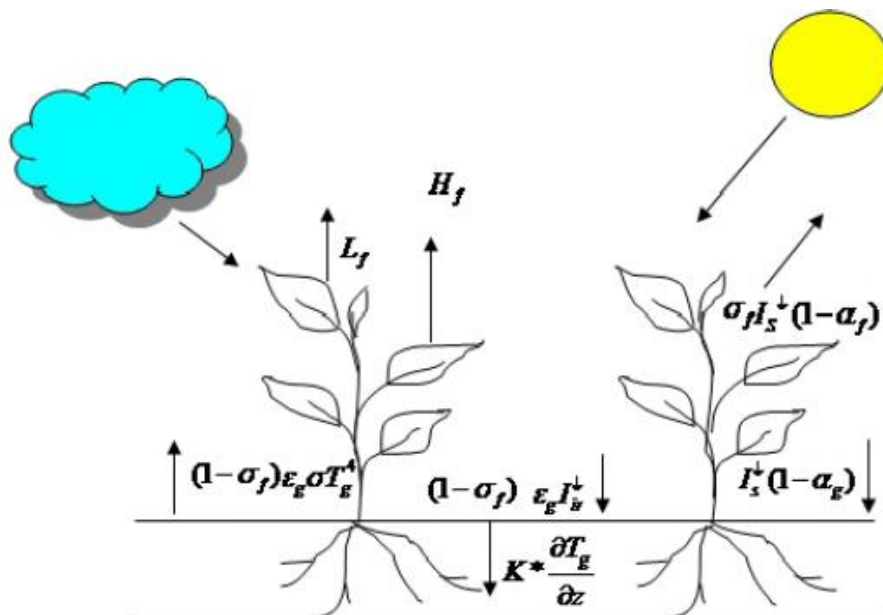
- Long wave and short wave radiative exchange within the plant canopy,
- Plant canopy effects on convective heat transfer,
- Evapotranspiration from the soil and plants, and
- Heat conduction (and storage) in the soil layer

As implemented in EnergyPlus the green roof module allows the user to specify "ecorooft" as the outer layer of a rooftop construction. The user can then specify various aspects of the green roof construction including growing media depth, thermal properties, plant canopy density, plant height, stomatal conductance (ability to transpire moisture), and soil moisture conditions (including irrigation). The model formulation includes the following:

- Simplified moisture balance that allows precipitation, irrigation, and moisture transport between two soil layers (top and root zone).
- Soil and plant canopy energy balance based on the Army Corps of Engineers' FASST vegetation models (Frankenstein and Koenig), drawing heavily from BATS (Dickenson et al.) and SiB (Sellers et al.).
- Soil surface (T_g) and foliage (T_f) temperature equations are solved simultaneously each time step, inverting the CTF to extract heat flux information for the energy balance calculation. The detailed energy balance analysis and resulting equations, being rather complicated, are summarized here. The interested reader is referred to the FASST documentation cited herein for the complete development. The end result is a set of two simultaneous equations for temperature—one for the soil surface and the other for the foliage.

Green Roof Model Description

As with a traditional roof, the energy balance of an green roof is dominated by radiative forcing from the sun. This solar radiation is balanced by sensible (convection) and latent (evaporative) heat flux from soil and plant surfaces combined with conduction of heat into the soil substrate. This energy balance is illustrated in the diagram below. The variables introduced in this figure are defined in the [EnergyPlus Engineering Document](#).



The energy balance for a green roof.

The energy budget analysis follows the Fast All Season Soil Strength (FASST) model developed by Frankenstein and Koenig for the US Army Corps of Engineers. FASST was developed, in part, to determine the ability of soils to support manned and unmanned vehicles and personnel movement. In order to accomplish this, however, FASST tracks the energy and moisture balance (including ice and snow) within a vegetated soil. It is a one-dimensional model that draws heavily from other plant canopy models including BATS (Dickinson et al.) and SiB (Sellers et al.). FASST is implemented in EnergyPlus with only a few modifications to adapt it for use with a relatively thin soil layer.

Green Roof Data

Moisture diffusion calculation method

Two moisture diffusion models can be selected:

- **1-Simple** is the original EcoRoof model - based on a constant diffusion of moisture through the soil. This model starts with the soil in two layers. Every time the soil properties update is called, it will look at the two soils moisture layers and asses which layer has more moisture in it. It then takes moisture from the higher moisture layer and redistributes it to the lower moisture layer at a constant rate.
- **2-Advanced** is the later EcoRoof model. If you use it, you will need to increase your number of timesteps in hour for the simulation with a recommended value of 20. This moisture transport model is based on a project which looked at the way moisture transports through soil. It uses a finite difference method to divide the soil into layers (nodes). It redistributes the soil moisture according the model described in **Marcel G Schaap and Martinus Th. van Genuchten, 2006, 'A modified Maulem-van Genuchten Formulation for Improved Description of the Hydraulic Conductivity Near Saturation', Vadose Zone Journal 5 (1), p27-34.**

Height of Plants

The average height of plants in the green roof.

Leaf Area Index (LAI)

This is the projected leaf area per unit area of soil surface. It is a dimensionless number between 0.001 and 5.0. The tables below gives some typical values for LAI.

The table below is reproduced from [Global Leaf Area Index Data from Field Measurements](#), 1932-2000

Table: Statistical distribution of leaf area index by biome, for the data set “Global Leaf Area Index Data from Field Measurements, 1932-2000”, and after removal of outliers following Inter-Quartile Range (IQR) statistical analysis. Of the original total of 1008 records, 77 were excluded from this analysis (e.g., because biome was not available). See the accompanying Technical Memorandum (Scurlock et al., 2001) for biome acronyms.

Biome	Original data					Number of outliers removed	Data after IQR analysis			
	Number of observations	Mean	Standard deviation	Min	Max		Mean	Standard deviation	Min	Max
All	931	5.23	4.08	0.002	47.0	53	4.51	2.52	0.002	12.1
Forest / BoDBL	58	2.64	1.03	0.28	6.0	5	2.58	0.73	0.6	4.0
Forest / BoENL	94	3.50	3.34	0.48	21.6	8	2.65	1.31	0.48	6.21
Crops	88	4.22	3.29	0.2	20.3	5	3.62	2.06	0.2	8.7
Desert	6	1.31	0.85	0.59	2.84	0	1.31	0.85	0.59	2.84
Grassland	28	2.50	2.98	0.29	15.4	3	1.71	1.19	0.29	5.0
Plantation	77	8.72	4.32	1.55	18.0	0	8.72	4.32	1.55	18.0
Shrub	5	2.08	1.58	0.4	4.5	0	2.08	1.58	0.4	4.5
Forest / BoTeDNL	17	4.63	2.37	0.5	8.5	0	4.63	2.37	0.5	8.5
Forest / TeDBL	187	5.12	1.84	0.4	16.0	3	5.06	1.60	1.1	8.8
Forest / TeEBL	58	5.82	2.57	0.8	12.5	1	5.70	2.43	0.8	11.6
Forest / TeENL	215	6.70	5.95	0.002	47.0	16	5.47	3.37	0.002	15.0
Forest / TrDBL	18	3.92	2.53	0.6	8.9	0	3.92	2.53	0.6	8.9
Forest / TrEBL	61	4.90	1.95	1.48	12.3	1	4.78	1.70	1.48	8.0
Tundra	13	2.69	2.39	0.18	7.2	2	1.88	1.47	0.18	5.3
Wetlands	6	6.34	2.29	2.50	8.4	0	6.34	2.29	2.5	8.4

The table below is reproduced from the PhD Thesis of Chen Yu entitled [The intervention of plants in the conflicts between buildings and climate - A case study in Singapore](#)

Plant description

Mean Leaf Area Index (LAI) Picture

"White flowers, spider lily"

3.07



"Pink flowers"

4.95



"Yellow green leaves"

3.75



Plant description

Mean Leaf Area Index (LAI) Picture

"Dark green long blades of leaves/grass"

5.82



"Pinkish red flowers"

2.44



"Fern-like"

6.59



Plant description

Mean Leaf Area Index (LAI) Picture

"Palm tree-like"

4.41



"White flowers with yellow center"

3.21



"Small yellow green leaves"

4.08



Plant description

Mean Leaf Area Index (LAI) Picture

"Long big leaves"

5.28



"Orange stems and leaves for those which are taller"

2.15



"No special features"

3.32



Plant description

Mean Leaf Area Index (LAI) Picture

"Light green edges with dark green
center leave blades"

5.83



"Red yellow tulip like flowers"

3.04



"Large red leaves"

2.33



Plant description

Mean Leaf Area Index (LAI) Picture

"Dark green leaf blades"

~0



Tree

1.69



Palm tree

2.37



Leaf Reflectivity

The fraction of incident solar radiation that is reflected by the individual leaf surfaces. Solar radiation includes the visible spectrum as well as infrared and ultraviolet wavelengths. Values for this field must be between 0.1 and 0.4.

Leaf Emissivity

This field is the ratio of thermal radiation emitted from leaf surfaces to that emitted by an ideal black body at the same temperature. This parameter is used when calculating the long wavelength radiant exchange at the leaf surfaces. Values for this field must be between 0.8 and 1.0 (with 1.0 representing “black body” conditions).

Minimum Stomatal Resistance

This field represents the resistance of the plants to moisture transport. It has units of s/m. Plants with low values of stomatal resistance will result in higher evapotranspiration rates than plants with high resistance. Values for this field must be in the range of 50.0 to 300.0.

Max volumetric moisture content of the soil layer (saturation)

Maximum volumetric moisture content of the soil depends on the properties of the soil and in particular the porosity.

Min (residual) volumetric moisture content of the soil layer

The minimum possible volumetric moisture content of the soil layer.

Initial volumetric moisture content of the soil layer

The volumetric moisture content of the soil layer at the start of the simulation. The moisture content will be updated during the course of the simulation based on surface evaporation, irrigation and precipitation.

Embodied Carbon

Embodied Carbon tab on [Materials](#) Dialog.

You can enter data on the embodied carbon that is associated with the material. This definition for embodied energy is provided on the Bath ICE website:

Embodied energy may be taken as the total primary energy consumed during resource extraction, transportation, manufacturing and fabrication of a product... to separate it from operational impacts.

Likewise **embodied carbon** is the carbon released in the above processes. It has become common practice to specify the embodied carbon as ‘Cradle to Gate’, which includes all carbon until the product leaves the factory gate. The final boundary condition is ‘Cradle to Site’, which includes all of the carbon emitted until the product has reached the point of use (i.e. the building site).

Embodied carbon available

If embodied carbon data is available for the material then this check box should be ticked.

Embodied carbon

Enter the amount of embodied carbon associated with the material. When using SI units, the units are kgCO₂/kg.

Conversion factor name

Enter the name of the conversion factor as defined in the source documentation. This can be used to help you to trace the conversion factor back within the source database.

Source

The source of the data in text form. Much of the data provided by DesignBuilder is derived from the Bath ICE database.

Assumption / factor boundary

Typically embodied energy is defined as the carbon emitted into the atmosphere in order to mine, process and produce the material. Some common definitions are:

- **Cradle to Gate** - the carbon emitted to bring the material to the factory gate (but not to transport it to the building site)
- **Cradle to Site** - as above but additionally includes consideration of transport from factory to the building site.

Equivalent carbon available

If data on equivalent carbon is available then check this box.

Equivalent carbon

Equivalent carbon is similar to **Embodied carbon** (above) but also includes the effects of other greenhouse gases so as to provide an equivalent amount of CO₂ that would cause the same amount of global warming as the actual greenhouse gases (which may include sulphur dioxide, methane etc) emitted by the processes involved in production of the material.

Embodied Carbon Report

Material, construction and glazing inventories and embodied carbon reports are included on the [Cost and Carbon](#) screen and in the [CSV report file](#).

Phase Change Materials

Phase Change tab on [Materials](#) Dialog.

Phase Change Materials (PCMs) are often impregnated into gypsum board, plaster, concrete or other materials in order to increase the thermal storage capacity of ceilings, walls and floors. Some PCM manufactures also provide mats of pure PCM "pockets" which can be included as layers within constructions.



Example of PCM Mat

By completing a transition between solid and liquid phases PCMs can absorb large quantities of latent heat. The high latent heat storage capacity of these materials works to effectively increase thermal mass which tends to moderate interior temperatures and improve comfort conditions. PCMs are frequently used to reduce the need for mechanical cooling, peak load shifting and improving solar energy utilisation.

DesignBuilder allows you to enter manufacturers phase change properties in the form of a temperature - enthalpy curve on the Phase change tab of the [Materials](#) dialog. A set example PCM datasets is provided with the materials database that comes with DesignBuilder.

DesignBuilder supports both types of PCM modelling methods provided by EnergyPlus:

- "Phase Change Material - Basic" on page 1390
- "Phase Change Material - Hysteresis" on page 1392

Solution Algorithm

Important note: The [Finite Difference solution algorithm](#) must be used to include the effect of material phase change properties in simulations. **If you use the CTF algorithm the material will behave as if its PCM option were not selected.** Note also that the CTF solution algorithm is the only option available for Heating and Cooling design so the phase change effect of the material will not be seen in these calculations. An error message to this effect will be generated for PCM simulations where the CTF solution method is selected.

Tip: With [Learning mode](#) active, the Info panel for the Phase change tab includes a **Set PCM Simulation options** link which allows a set of default simulation properties recommended for PCM analyses to be quickly loaded.

Recommended simulation parameters for PCM

[Timestep](#)

[Solution algorithm](#)

[Difference scheme](#)

[Space discretisation constant](#)

[Relaxation factor](#)

[Inside face surface temperature convergence criteria](#)

12 (higher values should increase accuracy at the expense of longer simulation times)

2-Finite difference

1-Fully implicit first order

3

1

0.01 (lower values should increase accuracy at the expense of longer simulation times)

Phase Change Material - Basic

Phase Change Properties

Phase change material

When the Phase change material check box is selected, this material is simulated as having the following temperature dependent material properties.

Phase change method

Choose from two methods to model PCMs:

- **1-Basic** the method described in this section where the same melting/freezing temperature/enthalpy curve is used
- **2-Hysteresis** which allows the melting/freezing process to follow different temperature/enthalpy curves, representing an effect that is commonly seen in actual building envelope phase change material applications.

Temperature coefficient for thermal conductivity

This field is used to enter the temperature dependent coefficient for thermal conductivity of the material. Units for this parameter are W/m-K². This is the thermal conductivity change per unit temperature excursion from 20°C. The conductivity value at 20°C is the one specified with the basic material properties of the regular material specified in the name field. The thermal conductivity is obtained from:

$$k = k_0 + k_1(T_i - 20)$$

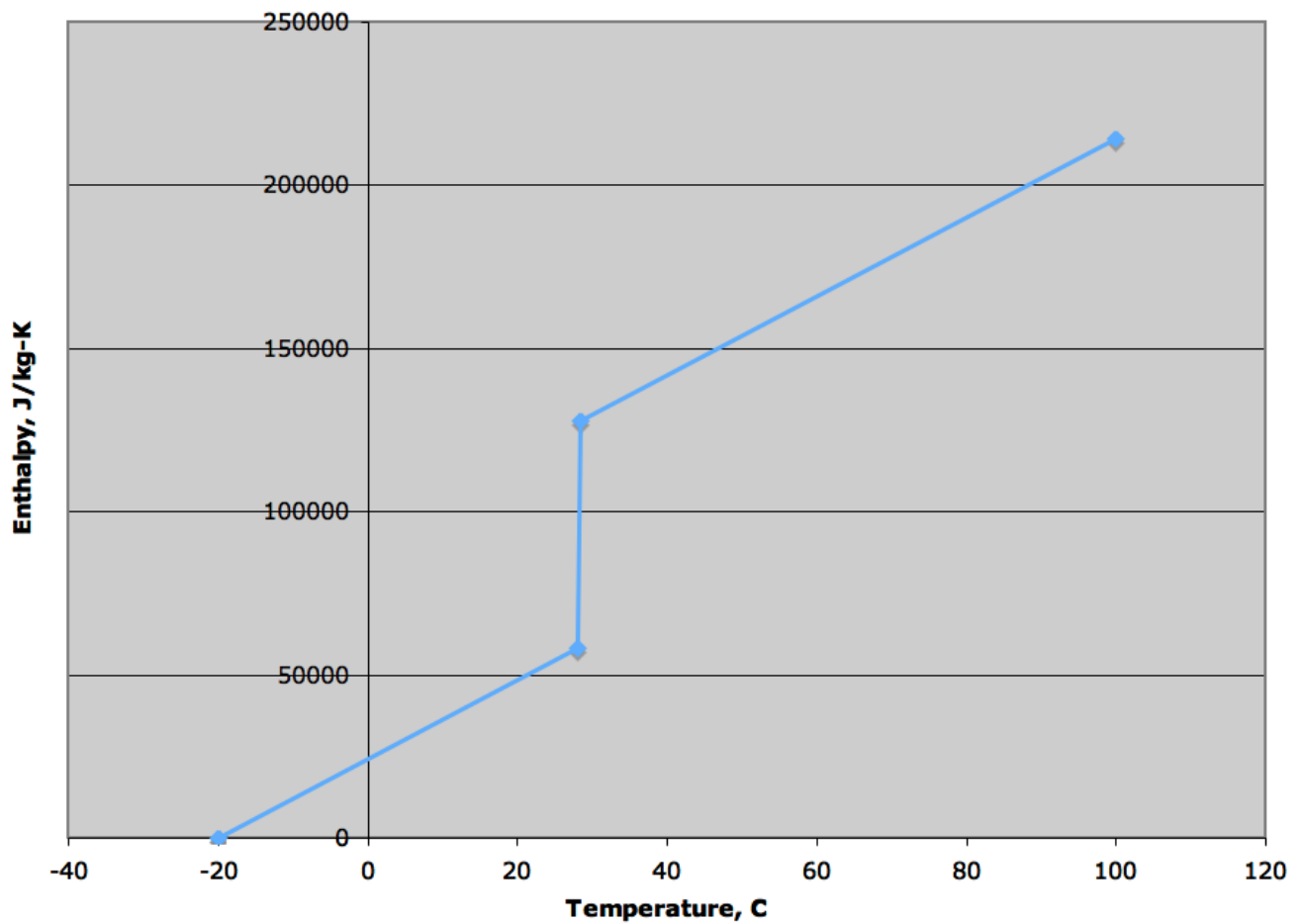
where:

k_0 is the 20°C value of [Thermal conductivity](#) entered on the Material properties tab of the Materials dialog.
 k_1 is the change in conductivity per degree temperature difference from 20°C (this field).

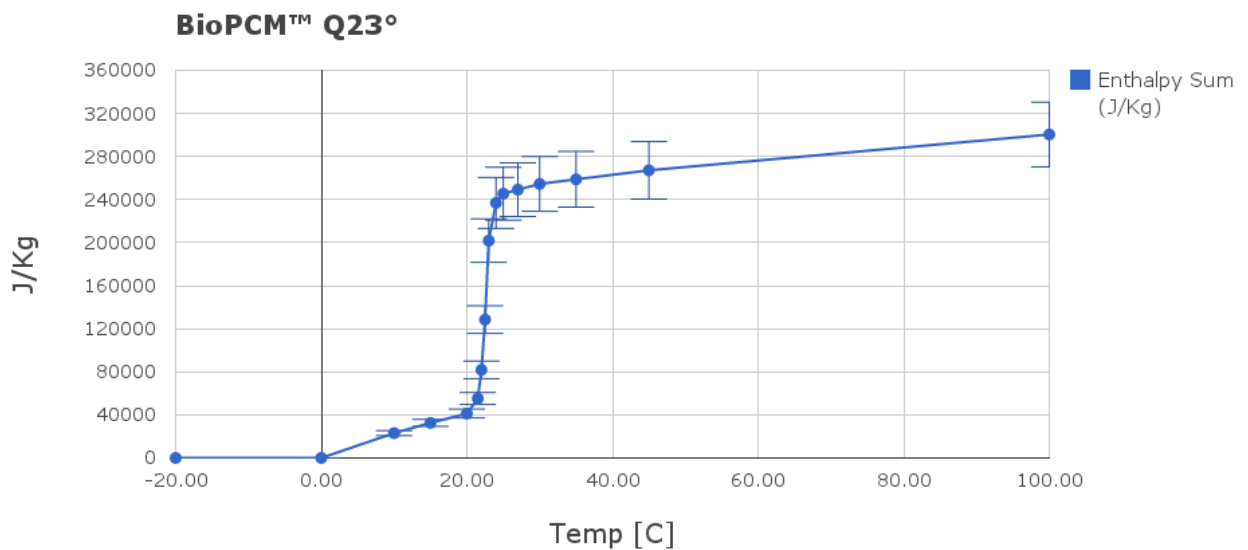
Temperature-Enthalpy Curves

The temperature – enthalpy set of inputs specify a two column tabular temperature-enthalpy function for the basic material. Sixteen pairs can be specified. Specify only the number of pairs necessary. The tabular function must cover the entire temperature range that will be seen by the material in the simulation. It is suggested that the function start at a low temperature (-20°C is typical), and extend to 100°C. Note that the function has no negative slopes and the lowest slope that will occur is the base material specific heat. Enthalpy contributions of the phase change are always added to the enthalpy that would result from a constant specific heat base material. Examples of simple generic enthalpy temperature function and also manufacturers data is shown below.

Temperature Enthalpy Data



Generic Data from EnergyPlus



Example of Manufacturers PCM Data

Number of points on the curve

Select a number from 1 to 16. The number of points selected here will determine the number of points available for data entry below.

Temperature x

This field is used to specify the temperature of the temperature-enthalpy function for the material. Units are °C or °F.

Enthalpy x

This field specifies the enthalpy that corresponds to the previous temperature of the temperature-enthalpy function. Units are J/kg or Btu/lb.

Phase Change Material - Hysteresis

The [Basic](#) phase change option describes a single process curve whereby a material moves from a crystallized to liquid state and back. This option models the same process but also includes a hysteresis effect, allowing the melting and freezing processes to follow different curves, representing an effect that is commonly seen in actual building envelope phase change material applications. This option also allows the characteristic properties of the processes to be entered instead of a detailed temperature/enthalpy curve, making it more amenable for studies in which the user does not have the detailed test data required to generate the temperature/enthalpy curve.

Phase Change Properties

Phase change material

When the Phase change material check box is selected, this material is simulated as having the following temperature dependent material properties.

Phase change method

Choose from two methods to model PCMs:

- [1-Basic](#) where the same temperature/enthalpy curve is used for both melting and freezing.
- [2-Hysteresis](#) the method described in this section where the melting and freezing processes follow different temperature/enthalpy curves, representing an effect that is commonly seen in actual building envelope phase change material applications.

Phase Change Hysteresis

Latent heat during the entire phase change process

This is the total amount of latent heat absorbed or discharged during the transition from solid to liquid or back (in J or Btu/lb). The shapes of the enthalpy curves differ based on direction, but the total amount of energy from one state to the other does not.

Liquid State

Liquid state thermal conductivity

This is the constant thermal conductivity used while the material is fully liquid (in W/m-K or Btu-in/h-ft²-°F).

Liquid state density

This is the constant density while the material is fully liquid (in kg/m³ or lb/ft³).

Liquid state specific heat

This is the constant specific heat while the material is fully liquid (in J/kg-K or Btu/lb-°F).

Melting Curve

High temperature difference of melting curve

This is the width of the enthalpy/specific heat melting curve, on the high side of the peak melting temperature (in Delta °C or Delta °F).

Peak melting temperature

This is the centre (peak) of the melting curve (in °C or °F).

Low temperature difference of melting curve

This is the width of the enthalpy/specific heat melting curve, on the low side of the peak melting temperature (in Delta °C or Delta °F).

Solid State

Solid state thermal conductivity

This is the constant thermal conductivity used while the material is fully solid (in W/m-K or Btu-in/h-ft2-°F).

Solid state density

This is the constant density while the material is fully solid (in kg/m3 or lb/ft3).

Solid state specific heat

This is the constant specific heat while the material is fully crystallized (in J/kg-K or Btu/lb-°F).

Freezing Curve

High temperature difference of freezing curve

This is the width of the enthalpy/specific heat freezing curve, on the high side of the peak freezing temperature (Delta °C or Delta °F).

Peak freezing temperature

This is the center (peak) of the freezing curve (in °C or °F). This will be higher than the peak melting temperature based on empirical data.

Low temperature difference of freezing curve

This is the width of the enthalpy/specific heat freezing curve, on the low side of the peak freezing temperature (Delta °C or Delta °F).

Moisture Transfer

General tab on [Materials](#) Dialog.

Data for the transfer of moisture that occurs during simulations is associated with a material component by checking the **Include moisture transfer settings** checkbox under the **Moisture Transfer** header on the [Materials](#) dialog and selecting a [Moisture transfer](#) data component.

Note: This data is not used for Condensation analysis. For that use the [Vapour resistance](#) data on the Materials dialog.

Material Cost Data

Cost tab on Materials Dialog

Cost type

You can define the cost of each material used in the building fabric in one of 3 ways through the cost type setting:

- **1-Cost per surface area** where the material cost is defined per area of the construction surface. This is the most commonly used way to define costs for most materials of low to medium thickness.
- **2-Cost per mass** where the cost of the material is defined based on mass (cost per kg or lb)
- **3-Cost per volume** where the cost of the material is defined based on volume (cost per m3 or ft3). This option is often used to define the cost for poured construction layers such as cast concrete.

Cost per surface area

When the cost type is **1-Cost per surface area** enter the cost of the material per area when applied to the building surface.

Cost per mass

When the cost type is **1-Cost per mass** enter the cost of the material per kg or lb when applied to the building surface.

Cost per volume

When the cost type is **1-Cost per volume** enter the cost of the material per m3 or ft3 when applied to the building surface.

Glazing Components

Glazing systems are simulated in EnergyPlus according to the ISO 15099 2003 standard.

There are 2 ways to enter Glazing component data depending on the **Definition method** setting:

1. **Material Layers** - where the properties of the glazing component is defined in detail through selection of a number of [glazing layers](#) ([Panels](#)) separated from each other by [window gas](#) layers. In this case there are two tabs on the Glazing data dialog:
 - [Layers](#) where the glazing panes and window gas layers are entered
 - [Calculated](#) where derived calculated data on the glazing system is displayed
 - **Cost** where the cost of the glazing system is defined per area of glazing.
2. **Simple** - the glazing properties for the overall glazing system are defined using summary U-value, Solar Heat Gain Coefficient (SHGC) and Visible transmittance (VT) data. This approach is recommended when only manufacturers summary glazing performance data is available or where only [whole assembly](#) glazing data is available. If more detailed data on the constituent panes is available then the more accurate Layers method should be used.

Glazing component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),

- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the glazing icon on the Openings tab in the model data,
- by clicking on the Edit button when selecting glazing components from a list.

See also the [Creating new glazing types](#) tutorials

Glazing Layer Data

Layer tab on [Glazing](#) Dialog.

General

Name

Enter a unique name for this glazing component.

Description

An optional description of the glazing component.

Source

A text field where you can enter the source of the glazing data. For example if it came from a national energy code or certification scheme then it might be useful to say something about that here.

Category

Choose a category for the glazing component.

Note: If the glazing includes integrated photovoltaic generation or thermochromic shading then it is important to select the appropriate category to make further settings available.

Region

If the glazing component is only relevant in a particular region then select that region here.

Colour

The colour selected here is used to shade any windows that use this glazing on the Constructions and Glazing [Model Data Visualisation](#) screen.

Definition Method

Definition method

Glazing performance can be defined using 2 methods:

- **1-Material layers** where glazing pane and window gas layers are selected.
- **2-Simple** where the performance of the overall glazing component is defined using 3 main global parameters: U-value, Solar Heat Gain Coefficient (SHGC) and Visible transmittance (VT).

The rest of this page describes the definition of glazing performance by the **1-Material layers** method.

Layers

Number layers

The DesignBuilder glazing data consists of layers of panes interspersed with 1 or more layers of window gas. Set the number of glazing layers first (e.g. 2 = double glazing, 3 = triple glazing etc), then select the [Pane](#) and [Window gas](#) material for each layer.

When the Thermochromic category has been set, each pane layer can be defined as either a Pane group or as a single Pane, depending on the Pane definition selection.

Pane definition

When using the Thermochromic category, you can choose to define the properties of each glazing layer either using a pane of constant properties or a pane group which allows temperature-varying properties to be defined. Select from:

- **1-Pane** where a single [Pane](#) is to be selected.
- **2-Pane group** where a [Pane group](#) is to be selected. The pane group allows temperature-varying pane performance characteristics to be modelled.

Note: **Low emissivity glazing systems** can be defined by specifying an appropriate 'LoE' Pane type for the Innermost Pane. You can find some typical low emissivity panes under the 'Clear ' Panes category.

When using the **1-Material layers** method, a set of summary derived data for the overall glazing system such as internal and external surface heat transfer coefficients, U-values etc is displayed on the [Calculated](#) tab.

Flip layer

Check this checkbox to reverse the direction of the associated glazing layer so that the inside face becomes the outside face and vice-versa.

Tip: If you aren't sure which side of the pane a **low-e coating** has been added you can find out by checking the Panes data. On the [Infra-red](#) tab of the Panes dialog you can find the Outside and Inside emissivity values for the pane. Uncoated emissivities are typically in the range 0.8 - 0.9, whereas coated emissivities are generally less than 0.1. In the example below you can see that the Outside emissivity is 0.85 while the Inside emissivity is 0.02 which tells us that the low-e coating is on the inside of the pane. In this case, to change the glazing configuration so that the coating is on the outside surface of the pane instead the **Flip layer** checkbox could be checked.

Edit pane - AGC Glass Europe Stopray Vision-40 8 mm <vision40_8.gvb>

Panes

General Thermal Solar Visible **Infra-red** Spectral data

Infra-Red Properties

Infra red transmittance	0.00000
Outside emissivity	0.02000
Inside emissivity	0.84100

Low-e coating on the outer surface

Photovoltaic Panel

When the **Glazing Integrated Photovoltaics** Glazing category is selected, you can define the properties of a BIPV film (which is assumed to be associated with the window outer surface) below the data for the outermost layer.

Tip: See "How to Model Photovoltaic (PV) Systems" on page 523 for a description of the process involved in modelling BIPV systems.

Performance type

Select from:

- **1-Simple**, and
- **2-Equivalent One-Diode**.

Depending on the selection you will be able to select from either a simple or equivalent one-diode definition of the panel.

Tip: BIPV panels are included in the electrical generation side of the model by specifying them on the [Generator list](#) tab of the Electric load centre dialog when one of the d.c. [Buss types](#) is selected.

Performance model

Choose the component to be used to define the performance of the collector. When the **1-Simple** performance type is selected, select from the list of previously defined [Simple Photovoltaic components](#), or when **2-Equivalent One-Diode** is selected then choose from one of the [Equivalent One-Diode components](#).

Heat transfer integration mode

The BIPV film is assumed to be laminated within the window outer pane and so the **3-Integrated Surface Outside Face** Heat transfer integration mode is used behind the scenes. With this integration mode, the temperature resulting from EnergyPlus' surface modelling is used as the PV cell temperature. Also the energy extracted in the form of electricity is removed from surface heat transfer calculations using a sink term.

See the corresponding help for [BIPV in opaque constructions](#) for more information on Heat transfer integration modes.

Modules in parallel

This field is the number of series-wired strings of PV modules that are in parallel to form the PV array. The product of this field and the next field should equal the total number of modules in the array.

The **Modules in parallel** and **Modules in series** data are only used for **2-Equivalent One-Diode** PV arrays.

Modules in series

This field is the number of modules wired in series (on each string) to form the PV array. The product of this field and the previous field should equal the total number of modules in the array.

Radiance Daylighting

If the glazing is to be used to model diffusing glazing in a Radiance daylighting simulation then the relevant parameters can be defined here.

Note: These settings will have no effect on EnergyPlus daylighting simulations.

Diffusing

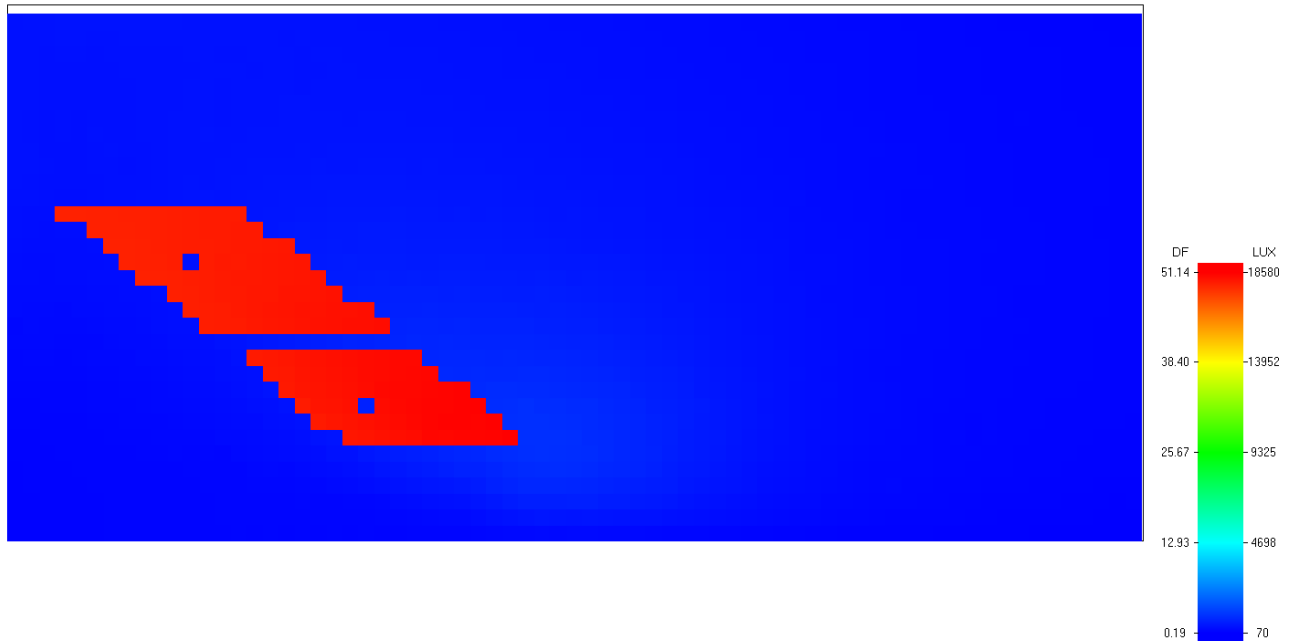
Check this checkbox if the glazing should diffusely scatter transmitted light in Radiance daylighting simulations. Checking this option exposes the following Fraction diffused checkbox.

Fraction diffused

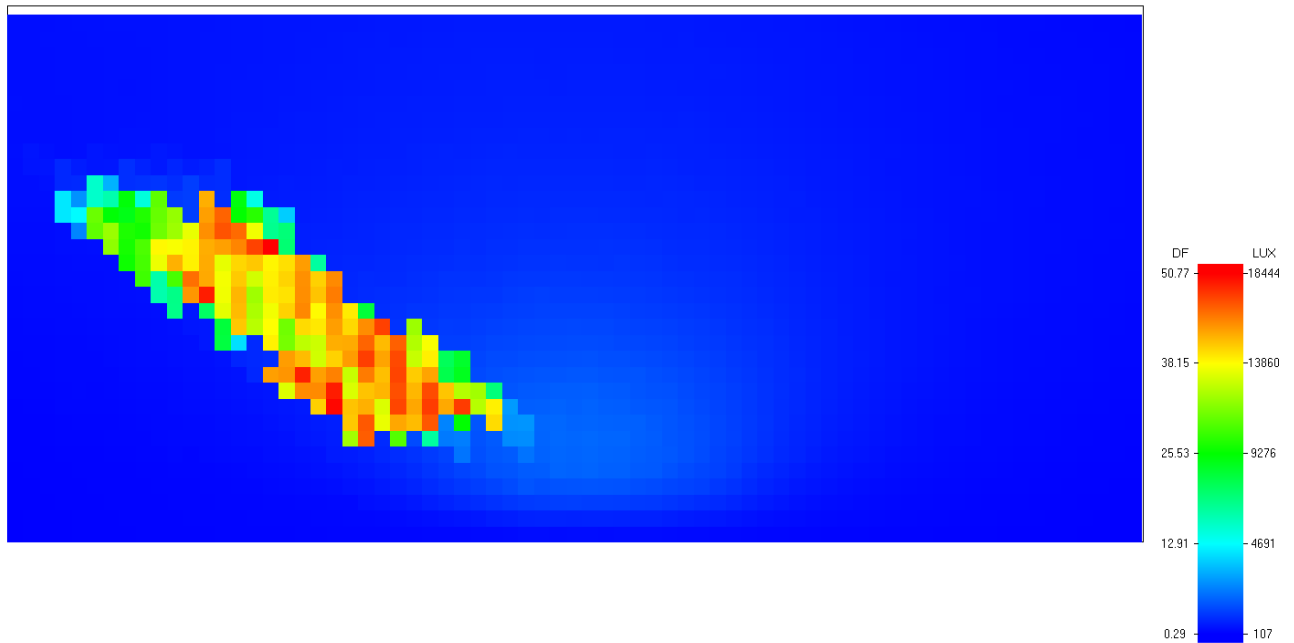
Enter a value between 0 and 1 to define the fraction of the light passing through the glazing that is diffusely scattered.

Sample Diffusing Glazing Outputs

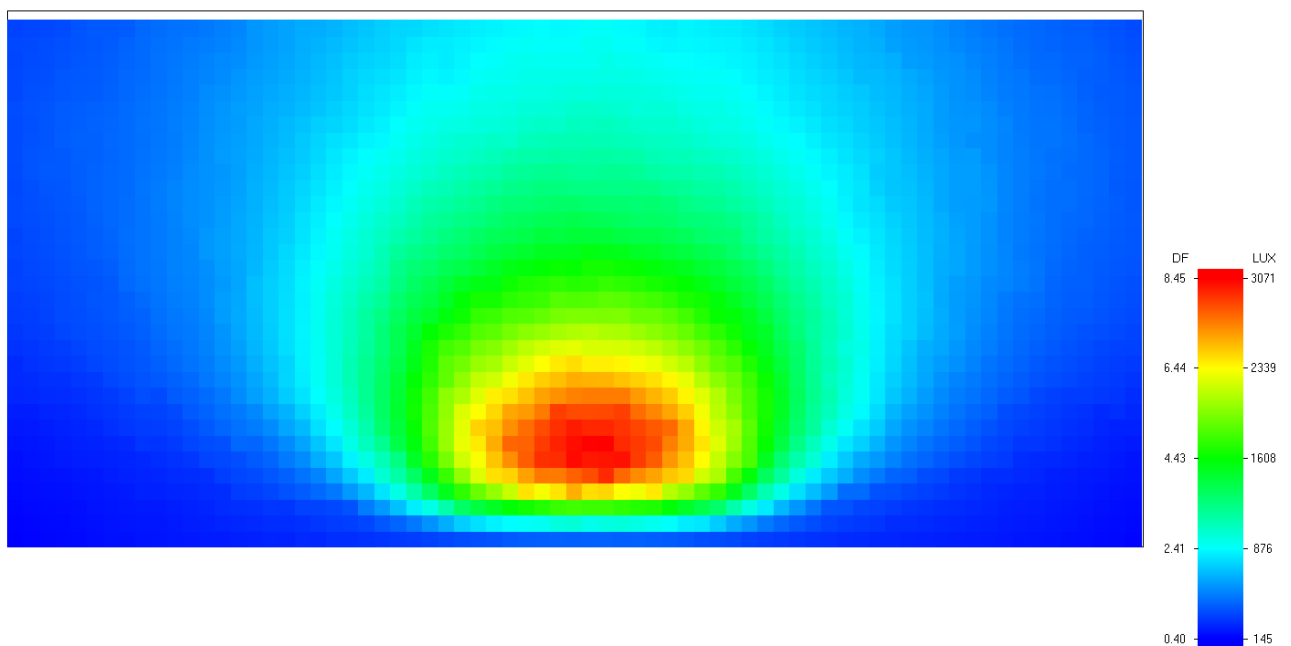
The sample outputs below show illuminance maps for the same simple test cell with varying degrees of diffusing glazing with a sunny clear sky applied.



Specular (Non-Diffusing) Glazing: The Diffusing option is switched off. Note the fairly clean projection of the window shape on the floor. The dividers can just about be seen.



Part Diffusing Glazing. The diffusion option is switched on and the **Fraction diffused** is set to 0.5. The projection of the window onto the working plane can still be made out but it is no longer clear due to the diffusion of light.



Fully Diffusing Glazing. The **Diffusing** option is switched on and the **Fraction diffused** set to 1 to give pure diffuse transmission. The projection of the window onto the working plane can no longer be seen due to the perfect diffusion of light.

Glazing Calculated Data

Calculated tab on [Glazing Dialog](#).

The Calculated data on the Glazing tab is based on the [Layer data](#) on the previous tab as well as some assumptions about the surface heat transfer properties. The calculated data can be used as indicative data for reporting and comparison with other glazing components. It can also be used to help set up glazing model descriptions in cases where global glazing system data is known rather than data on individual panes and window gases.

Calculated Values	
Total solar transmission (SHGC)	0.474
Direct solar transmission	0.358
Visible transmittance	0.661
U-value (ISO 10292/ EN 673) (W/m ² -K)	0.993
U-Value (W/m²-K)	0.982

Two different U-values are provided. The first uses the EN ISO 10292 / EN 673 standard method and the second, displayed in bold on the dialog uses the ASHRAE method.

Note: The U-values displayed on this tab are provided as indicative data for reporting and comparison with other glazing components. **It is important to understand that these values are not used during EnergyPlus simulations**, where the heat transfer through windows is calculated using more sophisticated algorithms based on dynamically varying convective heat transfer coefficients as well as physics-based simulation of heat transfer through the gas and pane layers.

For the ASHRAE calculation, the U-value is determined by EnergyPlus through a heat balance calculation on the glazing layers using the following standard ASHRAE winter conditions to calculate the convective heat transfer coefficients. These boundary conditions are based on the values used in the WINDOW program:

- Inside air temperature = 21°C (69.8°F)
- Outside air temperature = -18°C (-0.4°F)
- Wind speed = 5.5 m/s (12.3 mph)
- Outside convective air film conductance = 26 W/m²-K (4.58 Btu/h-ft²-F)
- Inside convective air film conductance: calculated by EnergyPlus
- No solar radiation

The **Total solar transmission (SHGC)** (Solar Heat Gain Coefficient) is the fraction of incident beam solar radiation that enters the zone. This includes the transmitted solar radiation and the inward flowing heat from the solar radiation that is absorbed by the glazing. SHGC applies only to the centre of the glazed part of a window construction; it does not include the effect of beam solar radiation absorbed by a window frame or divider. SHGC is calculated by EnergyPlus for the following standard summer condition:

- Inside air temperature = 24°C (75.2°F)
- Outside air temperature = 32°C (89.6°F)
- Wind speed = 2.8 m/s (6.2 mph)
- Incident beam solar radiation normal to glazing = 783 W/m² (248 Btu/h-ft²)

The **Visible transmittance (VT)** is the fraction of the radiation at normal incidence that is transmitted through the glazing, averaged over the solar spectrum and weighted by the response of the human eye. It applies only to the centre of the glazed part of the window construction. Visible transmittance is calculated by EnergyPlus.

Tip: You can manually refresh the Calculated data by clicking on the **Update** link in the Info panel on the right of the dialog. This causes EnergyPlus to be run behind the scenes to calculate the ASHRAE U-value, SHGC and VT which are read into DesignBuilder from the .eio file.

Technical Details on ISO 10292 / EN 673 U-value Calculations

The following excerpt is copied from [International Window Standards - Final Report, prepared for Canadian Homeowner Protection Office](#):

Standard EN 673 is referenced by ISO 10077-1 for determining the thermal transmittance of the glazing unit (the centre of glass U-value). EN 673 does not cover edge effects or solar heat gain. Important information in EN 673 includes standard external (h_e) and internal (h_i) surface film coefficients to use for the purpose of comparing glazing U-value: $h_e = 25 \text{ W/m}^2\text{-K}$ and $h_i = 7.7 \text{ W/m}^2\text{-K}$.

EN 673 was originally based on ISO 10292. ISO 10292 also covers rules for calculating thermal transmittance in the glazing central area (centre of glass U-value). This standard is similar to EN 673. The surface film coefficients in ISO 10292 are slightly different from EN 673: $h_e = 23 \text{ W/m}^2\text{-K}$ and $h_i = 8.0 \text{ W/m}^2\text{-K}$.

Simple Glazing Definition

If only basic manufacturers data on the glazing system is available then you may need to use the more approximate Simple glazing [definition method](#). In this case the glazing properties for the overall glazing system are defined using just three parameters:

- U-value,
- Solar heat gain coefficient (SHGC), and,
- Visible transmittance (VT) data.

You can read details of the method used behind the scenes in EnergyPlus to represent the simple glazing definitions in the report [Modeling Windows in Energy Plus with Simple Performance Indices](#) by Dariush Arasteh, Christian Kohler and Brent Griffith.

See also "Glazing Assemblies" on page 418

U-Value

This field describes the value for window system U-Factor, or overall heat transfer coefficient including inside and outside surface convective and radiant resistances. Units are in $\text{W/m}^2\text{K}$. This is the rated (NFRC) value for U-factor under winter heating conditions. The U-factor is assumed to be for vertically mounted products.

Tip: You can find some detailed information about the U-value to use and how the NFRC and EU format U-values differ on the [DesignBuilder b.i.g. website](#)

Solar heat gain coefficient (SHGC)

Enter the value for solar heat gain coefficient for the combined glazing system, a fractional value from 0 to 1 without units. This is the rated (NFRC) value for SHGC under summer cooling conditions and represents SHGC for normal incidence and vertical orientation.

Background information on solar transmission

- **g-value** is the coefficient commonly used in Europe to measure the solar energy transmittance of glass - called a Solar Factor on some window literature (%) i.e. 53% = 0.53;
- **Solar Heat Gain Coefficient (SHGC)** is used in the United States and most commonly refers to the solar energy transmittance of a window or door as a whole, factoring in the glass, frame material (wood, aluminium, etc.), sash (if present), divided lite bars (if present) and screens (if present). SHGC

may also refer to the solar energy transmittance of the glass alone (sometimes more specifically termed center-of-glass SHGC), in which case it is analogous to g-value.

- **Shading Coefficient** was succeeded by SHGC in the United States, but older windows and doors may still refer to their SC value. The relationship between SHGC and SC may be approximated as: $SHGC = SC \times 0.87$;

g-values and SHGC values ranges from 0 to 1, a lower value representing less solar gain. Shading coefficient values are calculated using the sum of the primary solar transmittance (T-value) and the secondary transmittance. Primary transmittance is the fraction of solar radiation that directly enters a building through a window compared to the total solar insolation, the amount of radiation that the window receives. The secondary transmittance is the fraction of inwardly flowing solar energy absorbed in the window again compared to the total solar insolation.

[Source Wiki Solar Gain](#)

Visible transmittance

Enter the visible transmittance for the glazing coefficient, a fractional value from 0 to 1. It is the rated (NFRC) value for visible transmittance at normal incidence.

Frame

Simple glazing data can either include or exclude the effect of the frame. If the glazing data includes the frame then you should make sure to switch off frames in the openings model data to ensure that they are not double-counted.

Note: When using the simple glazing definition it is important to understand that there may be significant differences in performance between the simple window system and the usual more detailed model. This option should therefore only be used when detailed layered data is not available.

Glazing Cost Data

Cost tab on [Glazing Dialog](#)

Glazing cost is defined as the cost per area of window.

Cost per surface area

The cost per surface area the overall cost of the window per area. Make sure to include labour and any other costs directly relating to this cost.

Pane Components

Pane components holds data on the [thermal](#), [solar](#), [visible](#) and [infrared](#) properties of individual window panes.

There are five tabs on the Pane dialog:

- [General](#)
- [Thermal](#)
- [Solar](#)
- [Visible](#)
- [Infra-red](#)
- [Spectral data](#)

Tip: The Info Panel on the right of the Dialog contains information on the naming convention is used with EnergyPlus glazing data.

International Glazing Database (IGDB)

DesignBuilder's Panels database includes all IGDB glass types with detailed spectral data for around 6,000 manufacturers glazing pane products from around the world. The IGDB is a comprehensive international glazing layer database compiled by NFRC and provided through the specialist WINDOW and OPTICS fenestration analysis software provided by LBNL. DesignBuilder is kept up to date with the latest version of the IGDB.

Note: DesignBuilder provides an option to [load Glass data](#) directly from WINDOW mdb files through a link on the Info panel.

Panel component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by clicking on the Edit button when selecting panels from a list.

Panels - General

General tab on [Panels Dialog](#).

Enter the name, description and category of the panel. Also enter the source of the data.

Data type

The Data type allows you to define whether the glazing pane has solar transmission and reflectance data in spectral or broadband form. The options are:

- **1-Broadband** - where transmission and reflection data is averaged across all wavelengths,
- **2-Spectral** - the more accurate option where solar transmission and reflectance data is defined for a range of wavelengths between 0.1 and 4 microns on the [Spectral data](#) tab of the dialog.

Note: Where possible you should use glazing panels which have Spectral data defined because the calculations in EnergyPlus are more accurate accounting for the way solar transmission varies with wavelength. In particular, the results for glazing systems using spectral data can be significantly more accurate relative to the equivalent broadband data for panels with selective coatings e.g. low-e glass.

Panels - Thermal Data

Thermal tab on [Panels Dialog](#).

Thickness

Enter the thickness of the panel.

Thermal conductivity

This field is used to enter the thermal conductivity of the material layer. Thermal conductivity must be greater than zero.

Panes - Solar Data

Solar tab on [Panes Dialog](#).

For uncoated glass, when alternative optical properties are available, such as thickness, solar index of refraction, and solar extinction coefficient, they can be converted to the equivalent solar transmittance and reflectance values using the equations given in [Glass Optical Properties Conversion](#).

This data is not used in simulations when the [Data type](#) is **2-Spectral** but it is still displayed because it gives a spectral-average overview of the pane's solar transmission and reflectance.

Solar transmittance

The solar transmittance for the pane at normal incidence averaged over the solar spectrum.

Solar reflectance

The inside (back-side) and outside (front-side) solar reflectance properties of the pane at normal incidence averaged over the solar spectrum.

Note: For uncoated glass, when alternative optical properties (thickness, solar index of refraction, and solar extinction coefficient) are available, they can be converted to equivalent solar transmittance and reflectance values using the equations given in the "Glass Optical Properties Conversion" on page 1406 topic.

Panes - Visible Data

Visible tab on [Panes Dialog](#).

For uncoated glass, when alternative optical properties are available—such as thickness, solar index of refraction, and solar extinction coefficient—they can be converted to equivalent solar transmittance and reflectance values using the equations given in [Glass Optical Properties Conversion](#).

Visible transmittance

The visible transmittance for the pane at normal incidence averaged over the solar spectrum, and weighted by the response of the human eye.

Visible reflectance

The inside (back-side) and outside (front-side) visible reflectance properties of the pane at normal incidence averaged over the solar spectrum, and weighted by the response of the human eye.

Panes - Long-wave (Infra-red) Properties

Infra-red tab on [Panes Dialog](#).

Long-wave transmittance

The long wave transmittance at normal incidence.

Note that although EnergyPlus data input allows a range of inputs between 0 and 1 the only value that is allowed in the calculations is zero so this value is now fixed at zero.

Emissivity

Enter the inside (back-side) and outside (front-side) hemispherical emissivity properties of the pane. Note that emissivity is 1 minus the infra red reflectance value.

Spectral Data

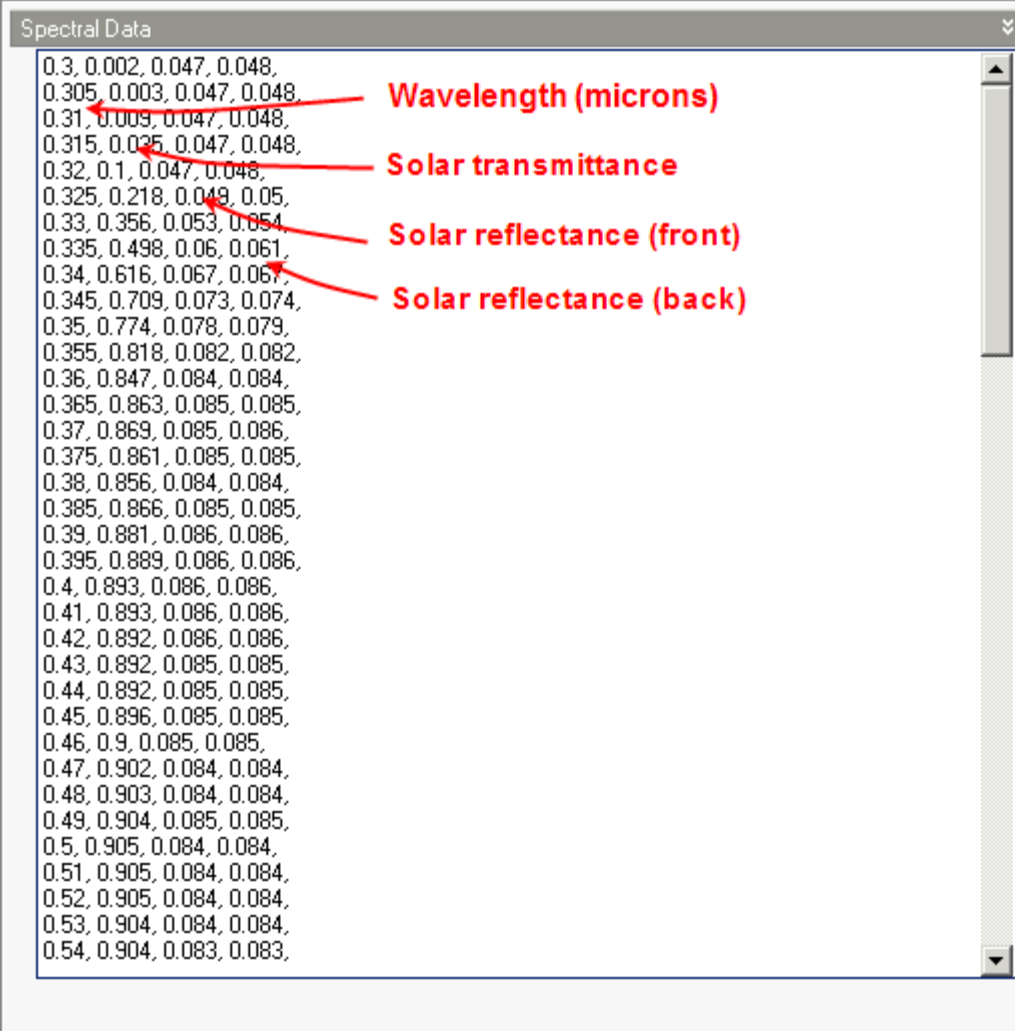
Spectral data tab on [Panels Dialog](#).

Solar transmission and reflectance data is entered for a range of wavelengths between 0.1 and 4 microns. This data will generally have been extracted from the [IGDB](#) (International Glazing Database) and supplied by DesignBuilder.

The data is arranged in comma-separated table form with a single line per wavelength, each line having 4 columns:

1. Wavelength (microns)
2. Solar transmittance (Tsol)
3. Solar reflectance, front (Rsol1)
4. Solar reflectance, back (Rsol2)

An example is shown in the screenshot below.



Wavelength (microns)	Solar transmittance	Solar reflectance (front)	Solar reflectance (back)
0.3	0.002	0.047	0.048
0.305	0.003	0.047	0.048
0.31	0.009	0.047	0.048
0.315	0.035	0.047	0.048
0.32	0.1	0.047	0.048
0.325	0.218	0.048	0.05
0.33	0.356	0.053	0.054
0.335	0.498	0.06	0.061
0.34	0.616	0.067	0.067
0.345	0.709	0.073	0.074
0.35	0.774	0.078	0.079
0.355	0.818	0.082	0.082
0.36	0.847	0.084	0.084
0.365	0.863	0.085	0.085
0.37	0.869	0.085	0.086
0.375	0.861	0.085	0.085
0.38	0.856	0.084	0.084
0.385	0.866	0.085	0.085
0.39	0.881	0.086	0.086
0.395	0.889	0.086	0.086
0.4	0.893	0.086	0.086
0.41	0.893	0.086	0.086
0.42	0.892	0.086	0.086
0.43	0.892	0.085	0.085
0.44	0.892	0.085	0.085
0.45	0.896	0.085	0.085
0.46	0.9	0.085	0.085
0.47	0.902	0.084	0.084
0.48	0.903	0.084	0.084
0.49	0.904	0.085	0.085
0.5	0.905	0.084	0.084
0.51	0.905	0.084	0.084
0.52	0.905	0.084	0.084
0.53	0.904	0.084	0.084
0.54	0.904	0.083	0.083

In cases where the pane layer has been flipped in the Glazing dialog then the last 2 columns (front and back reflectance) will be reversed in the data supplied to EnergyPlus for the pane layer.

Note: DesignBuilder allows spectral data to be edited either at library level or within the model.

Glass Optical Properties Conversion

Conversion from Glass Optical Properties Specified as Index of Refraction and Transmittance at Normal Incidence

The optical properties of uncoated glass are sometimes specified by index of refraction, n , and transmittance at normal incidence, T . The following equations show how to convert from this set of values to the transmittance and reflectance values required by DesignBuilder and EnergyPlus. These equations apply only to uncoated glass, and can be used to convert either spectral-average solar properties or spectral-average visible properties (in general, n and T are different for the solar and visible). Note that since the glass is uncoated, the front and back reflectances are the same and equal to the R that is solved for in the following equations.

Given n and T , find R :

$$r = \left(\frac{n-1}{n+1} \right)^2$$
$$\tau = \frac{\left[(1-r)^4 + 4r^2 T^2 \right]^{1/2} - (1-r)^2}{2r^2 T}$$
$$R = r + \frac{(1-r)^2 r \tau^2}{1 - r^2 \tau^2}$$

Example:

$$T = 0.86156$$

$$n = 1.526$$

$$r = \left(\frac{1.526-1}{1.526+1} \right)^2$$

$$\tau = 0.93974$$

$$R = 0.07846$$

Load Glass Data From Berkeley Lab WINDOW

Accessed from the [Panels Dialog](#)

DesignBuilder provides an option to load Glass data directly from an [Berkeley Lab WINDOW](#) mdb file into the current Pane dataset. The option is accessed either from the **Import** button at the bottom of the Panels dialog, or when using Learning mode, from a link on the Info panel. When selecting the Import option you will be prompted for the name of the mdb file containing the data to be loaded. Having selected the mdb filename a list of the Glass types is presented as shown in the screenshot below.

Select Glass from WINDOW6 Database

Manufacturer /

ID	Name	Source	SpectralData	AngularFunction	Thickness	Tsol	Rsol1	Rsol2	Tvis	Rvis1	Rvis2
3M Corporation											
2714	RE35AMAR.MMM	IGDB v16.2	1	0	3.13	0.1779262	0.5763801	0.7140068	0.3014347	0.5566363	0.6032E-02
2716	RE35NEAR.MMM	IGDB v16.2	1	0	3.13	0.3069355	0.1871686	0.194952	0.3622821	0.1995945	0.1790E-02
2711	RE20BRAR.MMM	IGDB v11.4	1	0	0.0381	0.2655831	0.1573364	0.1736665	0.2289016	9.983931E-02	0.156E-02
2712	RE20NEAR.MMM	IGDB v16.2	1	0	3.15	0.1243509	0.1948055	0.194025	0.1562207	0.20984	0.1862E-02
2719	RE50NEAR.MMM	IGDB v16.2	1	0	3.14	0.4608931	0.1169612	0.1107807	0.5286015	0.1246216	0.108E-02
2740	FXHP5.mmm	IGDB v16.0	1	0	5.7042	0.2434029	0.0601598	8.738703E-02	3.129333E-02	4.767797E-02	4.686366E-02
2739	FXHP35.mmm	IGDB v16.0	1	0	5.7042	0.4251789	7.241947E-02	8.659121E-02	0.4143532	6.635303E-02	8.047184E-02
2742	FXST20.mmm	IGDB v16.0	1	0	5.7042	0.4605078	5.467862E-02	0.0539287	0.2305886	4.712794E-02	4.554451E-02
2741	FXHP50.mmm	IGDB v16.0	1	0	5.7042	0.4543949	7.945746E-02	8.648749E-02	0.487134	7.702281E-02	8.162121E-02
2702	CM 40.mmm	IGDB v17.2	1	0	3.3	0.3440335	0.1907181	0.1829029	0.4572499	0.1443781	0.119E-02
2703	LE35AMAR.MMM	IGDB v16.2	1	0	3.14	0.182605	0.5587685	0.6831188	0.3055365	0.5334418	0.5641E-02
2700	CM 30.mmm	IGDB v17.2	1	0	3.3	0.2641097	0.236684	0.23221	0.3669764	0.1746716	0.1511E-02
2701	CM 50.mmm	IGDB v17.2	1	0	3.3	0.4353527	0.1492792	0.1373351	0.5454012	0.1204625	9.726869E-02
2704	P18AR.MMM	IGDB v16.2	1	0	3.16	0.1183051	0.5272442	0.6154482	0.1690424	0.5790399	0.578E-02
2707	PR90EXT.mmm	IGDB v18.1	1	0	3.3	0.6836252	0.2665618	0.2558249	0.9049547	9.009282E-02	0.0900E-02
2710	RE15SX.MMM	IGDB v11.4	1	0	0.0381	0.1168002	0.5980678	0.6153922	0.1593076	0.5785106	0.599E-02
2705	PR40EXT.mmm	IGDB v18.1	1	0	3.3	0.2544533	0.2404767	7.305787E-02	0.4316012	5.687163E-02	5.302092E-02
2706	PR70EXT.mmm	IGDB v18.1	1	0	3.3	0.3982	0.2458833	9.577186E-02	0.7285737	7.381196E-02	6.998923E-02
2727	NV-45.MMM	IGDB v13.6	1	0	0.051	0.5374409	6.252895E-02	6.876728E-02	0.5114503	6.219453E-02	6.535801E-02
2726	NV35.mmm	IGDB v17.0	1	0	3.17	0.3361699	0.1325934	8.238679E-02	0.3644603	0.1283528	6.549683E-02
2729	BC10.mmm	IGDB v16.0	1	0	5.7042	8.940493E-02	0.2382922	0.2189598	0.1037745	0.2210669	0.182E-02
2728	NV15.mmm	IGDB v17.0	1	0	3.16	0.1325114	0.3705041	0.1730767	0.1506019	0.388105	0.1061E-02
2725	NV25.mmm	IGDB v17.0	1	0	3.16	0.2295365	0.2030218	0.105289	0.2451216	0.1980986	7.117056E-02
2722	S35NEAR.MMM	IGDB v16.2	1	0	3.25	0.3027105	0.1881932	0.1883017	0.3557022	0.2086116	0.177E-02
2721	RE70NEAR.MMM	IGDB v16.2	1	0	3.14	0.6379107	8.541232E-02	7.611196E-02	0.704926	9.465799E-02	8.298125E-02
2724	SCLAR400.MMM	IGDB v11.4	1	0	0.1016	0.8447313	8.548506E-02	8.712485E-02	0.8703511	9.416591E-02	9.436654E-02
2723	SCLAR150.MMM	IGDB v11.4	1	0	0.0508	0.8573449	7.803981E-02	0.079541	0.882374	8.469272E-02	8.578935E-02
2736	CS50.mmm	IGDB v16.0	1	0	5.6768	0.5316811	5.495222E-02	7.117603E-02	0.5174848	5.378936E-02	6.671356E-02

OK Cancel Help Collapse All Expand All

Select the Glass type to be imported by clicking on the appropriate row of data and pressing the OK button. All Glass data supported by DesignBuilder, including spectral data, will be loaded from the database into the currently open Pane dialog.

Data is grouped by Manufacturer by default, but other column headers can be selected as the grouping category instead by dragging the existing group header back to the column header row and dragging the new grouping column category header to the group box. The screenshot below shows the data grouped by the IGDB version number (when the product was introduced) instead.

Select Glass from WINDOW6 Database

Source /

ID	Name	Manufacturer	Source /	SpectralData	AngularFunction	Thickness	Tsol	Rsol1	Rsol2	Tvis
IGDB v17.2										
IGDB v17.3										
IGDB v17.4										
IGDB v17.5										
IGDB v18.0										
IGDB v18.1										
IGDB v18.2										
IGDB v19.0										
IGDB v20.0										
3376	CG5527Emb3L3.grd	Guardian	IGDB v20.0	1	0	6.7056	0.2353461	0.4221701	0.4020498	0.
3377	CG5527Emb3L6.grd	Guardian	IGDB v20.0	1	0	7.4676	0.2328004	0.422098	0.3797325	0.
3374	CG5527Emb27L6.grd	Guardian	IGDB v20.0	1	0	6.8072	0.2343721	0.4308026	0.3871758	0.
3375	CG5527Emb27L9.grd	Guardian	IGDB v20.0	1	0	7.5692	0.2322608	0.4307303	0.3708819	0.
3378	CG5527Emb3L9.grd	Guardian	IGDB v20.0	1	0	8.2296	0.2306912	0.4220279	0.3637637	0.
3381	CG5527Emb31L9.grd	Guardian	IGDB v20.0	1	0	8.432799	0.2302191	0.4193899	0.3616125	0.
3382	CG5527Exp23L3.grd	Guardian	IGDB v20.0	1	0	5.334	0.2563404	0.2834943	0.5107805	0.
3379	CG5527Emb31L3.grd	Guardian	IGDB v20.0	1	0	6.908801	0.2348566	0.4195341	0.3996575	0.
3380	CG5527Emb31L6.grd	Guardian	IGDB v20.0	1	0	7.6708	0.2323262	0.4194581	0.3774819	0.
3373	CG5527Emb27L3.grd	Guardian	IGDB v20.0	1	0	6.0452	0.2369305	0.4308789	0.409936	0.
16080	SVSHLBrE.pfg	PFG Building Glass	IGDB v20.0	1	0	7.065	0.3200808	7.405759E-02	0.108335	0.
16079	SVSHLBr.pfg	PFG Building Glass	IGDB v20.0	1	0	6.075	0.4340155	0.0559384	5.490299E-02	0.
16082	SVSHLGrE.pfg	PFG Building Glass	IGDB v20.0	1	0	7.071	0.3266756	6.394526E-02	0.1064878	0.
16081	SVSHLGr.pfg	PFG Building Glass	IGDB v20.0	1	0	6.037	0.4162568	5.628137E-02	5.445027E-02	0.
16078	SVSHLBI.pfg	PFG Building Glass	IGDB v20.0	1	0	7.092	0.4482642	7.473046E-02	0.1094559	0.
3371	CG5527Emb23L6.grd	Guardian	IGDB v20.0	1	0	6.096	0.2361023	0.4404492	0.3954099	0.
3372	CG5527Emb23L9.grd	Guardian	IGDB v20.0	1	0	6.858	0.2339718	0.4403715	0.3787532	0.
16077	SVSHLBI.pfg	PFG Building Glass	IGDB v20.0	1	0	6.091	0.5033199	0.066608	6.327897E-02	0.
3370	CG5527Emb23L3.grd	Guardian	IGDB v20.0	1	0	5.334	0.2386776	0.4405221	0.4186561	0.
16075	SVSHLAq.pfg	PFG Building Glass	IGDB v20.0	1	0	6.044	0.4761205	6.807618E-02	6.526887E-02	0.

OK Cancel Help Collapse All Expand All

Buttons are provided to the right of the screen to allow you to expand or collapse all of the group headings.

You can sort the data by any column by clicking on the appropriate column header. Click again to toggle between ascending and descending sort order.

Importing New IGDB Data

DesignBuilder is supplied with recent IGDB (International Glazing Database) data, but the IGDB is updated at regular intervals and so the very latest glazing data may not be included in DesignBuilder. However, the latest IGDB data can be [downloaded from LBNL in the form of a database](#). This database can be imported to a WINDOW mdb file where it can be accessed from DesignBuilder as described above.

Note: There is no direct way to import data directly the IGDB database. The data must be imported as a 2-stage process: IGDB to WINDOW then WINDOW to DesignBuilder.

Pane Group Components

A Pane group is a list of panes used to model variable glazing pane layer performance characteristics. Pane groups are used to model Thermochromic glazing whose transmission properties depend on the temperature of the glass.

To define a pane group, first set the **Number of bands** to be used to define varying performance. Each band has a Temperature below which the simulated pane layer should use the properties of the corresponding pane.

The pane associated with a particular band becomes active when the temperature of the thermochromic layer rises above the band midpoint temperature. In the example below the performance of the thermochromic glazing is defined using 5 bands. Each band defines the transmission characteristics of a pane layer for a particular temperature range. When the temperature is at or below the first band mid-point $(24 + 34) / 2 = 29^{\circ}\text{C}$, the pane LBNL Thermochromic1_24... is used to define the glazing layer characteristics. For temperatures between the first band mid-point and the second band mid-point (29°C to 41°C) the pane LBNL Thermochromic1_34... is used etc.

General	
Name	Thermochromic Example
Category	Thermochromic
Number of bands	5
Band 1	
☐ Pane type	LBNL Thermochromic1_24 <Ther
Temperature (°C)	24
☐ Flip layer	
Band 2	
☐ Pane type	LBNL Thermochromic1_34 <Ther
Temperature (°C)	34
☐ Flip layer	
Band 3	
☐ Pane type	LBNL Thermochromic1_48 <Ther
Temperature (°C)	48
☐ Flip layer	
Band 4	
☐ Pane type	LBNL Thermochromic1_62 <Ther
Temperature (°C)	62
☐ Flip layer	
Band 5	
☐ Pane type	LBNL Thermochromic1_75 <Ther
Temperature (°C)	75
☐ Flip layer	

Flip layer

You can use the **Flip layer** checkboxes to reverse the direction of the associated glazing layer so that the inside face becomes the outside face and vice-versa.

Schedules

Schedules are used in DesignBuilder to define:

- Occupancy times
- Equipment, lighting HVAC operation
- Heating and Cooling temperature setpoints
- Transparency of component blocks (usually seasonal)

Apart from their use for defining [Transparency of component blocks](#), Schedules are only used in simulations when the [Timing](#) model option is set to 'Schedules'

You can define schedules in one of 2 ways:

- [7/12 Schedules](#) - where each day of the week and each month of the year has a unique daily variation defined using profiles.
- [Compact Schedules](#) - more flexible definition using EnergyPlus **Schedule:Compact** text-based data format.
- [Day schedules](#) - special type of schedule defining the profile for a single day. Used for defining [Cooling design day range multipliers](#).

Use the **Schedule type** drop list on the Schedules dialog to make the selection.

[Click here](#) for information on temperature setpoint schedule definition.

Tip: You can visualise all schedules used in the model by adding the following line to your IDF:

```
Output:Variable, *, Schedule Value, Hourly;
```

Then after a simulation has been run you can use the [Results Viewer](#) to view the schedule values output by EnergyPlus.

Extra IDF can be added using one of two methods: [EMS scripting](#) or the [Include IDF data](#) option.

Schedule data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the schedules icon in the model data,
- by clicking on the Edit button when selecting schedules from a list.

See also the [Timings, schedules and profiles](#) and [Compact schedules](#) tutorials

7/12 Schedules - Defined Using Profiles

General tab on [Schedules Dialog](#).

7/12 Schedules reference [Profiles](#) of daily variation, one Profile for each day of the week and for each month of the year (hence 7/12).

You can select multiple rows (months) or columns (days) and edit the data by clicking on the **Edit selected cells** link in the Info Panel. Likewise, you can select individual profiles by clicking on the appropriate cell and clicking **Edit selected cells**.

Design days

Winter and summer design day simulations are carried out when required for [autosizing](#) heating and cooling equipment prior to a Detailed HVAC simulation.

Design day definition method

With 7/12 Schedules there are 2 ways to define the schedule during these winter design day simulations, **1-End-use defaults** or **2-Profiles**:

1. End-use defaults

When the End use defaults option is selected you should define the intended use for the schedule using the options below and DesignBuilder controls whether the schedule is active during heating design using the options set on the [Heating Design](#) tab of the model options dialog:

- **General** - include general purpose schedules in Detailed HVAC heating [autosizing](#) calculations .
- **Occupancy** - defines occupancy
- **Lighting** - Schedule to be used for lighting
- **Equipment** - defines operation of equipment.
- **Heating demand** - defines operation of heating equipment
- **Cooling demand** - defines operation of cooling equipment

- **HVAC** - defines operation of general HVAC equipment
- **Natural ventilation demand** - defines operation of window opening and/or natural ventilation airflow operation
- **DHW** - defines operation of hot water equipment

It is also necessary to apply the [behaviour](#) for each category as defined on the [Heating design tab](#) of the Model options dialog.

Behaviour of the schedule during the **Summer design day** when autosizing cooling systems is simply taken from the profile for Wednesday, i.e. for any particular month, the schedule has the same behaviour during the cooling design day as it has for Wednesdays.

Note that if the schedule is to be used for heating or cooling operation then it is especially important to select the appropriate setting.

Warning: This system of categories can be confusing and has been underlying cause of many queries on the DesignBuilder Support Desk related to undersized HVAC systems. We therefore now recommend that you to use either the [2-Profiles](#) option where the schedule operation during heating and cooling are defined explicitly using a profile or use [Compact schedules](#) where again heating and cooling design operation is defined explicitly.

2. Profiles

In this option both heating and cooling design day operation are defined explicitly as a [Profile](#). This is more flexible and clearer than **End-use defaults** and is the preferred choice.

- **Heating design day profile** - the profile defining the schedule output during the heating autosizing design day.
- **Cooling design day profile** - the profile defining the schedule output during the cooling autosizing design day.

Important Note: When setting up schedules that will be used in Compact or Detailed HVAC simulations it is important to consider whether the schedule will be active or inactive during the heating and cooling sizing simulations used for autosizing equipment. Generally *internal gains must be excluded from heating sizing*. So you should make sure to select the "Off" schedule for the **Heating design day profile** if the schedule is to be applied to internal gains, occupancy, lighting etc.

Weekly and monthly variations

Other tools are available for rapid selection/de-selection and setting of schedule data:

- Select all weekdays
- Select all weekends
- Set all weekends 'Off'
- Select all
- Deselect all

General							
Name	Office_OpenOff_Occ						
Description	Building: OFFICE Area: OPEN PLAN OFFICE Occupancy schedule						
Source	UK NCM						
Category	OFFICE						
Region	General						
Schedule type	1-7/12 Schedule						
Design Days							
Design day definition method	1-End use defaults						
Use end-use default	2-Occupancy						
Profiles							
M...	Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday
Jan	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Feb	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Mar	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Apr	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
May	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Jun	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Jul	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Aug	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Sep	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Oct	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Nov	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off
Dec	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	8:00 to 18:00	Off

See also [EnergyPlus Compact Schedules](#)

See also the [Timings, schedules and profiles](#) tutorial

Compact Schedules

General tab on [Schedules](#) Dialog.

Compact Schedules are defined using a slightly modified version of the standard EnergyPlus **Schedule:Compact** format documented below.

Using the Compact Schedule definition all the features of the schedule components are accessed in a single command.

See also the [Compact schedules](#) tutorial

Name

This field is for your own use and is replaced by DesignBuilder with the name of the Schedule when generating data for EnergyPlus.

ScheduleType

ScheduleType should generally be **Fraction** even for schedules to be used for [temperature setpoint](#) definition.

You can enter **Any Number** to enter values greater than 1 but please bear in mind:

- The maximum gains values (e.g. people/m²) are multiplied by the values in the schedule to obtain the actual value to use at each time step in the simulation. So if you enter large schedule values the actual maximum value in the simulation will be higher than the 'maximum' value entered in DesignBuilder. There is no problem with this, just a possible concern that if the DesignBuilder interface indicates that the maximum occupancy density is say 0.1 people/m² but the schedule has values > 1 then the simulation results will, at first sight, appear incorrect. If you come back to the model 6 months later you may not remember the reason.
- [Temperature set-point schedules](#) should only have values 0, 0.5 or 1.0.

So generally we recommend using **Fraction** Compact schedules for clarity.

Field-Set (Through, For, Interpolate, Until, Value)

Each Compact Schedule must contain the elements **Through** (date), **For** (days), **Interpolate** (optional), **Until** (time of day) and **Value**.

Through

This field starts with 'Through:' and contains the [ending date](#) for the schedule period. You may define multiple schedule periods in this way, for example to define seasonal variations. If there are no seasonal variations then enter: Through: 31 Dec, to indicate that the data that follows applies to the whole year.

Note 1: each Compact Schedule must cover all the days for a year and it must have values for all 24 hours and all values for all day types.

Note 2: The dates in the Through fields must be provided consecutively, starting at the beginning of the year and finishing at the end of the year.

Note 3 only Month-Day combinations are allowed for this field.

For

This field starts with 'For: ' and contains the applicable days. Multiple choices may be combined on the line.

Choices are:

- Weekdays,
- Weekends,
- [Holidays](#),
- Alldays,
- [SummerDesignDay](#),
- [WinterDesignDay](#),
- Sunday,
- Monday,
- Tuesday,
- Wednesday,
- Thursday,
- Friday,
- Saturday,
- CustomDay1,
- CustomDay2.

In fields after the first 'For', 'AllOtherDays' may also be used.

Note: EnergyPlus has limit of line length of about 60 characters so lines like:

For: Weekdays SummerDesignDay WinterDesignDay Weekends Holidays AllOtherDays,
should be changed to:

For: AllDays,

Interpolate (optional)

Schedule values are input to the simulation at the zone time step, so there is also a possibility of 'interpolation' from the entries used in this object to the value used in the simulation. This field, if used, starts with 'Interpolate:' and contains the word 'Yes' or 'No'.

Until

This field contains the ending time for the current days and day schedule being defined. The data should represent clock time (standard) in the format HH:MM. 24 hour clock format (i.e. 1pm is 13:00). Note that 'Until: 7:00' includes all times up through 07:00 (7am). The time is followed by a comma and the value the schedule takes until the given time. See note above on usual values. Generally you should enter values between zero and 1.

Value

Finally, the value field is the schedule value for the specified time interval.

Note: you can use 0 for Temperature type Compact Schedules to indicate that the heating/cooling system is switched off during the specified period. See Example 2 below.

Example 1 - Operation Schedule

```
SCHEDULE:COMPACT,  
Office_CellOff_Light,  
Fraction,  
Through: 31 Dec,  
For: Weekdays SummerDesignDay,  
Until: 07:00, 0,  
Until: 19:00, 1,  
Until: 24:00, 0,  
For: Weekends,  
Until: 24:00, 0,  
For: Holidays WinterDesignDay AllOtherDays,  
Until: 24:00, 0;
```

This schedule is intended to be used to describe the lighting operation in an office. It indicates:

- Schedule is a fraction (0-1).
- No seasonal variation.
- Weekdays and [SummerDesignDays](#) the schedule is 1 (i.e. 'on') between 7am and 7pm and 0 (i.e. 'off') other times.
- Weekends, holidays, [WinterDesignDays](#) and all other days the schedule is 'off'.

Important Note on SummerDesignDays and WinterDesignDays. A common mistake is to inadvertently include internal gains from occupancy, computer gains or lighting in heating sizing simulations. This leads to heating systems to being undersized. The general rule is to avoid including any internal gains in heating

sizing simulations. This is done as shown in **Example 1** above, by using a value of 0 for WinterDesignDay for any schedule to be used to control internal gains.

Example 2 - Temperature Setpoint Schedule

```
SCHEDULE:COMPACT,
Bedroom_Cool,
Temperature,
Through: 31 Dec,
For: Weekdays SummerDesignDay,
Until: 05:00, 0.5,
Until: 09:00, 1,
Until: 17:00, 0.5,
Until: 24:00, 1,
For: Weekends,
Until: 05:00, 0.5,
Until: 24:00, 1,
For: Holidays WinterDesignDay AllOtherDays,
Until: 24:00, 0;
```

This schedule is intended to be used to describe the cooling temperature setpoint and setback in a bedroom. It indicates:

- Schedule is a temperature (in degC).
- No seasonal variation.
- Weekdays and [SummerDesignDays](#) the [main](#) temperature setpoint data (Activity tab) is used between 5am and 9am and also between 5pm and midnight. Cooling operates at [setback](#) temperature setpoint data at all other times on these days.
- Weekends the cooling setpoint is [main](#) temperature setpoint between 5am and midnight. Cooling operates at [setback](#) temperature setpoint data at all other times on these days.
- Holidays, [WinterDesignDays](#) and all other days the cooling is off.

IMPORTANT NOTE: even if you are using IP units for the main interface **you must use SI units for compact schedule values** as these are passed directly to EnergyPlus.

More on [Temperature setpoint schedules](#).

Example 3 - Heating Design Days

This example is intended to highlight a point about how WinterDesignDay (and by extension SummerDesignDay) values are defined. Looking at the compact schedule data below we can see that the WinterDesignDay term has been used in the "Through: 6 Jan" section (highlighted in yellow) and so at first sight you might expect that the winter design calculation will use a value of 1 for the heating design and heating sizing calculations. However, if the Winter design day date is set to 15 Jan (as it is by default for Northern Hemisphere locations in v5 and earlier) then the winter design day schedule value will come from the "Through: 31 Jan" section, where it is covered by the AllOtherDays day type and has a value of 0. This would result in incorrect system sizing for this heating schedule.

The general point is to be aware of the dates(s) used for winter and summer design days and to ensure that the values are set as required in the schedules, ideally for all Through sections so that if the design day dates change the schedule will still be valid.

```
SCHEDULE:COMPACT,
Heating Sch,
Fraction,
Through: 6 Jan,
For: Weekdays,
Until: 06:00, 0,
Until: 18:00, 1,
```

```

Until: 24:00, 0,
For: Weekends,
Until: 24:00, 0,
For: WinterDesignDay,
Until: 24:00, 1,
For: AllOtherDays,
Until: 24:00, 0,
Through: 31 Jan,
For: Weekdays,
Until: 06:00, 0,
Until: 18:00, 1,
Until: 24:00, 0,
For: Weekends,
Until: 24:00, 0,
For: AllOtherDays,
Until: 24:00, 0,

```

... etc

Day Schedule

General tab on [Schedules Dialog](#).

Day schedules are a special type of schedule covering only a single day. They have format similar to that of Compact schedules but because they only cover one day they don't include Through: and For: statements as shown below.

```

Schedule:Day:Interval,
Daily range multipliers for Design Days,
Any Number,
Yes
Until: 1:00, 0.88 ,
Until: 2:00, 0.92,
Until: 3:00, 0.95,
Until: 4:00, 0.98,
Until: 5:00, 1.00,
Until: 6:00, 0.98,
Until: 7:00, 0.91,
Until: 8:00, 0.74,
Until: 9:00, 0.55,
Until: 10:00, 0.38,
Until: 11:00, 0.23,
Until: 12:00, 0.13,
Until: 13:00, 0.05,
Until: 14:00, 0.00,
Until: 15:00, 0.00,
Until: 16:00, 0.06,
Until: 17:00, 0.14,
Until: 18:00, 0.24,
Until: 19:00, 0.39,
Until: 20:00, 0.50,
Until: 21:00, 0.59,
Until: 22:00, 0.68,
Until: 23:00, 0.75,
Until: 24:00, 0.82;

```

The syntax of Day schedule data is not checked by DesignBuilder in the same way as it is for Compact schedules, so it is important to check the data carefully before pressing OK on the Schedules dialog.

Note: Day schedules are only used for defining [Range multipliers](#) in cooling design data.

Defining Setpoint Temperature And RH Schedules

Schedules Timing Model Option

Operation of heating and cooling equipment is defined by selecting a [Schedule](#) under the Heating, Cooling and Operation headers on the HVAC tab. These schedules are used to generate both the heating/cooling availability schedule and the temperature setpoint schedules used in the simulations. Schedule values greater than 0.5 indicate the system is in full operation controlling temperature to the [main heating/cooling setpoint](#), values between 0 and 0.5 indicate the equipment in [setback](#) mode and values of 0 indicate that the equipment is switched off. See Table below.

Value	Heating/Cooling Equipment Operation
0	Equipment is off
> 0.5	Equipment is controlled using the main temperature setpoint data (Activity tab)
0 < Value <= 0.5	Equipment is controlled using the setback temperature setpoint data (Activity tab)

Notes

1. When using [7/12 Schedules](#), for 0.5 read 50% etc.
2. When using [2-Single period](#) profiles with [7/12 Schedules](#) to generate temperature setpoints the 'Off' periods are modelled using setback temperatures, i.e. the system is not switched off. If you wish the system to switch off outside main heating/cooling hours use the [3-Custom](#) profile type.
3. If the setback temperature set does not appear to affect results, this may be because the heating/cooling operation schedule selected uses only 0 and 1, not the 0.5 required to access setback temperature (see table above).

Typical Workday Timing Model Option

You can define the operation of heating and cooling equipment by entering [Typical workday](#) start and end times and seasonal operation under the Heating and Operation headers on the HVAC tab.

This data is used to generate both the heating/cooling availability schedule and the temperature setpoint schedule used in the simulations.

The equipment is switched on between the On and Off times (8am and 6pm in the example below). During this time the equipment is controlled using the [main](#) temperature setpoint (Activity tab). At other times and during weekends the equipment is controlled using the [setback](#) temperature setpoint. Weekends are defined on the Activity tab at the building level under [Occupancy](#) by setting the number of working days per week.

The Seasonal control data allows you to set times when the equipment is switched off altogether. For example the heating system data shown below indicates that the heating system will be switched on during the winter only and switched off in the summer.

Operation

On at 8:00

Off at 18:00

Seasonal control 3-Winter only

See also the [Timing, Schedules, Profiles and Holidays Tutorial](#)

Schedules And Design And Sizing Simulations

It is important to differentiate between the heating and cooling sizing calculations which are accessed from the [Heating design](#) and [Cooling design](#) screens and the HVAC autosizing simulations that are automatically carried out before EnergyPlus simulations when using the Detailed HVAC option and items of HVAC equipment are to be autosized.

Heating And Cooling Design Calculation

Schedules are not used for [Heating design calculations](#) which are based on a steady state analysis that does not account for timing. In cases where for example a maximum mechanical ventilation rate is defined and modified by a schedule for simulation purposes, for the steady-state Heating design calculations the same maximum mechanical ventilation rate is used continuously.

[Cooling design calculations](#) are of course dynamic and so schedules are required for these. It is possible to select the day type to be used for Cooling design calculations on the Calculation options dialog. By default the Compact schedule day type is SummerDesignDay for Cooling design calculations. For cooling sizing calculations carried out before Compact and Detailed HVAC simulations the **SummerDesignDay** day type is always used.

HVAC Autosizing Simulations

EnergyPlus provides [WinterDesignDay](#) and [SummerDesignDay](#) day types in Compact schedules to cover the behaviour of schedules during autosizing simulations. When setting up schedules that will be used in Detailed HVAC simulations it is important to consider whether the schedule will be active or inactive during the heating and cooling sizing simulations used for autosizing equipment. In particular internal gains must be excluded from heating sizing simulations to avoid undersizing the heating equipment.

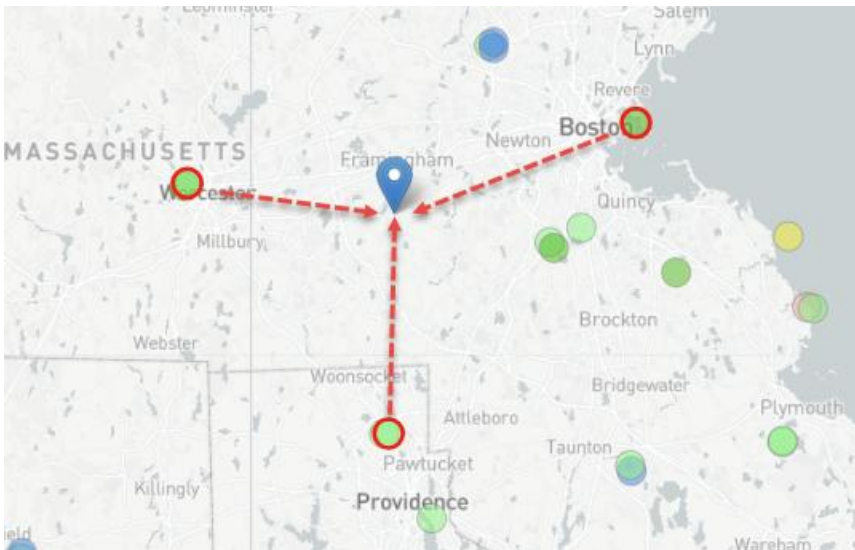
More specific advice on this can be found in the [7/12 schedules](#) and [Compact schedules](#) sections.

More general advice on HVAC autosizing can be found in the [Autosizing HVAC Components](#) section.

Simulation Hourly Weather Data

DesignBuilder uses [EnergyPlus format](#) hourly weather data to define external conditions during simulations. Each location has a separate file describing the external temperature, solar radiation, atmospheric conditions etc. for every hour of the year at that location. These hourly weather data sets are often "Typical Year" data derived from hourly observations at a specific location by National weather services. Examples of these Typical Year data sources include TMY2 and WYEC2 in the United States and Canada and TRY (CEC 1985) in Europe. See [Sources of EnergyPlus Hourly Weather Data](#) for more information on the hourly weather data itself and how to obtain it.

Because hourly weather data is not always available for every location, it may be necessary to use weather data for a nearby location which represents the weather of the actual site. Alternatively, with a licence for [Climate Analytics](#) it is possible to create hourly weather files for custom locations by interpolating between up to 4 existing stations.



The hourly weather data supplied with DesignBuilder has been obtained from the [Weather section of the EnergyPlus website](#) and processed using the [DesignBuilder weather data translator](#), but you can use any weather data provided it is in the correct [EnergyPlus epw file format](#).

See also the [Site level data and weather files](#) and [Hourly weather data](#) tutorials

Downloading Hourly Weather Data

DesignBuilder comes with a database of hourly weather data but the EnergyPlus epw files are not loaded onto your computer until you actually need them. The first time you require an hourly weather data file it is automatically downloaded from the DesignBuilder Website into the [Weather data folder](#) on your computer where it can be accessed by EnergyPlus for any subsequent simulations. You must be connected to the Internet to download weather data in this way.

Selecting hourly weather data

When you select an hourly weather data set on the [Location tab](#) at [Site level](#), you do not select the file directly, but rather an hourly weather data component, which in turn references the actual file. It works this way to allow us to supply a database of hourly weather data locations without needing to include hundreds of large hourly weather files.

When editing an hourly weather data component there are two tabs on the Hourly Weather Data dialog:

- [General](#)
- [Statistics](#)

See also:

- [Add new hourly weather data](#)
- [Edit/Translate hourly weather data](#)
- "Sources of EnergyPlus Hourly Weather Data" on page 1436

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Hourly Weather Data - General

General tab on [Hourly Weather data Dialog](#).

Select the Hourly weather data filename and the rest of the data is loaded directly from the .epw and .stat files.

Each file is named using the ISO standard three-letter country abbreviation (i.e. CUB for Cuba), followed by the location name, and the source format (CityUHK, CSWD, CTYW, CTZ2, CWEC, ISHRAE, IWEW, IGDG, INETI, KISR, SWEC, SWERA, TMY2, or TMY).

Thus, **CUB_Havana_IWEW.epw** is the EnergyPlus weather file (epw) from the ASHRAE International Weather for Energy Calculations (IWEW) data for Havana, Cuba (CUB).

There are three files associated with each location:

- **epw** - EnergyPlus weather files
- **stat** - a summary report on the data. Data from this file is loaded into the [Statistics tab](#) when an hourly weather data file is selected.
- **ddy** - design conditions.

Note: the above hourly weather data and associated statistics files are located in the [Weather data folder](#).

Hourly Weather Data - Statistics

Statistics tab on [Hourly Weather data Dialog](#).

The statistics displayed are generated by the EnergyPlus weather data translation utility.

The statistics data is used to identify useful simulation periods:

- **Summer start month, Summer end month** define the summer simulation period in **mmm** format.
- **Winter start month, Winter end month** define the winter simulation period in **mmm** format.
- **Extreme hot week, starting** defines the first day of the hottest week of the year.
- **Typical hot week, starting** defines the first day of a typical summer week.
- **Extreme cold week, starting** defines the first day of the coldest week of the year.
- **Typical cold week, starting** defines the first day of a typical winter week.

Note: week starting dates must be in the format **mmm dd** - any other format will generate an error when clicking on the links in the Simulation Options dialog.

This data can be accessed from the [Simulation Calculation options dialog](#) to set up typical simulation periods (screenshot right) .

Add New Hourly Weather Data

This topic explains how to add new hourly weather data to the library or the model.

See also the [Hourly weather data](#) tutorial

Add Hourly Weather Data To The Library

If you have hourly weather you wish to use in simulation you can add it to the *library* by following the steps below:

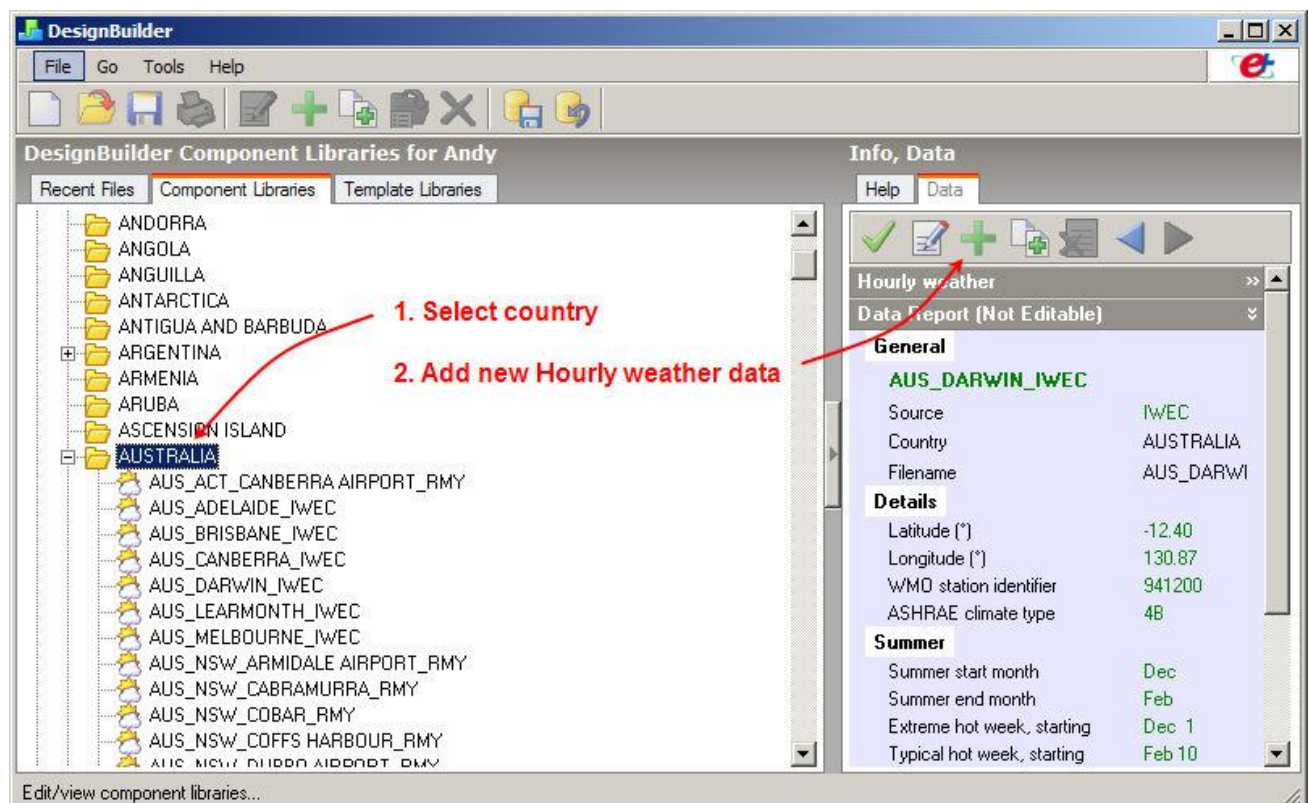
1. Copy the **EnergyPlus format** hourly weather data [.epw](#) file to the [weather data folder](#) (File > Folders > Weather data). If you have it, copy the [.stat](#) file there too.
2. From the Opening screen click on the Component libraries tab.
3. Click on the + to the left of Hourly weather data to open up the list of countries.
4. Select the country of the location (or a location within the country).
5. Add a new Hourly weather data set by clicking on the green + icon in the Toolbar.
6. Select the epw file and enter any extra information in the dialog as described below. Note that the location and other statistics are automatically loaded from the [.stat and .epw files](#) when you select the file.
7. Click OK to save the new Hourly weather data.

See further below for the alternative steps to follow if you have a file open and wish to create a new hourly weather data set just for that model.

Note: Any custom hourly weather epw file you select in your model will be saved with the dsb file so that the model can be used on other computers without the need to copy weather files manually.

Note: Although DesignBuilder can be used with any valid epw data sets, you are advised to [process the data](#) first to ensure that each hourly record has year 2002. This ensures that daily, hourly and sub-hourly data generated by EnergyPlus is correctly synchronised with the DesignBuilder graphics display. If you do not set the year to 2002 it will not affect the results but it will affect the day of the week display in DesignBuilder graphic output. In particular, daily, hourly and sub-hourly data will not necessarily show the correct day of the week in the results display graphics.

The steps are shown in the screenshots below.



In the Hourly weather data, enter the details of the location and select the EnergyPlus [epw](#) filename as shown below.

Hourly weather Data

General Statistics

General

Name New hourly weather data name

Source

Country AUSTRALIA

Filename

Details

Latitude (°)

Longitude (°)

WMO station identifier

ASHRAE climate type

Click here to select the EnergyPlus epw file

Once the epw has been selected, DesignBuilder loads the Latitude, Longitude and WMO station identifier.

Hourly weather Data

General Statistics

General

Name New hourly weather data name

Source IwEC

Country AUSTRALIA

Filename AUS_MELBOURNE_IwEC.epw

Details

Latitude (°) -37.67

Longitude (°) 144.83

WMO station identifier 948660

ASHRAE climate type 3C

It also searches for the equivalent [.stat](#) file and if it is found (in the [Weather data folder](#)) the data on the [Statistics tab](#) is also loaded (below).

Hourly weather Data	
General	Statistics
Summer	
Summer start month	Dec
Summer end month	Feb
Extreme hot week, starting	Jan 27
Typical hot week, starting	Feb 17
Cooling degree-days (Base 10°C) (Degree days)	1525
Winter	
Winter start month	Jun
Winter end month	Aug
Extreme cold week, starting	Jul 6
Typical cold week, starting	Jul 20
Heating degree-days (Base 18°C) (Degree days)	1733

Note: If a [stat](#) is not available you should enter the information on the [Statistics tab](#) by hand in the format shown above. The [stat](#) file is automatically generated by the [Hourly weather data translation](#) process.

When you click on the OK button you will be able to select this new Hourly weather data from:

1. [Location tab](#) at site level.
2. Add [New site](#) dialog.

Add Hourly Weather Data To The Model

The above process creates a *library* Hourly weather data component. If you have a dsb file open, you can follow an equivalent process to create a *model* Hourly weather data component. In this case the process is:

1. Copy the **EnergyPlus format** hourly weather data [epw](#) file to the [Weather data folder](#) (File > Folders > Weather data). If you have it, copy the .stat file there too.
2. Go to site level, Location tab, open the Simulation Weather Data header and click on the currently selected **Hourly weather data** selection.
3. In Learning mode, select the country of the location (or a location within the country) in the Info, data panel on the right of the screen.
4. Add a new Hourly weather data set by clicking on the green + icon in the Toolbar.
5. Select the epw file and enter any extra information in the dialog as described above.
6. Click OK to save the new Hourly weather data.

In this case you will need to export the hourly weather data you have just created from this model and reload the [ddf](#) file to the library if you wish to use the weather data in other models.

Note: When you export your own hourly weather data through ddf files the hourly weather data component will be exported but not the epw file itself.

If you don't have the data in epw format yet then it can be translated from a range of formats using the inbuilt [Weather Data Analysis](#) tool or, if the format is not supported, you can use the EnergyPlus translation tool that supports all weather formats. You can download this from the EnergyPlus website. The documentation can be found in the [Auxiliary Programs](#) document. It can be worth contacting EnergyPlus support about obtaining weather data for particular locations as they have a lot of contacts and sometimes can provide data that is not on the main website.

Weather file requirements

While EnergyPlus is quite flexible in the format of the data used in simulations, DesignBuilder requires:

- Each line to represent one hour - shorter time intervals are not supported.
- Each file must have exactly 8760 records, one for each hour of the year, starting on Jan 1 and ending Dec 31. The years do not have to be consecutive but should ideally be [set to 2002](#).

Simulating periods longer than a year

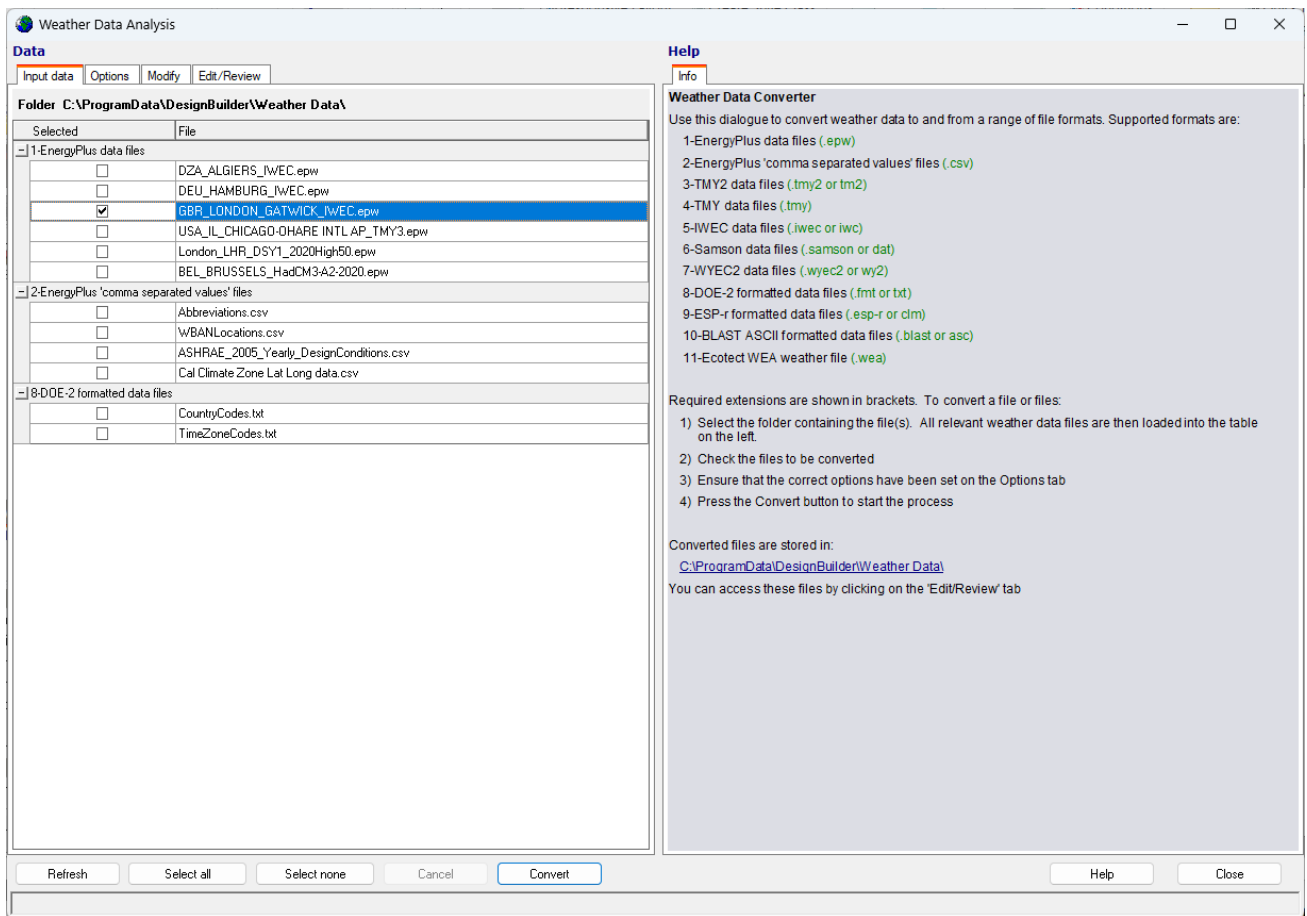
In some cases you may need to simulate a period longer than a year, possibly using custom weather data. Because DesignBuilder simulations can run for a maximum of a year, you would need to run 2 separate simulations to achieve this. Each simulation would use the appropriate weather file. For example to simulate the period September 2008 to December 2009 you could create 2 weather files, one for 2008 and the other for 2009 and run simulations for the 2 years separately. Results for the 2 simulations could be combined afterwards outside DesignBuilder (e.g. in a spreadsheet).

Weather Data Analysis

Accessed from the [Tools](#) menu

The **Weather Data Analysis** dialog can be used to convert hourly weather data to and from a range of [file formats](#). Behind the scenes, it uses the [EnergyPlus Weather Converter](#). It can be used for a variety of reasons, including:

- Convert EnergyPlus epw data to be compatible with DesignBuilder day of week results for daily, hourly and sub-hourly output ([set the year to 2002](#)).
- Translate weather data in other formats to the epw format so that it can be used in DesignBuilder.
- Generating modified weather data sets to represent extreme design cases such as climate change scenarios.
- Create additional report files from source .epw files including .stat (statistics), .ddy (design data) and .audit files.
- Correcting any invalid data points that may fall outside the typical range.



Tip: The Climate Analytics help includes a page which describes various ways to [view and edit EnergyPlus weather files](#).

In particular, the Rocky Mountain Institute provides a free .epw hourly weather data editing and translation tool called **Elements**. Users may find this to be a more convenient way to edit epw files than the processor built into DesignBuilder. It provides a spreadsheet environment and allows individual values to be edited or ranges of data can be pasted in from an external spreadsheet program. For more information see the [Elements downloads page](#).

To edit/convert a file or files:

1. On the **Input data** tab, select the folder containing the file(s) using the browse box at the top of the left-hand side of the dialog. All relevant weather data files are then loaded into the grid.
2. Check the check box next to each file that is to be converted.
3. Ensure that the correct options have been set on the Options tab.
4. Press the Convert button to start the process.

Converted files are stored in the [Weather data folder](#). You can access these files by clicking on the [Edit/Review](#) tab.

Input Data

On Input data tab you can select the source file (s) to be converted by checking the check boxes.

Tip: Use the **Select all** and **Select none** buttons at the bottom of the dialog to make quick bulk selections/deselections.

Click on the **Convert** button to start the conversion.

Options

On the Options tab you can select various options to be used during the conversion.

Output file format

Supported formats are:

- EnergyPlus data files (.epw)
- EnergyPlus 'comma separated values' files (.csv)
- TMY2 data files (.tmy2 or tm2)
- TMY data files (.tmy)
- IWECC data files (.iwecc or iwc)
- Samson data files (.samson or dat)
- WYEC2 data files (.wyec2 or wy2)
- DOE-2 formatted data files (.fmt or txt)
- ESP-r formatted data files (.esp-r or clm)
- BLAST ASCII formatted data files (.blast or asc)
- Custom format defined by .def file

Required extensions are shown in brackets.

Error Correction

Fix out of range data

You can automatically correct any values that are outside the typical range for the EnergyPlus converter by checking the **Fix out of range data** checkbox.

The [epw file format](#) includes minimum and maximum values for several of the fields. The weather converter program can ignore these (and just report them) or it can try to fix them with appropriate values. If this option is checked, then the processor will try to fix the data; if it is left unchecked, the processor will not fix the data though it will report any out of range data that it finds.

Error reporting

Depending on the quality control and accuracy of the weather data, hour to hour changes in some data values may make the data suspect for that time period. The Error reporting selection provides some control over the error reporting of any such outliers in the source data.

Note: this data is not “fixed”, merely reported by the program in the [audit report file](#).

Available options are:

- Use calculated trigger value (as primary trigger)
- Use 5°C
- Use 10°C
- Use 15°C
- Ignore calculated value

Although only one “trigger” value can be selected, consecutive values of dry-bulb temperature, dew-point temperature and wind speed are reported using appropriate calculated values. Both dew-point and wind speed use a calculated value based on mean of their values over the entire data period and standard deviation from that mean, heuristically derived.

An excerpt from the audit file illustrates:

Average Delta DB Change = 0.78°C ; Std Dev = 0.83°C
 Average Delta DP Change = 0.68°C ; Std Dev = 0.78°C
 Average Delta Relative Humidity Change = 4.02% ; Std Dev = 4.22%
 Average Delta Wind Speed Change = 0.91m/s ; Std Dev = 0.88m/s
 Hourly Dry Bulb temperature change trigger = minimum of 12.13°C and 10.°C
 12.13°C = calculated trigger based on mean change in dry-bulb temperature and standard deviation shown above
 10.°C = trigger set by user

Here the calculated DB trigger would be 12.13°C, but the user-chosen trigger is 10°C so hour by hour changes in dry bulb temperature greater than or equal to 10°C will be reported.

If the **Ignore calculated value** option is selected then the user-defined trigger value will be used directly for dry bulb temperature.

For example you might see a report such as:

Delta DB/DP Range Triggered for Apr 30
 Change in DB = 11.60°C, Hour = 14

Such detail allows the user to hand edit the incoming data, if desired.

Change filename

You can apply the standard convention used by DesignBuilder to rename the output file. If you select this option the output file will be named using the Country_Location_Source convention.

Modify

You can make some changes in the weather data as follows.

Fix year

You can fix the year to 2002. This is a recommended step for cases where you would like to view daily, hourly or sub-hourly EnergyPlus results within DesignBuilder or within the Results Viewer. The reasons for this are explained on the [Add new hourly weather data](#) page.

Modify climate

You can set the number of heating and/or cooling degree days or add a fixed delta to the dry-bulb outdoor air temperature values in the weather data. If you select the degree day option DesignBuilder adds or subtracts the same temperature difference to/from all dry-bulb outdoor air temperature values in the weather data so as to give the requested heating and cooling degree day values based on the supplied base temperature.

Technical Details

The algorithm used is as follows:

For Each Hour in the weather file:
 If Tdb < TDDBaseHeat Then
 Tdb = Tdb - HeatAdj
 Else If Tdb > TDDBaseCool Then
 Tdb = Tdb - CoolAdj

where:

Tdb is the Outdoor air dry bulb temperature

TDDBaseHeat is Heating base degree day temperature

TDDBaseCool is Cooling base degree day temperature

HeatAdj is the delta T required to be subtracted to give the correct overall number of heating degree days

CoolAdj is the delta T required to be subtracted to give the correct overall number of cooling degree days

$HeatAdj = 24 * (HeatDD - OldHeatDD) / HeatDDCount$

$CoolAdj = 24 * (CoolDD - OldCoolDD) / CoolDDCount$

HeatDDCount is the number of hours in the year where $Tdb < TDDBaseHeat$
HeatDD is the user defined required heating degree days
OldHeatDD is the number of heating degree days in the source data

CoolDDCount is the number of hours in the year where $Tdb > TDDBaseCool$
CoolDD is the user defined required cooling degree days
OldCoolDD is the number of cooling degree days in the source data

Edit/Review

You can view all files in the current hourly weather data folder on this tab. You can open any of the files for viewing/editing by clicking on the **Open selected file** link on the info panel or by double-clicking on it.

Converting From Other Formats

If you have hourly weather data in a format that is not supported by the DesignBuilder weather data translator there are 2 ways this can be used to generate an equivalent weather file in epw format.

1. Use the custom option where the format of the source data is defined in a .def file located in the same folder with the same filename stem. For example, if the input file is called London.dat then the .def file should be called London.def and stored in the same folder as London.dat. See EnergyPlus help for detailed descriptions on the [.def file format](#).
2. The second option is to use an existing epw file as a template and copy columns of data across from your weather data source file using the csv spreadsheet format.

The rest of this section provides details on the second of these methods.

To translate weather data from any format follow these instructions. Note that as a starting point you will need to have the main data required by EnergyPlus stored in a csv format file or spreadsheet. The data required is:

- Air temperature
- Dew point temperature
- Relative Humidity
- Atmospheric pressure
- Global horizontal solar radiation
- Diffuse Horizontal solar radiation
- Direct normal radiation
- Wind Speed
- Wind Direction
- Cloud cover

If any of the above parameters are missing and you are unsure about how to derive them, you may wish to consider using the EnergyPlus custom weather translator instead which has built-functions to derive some missing parameters based on other equivalent data. e.g. Direct normal radiation can be derived based on known solar position and global and diffuse horizontal radiation.

If you do have all the required data in columns of a csv file then you are ready. The process is as follows:

1. To start the process, generate a template for a similar location in csv format. To do this use the weather data translation tool, selecting the 'EnergyPlus 'comma separated values' files' output option on the Options tab.
2. Load the csv file created in step 1 into a spreadsheet to use as the template.
3. In a separate spreadsheet load the new data.
4. Copy and paste the data column by column from the new data spreadsheet into the equivalent columns in template spreadsheet. A 'missing value indicator' value can be entered in any of the places EnergyPlus doesn't use the data. The 'missing value indicator' value is different for each field and details can be found in the [Auxiliary Programs Guide](#).
5. Check that the header data at the top of the template spreadsheet is correct.
6. Ensure the year is set to 2002 in all rows.
7. Save the template as a csv file.

8. Use the weather file translator to translate the template csv file to epw format.
9. Rename the epw file as required.
10. Check that the data has been copied correctly by loading the file into DesignBuilder and clicking on the Simulation tab when at site level. You'll need to set up a new Hourly weather data component referencing the new epw file and select this first.
11. Check carefully that data output from simulations that use the new weather file makes sense. Pay particular attention to solar radiation. To do this you could check hourly incident solar radiation on a few surfaces in a model.

Note: If your regional settings aren't like those for UK/US, you could run into a couple of problems, such as the spreadsheet program changing the DataPeriods line from 1/1, 12/31 to 01-Jan -- you'd need to hand edit that before step 7

Note: Specific year data is not usually deemed appropriate for Energy Analysis. See this document for more information on selecting weather data: Drury B. Crawley. 1998. [Which Weather Data Should You Use for Energy Simulations of Commercial Buildings?](#) in ASHRAE Transactions, pp. 498-515, Vol. 104, Pt. 2. Atlanta: ASHRAE.

Hourly Weather Data Report Files

When using the [Weather Data Analysis](#) tool (aka Weather Converter) to create a new .epw or .csv file, 3 report files are generated:

- [Audit / Log File \(.audit\)](#) - shows details of the processing (including any errors) as well as the statistical report.
- [Statistical Report File \(.stat\)](#) - a general overview of the weather data in the file. This data is used by DesignBuilder to populate the [Statistics tab](#) of the Hourly weather data dialog to help identify design and typical weeks from the [Simulation Calculation options dialog](#).
- [Design Day Calculations Output \(.ddy\)](#) - a set of IDF data objects that can be used to define the site location, daylight saving periods and various types of design day conditions.

The Weather Converter checks the ASHRAE design condition files to see if a location with a corresponding WMO (World Meteorological Organization) Id can be found, and if so, the relevant ASHRAE design data is included in the reports.

Audit / Log File (.audit)

As an example, the initial portion of an audit file is shown (illustrating the error reporting) under the expanding header below.

Example Audit File Header

```
-Input File Type=EPW, with FileName=C:\Users\AndyT1\AppData\Local\Temp\WeatherDataTempFile 2.epw
-Out of Range Data items will be corrected.
- Start Date/End Date for Weather Source
Start Date=Jan 1; End Date=Dec 31
- Actual Data Years for Monthly Data**
Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
2002 2002 2002 2002 2002 2002 2002 2002 2002 2002 2002 2002
- ** Not all weather data sources represent contiguous years.
- ** Monthly data values may come from different years.
- Days per Month for Monthly Data**
Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
31 28 31 30 31 30 31 31 30 31 30 31
- ** Number of days per month seems okay.
- Data Sources should be checked for relevancy to these statistics.
Average Delta DB Change= 0.86°C ; Std Dev= 0.87°C
Average Delta DP Change= 0.62°C ; Std Dev= 0.73°C
Average Delta Relative Humidity Change= 3.86% ; Std Dev= 3.95%
Average Delta Wind Speed Change= 0.96m/s ; Std Dev= 0.90m/s
```

Hourly Dry Bulb temperature change trigger = minimum of 12.97°C and 10.°C
 12.97°C = calculated trigger based on mean change in dry-bulb temperature and standard deviation shown above
 10.°C = trigger set by user
 Changes indicated in delta temperatures, details follow.

 - Delta DB/DP Range Triggered for Feb 23
 - Change in DP=-10.60°C, Hour= 8

 - Delta DB/DP Range Triggered for May 5
 - Change in DB= 11.10°C, Hour=19
 -Output File Type=epw, with FileName=C:\ProgramData\DesignBuilder\Weather Data\USA_IL_CHICAGO-OHARE INTL AP_TMY3.epw

Statistical Report File (.stat)

The .stat file provides a useful summary of the weather data, including:

- A range of Design Conditions from "Climate Design Data 2013 ASHRAE Handbook
- Average Hourly Statistics for each month for key data.
- Monthly Calculated "undisturbed" Ground Temperatures.
- Heating/Cooling Degree Days/Hours.
- Köppen and ASHRAE Standard 196-2006 Climate Zone classifications, used to characterise the climate based on the source data file's contents.
- Typical/Extreme Periods.

In the "reporting" sections of the file, each line contains tab-delimited values , allowing you to easily copy/paste the data into a spreadsheet for further analysis.

Statistical Report File Header

```
Statistics for USA_IL_CHICAGO-OHARE INTL AP_TMY3
Location -- Chicago-OHare Intl AP IL USA
{N 41° 58'} {W 87° 55'} {GMT -6.0 Hours}
Elevation -- 201m above sea level
Standard Pressure at Elevation -- 98934Pa
Data Source -- TMY3
WMO Station 725300
- Using Design Conditions from "Climate Design Data 2005 ASHRAE Handbook"
- If the design condition source is ASHRAE, the design conditions are carefully generated
- from a period of record (typically 30 years) to be representative of that location and
- be suitable for use in heating/cooling load calculations. If the source is not ASHRAE,
- please consult the referenced source for the reasoning behind the data.
Design Stat Coldest month HDB 99.6% HDB 99% Hm-DP 99.6% Hm-HR 99.6% Hm-MCDB 99.6% Hm-DP 99% Hm-HR
99% Hm-MCDB 99% CMn-WS .4% CMn-MCDB .4% CMn-WS 1% CMn-MCDB 1% MCWS-99.6%DB PCWD-99.6%DB
Units {} {°C} {°C} {°C} {} {°C} {°C} {} {°C} {m/s} {°C} {m/s} {°C} {m/s} {deg}
Heating 1 -20.6 -17.3 -26.5 0.3 -19.7 -23.2 0.5 -16.8 12.8 -4.7 11.8 -4.1 4.9 270
Design Stat Hottest month HMn-DB Range CDB .4% CMCWB .4% CDB 1% CMCWB 1% CDB 2% CMCWB 2% ECWB .4%
EMCDB .4% ECWB 1% EMCDB 1% ECWB 2% EMCDB 2% MCWS-.4%DB PCWD-.4%DB D DP .4% D HR .4% D MCDB .4% D
DP 1% D HR 1% D MCDB 1% D DP 2% D HR 2% D MCDB 2% Ent .4% MCDB .4% Ent 1% MCDB 1% Ent 2% MCDB 2%
Units {} {°C} {°C} {°C} {} {°C} {°C} {} {°C} {°C} {°C} {} {°C} {°C} {} {°C} {°C} {} {°C} {m/s} {deg} {°C} {}
{°C} {°C} {} {°C} {°C} {} {°C} {kJ/kg} {°C} {kJ/kg} {°C} {kJ/kg} {°C}
Cooling 7 10.7 33.2 23.8 31.5 23 29.9 22.1 25.5 31.2 24.5 29.6 23.5 28.1 5.3 230 23.8 19 29 22.8
17.9 27.9 21.9 16.9 26.7 79 31.2 74.9 29.6 70.8 28.1
Design Stat X WS 1% X WS 2.5% X WS 5% X Max WB X Max DB X Min DB Sdev Max DB Sdev Min DB
Units {m/s} {m/s} {m/s} {°C} {°C} {°C} {°C} {°C}
Extremes 11.2 9.9 8.7 28.5 35.7 -24.2 1.7 4.2
```

If the source hourly weather file WMO station Id is not found in the ASHRAE design conditions database then the design statistics must be calculated from the source hourly data.

Header When the Weather Converter Must Calculate the Design Statistics

```
Statistics for UK_ LONDON HEATHROW AIRPORT _
Location -- London Heathrow Airport - UK
{N 51° 28'} {W 0° 26'} {GMT +0.0 Hours}
Elevation -- 25m above sea level
Standard Pressure at Elevation -- 101025Pa
Data Source -- CIBSE/Met Office
```

WMO Station *unknown*

- Using Design Conditions calculated from this weather file.
- The following design temperature statistics are calculated based on THIS weather file ONLY
- and may not be representative of a long-term period of record normally used for
- design temperatures. Also, note that dew point temperatures are listed where
- wet-bulb temperatures are normally presented.

Design Stat Coldest Month HDB 99.6% HDB 99%

Units {} {C} {C}

Heating 11 -2.7 -1.4

Design Stat Hottest Month CDB .4% CDB 1% CDB 2% CDP .4% CDP 1% CDP 2%

Units {} {C} {C} {C} {C} {C} {C} {C}

Cooling 7 32.1 30.7 29.3 19.4 18.7 17.9

Design Stat Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec

Units {m/s} {m/s} {m/s} {m/s} {m/s} {m/s} {m/s} {m/s} {m/s} {m/s} {m/s} {m/s}

Max WS 8.7 8.2 9.8 10.3 7.7 7.2 8.2 9.8 7.7 11.8 12.3 12.9

These are followed by groupings of Monthly temperature data.

Monthly Data

- Monthly Statistics for Dry Bulb temperatures °C

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec

Maximum 12.2 14.4 21.1 31.1 30.6 33.3 35.0 31.7 31.7 26.1 21.7 11.1

Day:Hour 21:15 23:10 27:15 15:15 5:13 8:13 19:15 3:14 27:16 21:14 2:13 7:15

Minimum -22.8 -21.7 -7.8 -3.9 1.7 6.7 11.7 11.7 6.7 -0.6 -16.1 -18.3

Day:Hour 7:07 6:05 18:03 4:24 9:05 25:04 3:04 7:01 25:06 24:03 26:07 20:07

Daily Avg -4.6 -2.5 3.8 10.0 15.3 21.1 24.1 21.8 18.1 11.0 4.7 -3.7

- Maximum Dry Bulb temperature of 35.0°C on Jul 19

- Minimum Dry Bulb temperature of -22.8°C on Jan 7

- Average Hourly Statistics for Dry Bulb temperatures °C

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec

0:01- 1:00 -6.3 -3.7 2.0 7.8 10.7 17.6 20.7 18.6 14.9 8.5 3.7 -4.9

1:01- 2:00 -6.2 -3.9 1.8 7.6 10.3 17.3 20.3 18.3 14.5 8.2 3.5 -4.8

2:01- 3:00 -6.1 -4.1 1.8 7.6 10.2 17.2 19.9 18.2 14.1 8.0 3.4 -4.9

3:01- 4:00 -6.3 -4.5 1.2 7.2 10.1 16.9 19.7 17.7 13.9 7.8 3.4 -5.2

4:01- 5:00 -6.5 -4.9 1.1 7.0 9.9 16.7 19.5 17.7 13.7 7.8 3.1 -5.3

5:01- 6:00 -6.8 -5.1 0.9 7.0 11.2 17.6 20.7 18.0 13.6 7.5 2.8 -5.6

6:01- 7:00 -6.8 -5.3 1.4 8.1 13.6 19.6 22.4 20.1 15.5 8.1 2.8 -5.7

7:01- 8:00 -6.6 -4.6 2.8 9.3 15.5 21.3 23.7 21.7 17.6 9.9 3.5 -5.4

8:01- 9:00 -5.6 -3.3 4.0 10.2 17.3 22.5 25.0 23.3 19.4 11.8 4.3 -4.5

9:01-10:00 -4.4 -2.1 5.1 10.9 18.7 23.6 26.1 24.6 20.5 13.2 5.5 -3.2

10:01-11:00 -3.5 -1.0 5.9 11.8 19.8 24.4 27.0 25.2 21.4 14.3 6.3 -2.1

11:01-12:00 -2.8 -0.2 6.8 12.3 20.6 24.8 27.8 25.4 22.2 14.9 7.0 -1.5

12:01-13:00 -2.0 0.3 7.2 12.8 21.1 25.2 28.4 25.9 22.8 15.2 7.2 -0.9

13:01-14:00 -1.7 0.4 7.3 13.1 20.9 25.5 28.6 25.7 23.2 15.4 7.4 -0.7

14:01-15:00 -1.6 0.5 7.2 13.0 20.6 25.4 28.5 25.4 22.9 15.3 7.3 -0.7

15:01-16:00 -1.9 0.3 6.8 12.8 20.0 25.1 28.0 25.2 22.5 14.6 6.7 -1.5

16:01-17:00 -2.9 -0.5 6.1 12.4 19.5 24.8 27.3 24.5 21.6 13.3 5.9 -2.5

17:01-18:00 -3.8 -1.4 5.0 11.5 18.1 24.0 26.5 23.8 20.0 11.8 5.3 -3.3

18:01-19:00 -4.5 -2.3 3.9 10.6 16.1 22.3 25.4 22.4 18.6 11.0 4.6 -3.7

19:01-20:00 -4.5 -2.6 3.3 9.9 14.5 20.7 24.0 21.5 17.8 10.4 4.2 -4.0

20:01-21:00 -4.7 -2.9 2.9 9.5 13.3 19.4 23.2 20.6 16.9 9.8 4.2 -4.2

21:01-22:00 -5.0 -2.9 2.6 9.2 12.5 18.8 22.6 20.3 16.5 9.3 4.1 -4.4

22:01-23:00 -5.2 -3.1 2.4 8.9 11.8 18.1 22.0 19.6 15.9 8.7 3.8 -4.5

23:01-24:00 -5.9 -3.4 2.1 8.4 11.4 17.9 21.6 19.0 15.3 8.5 3.7 -5.0

Max Hour 15 15 14 14 13 14 14 13 14 14 14 15

Min Hour 6 7 6 5 5 5 5 6 6 7 7

- Monthly Statistics for Extreme temperatures °C

#Days Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec

Max >= 32 3 8

Max <= 0 14 14 1 4 15

Min <= 0 29 22 20 5 3 13 29

Min <=-18 5 3 2

- Monthly Statistics for Dew Point temperatures °C

Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec

Maximum 6.1 11.1 13.9 18.3 20.6 23.3 25.0 25.6 20.0 13.9 17.2 6.7

Day:Hour 21:17 23:15 28:03 18:09 30:09 8:09 19:17 4:21 6:17 2:19 3:19 1:08

Minimum -28.3 -27.8 -13.3 -8.3 -6.1 -1.1 7.8 7.2 -0.6 -2.8 -20.6 -21.7

Day:Hour 27:06 7:07 5:19 4:09 6:20 25:03 3:11 17:04 30:15 14:19 26:07 30:01

Daily Avg -9.4 -8.2 -1.5 4.0 7.2 13.0 18.7 16.6 12.9 5.0 0.4 -7.6

- Maximum Dew Point temperature of 25.6°C on Aug 4
 - Minimum Dew Point temperature of -28.3°C on Jan 27
 - Monthly Statistics for Relative Humidity %
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 Maximum 100 92 100 100 100 100 100 100 100 100 93 100 100
 Day:Hour 2:18 18:08 13:10 1:16 1:24 8:22 1:17 19:24 9:04 1:24 7:01 1:02
 Minimum 35 31 30 24 17 25 36 37 28 32 39 47
 Day:Hour 27:14 16:16 21:14 30:13 4:17 3:14 7:12 30:14 27:16 13:13 26:13 11:13
 Daily Avg 71 66 71 69 63 62 74 74 75 69 75 75
 - Average Hourly Relative Humidity %
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 0:01- 1:00 78 69 76 77 76 70 87 85 90 77 77 79
 1:01- 2:00 78 69 76 78 77 72 89 87 91 79 77 79
 2:01- 3:00 77 70 76 78 79 72 90 87 92 79 79 80
 3:01- 4:00 77 70 77 79 80 74 91 87 92 80 79 81
 4:01- 5:00 78 71 78 80 81 76 91 88 92 80 81 81
 5:01- 6:00 77 70 81 79 81 76 88 86 92 80 82 82
 6:01- 7:00 76 71 80 75 72 72 82 80 86 81 83 82
 7:01- 8:00 74 72 76 71 64 68 77 75 76 76 80 81
 8:01- 9:00 71 71 72 67 58 65 72 70 69 68 78 78
 9:01-10:00 67 70 68 64 54 61 67 64 64 62 73 74
 10:01-11:00 64 65 67 61 50 57 63 62 61 57 70 69
 11:01-12:00 63 61 63 61 48 55 60 61 58 54 68 68
 12:01-13:00 61 60 61 59 47 54 58 59 55 54 66 65
 13:01-14:00 60 58 60 59 46 53 57 60 55 53 66 65
 14:01-15:00 60 58 60 60 47 52 56 60 55 53 66 64
 15:01-16:00 61 58 61 61 48 51 58 61 56 54 68 66
 16:01-17:00 65 59 62 62 49 50 61 63 59 59 72 70
 17:01-18:00 68 61 66 63 53 51 63 66 65 64 75 74
 18:01-19:00 72 65 70 66 57 54 66 71 71 68 76 76
 19:01-20:00 72 67 73 70 62 57 71 76 75 70 78 77
 20:01-21:00 73 69 73 72 66 60 76 80 80 72 78 76
 21:01-22:00 75 70 74 73 71 63 78 81 81 73 75 77
 22:01-23:00 75 70 75 74 74 66 82 83 85 76 77 77
 23:01-24:00 77 69 76 76 75 68 84 84 87 77 77 77
 Max Hour 5 8 6 5 5 6 5 5 5 7 7 6
 Min Hour 14 16 14 13 14 17 15 13 14 15 15 15
 - Monthly Indicators for Precipitation/Moisture (kPa)
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 0.3 0.4 0.6 0.9 1.1 1.7 2.2 2.0 1.5 0.9 0.8 0.4
 - Monthly Statistics for Wind Chill/Heat Index temperatures °C **
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 Minimum WC -43 -30 -23 -11 -2 3 5 -5 -32 -33
 Day:Hour 27:08 5:12 5:05 1:01 8:06 25:04 25:04 16:06 25:23 20:19
 Average WC -13 -11 -5 -1 5 6 8 1 -8 -11
 Avg Del WC 8 8 8 6 3 1 0 5 8 7
 # Hours WC 716 634 635 386 124 7 23 294 460 703
 Maximum HI 31 31 39 40 38 30
 Day:Hour 18:13 30:16 8:13 19:17 4:15 6:11
 Average HI 29 29 30 33 31 28
 Avg Del HI 0 1 1 3 2 0
 # Hours HI 15 9 76 201 95 16
 - **WindChill/HeatIndex Temps -- statistics...only those different from Air Temps
 - Monthly Wind Direction % {N=0 or 360,E=90,S=180,W=270}
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 North 11 10 7 14 15 14 15 12 15 9 11 10
 NorthEast 1 4 16 12 23 13 12 14 6 4 11 1
 East 2 0 8 10 14 6 5 5 4 2 1 0
 SouthEast 5 0 7 5 6 9 7 6 7 2 8 4
 South 17 15 13 18 12 19 18 12 15 25 23 15
 SouthWest 17 19 15 17 3 19 18 19 13 22 8 19
 West 23 32 21 11 11 11 16 19 24 19 23 23
 NorthWest 24 19 14 13 15 7 10 13 16 16 15 28
 - Monthly Statistics for Wind Speed m/s
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 Maximum 12.9 12.4 15.4 12.9 12.9 11.3 11.8 9.8 8.8 10.3 12.4 12.9
 Day:Hour 21:19 24:17 4:23 24:18 13:13 20:17 4:13 6:11 17:15 16:01 20:12 20:21
 Minimum 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
 Day:Hour 1:17 7:06 4:09 1:08 3:20 1:18 7:04 1:23 5:05 1:14 8:02 2:24

Daily Avg 4.9 5.1 5.5 4.8 3.7 4.9 4.2 3.8 3.4 4.8 5.3 4.3
 - Maximum Wind Speed of 15.4 m/s on Mar 4
 - Minimum Wind Speed of 0.0 m/s on Jan 1
 - Monthly Statistics for Liquid Precipitation mm
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 Total 0 0 0 0 0 0 0 0 0 0 0 0
 - Monthly Statistics for Albedo
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 Average 0.100 0.100 0.100 0.100 0.100 0.100 0.100 0.100 0.100 0.100 0.100 0.100
 - Monthly Statistics for Solar Radiation (Direct Normal, Diffuse, Global Horizontal) Wh/m²
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 Direct Avg 2335 2737 3146 3668 5014 5205 4829 4194 4004 3268 1880 2211
 Direct Max 6874 7472 8929 8384 10148 9706 8814 9321 7925 7690 5672 6563
 Day 27 15 15 23 3 24 5 18 4 27 14 5
 Diffuse Avg 956 1330 1740 2057 2539 2690 2800 2422 1861 1394 1089 799
 Global Avg 1764 2493 3440 4394 5976 6294 6177 5161 4193 2941 1819 1504
 - Maximum Direct Normal Solar of 10148 Wh/m² on May 3
 - Average Hourly Statistics for Direct Normal Solar Radiation Wh/m²
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 0:01- 1:00 0 0 0 0 0 0 0 0 0 0 0 0
 1:01- 2:00 0 0 0 0 0 0 0 0 0 0 0 0
 2:01- 3:00 0 0 0 0 0 0 0 0 0 0 0 0
 3:01- 4:00 0 0 0 0 0 0 0 0 0 0 0 0
 4:01- 5:00 0 0 0 0 6 23 5 0 0 0 0 0
 5:01- 6:00 0 0 0 15 132 192 108 59 15 1 0 0
 6:01- 7:00 0 3 63 196 283 325 273 227 187 74 4 0
 7:01- 8:00 38 81 220 253 377 411 322 321 316 241 80 65
 8:01- 9:00 182 188 284 317 432 477 387 360 411 384 211 230
 9:01-10:00 268 309 335 349 477 465 412 427 442 408 230 286
 10:01-11:00 304 370 384 356 475 494 425 408 440 408 277 327
 11:01-12:00 317 343 362 376 475 445 466 415 389 357 285 313
 12:01-13:00 333 371 381 346 492 433 503 388 412 376 266 307
 13:01-14:00 341 313 363 404 470 428 439 355 396 384 263 276
 14:01-15:00 291 307 292 377 412 422 430 338 366 299 164 245
 15:01-16:00 206 289 237 295 363 397 381 370 331 237 87 146
 16:01-17:00 55 152 176 222 327 307 342 282 223 96 12 15
 17:01-18:00 0 11 51 154 224 262 245 191 75 3 0 0
 18:01-19:00 0 0 0 9 70 119 89 52 3 0 0 0
 19:01-20:00 0 0 0 0 6 3 0 0 0 0 0 0
 20:01-21:00 0 0 0 0 0 0 0 0 0 0 0 0
 21:01-22:00 0 0 0 0 0 0 0 0 0 0 0 0
 22:01-23:00 0 0 0 0 0 0 0 0 0 0 0 0
 23:01-24:00 0 0 0 0 0 0 0 0 0 0 0 0
 Max Hour* 14 13 11* 14 13 11* 13 10* 10* 10* 12 11*
 Min Hour 1 1 1 1 1 1 1 1 1 1 1 1
 - Monthly Calculated "undisturbed" Ground Temperatures** °C
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 0.5 m -1.9 -3.1 -1.0 2.2 10.7 17.2 21.6 22.9 20.7 15.6 8.8 2.6
 2.0 m 2.4 0.3 0.7 2.5 8.1 13.2 17.3 19.5 19.0 16.2 11.5 6.6
 4.0 m 5.9 3.8 3.3 4.0 7.2 10.6 13.8 16.0 16.5 15.2 12.5 9.2
 - **These ground temperatures should NOT BE USED in the GroundTemperatures object to compute building floor losses.
 - The temperatures for 0.5 m depth can be used for GroundTemperatures:Surface.
 - The temperatures for 4.0 m depth can be used for GroundTemperatures:Deep.
 - Calculations use a standard soil diffusivity of 2.3225760E-03 {m**2/day}
 - Monthly Heating/Cooling Degree Days/Hours
 Jan Feb Mar Apr May Jun Jul Aug Sep Oct Nov Dec
 HDD base 10C 454 352 197 89 3 0 0 0 0 30 199 424
 HDD base 18C 702 575 439 268 104 12 1 2 38 218 398 672
 CDD base 10C 0 2 5 87 168 333 438 365 244 60 41 0
 CDD base 18C 0 0 0 27 21 105 191 119 42 1 0 0
 CDH base 20C 0 0 1 520 703 2077 3409 2029 845 69 7 0
 CDH base 23C 0 0 0 258 302 1043 1866 869 320 10 0 0
 CDH base 27C 0 0 0 71 47 285 599 155 50 0 0 0
 - 1743 annual cooling degree-days (10°C baseline)
 - 1748 annual heating degree-days (10°C baseline)
 - 506 annual cooling degree-days (18°C baseline)
 - 3430 annual heating degree-days (18°C baseline)
 - Climate type "Dfa" (Köppen classification)**

- Humid continental (hot summer, cold winter, no dry season, lat. 30-60°N)
- Unbearably humid periods in summer, but passive cooling is possible
- **Note that the Köppen classification shown here is derived algorithmically from the source weather data.
- It may not be indicative of the long term climate for this location.
- Climate type "5A" (ASHRAE Standards 90.1-2004 and 90.2-2004 Climate Zone)**
- Cool - Humid, Probable Köppen classification=Dfa, Humid Continental (Warm Summer)
- **Note that the ASHRAE classification shown here is derived algorithmically from the source weather data.
- It may not be indicative of the long term climate for this location.
- Typical/Extreme Period Determination
- Summer is Jun:Aug
- Extreme Summer Week (nearest maximum temperature for summer)
- Extreme Hot Week Period selected: Jul 13:Jul 19, Maximum Temp= 35.00°C, Deviation=| 8.451|°C
- Typical Summer Week (nearest average temperature for summer)
- Typical Week Period selected: Aug 24:Aug 30, Average Temp= 22.35°C, Deviation=| 0.095|°C
- Winter is Dec:Feb
- Extreme Winter Week (nearest minimum temperature for winter)
- Extreme Cold Week Period selected: Jan 27:Feb 2, Minimum Temp= -22.80°C, Deviation=|12.651|°C
- Typical Winter Week (nearest average temperature for winter)
- Typical Week Period selected: Dec 22:Dec 28, Average Temp= -3.65°C, Deviation=| 0.123|°C
- Autumn is Sep:Nov
- Typical Autumn Week (nearest average temperature for autumn)
- Typical Week Period selected: Oct 27:Nov 2, Average Temp= 11.28°C, Deviation=| 0.020|°C
- Spring is Mar:May
- Typical Spring Week (nearest average temperature for spring)
- Typical Week Period selected: Apr 26:May 2, Average Temp= 9.69°C, Deviation=| 0.357|°C

Design Day Calculations Output (.ddy)

Using the WMO field (or determining it from the WBAN field), the Weather Converter performs table look up in the Design Condition files to see if there are recorded design conditions for the subject location. If the location is found, then design day objects are produced on the resultant design day object (ddy extension) file - ready for inclusion in an EnergyPlus input data file. If no design conditions are located, then the design day object file will still include a location object for inclusion with EnergyPlus. However, statistics using the weather file are displayed to the statistics file - these "can" be used to create your own design day definitions but you should read the warning that is issued and take care if your weather file is only a "single instance" weather data representation.

The location objects as well as the design condition objects are constrained by the data source. Some data sources do not have elevation information - thus, a location object from such a source will have an elevation of 0.0. Likewise, the time zone of some locations may not be available from the source data nor other data resources that the weather converter uses. A time zone will be estimated from the standard meridian of the location (determined by the longitude) but it may not be accurate. A user needs to be aware of these limitations when taking the design day files from the weather converter.

Note that you can always include a "def" file with this data to assure accuracy regardless of input format limitations.

An excerpt of a design day output are shown under the expanding header below. Actual design day objects have been deleted for brevity. Note that with the 2009 ASHRAE HOF climate conditions, a DaylightSavingPeriod object may be included.

Example Design Day Report Extract

```
! The following Location and Design Day data are produced as possible from the indicated data
! source.
! Wind Speeds follow the indicated design conditions rather than traditional values (6.7 m/s
! heating, 3.35 m/s cooling)
! No special attempts at re-creating or determining missing data parts (e.g. Wind speed or
! direction)
! are done. Therefore, you should look at the data and fill in any incorrect values as you
! desire.
```

```
Site:Location,
Chicago-OHare Intl AP_IL_USA Design_Conditions, !- Location Name
```


41.98, !- Latitude {N+ S-}
 -87.92, !- Longitude {W- E+}
 -6.00, !- Time Zone Relative to GMT {GMT+/-}
 201.00; !- Elevation {m}

! WMO=725300 Time Zone=NAC: (GMT-06:00) Central Time (US & Canada)

! Data Source=ASHRAE 2005 Annual Design Conditions

RunPeriodControl:DaylightSavingTime,
 1st Sunday in April, !- StartDate
 Last Sunday in October; !- EndDate

! Using Design Conditions from "Climate Design Data 2005 ASHRAE Handbook"

! Chicago-OHare Intl AP_IL_USA Extreme Annual Wind Speeds, 1%=11.2m/s, 2.5%=9.9m/s, 5%=8.7m/s

! Chicago-OHare Intl AP_IL_USA Extreme Annual Temperatures, Max Drybulb=35.7°C Min Drybulb=-24.2°C

! Chicago-OHare Intl AP_IL_USA Annual Heating Design Conditions Wind Speed=4.9m/s Wind Dir=270

! Coldest Month=JAN

! Chicago-OHare Intl AP_IL_USA Annual Heating 99.6%, MaxDB=-20.6°C

SizingPeriod:DesignDay,

Chicago-OHare Intl AP Ann Htg 99.6% Condns DB, !- Name

-20.6, !- Maximum Dry-Bulb Temperature {C}
 0.0, !- Daily Temp Range {C}
 -20.6, !- Humidity Indicating Conditions at Maximum Dry-Bulb
 98934., !- Barometric Pressure {Pa}
 4.9, !- Wind Speed {m/s} design conditions vs. traditional 6.71 m/s (15 mph)
 270, !- Wind Direction {Degrees; N=0, S=180}
 0.00, !- Clearness {0.0 to 1.1}
 0, !- Rain {0-no,1-yes}
 0, !- Snow on ground {0-no,1-yes}
 21, !- Day of Month
 1, !- Month

WinterDesignDay,!- Day Type

0, !- Daylight Savings Time Indicator
 WetBulb; !- Humidity Indicating Type

!! Chicago-OHare Intl AP_IL_USA Annual Heating 99%, MaxDB=-17.3°C

SizingPeriod:DesignDay,

Chicago-OHare Intl AP Ann Htg 99% Condns DB, !- Name

-17.3, !- Maximum Dry-Bulb Temperature {C}
 0.0, !- Daily Temp Range {C}
 -17.3, !- Humidity Indicating Conditions at Maximum Dry-Bulb
 98934., !- Barometric Pressure {Pa}
 4.9, !- Wind Speed {m/s} design conditions vs. traditional 6.71 m/s (15 mph)
 270, !- Wind Direction {Degrees; N=0, S=180}
 0.00, !- Clearness {0.0 to 1.1}
 0, !- Rain {0-no,1-yes}
 0, !- Snow on ground {0-no,1-yes}
 21, !- Day of Month
 1, !- Month

WinterDesignDay,!- Day Type

0, !- Daylight Savings Time Indicator
 WetBulb; !- Humidity Indicating Type

! Chicago-OHare Intl AP_IL_USA Annual Humidification 99.6% Design Conditions DP=>MCDB, DP=-26.5°C

SizingPeriod:DesignDay,

Chicago-OHare Intl AP Ann Hum_n 99.6% Condns DP=>MCDB, !- Name

-19.7, !- Maximum Dry-Bulb Temperature {C}
 0.0, !- Daily Temp Range {C}
 -26.5, !- Humidity Indicating Conditions at Maximum Dry-Bulb
 98934., !- Barometric Pressure {Pa}
 4.9, !- Wind Speed {m/s} design conditions vs. traditional 6.71 m/s (15 mph)
 270, !- Wind Direction {Degrees; N=0, S=180}
 0.00, !- Clearness {0.0 to 1.1}
 0, !- Rain {0-no,1-yes}
 0, !- Snow on ground {0-no,1-yes}
 21, !- Day of Month
 1, !- Month

WinterDesignDay,!- Day Type

0, !- Daylight Savings Time Indicator
DewPoint; !- Humidity Indicating Type

Design day "definitions" originate in the ASHRAE Handbook of Fundamentals. Prior to 1997, these conditions were described for winter and summer (heating and cooling). They were based on seasonal percentages.

EnergyPlus uses the design day object values and creates an entire day of weather data - this is described more fully in the Input Output Reference under the DesignDay object. The weather converter program assigns "SummerDesignDay" and "WinterDesignDay" day types by default - these day types influence "scheduling" of various elements. How to use these effectively is described during the DesignDay and Schedule objects discussions in the EnergyPlus Input Output Reference.

Beginning in 1997, and continuing (the latest version was published in 2009), the design condition data is based on annual percentages. In addition, only locations with long-term hourly observations data (on which to form the basis) are included.

[From ASHRAE Handbook of Fundamentals, 2009]

Design data based on dry-bulb temperature represent peak occurrences of the sensible component of ambient outdoor conditions. Design values based on wet-bulb temperature are related to the enthalpy of the outdoor air. Conditions based on dew point relate to the peaks of the humidity ratio. The designer, engineer, or other user must decide which set(s) of conditions and probability of occurrence apply to the design situation under consideration.

The 99.6% and 99% Heating conditions are often used in the sizing of heating equipment.

The 0.4, 1.0, and 2.0% dry-bulb temperatures and mean coincident wet-bulb temperatures (i.e., DB = >MWB) often represent conditions on hot, mostly sunny days. These are often used in sizing cooling equipment such as chillers or air-conditioning units.

Design conditions based on wet-bulb temperatures (i.e., WB = >MDB) represent extremes of the total sensible plus latent heat of outdoor air. This information is useful for cooling towers, evaporative coolers, and fresh air ventilation system design.

Design conditions based on dew-point temperatures (i.e., DP = >MDB) are directly related to extremes of humidity ratio, which represent peak moisture loads from the weather. Extreme dew-point conditions may occur on days with moderate dry-bulb temperatures, resulting in high relative humidity. These values are especially useful for humidity control applications, such as desiccant cooling and dehumidification, cooling-based dehumidification, and fresh-air ventilation systems. The values are also used as a check point when analyzing the behavior of cooling systems at part-load conditions, particularly when such systems are used for humidity control as a secondary function. Humidity ratio values are calculated from the corresponding dew-point temperature and the standard pressure at the location's elevation.

The enthalpy design conditions (i.e. Enthalpy = > MDB) gives the annual enthalpy for the cooling season; this is used for calculating cooling loads caused by infiltration and/or ventilation into buildings. Enthalpy represents the total heat content of air (the sum of its sensible and latent energies). Cooling loads can be easily calculated knowing the conditions of both the outdoor ambient and the building's interior air.

Sources Of EnergyPlus Hourly Weather Data

All of the hourly weather data provided with DesignBuilder is supplied by DOE from:

<https://energyplus.net/weather>

But with the year changed to 2002 as described in [Add new hourly weather data](#).

Most of these "Typical Year" weather files are composite files from multiple years. Each month has been selected to be representative of that month for the period of record. Data is representative of the normal patterns for that month and is not intended to be the average. The selection of months is usually based on a weighting of temperature, humidity, wind, and solar. There are some variations among the weighting, most are similar to the TMY2 done in the US in the 1990s.

You can find details on the [EnergyPlus file format](#) in the Climate Analytics help.

Other Sources

Other sources of hourly weather data and associated tools are listed below.

International sources

- **DesignBuilder Climate Analytics** - a comprehensive source of hourly weather data with integrated analysis, selection and morphing tools to support building modelling projects. It provides "Design Years", "Actual Years" and "Typical Years" and base data can be morphed to account for future climate scenarios, urban heat islands and various extreme design cases.
- **Climate.OneBuilding** - a widely used repository of typical year TMYx weather data covering more than 17,476 locations worldwide. It is developed and curated by Dru Crawley and Linda Lawrie, the original developers of the EPW weather file format. Data from this site can be accessed at no charge.
- **White Box Technologies** - "Typical Year" and "Actual Year" weather files created from weather reports of over 13,000 weather stations around the world. The weather files on the White Box Technologies website were developed primarily from historical weather data archived in the NOAA NCDC ISD database. The IWEC2 weather files were produced from ISD data from 1986 to 2009. Historical weather files are available for the period 2001 to 2022.

National sources

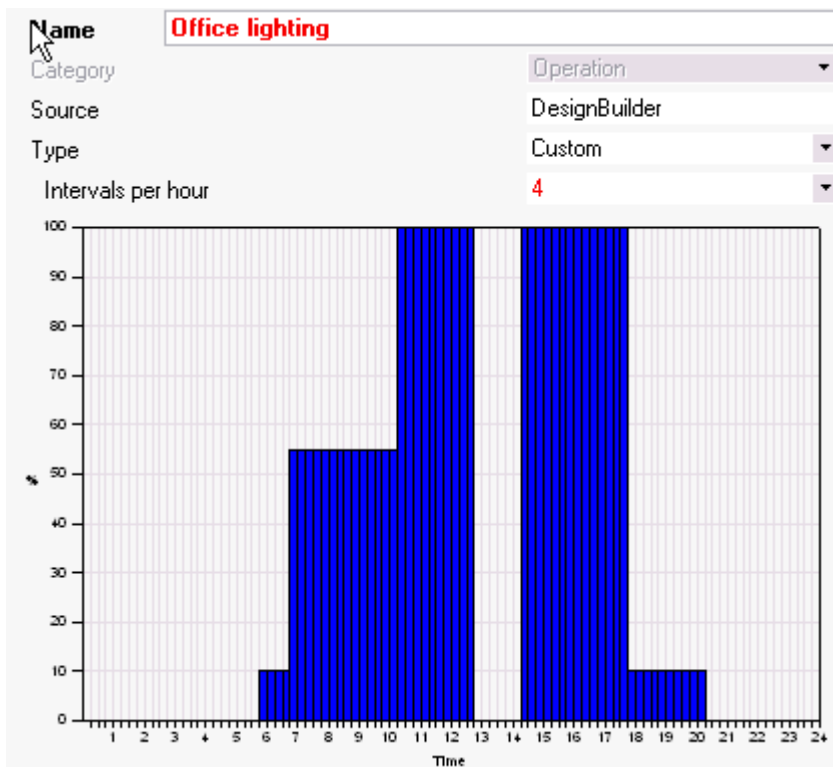
- **UK - CIBSE** TRY, DSY and future climate scenario data is available from the [DesignBuilder CIBSE Weather Data webpage](#).
- **Germany - DWD**, TRY 2010 weather data for Germany can be downloaded free of charge from the Europe page on the <https://climate.onebuilding.org> website.
- **Australia and new Zealand - NATHers**, Latest hourly weather data for Australia and New Zealand can be downloaded from NATHers at no cost. The link depends on your browser. Copy the links below into the address bar of your browser.
 - Internet Explorer: ftp://rNatHERS:MDkflWeZ@ftp.niwa.co.nz/ftp/NatHERS
 - Chrome/Firefox: ftp://rNatHERS:MDkflWeZ@ftp.niwa.co.nz
- **Portugal** - DGEg, Download a software tool which can generate hourly weather data for the various Portuguese regions and adjusted to the site height. See:
 - <https://www.dgeg.gov.pt/pt/areas-setoriais/energia/energias-renovaveis-e-sustentabilidade/sce-er/>
 - https://energyplus.net/weather-region/europe_wmo_region_6/PRT

Weather data editor/translator/viewers

- **DesignBuilder Results Viewer v4** and later allows hourly weather data in .epw files to be viewed and compared with data in other files.
- **Elements** is a free, open-source, cross-platform software tool for creating and editing custom weather files for building energy modelling.

Profiles

Profiles define the variation in occupancy, equipment usage or temperature over the course of a single day. They are used as building blocks in the definition of [Schedules](#).



There are 3 types of profile:

1. **Constant** - has the same value all day. Enter the constant value for the profile
2. **Single period** - define the 'on' period by setting the start and end times for the profile using the mouse. The operation is assumed to be 'off' for the rest of the day
3. **Custom** - the profile is defined by a number of intervals, each of which can have a different value. There are two ways to set interval values:
 - a. To change the value for a single interval (single histogram bar) select the top of the histogram bar and drag up/down using the LEFT mouse button.
 - b. To apply a constant value over a range of intervals select the top of the histogram bar which has the required value and drag left/right over the range of intervals to be set using the RIGHT mouse button.

The figure above shows an example custom profile.

Profile data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by clicking on the Edit button when selecting profiles from a list in the [Schedules dialog](#).

See also the [Timings, schedules and profiles](#) tutorial

Textures

Textures are used to represent the external surface of building [constructions](#) in the rendered views in the [Visualisation](#) Screen.

Select the texture file and the real-world width and height represented by the texture. Textures are associated with materials and used to render the model in the Visualisation Screen.

Note: If an image does not appear under the Image header this means that the selected file is not installed.

Adding your own textures

DesignBuilder comes with a library of textures, but you can also add your own custom textures. To add a texture to a model you should first get a list of textures and then click on the Add new icon. Then in the texture dialog, enter a name, the width and height represented by the texture and click on the browse button to bring up the list of texture files. In the texture file list click on the Add button to add a new texture file. Then select the corresponding texture file.

Important note: Textures in bitmap and jpeg format have been found to work best. Although DesignBuilder does allow other formats these can be unreliable.

The texture file you've just added will be stored in the dsb file itself so the texture will be available if you open the model on any other computer.

Note: When you [export](#) your own textures the texture *component* will be included in the DDF file but the texture *file* will not. So on its own, DDF export is not an ideal way to transfer texture data to another computer. To get round this you can follow these steps on the second computer:

1. Open a dsb file that contains the texture(s). This copies the custom texture files from the dsb file to the disk of the second computer.
2. Close the dsb file.
3. Import the DDF that contains the texture component(s) to any model. You should now have both the texture component and the file as required for visualisation.

Texture data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the texture icon in the model data,
- by clicking on the Edit button when selecting textures from a list.

Window Gas

Window Gas component data is used to define the thermal properties of the cavity fill gas between window [Panels](#) in [Glazing components](#).

The gas which fills the inter-glazing cavity affects the heat transfer through the assembly but has almost no effect on the solar heat gain or the visible light transmission. Air, of course, is the most common cavity fill gas, but the use of an inert gas (typically argon or krypton) can significantly reduce window heat transfer. Krypton and argon are colourless, odourless and non-toxic. Argon is the most commonly-used fill gas because it offers good thermal performance at low cost. Krypton is more effective at reducing heat loss, but is roughly 200 times more expensive than argon per unit volume. Because krypton works best at smaller pane spacings (8 mm), it is often used in triple and quadruple-glazed windows to minimize the overall thickness of the unit. Other types of gases are used (e.g. sulphur hexafluoride (SF₆) and carbon dioxide (CO₂)) to reduce sound transmission, but these gases do not offer the improved thermal performance of the inert gases.

Category

This is the gas type. Select from:

- Air
- Argon
- CO₂

- Krypton
- Other
- SF6
- Xenon

The window gas you are defining does not fit with the available options select **Other**. Select **Air** for an air gap.

Thickness

Enter the thickness of the inter-glazing cavity.

Custom material properties

The window gas properties of the gases Air, Argon, Krypton, Xenon are known by EnergyPlus and so no further data is required for these gases.

For CO2, Other and SF6 you must enter custom properties.

Custom Gases Only

If you elect to enter custom material properties then you must enter the details. The A and B coefficients are those in the following expression that gives a property value as a function of temperature in degrees K:

Property = Coefficient_A + Coefficient_B * Gas_Temperature (K)

Conductivity Coefficient A

The A coefficient for gas conductivity (W/m-K).

Conductivity Coefficient B

The B coefficient for gas conductivity (W/m-K2).

Viscosity Coefficient A

The A coefficient for gas viscosity (kg/m-s).

Viscosity Coefficient B

The B coefficient for gas viscosity (kg/m-s-K).

Specific Heat Coefficient A

The A coefficient for gas specific heat (J/kg-K).

Specific Heat Coefficient B

The B coefficient for gas specific heat (J/kg-K2).

Molecular Weight

The molecular weight (mass) of the gas (Kg/kmol).

Window gas component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by clicking on the Edit button when selecting window gases from a list.

Metabolic Rates

Metabolic rate data defines a category of human metabolic output. The metabolic rate can be selected in Model data on the Activity tab by choosing a Metabolic rate component.

The metabolic rate per person data is supplied to EnergyPlus which multiplies it by the number of people in the zone as the starting point for occupancy heat emission calculations.

Metabolic rate data can be found in the ASHRAE Handbook of Fundamentals, Chapter 8, Table 5.

DesignBuilder data is derived from this data and is for adult males having typical surface area of 1.8m². For women multiply the adult male value by 0.85 and for children multiply by 0.75.

Component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by clicking on the Edit button when selecting window gases from a list.

Background

Heat is generated in the human body by oxidation at a rate called the metabolic rate. This heat is dissipated from the body surface and respiratory tract by a combination of radiation, convection and evaporation. The relative proportions of sensible and latent heat from people is a complex function of the metabolic rate and the environmental conditions.

EnergyPlus uses a polynomial function to divide the heat gain into sensible and latent portions. More details can be found in the EnergyPlus EngineeringDoc.pdf documentation.

See also the section on metabolic activity data.

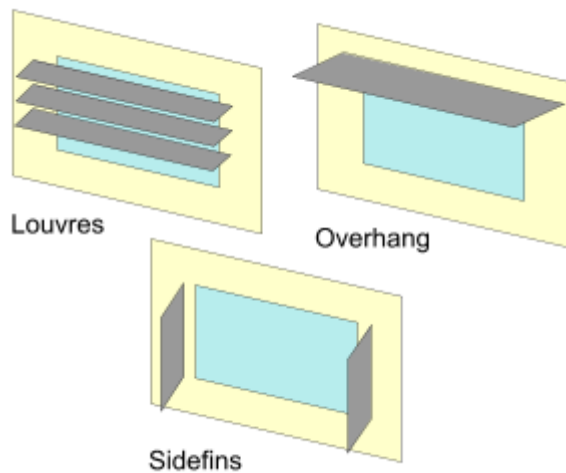
Local Shading Components

General tab on [Local Shading Dialog](#).

Local Shading Components are used to define simple generic shading devices to apply to the Windows in the model.

Enter the name and category for the local shading data set.

[Overhangs](#), [louvres](#) and [sidefins](#) can be specified either individually (e.g. just overhangs) or in combination (e.g. sidefins combined with louvres).



Local Shading component data can be selected from the [Openings tab](#) in Model Data and is applied to all windows on the surface, including custom windows.

The data applies only to external glazing - roof and internal glazing do not use local shading data.

Overhang, louvre and sidefin blades are all assumed to be 100% opaque and made of the material specified on this tab.

Note: the DesignBuilder Local shading devices allow a range of different overhangs, side fins and louvres to be defined, but if you need to model a shading device not covered by the standard mechanism, you can draw custom shading devices using [Component blocks](#).

See also the [Shading via louvres, overhangs, and side fins](#) tutorial

There are four tabs on the Local Shading dialog:

- [General](#)
- [Louvres](#)
- [Sidefins](#)
- [Overhangs](#)

Local shading data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the local shading icon on the Openings tab in the model data,
- by clicking on the Edit button when selecting local shading devices from a list.

Local Shading - General

General tab on [Local Shading Dialog](#).

Enter the Name, Category and Source for this Local shading data set.

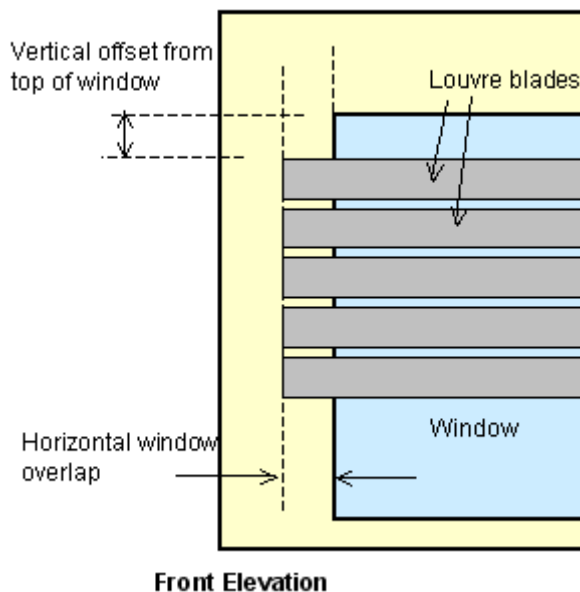
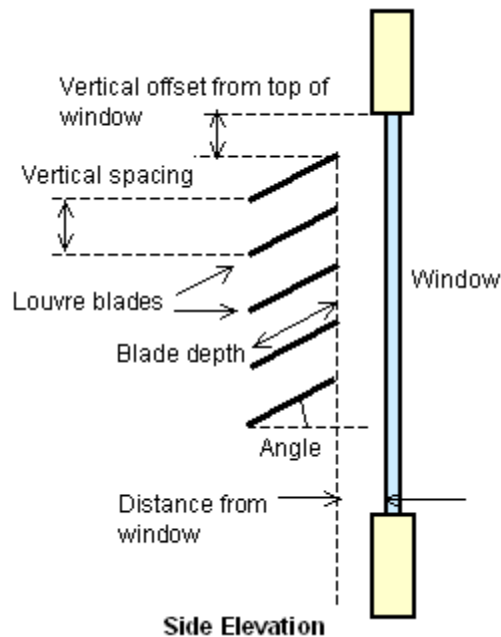
Blade material

Select the material used to manufacture this local shading device. Typical options may be steel or aluminium.

Local Shading - Louvres

Louvres tab on [Local Shading](#) Dialog.

Enter details on the louvres, if fitted.

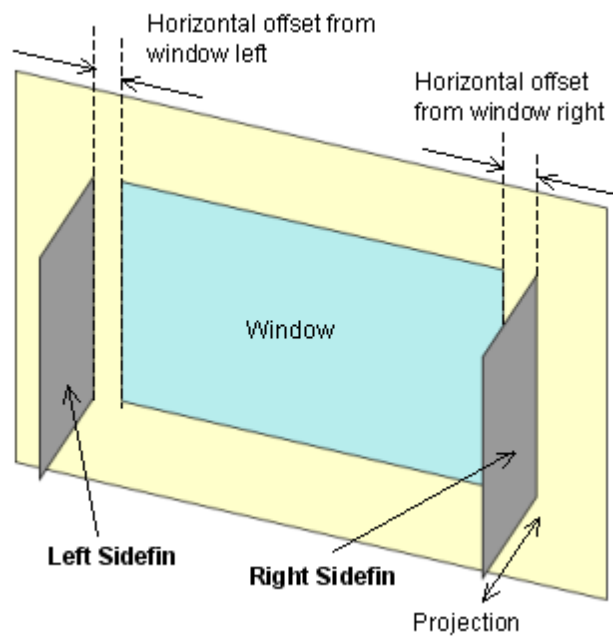


The number of louvres is a maximum value and if the window is not high enough to accommodate the requested number of louvres blades, any louvres blades which would be positioned below the window are not added.

Local Shading - Sidesfins

Sidesfins tab on [Local Shading](#) Dialog.

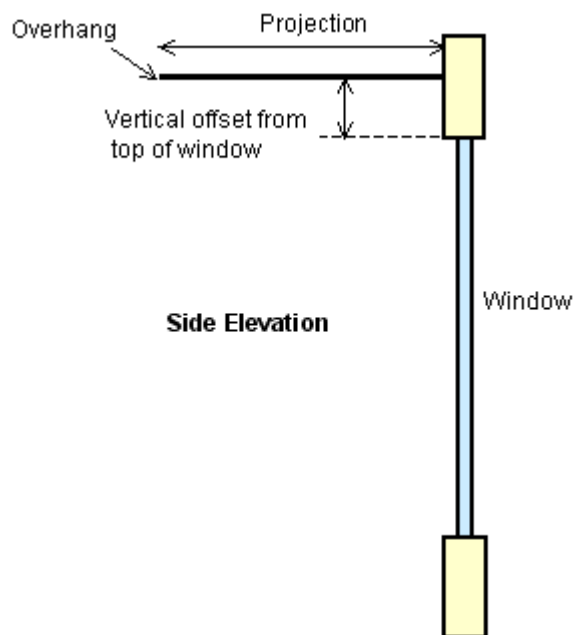
Enter details on the sidesfins, if fitted.

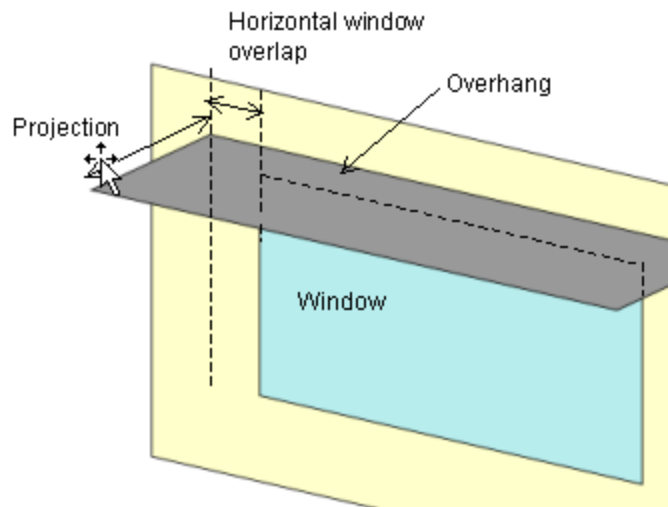


Local Shading - Overhangs

Overhangs tab on [Local Shading Dialog](#).

Enter details on the overhangs if fitted.





Window Blinds

Window blind components can be selected for external glazing and roof glazing from the Openings tab in the Model Data.

There are five types of window blinds, as selected in the Category data on the [General tab](#):

- [Slat](#) - slatted blinds which have different transmission properties with solar position.
- [Shade](#) - assumed perfectly diffusing.
- [Electrochromic switching](#) - switchable visible and solar properties based on solar gain.
- [SageGlass electrochromic](#) - multi-state electrochromic glazing systems.

Note: In the case where a Window blind is applied [mid-pane](#), any [dividers](#) that may have been specified for the window will not be applied in the model. EnergyPlus will generate a warning to this effect.

Window blind component data can be selected from the [window shading data](#) on the [Openings tab](#) in Model Data and is applied to all windows on the surface, including custom windows.

The data applies only to external and roof glazing - internal windows cannot have window blinds.

See also the [Shading via louvres, overhangs, and side fins](#) tutorial

Window blinds component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the window blinds icon on the Openings tab in the model data,
- by clicking on the Edit button when selecting window blinds from a list.

Window Blinds - General

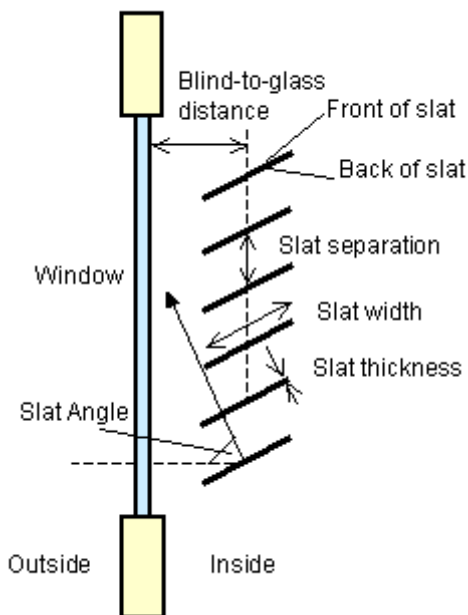
General tab on [Window Blinds](#) Dialog.

Enter the name, category and source for this Window blinds data set.

Window Blinds - Slat Data

Slat data tab on [Window Blinds](#) Dialog.

This object specifies the properties of a window blind consisting of flat, equally-spaced slats. Unlike window [shades](#), which are modelled as perfect diffusers, window blinds have solar and visible transmission and reflection properties that strongly depend on slat angle and angle of incidence of solar radiation. When in place, the blind is assumed to cover all of the glazed part of the window, including dividers; it does not cover any of the window frame, if present. The plane of the blind is assumed to be parallel to the glazing. When the blind is retracted it is assumed to cover none of the window. The solar and thermal effects of the blind's support strings, tapes or rods are ignored. Slat curvature, if present, is ignored.



Side Elevation

The slat angle is the angle between the glazing outward normal to the slat outward normal. A slat angle of 0° sets slats to be vertical and an angle of 90° sets them to be horizontal.

A slatted blind can be applied **1-Inside**, **2-Outside** or **3-Mid-pane** by selecting the appropriate [Position](#) on the Openings tab.

Slat Properties

Blind to Glass Distance

For interior and exterior blinds, the distance from the mid-plane of the blind to the adjacent glass (m). See schematic above. Not used for between-glass blinds. As for window [shades](#) this distance is used in calculating the natural convective air flow between glass and blind that is produced by buoyancy effects.

Slat Orientation

The choices are Horizontal and Vertical. "Horizontal" means the slats are parallel to the bottom of the window; this is the same as saying that the slats are parallel to the X-axis of the window. "Vertical" means the slats are parallel to Y-axis of the window.

Slat Width

The width of the slat measured from edge to edge.

Note: If the Blind is to be applied mid-pane then the gap between the inner pane and the second pane must be at least large enough to accommodate the slat width. The gap thickness is defined by the [Window gas](#) selected in the [Glazing](#) component.

Slat Separation

The distance between the front of a slat and the back of the adjacent slat.

Slat Thickness

The distance between the faces of a slat.

Slat Angle

The angle (degrees) between the glazing outward normal and the slat outward normal, where the outward normal points away from the front face of the slat (degrees).

Minimum Slat Angle

The minimum allowed slat angle (°) is used in the simulation only if one of the **2-Scheduled slat angle** or **3-Block beam solar Slat angle control types** have been selected for the window to which the blind has been applied. In this case, if the program tries to select a slat angle less than **Minimum slat angle** the slat angle will be reset to **Minimum slat angle**. Note that if the Minimum slat angle itself is less than the minimum allowed by **Slat width**, **Slat separation** and **Slat thickness**, it will be reset to that minimum.

Maximum Slat Angle

The maximum allowed slat angle (°) is used in the simulation only if one of the **2-Scheduled slat angle** or **3-Block beam solar Slat angle control types** have been selected for the window to which the blind has been applied. In this case, if the program tries to select a slat angle greater than **Maximum slat angle** the slat angle will be reset to **Maximum slat angle**. Note that if the Maximum slat angle itself is greater than the maximum allowed by **Slat width**, **Slat separation** and **Slat thickness**, it will be reset to that maximum.

Note: When using slatted blinds additional [Slat angle control parameters](#) also need to be set on the Openings tab to define how the slat angle varies (if at all).

Slat Conductivity

The thermal conductivity of the slat.

Slat Beam Solar Properties

Slat Beam Solar Transmittance

The beam solar transmittance of the slat, assumed to be independent of angle of incidence on the slat. Any transmitted beam radiation is assumed to be 100% diffuse (i.e., slats are translucent).

Front Side Slat Beam Solar Reflectance

The beam solar reflectance of the front side of the slat, assumed to be independent of angle of incidence (matte finish). This means that slats with a large specularly-reflective component (shiny slats) are not well modelled.

Back Side Slat Beam Solar Reflectance

The beam solar reflectance of the back side of the slat, assumed to be independent of angle of incidence (matte finish). This means that slats with a large specularly-reflective component (shiny slats) are not well modelled.

Slat Diffuse Solar Properties

Slat Diffuse Solar Transmittance

The slat transmittance for hemispherically diffuse solar radiation. This value should equal "Slat Beam Solar Transmittance."

Front Side Slat Diffuse Solar Reflectance

The front-side slat reflectance for hemispherically diffuse solar radiation. This value should equal "Front Side Slat Beam Solar Reflectance."

Back Side Slat Diffuse Solar Reflectance

The back-side slat reflectance for hemispherically diffuse solar radiation. This value should equal "Back Side Slat Beam Solar Reflectance."

Slat Beam Visible Properties

Slat Beam Visible Transmittance

The beam visible transmittance of the slat, assumed to be independent of angle of incidence on the slat. Any transmitted visible radiation is assumed to be 100% diffuse (i.e., slats are translucent).

Front Side Slat Beam Visible Reflectance

The beam visible reflectance on the front side of the slat, assumed to be independent of angle of incidence (matte finish). This means that slats with a large specularly-reflective component (shiny slats) are not well modeled.

Back Side Slat Beam Visible Reflectance

The beam visible reflectance on the front side of the slat, assumed to be independent of angle of incidence (matte finish). This means that slats with a large specularly-reflective component (shiny slats) are not well modeled.

Slat Diffuse Visible Properties

Slat Diffuse Visible Transmittance

The slat transmittance for hemispherically diffuse visible radiation. This value should equal "Slat Beam Visible Transmittance."

Front Side Slat Diffuse Visible Reflectance

The front-side slat reflectance for hemispherically diffuse visible radiation. This value should equal "Front Side Slat Beam Visible Reflectance."

Back Side Slat Diffuse Visible Reflectance

The back-side slat reflectance for hemispherically diffuse visible radiation. This value should equal "Back Side Slat Beam Visible Reflectance.."

Slat Infra Red Properties

Slat Infrared Hemispherical Transmittance

The slat Infrared transmittance. It is zero for solid metallic, wooden or glass slats, but may be non-zero in some cases (e.g., thin plastic slats).

Front Side Slat Infrared Hemispherical Emissivity

Front-side hemispherical emissivity of the slat. Approximately 0.9 for most materials. The most common exception is bare (unpainted) metal slats or slats finished with a metallic paint.

Back Side Slat Infrared Hemispherical Emissivity

Back-side hemispherical emissivity of the slat. Approximately 0.9 for most materials. The most common exception is bare (unpainted) metal slats or slats finished with a metallic paint.

Openings

The openings data allows you to define the distance between the blind and the window and also define the fraction of the blind surface that is open to air flow on each side of the blind. The following opening multipliers are defined in the same way as for window [shades](#). Unlike window shades, there is no input for Air-Flow Permeability; this is automatically calculated by the program from slat angle, width and separation.

Note: The opening multiplier data affects the flow of air through the cavity between the blind and the window. This flow is used in the calculation of the thermal resistance of the blind as part of the overall window / blind assembly resistance. Note in particular that **blinds do not affect any Calculated natural ventilation flows through the window during the simulation.**

Blind Top Opening Multiplier

Defined as for [Shades](#).

Blind Bottom Opening Multiplier

Defined as for [Shades](#).

Blind Left-Side Opening Multiplier

Defined as for [Shades](#).

Blind Right-Side Opening Multiplier

Defined as for [Shades](#).

Window Blinds - Shade Data

Shade data tab on [Window Blinds](#) Dialog.

Shade blinds can be used for diffusing materials such as drapery and translucent roller shades. For slat-type shading devices, like Venetian blinds, that have a strong angular dependence of transmission, absorption and reflection, it is better to use the [Slat option](#).

You can specify the properties of window shade materials. Reflectance and emissivity properties are assumed to be the same on both sides of the shade. Shades are considered to be perfect diffusers (all transmitted and reflected radiation is hemispherically-diffuse) with transmittance and reflectance independent of angle of incidence. When in place, the shade is assumed to cover all of the glazed part of the window, including

dividers; it does not cover any of the window frame, if present. The plane of the shade is assumed to be parallel to the glazing.

Transmittance and reflectance values for drapery material with different colour and openness of weave can be obtained from manufacturers or determined from 2001 ASHRAE Fundamentals, Chapter 30, Fig. 31.

A diffusing blind can be applied **1-Inside**, **2-Outside** or **3-Mid-pane** by selecting the appropriate [Position](#) on the Openings tab.

Shade Properties

Thickness

Thickness of the shade material (in m or in). If the shade is not flat, such as for pleated pull-down shades or folded drapery, the average thickness normal to the plane of the shade should be used.

Tip: If you plan to apply the shade **mid-pane** then you should enter a very low thickness value such as 0.0001m. In this case, if the shade performs an important insulation function (as well as shading) then you should adjust the conductivity accordingly. For example, in the case where the shade to be modelled is actually 0.025m thick and has conductivity 0.040 W/m-K you should enter the thickness as 0.0001m and the conductivity as $0.04 \times (0.001 / 0.025) = 0.00016$ W/m-K.

Conductivity

Shade material conductivity (in W/m-K or BTU-in/h-ft²-F).

Solar transmittance

Transmittance averaged over the solar spectrum. Assumed independent of incidence angle.

Solar reflectance

Reflectance averaged over the solar spectrum. Assumed same on both sides of shade and independent of incidence angle.

Visible transmittance

Transmittance averaged over the solar spectrum and weighted by the response of the human eye. Assumed independent of incidence angle.

Visible reflectance

Reflectance averaged over the solar spectrum and weighted by the response of the human eye. Assumed same on both side of shade and independent of incidence angle.

Thermal hemispherical emissivity

Effective long-wave emissivity. Assumed same on both sides of shade. We can approximate this effective emissivity, ϵ_{eff} , as follows. Let η be the “openness” the shade, i.e., the ratio of the area of openings in the shade to the overall shade area (see Air-Flow Permeability, below). Let the emissivity of the shade material be ϵ . Then

$$\epsilon_{\text{eff}} \approx \epsilon \cdot (1 - \eta)$$

For most non-metallic materials ϵ is about 0.9.

Thermal transmittance

Effective long-wave transmittance. Assumed independent of incidence angle. We can approximate this effective long-wave transmittance, τ_{eff} , as follows. Let η be the “openness” the shade, i.e., the ratio of the area of openings in the shade to the overall shade area. Let the long-wave transmittance of the shade material be τ . Then

$$T_{\text{eff}} \approx \eta + T_s(1 - \eta)$$

For most materials τ is very close to zero, which gives

$$T_{\text{eff}} \approx \eta$$

Openings

The openings data allows you to define the distance between the shade and the window and also define the fraction of the shade surface that is open to air flow on each side of the blind. The opening multipliers below are defined in a similar way as for window [blinds](#).

Note: The opening multiplier data affects the flow of air through the cavity between the shade and the window. This flow is used in the calculation of the thermal resistance of the shade as part of the overall window / shade assembly resistance. Note in particular that **shade do not affect any Calculated natural ventilation flows through the window during the simulation.**

Shade to glass distance

Distance from shade to adjacent glass (in m or in). This is denoted by s in Figures below. If the shade is not flat, such as for pleated pull-down shades or folded drapery, the average shade-to-glass distance should be used. (The shade-to-glass distance is used in calculating the natural convective air flow between glass and shade produced by buoyancy effects.). Note used for between-glass shades. In the following, H is the glazing height and W is the glazing width.

Shade top opening multiplier

Effective area for air flow at the top of the shade divided by sW , the horizontal area between glass and shade (see Figures below).

Shade bottom opening multiplier

Effective area for air flow at the bottom of the shade divided by sW , the horizontal area between glass and shade (see Figures below).

Shade left-side opening multiplier

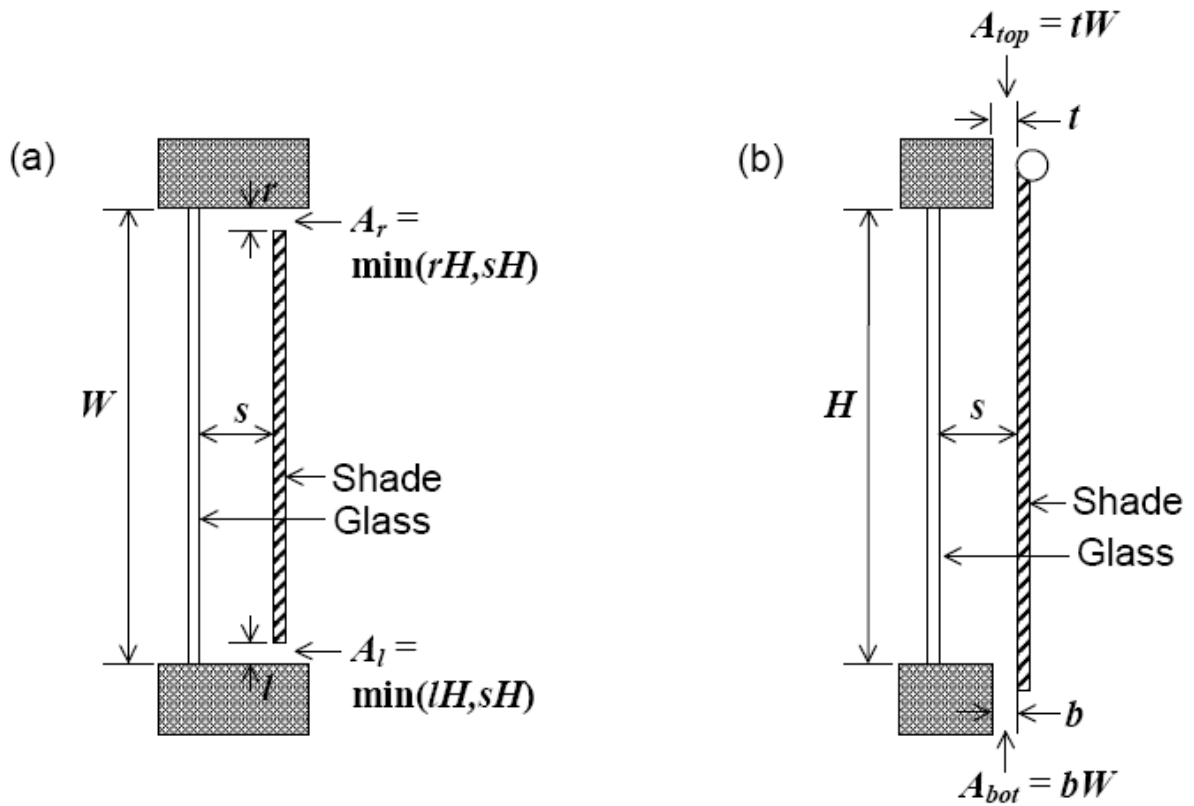
Effective area for air flow at the left side of the shade divided by sH , the vertical area between glass and shade (see Figures below).

Shade right-side opening multiplier

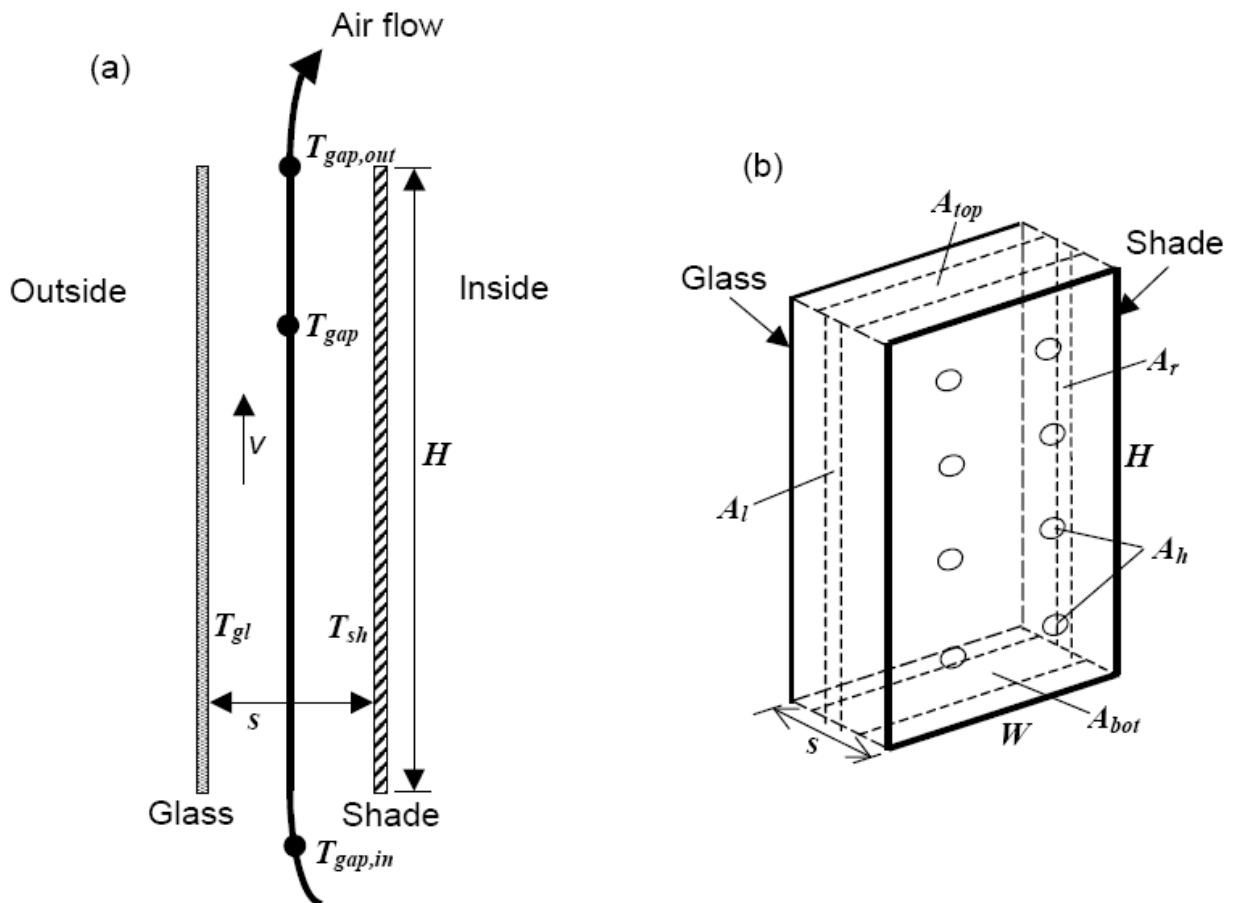
Effective area for air flow at the right side of the shade divided by sH , the vertical area between glass and shade (see Figures below).

Shade air-flow permeability

The fraction of the shade surface that is open to air flow, i.e., the total area of openings ("holes") in the shade surface divided by the shade area, HW . If air cannot pass through the shade material, Air-Flow Permeability = 0. For drapery fabric and screens the Air-Flow Permeability can be taken as the "openness" of the fabric (see 2001 ASHRAE Fundamentals, Chapter 30, Fig. 31), which is 0.0 to 0.07 for closed weave, 0.07 to 0.25 for semi-open weave, and 0.25 and higher for open weave.



Vertical section (a) and perspective view (b) of glass and interior shade layers showing variables used in the gap air flow analysis. In (b), the air-flow opening areas A_{bot} , A_{top} , A_l , A_r and A_h are shown schematically. See Engineering Manual for definition of thermal variables.



Examples of air-flow openings for an interior shade covering glass of height H and width W . Not to scale.

(a) Horizontal section through shade with openings on the left and right sides (top view).

(b) Vertical section through shade with openings at the top and bottom (side view).

In (a) Left-Side Opening Multiplier = $A_l / sH = \min(l/s, 1)$ and Right-Side Opening Multiplier = $A_r / sH = \min(r/s, 1)$. In (b) Top Opening

Multiplier = $A_{top} / sW = t/s$ and Bottom Opening Multiplier = $A_{bot} / sW = b/s$.

Note: The opening multiplier data affects the flow of air through the cavity between the blind and the window. This flow is used in the calculation of the thermal resistance of the blind as part of the overall window / blind assembly resistance. Note in particular that **blinds do not affect any Calculated natural ventilation flows through the window during the simulation.**

Window Blinds - Electrochromic Switching

Electrochromic tab on [Window Blinds](#) Dialog.

Enter the electrochromic switchable pane material.

This pane will be used in place of the standard outer glazing pane during the simulation for timesteps when shading is called for based on the Model data [control settings](#).

[More information on Electrochromic glazing.](#)

Note: that the blind [Position](#) in Openings Model Data must be **4-Switchable** to apply electrochromic switchable shading. This Position option becomes available when a [Window shading Type](#) from the **Electrochromic switchable** category has been selected.

SageGlass Electrochromic



[SageGlass](#) is a proprietary glazing system provided by Saint Gobain Electrochromic glass that tints automatically based on external conditions and, optionally a signal from a BMS system. It can dynamically change tint in response to the sun to provide unobstructed outdoor views and enhanced occupant comfort. The products control daylight, glare and energy use without the need for traditional solutions, like blinds, shades or low-e glass.

Detailed guidance on how to model SageGlass Electrochromic systems can be found in the [DesignBuilder SageGlass Modelling Guide](#).

Vent Components

Vent Components hold data on the [construction](#) to be used to represent heat conduction, a [coefficient of discharge](#) and a [texture](#) (used to represent a vent in the rendered view). They can be selected from the Openings tab.

[More information about vents](#).

There are two tabs on the Vents dialog:

- [General](#)
- [Airflow](#)

Vent component data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the vent icon on the Openings tab in the model data,
- by clicking on the Edit button when selecting vents from a list.

Vents - General

General tab on [Vents](#) Dialog.

Enter the name and category for this vent component.

General

Name

Enter a unique name to describe this vent.

Category

Select from one of the following categories:

- **Grill** - a vent normally only used with Calculated natural ventilation although can also be used with Scheduled natural ventilation in internal surfaces to include "Zone Mixing" air exchange. By default the Discharge coefficient is 0.5.
- **Opening** - a vent normally only used with Calculated natural ventilation although can also be used with Scheduled natural ventilation in internal surfaces to include "Zone Mixing" air exchange. By default the Discharge coefficient is 0.6.
- **Zone Exhaust Fans** - a special category that must be used with a corresponding [Zone exhaust fan](#) in the equivalent [HVAC zone](#) when using Detailed HVAC with Calculated natural ventilation.

Note: Vents with the **Zone Exhaust Fan** category must be placed on *external* surfaces.

Visual

Texture

Select the texture to be used to represent the vent in rendered views.

Conduction

Construction

Regardless of the [Natural ventilation](#) mode selected, heat conduction through the vent will be modelled separately from any airflow. Select the construction that will be used to model the heat conduction.

Vents - Airflow

[Airflow tab on Vents Dialog.](#)

Airflow

Coefficient of Discharge

Enter the discharge coefficient for this vent component.

DesignBuilder applies a default of 0.5 for **Opening** type vents and 0.6 for **Grill** type vents.

You can select this [vent](#) from the [Openings](#) model data.

Note: The discharge coefficient is only used in thermal simulations when the [Natural ventilation](#) Model option is set to **Calculated**.

Velocity Calculation

The velocity of the air moving through the vent is given by the standard orifice equation:

$$v(y) = C_d \sqrt{2 \frac{P_n(y) - P_m(y)}{\rho}}$$

Where:

- v : Velocity
- C_d : Discharge coefficient
- P : Pressure either side of the vent
- ρ density of the air going through the opening

Holiday Schedule

A 'Holiday schedule' is a list of intervals of one or more day which identify the days the building may be unoccupied. The [holiday intervals](#) are listed in order, with holidays most likely to be taken at the top of the list. The [Holidays per year](#) Model data on the [Activities](#) tab at building level defines how many holidays are taken and this 'Holiday schedule' defines which days are holidays. Days are selected from the top of the list working downwards. The data is used to specify EnergyPlus 'SpecialDayPeriods'.

There are two tabs on the Holidays dialog:

- [General](#)
- [List](#)

Holiday data can be accessed:

- from the [Components tab on the opening screen](#) (to view/edit library components if no file is open),
- from the [Components tab in the Navigator panel](#) (to view/edit model components if a file is open),
- by double-clicking on the holidays icon on the Activity tab in the model data,
- by clicking on the Edit button when selecting holidays from a list.

Holiday Schedule - General

General tab on [Holidays](#) Dialog.

Enter the [Region](#) and Category for the Holiday Schedule.

Holiday Schedule - List

List tab on [Holidays](#) Dialog.

Holidays are selected from the top of the list, so holidays which are most likely to be taken should be defined first (so they are positioned at the top of the list).

Use the controls in the Info panel to add, edit and delete holidays.

Valid holiday date formats are defined in [Compact Schedules: EnergyPlus Date Definition](#)

Ground Domain

Ground domain components are defined at site level under the Site Details > Ground > [Ground Domain](#) headers. The adjacency between ground domain components and the corresponding zone surface in the model is defined by selections made at surface level on the Construction tab under the [Ground Domain](#) header.

Tabs on the dialog are:

- "Ground Domain - General Tab" on page 1456
- "Ground Domain - Soil" on page 1460
- "Ground Domain - Ground Undisturbed Temperatures" on page 1460
- "Ground Domain - Insulation" on page 1462

Ground Domain - General Tab

General tab on [Ground Domain](#) Dialog

Name

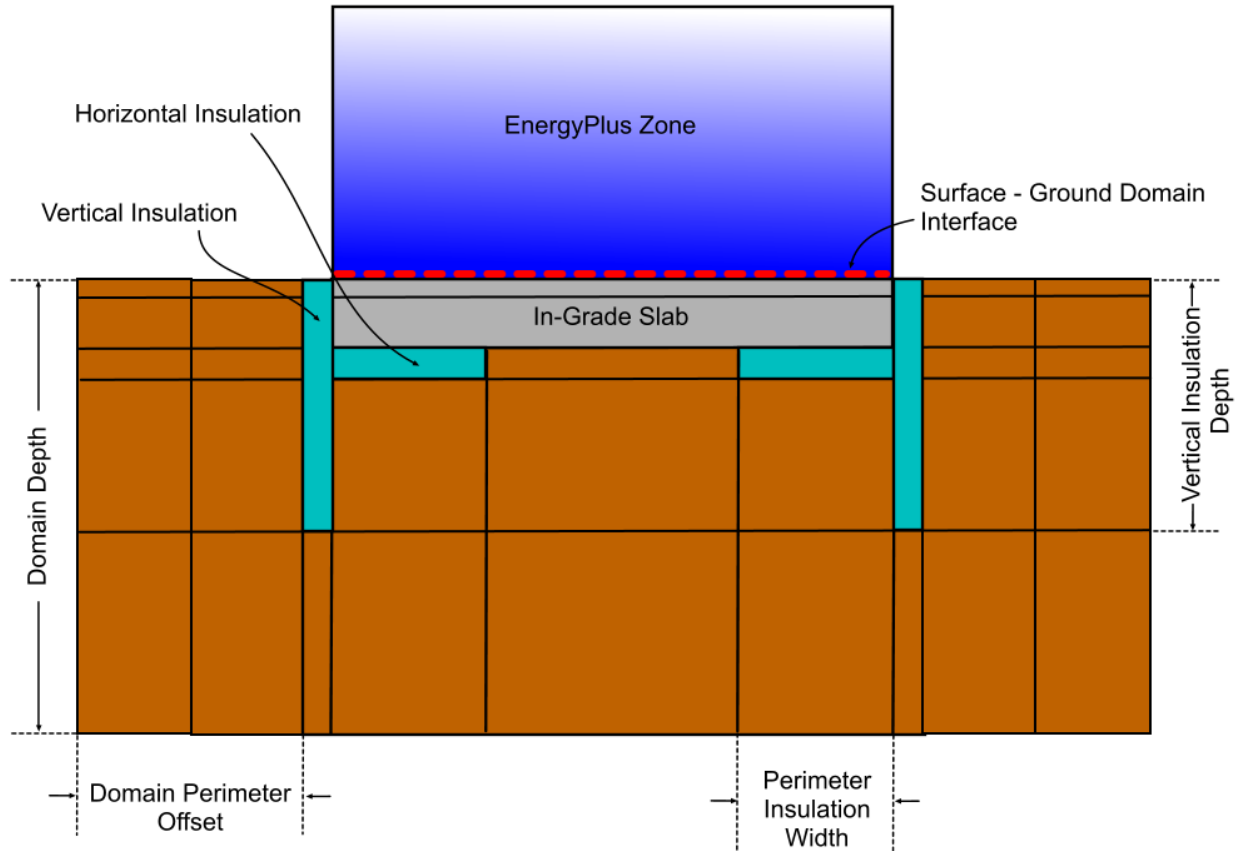
Enter a unique name for the ground domain component.

Category

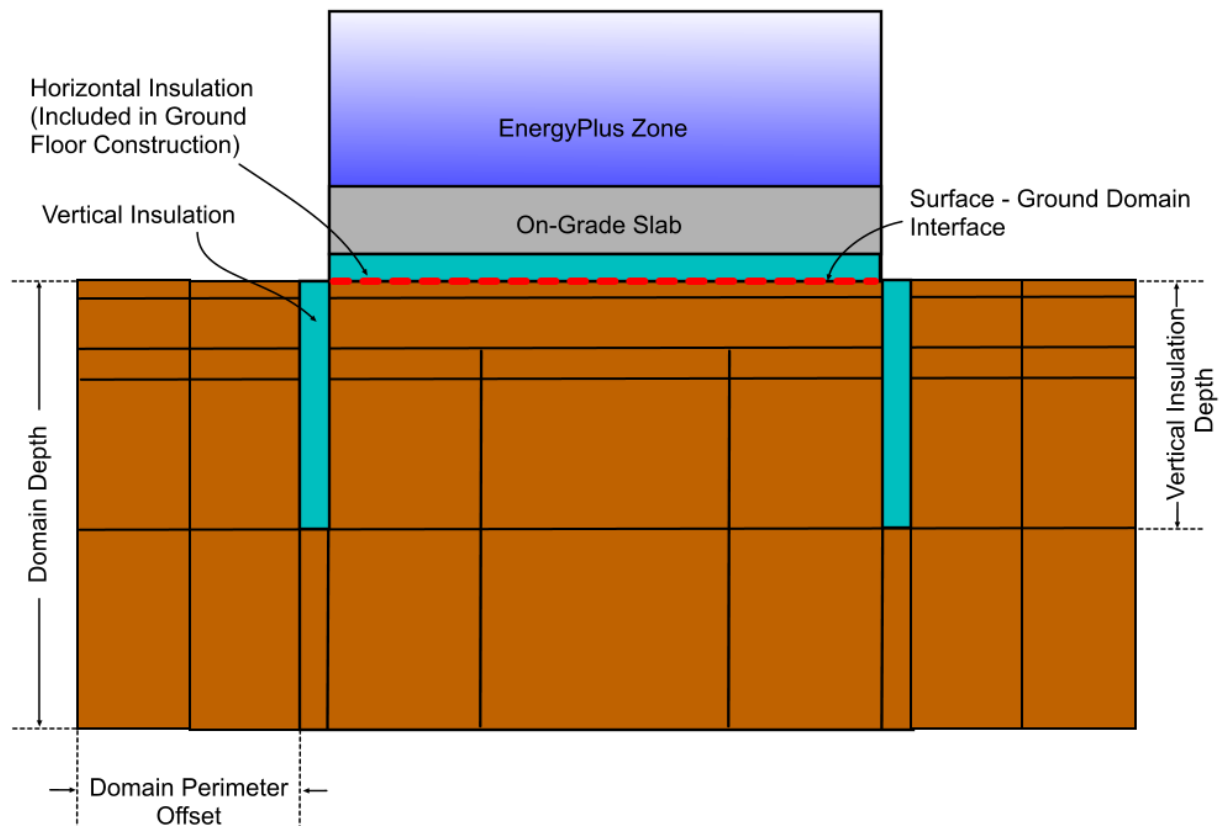
You can select 2 types of ground domain objects:

- **Slab** is a horizontal slab sitting either on or just below ground level. Slabs are used to simulate ground coupled heat transfer with horizontal building surfaces. Horizontal ground surfaces interact with the slab ground domain objects through adjacencies defined at [surface level on the Construction tab](#). Multiple horizontal ground floor surfaces can be coupled to the same ground domain object. Alternatively each horizontal surface may also have its unique ground domain, however, runtime will be adversely affected in this case.

Slab ground domain objects are equipped to model 2 different scenarios: in-grade slabs, and on-grade slabs. This selection is made in the [Slab section](#) below.



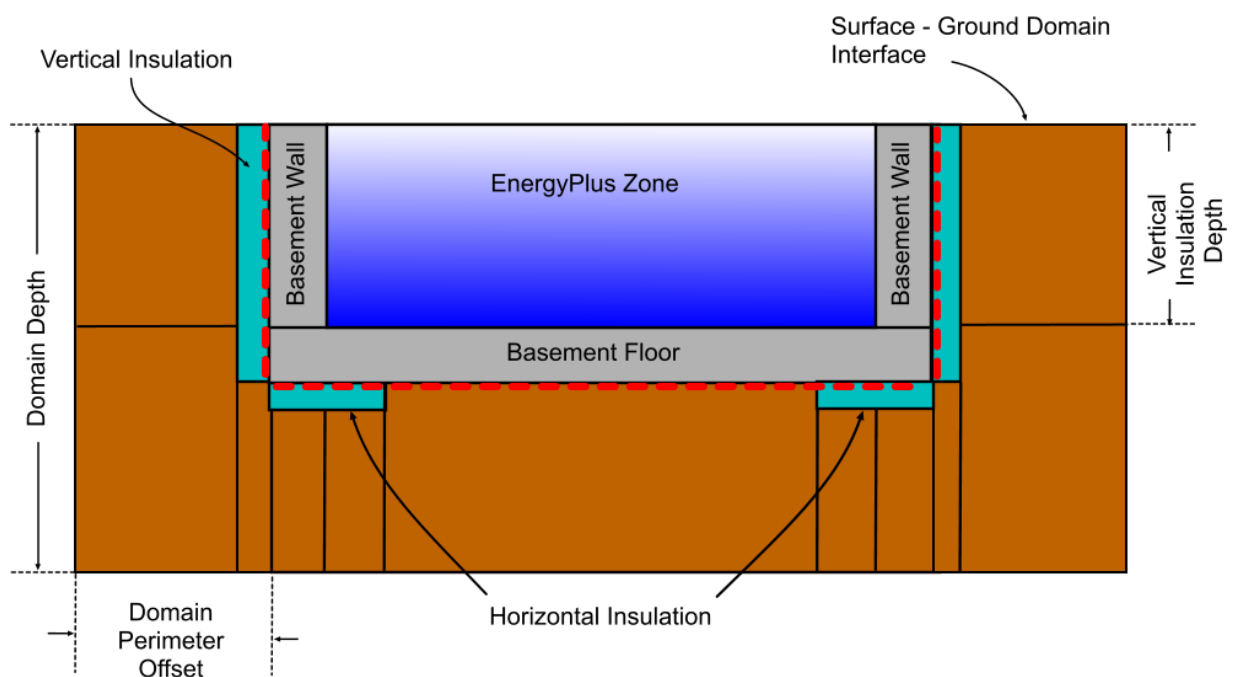
In Grade Slab



On Grade Slab

- **Basement** - for underground zones with underground wall and floors adjacent to the ground. Horizontal and vertical ground insulation are simulated by the ground domain and therefore should not be included in the wall and floor construction objects. Vertical wall surfaces and horizontal floor surfaces adjacent to ground interact with the basement ground domain objects by through connections made at [surface level on the Construction tab](#).

Note: When defining a basement ground domain it is important to define at least one wall and one ground floor as being adjacent to any particular basement domain object.



Please refer to the above diagrams when reading about the various dimensions described below

Ground domain depth

Depth of the simulation domain (in m or ft).

Aspect ratio

This is the of ratio of the slab or basement length to width (when viewed in plan), used to determine the aspect ratio of the overall domain. This field along with the total slab or basement floor area, which is taken as the combination of all surfaces connected to the floor of the basement, are used to determine the size and shape of the domain. Aspect ratios and the inverse of aspect ratios should produce identical results. i.e. Aspect ratio = 2 is equivalent to Aspect ratio = 0.5.

Perimeter offset

The distance from the slab/basement perimeter to the domain perimeter (in m or ft).

Evapo-transpiration ground cover parameter

Numeric field specifies the ground cover effects used in the evapo-transpiration model at the ground surface heat balance. The values range from 0 (solid, non-permeable ground surface) to 1.5 (wild growth).

Simulation timestep

Selection indicating whether the domain will update temperatures at each zone timestep, or at hourly intervals. Options are:

- **1-Hourly**
- **2-Timestep**

Slab

Slab location

The slab location indicates whether the slab is in-grade (top surface level with ground surface) or on-grade (bottom surface level with ground surface). Options include:

- **1-On grade** which is used to simulate situations where the *lower* slab surface is near the ground surface level. In this situation, the entire floor must be included within the floor construction object. Vertical insulation is modelled by the Ground domain in this scenario. Horizontal insulation can only be modelled as covering the full horizontal surface.
- **2-In grade** which is used to simulate situations when the *upper* slab surface is near the ground surface level. For this situation, slab's upper surface must interact with the zone via a boundary which is specified at surface level. Because of this, the Floor construction object for the zone floor surface must include a thin layer of the upper floor material. Horizontal and vertical insulation are modelled by the Ground domain in this scenario. Horizontal insulation can be modelled as covering the full horizontal surface, or it can be limited to the perimeter regions only. In the latter case, the perimeter insulation width must be specified.

Slab material

Select the material component representing the slab material thermal properties. Only required for in-grade situations.

Thickness

Thickness of the slab (in m or ft). Only required for in-grade situations.

Basement

Basement depth

Depth of basement floor surface referenced from the ground surface (in m or ft). This domain should be the distance from the ground surface down to the basement floor surface. In cases where the ground surface is below the main above-ground building level, a separate wall surface should be employed between the basement walls and the main level walls.

Mesh density parameter

Integer field indicating the density of the finite difference ground domain cells between the basement and the far field boundaries. Default value is 4. Total number of ground domain cells, insulation cells, and ground surface cells are indicated as outputs to the eio file.

Ground Domain - Soil

Soil tab on [Ground Domain](#) Dialog

On this tab you can select the Material defining the ground thermal properties and enter moisture content of the ground. The thermal properties of the material are used in calculation of the [undisturbed ground temperatures](#) as well as general heat conduction from building ground-adjacent surfaces connected to the ground domain object.

Soil Thermal Properties

Soil material

Select the material used to define the soil thermal properties.

Soil Moisture Content

Moisture and freezing effects are accounted for within EnergyPlus during the simulation by varying the specific heat value.

Soil moisture content volume fraction

A nominal value of soil moisture content to be used when evaluating soil thermal properties.

Soil moisture content volume fraction at saturation

A nominal value of soil moisture content when the soil is saturated, this is used in evaluating thermal properties of freezing soil.

Ground Domain - Ground Undisturbed Temperatures

Ground Undisturbed Temperature tab on [Ground Domain](#) Dialog.

During the simulation EnergyPlus needs to have access to the distribution of temperatures in the "undisturbed" ground at the site. This means the temperature that would exist in the ground over the year if the building(s) were not present. EnergyPlus provides three ways to calculate these undisturbed ground temperatures:

- [1-Finite difference](#), the default option.
- [2-Kusuda-Achenbach](#),
- [3-Xing](#).

More information on these options is provided below. Note that all of the calculation methods also require the ground thermal properties as defined by the [soil material](#) on the Soil tab.

Finite Difference

The Finite Difference method uses a 1-D finite difference heat transfer model which uses the weather file to obtain surface boundary conditions. An annual simulation is run on the ground model during its initialization until the annual ground temperature profile has reached steady periodic behaviour. Once steady periodic behaviour is reached, the ground temperatures are cached for retrieval during the rest of the simulation.

No additional data needs to be entered for this option.

Technical

This model uses a 1-D implicit finite difference heat transfer model to determine the steady periodic annual ground temperature. The model, which uses a daily timestep, is run through an annual simulation using the user provided weather file to determine daily averages for global horizontal radiation, air temperature, relative humidity, and wind speed. Once the steady-periodic ground temperature has been determined, the temperatures are cached for use later in the simulation. The basis for the model was taken from Xing, 2014, however, the numerical methods were adapted from those described in Lee, 2013; the latter uses an implicit numerically stable finite difference method, whereas the former utilized an explicit, conditionally stable method. Surface heat balance boundary conditions are similar what is described in Herb et al., 2008. Evapotranspiration is considered as described by Allen et al., 1998. Soil freezing given the assumed stagnant soil moisture content is also considered. The model does not consider the effects of vegetative canopy layers, snow cover, ground water flow, ground moisture transport, or surface runoff.

Kusuda-Achebach

The Kusuda-Achebach method provides an undisturbed ground temperature based on the correlation developed by Kusuda T. and P. Achenbach. 1965. The correlation uses three parameters for ground temperature at the surface to define a correlation for undisturbed ground temperatures as a function of depth and time. If one thinks of the ground temperature for a given depth as a sinusoid, the average ground temperature, amplitude (average difference between maximum ground temperature and minimum ground temperature), and the phase shift (day of minimum surface temperature) are all required to define the correlation.

Technical

$$T(z, t) = \bar{T}_s - \Delta \bar{T}_s \cdot e^{-z \cdot \sqrt{\frac{\pi}{\alpha \tau}}} \cdot \cos \left(\frac{2\pi t}{\alpha \tau} - \theta \right)$$

Where:

$T(z, t)$ is the undisturbed ground temperature as a function of time and depth.

$\bar{T}_s$ is the average annual soil surface temperature, in °C.

$\Delta \bar{T}_s$ is the amplitude of the soil temperature change throughout the year, in °C.

θ is the phase shift, or day of minimum surface temperature.

α is the thermal diffusivity of the ground.

τ is the time constant, 365.

Kusuda-Achebach annual average ground surface temperature

The average ground surface temperature throughout the entire year (in °C or °F)

Kusuda-Achebach average amplitude of annual ground surface temperature

The average amplitude of the ground surface temperature (in °DeltaC or °DeltaF)

Kusuda-Achenbach phase shift of minimum surface temperature

The day of the year which has the lowest ground surface temperature (in days).

Xing

This model uses the correlation developed by Xing, 2014 to predict undisturbed ground temperature. The correlation is a 5 parameter, 2 harmonic model based on the work of Lord Kelvin (Thomson, 1862). The average soil surface temperature and two sets of surface temperature amplitude and phase shift must be provided.

Technical

Parameters for 4000+ international locations can be found in Xing, 2014. Thomson, W. 1862. 'On the Reduction of Observations of Underground Temperature, with applications to Professor Forbes' Edinburgh Observations and the continued Calton Hill Series.' Proceedings of the Royal Society of Edinburgh. IV: 342-346. Xing, L. 2014. Estimations of Undisturbed Ground Temperatures using Numerical and Analytical Modeling. Ph.D. Diss. Oklahoma State University, Stillwater, OK.

The parameters were first determined by creating and validating a finite difference numerical model which used local weather data for boundary conditions. From the numerical model, the correlation parameters were determined to provide for this simplified design model.

$$T(z, t) = \bar{T}_s - \sum_{n=1}^2 \Delta \bar{T}_{s,n} \cdot e^{-z \cdot \sqrt{\frac{n\pi}{\alpha\tau}}} \cdot \cos \left[\frac{2\pi n}{\tau} (t - \theta_n) - z \sqrt{\frac{n\pi}{\alpha\tau}} \right]$$

Where:

$T(z, t)$ is the undisturbed ground temperature as a function of time and depth

$\bar{T}_s$ is the average annual soil surface temperature, in deg C

$\Delta \bar{T}_{s,n}$ is the n-th amplitude of the soil temperature change throughout the year, in °C

n is the n-th phase shift, or day of minimum surface temperature

α is the thermal diffusivity of the ground.

τ is the time constant, 365.

Average soil surface temperature

The average annual surface temperature of the soil (in °C or °F).

Soil surface temperature amplitude 1

First soil surface temperature amplitude parameter (in °C or °F).

Soil surface temperature amplitude 2

Second soil surface temperature amplitude parameter (in °C or °F).

Phase shift of surface temperature amplitude 1

Phase shift of surface temperature amplitude 1, in days.

Phase shift of surface temperature amplitude 2

Phase shift of surface temperature amplitude 2, in days.

Ground Domain - Insulation

Insulation tab on [Ground Domain](#) Dialog.

On this tab you can define whether there is horizontal and vertical insulation present and if so the extent and position of the insulation and its thermal properties.

Horizontal Insulation

Horizontal insulation is only applicable to [in-grade](#) situations.

Include horizontal insulation

Checkbox indicates whether horizontal insulation is present. When it is checked the data below must be entered.

Material

Where horizontal insulation is present, select the material representing the horizontal slab insulation thermal properties.

Thickness

Enter the thickness of the horizontal insulation material layer (in m or in).

Extent

Droplist selection indicating whether the horizontal slab insulation extends to cover the full horizontal area of the slab, or only covers the slab perimeter. Options are:

- **1-Full** and
- **2- Perimeter**

Perimeter insulation width

This setting indicates the width of the perimeter insulation measured from the slab edge (in m or ft). Valid range is from > 0 to $< 0.5 \times$ smallest slab width. It is not required when the Horizontal insulation extent

Vertical Insulation

Include horizontal insulation

Checkbox indicates whether vertical insulation is present. When it is checked the data below must be entered.

Material

Select the material representing the vertical slab insulation.

Thickness

Enter the thickness of the vertical insulation material layer (in m or in).

Depth

The depth measured (in m or ft) from the ground surface to which the vertical perimeter insulation extends. Valid range is from $>$ Slab thickness to $<$ Domain depth.

Kiva Foundation

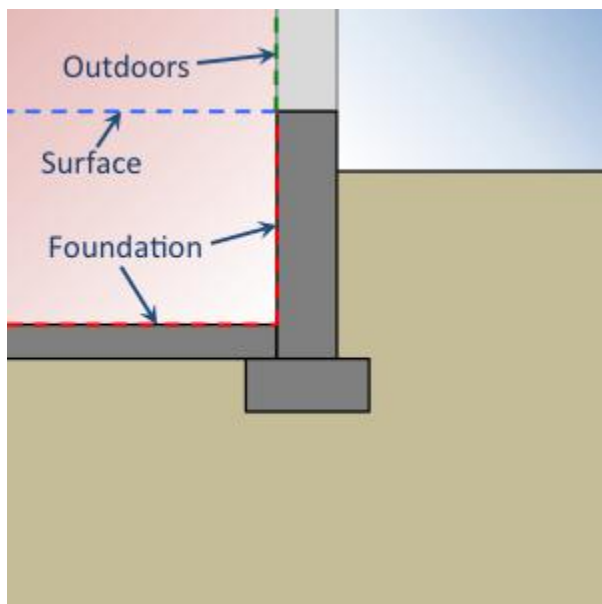
Kiva foundation objects describe boundary conditions for ground-coupled foundation surfaces when one of the Kiva [Ground modelling methods](#) have been selected (**2-Kiva Basic** or **4-Kiva Full**).

Technical Description

The model inputs for Kiva foundation objects are translated into Kiva's foundation heat transfer model. Kiva generates a two-dimensional heat transfer calculation to represent heat flow between a zone and the adjacent

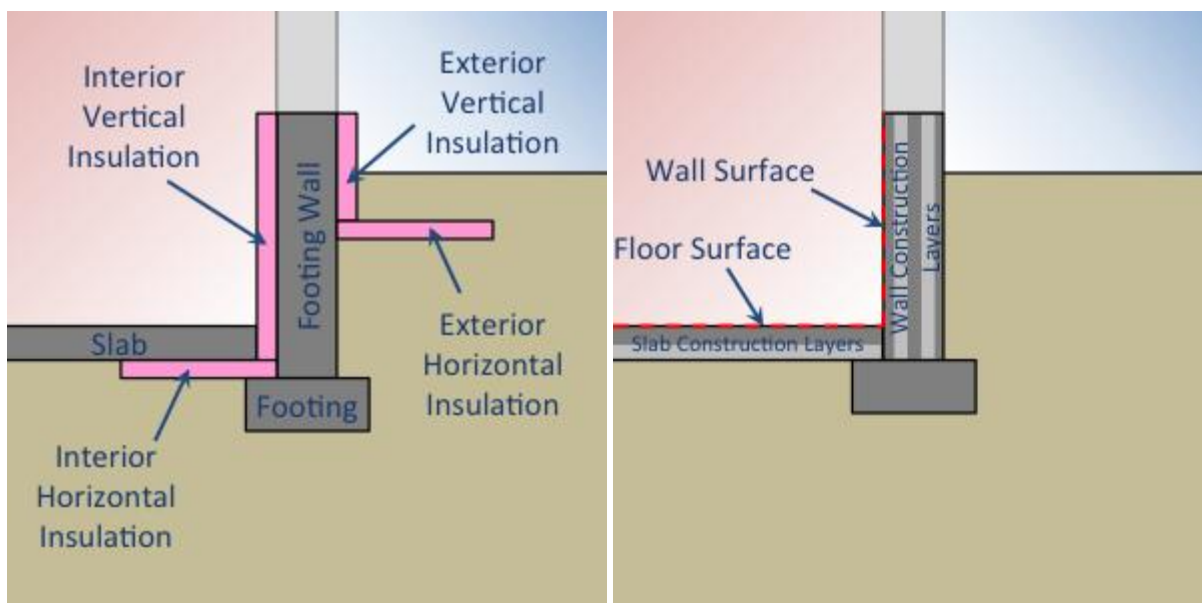
ground. Kiva foundation surfaces do not use the same [Solution Algorithm](#) (e.g. CTF, Finite difference etc) as the rest of the model.

Kiva foundation objects are used to describe the two-dimensional features that cannot be captured by the typical one-dimensional constructions used in EnergyPlus. The figure below illustrates Kiva's two-dimensional context for a basement where the basement slab and wall both refer to "Foundation" as the Outside Boundary Condition, the ceiling of the basement and the exterior wall of the zone above the basement refer to "Surface" (or "Zone") and "Outdoors", respectively. The non-foundation surfaces are shown in the figure below for context, but are not part of the Kiva model.



Outside Boundary Conditions for surfaces within Kiva's Two-dimensional context. Only surfaces referencing Foundation are simulated in Kiva

This context allows for a finer description of the structural and insulation components of a foundation that impact heat transfer (see figure below). Kiva foundation objects define only the aspects of the foundation that are not already defined by the one-dimensional constructions of the respective surfaces. That is, the "footing wall" and "slab" constructions and their relative dimensions are inferred from the respective ground floor construction and below grade wall constructions, whereas the "footing" and "insulation" objects are set here on the Kiva Foundation dialog. The insulation here is specific to edge insulations around the foundation. Continuous under slab or wall insulations (that are along the whole floor or along the full height of the wall) are modelled as part of surface constructions. Additionally, if the foundation wall is not exposed to a zone, then the [Footing wall](#) construction setting is used.

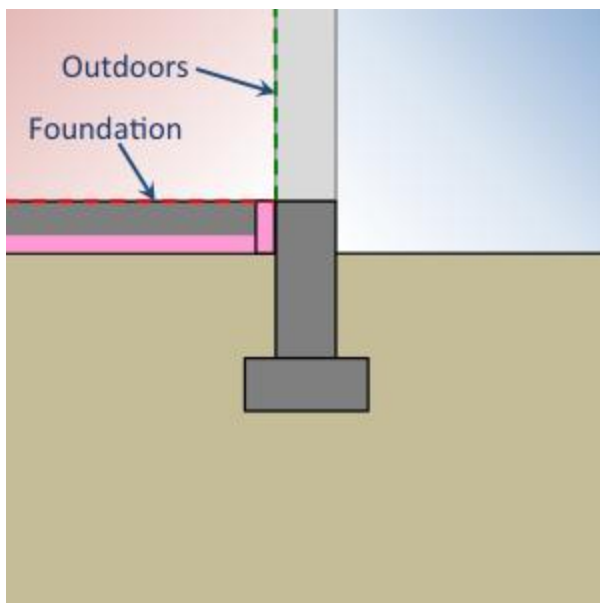


Structural and insulation components of Kiva Foundations and Two-dimensional interpretation of foundation surface data. These components are defined either as one-dimensional constructions of the respective surfaces or as part of Kiva Foundation objects.

Note: Because the floor slab is defined by the selection of [Ground floor construction](#) for the ground surface, any insulation that extends across the whole floor (above or below the concrete slab layer) should simply be added as additional material layers on the [Constructions](#) dialog.

The depth of the foundation is defined by the height of the wall surfaces that reference the Kiva foundation boundary condition object. For slab-on-grade foundations, a depth of zero is implied by having no associated wall surfaces. The figure below shows a slab-on-grade foundation with whole slab insulation. Notice there are no walls referencing the “Foundation” Outside Boundary Condition. In this case, the under-slab insulation is modelled as part of the slab construction, while the edge/gap insulation is modelled using the interior vertical insulation fields of a Kiva foundation object.

Note: Since there are no wall surfaces for slab foundations, the footing wall construction is defined within the Kiva foundation object.



Two-dimensional interpretation of foundation surface data

The width of the floor surface in the two-dimensional context is defined by the area and the exposed perimeter of the floor surface object. Details on this calculation can be found in the Engineering Reference document.

Other components of the two-dimensional context are defined by the [Kiva foundation settings](#) object and applied uniformly for all Kiva foundation objects. These components include:

- Far-Field width
- Deep Ground depth (and boundary type)
- Soil and ground surface thermal properties

See also: [Kiva Limitations](#) in the [Summary](#) article.

General

Name

The unique identifier of the Kiva foundation object.

Initial Indoor Temperature

When Kiva initialises its ground domains at the start of an EnergyPlus simulation it needs a temperature for the boundary between the building and the ground. The **Custom initial indoor air temperature** setting described below provides a way for users to control this aspect of domain initialisation.

See also the "Initialisation and Warm-Up" on page 309 section of the Kiva Summary.

Use custom initial indoor air temperature

Generally you should check this checkbox and enter an initial indoor air temperature in the **Custom initial indoor air temperature** text box below. Otherwise, if you leave this checkbox unchecked, indoor air temperatures during initialization will be estimated based on zone setpoints. However in some cases, such as when using Simple HVAC, setpoints are not available and EnergyPlus may set unrealistic initial temperatures and you will see unpredictable results.

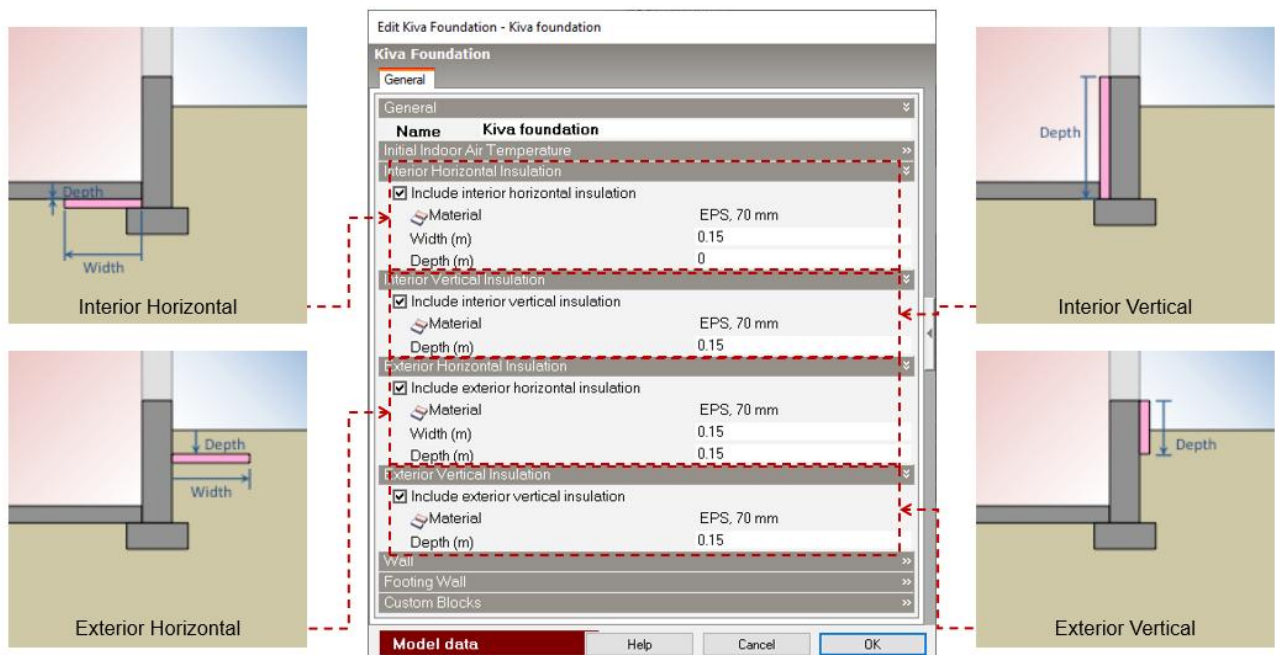
Custom initial indoor air temperature

Enter the indoor air temperature (in °C or °F) to be used for the Kiva domain initialization prior to the main EnergyPlus simulation warmup period.

Kiva Insulation Overview

Continuous insulation of the ground slab is modelled as a layer in the ground construction in the normal way, but Kiva also allows additional insulation elements to be included along the perimeter of the building. The thickness, length and position of these insulation elements are mostly defined explicitly on the Kiva Foundation dialog, or in the case of the thickness of the footing wall construction, they are taken from the layer data for the selected construction.

- **Interior horizontal insulation**, **Width** is measured from the wall interior to the edge of insulation. And **depth** is the distance from bottom of the slab which is typically 0.
- **Interior vertical insulation**, **Depth** is measured from the top of the wall to the edge of the insulation.
- **Exterior horizontal insulation**, **Width** is measured from the wall exterior to the edge of the insulation. And **depth** is the distance from the exterior grade.
- **Exterior vertical insulation**, **Depth** is measured from the top of the wall to the edge of the insulation.



Details on how to define additional internal and external horizontal and vertical insulation elements is provided below.

Also, some examples of how to enter Kiva Foundation data for some specific cases are provided on the "Kiva Foundation Example Cases" on page 1473 page.

Interior Horizontal Insulation

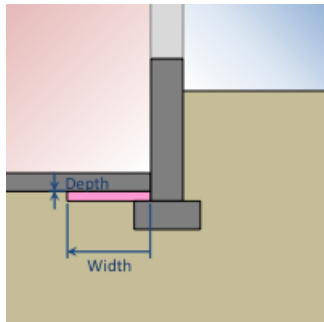
Include interior horizontal insulation

Check this checkbox if interior horizontal insulation is to be included with this Kiva foundation object.

Interior horizontal insulation material

Select the material associated with the interior horizontal insulation. The material [Default thickness](#) value defines the thickness of the insulation.

The following two fields define the placement of this material within Kiva's two-dimensional context as illustrated in the figure below.



Placement of interior horizontal insulation

Width

Extent of insulation as measured from the wall interior to the edge of interior horizontal insulation (in m or ft).

Depth

Distance from the bottom of the slab to the top of interior horizontal insulation (in m or ft). Generally, interior horizontal insulation is placed directly below the slab, and therefore, this value will usually be set to zero.

Interior Vertical Insulation

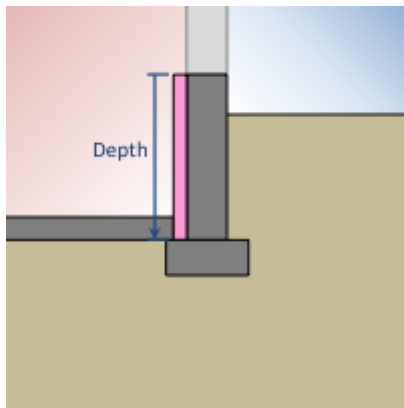
Include interior vertical insulation

Check this checkbox if interior vertical insulation is to be included with this Kiva foundation object.

Material

Select the material object associated with the interior vertical insulation. The material [Default thickness](#) value defines the thickness of the insulation.

The following field defines the placement of this material within Kiva's two-dimensional context as illustrated in the figure below.



Placement of interior vertical insulation

Depth

Extent of insulation as measured from the wall top to the bottom edge of the interior vertical insulation (in m or ft).

Exterior Horizontal Insulation

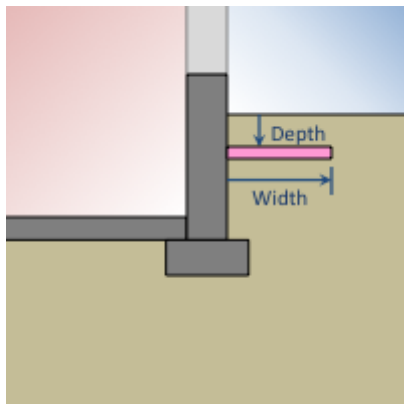
Include exterior horizontal insulation

Check this checkbox if exterior horizontal insulation is to be included with this Kiva foundation object.

Material

Select the material associated with the exterior horizontal insulation. The material [Default thickness](#) value defines the thickness of the insulation.

The following two fields define the placement of this material within Kiva's two-dimensional context as illustrated in the figure below.



Placement of exterior horizontal insulation

Width

Extent of insulation as measured from the wall exterior to the edge of exterior horizontal insulation (in m or ft).

Depth

Distance from the exterior grade to the top of exterior horizontal insulation (in m or ft).

Exterior Vertical Insulation

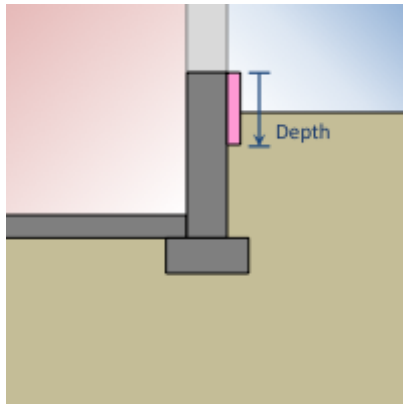
Include exterior vertical insulation

Check this checkbox if exterior vertical insulation is to be included with this Kiva foundation object.

Material

Select the material associated with the exterior vertical insulation. The material [Default thickness](#) value defines the thickness of the insulation.

The following field defines the placement of this material within Kiva's two-dimensional context as illustrated in the figure below.



Placement of exterior vertical insulation

Depth

Extent of insulation as measured from the wall top to the bottom edge of the exterior vertical insulation (in m or ft).

Footing Wall

Footing wall construction

Defines the construction of the foundation footing wall for slab foundations where the foundation wall is not exposed to the zone (and has no zone surface to explicitly assign a construction).

Wall height above grade

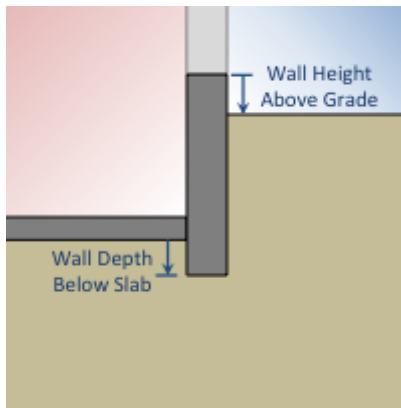
Distance from the exterior grade to the wall top (in m or ft).

Wall depth below slab

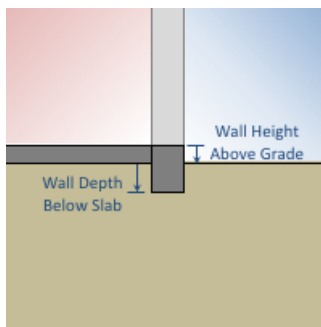
Distance from the slab bottom to the bottom of the foundation wall (in m or ft).

Details

The figure below illustrates the definition of both [Wall height above grade](#) and [Wall depth below slab](#). If the top of foundation wall is flush with the ground then the [Wall Height Above Grade](#) should be entered as zero.



Definition of exterior grade and footing wall depth relative to the wall surface



Definition of exterior grade and footing wall depth relative to the wall surface (for a slab foundation context)

Extending the wall below the slab provides a coarse approximation of the foundation footing. Alternatively, use the **Footing material**, **Footing width** and **Footing depth** to settings explicitly model the footing.

Note: Explicit modelling of the footing requires a higher spatial discretisation and, therefore, longer computation times.

Include foundation footing

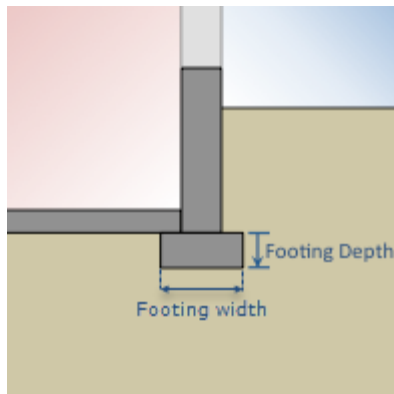
If a foundation footing is to be included then check this checkbox to reveal the further associated settings described below.

Footing material

Select the material associated with the foundation footing (typically some form of concrete).

Note: The **Thickness** property of the selected material will be overwritten by the **Footing width** setting entered below.

The following field defines the placement of this material within Kiva's two-dimensional context and are illustrated in the figure below.



Placement of footing

Footing width

Horizontal width dimension of the footing (in or ft).

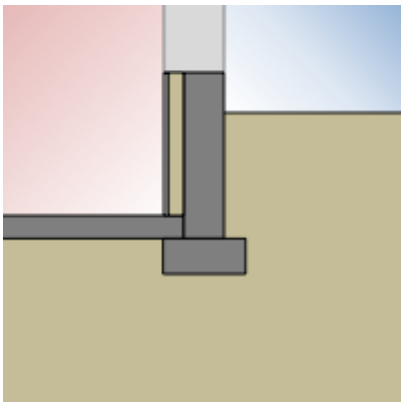
Note: The value entered here is used to define the **Thickness** property of the simulated Footing material. See above.

Footing depth

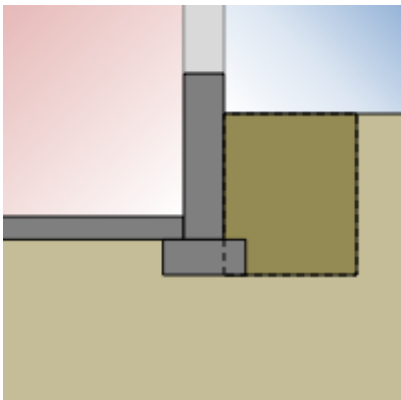
Top-to-bottom dimension of the footing (not to be confused with its depth in the ground) (in m or ft).

Custom Blocks

Custom blocks can be used to represent solid materials in the two-dimensional context that are not otherwise covered by the fields above. Examples of this might include interior finishings and insulation or backfill soil with different thermal properties (see figures below).



Custom blocks representing interior batt insulation and dry wall



Custom block representing exterior backfill

Number of custom blocks

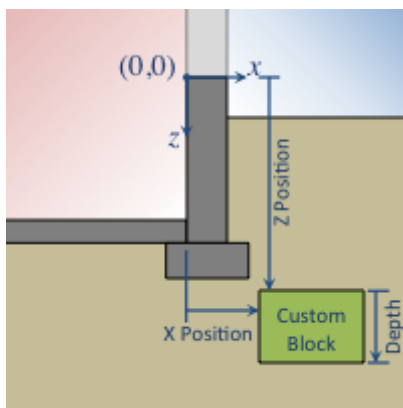
If custom blocks are to be associated with this Kiva foundation object then select the number of them here. Any value from 0 to 10 is allowed.

Custom block <x> material

Select the material associated with a custom block in the two-dimensional foundation context.

If two or more custom blocks overlap, the final properties are determined by the higher block number (e.g., Custom Block 4 in the input object supersedes properties defined by Custom Block 2). All custom blocks properties are superseded by the elements shown in the figure below.

The following fields defines the placement of this material within Kiva's two-dimensional context and are illustrated in the Figure below.



Placement of a custom block

Note: The Thickness property of the selected material will be overwritten by the **Custom block <x> width** setting entered below.

Custom block <x> width

Horizontal width of the custom block (in m or ft).

Note: The value entered here is used to define the **Thickness** property of the simulated **Custom block <x> material**. See above.

Custom block <x> depth

Top-to-bottom dimension of the block downward (in m or ft). The default is the depth from the wall top to the top of the slab to facilitate interior constructions.

Custom block <x> x Position

Position outward (+) or inward (-) relative to the foundation wall (in m or ft).

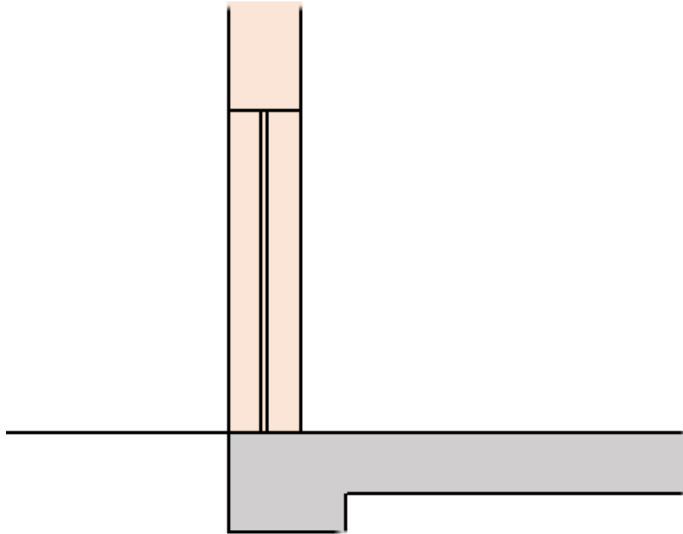
Custom block <x> z Position

Position downward (+) relative to the foundation wall top (in m or ft).

Kiva Foundation Example Cases

The examples below provide details on how to set up Kiva Foundations for some common cases.

Raft Slab



This example shows how to model a simple raft slab case.

1. The main slab including any continuous insulation layer is modelled within the ground floor construction element.
2. There is no above grade footing wall so **Wall height above grade** is set to zero.
3. In this example the raft beam extends 0.2m below the base of the slab so the **Wall depth below slab** should be entered 0.2m.
4. The footing wall construction thickness defines the width of the footing wall.
5. There is no foundation footing in this example.

Edit Kiva Foundation - Kiva foundation

Kiva Foundation

General

General

Name Kiva foundation

Initial Indoor Air Temperature >>

Interior Horizontal Insulation >>

Interior Vertical Insulation >>

Exterior Horizontal Insulation >>

Exterior Vertical Insulation >>

Wall

Wall height above grade (m) 0

Wall depth below slab (m) 0.2

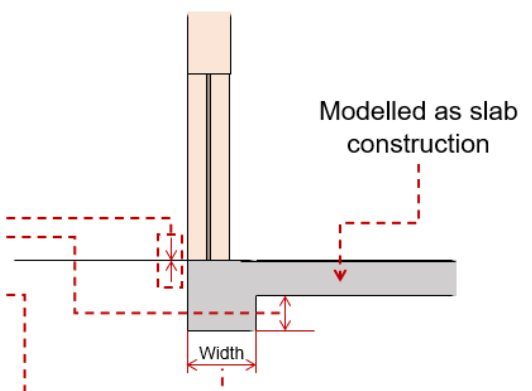
Footing Wall

Footing wall construction Kiva Foundation Footing Wall

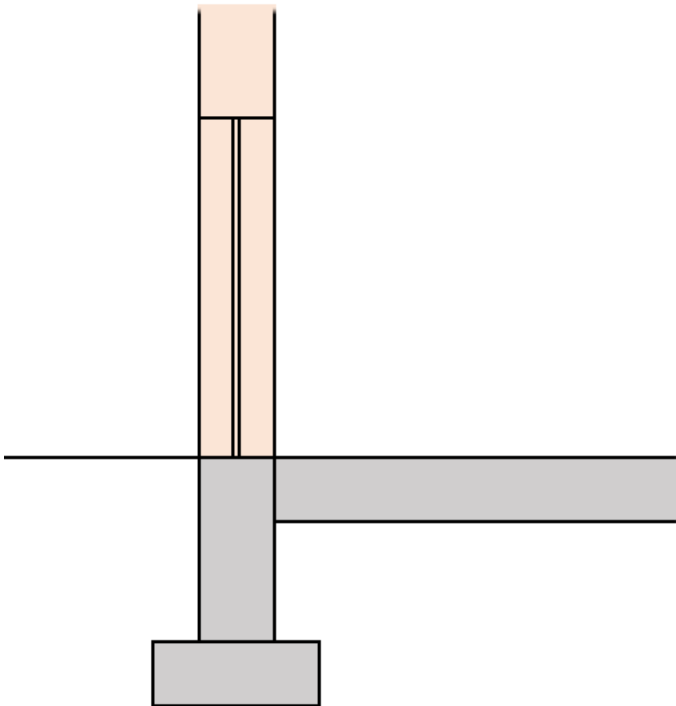
☐ Include foundation footing

Custom Blocks >>

Model data Help Cancel OK



Foundation Footings



In this example we will be modelling a traditional foundation with footing without any additional perimeter insulation. The top of the slab is level with the ground level.

1. The main slab, including its insulation, is modelled within the ground floor construction element.
2. Because the top of the slab is level with the top of the ground, the **Wall height above grade** is set to zero.
3. The footing wall extends below the base of the slab by 0.5m so the **Wall depth below slab** should be entered as 0.5m.
4. The footing wall construction thickness defines the width of the footing wall.
5. As there is additional concrete foundation footing below the footing wall, its details are defined separately with Footing width and Footing depth set to 0.5m and 0.2m respectively.

Edit Kiva Foundation - Kiva foundation

Kiva Foundation

General

General

Name **Kiva foundation**

Initial Indoor Air Temperature >>

Interior Horizontal Insulation >>

Interior Vertical Insulation >>

Exterior Horizontal Insulation >>

Exterior Vertical Insulation >>

Wall

Wall height above grade (m) 0

Wall depth below slab (m) 0.5

Footing Wall

☒ Footing wall construction Kiva Foundation Footing Wall

☒ Include foundation footing

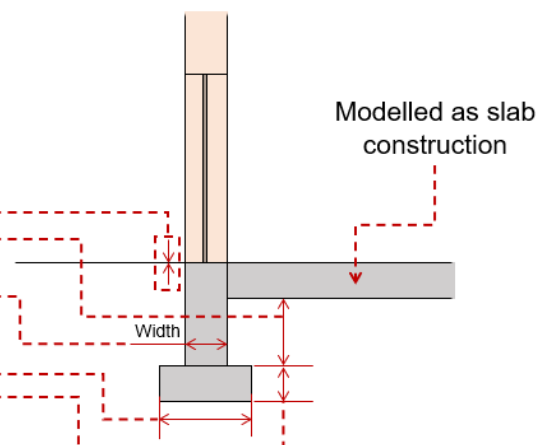
☒ Footing material 12 in. heavyweight concrete

Footing width (m) 0.5

Footing depth (m) 0.2

Custom Blocks >>

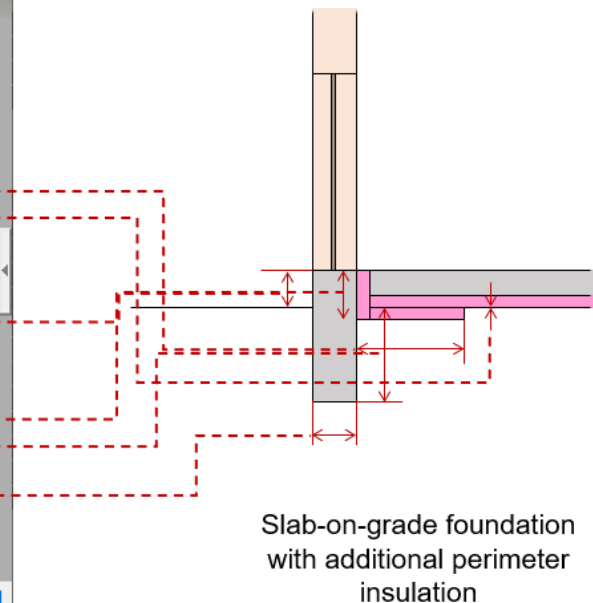
Model data Help Cancel OK



Slab-on-Grade Foundation With Additional Perimeter Insulation

In this example there is a slab with continuous insulation, and additional perimeter insulation elements, and a footing wall. Firstly, the continuous insulation will be modelled in the slab construction and will not there appear in the foundation data.

1. There is 0.5m of interior horizontal insulation around the perimeter of the foundation. It is measured from wall interior to the edge of insulation and is defined in the model by entering the Interior horizontal insulation **Width** as 0.5m.
2. The interior horizontal insulation is positioned directly below the slab and so the **Depth** is entered as 0.
3. There is also 0.25m depth of Interior vertical insulation around the perimeter of the foundation. It is measured from the top of wall to the edge of the insulation. So enter 0.25m as the Interior vertical insulation **Depth**.
4. The top of the footing wall is 0.2m above grade as the slab is on grade and goes into the ground, therefore the **Wall height above grade** should be entered as 0.2m.
5. The footing wall extends below the base of the slab by 0.5m so the **Wall depth below slab** should be entered as 0.5m.
6. The footing wall construction thickness defines the width of the footing wall.
7. There is no foundation footing in this example.



Kiva Foundation Settings

When the [Ground modelling method](#) has been set to **2-Kiva Basic** or **4-Kiva Full**, the Kiva foundation settings component [selected at site level](#) defines the soil thermal and surface properties, the extent of the domain and some calculation options that are to be applied across all Kiva foundation calculations. Just one Kiva foundation settings object is required for a simulation.

General

Name

Enter a unique name for this Kiva foundation settings component.

Soil Thermal Properties

Soil conductivity

The thermal conductivity of the soil (in W/m-K or Btu-in/h-ft²-°F).

Soil density

The bulk density of the soil (in kg/m³ or lb/ft³).

Soil specific heat

The specific heat of the soil (in J/kg-K or Btu/lb-°F).

Surface Properties

Enter the properties of the ground surface.

Ground solar absorptivity

Solar absorptivity of the exterior grade surface.

Ground thermal absorptivity

Long-wave absorptivity (emissivity) of the exterior grade surface.

Ground surface roughness

The relief roughness of the exterior ground surface in m. Some typical surface roughness for various surface types are shown in the Table below.

Example Surface	Roughness [m]
Concrete	0.002
Brick	0.003
Soil	0.005
Gravel	0.012
Grass	0.030

Extent

Far-field width

The distance from the wall interior to the zero horizontal heat flux (i.e., adiabatic) far-field boundary (in m or ft). This distance represents either the distance halfway between this foundation and a similar foundation of a neighbouring building, or a distance adequately far from the foundation such that it is isolated from the effects of the boundary (typically ≥ 40 m).

Deep-ground depth

The distance from the exterior grade to the deep ground boundary (in m or ft). This distance represents either the distance to the ground water level, or a distance adequately far from the foundation such that it is isolated from the effects of the boundary (typically ≥ 40 m).

If you know the water table depth, then it is generally more accurate to model the boundary at that depth using the water table temperature.

The recommended setting for **Deep-ground depth** is 40m.

The **Deep-ground depth** setting can be set to **Autocalculate** in which case Kiva will use a correlation to estimate how deep the ground water is based on the elevation of your location. If you use this option you should be aware that ground heat transfer can be very sensitive to the location of the water table which is much more geospatially variable than the correlation represents (e.g. elevation and water table depth at your weather station could be very different from your actual building site).

Options

Minimum cell dimension

The minimum cell dimension (in m or ft) used in the Kiva discretisation. Default: 0.02 m.

Maximum cell growth coefficient

The maximum ratio of growth between neighbouring cells in the direction away from the near-field area of interest. Default: 1.50.

Simulation timestep

This field allows the user to choose whether to calculate foundation loads on the zone timestep or at hourly intervals. Hourly intervals will allow for less overall computation time, but with less accuracy. Choices are:

- **1-Hourly** and
- **2-Timestep**

Deep-ground boundary condition

Defines the type of boundary condition to apply at the Deep-ground depth. Options are:

- **1-Autoselect** which applies either of the 2 boundary conditions below depending on the elevation of the building site (Williams and Williamson, 1989) as shown in the equation below.
- **2-Zero flux** in which case the water table is considered to be deep enough that it doesn't impact the heat loss from the building and Kiva applies a zero vertical heat flux (i.e. adiabatic) boundary condition. This is the recommended value.
- **3-Ground water** which applies a constant temperature boundary condition, with a temperature equal to the average outdoor air dry-bulb temperature from the environment(s).

Williams and Williamson, 1989 provide this simple correlation to help estimate the depth of the water table, d_{wt} from the [site elevation above sea level](#), d_{elev} :

$$d_{wt} = 0.1022 \cdot d_{elev}$$

If $d_{wt} \leq 40$ m., the **3-Ground water** boundary is applied, otherwise a **2-Zero flux** boundary is applied at 40 m. If the calculated ground water depth is shallower than any element of the foundation construction, then the **3-Ground water** option is applied at 1m below the lowest element. The default is **1-Autoselect**.

Moisture Transfer Data

The Moisture transfer component provides data for either HAMT or EMPD or both. It can be selected from the [Materials](#) dialog from under the [Moisture Transfer](#) header at the bottom of the General tab.

DesignBuilder is provided with a database of moisture transfer properties for a small range of representative materials. The database includes properties for both HAMT and EMPD calculation methods. The HAMT database is derived from [IEA Annex 24 c.1996](#).

For details on the HAMT algorithms used within EnergyPlus see the [Combined Heat and Moisture Transfer \(HAMT\) Model](#) section in the Engineering Reference.

Note: Moisture transfer data provided in the literature tends to be approximate, much more so than for thermal properties. This is due to large variations in these properties that have been measured in lab tests.

General

Name

Enter a unique name for this set of moisture transfer data.

Category

The **<All>** category is presently the only option.

Moisture transfer simulation method

You can choose to enter data for EMPD and/or HAMT. Select from:

- **1-EMPD** - the component includes data for EMPD only.
- **2-HAMT** - the component includes data for HAMT only.
- **3-EMPD and HAMT** - data for both EMPD and HAMT is included in the component.

When either of the **1-EMPD** or **3-EMPD and HAMT** options are selected the [EMPD tab](#) will be shown. If either of the **2-HAMT** or **3-EMPD and HAMT** options are selected the following tabs will be shown:

- [HAMT Settings](#)
- [Sorption Isotherm](#)
- [Suction](#)
- [Redistribution](#)
- [Diffusion](#)
- [Thermal Conductivity](#)

EMPD

The Effective Moisture Penetration Depth (EMPD) moisture model will be used when the appropriate EMPD moisture materials are specified and the Moisture transfer simulation method is one the EMPD options. When the EMPD option is being used, data on this tab is used to describe the moisture material properties that are used in the **5-Moisture Penetration Depth Conduction Transfer Function** solution algorithm. The EMPD algorithm is a simplified, lumped moisture model that simulates moisture storage and release from interior surfaces.

The model uses convective mass transfer coefficients that are determined by existing heat and mass transfer relationships, e.g. the Lewis relation. The EMPD model includes two fictitious layers of material with uniform moisture content: a surface layer, which accounts for short-term moisture buffering, and a deep layer, which accounts for more slowly responding moisture buffering. The model calculates the moisture transfer between the air and the surface layer and between the surface layer and the deep layer. This moisture transfer impacts the zone humidity, and also impacts the zone temperature through latent-to-sensible conversion from the heat of adsorption.

Water vapour diffusion resistance factor

The vapour diffusion resistance factor is the resistance to water vapour diffusion relative to the resistance to water vapour diffusion in stagnant air. In other words, equals 1 for air, and is generally greater than 1 for building materials. The equation for μ is:

$$\mu = \frac{\delta_{perm,air}}{\delta_{perm}}$$

where $\delta_{perm,air}$ is the permeability of water vapour in air [kg/m-s-Pa], and δ_{perm} is the permeability of water vapour in the material. The permeability of water vapour in air can be estimated as:

$$\delta_{perm,air} = \frac{2 \times 10^{-7} \cdot T^{0.81}}{P_{ambient}}$$

where T is the temperature [C] and $P_{ambient}$ the ambient atmospheric pressure [Pa].

Effective moisture depth

This field is used to enter the effective moisture penetration depth of the material layer. Units for this parameter are m or ft.

Surface layer penetration depth

The **Surface layer penetration depth** is the fictitious thickness of the surface layer (in m or in), and is used to calculate the volume of material that participates in short-term moisture transfer and storage. This layer has a uniform moisture content, and can be considered a lumped-capacitance. The penetration depth is based on the amount of material that interacts with the zone air when subject to a cyclic relative humidity variation. It also impacts the mass transfer resistance between the zone air and this layer, with thinner depths resulting in lower mass transfer resistances (ref: Effective Moisture Penetration Depth (EMPD) Model in the [Engineering Reference](#)). For this reason very small values can lead to instabilities depending on the timestep. The surface penetration depth can be estimated with the following equation:

$$d_{EMPD,surf} = \sqrt{\frac{\delta_{perm} P_{sat} \tau_{surf}}{\rho_{material} \frac{du}{d\phi} \pi}}$$

where:

δ_{perm} = water vapor permeability in the material, kg/m-s-Pa (see Vapour diffusion resistance factor above)

P_{sat} = saturated vapor pressure at some nominal temperature, Pa

τ_{surf} = cycle period of typical RH variations, s. 24 hours (86,400 s) is often used.

$\rho_{material}$ = dry density of material, kg/m³

$du/d\phi$ = slope of moisture sorption curve, $ab\phi^{b-1} + cd\phi^{d-1}$

If this field is set to **autocalculate**, the above equation will be used to calculate the surface layer penetration depth assuming a surf of 24 hours. To use a period different than 24 hours, the equation above can be used to calculate the penetration depth based on a different value of surf. The penetration depth can also be entered as an empirical value, as in Woods and Winkler, 2016. If calculating $d_{EMPD,surf}$, the assumed value of τ_{surf} should not be less than 4x the simulation timestep to ensure an accurate and stable solution.

Deep layer penetration depth

The **Deep layer penetration depth** is the fictitious thickness of the deep layer (in m or in), and is used to calculate the volume of material that participates in long-term moisture transfer and storage. This layer has a uniform moisture content, and can be considered a lumped-capacitance. The deep penetration depth is based on the amount of material that interacts with the surface layer when subject to cyclic relative humidity variation. The deep penetration depth can be estimated with the following equation:

$$d_{EMPD,deep} = \sqrt{\frac{\delta_{perm} P_{sat} \tau_{deep}}{\rho_{material} \frac{du}{d\phi} \pi}}$$

Each term is the same as for the surface layer, except that the cycle period is different. This is usually on the order of weeks for the deep layer. If this field is set to **autocalculate**, the above equation will be used to calculate the deep layer penetration depth assuming a deep of three weeks. To use a period different than 3 weeks, the equation above can be used to calculate the penetration depth based on a different value of deep. The penetration depth can also be entered as an empirical value, as in Woods and Winkler, 2016.

Coating layer thickness

The Coating Layer Thickness (in m or in) adds an additional resistance between the surface layer and the zone and represents a thin coating, such as paint, plaster, or other wall coverings. This input is optional, and an input of zero implies no coating.

Coating layer water vapour diffusion resistance factor

The vapour diffusion resistance factor of the coating is the coating's resistance to water vapour diffusion relative to the resistance to water vapour diffusion in stagnant air (see Vapour diffusion resistance factor section above). An input of zero implies no coating.

Moisture Equilibrium Constants

The next four fields, coefficients "a", "b", "c", and "d", help define the sorption isotherm curve used for building materials under equilibrium conditions. They are used to define the relationship between the material's moisture content and the surface air relative humidity (ref: Effective Moisture Penetration Depth (EMPD) Model in the [Engineering Reference](#)):

$$U = a\phi^b + c\phi^d$$

where:

a,b,c,d = Coefficients to define the relationship between the material's moisture content and the surface air relative humidity

U = Moisture content defined as the mass fraction of water contained in a material (kg/kg or lb/lb)

ϕ = Surface air relative humidity [0 to 1].

The next four fields are dimensionless coefficients:

Coefficient a

Coefficient b

Coefficient c

Coefficient d

HAMT Settings

Porosity

The porosity of a material is the maximum fraction, by volume, of a material that can be taken up with water. The units are m³/m³ or ft³/ft³.

Water content

Enter the initial water content ratio (in kg/kg or lb/lb). For the HAMT solution algorithm the initial water content is assumed to be distributed evenly through the depth of the material.

Sorption Isotherm

The Sorption Isotherm data relates the moisture, or water content of a material with the relative humidity (RH). The water content is expected to increase as relative humidity increases, starting at zero content at 0.0 RH fraction and reaching a maximum, defined by the porosity, at 1.0 RH fraction, which corresponds to 100% relative humidity. Relative humidities are entered as fraction ranging from 0.0 to 1.0. These two extremes (0.0 and 1.0) are automatically set by the HAMT solution. However, if they are entered they will be used as extra data points. Data should be provided with increasing RH and moisture content up to as high an RH as possible to provide a stable solution.

One possible reason for the following error message may be that a material has a very rapid increase in water content for a small change in RH, which can happen if the last entered water content point is at a low RH and the material has a very high porosity.

** Warning ** HeatAndMoistureTransfer: Large Latent Heat for Surface ROOF

Number of data coordinates

A maximum of 25 coordinates can be specified.

Data Coordinate n

Relative humidity fraction x

The relative humidity of the nth coordinate. The relative humidity is entered as fraction, not in percent.

Moisture content

The Moisture content of the nth coordinate. The units are kg/m³ or lb/ft³

Suction

The suction data relates the liquid transport coefficient, under suction, to the water content of a material. A data point at zero water content is required. The liquid transport coefficient at the highest entered water content value is used for all liquid transport coefficient values above this water content. These coefficients are used by HAMT when the rain flag is set in the weather file.

Number of Suction points

A maximum of 25 points can be specified.

Suction Pair n

Moisture content n

The moisture content of the xth point. The units are kg/m³ or lb/ft³.

Liquid transport coefficient n

The Liquid Transport Coefficient of the xth point. The units are m²/s or ft²/s.

Redistribution

The redistribution data relates the liquid transport coefficient to the water content of a material under normal conditions. A data point at zero water content is required. The liquid transport coefficient at the highest

entered water content value is used for all liquid transport coefficient values above this water content. These coefficients are used by the Heat and Moisture Transfer algorithm when the rain flag is NOT set in the weather file.

Number of Redistribution Points

A maximum of 25 points can be specified.

Redistribution Point n

Moisture content

The moisture content of the nth point. The units are kg/m³ or lb/ft³.

Liquid transport coefficient

The Liquid transport coefficient of the nth point. The units are m²/s or ft²/s.

Diffusion

The MU data relates the vapor diffusion resistance factor (dimensionless) to the relative humidity as fraction (RH). A data point at zero RH is required. The vapor diffusion resistance factor at the highest entered relative humidity (RH) value is used for all vapor diffusion resistance factor values above this RH. The relative humidity maximum value in fraction is 1.0.

Number of data pairs

A maximum of 25 pairs can be specified.

Data Pair n

Relative humidity fraction

The moisture content of the nth pair. The relative humidity is entered as fraction, not in percent.

Vapor diffusion resistance factor

The Liquid transport coefficient of the nth pair.

Thermal Conductivity

The thermal data relates the thermal conductivity of a material to the moisture or water content. A data point at zero water content is required. The thermal conductivity at the highest entered water content value is used for all thermal conductivity values above this water content.

Number of thermal coordinates

A maximum of 25 coordinates can be specified.

Thermal Coordinate n

Moisture content

The moisture content of the nth coordinate. The units are kg/m³ or lb/ft³.

Thermal conductivity

The Thermal conductivity of the nth coordinate. The units are W/m-K or Btu-in/h-ft-°F.

HAMT Limitations

- HAMT is incompatible with linear thermal bridging because it does not support use of R-value materials used for bridging.

Fuel Emission Factors

Fuel emission factors are used as multipliers to calculate the emission of pollutants from fuel energy consumption. This dialog, accessed from the [Regions](#) dialog, allows you to define the pollution caused by each fuel treated by EnergyPlus. Data for each fuel is accessed on its own tab.

General

On the **General** tab you can enter a name for this set of fuel emission factors and a definition method to define whether fixed or time-varying factors will be entered.

Name

Enter the name of the set of fuel factors. This might be the region covered by this set of emission factors plus the date the data applies to.

Definition method

Emission factors can be defined either as constant or time-varying values:

- **1-Constant** where fixed fuel emission factors are provided for each fuel.
- **2-Scheduled** where one or more fuel emission factors varies with time. This option might be used when for example there are seasonal variations (e.g. more hydro-electric generation in the mix during winter).

Electricity, Natural Gas, Fuel Oil 1 Etc Tabs

To the right of the **General** tab are a set of 10 further tabs, one for each type of fuel supported:

- Electricity
- Natural Gas
- Fuel Oil 1
- Fuel Oil 2
- Coal
- Gasoline
- Propane
- Diesel
- Other Fuel 1
- Other Fuel 2

Fuel Emission Factors Data											
General	Electricity	Natural Gas	Fuel Oil 1	Fuel Oil 2	Coal	Gasoline	Propane	Diesel	Other Fuel 1	Other Fuel 2	
Electricity											
Source	Grid supplied electricity NCM 2013										
Units of measure	1-Mass										
Energy per unit factor	0.000000000000000										
Source energy factor	3.070000000000000										
Emission Factors											
CO2 (g/MJ)	144.16670000000000										
CO (g/MJ)	0.042061600000000										
CH4 (g/MJ)	0.001398580000000										
NOx (g/MJ)	0.410753000000000										
NO2 (g/MJ)	0.002419160000000										
SO2 (g/MJ)	0.865731000000000										
PM (g/MJ)	0.029582700000000										
PM10 (g/MJ)	0.018045000000000										
PM25 (g/MJ)	0.011537700000000										
NH35 (g/MJ)	0.001108370000000										
NM VOC (g/MJ)	0.003723320000000										
Hg (g/MJ)	0.00000336414000										
Pb (g/MJ)	0.000000000000000										
Water (g/MJ)	2.100740000000000										
Nuclear high level (g/MJ)	0.000000000000000										
Nuclear low level (g/MJ)	0.000000000000000										

On each of these tabs the following data can be entered.

Source

Enter text describing the source of the data.

Source energy factor

EnergyPlus uses the concept of "Source Energy" to account for the energy consumed in generating and delivering fuel to a building. This helps to provide a more complete picture of a building's environmental impact than just the energy consumed on-site.

What is Source Energy?

The source energy defines the full upstream energy costs associated with the energy consumed downstream in the building, helping to look beyond the meter and consider the full life cycle of energy delivery. When electricity is used in a building, the power plant that generated that electricity had to burn fuel such as coal or natural gas to produce it. Then, some energy is lost when the electricity travels through transmission lines to the building. Source energy accounts for such losses. It can be thought of as the energy at the source (the power plant, the well where natural gas is extracted etc.) that was required to deliver the energy eventually used in the building.

The **Source energy factor** is a multiplier that tells you how many units of energy had to be consumed at the original source for every one unit of energy delivered to the building.

For example:

- **Electricity:** When you use 1 kWh of electricity in your building, the power plant might have had to burn enough fuel to generate, say, 3 kWh of primary energy. This accounts for:
 - **Generation Losses:** The inefficiency of converting fuel into electricity at the power plant. A significant amount of energy is lost as heat.
 - **Transmission and Distribution Losses:** Energy lost as heat when electricity travels through wires from the power plant to the building.

- **Fuel Extraction and Processing:** Energy used to extract the coal or natural gas, transport it to the power plant, and process it.

So, for electricity, a typical source-site ratio might be around 3.0. This means for every 1 unit of electricity used in the building, 3 units of energy were consumed at the source.

- **Natural Gas:** Natural gas has a much lower source-site ratio, often close to 1.1. It's usually lower because:
 - It's a "primary" fuel delivered directly to the site with minimal processing and transportation losses compared to electricity generation.
 - The primary losses are related to extraction, processing, and pipeline transportation.
- **Other Fuels:** Other fuels like fuel oil or district heating/cooling also have their own source-site ratios, reflecting the energy needed for their production and delivery.

If using the default [1-Constant Definition method](#) then this fixed value is multiplied by the fuel consumption to compute the source energy contribution for the fuel.

Note: If the TDV files are used the value should be 0.293 for electricity and 0.01 for natural gas to account for the units used in the files. Units of J/J.

Source energy schedule

If using the [2-Scheduled Definition method](#) then the schedule specified here represents the source energy conversion factor.

Emission Factors

Enter the emission factors for each pollutant (in g/MJ).

CO₂

The environmental impact coefficient for the fuel for calculating the mass of carbon dioxide (CO₂) released into the atmosphere. The units are g/MJ. Carbon dioxide gas is naturally produced by animals during respiration and through decay of biomass, and used by plants during photosynthesis. Although it only constitutes 0.04 percent of the atmosphere, it is one of the most important greenhouse gases. The combustion of fossil fuels is increasing carbon dioxide concentrations in the atmosphere, which contributes to global warming.

If using the [2-Scheduled Definition method](#) then select the schedule which defines the emission factor using SI units.

CO

The environmental impact coefficient for the fuel for calculating the mass of carbon monoxide (CO) released into the atmosphere. The units are g/MJ. Carbon monoxide is a colorless, odorless and poisonous gas produced by incomplete fossil fuel combustion. Carbon monoxide combines with the haemoglobin of human beings, reducing its oxygen carrying capacity, with effects harmful to human beings.

If using the [2-Scheduled Definition method](#) then select the schedule which defines the emission factor using SI units.

CH₄

The environmental impact coefficient for the fuel for calculating the mass of methane (CH₄) released into the atmosphere. The units are g/MJ. Methane is a colorless, nonpoisonous, flammable gas created by anaerobic decomposition of organic compounds and is one of the more potent greenhouse gases. A major component of natural gas used in the home.

If using the [2-Scheduled Definition method](#) then select the schedule which defines the emission factor using SI units.

NO_x

The environmental impact coefficient for the fuel for calculating the mass of nitrogen oxides (NO_x) released into the atmosphere. The units are g/MJ. Nitrogen oxides refers to nitric oxide gas (NO) and nitrogen dioxide gas (NO₂) and many other gaseous oxides containing nitrogen. The main source of these gases in urban areas are motor vehicle exhaust and indoor gas stoves and kerosene heaters. The brown haze sometimes seen over cities is mainly nitrogen oxides. These gases are also partly responsible for the generation of ozone, which is produced when nitrogen oxides react with other chemicals in the presence of sunlight. Exposure to high levels of nitrogen dioxide can interfere with the ability of blood to carry oxygen, leading to dizziness and shortness of breath. Prolonged exposure can lead to respiratory failure.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

N₂O

The environmental impact coefficient for the fuel for calculating the mass of nitrous oxide (N₂O) released into the atmosphere. The units are g/MJ. Relatively inert oxide of nitrogen produced as a result of microbial action in the soil, use of fertilizers containing nitrogen, burning of timber and coal, chemical industry, and so forth. This nitrogen compound may contribute to greenhouse and ozone-depleting effects.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

SO₂

The environmental impact coefficient for the fuel for calculating the mass of sulfur dioxide (SO₂) released into the atmosphere. The units are in grams per MegaJoule. Sulfur dioxide gas is formed when fuel containing sulfur, such as coal and oil, is burned, and when gasoline is extracted from oil, or metals are extracted from ore. Sulfur dioxide reacts with other chemicals in the air to form tiny sulfate particles, associated with increased respiratory symptoms and disease, difficulty in breathing, and premature death. Sulfur dioxide and nitrogen oxides react with other substances in the air to form acids, which fall to earth as rain, fog, snow, or dry particles. Acid rain damages forests and crops, changes the makeup of soil, and makes lakes and streams acidic and unsuitable for fish. Sulfur dioxide accelerates the decay of building materials and paints.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

PM

The environmental impact coefficient for the fuel for calculating the mass of particulate matter (PM) released into the atmosphere. The units are g/MJ. PM is the sum of all particular matter emitted, including PM₁₀ and PM_{2.5}. Particulate matter, or PM, are particles found in the air, including dust, dirt, soot, smoke, and liquid droplets, which can be suspended in the air for long periods of time. Some particles are large or dark enough to be seen as soot or smoke. Others are so small that individually they can only be detected with an electron microscope. Breathing particulate matter is linked to significant respiratory health problems.

PM₁₀

The environmental impact coefficient for the fuel for calculating the mass of particulate matter 10 (PM₁₀) released into the atmosphere. The units are g/MJ. PM₁₀, includes particles with an aerodynamic diameter of less than 10 microns. These smaller particles are most likely responsible for the adverse health effects on humans because particles so small can reach the thoracic or lower regions of the respiratory tract.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

PM_{2.5}

The environmental impact coefficient for the fuel for calculating the mass of particulate matter 2.5 (PM_{2.5}) released into the atmosphere. The units are g/MJ. EPA's national air quality standards for fine particles, also known as "PM_{2.5} standards," are levels allowed in the outdoor air for particulate matter 2.5 microns in diameter or smaller. EPA issued the PM_{2.5} standards in 1997 to protect human health and the environment.

Studies have linked increased exposure to PM_{2.5} to increases in premature death as well as a range of serious respiratory and cardiovascular effects.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

NH₃

The Environmental impact coefficient for the fuel for calculating the mass of ammonia (NH₃) released into the atmosphere. The units are g/MJ. Ammonia reacts with nitrogen and sulfur compounds in the atmosphere, mainly nitric and sulfuric acids, to form particulate matter.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

NM VOC

The Environmental impact coefficient for the fuel for calculating the mass of non-methane volatile organic compounds (NMVOC) released into the atmosphere. The units are g/MJ. Non-methane volatile organic compounds (NMVOC), which include propane, butane, and ethane, are emitted primarily from transportation, industrial processes, and non-industrial consumption of organic solvents. Volatile organic compounds react with nitrogen oxides in the atmosphere to form ozone.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

Hg

The Environmental impact coefficient for the fuel for calculating the mass of mercury (Hg) released into the atmosphere. The units are g/MJ. This heavy metal can accumulate in the environment and is highly toxic if breathed or swallowed. In the US, primary sources of mercury air emissions are coal-fired power plants.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

Pb

The Environmental impact coefficient for the fuel for calculating the mass of lead (Pb) released into the atmosphere. The units are g/MJ. A heavy metal that is hazardous to health if breathed or swallowed. Its use in gasoline, paints, and plumbing compounds has been sharply restricted or eliminated by federal laws and regulations.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

Water

The environmental impact coefficient for the fuel for calculating the volume of water (H₂O) consumed or evaporated in the generation of electricity. The units are g/MJ. This is the water consumed in the production of the energy, ie. electricity off-site evaporated in cooling towers or scrubbers, or in the production or processing of the fuel itself, i.e., refinery for gasoline or diesel.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

Nuclear high level

The environmental impact coefficient for the fuel for calculating the mass of high-level nuclear waste, removed as spent nuclear fuel from a nuclear reactor once it no longer is efficient at powering the reactor.. The units are g/MJ. Once a year, approximately one-third of nuclear fuel is replaced with new fuel. This used fuel is called spent nuclear fuel and is highly radioactive; containing plutonium and other radionuclides. Although there is little information on quantities of high-level nuclear waste, a few utilities are beginning to publish this information.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

Nuclear low level

The environmental impact coefficient for the fuel for calculating the volume of low-level nuclear waste, removed from a nuclear reactor after radiation contamination. The units are g/MJ or a volume measurement. Low-level waste can come from nuclear reactors or other users of radioactive material, like hospitals or research institutes. Low-level waste is less hazardous than high-level waste.

If using the **2-Scheduled Definition method** then select the schedule which defines the emission factor using SI units.

Emissions data for the other fuels is entered in the same way as for electricity.

Fuel Emissions Library Data

DesignBuilder is provided with a built-in library of fuel emissions data for a range of locations. This mostly comes from the data provided with EnergyPlus which includes specific emissions rates for each US state and also a US National average dataset in:

- ElectricityUSAEnvironmentalImpactFactors.idf
- FossilFuelEnvironmentalImpactFactors.idf

Fuel emission factors for the UK are provided from the 2013 UK National Calculation Methodology (NCM).

For other global locations, a "generic" data set is used based on the US average National dataset:
United States 1999 national average electricity emissions factors based on eGRID, 1605, AirData

United States Water Emission Fuel Factors are the combined thermoelectric and hydroelectric weighted averages from:

- Torcellini, Paul; Long, Nicholas; Judkoff, Ron; "Consumptive Water Use for U.S. Power Production"; NREL Report No. TP-550-33905. Golden, CO; 2003;
<http://www.nrel.gov/docs/fy04osti/33905.pdf>;

or:

- Torcellini, Paul; Long, Nicholas; Judkoff, Ron; "Consumptive Water Use for U.S. Power Production"; ASHRAE Transactions 2003, Vol 110, Part 1. Atlanta, GA; January 2004;

Note: If more specific fuel emissions data is available for your project location, you should use this instead of the generic library data provided.

Outputs

To obtain detailed outputs of pollution caused by fuel consumption use the [check boxes](#) on the Region tab at Site level.

The screenshot shows the 'Site' dialog box with the 'Region' tab active. The 'Legislative Region' dropdown is set to 'England and Wales'. Below it, the 'Include Fuel Emissions' section has a list of fuel types with checkboxes. 'Electricity' and 'Natural gas' are checked, while all other fuel types are unchecked.

Horizon Component

This component is used to define a horizon to include the effects of terrain and/or more distant tall buildings on the shading of the simulated building. It can be selected at when at site level using the [Horizon component](#) browse control on the Location model data tab, under the **Site Details > Horizon** headers.

A horizon elevation angle is defined for each direction (orientation) around the building.

Horizon

Increment

The Increment angle indicates the level of granularity you wish to use for defining different horizon elevations for each orientation. The lower the increment the more detailed the horizon. Select an increment from the list of options:

- 90 - define only 4 horizon elevation values, one for each cardinal direction.
- 45
- 30
- 22.5
- 15
- 12
- 10 - define 36 different elevation values at 10 degree orientation intervals.

For most accurate modelling select 10° increments.

Distance to horizon

DesignBuilder models the horizon by adding a series of vertical rectangular shading surfaces to the EnergyPlus input file using the increment and elevation data defined on this dialog. These shading surfaces are added in a circle around the model centre at a radius defined by this **Distance to horizon** setting. The default distance is 100 m or 328.08 ft.

Elevation for each Orientation

Enter the horizon elevation for each orientation. Orientation angles are defined as clockwise East from North, so the elevation for 90° represents the elevation of the horizon when looking East for example.

The Increment set above defines the list of orientations displayed.

Currency

DesignBuilder is provided with a list of all the major international currencies. Each item in the list is a currency component which defines some key parameters associated with the currency, including the symbol, 3-letter code and conversion rate relative to GBP.

General

Currency symbol

The currency symbol is used as a prefix when reporting costs within EnergyPlus cost outputs.

Date

The date when the currency was added to the library.

3-letter currency code

This is the standard 3-letter code used to define each currency, such as USD, GBP, EUR etc.

This 3 letter code is used in DesignBuilder's [internal construction cost reports](#).

Conversion rate

The conversion rate used to convert from this currency to the internal "native " currency of GBP used to store all cost data. Units are GBP/Unit.

Local currency = GBP / Conversion rate

Templates

DesignBuilder templates allow you to quickly load data into your model in bulk. If you often work with similar types of buildings you may find it useful to create your own templates. This will allow you to load the same data into any models you work with in the future.

DesignBuilder Templates are databases of typical generic data. The following templates are available:

- [Activity templates](#)
- [Construction templates](#)
- [Glazing templates](#)
- [Facade templates](#)
- "Lighting Templates" on page 1498
- [HVAC templates](#)
- [Location Templates](#)
- [Crack templates](#)
- [Legislative region templates](#)
- [Energy code templates](#)
- [Sector templates](#)
- [Time zones templates](#)
- [Wind Pressure coefficient templates](#)

Library Templates are loaded into the model at the time the dsb file is created and from then on, they are associated specifically with this model and so are known as **Model Templates**. Any changes made to the Library Templates do not affect existing building models because they reference their own set of Model Templates. The latest set of library templates can be loaded into a model using the Import library components and templates to model command.

Also templates are used for loading data into models, so ***a change to a template will not affect the model until the data is loaded.***

Note: Templates are loaded to the model in the same way that components are so that any edit to Model template data will be associated with that model only.

See also the [Data templates](#) tutorial

Managing template data

You can edit and move template data around in various ways:

1. Add, Edit, Delete, [Import](#) and [Export](#) Template library data from the [Opening Screen - Template Libraries](#) tab.
2. If a model file is loaded you can edit Model templates from the Templates tab in the Navigator panel ([Learning mode](#) switched off) or by using the Data management panel on the right of the screen ([Learning mode](#) switched on).
3. In either case, **to add a template**, open up the template type you wish to define by clicking on the + to the left of the template icon, then click on the category of template you wish to define. Now click on the Green '+' to add a template of the selected category.
4. Import/export Model template using the **File > Export > Library data** menu command to create a .ddf file with the .dsb model file open.

5. To import the previously exported Model template data to the Template library close the .dsb file and [import](#) the .ddf file to the library.
6. Load the latest library templates into your model using the **Import library components and templates to model** command from the Tools menu when a model is loaded. Any new data that has been added to the library since the model file was created will be imported to the model. Also if there have been any changes to library template data, this new data will overwrite the original data.

Note: Any template you add while a model is open will be added to Model templates and not to the Template library. It will only be added to the library and hence to new models if you export the new component model data in .ddf form and import the .ddf file to the library before creating the new model.

Note: As well as importing DDF files created directly by DesignBuilder more advanced users can also create their own component and template data sets (outside DesignBuilder) for importing. See the [Importing Custom Templates and Components](#) topic for more information on the process involved.

Loading to Model Data

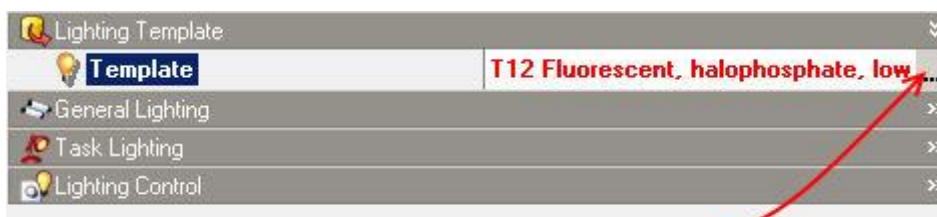
Each tab in the Model data has the name of the template last used to load the data at the top (shown in bold). At the top of the HVAC tab is the HVAC Template, at the top of the Lighting tab is the Lighting Template etc. This is the template that was previously used to load the default data, but you can load different data from a different template by clicking on the template data, then clicking again on the '...' to the right of the template data and making a new selection. When you make the selection, the appropriate data is loaded from the template into the model.

Note 1: It is important to understand that the template data itself does not contribute to the model until it is loaded in.

Note 2: Changes to the model data relative to the data in the template last loaded are highlighted in bold to show that it overrides the template data previously loaded.

Note 3: Another way to load data to a model from templates is to use the [Load Data From Template](#) tool.

Note 4: If you change the data in a template and you would like those changes to be updated to the model then you should use the [Load Data From Template](#) tool. There is an option to **Reload templates** which can be useful for this purpose.



Template selection in Model data

Colour codes

Templates are coded by colour in all selection lists as follows:

- **Red** shows user data, i.e. data that was not supplied by DesignBuilder.
- **Green** indicates that the data is derived from a reputable national or international source (e.g. ASHRAE, CIBSE).
- **Blue** indicates that the data is generated internally by DesignBuilder.
- **Black** is used for all other data.

See also [Model Data Colour](#).

Activity Templates

Activity templates are used to bulk-assign typical activity (usage) data to building models.

The data covers occupancy, equipment usage and suitable design internal temperatures, illuminance levels and ventilation rates per person.

The Activity template data provided with DesignBuilder is mainly sourced from the UK National Calculation Methodology and from ASHRAE 90.1 and 62.1.

See also the [Activity data templates and time schedules](#) tutorial

There are five tabs on the Activity Template dialog:

- [General](#)
- [All gains](#)
- [Occupancy](#)
- [Other gains](#)
- [Environmental](#)

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Activity Templates - General

General tab on [Activity Templates](#) Dialog.

Enter the name, category, sector and region for this activity template.

Enter the region as 'General' if you want the data to be generally available (i.e. not restricted to a particular region).

The floor shade colour is used to shade the floor of the model at block and zone levels to give a quick indication of the usage of the building in various areas.

Activity Templates - All Gains

All Gains tab on [Activity Templates](#) Dialog.

The lumped gains into the space from people, equipment, lights etc. Data on this tab is only used when the Gains option is set to 'Lumped'.

Activity Templates - Occupancy

Occupancy tab on [Activity Templates](#) Dialog.

Occupancy levels and times, metabolic activity on this tab is used when the 'Gains' option is set to 'Early' or 'Detailed'.

Set the metabolic rate according to the level of activity within the space. The metabolic factor accounts for people of various sizes. Enter 1.00 for men, 0.85 for women, 0.75 for children, or you can use an average value if there is a mix of sizes.

The Workday profile is used when the 'Timing' option is set to 'Typical workday' and the Schedule data is used when the Timing option is set to '7/12 Schedules'.

Activity Templates - Other Gains

Other Gains tab on [Activity Templates](#) Dialog.

Data on this tab is used when the 'Gains' option is set to 'Early'. DHW Data is always used.

The default gains into the space from computers, office equipment, catering, process and miscellaneous equipment for this activity.

Also hot water consumption rate per person per day and the operation schedules are defined here.

Activity Templates - DHW

Enter the Domestic Hot Water (DHW) volumetric consumption per unit of floor area per day and the operation schedule.

In DesignBuilder the amount of DHW consumed is considered to be a function of the activity. For example, there is a demand assumed to arise from the occupants of an office for activities such as washing hands and washing up cups. This demand is associated with the office rather than the toilet or tea room. Thus the demand from each space needs to be assigned to a DHW system even if the system itself is not present in the space.

Activity Templates - Environmental/Comfort

Environmental tab on [Activity Templates](#) Dialog.

Enter the data related to the environmental and comfort requirements of this activity:

- Heating set-point temperature (if the space is heated).
- Cooling set-point temperature (if the spaces cooled).
- Ventilation set-point temperature
- Illuminance levels required.
- Fresh air levels required per person - defines mechanical ventilation air change rates.

If there is never any heating or cooling, you can leave these set-points.

See [Environmental Control](#) under [Activity](#) Model Data for more details.

Construction Templates

Construction templates are used as a source of generic construction data for loading into building models.

The template provides data on the [construction](#) of external walls, roofs, floors, partitions, roofs etc and [air-tightness](#) data.

There are four tabs on the Construction template dialog:

- [General](#)
- [Simple](#)
- [Standard](#)
- [Airtightness](#)

Tip: DesignBuilder is provided with a number of system "Early Design" Construction templates. You can read about how to control these on the "Legislative Region Templates - Standard Insulation" on page 1510 help page.

See also the [Construction data](#) tutorial

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Construction Templates - General

General tab on [Construction Templates](#) Dialog.

Enter the name, source, category and region for this construction template.

Construction Templates - Simple

Simple tab on [Construction Templates](#) Dialog.

Data on this tab is used when the Construction model detail is set to **Pre-design**.

Enter default levels of insulation and thermal mass for this construction type.

Construction Templates - Walls

Walls tab on [Construction Templates](#) Dialog.

Data on this tab is used when the [Construction](#) model detail is set to **Pre-design** or **General** (i.e. it is always used).

Select default walls, partitions for exposed and semi-exposed surfaces for this Constructions template.

You can also select the external and internal wall sub-surface constructions here.

Construction Templates - Roofs/Floors/Slabs/Ceilings

Roofs/Floors/Slabs/Ceilings tab on [Construction Templates](#) Dialog.

Data on this tab is used when the [Construction](#) model detail is set to **Pre-design** or **General** (i.e. it is always used).

Select default constructions for roofs, floors, slabs and ceiling surfaces for this Constructions template.

You can also select the roof sub-surface construction here.

Construction Templates - Doors

Doors tab on [Construction Templates](#) Dialog.

Data on this tab is used when the [Construction](#) model detail is set to **Pre-design** or **General** (i.e. it is always used).

Select default door constructions for internal and external doors for this Constructions template.

You can also select the external and internal wall sub-surface constructions here.

Construction Templates - Airtightness

Airtightness tab on [Construction Templates](#) Dialog.

Cracks template

The Cracks template setting indicates the Cracks data template to use as a source for crack data when the [Calculated](#) Natural ventilation model option is selected.

Infiltration

The default infiltration rate used when the 'Natural ventilation' model detail level is 'Simple'.

Construction Templates - Thermal Mass

Thermal mass tab on [Construction Templates](#) Dialog.

Data on this tab is used when the [Construction](#) model detail is set to **Pre-design** or **General** (i.e. it is always used).

Enter the default thermal mass construction for this construction template.

See [Internal Thermal Mass](#) for more information on defining thermal mass in a model.

Glazing Templates

Glazing templates are used as a source of default [glazing](#), frame and [shading](#) construction data for building models.

There are three tabs on the Glazing Template dialog:

- [General](#)
- [Glazing](#)
- [Shading](#)

See also the [Glazing and openings](#) tutorial

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Glazing Templates - General

General tab on [Glazing Templates](#) Dialog.

Enter the name, category and region for this glazing template.

Enter the region as 'General' if you want the data to be generally available (i.e. not restricted to a particular region). Glazing templates with category '<System>' have their data set up at the time a file is created or when the Legislative region is changed.

Glazing Templates - Glazing

Glazing tab on [Glazing Templates](#) Dialog.

Enter data on the glazing construction, frame and opening area for external glazing, internal glazing and roof glazing.

Glazing Templates - Shading

Shading tab on [Glazing Templates](#) Dialog.

Select any shading devices to be applied when this glazing templates is loaded to the model.

Facade Templates (Types)

Facade templates are used to provide a choice of typical simple facade layouts and frame definitions which can be easily loaded into a building model.

There are two tabs on the Facade Template dialog:

- [General](#)
- [Openings](#)

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Facade Templates - General

General tab on [Facade Templates](#) Dialog.

Enter the name and category for this Facade Template.

Facade Templates - Openings

Openings tab on [Facade Templates](#) Dialog.

Enter the glazing and frame data for the facade. There are a number of standard facade types:

- None - there is no glazing.
- Continuous horizontal - glazing is generated in a continuous horizontal strip using sill height and % glazing.
- Fixed height - glazing is generated with a fixed height at the specified sill height, and width is calculated based on the % glazing. This option uses % glazing but prioritises window height.
- Preferred height - glazing is generated using the window height data and the % glazing, but the window height may be adjusted to achieve the required % glazing. This option uses window and sill height but prioritises % glazing.
- Fixed width and height - windows have fixed width and height. This option uses % glazing but prioritises window width and height.
- Fill surface (100%) - the entire surface is filled with glazing and there is no frame.
- If the glazing does not have a frame, uncheck the 'Has a frame?' box.

Lighting Templates

Lighting templates data sets are used as a source of generic lighting systems for loading into the building data.

There are three tabs on the Lighting Template dialog:

- [General](#)
- [Output](#)
- [Control](#)

See also the [Lighting data and daylight controls](#) tutorial

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),

- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Lighting Templates - General

General tab on [Lighting Templates](#) Dialog.

Enter the name, category and region for this Lighting Template.

Enter the region as 'General' if you want the data to be generally available (i.e. not restricted to a particular region).

See also Building Model [Lighting Data](#) for more information on the use of this template data in the model.

Lighting Templates - Output

Output tab on [Lighting Templates](#) Dialog.

Enter the lighting energy for task and general lighting in W/m2/100 lux (W/ft2/100 lux). This data is used for setting up the actual Lighting energy W/m2 data in each zone using:

Lighting energy (W/m2) = Lighting (W/m2/100 lux) x Design lux level (Activity tab) / 100

Substitute ft2 for m2 for equation in IP units.

The 'Output' data is used for generating default lighting loads based on lighting type and required illuminance levels as set in Activity data.

See [General Lighting](#) under Building Model [Lighting Data](#) for more information on the use of this template data in the model.

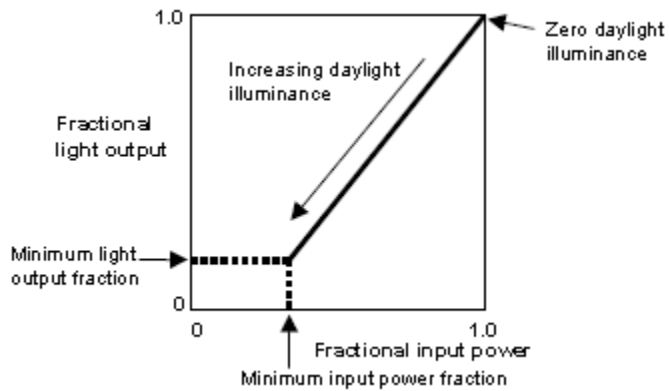
Lighting Templates - Control

Control tab on [Lighting Templates](#) Dialog.

If the lighting is to be controlled by the availability of natural daylight, check the 'control' option box and select the type of control.

Continuous Control

For 'continuous' control, the overhead lights dim continuously and linearly from maximum electric power, maximum light output to minimum electric power, minimum light output as the daylight illuminance increases. The lights stay at the minimum point with further increase in the daylight illuminance.



The minimum input power fraction for 'continuous' control type is the lowest power the lighting system can dim down to, expressed as a fraction of maximum input power. For 'continuous/off' lighting control, this is the power fraction reached just before the light switch off completely.

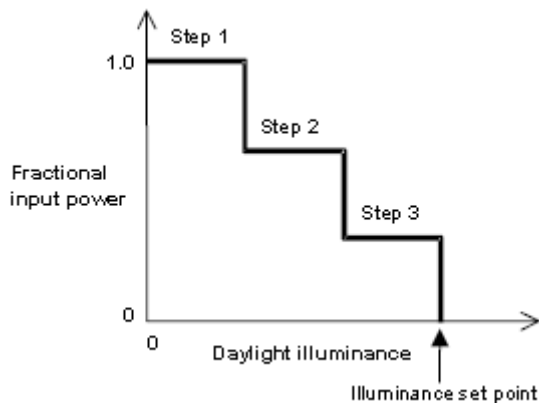
The minimum output fraction for 'continuous' control type, is the lowest lighting output the lighting system can dim down to, expressed as a fraction of maximum light output. This is the fractional light output that the system produces at minimum input power. For 'continuous/off' lighting control, this is the power fraction reached just before the light switch off completely.

Continuous/Off Control

'Continuous/off' control is the same as continuous control except that the lights switch off completely when the minimum dimming point is reached.

Stepped Control

For 'stepped' control, the electric power input and light output vary in discrete, equally spaced steps. The number of steps can be set.



HVAC Templates

HVAC templates data sets are used as a source of generic [heating/cooling](#) and [ventilation](#) systems for loading into the building data.

There are four tabs on the HVAC Template dialog:

- [General](#)
- [Ventilation](#)
- [System](#)
- [Air temperature distribution](#)

See also the [Simple HVAC Data](#) tutorial

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

HVAC Templates - General

General tab on [HVAC Templates](#) Dialog.

Enter the name, category, sector and region for this HVAC template.

Auxiliary energy

This setting defines the energy consumption of fans, pumps and other auxiliary equipment when using the Simple HVAC model option together with the [1-NCM Simple HVAC auxiliary energy calculations](#) option. This energy is assumed to be consumed during periods when either heating or cooling systems are operating.

Region

Select the region as **General** if you want the data to be generally available (i.e. not restricted to a particular region).

Floor shade colour

The floor shade colour is used to shade the floor of the model at block and zone levels to give a quick indication of the system type fitted in various areas.

HVAC Templates - Ventilation

Ventilation tab on [HVAC Templates](#) Dialog.

Natural Ventilation

Natural ventilation On

Check the **On** checkbox under the Natural Ventilation header to activate natural ventilation. This causes the other natural ventilation data to become accessible.

Rate

Enter the natural ventilation rate in ac/h. Note that this will only be used when the Outside air definition method on the HVAC model data tab is set to **1-By zone**.

Mixed mode

Check this option to enable mixed mode for this template. The other mixed model settings are not accessible through the HVAC template mechanism.

Mechanical Ventilation

Check the mechanical ventilation checkbox to indicate that outside air and/or re-circulated air is delivered to the zone. You can use this setting to define mechanical ventilation and air delivered through centrally ducted air conditioning systems or local fresh air systems. The specification of outside air delivery rates is the same for both options as described in the [Outside air](#) section.

Fan Type

When using Simple HVAC, enter the type of fan. Select from:

- **1-Supply** - fan blows outside air into the zone and the zone receives heat from the fan if **Fan in air** (below) > 0
- **2-Extract** - fan exhausts air from the zone. In this case there is no fan heat pickup. It is assumed that makeup air comes from outside.

Note that in Simple HVAC the [Auxiliary energy](#) data accounts for all electric fan and pump distribution energy plus controls and any other electrical energy use associated with HVAC that is not already accounted for elsewhere.

Fan pressure rise

Enter the pressure rise at full flow and standard conditions. Standard conditions are considered 20°C at sea level, 101325 Pa.

See also [Calculating Fan Pressure Rise](#).

Fan total efficiency (%)

Enter the product of the fan motor and impeller efficiency of the supply fan. This is the ratio of the power delivered to the air to the electrical input power at maximum flow expressed as a percentage. The motor efficiency is the power delivered to the shaft divided by the electrical power input to the motor. The fan efficiency is power delivered to the air divided by the shaft power. The power delivered to the fluid is the mass flow rate of the air multiplied by the pressure rise divided by the air density. Must be greater than 0 and less than or equal to 100.

Economiser

Economisers are used to provide cooling when the outdoor temperature is lower than the indoor temperature. An economiser is a damper opening that draws up to 100% outside air when the outside air is cooler than the temperature inside the building, thereby providing **free cooling**. An outdoor air economy cycle can reduce cooling energy requirements by some 20% to 30%, or around 5% of the air conditioning energy use and are often required by energy codes for larger air conditioning units.

Choose from 3 options:

- **1-None** where no economiser operation will be simulated and outside fresh air is based purely on the Mechanical ventilation Outside air rate and associated operation schedule.
- **2-Differential dry bulb** the economiser increases the outdoor air flow rate above the minimum outdoor air flow when there is a cooling load and the outdoor air temperature or enthalpy is below the zone exhaust air temperature.
- **3-Differential enthalpy** the economiser increases the outdoor air flow rate above the minimum outdoor air flow when there is a cooling load and the outdoor air enthalpy is below that of the zone exhaust air.

Note: The economiser works together with the cooling system and if cooling is not selected the economiser will not operate.

Max outdoor air rate when economiser operates

The upper limit of outside airflow when the economiser is working in ac/h. The default is 15 ac/h.

Note: This value is not used a) when the [Cooling limit type](#) is set to **2-Limit capacity** and b) when the [HVAC Autosize](#) model option is **1-Adequate**.

Heat recovery

When heat recovery is active you can choose the type of heat recovery. Select from:

- **1-Sensible** which provides sensible heat recovery whenever the zone exhaust air temperature is more favourable than the outdoor air temperature.
- **2-Enthalpy** where latent and sensible heat recovery are provided whenever the zone exhaust air enthalpy is more favourable than the outdoor air enthalpy.

With the EnergyPlus Ideal loads system used in Simple HVAC heat recovery heating is only available when a zone is heated, i.e. when the **Heated** checkbox is checked on the HVAC tab. Likewise cooling heat recovery will only be available if the **Cooled** checkbox is checked. That is not to say that heat recovery isn't available for zones with heating at times when the heating system is not operating - heat recovery heating is available at any time in this case when the outside air is cooler than zone exhaust air, regardless of whether heating is operating or not. If you need to model heat recovery for systems without heating then you should use Detailed HVAC which is more flexible.

Sensible heat recovery effectiveness

The sensible heat recovery effectiveness, where effectiveness is defined as the change in supply temperature divided by the difference in entering supply and relief air temperatures. The default is 0.70.

Latent heat recovery effectiveness

The latent heat recovery effectiveness, where effectiveness is defined as the change in supply humidity ratio divided by the difference in entering supply and relief air humidity ratios. The default is 0.65.

HVAC Templates - Heating And Cooling

Heating and Cooling tab on [HVAC Templates](#) Dialog.

Heating

Heating On

If the template includes heating then check this checkbox. When it is checked the rest of the items below become available.

Heating fuel

Select the type of fuel used to generate the heating energy - choose from:

Interface Fuel Name

- 1-Electricity from grid
- 2-Natural Gas
- 3-Oil
- 4-Coal
- 5-LPG
- 6-Biogass
- 7-Anthracite
- 8-Smokeless Fuel (inc Coke)
- 9-Dual Fuel Appliances (Mineral + Wood)
- 10-Biomass
- 11-Waste Heat

Fuel in results

- Electricity
- Gas
- Oil
- Solid fuel
- Bottled gas
- Bottled gas
- Solid fuel
- Solid fuel
- Other
- Other
- Other

You should select **1-Electricity from grid** you wish the results displayed in DesignBuilder to agree with those calculated by EnergyPlus directly for District heating and displayed on the Summary tab of the Simulation screen.

Heating system CoP

The heating system Coefficient of Performance is used to calculate the fuel consumption required to meet heating demand. The value represents the total seasonal efficiency of the entire heating system and should include the effect of all energy consumption associated with building heating such as any boiler fan energy, boiler inefficiency, control equipment etc. It does not include energy involved in air or water distribution (fans or pumps) or control gear, which is accounted for in the [Auxiliary energy](#).

Heating type

There are two heating types available:

- **1-Convective** - the space is heated by an air system and controlled to the air temperature set point. The system is modelled using the EnergyPlus "Ideal Loads" system.
- **2-Radiative/convective units** - used for modelling general systems where a radiant heat is a factor. Allows modelling of convective systems, hot water radiator systems, underfloor heating systems, baseboard heating etc. using EnergyPlus 'High Temp Radiant System'.

Heating Supply Air Conditions

Maximum supply air delivery temperature

The maximum dry-bulb air temperature of the air supplied for heating the zone (in °C or °F).

Maximum supply air humidity ratio

The constant humidity ratio (mass of water per mass of dry air) of the warm supply air to be delivered when heating is required. The default humidity ratio is 0.0156. You should be careful to enter a value that is a valid condition, i.e. it is under the 100% saturation line on the [Psychrometric Chart](#).

Cooling

Cooling On

If the template includes cooling then check this checkbox. When it is checked the rest of the items below become available.

Cooling system CoP

The cooling system coefficient of performance is used to calculate the fuel consumption required to meet cooling demand. The value represents the total seasonal efficiency of the entire cooling system and should include the effect of all energy consumption associated with building cooling such as fan and pump energy, chiller inefficiency, control equipment etc.

Cooling Supply Air Conditions

When using [Simple HVAC](#) you can specify the cooling supply air conditions of the air.

Minimum supply air delivery temperature

The minimum dry-bulb air temperature of the air supplied for cooling the zone (in °C or °F).

Minimum supply air humidity ratio

The minimum humidity ratio (kg of water per kg of dry air) of the cool supply air. The default is 0.0077 kg-H₂O/kg-air which corresponds to a 10C (50F) dew point. You should be careful to enter a value that is a valid condition, i.e. it is under the 100% saturation line on the [Psychrometric Chart](#).

HVAC Templates - Humidity Control

Humidification

If there is humidification control check the check box and select the control type below.

Humidification control type

When humidification is selected there are 2 further options:

- **1-Constant supply humidity ratio** means that during heating the supply air will always be at the Maximum heating supply humidity ratio.
- **2-Humidistat** means that the humidity in the zone is controlled using the [Humidification setpoint](#) defined on the Activity tab. The ideal loads system will attempt to meet the humidistat request, i.e. it will humidify according to the setpoint.

Dehumidification control type

When dehumidification is selected there are 3 further options:

- **1-Constant supply humidity ratio** means that during cooling the supply air will always be at the Minimum cooling supply humidity ratio.
- **2-Humidistat** means that the humidity in the zone is controlled using the [Dehumidification setpoint](#) defined on the Activity tab. The ideal loads system will attempt to meet the humidistat request, i.e. it will dehumidify according to the setpoint.
- **3-Constant sensible heat ratio** means that the ideal loads system will be controlled to meet the sensible cooling load, and the latent cooling rate will be computed using a constant sensible heat ratio (SHR).

For **3-Constant sensible heat ratio** and **2-Humidistat** options, if the mixed air humidity ratio is less than the target humidity ratio, then the mixed air humidity ratio will be used. For all options, the supply air humidity ratio will never be allowed to exceed saturation at the supply dry bulb temperature. The selected dehumidification control type is always applied when the unit is in cooling mode. If the unit is in deadband mode (not actively heating the supply air) control type Humidistat will be active. If the unit is in heating mode, control type Humidistat will be active if the Humidification Control Type field below is set to Humidistat or None. This allows the ideal loads system to heat and dehumidify at the same time.

HVAC Templates - Air Temperature Distribution

Air temperature distribution tab on [HVAC Templates](#) Dialog.

By default EnergyPlus assumes that air temperature within a zone is completely uniform (i.e. the air is fully mixed). The data on this tab allows you to set up a temperature gradient which varies dynamically depending on:

- Outside temperature.
- Inside temperature.
- Inside-outside temperature difference.
- Heating load.
- Cooling load.

See [Air Temperature Distribution](#) for more information on defining vertical zone temperature profiles (temperature gradients).

HVAC Templates - Cost

Cost tab on [HVAC Templates Dialog](#).

The HVAC cost is calculated zone by zone by multiplying the zone floor area by the cost per floor area (GIFA) for the zones on the HVAC tab. This data will normally be loaded to model data from the HVAC template to ensure that HVAC performance and cost data are consistent.

This cost should include any contributions to the total cost of the building HVAC system from plant components and not only the components within the particular zone.

Note: This data is used for both Simple and Detailed HVAC for costing purposes.

Tip: The [Cost data](#) help topic provides an overview of the places in DesignBuilder where costs are entered.

Cost per area

Enter the total cost per floor area (GIFA) for the HVAC system.

HVAC cost per cooling load

Enter the indicative cost of the cooling element of this HVAC system type (in GBP/kW).

HVAC cost per heating load

Enter the indicative cost of the heating element of this HVAC system type (in GBP/kW).

Location Templates

Location Templates are a database of locations from around the world based on the latest ASHRAE design weather data used by EnergyPlus.

Each Location data set is located within a 'Legislative Region'. Most Regions in Europe include some basic data on the energy codes which apply.

There are four tabs on the Location Template dialog:

- [Location](#)
- [Winter design weather](#)
- [Summer design weather](#)
- [Simulation weather](#)

Note: There is no direct link between hourly weather data and the ASHRAE design values on the Winter and Summer design data tabs.

See also the [Site level data and weather files](#) tutorial

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Location Templates - Location

Location tab on [Location Templates](#) Dialog.

DesignBuilder Location Templates are located within 'Legislative Regions'. Most regions in Europe include some basic data on the energy codes which apply.

See the corresponding topics for model data for details:

- "Site location" on page 287
- "Site details" on page 288
- "Time and daylight saving" on page 314

Location Templates - Winter Design Weather

Winter design weather tab on [Location Templates](#) Dialog.

The weather data used in winter design calculations is:

- Minimum outside dry-bulb temperature
- Co-incident wind speed and direction.

There are options for 99.6 or 99% confidence (i.e. 0.4 or 1% chance of more extreme winter weather occurring).

See the corresponding topics for model data for details:

- "Winter design weather data" on page 316

Location Templates - Summer Design Weather

Summer design weather tab on [Location Templates](#) Dialog.

The weather data used in summer design calculations is:

- Maximum outside dry-bulb temperature
- Minimum outside dry-bulb temperature
- Wet-bulb temperature at the time of the maximum dry-bulb temperature.

Risk can be specified based on dry or wet-bulb temperatures, for 99.6, 99 and 98% confidence (i.e. 0.4, 1 or 2% chance of more extreme weather occurring).

See also: "Summer Design Weather Data" on page 317

Location Templates - Simulation Weather

Simulation weather tab on [Location Templates](#) Dialog.

DesignBuilder will only use hourly weather data for simulations (winter and summer design calculations use design weather data).

See also: [Simulation weather data](#)

You can [add your own Hourly Weather data](#).

Crack Templates

Crack data is used when the 'Calculated' natural ventilation option is set. In this case every surface in the model has a crack and its size (characterised by flow coefficient and exponent) is determined by [Airtightness](#) as set on the [Constructions tab](#) in the Model Data. The value of Airtightness is used as a key to look up corresponding crack sizes in the Cracks database.

The cracks database has been set up empirically to give typical air change rates for each of the five Airtightness categories in a range of building types.

There are five tabs on the Crack Template data dialog:

- [General](#)
- [Openings](#)
- [Walls](#)
- [Floors/ceilings](#)
- [Roofs](#)

A source of additional information on crack data from AIVC is [Numerical Data for Air Infiltration & Natural Ventilation Calculations](#).

See also:

- "Natural Ventilation and Infiltration model Options" on page 1272

Crack Templates - General

General tab on [Crack Templates Dialog](#).

Tab contains the name and category of the crack template.

Crack Templates - Openings

Openings tab on [Crack Templates Dialog](#).

Enter the flow characteristics of the cracks around internal and external windows, vents and doors.

Flow coefficient

Flow coefficients for openings define the flow through the cracks between the opening and the parent surface. In practice this may include any gaps between in openable sections of windows, doors and vents and their frame as well as the gaps between the opening component frame and the surface fabric. If using published data for window, door and vent components, be careful to ensure that both these effects are included in the data provided in DesignBuilder. For example in the AIVC document: is [Numerical Data for Air Infiltration & Natural Ventilation Calculations](#), window leakage data is provided only for the windows themselves and not for the gaps between fabric and the window frames.

Note that the opening crack characteristics are normalised by opening perimeter *lengths*.

Crack Templates - Walls

Walls tab on [Crack Templates](#) Dialog.

In reality the porosity of walls is caused by a very large number of small cracks and holes.

DesignBuilder models this porosity using a single equivalent crack. Enter the characteristics this equivalent single crack for internal and external walls.

Note that the crack characteristics are normalised by *surface area*.

Crack Templates - Floors/Ceilings

Floors/Ceilings tab on [Crack Templates](#) Dialog.

In reality the porosity of floors is caused by a very large number of small cracks and holes.

DesignBuilder models this porosity using a single equivalent crack. Enter the characteristics this equivalent single crack for internal and external floors.

Note that the crack characteristics are normalised by *surface area*.

Crack Templates - Roofs

Roofs tab on [Crack Templates](#) Dialog.

In reality the porosity of roofs is caused by a very large number of small cracks and holes.

DesignBuilder models this porosity using a single equivalent crack. Enter the characteristics this equivalent single crack for roofs.

Note that the crack characteristics are normalised by *surface area*.

Legislative Region Templates

A Legislative region in DesignBuilder is typically a whole country or a region, province or state within a country which has its own building control powers.

In current versions of the software a Legislative Region contains information on:

- Local energy codes/building regulations.
- Carbon dioxide emission factors are provided for a range of fuels.

The current regions database is only sparsely populated with useful data (most of that is from countries within Europe). But we expect to expand this in time.

There are three tabs on the Legislative Region Template dialog:

- [General](#)
- [Standard insulation](#)
- [Emissions](#)

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),

- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Legislative Region Templates - General

General tab on [Legislative Region Templates Dialog](#).

Enter the name of the Legislative region, the category (country) and a description of the Region.

Legislative Region Templates - Standard Insulation

Standard insulation tab on [Legislative Region Templates Dialog](#).

Data on this tab specifies 4 standard insulation levels which apply to the region:

- **Uninsulated** - typical U-values for buildings in the region which are not insulated. In general the same standard is applied for all regions, though the capacity exists to apply different standards.
- **Typical** - Average (estimated) insulation standards for real buildings in the region. This data is used by default to define default construction and glazing performance settings when adding a new building.
- **Mandatory energy code** - Defines the maximum U-values allowed in the energy code/building regulations in force in the region.
- **Best practice** - Defines the U-values to be expected in a state-of-the-art building in the region. In general the same standard is applied for all regions, though the capacity exists to apply different standards.

The [energy code templates](#) referenced here indicate the required maximum U-values.

Regions, Energy Codes, Construction Templates And Constructions - How They Interact

The way the system works is explained below.

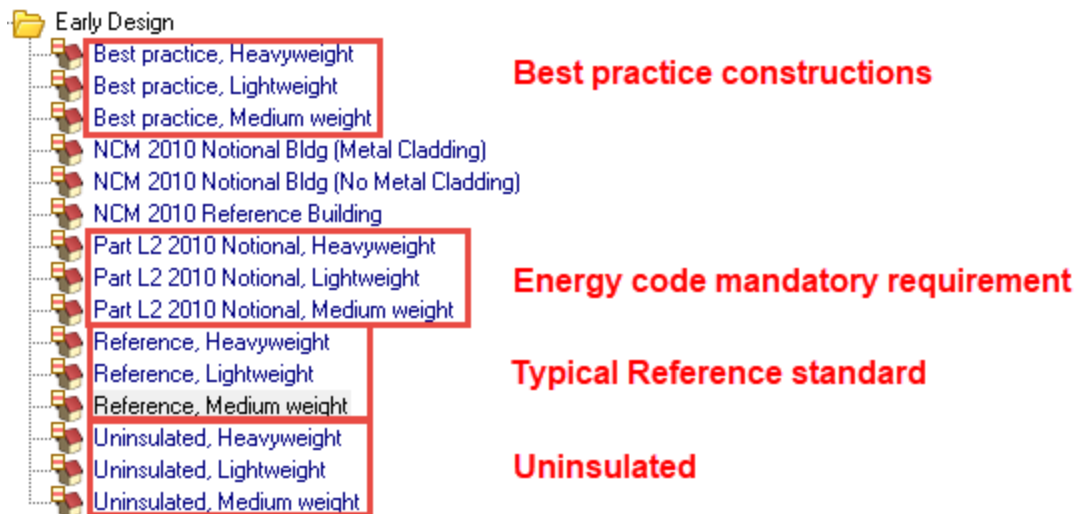
1. When a project is first created, DesignBuilder uses the 4 different standards of insulation for the [Region](#) associated with the [Location](#) selected on the [New site](#) dialog to create the corresponding 12 x system [Constructions](#) and 12 x "Early Design" system [Construction templates](#).
2. Because the 12 x "Early Design" Constructions are set up at the point the project is created, the easiest way to fully control the U-values that will be applied to the corresponding library Constructions, is to create appropriate [library components and templates](#) before the file is created:

1. Location template,
2. Region, and,
3. Energy code with the required levels of insulation defined.

The Region should reference the new Energy codes on the [Standard insulation](#) tab and the Location template should reference the new Region on the Location tab. Select your custom Location template on the [New site](#) dialog.

3. The 12 constructions and templates referred to above cover the 4 insulation standards defined by the Energy codes associated with the region multiplied by 3 different levels of thermal mass. The "Early Design" system construction templates for the England and Wales region are illustrated in the screenshot below.

"Early Design" System Construction Templates



- When adding a [new building](#) you can select the construction, glazing, lighting templates to be applied as [default settings for the new building](#). The default construction template will be the **Reference** template for the Region's energy code but this and the other settings will typically be overridden at some stage in the model development process to provide project-specific settings. However for early stage analysis you may wish to start your modelling process right away based on the default constructions etc and refine the detail at a later stage.
- There are 2 alternative ways to select/load construction templates depending on the [Construction and glazing](#) data model option selection. When the (default) **General** option is selected then the above 12 construction templates can be loaded from a normal browse list, or any other construction template can be loaded. Or, when the **Pre-design** option is selected, the above 12 construction templates can be loaded by making appropriate settings on the **Insulation** and **Thermal mass** sliders. In this case only the 12 "Early Design" construction templates are available for selection (via the 2 sliders).

Legislative Region Templates - Emissions

Emissions tab on [Legislative Region Templates](#) Dialog.

CO2 Emissions

Emissions factor

The emissions tab contains a single browse selection to define the [Fuel emissions factors](#) for each fuel. The emissions factors define the amount of CO₂, CO, CH₄, NO_x and several other pollutants emitted per unit of energy consumed for each of the major fuel types in the region. The emission factors depend on the mix of technology used to generate the electricity, the quality of the fuel and other factors such as production carbon cost.

CO₂ emissions are calculated within DesignBuilder by multiplying simulated fuel consumption by the CO₂ emission factor for that fuel. Conversion factors can be made specific to the region by changing the data for the region.

Note: The emission factors for each fuel are only actually used in the simulation when the corresponding selection has been made for that fuel type on the Region tab at site level under the [Include Fuel Emissions](#) header.

Energy Code Templates

Energy code data is used within DesignBuilder to indicate the mandatory energy performance of building components and/or the whole building.

There are three types of heat transmission code:

1. Envelope component performance
2. Whole building envelope performance
3. Whole building energy performance

You can define the type of energy code by making selections on the next 3 tabs.

If this energy code requires for example that minimum component performance and whole building energy performance then check both these options.

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Energy Code Templates - General

General tab on [Energy Code Templates](#) Dialog.

Enter a name and description of this Energy code.

Energy Code Templates - Envelope (component)

Envelope (component) tab on [Energy Code Templates](#) Dialog.

The U-value data on this tab specifies the mandatory maximum U-values through the individual components (walls, windows, roofs, floors etc.) for the energy code.

Maximum U-Values (Exposed Elements)	
Wall (W/m ² -K)	0.260
Below grade wall (W/m ² -K)	0.260
Flat roof (W/m ² -K)	0.180
Pitched roof (W/m ² -K)	0.160
Roofspace floor (W/m ² -K)	0.250
Floor (W/m ² -K)	0.220
Solid floor to ground (W/m ² -K)	0.220
Window (W/m ² -K)	2.000
Rooflight (W/m ² -K)	2.200
Vehicle access and other large doors (W/...	1.500
Personnel doors (W/m ² -K)	2.200
High usage entrance doors (W/m ² -K)	6.000
Maximum U-Values (Semi-Exposed Elements)	
Window (W/m ² -K)	2.200
Wall (W/m ² -K)	0.350
Floor (W/m ² -K)	0.250

The U-values are used to define the system constructions at the point when a new project is created. These in turn are referenced by corresponding construction templates.

Energy Code Templates - Envelope (building)

Envelope (building) tab on [Energy Code Templates Dialog](#).

Sets a limit on the rate of heat transfer through the overall building envelope. The limit is typically set as an average U-value for the overall building envelope.

Energy Code Templates - Energy Performance

Energy performance tab on [Energy Code Templates Dialog](#).

Sets a limit on the rate of heat transfer through the overall building envelope. The limit is typically set as an average U-value for the overall building envelope.

Sector Templates

Sectors are used to categorise buildings according to their use/activity.

A sector is located within a legislative region and can only be accessed if the building is situated within the region containing the sector.

The 'General' sector is an exception to this rule as it can be accessed from all regions if the 'List region data' Program Options setting is 'All or 'General+Local'.

You can also associate a holiday schedule with the sector.

Most of the sectors provided with DesignBuilder come from the UK NCM and are used for UK compliance and certification calculations. However users can create their own sectors if necessary, or otherwise simply use the generic "General" sector.

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Times Zone Templates

DesignBuilder comes with a database of all of the world's time zones including information on daylight savings. This data is sourced from ASHRAE.

If the time zone does not have daylight saving then the text **N/A** is entered in the **Daylight saving from** field.

Time zone

This field represents the time zone of the facility (relative to Greenwich Mean Time or the 0th meridian). Time zones west of GMT (e.g. North America) are represented as negative; east of GMT as positive. Non-whole hours can be represented in decimal (e.g. 6:30 is 6.5).

The time zone is selected in model data under [Time and Daylight Saving](#) on the [Location tab](#) at site level.

Note: you can override daylight savings to be off in the model data by unchecking the [Use daylight saving](#).

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Wind Pressure Coefficient Templates

[Wind pressure coefficients](#) are used when [Natural ventilation model option](#) is set to 'Calculated'. The EnergyPlus Airflow Network calculations use pressure coefficients when calculating [wind-induced pressure](#) on each surface during simulations when the Calculated Natural ventilation option is selected.

DesignBuilder is supplied with a database of wind pressure coefficients based on data from [Martin Liddament, Air Infiltration Calculation Techniques, An Applications Guide, AIVC](#). The Cp data is buildings of 3 storeys or less, with square surfaces and for 3 levels of site exposure. The data is given in 45° increments.

Another source of information on wind pressure coefficients from AIVC is [Numerical Data for Air Infiltration & Natural Ventilation Calculations](#).

Note: The AIVC wind pressure coefficient data provides a good first level of approximation for basic design purposes. However, ideally you would override this default data with pressure coefficient data specific to your application. This data might be calculated from CFD simulations or measured from wind tunnel studies using physical models.

Template data can be accessed:

- from the [Templates tab on the opening screen](#) (if no file is open),
- from the [Templates tab in the Navigator panel](#) (if a file is open),
- by double-clicking on the template icon in the model data,
- by clicking on the Edit button when selecting templates from a list.

Import Components/Templates

Import component/template [.DDF](#) files previously [exported](#) using the **File > Import > Library data** menu command:

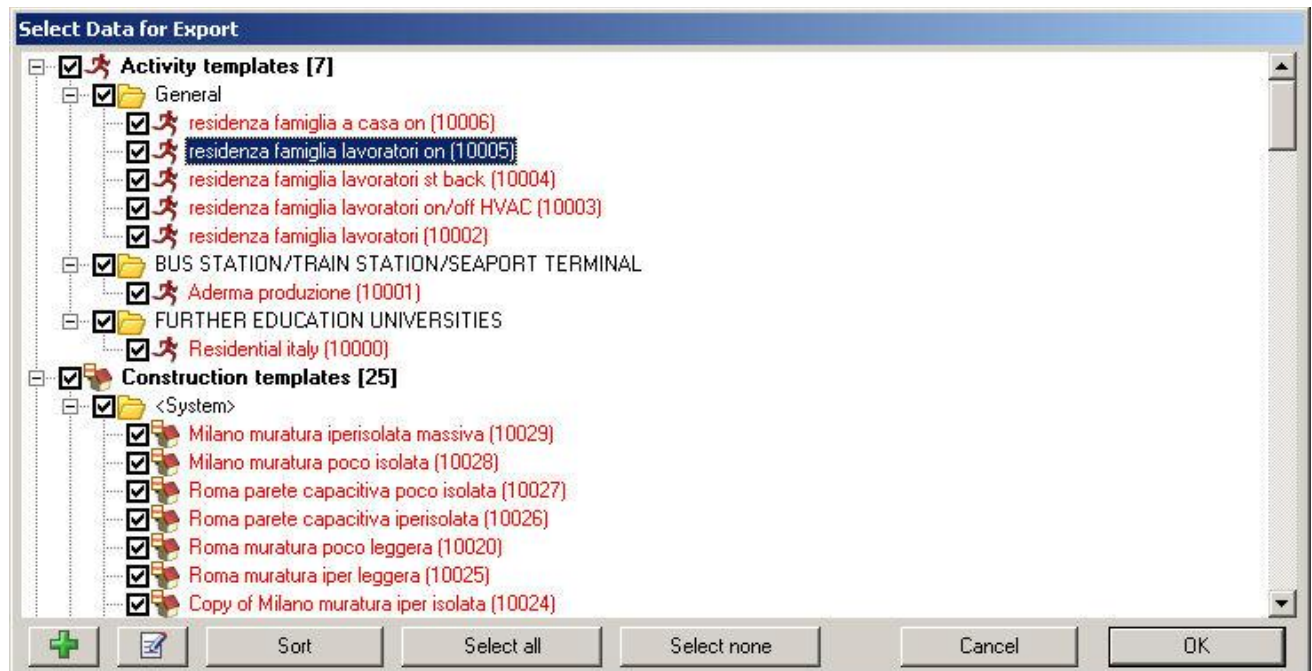
- Import the data to the *model* by selecting the Import menu command with a file open.
- Import to *libraries* by working from the Opening screen.

This command imports **all** components/templates in the DDF file.

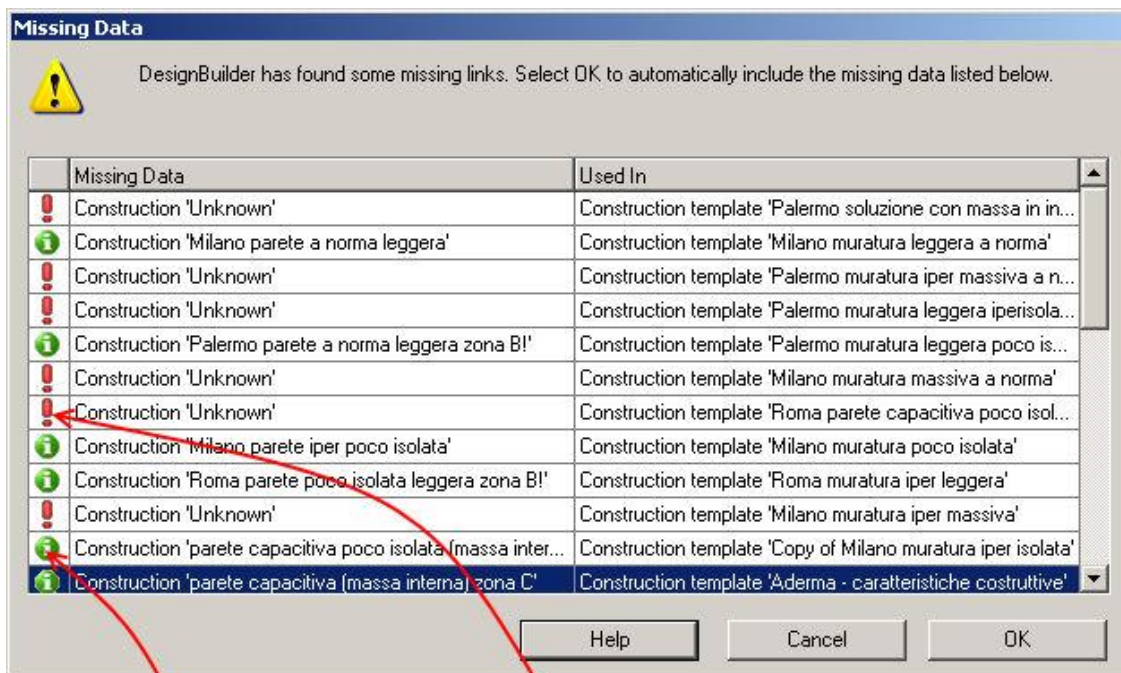
Note: As well as importing DDF files created directly by DesignBuilder you can also create your own component and template data sets (outside DesignBuilder) for importing. See the [Importing Custom Templates and Components](#) topic for more information on the process involved.

Export Components/Templates

You can export your user DesignBuilder Component/Template libraries by selecting the **File > Export > Library data** menu command to create a [.DDF](#) file. This file can be shared with other users, [imported](#) on other computers in your office or simply used as a backup.



When exporting a complex set of data, DesignBuilder will check that where user data references other user components, these referenced components are also selected. For example, if you select a construction for export which uses your own materials, you must make sure that all materials used in this construction are also selected. If any required user components are not selected DesignBuilder displays a dialog showing the missing components. See below.



Data marked with a green 'info' icon is automatically included

Data marked with a red '!' icon is missing and should be either recreated or the selection changed

Press the OK button to continue with the export automatically including the missing data.

Note: if there is any data that is referenced but does not exist it is flagged with a red '!' icon (above). In this case you should press the Cancel button and make the necessary edits to ensure that all referenced data exists. DesignBuilder will not prevent you from continuing so it is possible to create DDF files with missing data.

When you use the library data **Export** command *with a model data file open*, the data exported will be user **Model components/templates**, not **Library components/templates**.

From the **Opening** screen you can also export entire data tables as **csv-spreadsheet files** suitable for viewing in a spreadsheet. This data cannot be read back into DesignBuilder.

Export Limitations

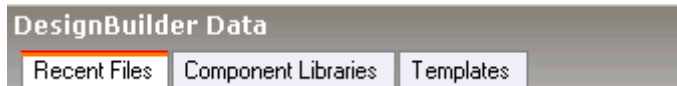
Some components rely on external file resources to complete the data set. In these cases the external file is not included in the ddf file and must be copied manually. This applies to:

- [Textures](#)
- [Hourly weather data](#)

For both the above cases the external texture and epw files are included in the dsb file though, so you may find it easier to save example template dsb models to transfer these datasets.

Library Management

DesignBuilder is supplied with libraries of [Templates](#) and [Components](#) and these can be managed from the opening screen (i.e. when no dsb file is loaded).



Click on [Component Libraries](#) or [Templates](#) tabs (above) to manage library data.

Tip: you can load the latest library components and templates into your model using the **Import library data to model** command from the Tools menu when a model is loaded. Any new data that has been added to the library since the model file was created will be imported to the model. Also if there have been any changes to library component and template data, this new data will overwrite the original data.

Keeping Templates and Component Libraries

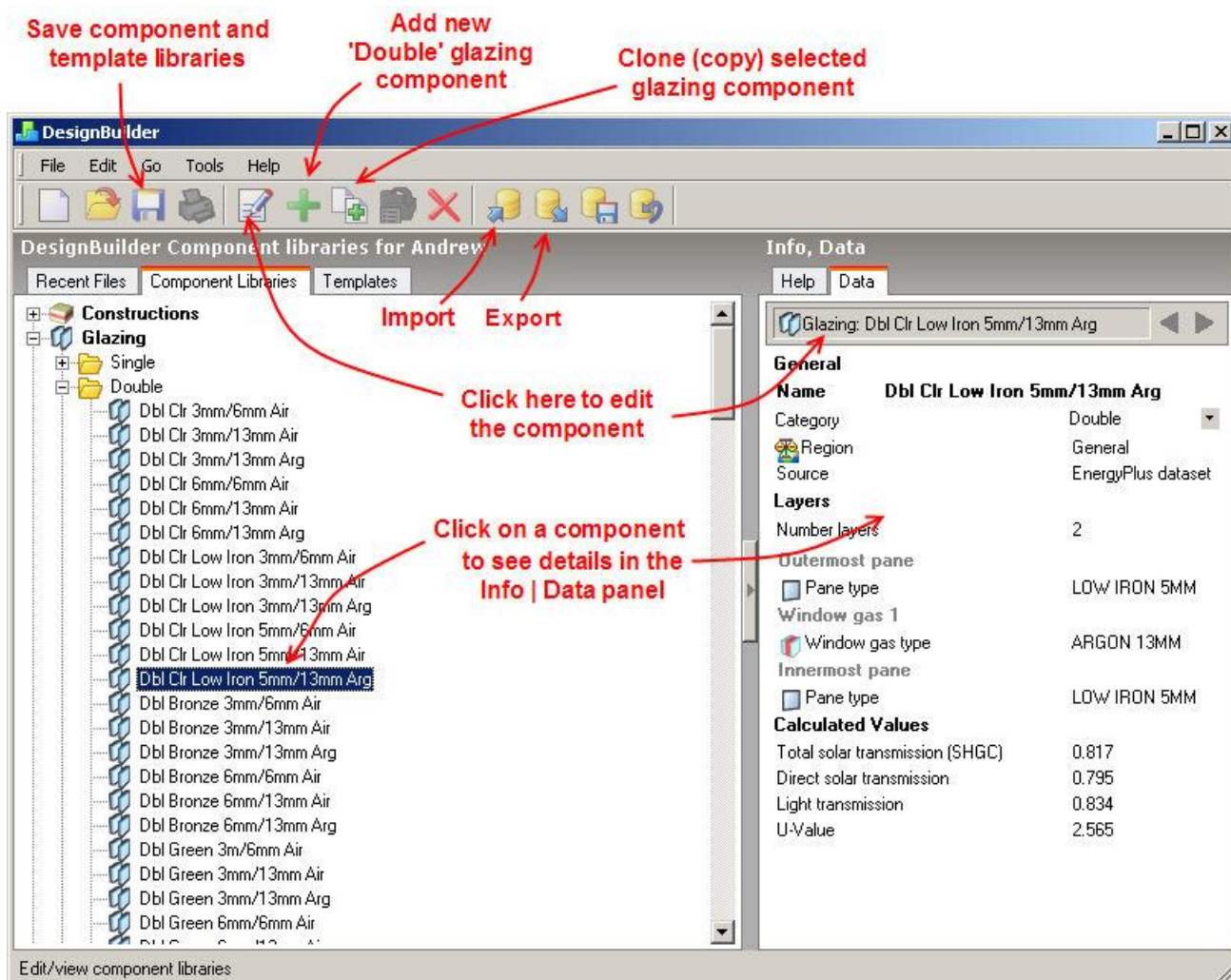
You may build up a large database of your own templates and components and to avoid losing this data between version upgrades, DesignBuilder backs up your library data in a series of .dat files which you can access from the **File > Folders > Library data** menu command. These files are retained when the software is uninstalled. When you install a new version of DesignBuilder and run it for the first time, any data you entered in the old libraries is merged with the new libraries. This ensures that you keep any data you created, while receiving the latest DesignBuilder system template and component data.

Note: DesignBuilder keeps separate program options and template and component libraries for each user on the computer system.

Opening Screen - Component Libraries

From the Component Library screen you can add, clone (copy), delete, view, edit, [import](#), [export](#), [backup](#), [restore](#) components as shown below. You can also save the libraries by clicking on the Save toolbar icon (libraries are also saved when the program exits).

Library [Components](#) are automatically loaded into the model *at the time the file is created* and from then on, they are associated specifically with this model and so are known as 'Model Components'. Any change made to the Library Components does not affect existing building models because they reference their own set of Model Components.



Add New Component

To add a component:

1. Click on the + to the left of the component type (Glazing in the above screen shot).
2. Click on the category (folder icon) or on a component within a category.
3. Click on the green + on the toolbar to create a new component of the selected category.
4. Fill in the details on the dialog and press OK.

The component has been added.

Clone (Copy) Component

Cloning a component creates an exact copy of the selected component gives the copy name 'Copy of ...' followed by the name of the original component.

1. Click on the + to the left of the component type (Glazing in the above screen shot).
2. Click on the component you wish to copy.
3. Click on the clone icon on the toolbar to create a new component identical to the selected one. This creates the copy and selects it,
4. To edit the new component click on the Edit toolbar button
5. Fill in the details on the dialog and press OK.

View Component

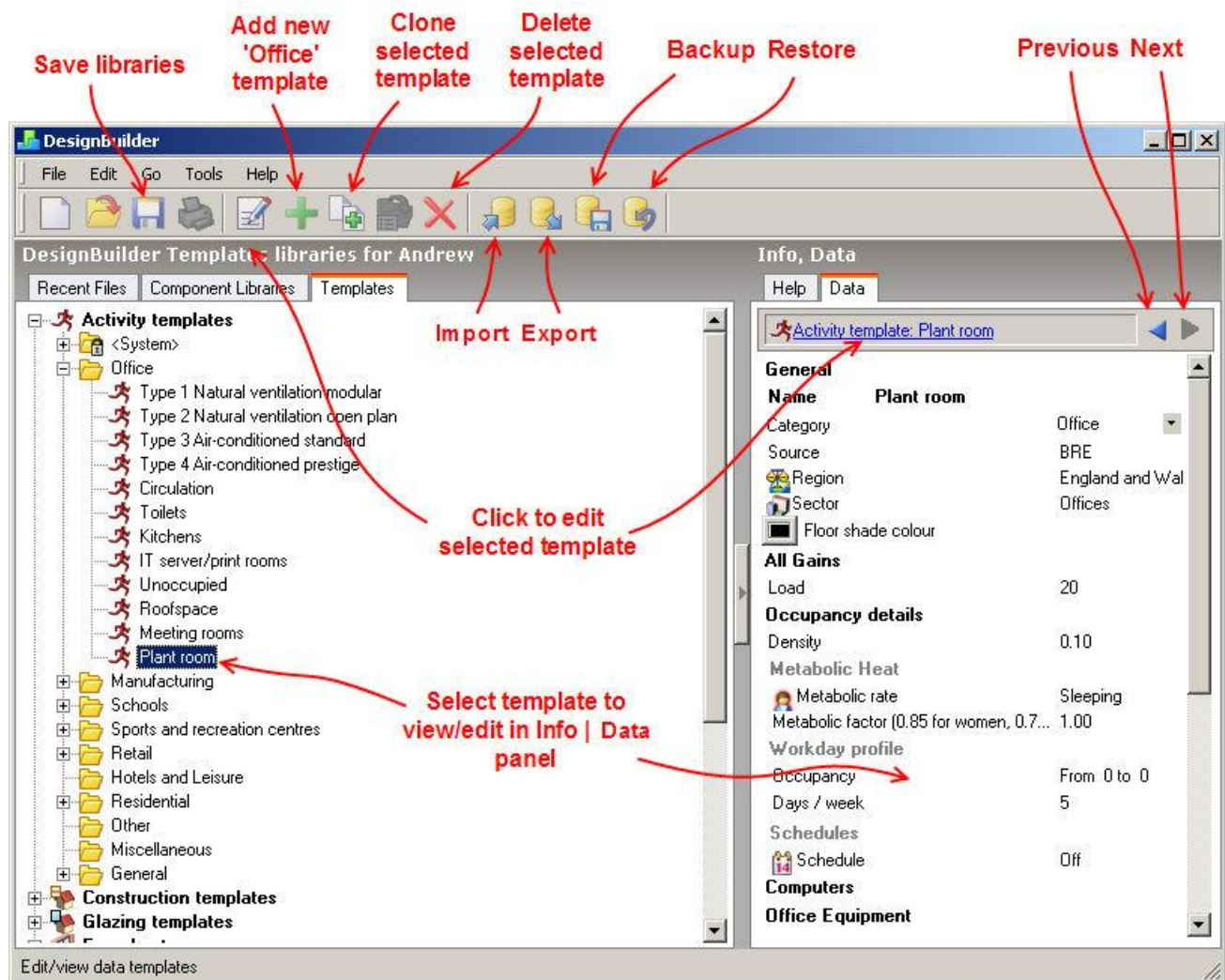
You can view the details of a component in the Info | Data panel on the right of the screen as shown in the diagram above. Note that icons in the View panel indicate that you can click on the component to see its'

details. You can also use the Previous and Next Navigation buttons above the Info panel to navigate through previously viewed components and templates.

Template Library Management

Use the opening screen to work with [template libraries](#). The whole screen is dedicated to the job and you get access to the Info Data panel which allows you to review the data for the currently selected template (screen shot below).

You can add, clone (copy), delete, view, edit, [import](#), [export](#), [backup](#), [restore](#) library templates as shown below. You can also save the libraries by clicking on the Save toolbar icon (libraries are also saved when the program exits)



See also the [Data templates](#) tutorial

Add New Template

To add a template:

1. Click on the + to the left of the template type (Activity templates in the above screen shot).
2. Click on the category (folder icon) or on a template within a category.
3. Click on the green + on the toolbar to create a new template of the selected category.

4. Fill in the details on the dialog and press OK.

The template has been added.

Clone (Copy) Template

Cloning a template creates an exact copy of the selected template gives the copy name 'Copy of ...' followed by the name of the original template.

1. Click on the + to the left of the template type (Activity templates in the above screen shot).
2. Click on the template you wish to copy.
3. Click on the clone icon on the toolbar to create a new template identical to the selected one. This creates the copy and selects it,
4. To edit the new template click on the Edit toolbar button
5. Fill in the details on the dialog and press OK.

View Template

You can view the details of a template in the Info | Data panel on the right of the screen as shown in the diagram above. Note that icons in the View panel indicate that you can click on the template to see its' details. You can also use the Previous and Next Navigation buttons above the Info panel to navigate through previously viewed components and templates.

Note: You can work with templates with a file open from the Templates tab on the Navigator panel but in this case you're working with [Model Templates](#), i.e. templates specifically associated with a particular model not the library.

Backup Component/Template Libraries



Back up your Component and Template libraries by clicking on the Backup toolbar icon. Backups created in this way are stored in the 'Backup\Manual' folder beneath the rest of the [library data](#).

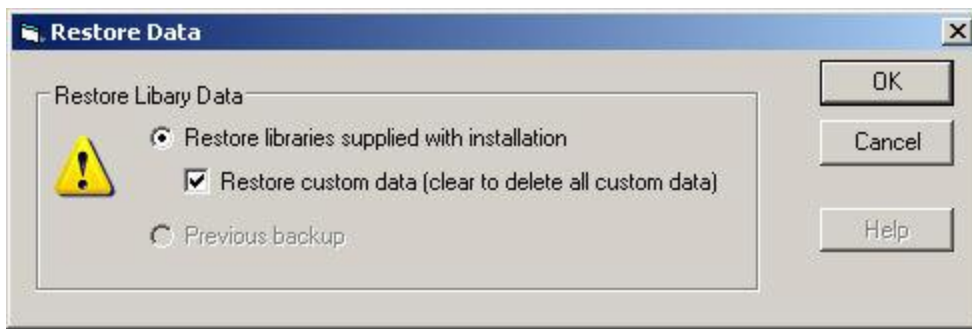
As well as backing up the compiled .dbt files, DesignBuilder also exports the files in .dat format and packages the whole backup into a zip file. If you have problems with your libraries you may find it helpful to send this zip file to DesignBuilder Support for analysis.

You can use this data as a backup of your library data and can restore it at a later date using the [Restore libraries](#) command.

Restore Component/Template Libraries



Click on the Restore icon to open the Restore Data dialog which allows you to Restore previously backed up Component/Template libraries. Select the source of the restore - you can either restore the libraries to their state when you first installed DesignBuilder or you can restore library data from a [previous backup](#) (if one exists). In the screenshot below, no backup had been made so the [Previous backup](#) option is not available.



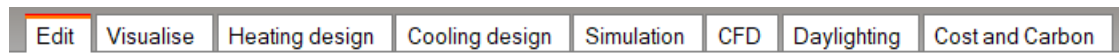
If you choose to restore libraries supplied with installation you can use the **Restore custom data (clear to delete all custom data)** option to either:

1. Restore the custom data which existed on the computer at the time the software was installed, or,
2. Clear all custom data to keep only the data supplied by DesignBuilder.

Note to restore the libraries supplied with the installation and clear any custom data select **Restore libraries supplied with installation** and uncheck the **Restore custom data (clear to delete all custom data)** option.

Outputs And Calculations

To the right of the [Edit screen](#) tab are a further set of screen tabs which provide access to calculations and output reports. The image below shows the screen tabs at the top of the DesignBuilder main form, as seen when in non-Learning mode. When Learning mode is active the same list is provided at the bottom of the screen.



- "Visualisation" on page 1525
- [Heating design simulation](#)
- [Cooling design simulation](#)
- [Simulation using hourly weather data](#) (simply referred to as 'simulation' in DesignBuilder documentation)
- [CFD](#)
- [Daylighting](#)
- [Cost and Carbon](#)
- **Calculate** - for SBEM and DSM calculations (not covered in this Program help).

[Display Options](#) in the Panel in the bottom left of the screen allow you to change the data displayed and the presentation style.

Results can be [Reported](#) in a variety of file formats.

See also the [DesignBuilder User Interface – Simulation Screens: Overview of visualisation and calculation screens](#) tutorial

Visualisation

The Visualisation screen provides various ways to visualise the model inputs either using false colours to represent the various option settings and also with full 3-D rendering including textures, glazing shading etc.

From the Visualisation screen you can:

- [View model data](#) graphically using false colours.
- [View a detailed rendered image](#) of the model with textures and shading.

Visualisation - Model Data

The Model Data tab of the Visualisation screen provides tools to display the model data in graphical form.

The following options are available by making an appropriate **Plot type** selection in the Display options panel in the bottom left of the screen.

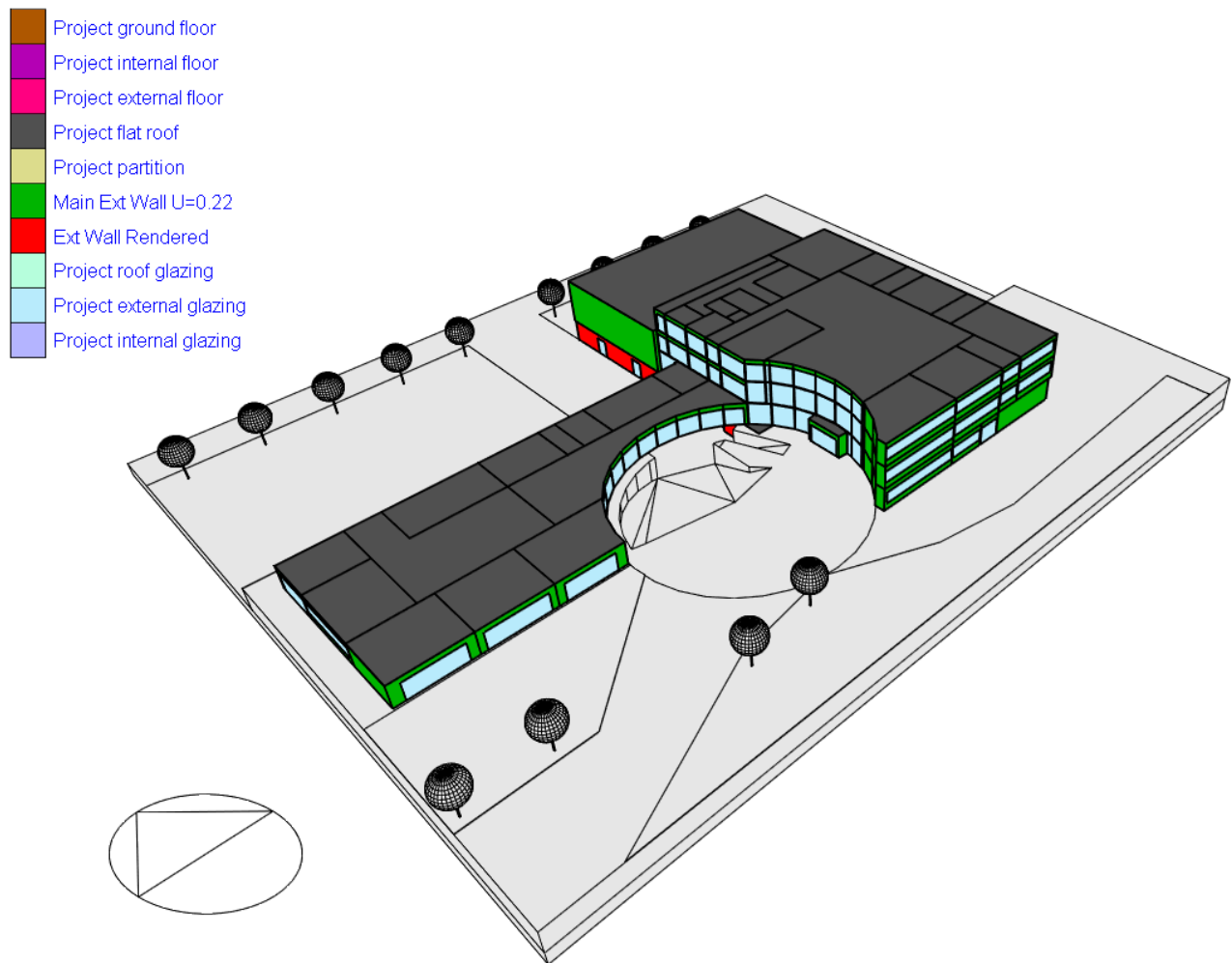
- [1-Constructions and glazing](#) - displays the Construction and Glazing components used for each surface and window in the model
- [2-Linear thermal bridging](#) - displays the linear thermal bridges for the various edges in the model.

Constructions And Glazing

When the **1-Constructions and glazing** Plot type option is selected, the view on the **Model Data** tab shows the model with each surface rendered using a colour that defines the construction or glazing component applied.

This data can help provide a global view to check that all construction and glazing settings across the whole model have been made as required.

The screenshot below shows a typical external view of the constructions and glazing model data visualisation.

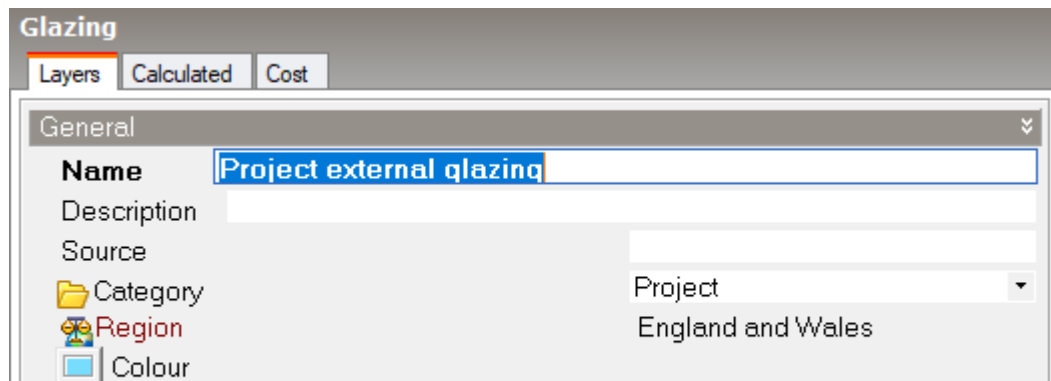


The legend on the left of the view provides the list of constructions and glazing components and the colour associated with each. In large models the list can be long and in this case you may find a scroll arrow at the bottom and/or top of the legend list to allow you to access the next/previous page.

Tip: The [Constructions](#) and [Glazing](#) dialogs both allow you to select a colour to represent that component when applied to surfaces and openings. This is the colour used to shade surfaces and windows on this view.



Select colour to represent this construction in model data visualisations



Select colour to represent this glazing type in model data visualisations

Using the Data Visualisation Tools

As well as all the usual view control tools (orbit, zoom etc), some extra tools are provided to help you get the most out of the model data visualisation and these are described below. You will need to use either or both of the Surface removal or Section cut tools, described below, to view data for internal surfaces.

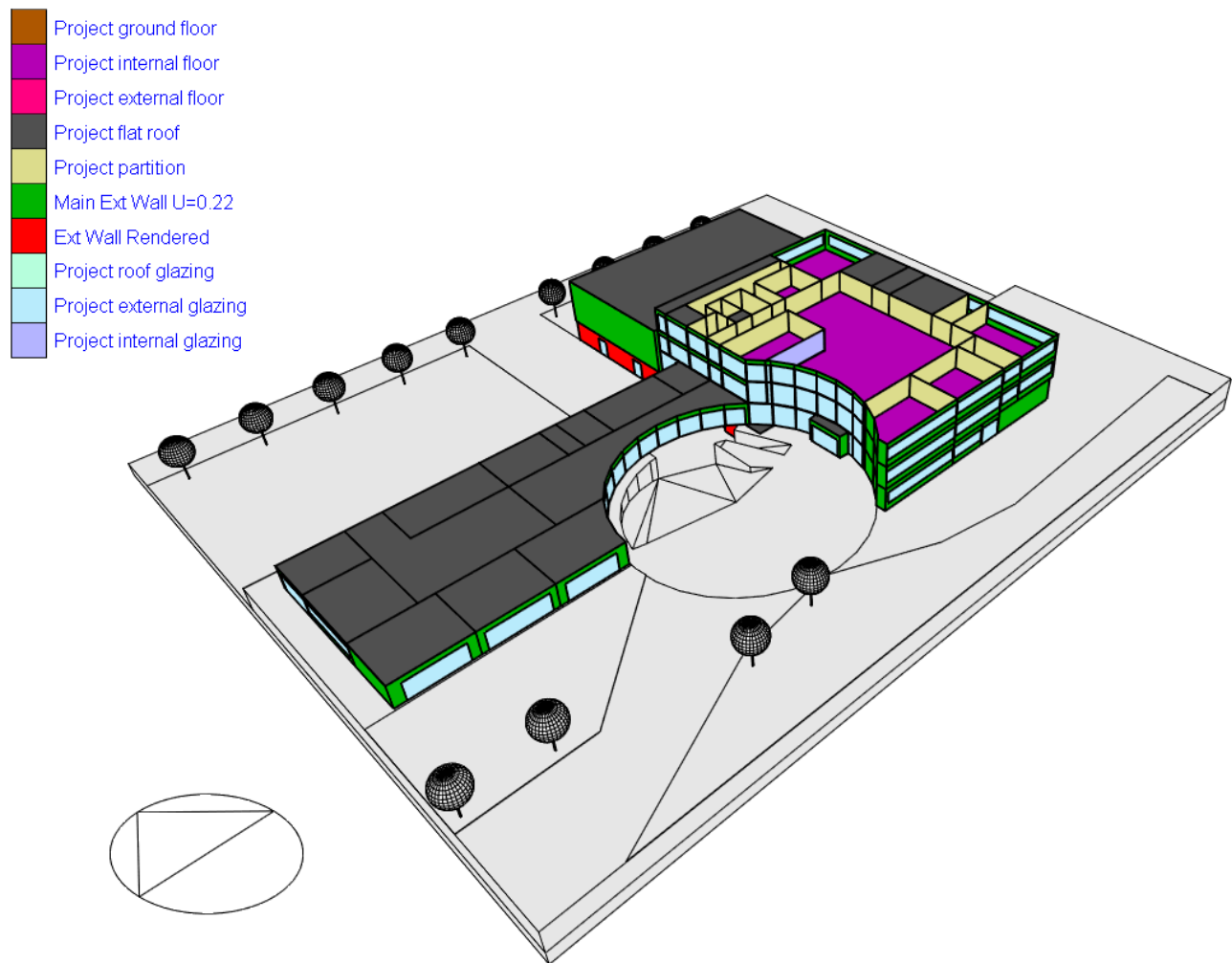
Surface removal

You can remove surfaces to allow internal surface data to be viewed by clicking with the mouse on any external or internal surface that you would like to be removed. You can use this tool to selectively expose data deep within a building without losing other data that you might also wish to view.

To restore a surface that was removed hold the <Ctrl> key down and click with the mouse again on the space where the removed surface should be.

To restore all removed surfaces press the <Esc> key.

The example screenshot below shows the same model as above but with some of the roof surfaces removed as described above to reveal internal surfaces.

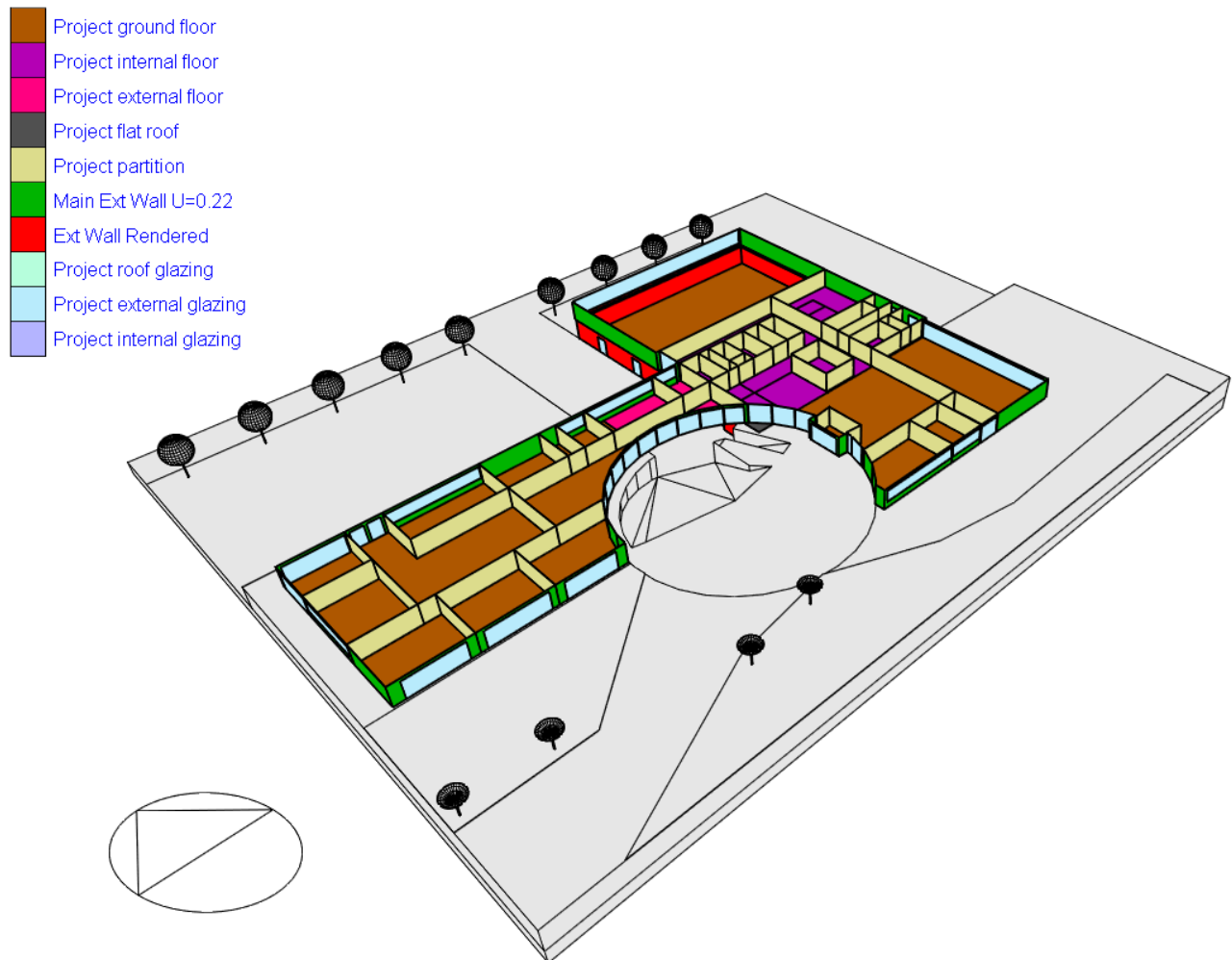


Note: Removing surfaces in this way only affects the model data visualisation view and does not affect the model itself.

Section cut

The [Section cut](#) tool can be used to cut a slice through the model to allow internal data to be viewed. This tool can be especially useful for viewing data across a whole floor, or across a whole façade.

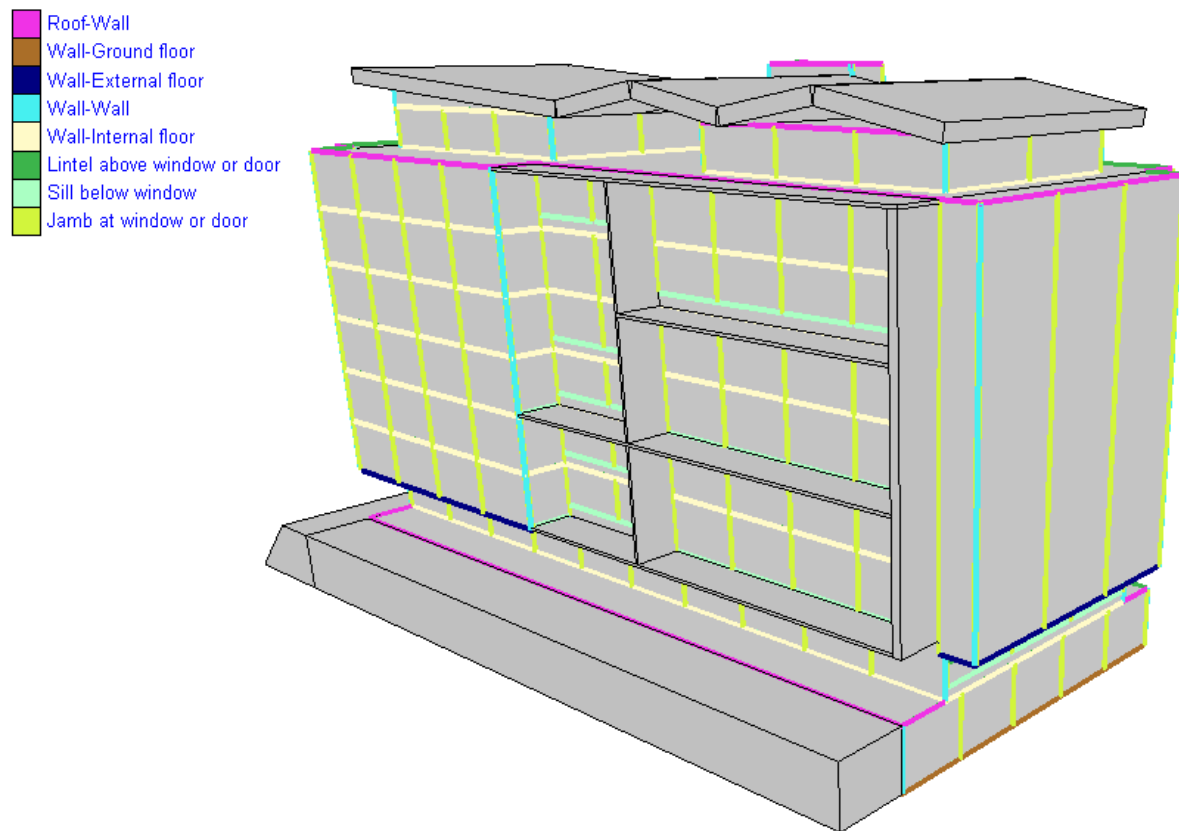
The example screenshot below shows the application of the section cut tool to allow the construction and glazing properties of internal constructions on the top floor to be viewed, as well as those for external surfaces.



Linear Thermal Bridging

When the **2-Linear thermal bridging** Plot type option is selected, the view on the **Model Data** tab shows the model with each edge rendered using a colour that defines the class of [linear bridge](#) applied. This enables a visual inspection of the thermal bridges at specific geometric junctions (like wall-to-wall corners) and can help clarify how the various classes of linear bridging are applied to the building geometry.

The screenshot below shows a typical external view of the Linear thermal bridging data visualisation.



Using the Data Visualisation Tools

Identifying bridge lengths

To identify the length of any individual bridge, simply click on it using the mouse, and DesignBuilder will display its length in a tooltip.

Visualisation - Rendered View

From the Rendered View tab on the Visualisation screen you can:

- Access a realistic rendered view of the model showing surfaces with textures.
- View the effects of [solar shading](#) for any day of the year
- [Export](#) rendered images in various formats
- Generate AVI [movies](#) of the scene.

Visualisation Controls

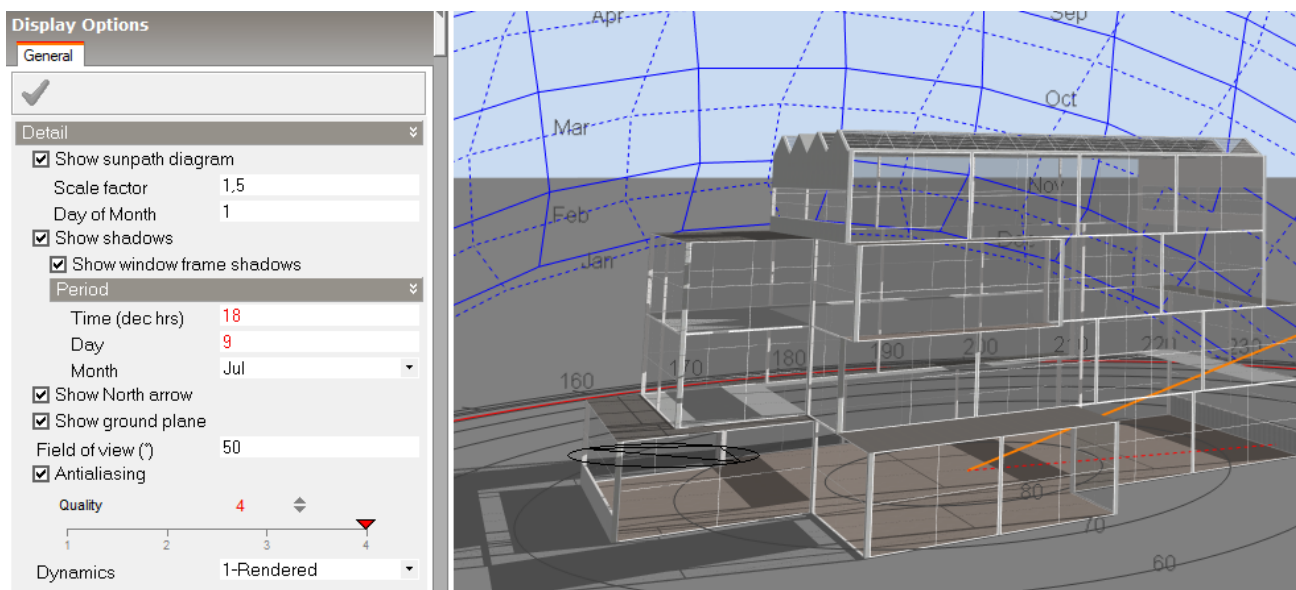
Controls are also provided to customise the rendered image:

[Ref1187362187](#)

- "Sunpath" on page 1531
- [Show shadows](#)
- [Show North arrow](#)
- "Show Ground Plane" on page 1535
- "Field Of View" on page 1536
- [Antialiasing](#)

- [Dynamics](#)

These controls are accessed from the Display Options panel on the Visualisation screen:



The following controls are available to customise the view:



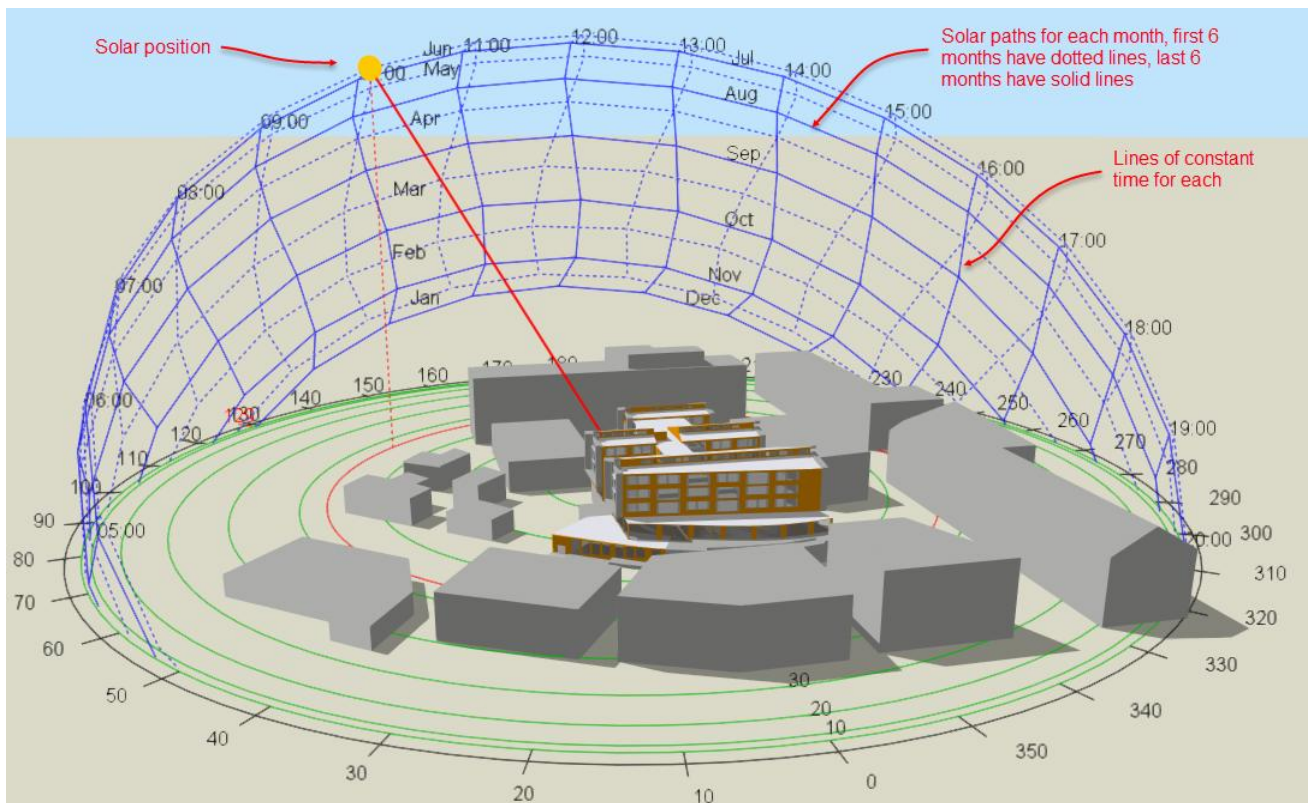
- [View Rotation](#)
- [Orbit](#)
- [Zoom](#)
- [Fit view](#)
- [Pan view](#)
- [Zoom window](#)
- "Section Cut and Clipping Plane" on page 1540

"Visualisation Rendered View Limitations" on page 1540

Sunpath

Display options on [Visualisation](#) screen

The Sunpath diagram helps to visualise the paths taken by the sun as it moves through the sky at all times of the year. You can display a Sunpath diagram for the building at its current location by checking the **Show Sunpath diagram** Display option and pressing the Apply button. This will display a Sunpath diagram similar to that shown below.



Sunpath diagram scale factor

There is an option to override the default scale factor applied to the Sunpath diagram. With a scale factor of 1 the Sunpath diagram fits snugly round the building and any building-level component blocks. The default scale factor is 1.3.

Sunpath day of month

By default DesignBuilder uses the first day of the month when generating the sunpath lines for each month of the year. This option allows you to override that and use any day of the month from 1 to 28.

Change solar position tool



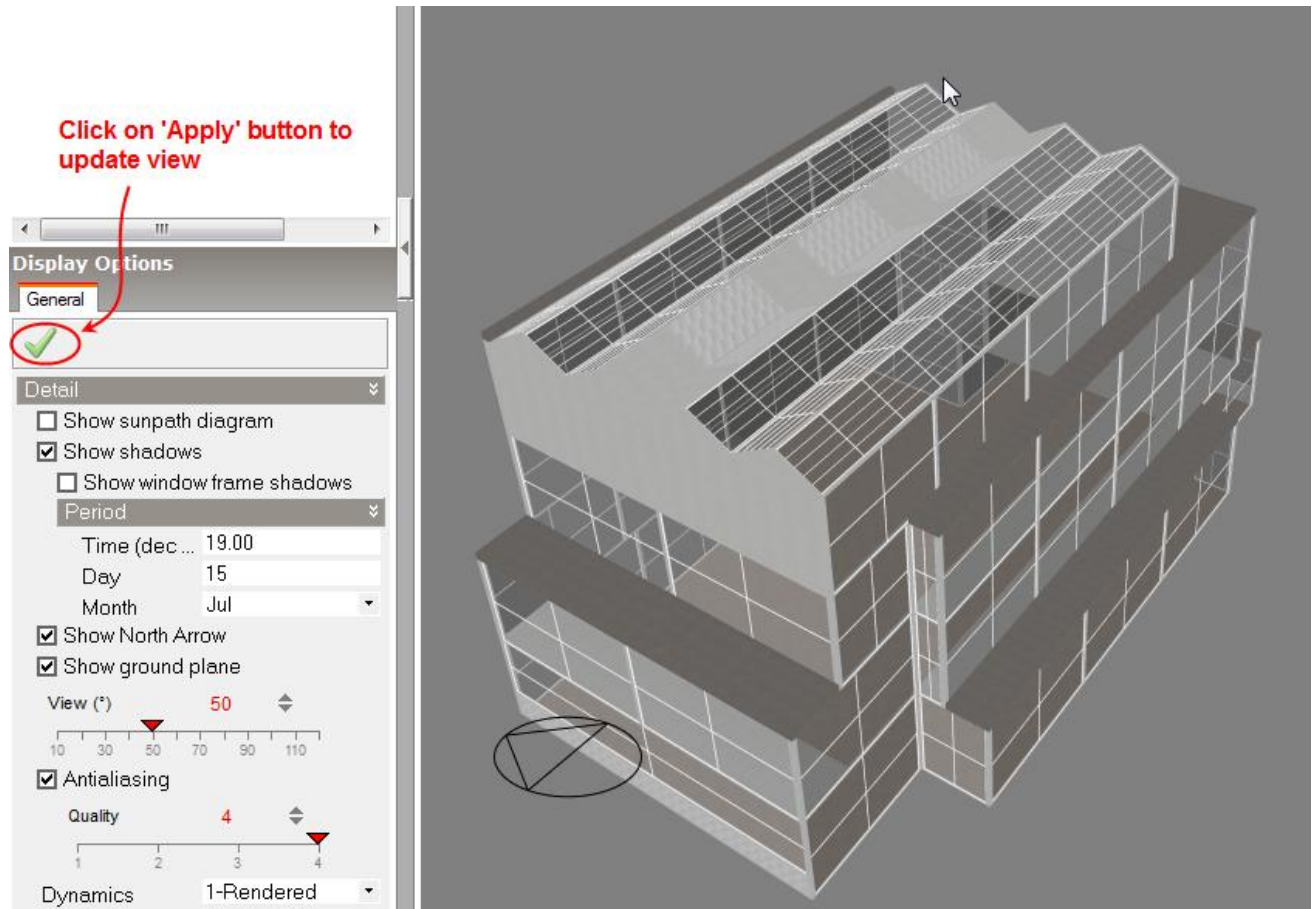
When both the **Show Sunpath diagram** and **Show shadows** Display options are set, the **Change solar position** tool becomes available. This allows you to move the sun around the diagram. To do this first click on the **Change solar position** toolbar icon (above). When this tool is active the cursor become a cross and you can use it to click on a junction of time/month sunpath lines to set the new solar position. You can continue to move the sun around the sky by clicking on the diagram in the same way. An alternative way to use this tool is to drag the sun around the sky. You will notice that the Time of day and Month data in the Display options panel are updated along with shadows as the sun moves through the sky. When you have finished press the <Escape> key to return to Select mode.

Tip: You can use the Sunpath diagram with the **Create movie** tool to show the sun moving through the sky for the current design month. To do this select the **2-Shading for design day** option on the **Movie options data** dialog box

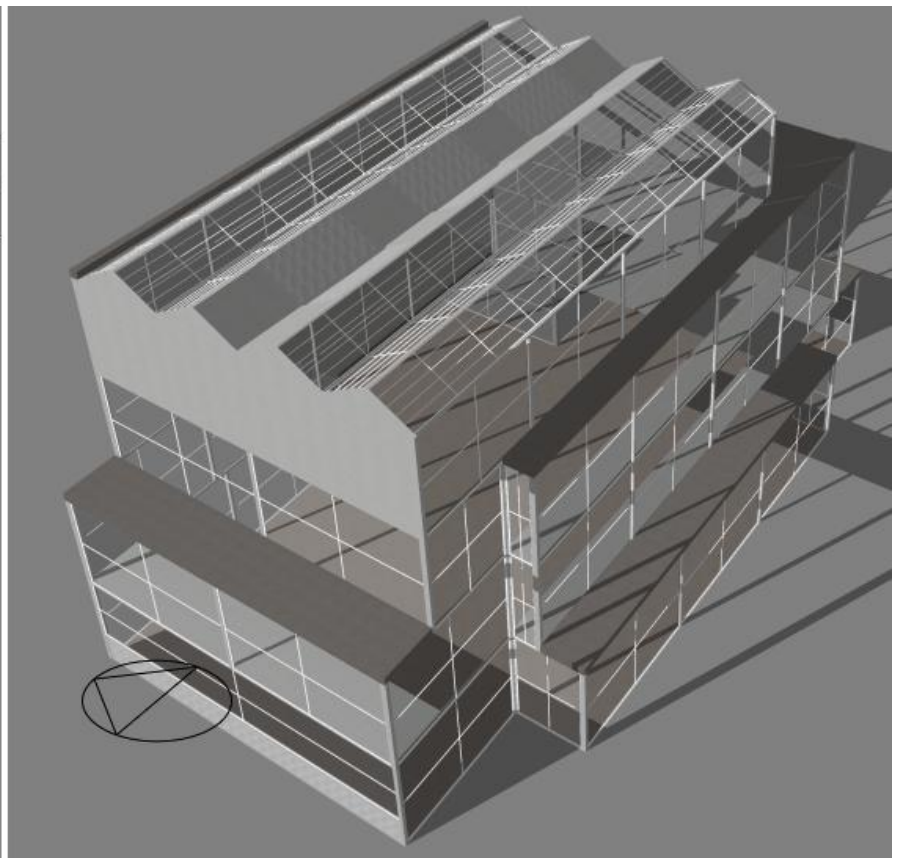
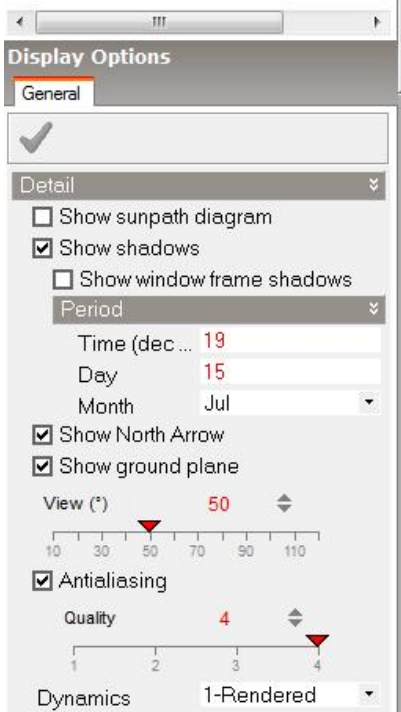
Show Shadows, Time, Day, Month

Display options on [Visualisation](#) screen

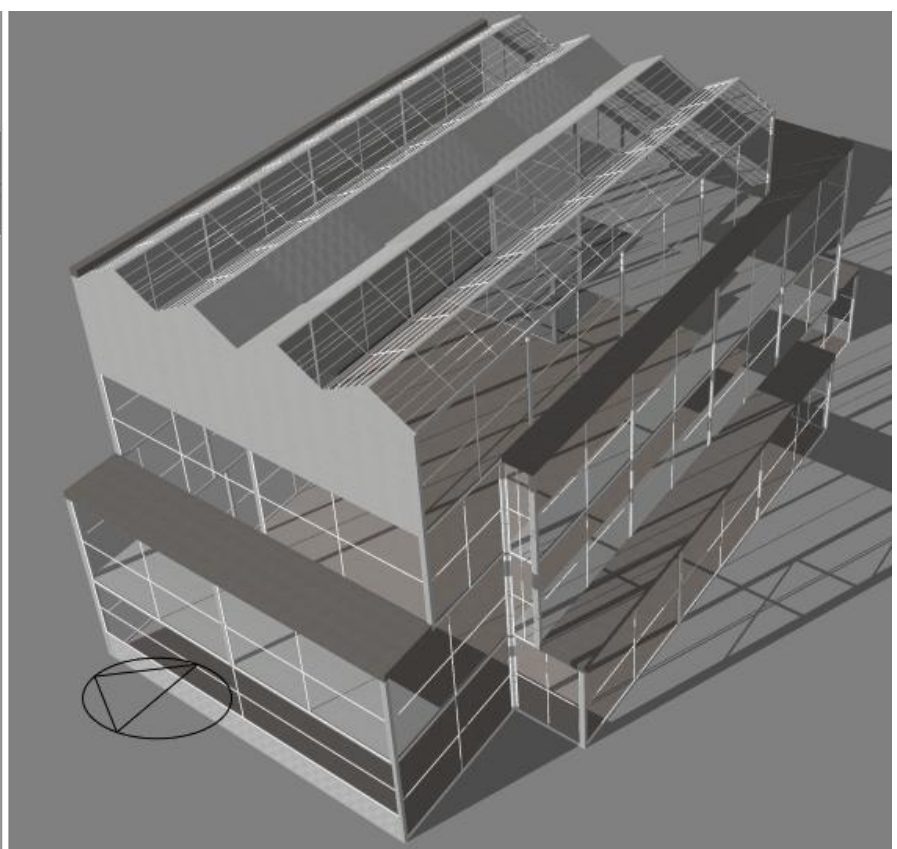
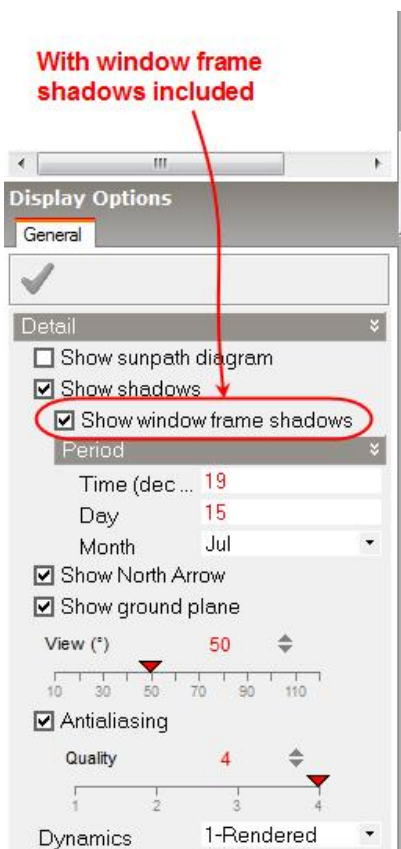
You can incorporate solar shading within the rendered model view by switching the **Show shadows** check box on and then clicking on the **Apply** button:



After clicking on the **Apply** button, a progress control appears to inform of shadow generation and then the view is updated:



If you want to look at more detailed shading, you can include the window frame dividers in the shading calculations by checking the **Show window frame shadows** check box and clicking the **Apply** button:

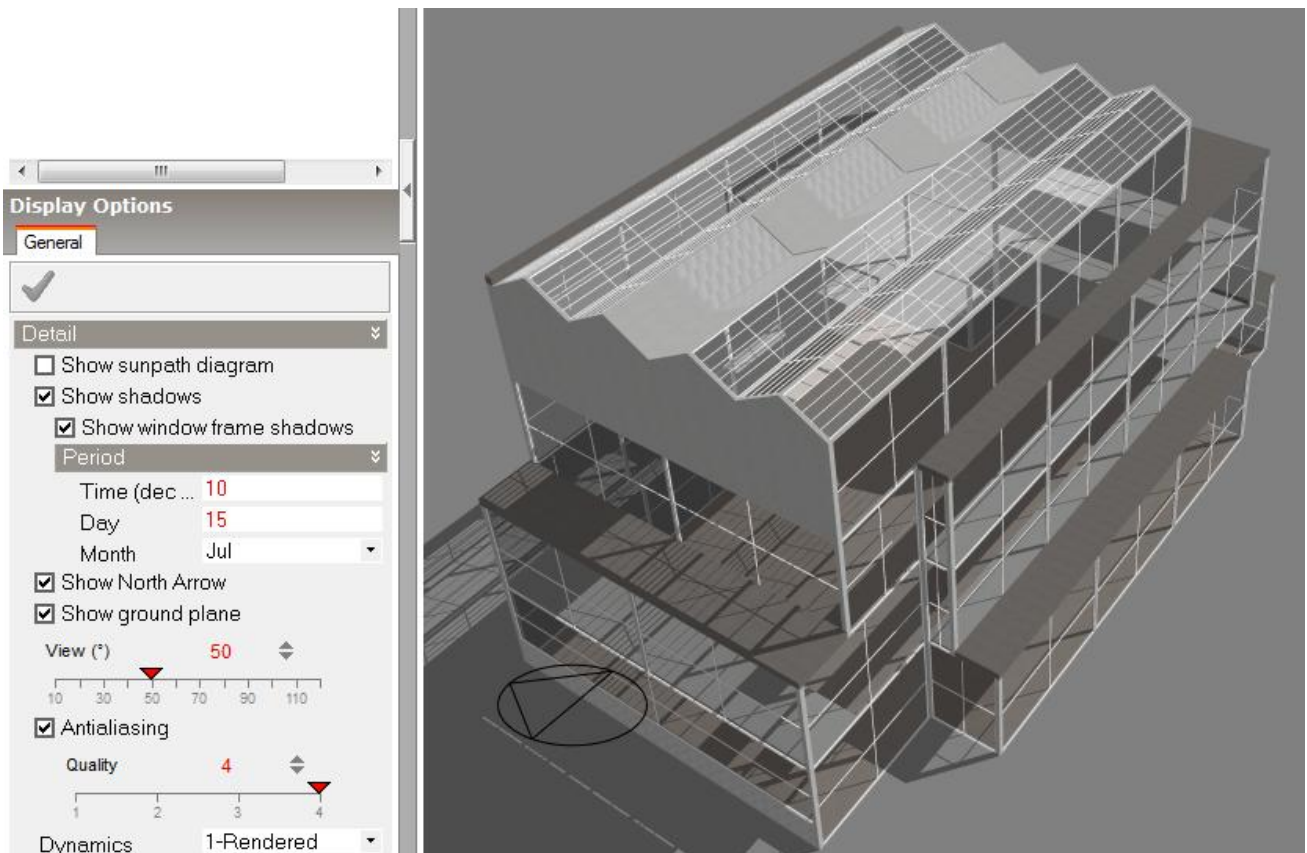


Period

The month, day and time-of-day can be controlled using the controls under the **Period** header. Note that the time specified here is local mean time excluding any daylight saving. You can account for daylight saving time by adding 1 hour to the local mean time during summer months to see shading for clock time.

Note: Time is defined in decimal hours, e.g. '10.5' is the same as 10:30 a.m..

In the screenshot below the time has been changed to 10:00 and the shadows adjust accordingly.



Show North Arrow

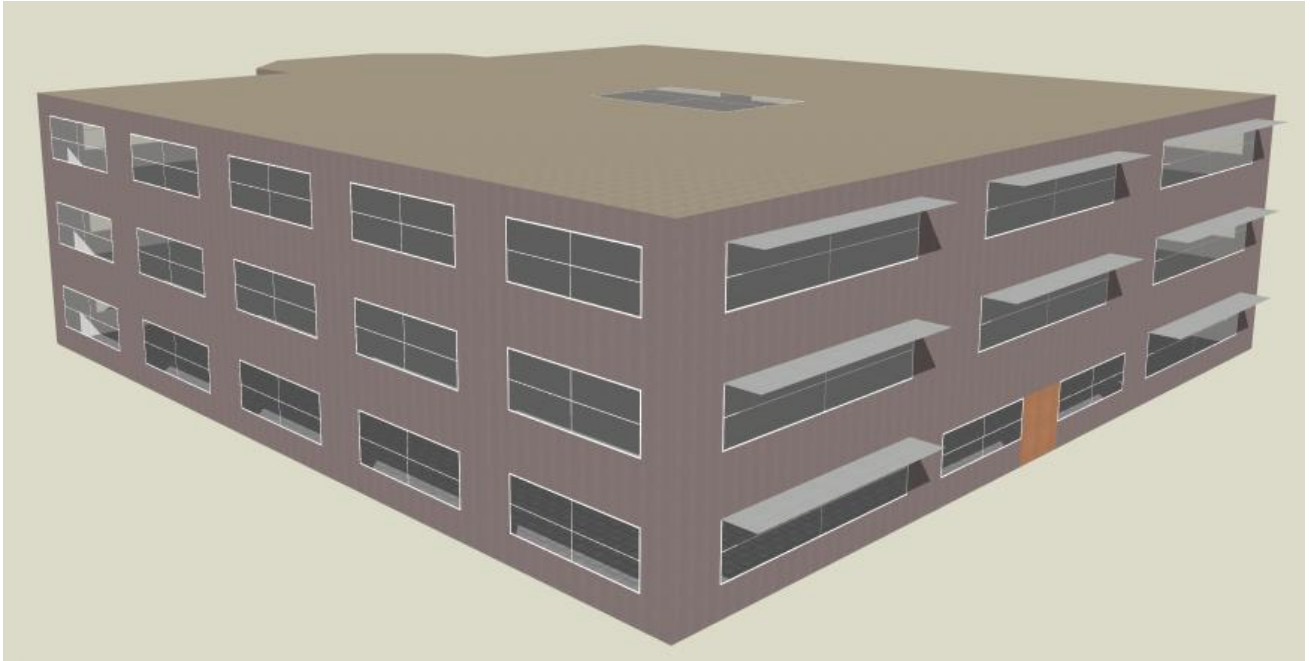
Display options on [Visualisation screen](#)

Use this control to switch the display of the North direction arrow on or off.

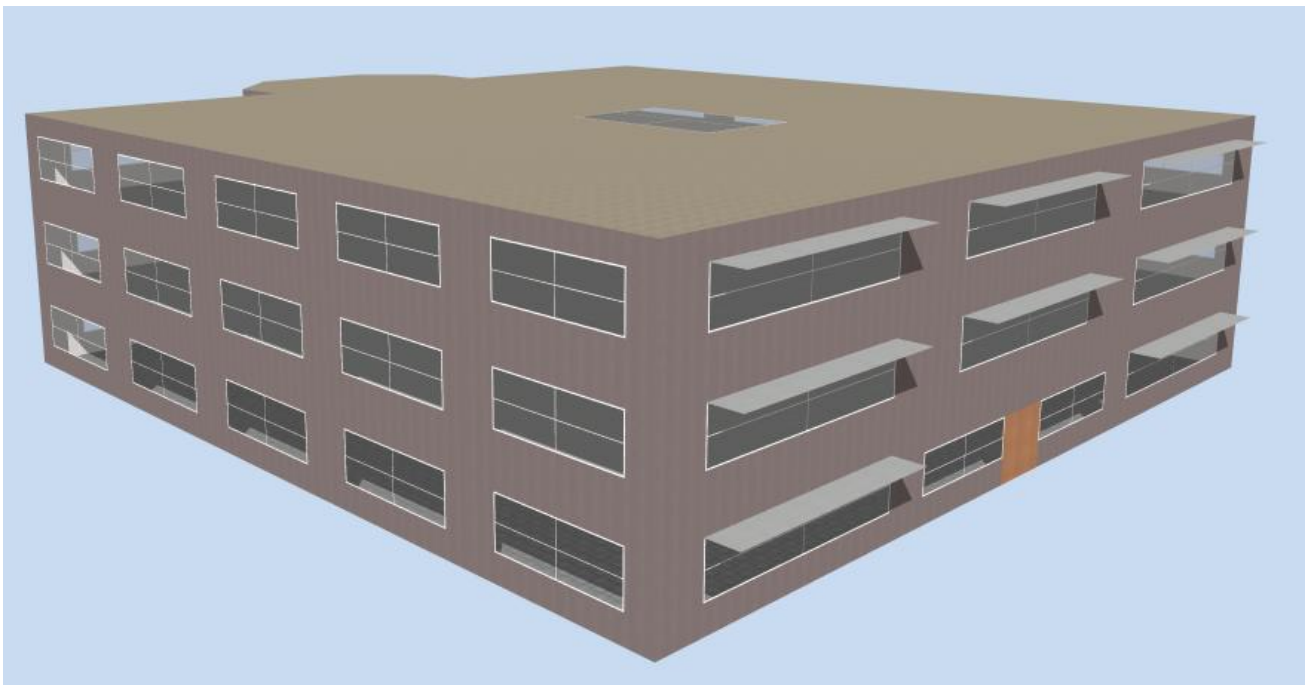
Show Ground Plane

Display options on [Visualisation screen](#)

Check this option to see the ground plane on the Visualisation screen, or uncheck it to remove the ground plane. It can sometimes be useful to remove the ground plane to gain a clearer view of basement blocks. Hiding the ground plane can also make it easier to obtain a "building only" view, allowing images taken from the model to be placed on top of custom backgrounds using image processing software.



Ground plane displayed

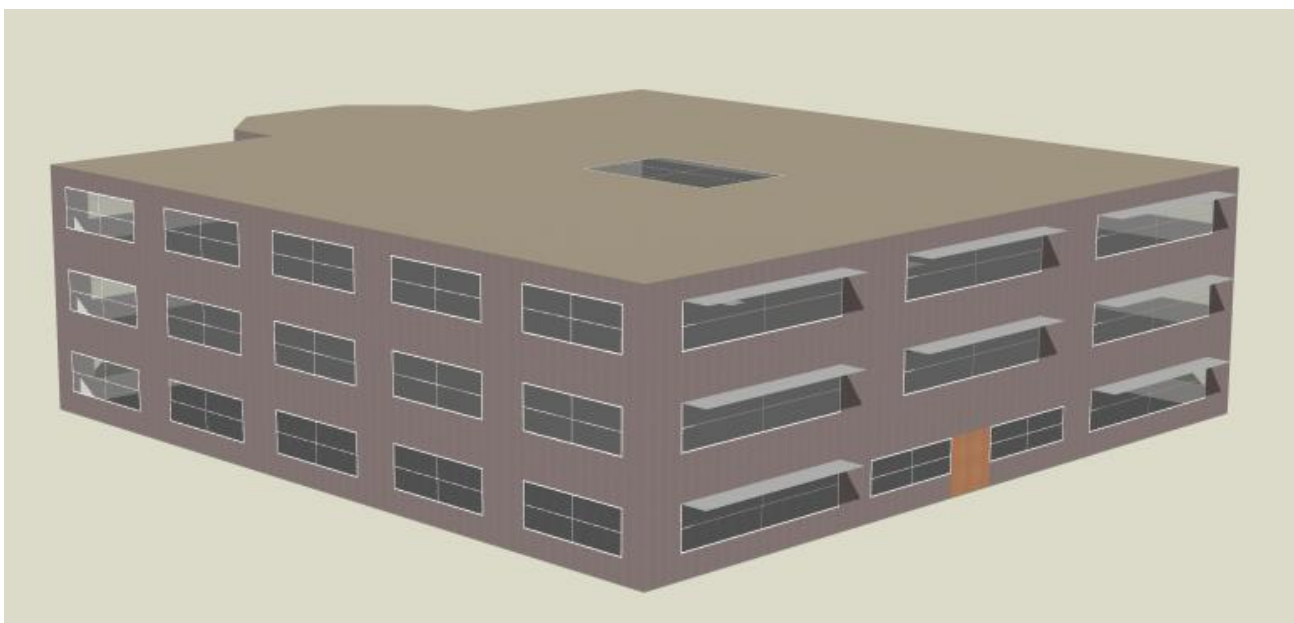


Ground plane hidden

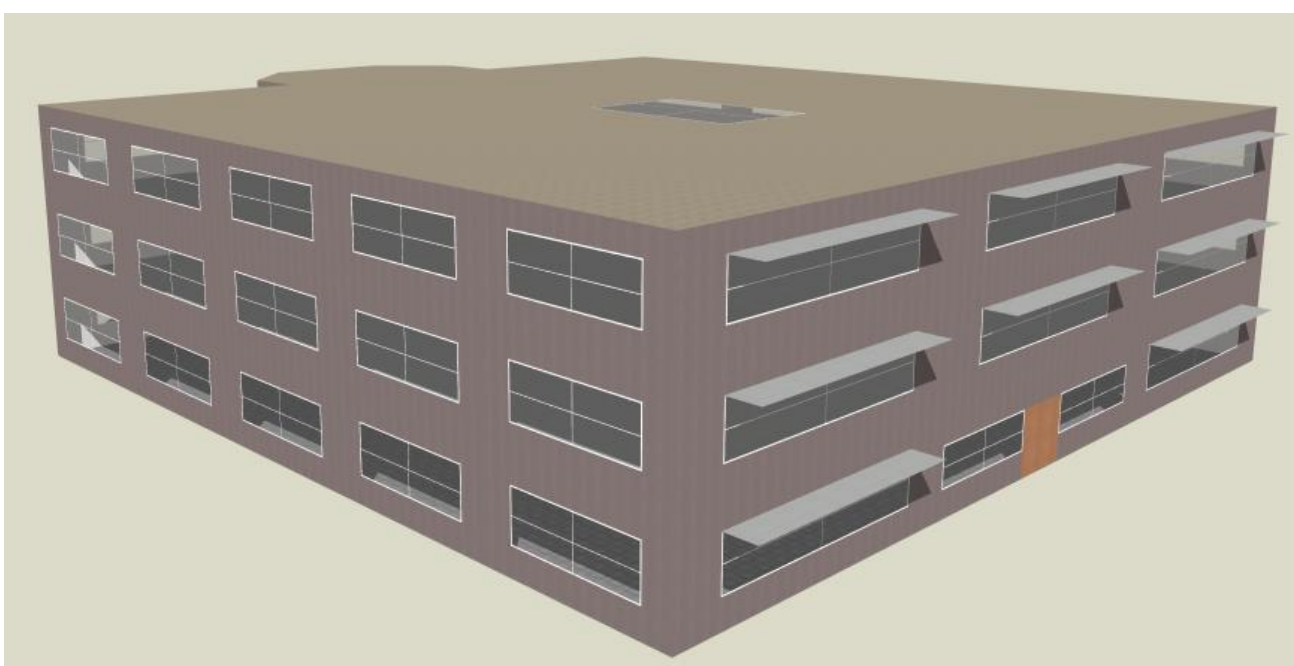
Field Of View

Display options on [Visualisation](#) screen

The **Field of view** is the extent of the rendered model that is seen on the display at any given moment. It affects perspective in the view. A low **Field of view** value such as 10° is similar in effect to using a telephoto lens which "flattens" perspective, giving an effect more like the isometric view used on the model edit screen. A high value like 90° allows more of the model to be seen on the screen at any one time which can be especially useful for interior views. However this does exaggerate the effect of perspective giving distorted angles. These effects can be seen in the screenshots below.



Field of view = 10° (telephoto with flattened perspective)



Field of view = 50° (normal view, default setting)



Field of view = 90° (wide angle with exaggerated perspective)

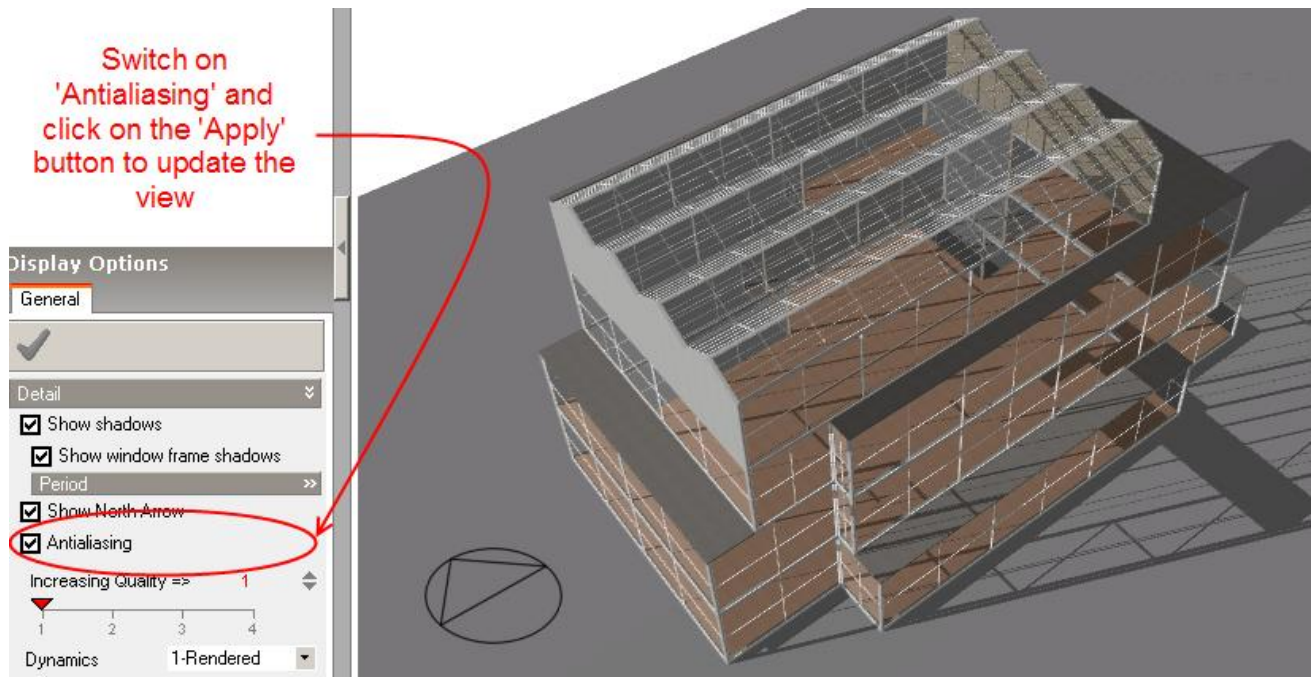
Antialiasing

Display options on [Visualisation](#) screen

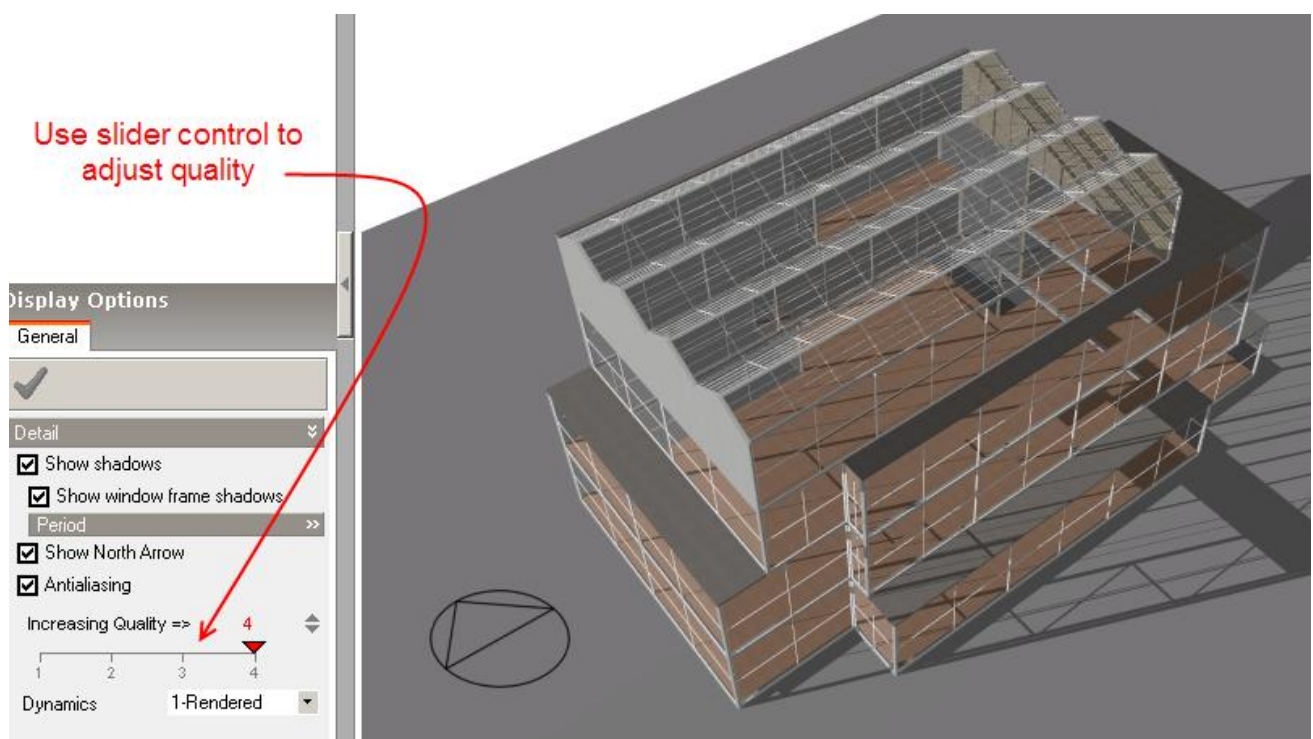
The antialiasing option can be used to improve image quality. You may need to experiment with antialiasing to confirm whether or not the graphics card fitted to your computer will facilitate this feature - with some earlier cards, antialiasing can be very slow.

Tip: more advanced graphics cards can provide [hardware antialiasing](#) and in this case this software antialiasing option may only provide marginal benefit.

To enable antialiasing, switch the **Antialiasing** check box on and click on the **Apply** button:



You can improve the quality of the image using the **Increasing Quality** slider control:



Bear in mind that as you increase image quality, the display speed will decrease.

See also the [Shading via louvers, overhangs, and side fins](#) tutorial

Dynamics

Display option on [Visualisation](#) screen

With larger and more complex models, orbit and zoom control speed can be increased very significantly by selecting 'Wireframe' from the 'Dynamics' control drop list.

Visualisation Rendered View Limitations

Visualisations are based on the same model as that used for the calculations but you may notice some slight differences in the way the model is displayed vis-a-vis the floor constructions and [Zone volume calculation](#) options.

1. DesignBuilder uses a single floor 'slab' for each block even when some floors in the block may have different constructions. This is done for simplicity and also to ensure the most realistic images are produced without step changes in the floor level throughout the block. So there is only limited scope for adjusting the height of floor level in the visualisation.
2. Where the **Ground floor construction is below ground and is not subtracted from zone volume** option is selected, the top of the ground floor will be shown in the visualisation screen as being at ground level - this is consistent with the data. However you will not be able to see the ground construction below the ground if you adjust the camera level to be below ground.

Section Cut And Clipping Plane

Accessible from the [Visualisation](#), [CFD](#) and [Simulation Data Visualisation](#) screens.



The **Section cut** tool can be used to create a "clipping plane" through the building to allow the inside of the model to be viewed. It can be especially useful for viewing data across a whole floor or façade.

To use Section cut:

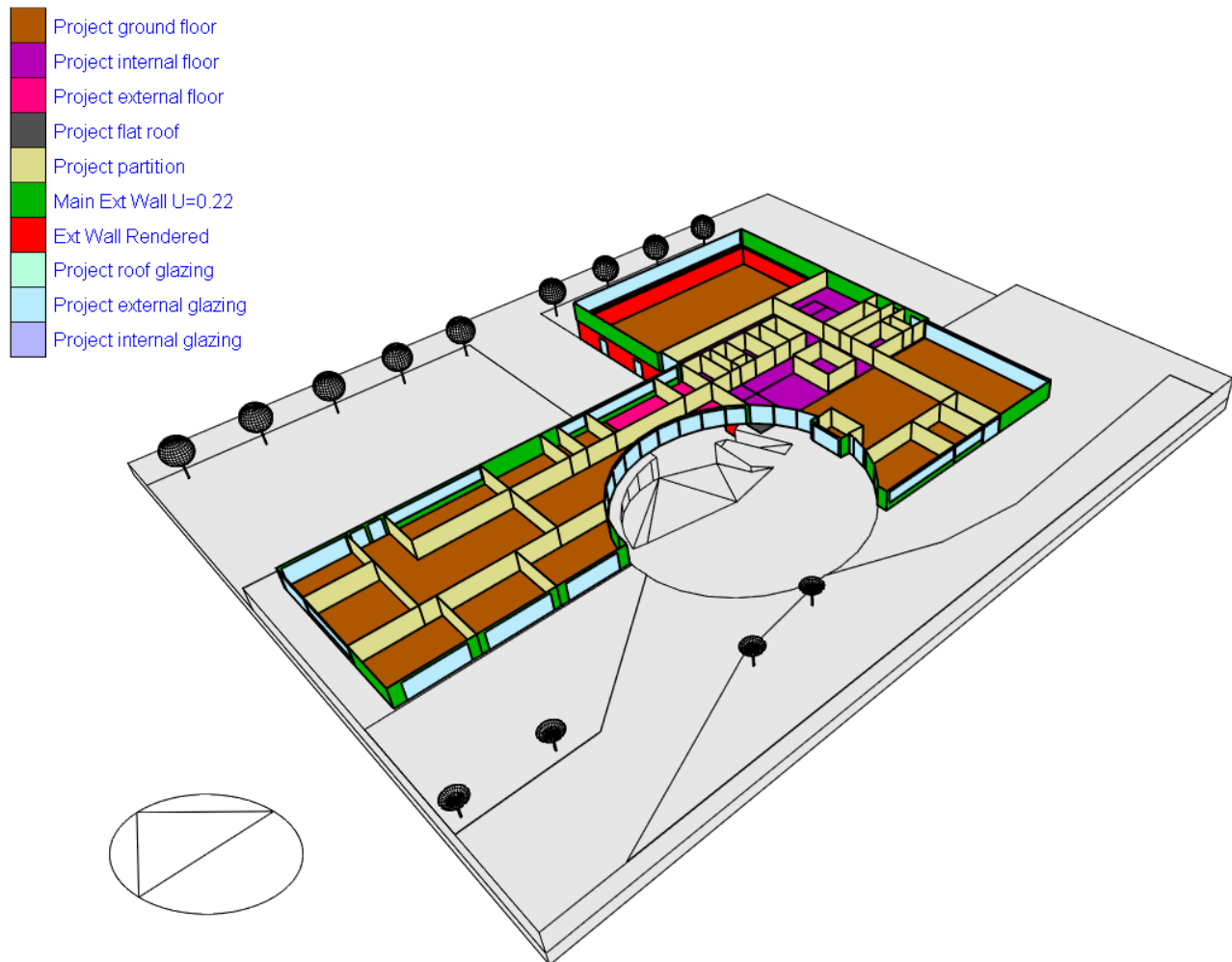
1. Click on the Section cut toolbar icon,
2. Select the direction of the cut using the 3 axis arrows in the top right of the screen and then
3. Move the mouse over the model to adjust the position of the cut.
4. Once you are happy with the cut position, click once with the mouse.

You can continue to use other view tools with the cut in place.

To subsequently change the position or direction of the cut press the <Esc> key and repeat the steps above to select the new cut position and direction.

You can change the direction of the section cut by using the Section Cut, **Orientation** setting at the top of the Display Options panel when the Section cut command is active.

The example screenshot below shows the application of the section cut tool to allow the construction and glazing properties of internal constructions on the top floor to be viewed, as well as those for external surfaces.



Note: with the cut in place you can view the shading (rendered view) and other results (data visualisation view) within the building. The section cut does not affect shading or other results - it just affects the view that you have on the model.

See also the [Drawing holes and merging zones](#) tutorial

Clipping plane height

Also, when at block level on the Visualise and CFD screens it is possible to enter a clipping plane height in the Display Options panel simply by entering a height above the block floor level at which the cut is to be made. The default clipping plane height is 2.5m.

Show Surface Edge Lines

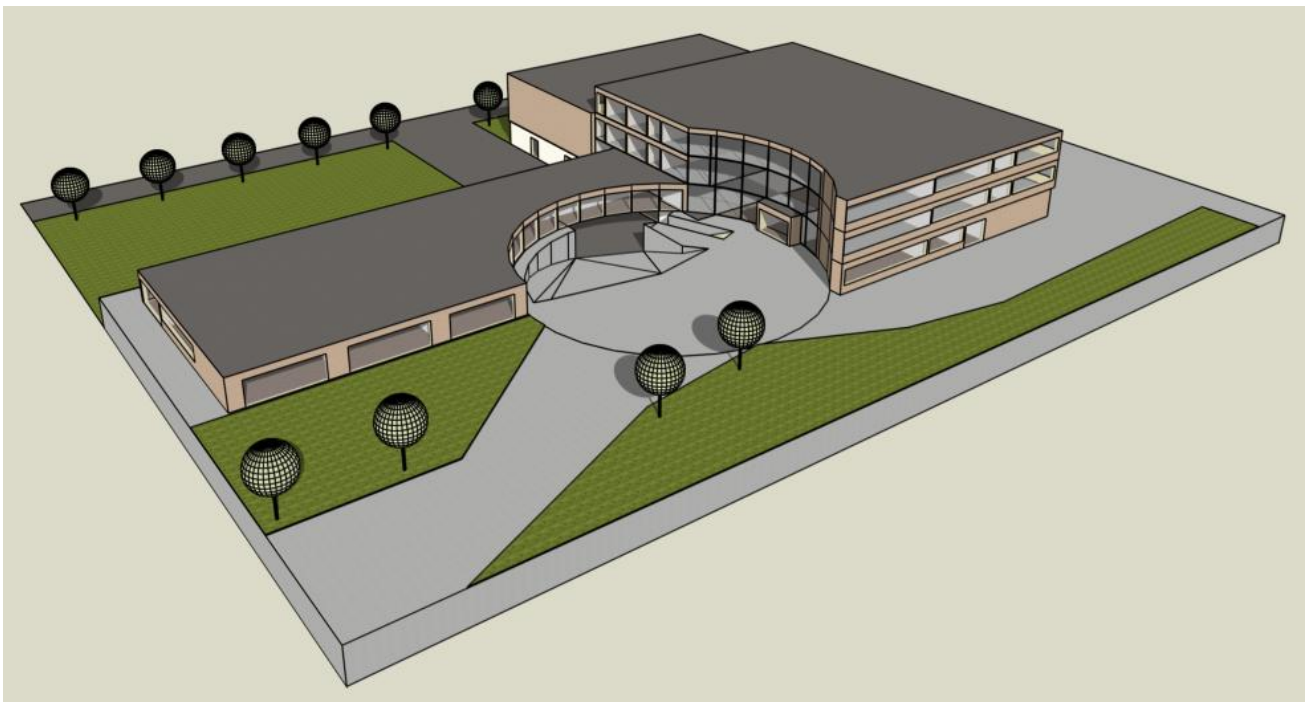
For models where adjacent surfaces in a model have the same texture it can be useful to show black lines around the edge of surfaces to help distinguish between the different surfaces. You can achieve this by checking one or both of the **Show external surface edges** and **Show internal surface edges** checkboxes in the Display options panel in the bottom left of the screen. These options are described in more detail below.

Show external surface edges

When viewing the model from *outside* the **Show external surface edges**, when checked, helps ensure that all external block edge surfaces are clearly delineated. The screenshots below illustrate the effect of changing this setting.



Show external surface edges: Off



Show external surface edges: On

Show internal surface edges

When viewing the model from *inside* the **Show internal surface edges**, when checked, helps ensure that all zone internal surfaces are clearly delineated. The screenshots below illustrate the effect of changing this setting.

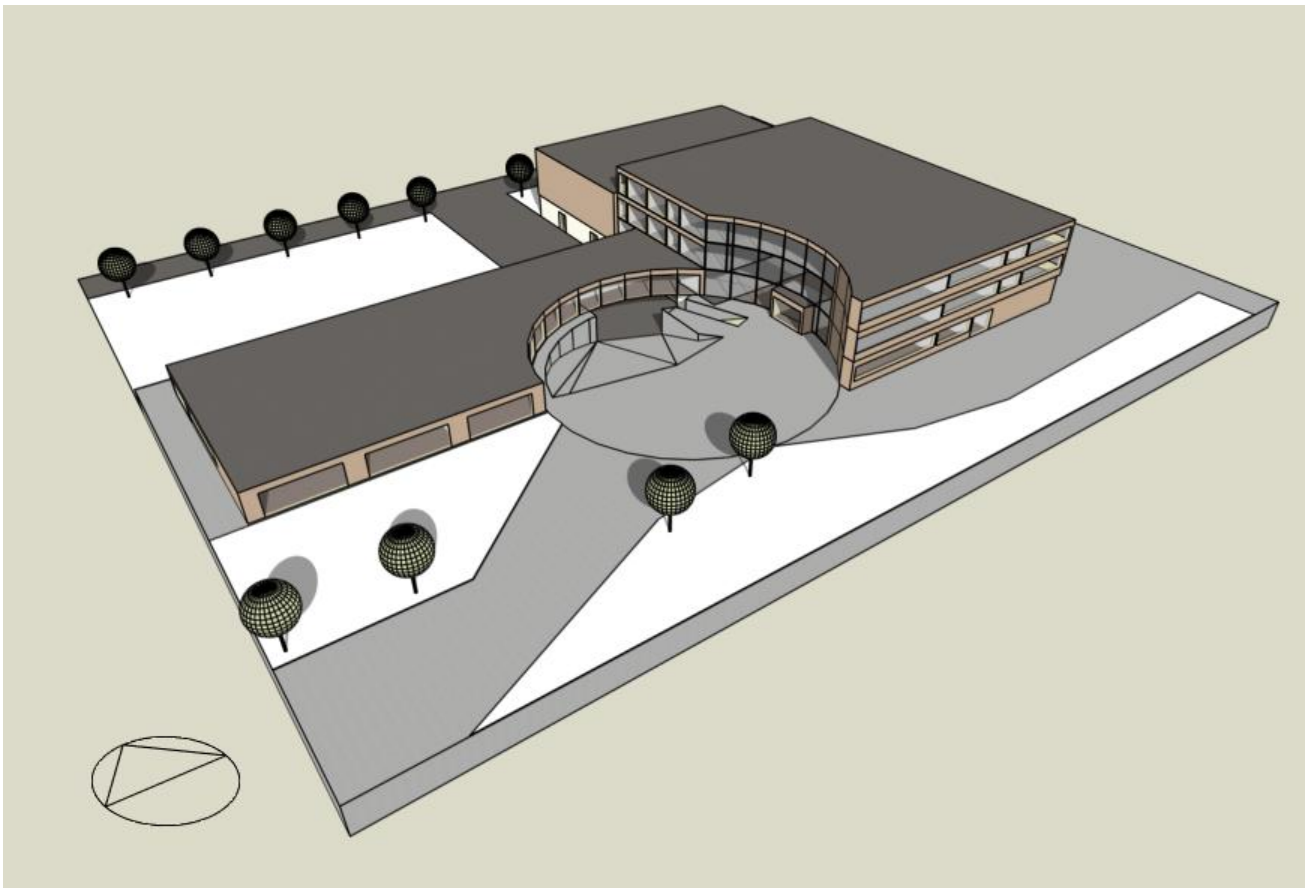


Show internal surface edges: Off

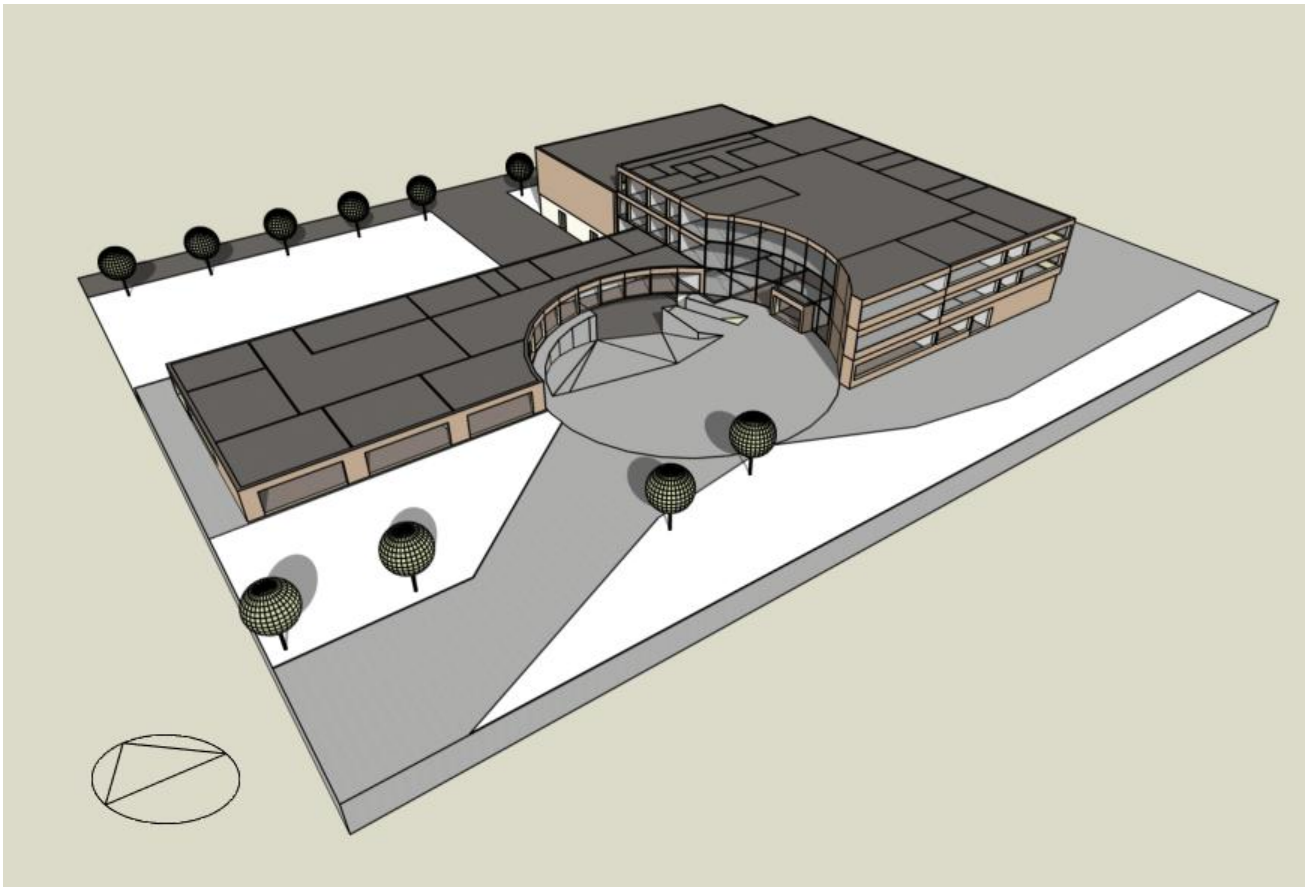


Show internal surface edges: On

Note: When a model includes external surfaces with zero thickness surfaces, it can be better to uncheck this option when viewing from the outside to avoid any lines from internal surfaces such as partitions bleeding through to the outside. See screenshots below.



Show internal surface edges: Off



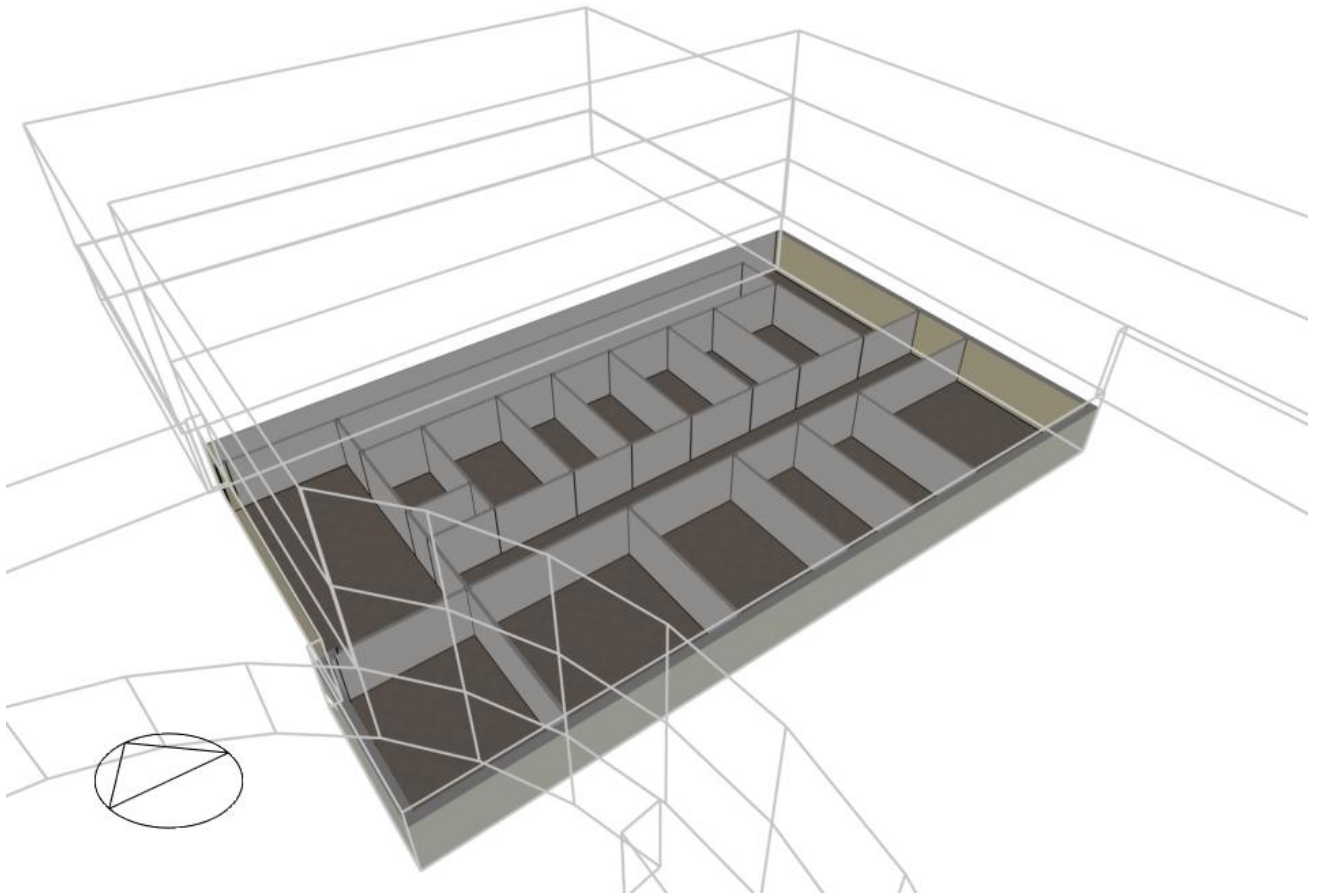
Show internal surface edges: On

Show Surrounding Blocks/Zones/Surfaces

When on the Visualisation screen, for rendered views of blocks, zones and surfaces, it is possible to view surrounding objects by the using the Show Surrounding Blocks/Zones/Surfaces controls as described below. This can help to clarify the context of the data being shown.

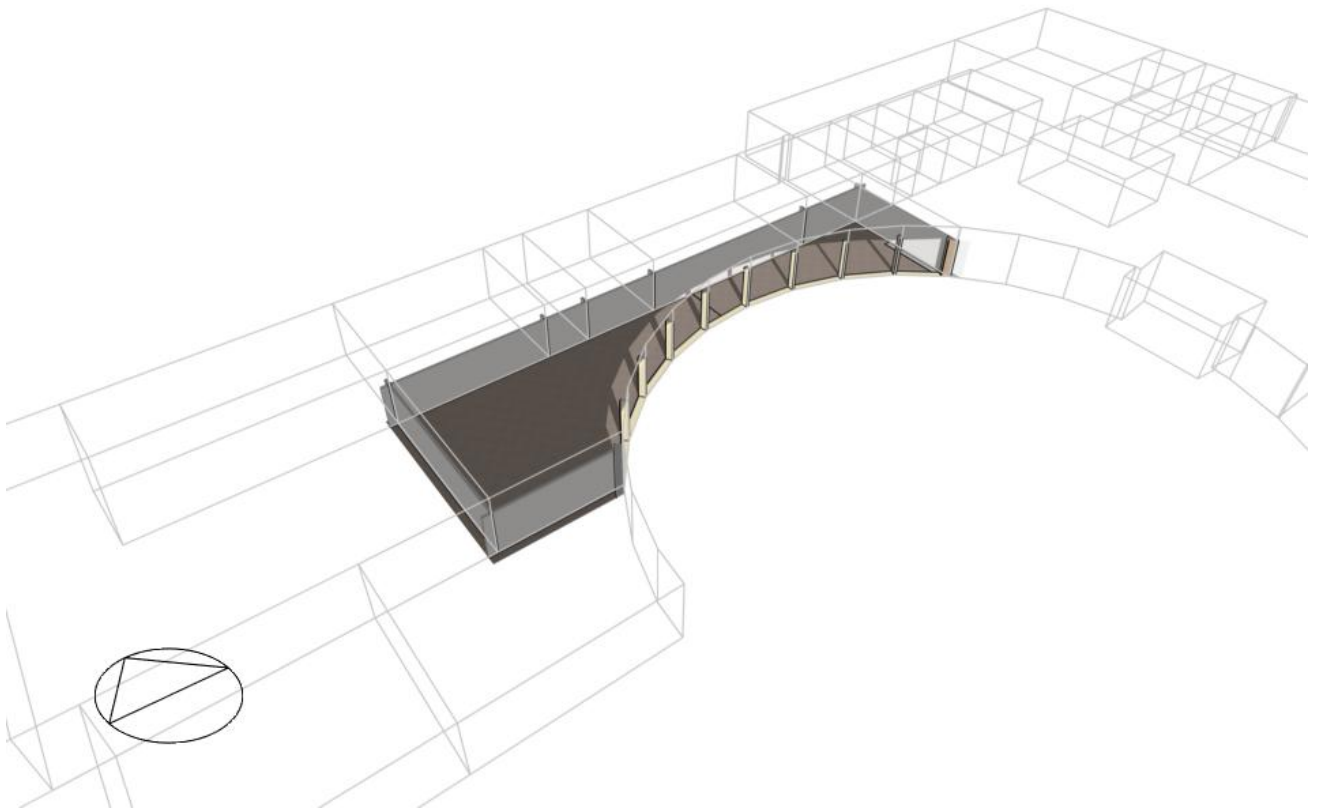
Show surrounding blocks

When at block level you can check the **Show surrounding blocks** checkbox to view the other blocks in the building in wire-frame as shown below.



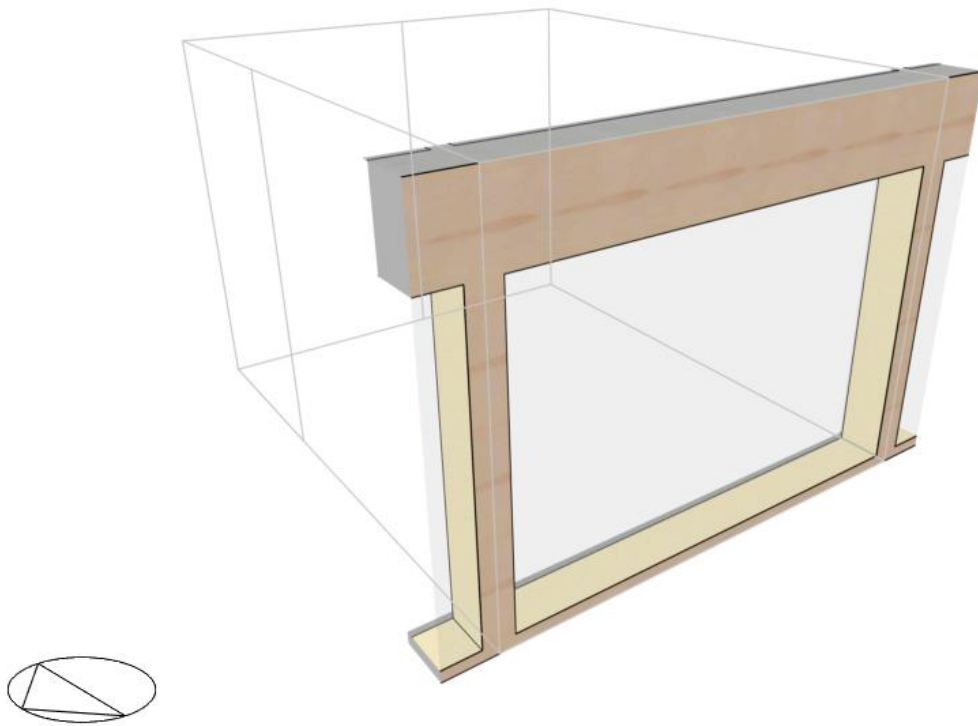
Show surrounding zones

When at zone level you can check the **Show surrounding zones** checkbox to view other zones in the same block in wire-frame as shown below.



Show surrounding surfaces

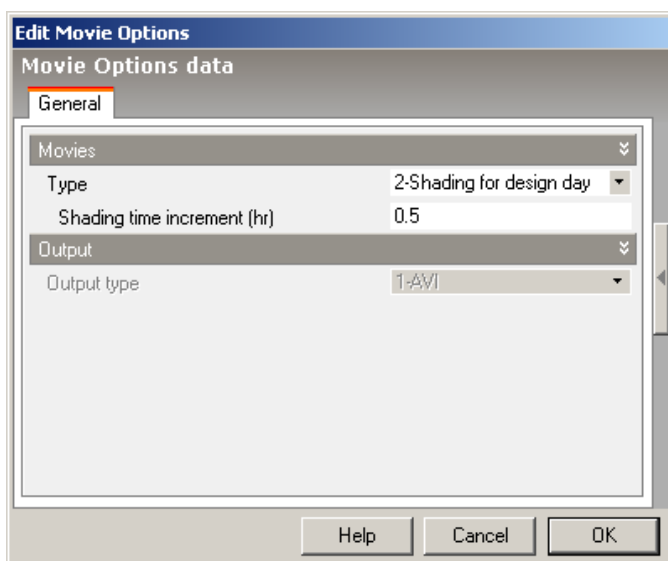
When at surface or opening level you can check the **Show surrounding surfaces** checkbox to view other surfaces in the same zone in wire-frame as shown below.



Creating Movies



When on the [Visualisation](#) or [CFD](#) screens you can create AVI movies of your model. To create a video file from a rendered view, click on the **Generate AVI movie** tool which opens the **Movie options data** dialog box:



Use the **Type** drop list to select either:

- **1-Orbit** - 360° orbit movie. This option generates a series of images, one for each angular Orbit increment and then allows you stitch them together to create an AVI file.
- **2-Shading for design day** - fixed camera position, varying the solar position over the [design day](#) as specified on the Visualisation Display options panel.

Tip: The **2-Shading for design day** option works particularly well with the [Sunpath diagram](#) displayed

Click on the OK button to generate the movie. After the movie generation has completed, a file save dialogue will be displayed to allow you to save the movie file to any required location.

Note: smaller **Shading time increments** and **Orbit increments** can generate a lot of data and take a long time to complete movie generation, especially for large models when software [antialiasing](#) is switched on.

Viewing AVI Files

To view AVI files on Windows 10 machines we recommend using **Windows Media Player** for best results. Otherwise if you use the default **Films and TV** program that is usually associated with the .AVI extension you are likely to see an error message like the one below:

Can't play

This item was encoded in a format that's not supported.

0xc00d5212

[Send feedback](#)

Close

This error means that there are missing codecs on the default video player application.

Alternately you could use another video player that supports the .AVI extension.

Heating Design Calculation

Heating design calculations are carried out to determine the size of heating equipment required to meet the coldest [winter design weather](#) conditions likely to be encountered at the site location. These design calculations are traditionally carried out using steady-state methods such as those offered by ASHRAE and CIBSE. You can use DesignBuilder to carry out effectively the same calculation using the [EnergyPlus](#) dynamic thermal simulation engine.

Click on the **Heating design** Screen tab to view Heating Design data. If the necessary data has not yet been generated, a Heating Design calculation is automatically started to generate the data.

See also the [Heating Design Calculations](#) tutorial

Heating design simulations using EnergyPlus have the following characteristics:

- Constant (Steady-state) external temperature set to the [winter design external temperature](#).
- [Wind speed and direction](#) set to design values.
- No solar gain.
- No internal gains (lighting, equipment, occupancy etc).
- Heated zones are heated constantly to achieve the heating temperature set point using a simple convective heating system.
- Includes consideration of heat conduction and convection between zones of different temperatures.
- [Schedules are not used for Heating design calculations](#) which are based on a steady state analysis which does not account for timing.

Simulation continues until temperatures/heat flows in each zone have converged. If convergence does not occur then simulation continues for the maximum number of days as specified in the [calculation options](#).

The simulation calculates heating capacities required to maintain the temperature set points in each zone and displays the total heat loss broken down as:

- Glazing
- Walls
- Partitions
- Solid floors
- Roofs
- External infiltration
- Internal natural ventilation (i.e. heat lost to other cooler adjacent spaces through windows, vents, doors, holes)

The total heat loss in each zone is multiplied by a Safety factor (1.5 by default) to give a recommended heating [Design Capacity](#). This heating capacity can be written directly into the **Heating capacity** model data depending on the setting of the [Plant sizing](#) model option on the Data tab under HVAC.

Customise the data displayed and display style using the [Display Options](#) panel.

Note: heating design calculations do not use any Calculated natural ventilation data that may have been set up, however they can be set up to use Scheduled natural ventilation data through the Sizing zone equipment > [Natural ventilation load](#) setting on the HVAC tab.

EnergyPlus IDF data

You can view the EnergyPlus IDF data used to carry out the calculation by clicking on the 'View the input data' link in the info panel (Learning mode only) or from the Tools menu.

Heating Design Calculation Options Dialog

These calculation options can be accessed from the [Heating design](#) calculation options tab of the [Model Options](#) dialog and from the Heating design calculation options dialog.

- [Calculation Options](#)
- "Output Options - Heating Design" on page 1281
- [Advanced Calculation Options](#)

Calculation Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog and General tab on Heating Design dialog.

Calculation Options

Calculation description

Enter some text to identify the calculation. This will be used in reports and in the filename for [automatically generated ESO files](#).

Temperature control

Heating and cooling systems control internal temperatures to meet the setpoint temperatures specified on the [Activity tab](#). These setpoint temperatures can be interpreted as air, operative or some other radiant fraction and DesignBuilder provides corresponding options to allow HVAC systems to be controlled by:

- **1-Air temperature** - control the zone mean air temperatures to the [heating and cooling setpoint temperatures](#) specified on the [Activity tab](#).
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this option.
- **3-Other** - you can enter the radiant temperature control fraction.

When using the **3-Other** option the radiant fraction should be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Radiative Fraction vs Air Velocity

Air Velocity (m/s)	<0.2	0.2 - 0.6	0.6 - 1.0
Radiant fraction	0.5	0.4	0.3

Note: The temperature control settings used in heating design and cooling design will also apply in simulation autosizing calculations for the winter and summer design days respectively.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

You can think of the Temperature control option as:

- The radiant fraction sensed by the room thermostat, or,
- Given a room thermostat that mainly senses air temperature, the extent of automatic modification of the setpoint temperature to ensure occupant comfort.

Note: This option does not affect natural and mechanical [ventilation setpoints](#) - these always use **air temperature** set points.

Tip: See the equivalent topic for Simulation for an [advanced discussion on Operative vs Air temperature control](#).

System Sizing

Design margin

The Heating Design margin is used to multiply calculated steady state heating loads in each zone to give a recommended heating system size. It accounts for the additional heat required to bring the building up to

temperature in a reasonably short preheat period and allows you to be confident that comfort conditions will be maintained in all but the most extreme winter conditions.

The default heating design margin is 1.25 which results in heating systems being oversized by 25% relative to the steady state load calculated by EnergyPlus. This value is based on ASHRAE Standard 90.1. For projects not subject to an energy code or standard that specifies a mandatory safety factor or design margin, the selection of an appropriate value is ultimately at the discretion of the designer.

Output Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog, General tab on [Heating Design Calculation Options](#) dialog and [Miscellaneous](#) model data tab

Heating design output options apply to the whole building and are only visible at building level when accessed from the [Outputs](#) model data tab.

Include unoccupied zones in block and building totals and averages

By default data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Store surface output

Causes the following output to be stored for all walls, floors, roofs, ceilings, partitions etc:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Loss
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Store opening output

Only visible when the [Store surface output](#) option is selected, this option causes the following data to be stored for windows vents, doors, holes, sub-surfaces:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Loss
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Warning: Use the surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be stored.

Advanced Calculation Options - Heating Design

[Heating Design](#) tab on [Model Options](#) dialog and General tab on the [Heating Design Calculation](#) options dialog.

General Solution

Temperature and loads convergence

The temperature and loads convergence values represent the maximum allowable difference in zone temperature and loads between successive daily iterations before convergence is considered to have been reached during warmup.

Convergence of the simultaneous heat balance/HVAC solution is reached when both the loads and temperature criteria are satisfied. Both tolerances work the same way, one looks at temperatures and one looks at heating and cooling loads. After the second warm-up day, the program compares the maximum temperature experienced in a space with the maximum temperature from the previous day. If those two temperatures are within the tolerance, then it has passed the first warm-up check. It does a similar comparison with lowest temperatures experience within all the zones. If the current simulation day and the previous day values are within the tolerance, then it has passed the second warm-up check. A similar comparison is carried out with the loads tolerance and the maximum heating and cooling loads that are experienced within the spaces. Those are compared individually to the values for the previous day. If they are both in tolerance, then the simulation has passed the third and fourth warm-up check. The simulation stays in the warm-up period until ALL FOUR checks have been passed.

Inside and Outside convection algorithms

You can select from a range of EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. More details on this and external convection can be found under [Constructions](#) Model Data > [Surface Convection](#) header.

Other

Surfaces within zone treated as adiabatic

EnergyPlus support recommend modelling surfaces wholly contained within a zone as adiabatic and this option allows you to follow this advice. We have found that in practice this option does not make much difference in results or in simulation speed so for most cases you can leave it in its default state.

See also [EnergyPlus Errors and Warnings](#)

Note: 'surfaces contained within a zone' are frequently generated when one of the [zone merging](#) options is used where a partition or floor which would have separated two zones actually sits within the merged zone. These surfaces *do not* refer to [hanging partitions](#) which are modelled using [Internal thermal mass](#).

Heat Loss Breakdown Results

The 'Steady-State' tab has detailed [heating design](#) results (displayed as a histogram and table by default).

For more on how the output is calculated see: [Calculation of DesignBuilder Output from EnergyPlus Report Variables](#).

- **Air Temperature** - the calculated average temperature of the air.
- **Radiant Temperature** - the (area * emissivity) weighted average of the zone inside surface temperatures.
- **Operative Temperature** - The mean of the internal air and radiant temperatures.
- **Outside Dry-Bulb Temperature**.
- **Glazing** - heat loss through all glazing.
- **Walls** - heat loss through all external walls plus all heat loss through [linear thermal bridges](#).
- **Ceilings (int)** - heat loss through internal ceilings (e.g. zone above is colder).
- **Floors (int)** - heat loss through internal floors (e.g. zone below is colder).
- **Floors (ext)** - heat loss through external floors (not ground floor, e.g. floor in cantilevered space, roof eaves etc).
- **Ground Floors** - heat loss through ground floors.
- **Doors and Vents** - conduction heat loss through doors and vents.
- **Partitions (int)** - heat loss through all internal partitions (e.g. adjacent zone is colder).
- **External Infiltration** - heat loss through air infiltration (non-unintentional air entry through cracks and holes in building fabric).
- **External Mechanical Ventilation** - is the heat loss due to the entry of outside air through the air distribution system. View can exclude mechanical ventilation from the heating design calculations by unchecking the 'Mechanical ventilation On' Model Data.
- **Internal Natural Ventilation** - heat loss due to air exchange through open internal windows, doors, vents, holes and virtual partitions to adjacent spaces.
- **Zone Sensible Heating** - heat delivered to maintain the internal heating design temperature.

Note: The **Heating delivered** is the sum of the component heat loss values.

Surface heat loss data (walls, ceilings, floors etc) refers to heat transfer from the zone to the inside surface of the building elements.

You can also customise the data displayed and display style using the [Display Options](#) panel.

Reviewing Heating Design Output

You can review heating design output in 2 ways:

- [Heat loss breakdown](#) - on the Steady State tab.
- [Summary table](#) - heating capacity data for each zone on the Summary tab.

To update the data using different calculation options, click Update (Ctrl-U).

The General tab on the Display Options panel provides options for controlling the data displayed in the Heating design data display screen. General Display Options are grouped under three headers:

- **Data** - control the data displayed.
- **Y-axis** - for customising the layout of the Y-axis.
- **Appearance** - for customising the style and appearance of the graphics.

The [Detailed](#) tab on the Display Options panel provides controls which allow you to select exactly the data items you wish to display.

Data Options

Data

Select an option to indicate the data you wish to display:

- **1-All** - fabric and ventilation heat gains/losses, internal gains (not Heating Design), temperatures and outside dry-bulb air temperature.
- **2-Site data** - all site data.
- **3-Comfort** - inside air, the radiant and comfort temperatures.
- **4-Internal gains** - internal gains including equipment, lighting, occupancy, solar and HVAC heating/cooling delivery.
- **5-Fabric and ventilation** - heat gains to the space from the surface element (walls, floors, ceilings etc.) and ventilation. Negative values indicate heat loss from the space.

Tip: to update the data using different calculation options, click Update (Ctrl-U).

Show as

You can display the data in one of four ways:

- **1-Graph** - displays the data on the screen in graphical format. The default option.
- **2-Grid** - you should select this option to [export](#) data in csv spreadsheet format. This [export format](#) can be loaded to Results Viewer for further analysis and comparison.
- **3-Graph and table** - the same as **1-Graph** but also includes a basic tabular display below the graph.
- **4-Table** - the same basic table format displayed for the **3-Graph and table** option, but without the graph.

Normalise by area

Check this option if you wish building, block and zone data to be displayed 'per floor area', i.e. /m² or /ft², and surface and opening data to be displayed 'per surface or opening area'.

At building, block or zone level you can select the floor area to use as the denominator from the options:

- **1-All areas** - Total area is used,
- **2-Occupied area** - just occupied areas are used.

Note: you can only select **1-All areas** if the building and block data *includes* occupied areas as set in [Include unoccupied zones in block and building totals and averages](#) option in Heating design [Output options](#). Likewise you can only select **2-Occupied area** if the building and block data *excludes* occupied areas.

Y-Axis Options

These options give you control over how the Y-axis is displayed.

Lock min/max

This option locks the extent of the Y-axes so that they keep their current maximum and minimum values. This allows consistent comparison with the future results display.

The Y-axis keeps its current settings until you unlock it again.

You can achieve the same effect by clicking on the 'Lock Y-axis min/max values' toolbar command



When you lock the Y-axis the current settings are stored and displayed for each of the available Y-axes. You can edit any of these to get a more precise setting. Remember to click on the Apply button when you are ready to register a change to any numeric display option data.

Overlap multiple axes

Option allows you to save space on the Y-axis allowing different axes to overlap.

Separate axes

Check this option to add a vertical space between each of the displayed axes.

Normalise by floor area

Check this option if you wish data to be displayed 'per floor area' i.e. /m2 or /ft2.

Appearance

You can control the details of the presentation of the graphical output by making selections on the Display Options panel under the [Appearance](#) header.

EnergyPlus IDF Input Data

When the [Store IDF files](#) Program option is selected, you can view the IDF data used to calculate Heating design results by selecting the **Tools | Display simulation input script** menu option or the **View the input data** Info panel link (in [Learning mode](#)).

If you do not have the EnergyPlus IDF editor installed, then you should associate the [.idf file extension](#) with a text editor to make this option work.

Note: this shows a snapshot of IDF data just before the simulation *not* the [_Ref-1140576427in.idf](#) file currently stored in the [EnergyPlus folder](#) which is generally from the most recent EnergyPlus simulation.

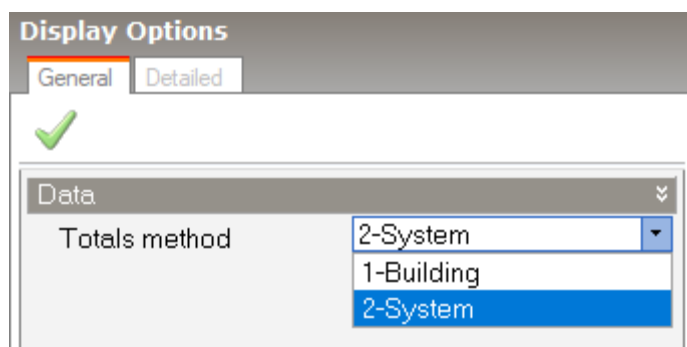
Heating Design Summary Table

Summary Display Options

Totals method

You can sum total heating loads displayed on the Summary tab in 2 different ways:

- **1-Building** - loads are summed for the whole building.
- **2-System** - loads are summed for each [System](#) type allowing you to calculate system (AHU)/block loads for sizing AHUs.



Summary Results

The Summary tab shows a range of Design heating loads, ventilation and fabric gains for each zone and other related data. Outputs for each zone are summed based on the **Totals method** Display option (see above):

- **Zone multiplier** is the [Zone multiplier](#) factor entered on the Activity tab. It is used to define the number of zones represented by this zone in the model. By default, it has a value of 1.
- **Comfort temperature** - The mean of the internal air and radiant temperatures.
- **Steady-state heat loss** (kW or kBtu/h) is the Zone Sensible Heating calculated by EnergyPlus, the heat delivered to maintain the internal heating design temperature for the outdoor design conditions.
- **Design capacity** (kW or kBtu/h) is the Steady-state heat loss (see above) multiplied by the [Design margin](#) to give the design heating capacity of the equipment.
- **Design capacity** (kW or kBtu/h) is the Steady-state heat loss (see above) normalised by floor area and multiplied by the [Design margin](#) to give the design heating capacity per floor area of the equipment.
- **Glazing Gains** (kW or kBtu/h) are the gains through windows (excluding solar gains).
- **Wall Gains** (kW or kBtu/h) are the gains through all external and internal wall (partition) surfaces.
- **Floor Gains** (kW or kBtu/h) are the gains through all floor surfaces.
- **Roof and Ceiling Gains** (kW or kBtu/h) are the gains through all roof + ceiling surfaces
- **Ventilation Gains** (kW or kBtu/h) are the mechanical + natural ventilation gains.
- **Infiltration Gains** (kW or kBtu/h) are the gains due to infiltration of outside air into the building through cracks and fabric porosity.

Tip: Use the [Export](#) command to export this data in spreadsheet format.

Cooling Design Calculation

Cooling design calculations are carried out to determine the capacity of mechanical cooling equipment required to meet the hottest [summer design weather](#) conditions likely to be encountered at the site location. In zones which are not mechanically cooled free-floating temperatures are calculated including the effects of [natural](#) or [mechanical ventilation](#) if these options are selected on the zone [HVAC tab](#). These design calculations are traditionally carried out using periodic steady-state methods such as the admittance and response factor methods provided by CIBSE and ASHRAE. You can use DesignBuilder to carry out effectively the same calculation using the [EnergyPlus](#) dynamic thermal simulation engine.

Click on the **Cooling design** Screen tab to view Cooling Design data. If the necessary data has not yet been generated, a Cooling Design calculation is automatically started to generate the data.

See also the [Cooling Design Calculations](#) tutorial

Cooling design simulations using EnergyPlus have the following characteristics:

- Periodic steady-state external temperatures calculated using maximum and minimum [design summer weather](#) conditions.
- No wind.
- Includes solar gains through windows and [scheduled](#) natural ventilation.
- includes internal gains from occupants, lighting and other equipment.
- Includes consideration of heat conduction and convection between zones of different temperatures.

For buildings situated in the Northern Hemisphere, Cooling Design calculations are made for the month of July and for buildings in the Southern Hemisphere they are made for the month of January.

Simulation continues until temperatures/heat flows in each zone have converged. If convergence does not occur then simulation continues for the maximum number of days as specified in the calculation options.

The simulation calculates half-hourly [temperatures and heat flows](#) for each zone and determines [cooling capacities](#) required to maintain any cooling temperature set points in each zone.

The maximum cooling load in each zone is multiplied by a [Safety factor](#) (1.3 by default) to give a Design Cooling Capacity. This [design cooling capacity](#) can be written directly into the **Cooling capacity** model data depending on the setting of the [Plant sizing](#) model option on the Data tab under HVAC.

Customise the data displayed and display style using the [Display Options](#) panel.

Selecting Summer Design Weather Data and Solar Gains

DesignBuilder offers two approaches to determine summer design peak cooling conditions:

1. Multiple Design Months (Default)

- Automatically tests a range of monthly temperatures and solar radiation profiles.
- Identifies the peak cooling month without manual intervention.

2. Single Design Month

- Uses one user-defined design month for temperatures (Outside Air, Mean Coincident Wet-Bulb, and Dry Bulb Temperature) and solar gains data.

Note: In spaces with large windows, peak cooling loads may occur in months other than the hottest due to solar gains

Where to Adjust Settings:

- Define whether to use single or multiple months and enter summer design weather data at site level under the [Summer Design Weather Data](#) header on the Location tab.
- Select the summer design day(s) on the Cooling Design Calculation Options dialog under the [Summer Design Day](#) header

Note: Cooling Design calculations use Scheduled Natural ventilation model data set on the HVAC tab even when the [Calculated](#) natural ventilation option is set. This is to help you to document the input data for your Cooling Design calculations. Calculated natural ventilation data input is too complex to be adequately summarised.

Cooling Design Calculation Options Dialog

These calculation options can also be accessed from the [Cooling design calculation options](#) tab of the [Model Options](#) dialog.

- "Summer Design Day" on page 1285
- "Cooling system sizing" on page 1286
- "Solar Options - Cooling Design" on page 1288
- "Output Options - Cooling Design" on page 1288
- "Advanced Calculation Options - Cooling Design" on page 1289

Calculation description

Enter some text to identify the calculation. This will be used in reports and in the filename for [automatically generated ESO files](#).

Click on the OK button to start the Cooling design calculation.

Calculation Options - Cooling Design

[Ref641103961](#) [Cooling Design](#) tab on [Model Options](#) dialog and [General](#) tab on [Cooling Design Calculation Options](#) dialog.

Calculation Description

Enter some text to identify the calculation. This will be used in reports and in the filename for [automatically generated ESO files](#). The Calculation description is only available on the Cooling design calculation options dialog.

Calculation Options

Temperature control

Heating and cooling systems control internal temperatures to meet the setpoint temperatures specified on the [Activity tab](#). These setpoint temperatures can be interpreted as air, operative or some other radiant fraction and DesignBuilder provides corresponding options to allow HVAC systems to be controlled by:

- **1-Air temperature** - control the zone mean air temperatures to the [heating and cooling setpoint temperatures](#) specified on the [Activity tab](#).
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this option.
- **3-Other** - you can enter the radiant temperature control fraction.

When using the **3-Other** option the radiant fraction should be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Radiative Fraction vs Air Velocity

Air Velocity (m/s)	<0.2	0.2 - 0.6	0.6 - 1.0
Radiant fraction	0.5	0.4	0.3

Note: The temperature control settings used in heating design and cooling design will also apply in simulation autosizing calculations for the winter and summer design days respectively.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

You can think of the Temperature control option as:

- The radiant fraction sensed by the room thermostat, or,
- Given a room thermostat that mainly senses air temperature, the extent of automatic modification of the setpoint temperature to ensure occupant comfort.

Note: This option does not affect natural and mechanical [ventilation setpoints](#) - these always use **air temperature** set points.

Tip: See the equivalent topic for Simulation for an [advanced discussion on Operative vs Air temperature control](#).

Exclude all zone natural ventilation (infiltration is always included)

This option should be selected when natural ventilation is to be excluded from the cooling calculations in all zones. Infiltration is always included regardless of this setting.

Exclude all zone mechanical ventilation

Select this option if you wish to exclude the zone mechanical ventilation load from the cooling loads in all zones. The option is useful in cases where the AHU is to meet mechanical ventilation loads and these loads are to be calculated separately when sizing the AHU coil capacity.

Exclude heat recovery

Check this option if you would like to exclude the savings introduced by heat recovery in any of the zones in the model that have heat recovery selected on the HVAC tab.

Summer Design Day

[Cooling Design](#) tab on [Model Options](#) dialog.

Day

Select the day of the month for the design day. The default day is 15, i.e. the middle of the month.

This data is only used to define the solar path over the design day and does not affect [temperatures](#) etc which are defined at site level.

Note: This same day of the month will be used for all months selected below when using the **2-Multiple design months** Design temperature period.

Month (or Start month and End Month)

Depending on the selection at site level for [Design temperature period](#), you can either select a single design month (when it is set to **1-Single design month**) or a month range (when it is set to **2-Multiple design months**).

When a single month is being used, the default design month will be 1 month after the month of mid-summer by default (July in the Northern Hemisphere, January in the Southern Hemisphere). When multiple months are being used, the default start month is the mid-summer month and the end month is 2 months later. This covers most design cases where peak conditions may occur significantly later after mid summer, especially when there are large amounts of South-facing glazing (in the Northern Hemisphere).

Important Note: If you are using the default **1-Single design month** [Design temperature period](#) option the design temperature and humidity values used in the calculations will be the “Yearly” design temperatures set at site level under Summer Design Weather Data. This is true regardless of the month and day of month selected on the Cooling design calculation options dialog. You will therefore find that the outdoor air temperatures won’t change when changing the month with this option set. You must use the **2-Multiple design months** option to see seasonal variations of design outside air conditions.

Tip: Use the **2-Multiple design months** [Design temperature period](#) option for a thorough analysis. DesignBuilder will sequentially test various monthly temperature and solar radiation profiles to automatically identify the peak cooling month on your project.

Day of week

This selection is used to identify the appropriate daily profile within the schedules to use for the Cooling design calculations. The day type should be the day when the most extreme conditions and highest cooling loads are expected. You should normally keep the **9-SummerDesignDay** default option, especially when also running simulations. Select the day of week from the options:

- **1-Sunday** - not recommended if the cooling system does not operate on Sundays.
- **2-Monday**,
- **3-Tuesday**,
- **4-Wednesday**,
- **5-Thursday**,
- **6-Friday**,
- **7-Saturday** - not recommended if the cooling system does not operate on Saturdays.
- **8-Holiday** - not recommended if the cooling system does not operate on holidays.
- **9-SummerDesignDay** - the default recommended setting.
- **10-WinterDesignDay** - not recommended.
- **11-CustomDay1**,
- **12-CustomDay2**

Cooling System Sizing

[Cooling Design](#) tab on [Model Options](#) dialog.

Design margin

The Cooling Design margin is a 'safety factor' used to multiply calculated cooling loads in each zone to give a recommended maximum cooling equipment capacity. It accounts for the additional cooling which might be required to cool the building down in a reasonably short pre-cool period and allows you to be confident that comfort conditions will be maintained in all but the most extreme summer conditions. The risk depends on the [Summer Design Weather Data](#) options selected at site level.

The default design margin is 1.15 which means that the cooling system will be oversized by 15%. This value is based on ASHRAE Standard 90.1. For projects not subject to an energy code or standard that specifies a mandatory safety factor or design margin, the selection of an appropriate value is ultimately at the discretion of the designer.

Sizing method

Sizing methods available are:

- **1-ASHRAE** - standard 'Heat Balance' method implemented in EnergyPlus.
- **2-Unitary DX** - calculations are made through a detailed EnergyPlus simulation using Unitary DX equipment.

Airflow calculation method

When calculating the design airflow rate from the cooling load DesignBuilder can use one of 2 methods:

- **1-Sensible only** the default option where the airflow rate is simply calculated from the cooling load, the supply air temperature and the zone air temperature setpoint. This method is widely used in the industry to calculate the airflow rate in cases where the latent load is a relatively low proportion of the total.

$$\text{Design flow rate} = \text{SizingFactor} \cdot \text{DesignCoolingLoad} / (T_{\text{zoneairdb}} - T_{\text{supplyair}}) \cdot \text{CP}_{\text{air}} \cdot \text{Density}_{\text{air}}$$

where:

CP_{air} is the specific heat capacity of air

$T_{\text{zoneairdb}}$ is the zone dry bulb temperature

$T_{\text{supplyair}}$ is the supply air dry-bulb temperature

- **2-Sensible + latent** where a similar enthalpy calculation is used. This method may be adopted in some cases where latent loads predominate:

$$\text{Design flow rate} = \text{SizingFactor} \cdot \text{DesignCoolingLoad} / (E_{\text{zoneair}} - E_{\text{supplyair}}) \cdot \text{Density}_{\text{air}}$$

where:

E_{zoneair} is the zone air enthalpy

$E_{\text{supplyair}}$ is the supply air enthalpy

ASHRAE Cooling Sizing

In the ASHRAE cooling sizing method the cooling design load calculations are carried out using the standard ASHRAE Heat Balance method implemented in EnergyPlus using the 'ZoneHVAC:IdealLoadsAirSystem' system type. You can define the conditions of the supply air delivered to cooled zones using Design supply temperature and Supply air humidity ratio. The total (sensible + latent) load is calculated as the enthalpy change between the zone return air and the specified supply air condition for the current load.

The ASHRAE system is modelled assuming:

- No economiser,
- No heat recovery,
- No heating,
- Zero pressure rise fan,
- No humidification/dehumidification,
- Zone plenums will not be used if specified,
- Outside airflow rate and operation schedule defined by the data under the [Mechanical Ventilation](#) and [Natural ventilation](#) headers on the [HVAC](#) tab at the zone level,
- Cooling coil availability defined by the schedule under the [Cooling](#) header on the [HVAC](#) tab at the zone level,
- Zone design cooling temperature setpoints are defined by the data under [Environmental Control](#) on the [Activity](#) tab.

Supply Air Conditions

In DesignBuilder v3.1 and later the supply air conditions used in Cooling design come from the [HVAC tab](#) of model data.

Unitary DX Cooling Sizing

When using the Unitary DX cooling sizing method the Cooling design calculation is made through a detailed EnergyPlus simulation using the EnergyPlus HVAC Template Single zone Unitary DX equipment.

Each zone has its own Unitary system with a fan (generating no pickup), a DX cooling coil, an outside air controller and a damper to reduce supply air flow. More specifically:

- No economiser,
- No heat recovery,

- No heating coil,
- Zero pressure rise fan,
- No humidification/dehumidification,
- Zone plenums will be used if specified,
- Outside airflow rate and operation schedule defined by the data under the [Mechanical Ventilation](#) and [Natural ventilation](#) headers on the [HVAC](#) tab at the zone level,
- Cooling coil availability defined by the schedule under the [Cooling](#) header on the [HVAC](#) tab at the zone level,
- Zone design cooling temperature setpoints are defined by the data under [Environmental Control](#) on the [Activity](#) tab.

Note: Unitary DX will tend to calculate higher latent loads than the other 3 cooling sizing methods and the zone air humidity will generally be lower. This is because DX coils run colder than water-cooled coils and will condense out more water from the supply air. Unitary DX can be used to provide a conservative calculation for the latent cooling loads.

Solar Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog and General tab on [Cooling Design Calculation Options](#) dialog.

Solar Options allow you to control aspects of the model related to solar gains.

Include all buildings in shading calcs

The standard wall, floor, roof surfaces defining the current building provide shading but by default the other buildings on the site are excluded from the analysis. Use this option to include other buildings on the site as shading obstructions.

Note: there is no way to simulate more than 1 building at a time. All external surfaces in buildings other than the current one are modelled as shading surfaces. If you need to model the adjacency of another building in contact with the current one, you can use the [adiabatic](#) adjacency option to model the touching surfaces.

Note: if one or more 'other buildings' are large/complex then you could generate a large number of shading elements and the [simulation could be slow](#) (even if the current building being simulated is simple).

Include shading from excluded zones in simulation

When zones are excluded from simulations, by unchecking the "Include Zone " on page 330 checkbox, it is often still necessary for the excluded the zone's surfaces to be used to shade other parts of the building. Checking this checkbox ensures that all building surfaces are used in shading calculations, even those that have been excluded.

Model all external reflections and shading of ground reflected solar

See [help](#) for equivalent topic for simulation.

Solar distribution

See [help](#) for equivalent topic for simulation.

Output Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog, General tab on [Cooling Design Calculation Options](#) dialog and [Miscellaneous](#) model data tab

Cooling design output options apply to the whole building and are only visible at building level when accessed from the [Outputs](#) model data tab.

Include unoccupied zones in block and building averages and totals

By default, data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Store surface output

Causes the following output to be stored for all walls, floors, roofs, ceilings, partitions etc:

- Inside Surface Temperature
- Outside Surface Temperature
- Surface Heat Gain
- Solar Incident
- External Sunlit Fraction
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Store opening output

Only visible when the [Store surface output](#) option is selected, this option causes the following data to be stored for windows vents, doors, holes, sub-surfaces:

- Inside Surface Temperature
- Outside Surface Temperature
- Glazing Heat Gain
- Solar Incident
- Solar Transmitted
- External Sunlit Fraction
- Internal Convective Heat Transfer Coefficient
- External Convective Heat Transfer Coefficient

Warning: Use the surface and opening output options with caution on large models as, when selected, they will cause large amounts of data to be stored.

Advanced Calculation Options - Cooling Design

[Cooling Design](#) tab on [Model Options](#) dialog and General tab on the [Cooling Design Calculation](#) options dialog.

General Solution

Inside face surface temperature convergence criteria

The surface heat balance model at the inside face has a numerical solver that uses a convergence parameter for a maximum allowable differences in surface temperature. This field can optionally be used to modify this convergence criteria. The default value is 0.002 and was selected for stability. Lower values may further increase stability at the expense of longer runtimes, while higher values may decrease runtimes but lead to possible instabilities. The units are °C or °F.

Temperature and loads convergence

The temperature and loads convergence values represent the maximum allowable difference in zone temperature and loads between successive daily iterations before convergence is considered to have been reached during warmup.

Convergence of the simultaneous heat balance/HVAC solution is reached when both the loads and temperature criteria are satisfied. Both tolerances work the same way, one looks at temperatures and one looks at heating and cooling loads. After the second warm-up day, the program compares the maximum temperature experienced in a space with the maximum temperature from the previous day. If those two temperatures are within the tolerance, then it has passed the first warm-up check. It does a similar comparison with lowest temperatures experience within all the zones. If the current simulation day and the previous day values are within the tolerance, then it has passed the second warm-up check. A similar comparison is carried out with the loads tolerance and the maximum heating and cooling loads that are experienced within the spaces. Those are compared individually to the values for the previous day. If they are both in tolerance, then the simulation has passed the third and fourth warm-up check. The simulation stays in the warm-up period until ALL FOUR checks have been passed.

Convection

Inside and Outside convection algorithms

You can select from a range of EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. More details on this and external convection can be found under [Constructions](#) Model Data > [Surface Convection](#) header.

Shading

Maximum number of shadow overlaps

Enter the maximum number of figures per shadow overlap. The shadow overlaps is a measure of the amount of complexity in the shading calculation and this maximum value allows you to limit the amount of time spent in the solar initialisation calculations.

Note: entering a small value here can speed up simulations in complex buildings. If you plan to use [Maximum shadow overlaps](#) below the default value of 15000 you should check accuracy of solar gains relative to results using default value

Polygon clipping algorithm

This is an advanced feature. Prior to V7, the internal polygon clipping method was a special case of the Weiler-Atherton method. Now, two options are available:

- **1-Sutherland Hodgman** (default)
- **2-Convex Weiler Atherton**

Theoretically, Sutherland-Hodgman is a simpler algorithm but it works well in cases where receiving surfaces (of shadows) are non-convex. The Weiler-Atherton implementation is only accurate where both casting and receiving surfaces are convex. Warnings/severe errors are displayed when necessary. More details on polygon clipping are contained in the [Engineering Reference](#).

Other

Surfaces within zone treated as adiabatic

EnergyPlus support recommend modelling surfaces wholly contained within a zone as adiabatic and this option allows you to follow this advice. We have found that in practice this option does not make much difference in results or in simulation speed so for most cases you can leave it in its default state.

See also [EnergyPlus Errors and Warnings](#)

Note: 'surfaces contained within a zone' are frequently generated when one of the [zone merging](#) options is used where a partition or floor which would have separated two zones actually sits within the merged zone. These surfaces *do not* refer to [hanging partitions](#) which are modelled using [Internal thermal mass](#).

Cooling Design Detailed Results

The 'Analysis' tab has detailed half-hourly [cooling design](#) results (displayed as a line graph by default).

For more on how the output is calculated see: [Calculation of DesignBuilder Output from EnergyPlus Report Variables](#).

- **Air Temperature** - the calculated average temperature of the air.
- **Radiant Temperature** - the (area * emissivity) weighted average of the zone inside surface temperatures.
- **Operative Temperature** - The mean of the mean internal air and mean radiant temperatures.
- **Outside Dry-Bulb Temperature.**
- **Glazing** - the total heat flow to the zone from the glazing, frame and divider of exterior glazing excluding transmitted short-wave solar radiation (which is accounted for in **Solar Gains Exterior Windows** below).

For windows **without an interior shading device** this heat flow is equal to:

+ [Convective heat flow to the zone from the zone side of the glazing]
+ [Net IR heat flow to the zone from zone side of the glazing]
– [Short-wave radiation from zone transmitted back out the window]
+ [Conduction to zone from window frame and divider, if present]

Here, short-wave radiation is that from lights and diffuse interior solar radiation.

For windows **with an interior shading device** this heat flow is equal to:

[Convective heat flow to the zone from the air flowing through the gap between glazing and shading device]
+ [Convective heat flow to the zone from the zone side of the shading device]

- + [Net IR heat flow to the zone from the zone side of the glazing]
 - + [Net IR heat flow to the zone from the zone side of the shading device]
 - [Short-wave radiation from zone transmitted back out the window]
 - + [Conduction to zone from window frame and divider, if present]
- **Walls** - heat gain due to conduction through all external walls, including the effect of solar radiation and longwave radiation to the sky. This output also includes all heat gain through [linear thermal bridges](#).
 - **Roofs** - heat gain due to conduction through all external roofs, including the effect of solar radiation and longwave radiation to the sky.
 - **Ceilings (int)** - heat conduction gain through internal ceilings (e.g. zone above is colder).
 - **Floors (int)** - heat conduction gain through internal floors (e.g. zone below is colder).
 - **Floors (ext)** - heat conduction gain through external floors (not ground floor, e.g. floor in cantilevered space, roof eaves etc).
 - **Ground floors** - heat conduction gain through ground floors.
 - **Partitions (int)** - heat conduction gain due to heat conduction through all internal partitions from adjacent zones at different temperatures.
 - **Doors and vents** - conduction heat gain through doors and vents.
 - **External Infiltration** - heat gain through air infiltration (non-unintentional air entry through cracks and holes in building fabric).
 - **External Vent** - heat gain due to the entry of outside air through natural and mechanical ventilation (as defined on the HVAC tab). You can exclude mechanical ventilation from the cooling design calculations by unchecking the 'Mechanical ventilation On' Model data.
 - **Internal Natural Ventilation** - heat gain from other zones due to air exchange through open internal windows, doors, vents, holes and virtual partitions.
 - **Task Lighting** - heat gain due to task lighting.
 - **General Lighting** - heat gain due to general lighting.
 - **Miscellaneous** - heat gain due to miscellaneous equipment.
 - **Process** - heat gain due to process equipment.
 - **Catering** - heat gain due to cooking.
 - **Computer and Equipment** - heat gain due to computer and other IT-related equipment.
 - **Occupancy** - sensible gain due to occupants. Please note that this can vary depending on the internal conditions. With very high temperatures the sensible gain can drop to zero with all cooling effects taking place through latent heat transfer.
 - **Solar Gains Exterior Windows** - Short-wave solar radiation transmission through *external* windows. For a bare window, this transmitted radiation consists of solar radiation passing through the glass and diffuse radiation from solar reflected from the outside window reveal, if present. For windows with a shade, this transmitted radiation is totally diffuse (shades are assumed to be perfect diffusers). For windows with a blind, this transmitted radiation consists of beam + diffuse radiation that passes between the slats and diffuse radiation from beam-to-diffuse reflection from the slats. *Any solar radiation re-reflected back out of the external window or transmitted through interior windows is not subtracted.*
Solar Gains Interior Windows - Total beam + diffuse solar radiation transmission through *interior* windows. Requires the [3-Full interior and exterior](#) solar model option to be set.
 - **Zone Sensible Cooling** - is the sensible cooling effect *on the zone* of any air introduced into the zone through the HVAC system. It includes any 'free cooling' due to introduction of relatively cool outside air. Cooling always shows as a negative heat gain in the results. *It is best thought of as part of the zone heat balance.*

Airflow

- **Mech Vent + Nat Vent + Infiltration** - The sum of **outside air** (in ac/h) flowing into the zone through:
 - The HVAC air distribution system +
 - Infiltration +
 - Natural ventilation

System Heat Flows

These heat flows are plotted on the System Energy axes:

- **Zone Heating** - energy supplied by local room heaters and reheat coils to maintain room internal heating temperature setpoint temperature when using Detailed HVAC data.

- **Total Cooling** - depending on the [Sizing method](#):
 - for the default [ASHRAE sizing method](#) **Total Cooling** is the rate at which total energy (sensible and latent) is removed from the mixed outside and return air stream in order to bring the mixed air stream to the [specified temperature and humidity ratio](#) of the supply air stream.
 - for the **2-Unitary DX Sizing method** - at *building level*: sensible + latent cooling transfer to the supply air from the AHU cooling coil + any single zone unitary and fan coil units in the building. At *zone level*: sensible + latent cooling transfer to the supply air from a single zone unitary or fan coil unit.
- **Sensible Cooling** - depending on the [Sizing method](#):
 - for the default [ASHRAE sizing method](#) **Sensible Cooling** is the rate at which sensible energy is removed from the mixed outside and return air stream in order to lower its temperature to the [specified temperature](#) of the supply air stream. Any energy needed for moisture addition or removal is ignored.
 - for the **2-Unitary DX Sizing method** **Sensible Cooling** is sensible only cooling heat transfer from the cooling coil to the supply air. Latent coil heat can be calculated as the difference: **Total Cooling - Sensible Cooling**.

Surface heat gain data refers to heat transfer from the inside surface of the building elements to the zone.

You can also customise the data and the way it is displayed using the [Display Options](#) panel.

Reviewing Cooling Design Output

You can review Cooling design output in 2 ways:

- [Detailed results](#) - half-hourly heat balance and inside conditions displayed on **Analysis** tab.
- [Summary table](#) - cooling capacity data for each zone on the **Summary** tab.

To update the data using different calculation options, click Update (Ctrl-U).

The General tab on the Cooling design Display options panel provides options for controlling the data displayed in the Cooling design data display screen. General Display options are grouped under three headers:

- Data - control the data displayed.
- Y-axis - for customising the layout of the Y-axis.
- Appearance - for customising the style and appearance of the graphics.

The [Detailed](#) tab on the Display Options panel provides controls which allow you to select exactly the data items you wish to display.

Data Options

Data

Select an option to indicate the data you wish to display:

- **1-All** - fabric and ventilation heat gains/losses, internal gains (not Heating Design), temperatures and outside dry-bulb air temperature.
- **2-Site data** - all site data.
- **3-Comfort** - inside air, the radiant and comfort temperatures, relative humidity, ASHRAE 55 and various comfort indices.
- **4-Internal gains** - internal gains including equipment, lighting, occupancy, solar and HVAC heating/cooling delivery.
- **5-Fabric and ventilation** - heat gains to the space from the surface element (walls, floors, ceilings etc.) and ventilation. Negative values indicate heat loss from the space.

Tip: to update the data using different calculation options, click Update (Ctrl-U).

Show as

You can display the data in one of four ways:

- **1-Graph** - displays the data on the screen in graphical format. The default option.
- **2-Grid** - you should select this option to [export](#) data in csv spreadsheet format. This [export format](#) can be loaded to Results Viewer for further analysis and comparison.
- **3-Graph and table** - the same as **1-Graph** but also includes a basic tabular display below the graph.
- **4-Table** - the same basic table format displayed for the **3-Graph and table** option, but without the graph.

Show month

If you ran cooling design calculations for more than 1 month then you can select the month to view detailed results from the droplist. The first item on the droplist of available months is **0-Peak** which, when selected, is used by DesignBuilder to automatically select the month of peak cooling load. In this case a further **Peak definition** option is displayed allowing you to select how the peak month is defined. See below.

Peak definition

When the **0-Peak** month has been selected in the **Show month** droplist (above) and you are at zone level or below, this additional selection is displayed to allow you to define whether "peak month" refers to the peak for the current zone or for the whole building. Select from:

- **1-Building peak** where data for the month of the building peak sensible cooling load is displayed.
- **2-Zone peak** where data for the month of the currently selected zone peak sensible cooling load is displayed.

Normalise by floor area

Check this option if you wish building, block and zone data to be displayed 'per floor area', i.e. /m² or /ft², and surface and opening data to be displayed 'per surface or opening area'.

At building, block or zone level you can select the floor area to use as the denominator from the options:

- **1-All areas** - Total area is used,
- **2-Occupied area** - just occupied areas are used.

Note: you can only select **1-All areas** if the building and block data *includes* occupied areas as set in [Include unoccupied zones in block and building totals and averages](#) option in Cooling design [Output options](#). Likewise you can only select **2-Occupied area** if the building and block data *excludes* occupied areas.

Y-Axis Options

These options give you control over how the Y-axis is displayed.

Lock min/max

This option locks the extent of the Y-axes so that they keep their current maximum and minimum values. This allows consistent comparison with the future results display. The Y-axis keeps its current settings until you unlock it again.

You can achieve the same effect by clicking on the 'Lock Y-axis min/max values' toolbar command.



When you lock the Y-axis the current settings are stored and displayed for each of the available Y-axes. You can edit any of these to get a more precise setting. Remember to click on the Apply button when you are ready to register a change to any numeric display option data.

Overlap multiple axes

Option allows you to save space on the Y-axis allowing different axes to overlap.

Separate axes

Check this option to add a vertical space between each of the displayed axes.

Normalise by floor area

Check this option if you wish data to be displayed 'per floor area' i.e. /m² or /ft².

Appearance

You can control the details of the presentation of the graphical output by making selections on the Display Options panel under the [Appearance](#) header.

EnergyPlus IDF Input Data

When the [Store IDF files](#) Program option is selected, you can view the IDF data used to calculate Cooling design results by selecting the **Tools | Display simulation input script** menu option or the **View the input data** Info panel link (in [Learning mode](#)).

If you do not have the EnergyPlus IDF editor installed, then you should associate the [.idf file extension](#) with a text editor to make this option work.

Note: this shows a snapshot of IDF data just before the simulation *not* the [_Ref-1140576427in.idf](#) file currently stored in the [EnergyPlus folder](#) which is generally from the most recent EnergyPlus simulation.

Summary Cooling Design Table

The Summary tab on the Cooling design screen shows a summary of maximum loads with some display options.

Summary Display Options

Totals method

You can sum total loads in 2 different ways

- **1-Building** - loads are summed for the whole building.
- **2-System** - loads are summed for each [System](#) type allowing you to calculate system (AHU)/block loads for sizing AHUs.

Peak method

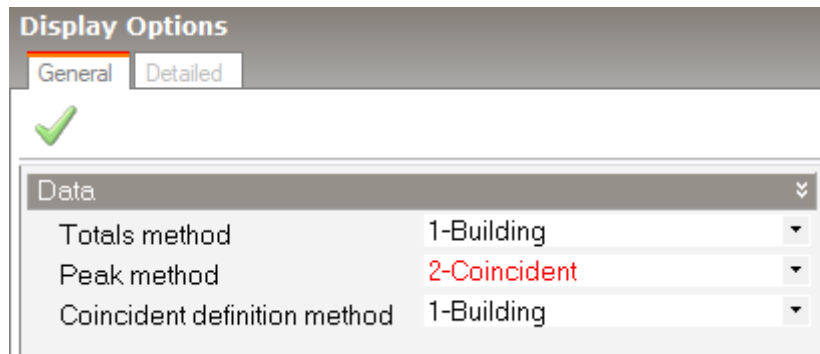
Summary results can be displayed either for the time and month of the overall maximum building cooling load or for the individual zone.

- **1-Non-coincident** - Summary results are for the time and month of maximum cooling load for each zone.
- **2-Coincident** - All summary results are for the time and month of the maximum building total cooling load.

Coincident definition method

When the **Peak method** is set to **2-Coincident** you can specify whether the coincident time is calculated for the building as a whole, i.e. for all zones, or whether it is calculated separately for each system. Select from:

- **1-Building**, where the time of peak cooling for zones is calculated by summing hourly totals for all zones in the building, or,
- **2-System**, where the time of peak cooling for zones is defined considering all zones in each system.



Display Options	
General Detailed	
✓	
Data	▼
Totals method	1-Building ▼
Peak method	2-Coincident ▼
Coincident definition method	1-Building ▼

Summary Results

The Summary tab shows a range of design cooling loads, flow rates internal and fabric gains for each zone and other related data. Outputs for each zone are summed based on the **Totals method** Display options described above:

- **Zone multiplier** is the [Zone multiplier](#) factor entered on the Activity tab. It is used to define the number of zones represented by this zone in the model. By default, it has a value of 1.
- **Design Capacity** (kW or kBtu/h), shown in bold, is the maximum value of **Total Cooling Load** and **Sensible** multiplied by the **Design margin** to give the design cooling capacity of the equipment. This value is written into the Cooling capacity Model data on the HVAC tab if the appropriate [Plant sizing](#) Model Option is set.
- **Design Flow Rate** (m3/s or ft3/min), shown in bold, is the supply air flow rate required to deliver the **Sensible** cooling load using the **Design supply temperature** and the zone air temperature at the time of the maximum load multiplied by the **Design margin**.
- **Total Cooling Load** (kW or kBtu/h) is the total maximum **Sensible** + **Latent** loads for the zone at the time of maximum [Sensible Cooling](#) load. If latent load is negative then it is ignored here and the Total Cooling Load is the same as the Sensible load.
- **Sensible** (kW or kBtu/h) is the maximum [Sensible Cooling](#) load for the zone in the Design day.
- **Latent** (kW or kBtu/h) is the latent load for the zone at the time of maximum Sensible load, calculated as [Total Cooling](#) - [Sensible Cooling](#).
- **Air Temperature** (°C or °F) is the Air temperature in the zone at the time of maximum Sensible load.
- **Humidity** (%) is the humidity in the zone at the time of maximum Sensible load.
- **Time of Max Cooling** - the time at which the maximum sensible cooling occurs.
- **Max Op Temp in Day** (°C or °F) is the maximum operative temperature in the zone (using radiant fraction = 0.5) over the design day including periods when the zone may be unconditioned.
- **Floor area** is the [floor area](#) of the zone.
- **Volume** (m3 or ft3) is the [volume](#) of the zone.
- **Flow/Floor area** (l/s-m2 or cfm/ft2) is the **Design Flow Rate** divided by the **Floor area** of the zone. It provides a useful check for the flow rates calculated.
- **Design Cooling Load Per Floor Area** (W/m2) is the **Design Capacity** divided by the **Floor area** of the zone. It provides a useful check for the design capacities calculated.
- **Outside Temperature** (°C or °F) is the outside temperature at the time of maximum Sensible load.
- **Glazing Gains** (kW or kBtu/h) are the gains through windows (excluding solar gains) at the time of maximum Sensible load.

- **Wall Gains** (kW or kBtu/h) are the gains through all external and internal wall (partition) surfaces at the time of maximum Sensible load.
- **Floor Gains** (kW or kBtu/h) are the gains through all floor surfaces at the time of maximum Sensible load.
- **Roof and Ceiling Gains** (kW or kBtu/h) are the gains through all roof + ceiling surfaces at the time of maximum Sensible load.
- **Ventilation Gains** (kW or kBtu/h) are the mechanical + natural ventilation gains at the time of maximum Sensible load.
- **Infiltration Gains** (kW or kBtu/h) are the infiltration gains at the time of maximum Sensible load.
- **Electric Equipment Gains** (kW or kBtu/h) are gains due to electric equipment (Computers, Office equipment, Miscellaneous, Catering, Process) at the time of maximum Sensible load.
- **Lighting Gains** (kW or kBtu/h) are gains due to lighting (General + Task) at the time of maximum Sensible load.
- **People Gains** (kW or kBtu/h) are the sensible gains due to people at the time of maximum Sensible load.
- **Solar Gains** (kW or kBtu/h) are the short and long wave solar gains through windows at the time of maximum Sensible load.
- **Mechanical Ventilation Fresh Air Rate** (m3/s or ft3/min) is the fresh air supply air flow rate at the time of maximum load.
- **Fresh air % of supply air** (%) is the % of the mechanical cooling design supply air that is from outside at the time of maximum load. The remainder of the supply air is assumed to be recirculated.

The image below shows an example of the summary results grid data when the [Totals method](#) is set to **1-Building**. Note the totals and averages listed for the whole building at the bottom of the grid.

Analysis Summary											
Zone	Block	Design Capacity (kW)	Design Flow Rate (m3/s)	Total Cooling Load (kW)	Sensible (kW)	Latent (kW)	Air Temperature (°C)	Humidity (%)	Time of Max Cool...	Max Op Temp in Day (°C)	Floor Area (m2)
Building 1											
Off NW	L2	13.49	0.799	11.73	10.02	1.71	24.00	50.0	Aug 15:00	26.72	140.948
Off E	L2	28.51	1.703	24.79	21.37	3.42	24.00	49.8	Aug 15:00	27.23	284.090
Off SW	L2	16.17	0.980	14.06	12.29	1.77	24.00	49.4	Aug 15:00	27.53	140.948
Off SW	L1	15.03	0.902	13.07	11.32	1.75	24.00	49.6	Aug 15:00	27.12	140.948
Off E	L1	26.44	1.565	22.99	19.64	3.35	24.00	50.1	Aug 15:00	26.81	284.090
Off NW	L1	12.20	0.714	10.61	8.96	1.65	24.00	50.4	Aug 15:00	26.22	140.948
Off NW	L0	12.47	0.731	10.85	9.17	1.68	24.00	50.3	Aug 15:00	26.26	140.948
Off E	L0	25.46	1.501	22.14	18.83	3.31	24.00	50.2	Aug 15:00	26.62	284.090
Off SW	L0	15.50	0.934	13.48	11.72	1.76	24.00	49.6	Aug 15:00	27.20	140.948
Totals	-	165.28	9.828	143.72	123.32	20.41	24.00	50.0	Aug 15:00	27.53	1697.955

The image below shows an example of the summary results grid data when the [Totals method](#) is set to **2-System**. Note the totals and averages listed for each system.

Analysis Summary											
Zone	Block	Design Capacity (kW)	Design Flow Rate (m3/s)	Total Cooling Load (kW)	Sensible (kW)	Latent (kW)	Air Temperature (°C)	Humidity (%)	Time of Max Cool...	Max Op Temp in Day (°C)	Floor Area (m2)
East											
Off E	L2	28.51	1.703	24.79	21.37	3.42	24.00	49.8	Aug 15:00	27.23	284.090
Off E	L1	26.44	1.565	22.99	19.64	3.35	24.00	50.1	Aug 15:00	26.81	284.090
Off E	L0	25.46	1.501	22.14	18.83	3.31	24.00	50.2	Aug 15:00	26.62	284.090
Totals	-	80.41	4.769	69.92	59.84	10.08	24.00	50.0	Aug 15:00	27.23	852.270
NW											
Off NW	L2	13.49	0.799	11.73	10.02	1.71	24.00	50.0	Aug 15:00	26.72	140.948
Off NW	L1	12.20	0.714	10.61	8.96	1.65	24.00	50.4	Aug 15:00	26.22	140.948
Off NW	L0	12.47	0.731	10.85	9.17	1.68	24.00	50.3	Aug 15:00	26.26	140.948
Totals	-	38.16	2.243	33.19	28.14	5.04	24.00	50.2	Aug 15:00	26.72	422.843
SW											
Off SW	L2	16.17	0.980	14.06	12.29	1.77	24.00	49.4	Aug 15:00	27.53	140.948
Off SW	L1	15.03	0.902	13.07	11.32	1.75	24.00	49.6	Aug 15:00	27.12	140.948
Off SW	L0	15.50	0.934	13.48	11.72	1.76	24.00	49.6	Aug 15:00	27.20	140.948
Totals	-	46.71	2.816	40.61	35.33	5.28	24.00	49.5	Aug 15:00	27.53	422.843

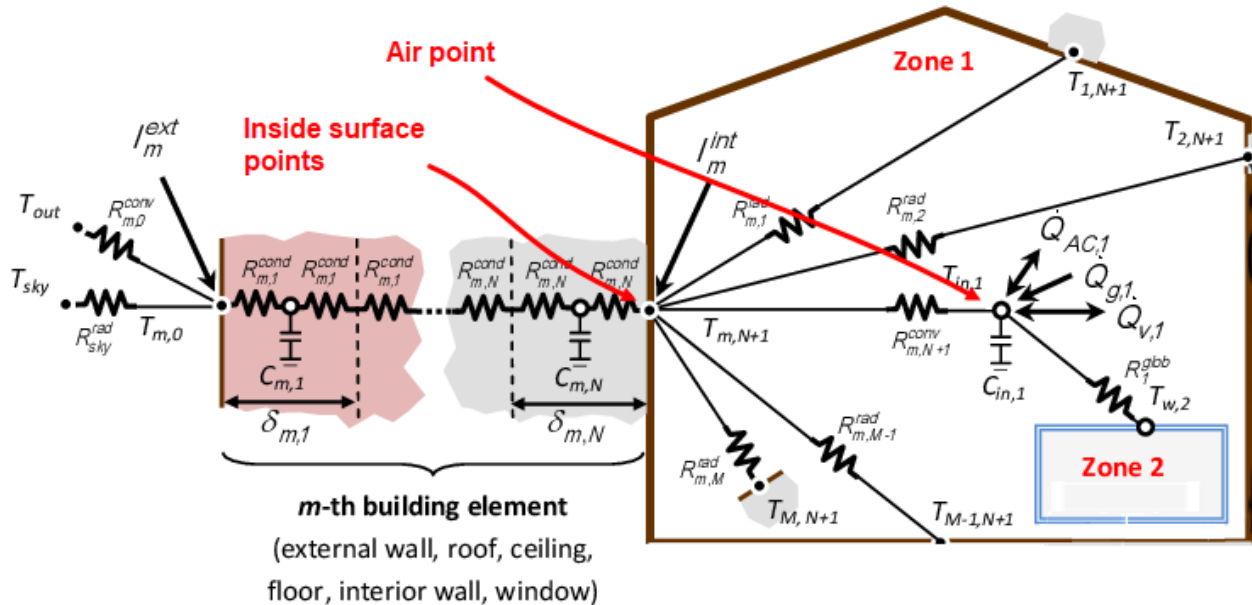
Tip: Use the **File > Export > Export HVAC loads report** command to view results in an industry standard PDF format. Alternatively, use the [Export](#) command to export summary data to spreadsheet format.

Why do the sum of the gains not equal the sensible cooling load?

It might be expected that the sum of the Glazing + Wall + Floor + Roof/Ceiling + Ventilation + Infiltration + Equipment + Lighting + People + Solar gains at design conditions would equal the Sensible cooling load. However, it is unrealistic to expect the sum of a mix of convective and radiant gains to equal the (purely convective) cooling load.

To understand more clearly we need to consider the nature of the building thermal resistance/capacitance network used in E+ (actually E+ uses a response factor method by default but the same principle applies).

For each zone, you can think of the thermal network as a single zone air node and a series of nodes for each surface. All zone convective gains are accumulated at the air node and radiant gains at the internal surface of each wall, window, roof etc. Conceptually speaking there is a resistance between the zone air and surface nodes and heat flows between them (through R_{conv} in the diagram below). The key to understanding the apparent discrepancy is that heat transferred to/from the surface nodes is not equivalent to heat exchanged at the air node. So the surface heat gains can't simply be combined together with gains at the air point to calculate cooling system sensible gain.



Why are some cells displayed in brown bold text?

In some cases, DesignBuilder may not be able to calculate a supply air flow rate, and in these cases, results may be displayed in brown bold text. This can occur when the [cooling supply air temperature](#) is the same as, or very close to, the zone air [Cooling setpoint temperature](#) which would lead to a divide by zero error. In such cases, the [Design flow rate](#) is set to zero and summary results are displayed in brown, even if there is actually a very small cooling load.

The screenshot below illustrates.

Zone	Block	Zone Multiplier	Design Capacity (kW)	Design Flow Rate (m3/s)	Total Cooling Load (kW)	Sensible (kW)	Latent (kW)	Air Temp
Pucities								
Zone 1	Chimney	1	0.00	0.000	0.00	0.00	0.00	-
Room 2.9	2nd floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 2.3	2nd floor	1	1.44	0.068	1.44	0.89	0.54	23.09
Room 2.4	2nd floor	1	0.40	0.456	0.40	0.40	0.00	22.73
Room 1.9-upper part	2nd floor	1	2.17	0.000	2.17	2.17	0.00	22.00
Room 2.1	2nd floor	1	0.14	0.062	0.14	0.14	0.00	23.95
Unconditioned area	2nd floor	1	0.00	0.000	0.00	0.00	0.00	-
ZoneRoom 1.1, 1.4-1.7 2...	2nd floor	1	4.99	0.274	4.99	3.40	1.60	22.40
Room 2.5	2nd floor	1	0.78	0.542	0.78	0.78	0.00	23.20
Room 2.2	2nd floor	1	0.33	0.306	0.33	0.33	0.00	22.90
Room 2.6	2nd floor	1	0.00	0.000	0.00	0.00	0.00	-
ZoneRoom 1.1, 1.4-1.7 2...	2nd floor	1	0.19	0.084	0.19	0.19	0.00	23.85
Room 1.12	1st floor	1	3.68	24.825	3.68	3.68	0.00	24.12
Room 1.3	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.15-front room	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.1, 1.4-1.7	1st floor	1	2.70	2.783	2.70	2.70	0.00	22.81
Room 1.2	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.13	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.10	1st floor	1	0.52	0.569	0.52	0.52	0.00	22.76
Room 1.8	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.11	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.15	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Room 1.9	1st floor	1	0.00	0.000	0.00	0.00	0.00	-
Totals	-	-	17.33	29.967	17.33	15.19	2.14	22.85

Simulation Using Hourly Weather Data

You can generate detailed building energy performance data based on simulations using hourly weather data.

Click on the **Simulation** Screen tab to view simulation data. If the necessary data has not yet been generated, a simulation is automatically started to generate the data.

Simulations using EnergyPlus have the following characteristics:

- Weather data comes from the selected [Hourly weather data file](#).
- Includes consideration of heat conduction and convection between zones of different temperatures.
- Includes solar gain through windows.
- Simulation of HVAC equipment ([options](#))
- Includes one or more 'warmup' (or pre-conditioning) days to ensure correct distribution of heat in building thermal mass at the start of the simulation. Warmup continues until temperatures/heat flows in each zone have converged. If convergence does not occur then simulation continues for the maximum number of days as specified in the calculation options.

You can customise the data displayed and display style using the [Display Options](#) panel.

- [Calculation Options](#)
- "Simulation Manager" on page 1609
- [Display Options](#)
- [Detailed Results](#)
- [Comfort Analysis](#)
- [Summary Output](#)
- [Parametric Analysis](#)
- [Optimisation](#)
- "Uncertainty and Sensitivity Analysis" on page 1783
- "ASHRAE 90.1 Reporting" on page 1898
- "Data Visualisation" on page 1898

See also the [Setting up simulations](#) and [Running Detailed HVAC simulations](#) tutorials

Simulation Calculation Options Dialog

The settings on the Simulation calculation options dialog control the simulation and the output produced.

Data is entered on the following tabs:

- "Simulation Calculation Options - Simulation Options" on page 1291
- "Options - Simulation" on page 1576
- "Simulation Calculation Options - Output" on page 1306
- "Simulation Manager" on page 1609

Click on the OK button to start the Simulation.

Auto-Update

This dialog is always shown when you select the **Update** toolbar icon and will also be shown before all simulations if the **Don't show this dialog** checkbox at the bottom of the dialog is cleared.

Simulation Calculation Options - Simulation Options

[Simulation tab](#) on [Model Options](#) dialog and [General tab](#) on [Simulation Options](#) dialog.

Calculation Description

Enter some text to identify the calculation. This will be used in reports, on Simulation Manager dialog and in the filename for [automatically generated ESO files](#).

Tip: If you forget to update the Description before running a simulation using the Simulation Manager you can [rename](#) it later in the Simulation Manager dialog.

Simulation Period

Start day, Start month, End day, End month

Select the start and end days for the simulation by selecting the start day and month and the end day and month.

Simulation Period

From

Start day: 17

Start month: Aug

☒ Specify year

Start year: 2024

To

End day: 23

End month: Aug

☐ Run simulation for multiple years

Alternatively select a typical period using the links on the Info panel:

- **Annual simulation** - all 365 days of the year.
- **Summer design week** - a week identified by the weather data translator as being the hottest of the year.
- **Summer typical week** - a week identified by the weather data translator as being typical of the summer.
- **All summer** - simulate the whole summer (as identified by the weather data translator).
- **Winter design week** - a week identified by the weather data translator as being the coldest of the year.
- **Winter typical week** - a week identified by the weather data translator as being typical of the winter.
- **All winter** - simulate the whole winter (as identified by the weather data translator).

Note: The above typical simulation periods are derived from the [Statistics data](#) from the currently selected [Hourly weather data](#). If the hourly weather data was loaded from a custom weather file that did not include statistics data (the .stat file) then data may not be available to allow these links to typical periods to work. In this case you must select the dates manually.

Specify year

For some modelling work it is necessary to simulate a specific year. For example when carrying out operational analysis on an occupied building and comparing simulation results with site measurements for a specific period. In such cases a "Typical Year" weather file cannot be used and the hourly weather file should represent a specific year. In these cases, you should check this checkbox and enter the year in the **Start year** textbox below.

Start year

When the **Specify year** checkbox has been checked, you must enter the year represented by the currently selected [Hourly weather file](#).

Note: When the **Start year** is specified here it overrides the [Day of week for start day](#) site level setting, and the year set here defines the day of the week of the start day.

Run simulation for multiple years

If you would like to run the simulation for more than one year then check this box. This can be useful for models where there is very heavy thermal mass involved anywhere in the model. A typical example is when modelling ground domains where the thermal mass of the ground itself is simulated in detail. Similarly, when simulating a ground heat exchanger it can take several years for the ground temperature fields to stabilise to periodic steady-state.

Number of years

When the Run simulation for multiple years checkbox is checked you must enter the number of years to be simulated. Enter a number 2 or greater.

Note: When simulating multiple years it is only possible to run for a whole number of years, each of which start in January and end in December. So it isn't possible to simulate from May one year to June 3 years later for example.

Output Intervals For Reporting

Select one or more intervals for reporting from the available list:

- Monthly and Run period
- Daily
- Hourly
- Sub-hourly

Warning: selecting output at hourly or sub-hourly intervals can produce large amounts of data which slows processing, results in large file sizes and in extreme cases can cause DesignBuilder to fail when attempting to load results back into the model for reporting. If you want to generate hourly or sub-hourly data you may wish to consider switching off some of the [Output options](#) to ensure you are only generating the data you require.

Options - Simulation

[Simulation tab](#) on [Model Options](#) dialog and [Options tab](#) on [Simulation Options](#) dialog.

On the Options tab of the Simulation options dialog you can enter settings for the following.

- "Simulation Calculation Options - Calculation Options" on page 1293
- "Simulation Calculation Options - Solar" on page 1296
- "Simulation Calculation Options - Advanced" on page 1301

Simulation Calculation Options - Calculation Options

[Simulation](#) tab on [Model Options](#) dialog and Options tab on [Simulation Options](#) dialog.

Time steps per hour

The Time steps per hour setting specifies the overall timestep for the simulation. The value entered here is also known as the Zone Timestep. This is used in the Zone Heat Balance Model calculation as the driving timestep for heat transfer and load calculations. The value entered here is the number of timesteps to use within an hour. Longer length timesteps have lower values for Number of timesteps per hour. For example a value of 6 entered here directs the program to use a zone timestep of 10 minutes and a value of 60 means a 1 minute timestep.

The user's choice for Number of timesteps per hour must be evenly divisible into 60 and the allowable choices are 1, 2, 4, 6, 10, 12, 30, and 60.

The choice made for this field has important implications for modelling accuracy and the overall time it takes to run a simulation. Here are some considerations when choosing a value:

- The solution technique used in EnergyPlus has been designed to be stable with zone timesteps of up to sixty minutes (Number Timesteps in Hour = 1). However, 60 minutes is considered a "long" timestep and it should only be used in rare occasions where there is no HVAC system, accuracy is not a concern, and short run times are critical. Such long timesteps are not recommended to use because simulation results are more accurate for shorter timesteps, of say 10 minutes or less (Number of timesteps per hour of 6 or more). Shorter zone timesteps improve the numerical solution of the Zone Heat Balance Model because they improve how models for surface temperature and zone air temperature are coupled together. Longer timesteps introduce more lag and lead to more a dampened dynamic response.
- Simulation run time increases with shorter timesteps or larger values for Number of timesteps per hour. The effect varies with the nature of the model. The user can test out different values on their particular model to understand the implications for his or her particular case. Sometimes large models with multizone HVAC and Plant systems execute nearly as fast with 15 minute timesteps as with 60 minute timesteps because fewer iterations are required in the system modelling since the prior timestep's results are close to the final outcome of next timestep.
- The weather data files usually have 60-minute (or hourly) data. However, it does not follow that this should be used as the basis for choosing the zone timestep because:
 - o EnergyPlus carefully interpolates the weather data between data points for use at shorter timesteps.
 - o Many aspects of a model have time scales that differ from the that of the weather data. A goal of the modelling is to predict how the building will respond to the weather. However, the building's response is not governed by the time scale that the weather data are available at, but rather the time scales of the dynamic performance of the thermal envelope as well as schedules for internal gains, thermostats, and equipment availability.
- If the model will also be used to calculate the cost of electricity then you should be aware that many electric utility tariffs base charges on demand windows of a specified length of time. If the choice of Number of timesteps per hour is not consistent with the demand window, then unexpected results may be obtained. For reasonable prediction of the maximum rates for electricity use for in calculating demand charges, the length of the zone timestep needs to be consistent with the tariff's demand window. The following table lists what values are consistent with various demand windows.

Demand Window Applicable	Number of timesteps per hour
Quarter Hour	4, 12, or 60
Half Hour	2, 4, 6, 10, 12, 30, or 60
Full Hour, Day, Week	Any

There is also second type of timestep inside EnergyPlus that is known as the System or HVAC timestep. This is a variable-length timestep that governs the driving timestep for HVAC and Plant system modelling. The user cannot directly control the system timestep (except by use of the Convergence limits data). When the HVAC portion of the simulation begins its solution for the current zone timestep, it uses the zone timestep as its maximum length but then can reduce the timestep, as necessary, to improve the solution. The technical details of the approach are explained in the Engineering Documentation under "Integrated Solution Manager".

Advanced EnergyPlus users can obtain and view data at intervals of the HVAC time step used if they select the 'detailed' option on an HVAC report variable when working directly with IDF data.

Though many buildings can be successfully simulated with 1 or 2 time steps per hour, EnergyPlus suggest a minimum of 4 for non-HVAC simulations and 6 for simulations with HVAC.

20 Timesteps per hour is the minimum when using the Finite difference solution method.

[Green roof](#) simulations may also require more timesteps.

Note 1: In general, increasing the number of time steps improves accuracy but slows the simulation (and generates more data if output is requested at the 'sub-hourly' interval).

Note 2: When using **1 time steps per hour** you will not be able to access [Temperature distribution](#) results

Temperature control

Heating and cooling systems control internal temperatures to meet the setpoint temperatures specified on the [Activity tab](#). These setpoint temperatures can be interpreted as air, operative or some other radiant fraction and DesignBuilder provides corresponding options to allow HVAC systems to be controlled by:

- **1-Air temperature** - control the zone mean air temperatures to the [heating and cooling setpoint temperatures](#) specified on the [Activity tab](#).
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this option.
- **3-Other**- you can enter the radiant temperature control fraction.
- **4-Fanger comfort** - this option is only available when using Detailed HVAC. It allows you to control zone comfort conditions using [Fanger PMV heating and cooling comfort setpoints](#) instead of the usual temperature setpoints. The PMV setpoint values are defined under the [Environmental control](#) header on the Activity tab when using the default **1-Simple HVAC Detailed HVAC Activity data** option. Alternatively, when using the **2-Detailed HVAC Detailed HVAC Activity data** option, time-varying PMV setpoints for heating and cooling are defined by Fanger PMV comfort schedules selected on the HVAC zone dialog .

When using the **3-Other** option the radiant fraction should be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Radiative Fraction vs Air Velocity

Air Velocity (m/s)	<0.2	0.2 - 0.6	0.6 - 1.0
Radiant fraction	0.5	0.4	0.3

Note: When the **2-DesignBuilder Simple HVAC autosize method** is selected, the temperature control settings used in [heating design](#) and cooling design will also apply in simulation autosizing calculations for the winter and summer design days respectively.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

You can think of the Temperature control option as:

- The radiant fraction sensed by the room thermostat, or,
- Given a room thermostat that mainly senses air temperature, the extent of automatic modification of the setpoint temperature to ensure occupant comfort.

Note: This option is overridden when using radiant heating systems by the [radiant heating system control setting](#).

Note: This option does not affect natural and mechanical [ventilation setpoints](#) - these always use **air temperature** set points.

Operative vs Air temperature control - Advanced discussion

There has been much debate amongst simulation experts over the years on the extent to which radiant effects should be included on the simulated thermostat. Most real-world thermostats probably don't actually sense more than about 20% radiant heat transfer - the temperature sensor will mainly be sensitive to the temperature of the nearby room air. So you might think that air temperature (or 20% radiant) control is the best choice. But Operative control (radiant fraction = 0.5) can be useful for calculating realistic heating and cooling energy based on published summer and winter temperature requirements for the activities in each zone. This is because HVAC systems controlled using the operative temperature continue to condition the building until comfort conditions are met (just like they are in real buildings where occupants may adjust thermostats until they are comfortable). Also, the default [temperature set points](#) from the [Activity templates](#) are generally derived from sources quoting operative temperatures. With Air temperature control the room *air* temperature is controlled to the set point temperature, which (depending on internal radiant temperatures) may not necessarily be comfortable.

The disadvantage of using **Operative** temperature control is that start up loads can be unrealistically high due to the lag in thermal response of the walls, floor, ceilings. The slow temperature response of the building fabric part governs the output of the operative thermostat and hence the operation of the heating/cooling equipment. If this effect dominates it can lead to an overestimate of the required design loads. You should be familiar with this issue before using operative temperature control to size heating and cooling equipment. In our experience, using Operative temperature control usually leads to significantly higher peak loads in Heating and Cooling design calculations and higher heating and cooling energy consumption in Simulations. DesignBuilder therefore uses air temperature control for heating and cooling autosizing simulations, even when the Operative or the Other control option is selected to avoid using unrealistically high capacities in the simulations. You should be aware that this can lead to undersizing and unmet hours when running ASHRAE 90.1 simulations. An alternative solution to avoid undersizing when using the Operative option is to make appropriate increases sizing factors or HVAC sizing settings. Some trials and error may be needed with this.

Caution: Operative temperature control can cause [EnergyPlus Error 3](#) when using Simple HVAC, or Cooling design calculations with Operative control in zones with strong radiant heat gains causing high radiant temperatures (e.g. uninsulated roof or zone is highly glazed). The error is caused by the fixed supply air temperature being higher than the zone air temperature required to give the operative setpoint (e.g. 24°C). The solution may be to use Air temperature control and to manage the high radiant temperatures using solar shading/insulation as appropriate.

Air temperature control is easier to use as none of the aforementioned problems apply, but it can lead to inadequate equipment sizes peak loads in Heating and Cooling design calculations when not used with a design safety factor. This is especially true when radiant temperatures are very different from air temperatures for example in poorly insulated buildings, buildings with large unshaded glazing areas or high ventilation rates. Generally using air temperature control in Simulations of such buildings will underestimate energy consumption.

Autosizing simple HVAC convective systems with operative temperature control

Another issue to bear in mind with operative temperature control is that in simple HVAC convective systems, autosized cooling systems use a different algorithm to calculate the maximum supply airflow rate used in the EnergyPlus **Ideal Loads** system. In convective systems the equation used is:

$\Delta T = \text{ZoneCoolingSetpointTemperature} - \text{HVACCoolingCoilSetpointTemp}$ (*difference in temperature between zone air and supply air*)

$\text{DesignSupplyAirFlowM3PerS} = \text{DesignMaxCoolingLoad} / (\text{Cp} * \Delta T * \text{AirDensity})$

This equation works because the temperature of the air in the zone can safely be assumed to be the zone cooling set point and so there is fixed difference in temperature between zone air and supply air. Calculating the design supply cooling airflow rate in this way does not work for operative temperature control because the air temperatures in the space are often much lower than the zone cooling setpoint temperature and sometimes in zones having very high radiant temperatures, the air temperature in the zone approaches the supply air temperature. In other words the difference in temperature between zone air and supply air in the simulated system becomes very low and therefore very large airflow rates are required to meet cooling loads.

Tip: As a general rule you should prioritise checking building **comfort** levels when using Air temperature control and **realistic plant operation** (oversized equipment, supply temperatures very low) when using Operative temperature control.

Simulation Calculation Options - Solar

[Cooling Design](#) and [Simulation](#) tabs on [Model Options](#) dialog and Options tab on [Simulation Options](#) dialog.

Solar Options allow you to control aspects of the model related to solar gains.

Include all buildings in shading calcs

The standard wall, floor, roof surfaces defining the current building provide shading but by default the other buildings on the site are excluded from the analysis. Use this option to include other buildings on the site as shading obstructions.

Note: there is no way to simulate more than 1 building in the same simulation. All external surfaces in buildings other than the current one are modelled as shading surfaces. If you need to model the adjacency of another building in contact with the current one, you can use the [adiabatic](#) adjacency option to model the touching surfaces.

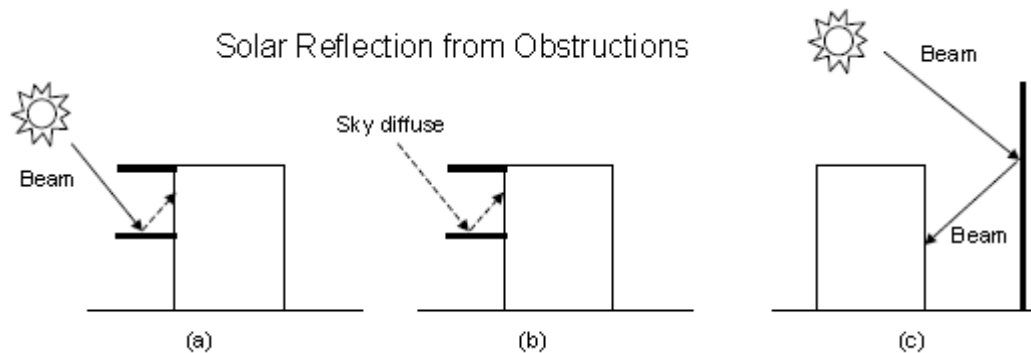
Note: if one or more of the 'other buildings' are large and/or complex then using this option could result in a large number of shading elements and the [simulation could run slowly](#) (even if the current building being simulated is simple).

Include shading from excluded zones in simulation

When zones are excluded from simulations, by unchecking the "Include Zone " on page 330 checkbox, it is often still necessary for the excluded the zone's surfaces to be used to shade other parts of the building. Checking this checkbox ensures that all building surfaces are used in shading calculations, even those that have been excluded.

Model all external reflections and shading of ground reflected solar

When this option is selected, the program calculates the beam and sky solar radiation that is reflected from exterior surfaces and then strikes the building. Also, this option enables the shadowing of ground-reflected radiation by obstructions such as the building itself and component blocks.

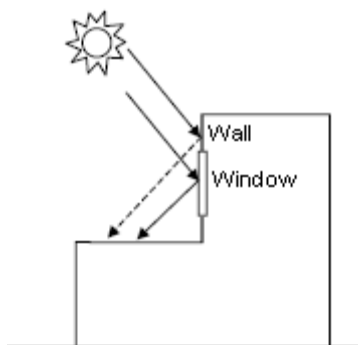


The above schematic shows solar reflection from various surface types. Solid arrows are beam solar radiation; dashed arrows are diffuse solar radiation. (a) Diffuse reflection of beam solar radiation from the top of an overhang (point 1 below). (b) Diffuse reflection of sky solar radiation from the top of an overhang (point 1 below). (c) Beam-to-beam (specular) reflection from the façade of an adjacent highly-glazed building represented by a vertical shadowing surface (point 2 below).

Reflecting surfaces fall into 3 categories:

1. Shadowing surfaces. These are surfaces of objects like overhangs, component blocks, assemblies and shading planes. These surfaces have diffuse reflectance derived from the associated material properties.

2. Exterior building surfaces. In this case one section of the building reflects solar radiation onto another section (and vice-versa). See schematic below. The building surfaces are assumed to be diffusely reflecting if they are opaque (walls, for example) and specularly reflecting if they are windows (or glass doors defined as windows). The reflectance values for opaque surfaces are calculated by the program from the [Solar Absorptance](#) and [Visible Absorptance](#) of the outer material layer of the surface's construction. The reflectance values for windows and glass doors are calculated by the program from the reflectance properties of the [individual glass layers](#) that make up surface's construction assuming no shading device is present and taking into account inter-reflections among the layers.

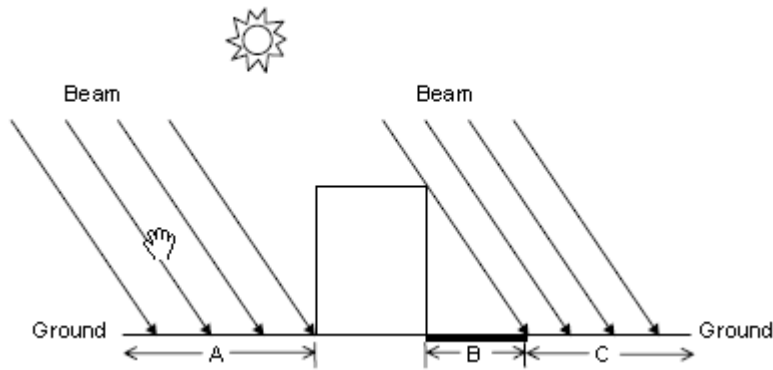


Solar reflection from building surfaces onto other building surfaces.

In this example beam solar reflects from a vertical section of the building onto a roof section. The reflection from the window is specular. The reflection from the wall is diffuse.

3. The ground surface. Reflection from the ground is calculated even if the **Model all external reflections and shading of ground reflected solar** option is not used. In this case the shadowing of the ground by the building itself or by obstructions such as neighbouring buildings is ignored. However if this option is applied then:

- Shadowing of ground reflected solar radiation by louvres, component blocks etc is taken into account, and,



- Shadowing from the building affects beam solar reflection from the ground. Beam-to-diffuse reflection from the ground onto the building occurs only for sunlit areas, A and C, not from shaded area, B.

Solar distribution

This option determines how EnergyPlus treats beam solar radiation and reflectance from exterior surfaces that strike the building and, ultimately, enter the zone. There are 3 choices:

1-Minimal shadowing - In this case, there is no exterior shadowing except from window and door reveals. All beam solar radiation entering the zone is assumed to fall on the floor, where it is absorbed according to the floor's solar absorptance. Any reflected by the floor is added to the transmitted diffuse radiation, which is assumed to be uniformly distributed on all interior surfaces. If no floor is present in the zone, the incident beam solar radiation is absorbed on all interior surfaces according to their absorptances. The zone heat balance is then applied at each surface and on the zone's air with the absorbed radiation being treated as a flux on the surface.

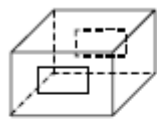
2-Full Exterior - in this case, shadow patterns on exterior surfaces caused by detached shading, wings, overhangs, and exterior surfaces of all zones are computed. As for Minimal shadowing, shadowing by window and door reveals is also calculated. Beam solar radiation entering the zone is treated as for 'Minimal shadowing' - all beam solar radiation entering the zone is assumed to fall on the floor, where it is absorbed according to the floor's solar absorptance. Any reflected by the floor is added to the transmitted diffuse radiation, which is distributed among interior surfaces according to view factors. If no floor is present in the zone, the incident beam solar radiation is absorbed on all interior surfaces according to their absorptance.

Note: If you use **2-Full Exterior** you should make sure that all of your zones have a floor. If your building model has any zones of unusual shape without floors then the EnergyPlus simulation may become unstable because it will not have a surface to apportion solar gains to.

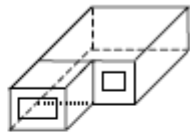
3-Full interior and exterior - this is the same as **Full exterior** except that instead of assuming all transmitted beam solar falls on the floor the program calculates the amount of beam radiation falling on each surface in the zone, including floor, walls and windows, by projecting the sun's rays through the exterior windows, taking into account the effect of exterior shadowing surfaces and window shading devices. If this option is used, you should be sure that the surfaces of the zone totally enclose a space, i.e. that all blocks are [closed manifold](#).

Also, unless you are using the **2-Pixel counting** [Shading calculation method](#), you should ensure that all zones in the model are convex when using this option. Examples of convex and non-convex zones are shown below. A common non-convex zone is an L-shaped zone. A formal definition of convex is that any straight line passing through the zone intercepts at most two surfaces. Another definition is that a zone is convex if every surface in a zone can 'see' every other surface - no surfaces 'around the corner' from each other.

If the above requirements are not met you should use **Full exterior** instead.



Convex zones



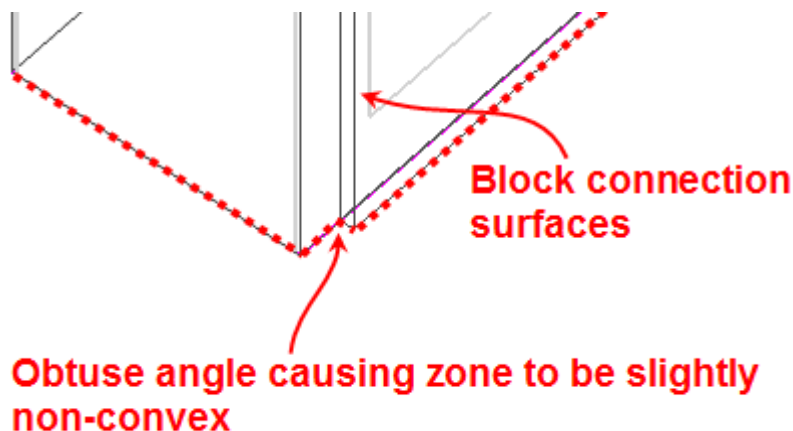
Non-Convex zones

If you use **3-Full Interior and exterior** the program will also calculate how much beam radiation falling on the inside of an exterior window (from other windows in the zone) is absorbed by the window, how much is reflected back into the zone, and how much is transmitted to the outside. In this calculation the effect of a shading device, if present, is accounted for.

Note: You must use **3-Full Interior and exterior** to account for direct solar and light transmission through internal windows.

Check for non-convex zones

When the solar distribution selection is **3-Full interior and exterior** (above) you can choose whether DesignBuilder is to check for non-convex zones. The DesignBuilder check for non-convex zones is very strict whereas the EnergyPlus check is less so. Generally EnergyPlus will tolerate a small degree of non-convex geometry. By default this option is switched on and the strict internal DesignBuilder convex check is carried out, but if the zones are only slightly non-convex you can switch off this check and allow EnergyPlus to continue calculations. You will probably receive [error message 2](#) if you switch this option off and have non-convex zones.



One common reason for the strict DesignBuilder check to flag a zone as being non-convex even when at first sight it is convex is the use of block connection geometry. In the above example of a double skin facade (DSF) cavity zone, you can see when zooming right in on the corner of a set of [block connection surfaces](#) that these surfaces cause obtuse angles to the zone geometry resulting in very slight non-convex geometry. These are exactly the sort of small deviations from convex that the DesignBuilder checker will fail and the EnergyPlus checks will pass. In this case it is perfectly safe to switch off this check to allow EnergyPlus calculations to proceed.

Shadow Calculations

The shadow calculation options are used to control some details of EnergyPlus' solar, shadowing and daylighting models. There are two basic methods available for the calculations. In order to speed up the

calculations, shadowing calculations (sun position, etc.) for the default **1-Average over days in frequency** method are performed over a period of days defined by the [Shadowing interval](#).

Shadow calculation method (Simulation only)

This setting is important for determining the amount of sun entering the building and by inference the amount of cooling or heating load needed for maintaining the building. Though termed “shadowing” calculations it, in effect, determines the sun position for a particular day in a simulation (each design day will use the date of the design day object). Even though weather file data contains the amount of solar radiation, the internal calculation of sun position will govern how that affects various parts of the building. By default, the calculations are done for every 20 days (depending on the setting in [Shadowing interval](#)) throughout a weather run period; an average solar position is chosen and the solar factors (such as sunlit areas of surfaces) remain the same for that number of days. When more integrated calculations are needed for controlling dynamic windows or shades (e.g. SageGlass), a second method is available where solar calculations are performed at each zone timestep.

Choose from:

- **1-Average over days in frequency**, the default and fastest method.
- **2-Timestep frequency**, a more detailed and slower method that must be used for modelling dynamic fenestration like SageGlass and shading surfaces

Shadowing interval (Simulation only)

This numeric field defines the interval between shadowing calculations in days. It only applies if **1-Average over days in frequency** [Shadow calculation method](#) is used. Using this field will allow you to synchronize the shadowing calculations with changes in shading devices. Using the default of 20 days in each period is the average number of days between significant changes in solar position angles. For these shadowing calculations, an “average” (over the time period) of solar angles, position, equation of time are also used.

Enter the number of days (1-365) covered by each shadowing calculation.

Enter 1 if you want shadowing to be calculated every day (for greater detail but slower calculations).

Shading calculation method

Select between CPU-based polygon clipping method or the GPU-based pixel counting method. Options are:

- **1-Polygon clipping** is the default option where shading is calculated on the CPU.
- **2-Pixel counting** where the shading is calculated using the GPU hardware if it is available. If GPU hardware (or GPU emulation) is not available, a warning will be displayed and EnergyPlus will revert to Polygon clipping. Unlike Polygon clipping, Pixel counting has no limitations related to zone concavity when used with the **3-Full Interior and exterior** [Solar distribution](#) option (i.e., it can accommodate both concave and convex zones equally). Use of the Pixel counting method requires some overhead in passing instructions between the CPU and the GPU. For low numbers of shading surfaces (less than about 200 for most hardware), Polygon clipping requires less runtime than Pixel counting. However, Pixel counting runtime scales significantly better at higher numbers of shading surfaces.

Important Note: When using the [Simulation Manager](#) to run your simulations, it is not straightforward to assign a specific graphics adapter to EnergyPlus because the program that executes the simulation is set up dynamically and so cannot be selected from the Windows Graphics Settings dialog. You should consider this when attempting to use the **2-Pixel counting** method and check the EnergyPlus error file for a possible report that the Polygon clipping method was used instead.

Note: Because the [JESS](#) simulation server doesn't include the necessary GPU hardware, the **2-Pixel counting** option will not work when simulations are run on this service, and the **1-Polygon clipping** option will be used instead.

Sky diffuse modelling algorithm (Simulation only)

Two choices are available:

- **1-Simple sky diffuse modelling**, the default option performs a one-time calculation for sky diffuse properties.
- **2-Detailed sky diffuse modelling**, the required setting if you have shadowing surfaces with changing transmittance (i.e. not all opaque or not all transparent) during the year. The program checks to see if this might be the case and automatically selects this option if the shading transmittance varies. When this option is selected, a warning is generated if the [Shadowing interval](#) (above) is > 1. In general (and you should also read the previous field description), if shadowing surfaces are used with the transmittance property, you should be careful to synchronize this calculation with the scheduled occurrence of the transmittance (if any), or use 1, which will be the most accurate but will cause more time in the calculations.

This field only applies to the **1-Average over days in frequency** [Shadow calculation method](#). When the **2-Timestep frequency** method is used, the **2-Detailed sky diffuse modelling** option is always used.

Simulation Calculation Options - Advanced

[Simulation](#) tab on [Model Options](#) dialog and Options tab on [Simulation Options](#) dialog

General Solution

Solution algorithm

This option controls the overall algorithm used for all the surfaces in the simulation apart from any that may be overridden through use of the construction [Simulation solution algorithm](#) selection for particular surfaces. Select from:

- **1-CTF** - The default method used in EnergyPlus for CTF calculations is known as the state space method (Ceylan and Myers 1980; Seem 1987; Ouyang and Haghighat 1991). CTF is a sensible heat only solution not taking into account moisture storage or diffusion in the construction elements.
- **2-Finite difference** - This solution technique uses a 1-D finite difference solution in the construction elements. It is a sensible heat only solution and does not take into account moisture storage or diffusion in the construction elements. This option is required for [PCM](#) simulations and may improve accuracy for sheet metal material layers in constructions and for chilled ceilings.
- **4-Combined Heat And Moisture Finite Element (HAMT)** - A coupled heat and moisture transfer and storage solution uses a one dimensional finite difference solution in the construction elements. It requires further moisture transfer property data inputs which are defined in the "Moisture Transfer" on page 1393 component.
- **5-Moisture Penetration Depth Conduction Transfer Function (EMPD)** - A sensible heat diffusion and an inside surface moisture storage algorithm that also needs additional moisture material property data inputs which are defined in the "Moisture Transfer" on page 1393 component.

Allow individual constructions to override solution method

Check this option to allow [Simulation solution algorithm](#) settings made for Constructions to override the general setting above.

Finite Difference Settings

The settings below are required when the general [Solution algorithm](#) is set to **2-Finite Difference** or if any constructions used in the simulation [override](#) the general setting to use the finite difference algorithm.

Difference Scheme

This field determines the solution scheme used by the Conduction Finite Difference model. There are two options:

- **1-Fully implicit first order** scheme which is first order in time and is more stable over time. But it may be slower than option 2.

- **2-Crank Nicholson 2nd order** which is second order in time and may be faster than option 1 but it can be unstable over time when boundary conditions change abruptly and severely.

Space discretisation constant

The Space discretisation constant controls the how the model determines the number of nodes used to represent each material layer in the construction. The model calculates the nominal distance associated with a node, Δx , using:

$$\Delta x = (C\alpha\Delta t)^{0.5}$$

Where:

- α is the thermal diffusivity of the material layer, in m²/s
- Δt is the length of the timestep in seconds.
- C is a constant set by this field.

The default is 3. Typical values are from 1 to 3. Lower values for this constant lead to more nodes and finer-grained space discretisation.

Relaxation factor

The finite difference solver includes under-relaxation for improved stability for interactions with the other surfaces. This input field can optionally be used to modify the starting value for the relaxation factor. Larger numbers may solve faster, while smaller numbers may be more stable. The default is 1.0. If the program detects numerical instability, it may reduce the value entered here to something lower and more stable.

Inside face surface temperature convergence criteria

The surface heat balance model at the inside face has a numerical solver that uses a convergence parameter for a maximum allowable differences in surface temperature. This field can optionally be used to modify this convergence criteria. The default value is 0.002 and was selected for stability. Lower values may further increase stability at the expense of longer runtimes, while higher values may decrease runtimes but lead to possible instabilities. The units are °C or °F.

Airflow Network

Maximum iterations

Enter the maximum number of iterations allowed in finding an Airflow Network solution. If the number of iterations at each simulation timestep is above the maximum number of iterations defined by this field, the program could not find the solution and a Severe error is issued and the program is aborted. The default value is 1000.

Relative airflow convergence tolerance

The [calculated natural ventilation](#) solution is assumed to have converged when the absolute value of the sum of the mass flow rates divided by the sum of the absolute value of the mass flow rates is less than the tolerance value entered here. The mass flow rates refer to the flow at all nodes in the airflow network model. The solution converges when both this tolerance and the tolerance in the **Absolute airflow convergence tolerance** are satisfied. The default value is 1×10^{-4} . Units are kg/s or lb/s.

Note: In cases where a large opening exists (in particular horizontal openings) this value and the absolute airflow convergence below may need to be increased by a factor of 10 or more to allow convergence to take place.

Absolute airflow convergence tolerance

The solution is assumed to have converged when the summation of the absolute value of all network airflows is less than the value specified for this input field. The default value is 1×10^{-6} .

Convection

Inside and Outside convection algorithms

You can select from a range of EnergyPlus inside convection algorithms for calculating the convection between internal zone surfaces and the rest of the zone air in the simulation calculations. More details on this and external convection can be found under [Constructions](#) Model Data > [Surface Convection](#) header.

Warmup

Warmup or pre-conditioning is the process of repeatedly simulating the first day before the simulation proper starts to ensure that the temperatures in the building fabric are realistic. Warmup continues until temperatures and heat flows in each zone have converged. If convergence does not occur then simulation continues for the maximum number of days as specified in the calculation option below.

Warmup and starting simulations on a weekend

Preconditioning of the building by repeated simulation of the first day until convergence is an important part of the simulation but it can cause issues if the first day in the simulation is extreme in any way. For example if the first day in the simulation is a Sunday and the building is not heated or cooled at all on that day and extreme overheating occurs then the fabric of the will be "charged" up with heat as if those conditions had been occurring repeatedly for many days (or even weeks in some heavyweight buildings). This can mean that on the next day, the Monday the building model is far more liable to overheat than would have been the case if the simulation had started on a different day. In cases where this may be an issue, it may be worth starting simulations on the first occupied day rather than on a weekend day.

Minimum number of warmup days

This data specifies the minimum number of "warmup" days before EnergyPlus will check if it has achieved convergence and can thus start running the simulation proper.

Value to use

Research into the minimum number of warmup days indicates that 6 warmup days is generally enough on the minimum end of the spectrum to avoid false predictions of convergence and thus to produce enough temperature and flux history to start EnergyPlus simulation. This was based on a study that used a set of benchmark reference buildings. It also was observed that convergence performance improved when the number of warmup days increased. As a result, the default value for the minimum warmup days has been set to 6. You should decrease this number only if you have knowledge that a specific model converges more quickly than 6 days. You may wish to increase the value in certain situations when, based on review of simulation outputs, it is determined that EnergyPlus has not converged. While this parameter should be less than the previous parameter, a value greater than the value entered in the field **Maximum number of warmup days** above may be used when users wish to increase warmup days more than the previous maximum number of days. In this particular case, the maximum value will be automatically reset to the value entered here and EnergyPlus will run exactly the number of warmup days specified here.

See also Temperature and loads convergence below.

Maximum number of warmup days

The maximum number of "warmup" or pre-conditioning days that can be used in the simulation before the simulation itself starts. A warning message will occur when the warmup simulation does not converge in the allowed maximum number of warmup days:

```
Loads Initialization did not Converge (CheckWarmupConvergence)
```

This error is usually caused by using very thick constructions (e.g. ground floors). You may need to increase the **Maximum number of warmup days** to get convergence, but some anomalous buildings may still not converge.

See also Temperature and loads convergence below.

Temperature and loads convergence

The temperature and loads convergence values represent the maximum allowable difference in zone temperature and loads between successive daily iterations before convergence is considered to have been reached during warmup.

Convergence of the simultaneous heat balance/HVAC solution is reached when both the loads and temperature criteria are satisfied. Both tolerances work the same way, one looks at temperatures and one looks at heating and cooling loads. After the second warm-up day, the program compares the maximum temperature experienced in a space with the maximum temperature from the previous day. If those two temperatures are within the tolerance, then it has passed the first warm-up check. It does a similar comparison with lowest temperatures experience within all the zones. If the current simulation day and the previous day values are within the tolerance, then it has passed the second warm-up check. A similar comparison is carried out with the loads tolerance and the maximum heating and cooling loads that are experienced within the spaces. Those are compared individually to the values for the previous day. If they are both in tolerance, then the simulation has passed the third and fourth warm-up check. The simulation stays in the warm-up period until ALL FOUR checks have been passed.

Note: The [maximum number of warmup days](#) will override the above convergence criteria, i.e. the simulation proper will start even if convergence has not occurred after the maximum number of warmup days.

Shading

Maximum number of shadow overlaps

Enter the maximum number of figures per shadow overlap. The shadow overlaps is a measure of the amount of complexity in the shading calculation and this maximum value allows you to limit the amount of time spent in the solar initialisation calculations.

Note: entering a small value here can speed up simulations in complex buildings. If you plan to use [Maximum shadow overlaps](#) below the default value of 15000 you should check accuracy of solar gains relative to results using default value

Polygon clipping algorithm

This is an advanced feature. Prior to V7, the internal polygon clipping method was a special case of the Weiler-Atherton method. Now, two options are available:

- [1-Sutherland Hodgman](#) (default)
- [2-Convex Weiler Atherton](#)

Theoretically, Sutherland-Hodgman is a simpler algorithm but it works well in cases where receiving surfaces (of shadows) are non-convex. The Weiler-Atherton implementation is only accurate where both casting and receiving surfaces are convex. Warnings/severe errors are displayed when necessary. More details on polygon clipping are contained in the [Engineering Reference](#).

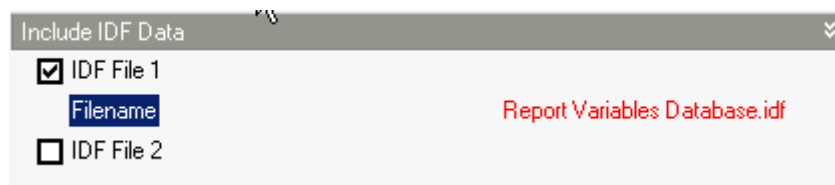
Include IDF Data

You can include up to 2 custom IDF data in your EnergyPlus simulations by checking the [IDF File 1](#) or [IDF File 2](#) check boxes and entering the name of the IDF data source file. The contents of any files specified are copied to the end of the IDF data generated by DesignBuilder before simulation. This can be a useful way to add in extra output reports, simulation objects etc, that do not interfere with the main IDF data.

For example if you needed to access output reports data not normally provided by DesignBuilder such as "Time Heating Setpoint Not Met While Occupied" then you might save a small IDF file in the DesignBuilder [EnergyPlus folder](#) called **Reports Variables Database.idf** containing this data:

```
Output:Variable, *, Time Heating Setpoint Not Met While Occupied, monthly;
Output:Variable, *, Time Cooling Setpoint Not Met While Occupied, monthly;
```

The file must be selected on this dialog as shown below.



After the simulation is finished the above setpoint not met monthly data would be included in the eso file which can be viewed outside DesignBuilder to obtain the extra outputs.

Important Note: Included IDF files must be located in the [EnergyPlus folder](#).

Tip: A more flexible way to include IDF is available through [EMS Scripting](#). Scripts are saved with the dsb file so they are completely portable. Also you can store and manage a list of various IDF datasets with EMS which makes it easier to switch various models and/or outputs in and out very easily. See "How to Generate and View Extra EnergyPlus Outputs" on page 2205 for more details on the recommended method.

Other

Surfaces within zone treated as adiabatic

EnergyPlus support recommend modelling surfaces wholly contained within a zone as adiabatic and this option allows you to follow this advice. We have found that in practice this option does not make much difference in results or in simulation speed so for most cases you can leave it in its default state.

See also [EnergyPlus Errors and Warnings](#)

Note: 'surfaces contained within a zone' are frequently generated when one of the [zone merging](#) options is used where a partition or floor which would have separated two zones actually sits within the merged zone. These surfaces *do not* refer to [hanging partitions](#) which are modelled using [Internal thermal mass](#).

Simulation Calculation Options - Output

[Simulation](#) tab on [Model Options](#) dialog and Output tab on [Simulation Options](#) dialog and [Outputs](#) model data tab.

Output Data

You can control the data generated from simulations by making selections on the Simulation tab of the Model options dialog, on the Output tab of the Simulation calculation options dialog or on the [Miscellaneous](#) model data tab.

Building and block output of zone data

This option causes average and total zone data to be generated for building and blocks. For example average temperatures, comfort conditions, total solar radiation, internal gains, fabric transmission losses etc. It does not affect the generation of energy consumption and CO2 data at the building level.

Note: It is not possible for DesignBuilder to display block level results when any type of [zone merging](#) has been enabled regardless of this setting.

Include unoccupied zones in block and building totals and averages

By default, data for unoccupied zones is *excluded* from block and building totals and averages. Check this option to include data for unoccupied zones.

A zone is defined as unoccupied if it has:

- No occupancy and,
- No cooling and,
- No heating and,
- No mechanical ventilation

The way building and block averages are calculated is described in the [Comfort Analysis](#) section.

Note: if you uncheck this option and all zones in the model are unoccupied then you will get a 'Zero floor area error' message

Allow custom outputs

By default this option is checked and it is possible to select specific outputs for building, zones, surfaces and openings on the [Outputs](#) model data tab. The disadvantage of specifying specific output selections for each EnergyPlus object is that it can slow the simulations down, especially the initialisation period before the main simulation starts. For large models this can cause extremely long delays and in this case it is advisable to switch this option off.

When this option is switched off the "*" character is used in the Output:Variable IDF data to mean "all objects", instead of specifying outputs for each object one by one. Because of the way that EnergyPlus is structured this results in much more efficient processing.

The following limitations apply when **Allow custom outputs** is deselected:

- The Simulation output options on the [Outputs](#) model data tab can only be selected at building level and building level options apply to all zones.
- Glazing results aren't displayed on the Heat Balance output graph on the Simulation Screen.

Tip: For very large models it is usually best to uncheck this option to reduce simulation times.

Graphable Outputs - Energy

Surface heat transfer

Select this option to generate heat flow at surface boundaries. Note that selecting this option results in large amounts of data being generated as EnergyPlus must generate output for every surface and window in the

zone and you should only select it if you need to look at the zone heat balance. The following EnergyPlus reports are generated:

- Opaque Surface Inside Face Conduction
- Window Heat Gain
- Window Heat Loss
- Window Transmitted Solar

For more information, see under the [Fabric and Ventilation](#) header of the "Simulation Detailed Results" on page 1622 page.

Internal gains including solar

Select this option to generate internal gains-related output and any associated energy consumption. The following EnergyPlus reports are generated:

- Zone Lights Total Heat Gain
- Zone Task Lights Total Heat Gain
- Zone People Sensible Heat Gain
- Zone Electric Equipment Cat02 Electric Consumption
- Zone Electric Equipment Cat03 Electric Consumption
- Zone Electric Equipment Cat04 Electric Consumption
- Zone Electric Equipment Cat05 Electric Consumption
- Zone Transmitted Solar
- Zone Infiltration Sensible Heat Loss
- Zone Infiltration Sensible Heat Gain
- Zone Ventilation Sensible Heat Loss
- Zone Ventilation Sensible Heat Gain

For more information, see under the [Internal Gains](#) header of the "Simulation Detailed Results" on page 1622 page.

Energy, HVAC etc

Select this option to generate energy consumption data for the HVAC system.

For more information, see under the [System Heat Flows](#) and [Fuel breakdown](#) headers of the "Simulation Detailed Results" on page 1622 page.

Latent loads

Select this option to include output of latent gains from occupants and other internal gains.

For more information, see under the [Internal Gains](#) header of the "Simulation Detailed Results" on page 1622 page.

Graphable Outputs - Comfort And Environmental

Reporting period

You can restrict the simulation *comfort and environmental* outputs using the options:

- **1-All periods** - the default option where data is generated for all periods including times when the zone is unoccupied.
- **2-Just occupied periods** - data is only output during times when the zone is occupied so daily and monthly comfort statistics become more meaningful.
- **3-User-defined schedule** - data is only output during times when the value of the **Reporting schedule** selected below has a value greater than zero.

Note that temperature distribution outputs (below) are unaffected by this setting as they have their own specific reporting period and schedule options.

Reporting schedule

Select the schedule to be used to restrict outputs when the **Reporting period** (above) is set to **3-User-defined schedule**.

Environmental

This option generates temperatures and humidity data. The following EnergyPlus reports are generated:

- Zone Mean Air Temperature
- Zone Mean Radiant Temperature
- Zone Air Relative Humidity
- Outdoor Dry Bulb

For more information, see under the [Environmental/Comfort Output](#) header of the "Simulation Detailed Results" on page 1622 page.

Fresh air supply

Select this option to include total infiltration, natural ventilation plus mechanical fresh air supply outputs.

For more information, see under the [Fabric and Ventilation](#) header of the "Simulation Detailed Results" on page 1622 page.

Simple ASHRAE Standard 55

- Time Not Comfortable Summer Clothes
- Time Not Comfortable Winter Clothes
- Time Not Comfortable Summer or Winter Clothes

For more information, see under the [Environmental/Comfort Output](#) header of the "Simulation Detailed Results" on page 1622 page.

Adaptive ASHRAE Standard 55

Checking this option causes these outputs to be generated:

- **Zone Thermal Comfort ASHRAE 55 Adaptive Model 80% Acceptability Status** - Reports whether the operative temperature falls into the 80% acceptability limits of the adaptive comfort in ASHRAE 55-2010. A value of 1 means within (inclusive) the limits, i.e. the zone operative temperature is no more than 2.5°C from than the ASHRAE 55 Adaptive model temperature. A value of 0 means outside the limits (i.e. uncomfortable), and a value of -1 means not applicable, i.e. the running average outdoor air temperature is below 10°C.
- **Zone Thermal Comfort ASHRAE 55 Adaptive Model 90% Acceptability Status** - Reports whether the operative temperature falls into the 90% acceptability limits of the adaptive comfort in ASHRAE 55-2010. A value of 1 means within (inclusive) the limits, i.e. the zone operative temperature is no more than 3.5°C from than the ASHRAE 55 Adaptive model temperature. A value of 0 means outside the limits (i.e. uncomfortable), and a value of -1 means not applicable, i.e. the running average outdoor air temperature is below 10°C.
- **Zone Thermal Comfort ASHRAE 55 Adaptive Model Running Average Outdoor Air Temperature** - the running average of the daily average outdoor dry-bulb temperatures of the previous 30 days.
- **Zone Thermal Comfort ASHRAE 55 Adaptive Model Temperature** - the ideal indoor operative temperature, or comfort temperature, as determined by the ASHRAE-55 adaptive comfort model. The 80% acceptability limits for indoor operative temperature are defined as no greater than 2.5°C from the adaptive comfort temperature. The 90% acceptability limits are defined as no greater than 3.5°C from the adaptive comfort temperature.

Note: ASHRAE 55 Adaptive outputs must be viewed in the Results Viewer - they are not available on DesignBuilder Simulation screen.

Note: Since valid ASHRAE 55 Adaptive outputs are only generated when the running mean outside air temperature is higher than 10°C, they apply mainly to summer overheating comfort analyses and so do not pick up winter underheating.

Adaptive CEN Standard 15251

Checking this option causes the following outputs to be generated:

- **Zone Thermal Comfort CEN 15251 Adaptive Model Category I Status** - reports whether the operative temperature falls into the Category I (90% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits, and a value of -1 means not applicable.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Category II Status** - reports whether the operative temperature falls into the Category II (80% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits, and a value of -1 means not applicable.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Category III Status** - reports whether the operative temperature falls into the Category III (65% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits, and a value of -1 means not applicable.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Running Average Outdoor Air Temperature** - reports the weighted average of the outdoor air temperature of the previous five days, an input parameter for the CEN-15251 adaptive comfort model.
- **Zone Thermal Comfort CEN 15251 Adaptive Model Temperature** - reports the ideal indoor operative temperature, or comfort temperature, as determined by the CEN-15251 adaptive comfort model. Category I, II, and III limits for indoor operative temperature are defined as no greater than 2, 3, and 4°C from this value respectively.

Note: CEN Standard 15251 Adaptive outputs must be viewed in the Results Viewer - they are not available on DesignBuilder Simulation screen.

CIBSE TM52

Select this option to generate [TM52 outputs](#).

Building category

Select the building category. See "Comfort Analysis" on page 1628 for more details.

Note: [CIBSE TM52 outputs](#) must be viewed in the Results Viewer - they are currently not available on DesignBuilder Simulation screen.

CIBSE TM59

Select this option to generate [TM59 overheating analysis outputs](#). When this option is checked the following further options can be selected.

Note: TM59 results are not available on DesignBuilder Simulation screen. CIBSE [TM59:2017 outputs](#) must be viewed in the Results Viewer, and [TM59:2026 outputs](#) must be reviewed through the dedicated plugin.

Vulnerable occupants

Selecting this output will change the building category for predominantly naturally ventilated buildings according to TM59 chapter 4.2 as described in the [TM59 section](#).

Natural ventilation rules

There are 4 options to choose from:

- **1-Standard EnergyPlus controls** - applies standard natural ventilation controls as explained in the main help sections.
- **2-TM59 rules** - overrides default EnergyPlus natural ventilation behaviour in favour of the implied TM59 rule to open windows when the inside temperature is greater than 22°C, regardless of outdoor air temperature.
- **3-Approved Document O rules** - applies TM59 rules but with the [limits imposed by Approved Document O](#) automatically applied.
- **4-TM49 Scotland** - For Scotland-specific TM59 calculations.

Note: When choosing options 2 or 3, please be sure to read the [detailed discussion on this option](#) in the TM59 temperature control section, especially if you plan to use the model for energy analysis as well as for TM59 overheating analysis.

Corridors

Selecting this output will produce results according to TM59 chapter 4.5 as described in the [TM59 section](#).

Restrict venting factor by schedule

This option allows you to vary an opening's free aperture through the day, typically to hold windows more closed overnight than during the day for security and/or noise reasons. When this option is selected, the [Opening operation schedule](#) defines a fraction that multiplies the window's free aperture, so the effective opening at any hour is % Glazing area opens × schedule fraction.

Note: The [Restrict venting factor by schedule](#) option is only available for the **2-TM59 Rules** and **3-Approved Document O Rules** natural ventilation rules outputs. It is not available for the other TM59 rules options. Also, this functionality does not apply to non-TM59 natural ventilation simulations, where the natural ventilation operations schedule only defines availability, in which case, schedule values of 0 mean the window is closed, and values > 0 mean that it is fully open.

Fanger

Select this option to generate Fanger PPD and PMV outputs.

Note: PMV outputs should only be viewed on a zone by zone basis at hourly or sub-hourly intervals. When viewing averages over longer intervals the meaning of this output can easily become distorted. For example consider a simple case where there is extreme overheating in the summer (e.g. average PMV = +3) and similarly extreme underheating in the winter (e.g. average PMV = -3). The average over the year might be close to zero, but that does not mean that the building was comfortable throughout the year! For similar reasons the building average PMV should be used with extreme caution.

Pierce 2-Node

Select this option to generate the following Pierce 2-Node outputs:

- Pierce Model Effective Temperature PMV
- Pierce Model Standard Effective Temperature PMV
- Pierce Model Discomfort Index
- Pierce Model Thermal Sensation Index

Kansas State University two-node

Select this option to generate KSU Model Thermal Sensation Vote outputs using the KSU two-node thermal comfort model.

Temperature distribution

Select this option to generate [Temperature distribution](#) data for each zone.

Reporting period

You can restrict the outputs for simulation *temperature distribution* only in 2 ways using the options:

- **1-All periods** - data is included for all periods including times when the zone is unoccupied.
- **2-Just occupied periods** - the default option where data is only included for times when the zone is occupied.
- **3-User-defined schedule** - data is only included for times when the value of the **Reporting schedule** selected below has a value greater than zero.

Reporting schedule

Select the schedule to be used to restrict outputs when the **Reporting period** (above) is set to **3-User-defined schedule**.

Detailed Daylight Outputs

Daylight map output

Generates the EnergyPlus daylight map outputs.

Details of this report are shown in the [EnergyPlus Output Details document](#).

Daylight factor, illuminance and glare data

When this option is selected additional daylighting data is generated as described below.

Daylight factor data is written to the .eio file and the daylight illuminance and glare data is written to the eso file for viewing in the [Results Viewer](#).

Daylight factors

Daylight factors are ratios of interior illuminance at a specific location to exterior horizontal illuminance. In EnergyPlus, they are pre-calculated hourly for every shadow calculation day for every exterior window of a daylight zone. The pre-calculated daylight factors are interpolated between hours and for the actual sky conditions in the time-step calculation of the daylighting performance of daylight zones.

EnergyPlus reports the daylight factors for four sky types (clear, turbid clear, intermediate, and overcast) at noon time (12:00pm) for each exterior windows (base window without shading) in the eio file when the daylight factors are first calculated. This reports all hourly pre-calculated daylight factors.

Details of this report are shown in the [EnergyPlus Output Details document](#).

Daylighting Reference Point <x> Illuminance

The total daylight illuminance at the first and second reference points from all of the windows in a daylit zone.

Daylighting Reference Point <x> Glare Index

The daylight glare index at the first and second reference points in a daylit zone.

Summary Outputs

Summary output units

EnergyPlus internally calculates energy in the form of J, but since the magnitude of those numbers for most buildings is very large, conversion options are provided for the summary report tables.

Select the units for energy to be reported in the EnergyPlus summary table. Options are:

- 1-kWh - J converted into kWh (1 / 3,600,000).
- 2-None - no conversions performed, i.e. energy data is provided in J.
- 3-MJ - J converted into MJ (1 / 1,000,000)
- 4-GJ - J converted into GJ (1 / 1,000,000,000)
- 5-IP - convert all annual, monthly, economics and timebins tabular values to common Inch-Pound equivalents.

The **1-kWh**, **3-MJ** and **4-GJ** unit conversion input options apply only to the Output:Table:Monthly reports and partially to the ABUPS report. For ABUPS, the **1-kWh** option changes the report but the **3-MJ** and **4-GJ** options do not since they it is already in MJ/m2 and GJ. In addition, the **1-kWh** option also changes results in the LEED Summary report, changing GJ to kWh and MJ/m2 to kWh/m2.

Summary Annual Reports

The easiest option is to select the **All Summary** report option which will produce all of the annual summary reports, except the Component Load Summary reports.

Including the **Component Load Summary** report will increase the simulation run time.

Also note that the **Envelope Summary**, **Surface Shadowing Summary** and **Shading Summary** reports can be very long for large models since they include entries for every surface / opening.

The screenshot below shows the summary annual reports available for selection.

Summary Annual Reports

- ☐ All Summary
- ☐ LEED Summary
- ☒ Annual Building Utility Performance Summary (ABUPS)
- ☒ Demand End Use Components Summary
- ☒ Sensible heat gain summary
- ☒ Input Verification and Results Summary
- ☐ Source Energy End Use Components Summary
- ☒ Adaptive Comfort Summary
- ☐ Zone Component Load Summary
- ☒ Standard 62.1 Summary
- ☐ Energy Meters
- ☒ Climatic Data Summary
- ☒ Equipment Summary
- ☒ Envelope Summary
- ☐ Initialization Summary
- ☐ Surface Shadowing Summary
- ☐ Shading Summary
- ☒ Lighting Summary
- ☒ HVAC Sizing Summary
- ☒ System Summary
- ☒ Component Sizing Summary
- ☒ Outdoor Air Summary
- ☒ Object Count Summary
- ☐ Component Cost Economics Summary

Tip: DesignBuilder v2025 doesn't provide specific options for the more recent EnergyPlus summary tables listed below, but these can still be generated by selecting the **All Summary** report option:

- Tariff Report
- Life Cycle Cost Report
- Heat Emissions Report
- Thermal Resilience Summary
- CO2 Resilience Summary
- Visual Resilience Summary

Details on the Summary Annual Reports

Annual Building Utility Performance Summary (sometimes called ABUPS)

The **Annual Building Utility Performance Summary** report often called ABUPS produces a report with an overall summary of the utility consumption of the building. It contains a number of sub tables that are each described below.

- **Site and Source Energy** – Indicates the total site and source energy use. For electricity the net electricity from the utility is used for the electric contribution. The site to source conversion factors are based on those entered by the user. These are entered in the Environmental Impact Factors and Fuel Factors data.
- **Building Area** – Shows the total floor area of the building and the conditioned floor area.
- **End Uses** – This shows the total use of electricity, natural gas, other fuels, purchased cooling, purchased heating and water for each major end-use category. The end-use categories are Heating,

Cooling, Interior Lighting, Exterior Lighting, Interior Equipment, Exterior Equipment, Fans, Pumps, Heat Rejection, Humidification, Heat Recovery, Hot Water, Refrigeration, and Generators. Not all fuels have corresponding end uses. The values in this sub-table are from output meters. To determine which components are attached to each end-use meter, consult the meter details output file (*.mtd). The source of the resource does not affect this table – the amount of electricity used for lights does not change if the electricity is from the utility or from an on-site generator. The district heating column also includes steam.

- **End Uses By Subcategory** – Shows a breakdown of the major end uses by user-defined category. If an end-use subcategory was not input for an object, it is automatically added to the General subcategory for that major end-use category.
- **Utility Use Per Floor Area** – These two sub-tables show the results from the end-uses table divided by the total floor area defined for the building and for the total conditioned floor area. Only three categories for end-uses are used for these sub-tables, lighting, HVAC and other. HVAC includes fans, pumps, heating, cooling, heat rejection, humidification, and domestic hot water heating. The district heating column also includes steam.
- **Electric Loads Satisfied** – Shows the different methods that electric loads are satisfied in the building. The values shown for on site power generation are: Fuel-Fired Power Generation, High Temperature Geothermal, Photovoltaic Power, and Wind Power. The flows to and from the electric utility are shown next and finally the total electricity used at the site is compared to the total generated on site plus the net amount from the electric utility. The percentages shown are based on the total electricity used by the end-uses.
- **On-Site Thermal Sources** – Shows the on-site thermal sources of energy such as Water-Side Heat Recovery, Air to Air Heat Recovery for Cooling, Air to Air Heat Recovery for Heating, High-Temperature Geothermal, Solar Water Thermal, Solar Air Thermal.
- **Water Loads Summary** – Shows the different methods the water loads were satisfied.

Input Verification and Results Summary (or IVRS)

The [Input Verification and Results Summary](#) report produces a report with several tables including:

- **General** which includes general information like the Program Version and Build, Weather, Latitude, Longitude, Elevation, Time Zone, North Axis Angle, and Hours Simulated.
- **Window-Wall Ratio** table for envelope which includes the wall area, the window area and the ratio of the two. These are computed for all walls and for walls that are oriented generally north, south, east and west. All walls are categorized into one of these four cardinal directions. This is computed for walls that have a tilt of 60 to 120 degrees.
- **Skylight-Roof Ratio** table for envelope which includes the roof area and the skylight area and the ratio of the two. This includes all surfaces with a tilt of less than 60 degrees.
- **Zone Summary** includes internal load summary for each zone including area, if conditioned, volume, multipliers, above ground gross wall area, underground gross wall area, window area, design lighting, design people, and design plug and process.
- **Hybrid Model: Internal Thermal Mass** table is only shown when hybrid model: internal thermal mass simulation. This includes Hybrid Modeling (Y/N) for internal mass and Temperature Capacitance Multiplier values for each zone.

Source Energy End Use Components Summary

The [Source Energy End Use Components Summary](#) report produces a report that includes three tables. These tables display source energy by fuel type that is calculated based on site to source energy factors specified by the user in the EnvironmentalImpactFactors and FuelFactors objects. The last two tables display the source energy in terms of area normalized metrics. Following is a description of each table:

- **Source Energy End Use Components** – This shows the total use of source electricity, source natural gas, source values of other fuels, source purchased cooling and purchased heating for each major end-use category. The end-use categories are Heating, Cooling, Interior Lighting, Exterior Lighting, Interior Equipment, Exterior Equipment, Fans, Pumps, Heat Rejection, Humidification, Heat Recovery, Hot Water, Refrigeration, and Generators. Not all fuels have corresponding end uses. The values in this sub-table are from output meters. To determine which components are attached to each end-use meter, consult the meter details output file (*.mtd). The source of the resource will affect this table – the amount of electricity used for lights will change if the electricity is from the utility or from an on-site generator. The district heating column also includes steam.

- **Source Energy End Use Components normalized by Conditioned Floor Area** – This table shows the total end uses in source energy normalized by conditioned floor area.
- **Source Energy End Use Components normalized by Total Floor Area** – This table shows the total end uses in source energy normalized by total floor area.

Climatic Data Summary

The **Climatic Data Summary** includes the main details on each of the design days including: maximum dry-bulb, daily temperature range, humidity value, humidity type, wind speed, and wind direction.

Envelope Summary

The **Envelope Summary** includes the following tables:

- **Opaque** which includes all opaque surfaces and includes the name of the construction, reflectance, U-Factor, gross area, azimuth, tilt, cardinal direction.
- **Fenestration** which includes all non-opaque surfaces and includes the name of the construction, areas (glass, frame, divider, single opening, multiplied openings), U-Factor, SHGC (the solar heat gain coefficient based on summer conditions), visible transmittance, conductance (frame, divider), indication of shade control, the name of the parent surface, azimuth, tilt, cardinal direction.

Surface Shadowing Summary

The **Surface Shadowing Summary** includes two tables. Note that surfaces starting with “Mir-” are automatically generated by EnergyPlus and are the mirror images of user entered surfaces.

- **Surfaces** (Walls, Roofs, etc) that may be Shadowed by Other Surfaces and includes the name of the surface and a list of surfaces that may possibly cast shadows on that named surface. The list of possible shadow casters does not necessarily mean that they do cast shadows during the simulation, only that their relative position makes it possible that shadows from a surface in the list may fall on the named surface.
- **Subsurfaces** (Windows and Doors) that may be Shadowed by Surfaces, includes the name of the subsurface such as a window or a door and a corresponding list of surfaces that may be casting shadows on the windows and doors.

Shading Summary

The **Shading Summary** includes the following tables:

- **Sunlit Fraction** which shows a list of windows and the fraction of the window that is sunlit for nine specific times of the year. The nine specific times include 9am, noon and 3pm on March 21, June 21, and December 21. These nine times were chosen to represent the range of sun angles. The simulation must include those times for the value to be included in the report.
- **Window control** includes the names of all windows that have a window shading control (see WindowShadingControl) and includes the name of the control, the type of shading, the shaded construction, the kind of control, and if glare control is used.

Lighting Summary

The **Lighting Summary** includes the following tables:

- **Interior Lighting** which includes the name of the lights object, the zone that it is used in, the lighting power density, zone area, total power, end use subcategory, schedule name, average hours per week, return air fraction, and whether the zone is conditioned.
- **Daylighting** which includes the names of the daylighting objects, the zone they are used in, the type of daylighting being used, the control type, the fraction of the lighting controlled, the total power of the lighting installed in the zone and the lighting power that is controlled by the daylighting object.
- **Exterior Lighting** which includes the name of the ExteriorLights object, the total watts described by the object, if the ExteriorLights uses an astronomical clock or just a schedule, the name of the

schedule used, and the average hours per week for the named schedule for the year. The effect of the astronomical clock does not get included in the averaged hours per week shown.

Equipment Summary

The **Equipment Summary** includes some details on the major HVAC equipment present. The report has seven parts.

- **Central Plant** includes details on chillers, boilers, and cooling towers including the capacity and efficiency. For Chiller:Electric:EIR and Chiller:Electric:ReformulatedEIR, IPLV at AHRI standard test conditions is reported.
- **Cooling Coils** includes the nominal total, sensible and latent capacities, the nominal sensible heat ratio, the nominal efficiency, nominal UA value, and nominal surface area for each cooling coil. These values are calculated by calling the cooling coil simulation routine with the rated inlet conditions: inlet air dry bulb temperature = 26.67C, inlet air wet bulb temperature = 19.44C, inlet chilled water temperature = 6.67C.
- **DX Cooling Coils** summarizes the Standard Rating (Net) Cooling Capacity, SEER, EER and IEER values at AHRI standard test. Currently, these values are only reported for coil type = Coil:Cooling:DX:SingleSpeed and Coil:Cooling:DX:MultiSpeed with condenser type = AirCooled. However, the EER value is not reported for the multi-speed DX cooling coil. There are two SEER values reported: SEER and SEER Default. The SEER value is calculated using user specified Part Load Factor (PLF) curve used for energy performance calculation and the SEER Default value is calculated using the AHRI Standard 210/240-2008 default PLF curve and cooling coefficient of degradation value of 0.25.
- **DX Heating Coils** summarizes the High Temperature Heating Standard (Net) Rating Capacity, Low Temperature Heating Standard (Net) Rating Capacity and Heating Seasonal Performance Factor (HSPF) values at AHRI standard test. Currently, these values are only reported for coil type = Coil:Heating:DX:SingleSpeed.
- **Heating Coils** includes the nominal capacity and efficiency for each heating coil. The capacity is calculated by calling the heating coil simulation routine at the rated inlet conditions: inlet air dry bulb temperature = 16.6C, inlet relative humidity = 50%, inlet hot water temperature = 82.2C.
- **Fan** includes the type of fan, the total efficiency, delta pressure, max flow rate, motor heat in air fraction, and end use.
- **Pumps** includes the type of pump, control type, head pressure, electric power, and motor efficiency for each pump.
- **Service Water Heating** includes the type of water heater, the storage volume, input, thermal efficiency, recovery efficiency, and energy factor.

HVAC Sizing Summary

The **HVAC Sizing Summary** includes the following tables:

- **Zone Cooling** which includes the following columns for each zone: the calculated design load, the user specified design load, the calculated design air flow, the user specified design air flow, the name of the sizing period, the time of the peak load during the sizing period, the temperature at the time of the peak load during the sizing period, and the humidity ratio at the time of the peak load during the sizing period used.
- **Zone Heating** which includes the following columns for each zone: the calculated design load, the user specified design load, the calculated design air flow, the user specified design air flow, the name of the sizing period, the time of the peak load during the sizing period, the temperature at the time of the peak load during the sizing period, and the humidity ratio at the time of the peak load during the sizing period used.
- **System Design Air Flow Rates** which includes the following columns for each air loop: the calculated cooling air flow rate, the user specified air flow rate for cooling, the calculated heating air flow rate, the user specified air flow rate for heating.
- **Coil Sizing Summary** which includes the following types of information for each coil: coil type, HVAC location, capacity and flow rates at the zone peak, system peak, and final values, design day name and time for the zone peak and system peak, zone conditions at the system peak, etc. This table is a subset of the Coil Sizing Details table. See the Output Details and Examples document for a complete description of each column.

Note: values listed as “calculated” are the unaltered result of the zone or system sizing calculations, using the design sizing period weather and schedules specified in the input. Values listed as “user specified” are either the calculated values modified by global or zone sizing factors or values specified with the flow/zone or flow/system design air flow method.

Component Sizing Summary

The **Component Sizing Summary** includes different tables depending on the kinds of HVAC components that exist in the input file. A table is shown for each type of HVAC component that is sized. The table lists the objects of that type of component that appear in the input file and one or more parameters related to that component. For example, the AirTerminal:SingleDuct:VAV:Reheat component creates a table showing the maximum air flow rate from the sizing calculations and the maximum reheat water flow rate. Another example is the Fan:VariableVolume object which shows a table with both the maximum and minimum flow rates for each fan based on the results from the sizing calculations.

Coil Sizing Details

The **Coil Sizing Details** table includes the following information for each coil: coil type, HVAC location, capacity and flow rates at the zone peak, system peak, and final values, design day name and time for the zone peak and system peak, zone conditions at the system peak, coil entering and leaving conditions, supply fan information, related plant sizing information, etc. See the Output Details and Examples document for a complete description of each column.

Outdoor Air Summary

The **Outdoor Air Summary** report includes the following tables:

- **Average Outside Air During Occupied Hours** table shows for each zone the average and nominal number of occupants, the zone volume, the average air change rate based on mechanical ventilation, infiltration and simple ventilation during occupied hours.
- **Minimum Outside Air During Occupied Hours** table shows for each zone the average and nominal number of occupants, the zone volume, the minimum air change rate based on mechanical ventilation, infiltration and simple ventilation during occupied hours.

System Summary

The **System Summary Report** includes the following tables:

- **Economizer** which includes the following columns for each Controller:OutdoorAir object: the high limit shutoff control, the minimum outdoor air flow, the maximum outdoor air flow, if the return air temperature has a control limit, if the return air has an enthalpy limit, the outdoor air temperature limit, and the outdoor air enthalpy limit.
- **Demand Controlled Ventilation** table is for each Controller:MechanicalVentilation object and shows the name, the nominal outdoor air per person and the nominal outdoor air per zone area.
- **Time Not Comfortable Based on Simple ASHRAE 55-2004** table shows how many hours that the space is not comfortable for each zone under the criteria of assuming winter clothes, summer clothes or both summer and winter clothes. See the People object for more information about this thermal comfort calculation.
- **Time Setpoint is Not Met** table shows how many hours the space is more than 0.2C from the setpoint during heating and during cooling. The last two columns indicate those hours that the setpoint is not met while the space is occupied.

Adaptive Comfort Summary

The **Adaptive Comfort Summary** report tabulates the sum of occupied hours not meeting adaptive comfort acceptability limits for each relevant People object (People objects for which adaptive comfort calculations are requested). These acceptability limits include ASHRAE Std. 55 80%, ASHRAE Std. 55 90%, CEN-15251 Category I, CEN-15251 Category II, and CEN-15251 Category III.

Sensible Heat Gain Summary

The **Sensible Heat Gain Summary** provides results for each zone and the overall building for some of the major heat gain components. The first four columns show the loads satisfied by sensible air heating and cooling as well as radiant heating and cooling surfaces in the zone. The heat gains from people, lighting, equipment, windows, interzone air flow, and infiltration are shown when adding heat to the zone and separately when removing heat from the zone (for applicable components). Finally the balance is shown as “Opaque Surface Conduction and Other Heat Addition” and “Opaque Surface Conduction and Other Heat Removal” which is a term indicating the affect of the walls, floors and ceilings/roof to the zone as well as the impact of the delay between heat gains/losses and loads on the HVAC equipment serving the zone. The following shows each output variable that is used for each column. For each timestep in the simulation, positive values are shown as additions and negative values are shown as removal for most variables.

HVAC Input Sensible Air Heating

- Zone Air Heat Balance System Air Transfer Rate
- Zone Air Heat Balance System Convective Heat Gain Rate

HVAC Input Sensible Air Cooling

- Zone Air Heat Balance System Air Transfer Rate
- Zone Air Heat Balance System Convective Heat Gain Rate

HVAC Input Heated Surface Heating

- Zone Radiant HVAC Heating Energy
- Zone Ventilated Slab Radiant Heating Energy

HVAC Input Cooled Surface Cooling

- Zone Radiant HVAC Cooling Energy
- Zone Ventilated Slab Radiant Cooling Energy

People Sensible Heat Addition

- Zone People Sensible Heating Energy

Lights Sensible Heat Addition

- Zone Lights Total Heating Energy

Equipment Sensible Heat Addition & Equipment Sensible Heat Removal

- Zone Electric Equipment Radiant Heating Energy
- Zone Gas Equipment Radiant Heating Energy
- Zone Steam Equipment Radiant Heating Energy
- Zone Hot Water Equipment Radiant Heating Energy
- Zone Other Equipment Radiant Heating Energy
- Zone Electric Equipment Convective Heating Energy
- Zone Gas Equipment Convective Heating Energy
- Zone Steam Equipment Convective Heating Energy
- Zone Hot Water Equipment Convective Heating Energy
- Zone Other Equipment Convective Heating Energy

Window Heat Addition & Window Heat Removal

- Zone Windows Total Heat Gain Energy

Interzone Air Transfer Heat Addition & Interzone Air Transfer Heat Removal

- Zone Air Heat Balance Interzone Air Transfer Rate

Infiltration Heat Addition & Infiltration Heat Removal

- Zone Air Heat Balance Outdoor Air Transfer Rate

The Opaque Surface Conduction and Other Heat Addition and Opaque Surface Conduction and Other Heat Removal columns are also calculated on a timestep basis as the negative value of the other removal and gain columns so that the total for the timestep sums to zero. These columns are derived strictly from the other columns.

Component Load Summary

Three different component load summary reports can be generated:

- Zone Component Loads Summary
- AirLoop Component Loads Summary
- Facility Component Loads Summary

The AirLoop and Facility level reports are generally aggregations of the results reported in the Zone Component Loads Summary report. The Component Loads Summary reports provide an estimate of the heating and cooling peak loads for each zone, airloop or the entire facility broken down into various components. These reports may help determine which components of the load have the largest impact for the heating and cooling peak design conditions. When specified, the Zone Component Loads Summary report is created for each zone that is conditioned. Similarly, the AirLoop Component Load Summary is generated for each AirLoop and a single Facility Component Loads Summary is generated for the entire facility. The difference between the peak design sensible load and the estimated instant + delayed sensible load (as shown in the Peak Conditions subtable) is an indication of how consistent the overall total estimate may be to the computed total peak loads for the zone. When the report is called the zone sizing calculations are repeated twice so this may result in longer simulation times. The keys used to obtain these reports are ZoneComponentLoadSummary, AirLoopComponentLoadSummary, and FacilityComponentLoadSummary. Since including this report may increase the simulation time, new key options have been added that will display all reports but the Zone Component Load Summary those keys used are AllSummaryButZoneComponentLoad and AllSummaryAndMonthlyButZoneComponentLoad.

The report has six parts:

1. Estimated Cooling Peak Load Components

Contains the sensible-instant, sensible-delay, sensible-return air, latent, total and %grand total for people, lights, equipment, refrigeration, water use equipment, HVAC equipment loads, power generation equipment, infiltration, zone ventilation, interzone mixing, roof, interzone ceiling, other roof, exterior wall, interzone wall, ground contact wall, other wall, exterior floor, interzone floor, ground contact floor, other floor, fenestration conduction, fenestration solar, opaque door. The values in the sensible-delay column are estimated using a procedure shown in the Engineering Reference. Also shown are the related areas for each type of component. For People and Lights the floor area is shown but for walls, fenestration. and other surfaces, the area of that surface is used.

2. Cooling Peak Conditions

Contains the time of the peak load and the outside dry bulb and wet bulb temperatures as well as the outside humidity ratio for that time. It also shows the zone temperature and relative humidity and humidity ratio for that time. The airflow and outside airflow as well as the supply air temperature are shown. Also the sensible peak load accounting for the sizing factor is shown along with the difference due to the sizing factor.

3. Engineering Checks for Cooling

This table shows some ratios that may be handy in evaluating the results. It contains the percentage of outside air, the airflow per floor area, the airflow per capacity, the floor area per capacity and the capacity per floor area. It also shows the number of people.

4. Estimated Heating Peak Load Components

Contains the sensible-instant, sensible-delay, sensible-return air, latent, total and %grand total for people, lights, equipment, refrigeration, water use equipment, HVAC equipment loads, power generation equipment, infiltration, zone ventilation, interzone mixing, roof, interzone ceiling, other roof, exterior wall, interzone wall, ground contact wall, other wall, exterior floor, interzone floor, ground contact floor, other floor, fenestration conduction, fenestration solar, opaque door. The values in the sensible-day column are estimated using a procedure shown in the Engineering Reference. Also shown are the related areas for each type of

component. For People and Lights the floor area is shown but for walls, fenestration. and other surfaces, the area of that surface is used.

5. Heating Peak Conditions

Contains the time of the peak load and the outside dry bulb and wet bulb temperatures as well as the outside humidity ratio for that time. It also shows the zone temperature and the relative humidity and humidity ratio for that time. The airflow and outside airflow as well as the supply air temperature are shown. Also the sensible peak load accounting for the sizing factor is shown along with the difference due to the sizing factor

6. Engineering Checks for Heating

This table shows some ratios that may be handy in evaluating the results. It contains the percentage of outside air, the airflow per floor area, the airflow per capacity, the floor area per capacity and the capacity per floor area. It also shows the number of people.

The "Air Loop Component Summary" tables also include tables that indicate the zones included in the aggregated results for heating and cooling.

If the time of the peak load for each Zone for cooling exactly matches the time of the peak load for the AirLoop or Facility than the Estimated Cooling Peak Load Components will represent a sum of the values from the corresponding zones. Likewise the Estimated Heating Peak Load Components will add up for the AirLoop or Facility if the times of the heating peaks exactly match. This is not necessarily the case for Peak Conditions or the Engineering Checks tables. Since the sizing of Air loops is based on the system sizing they usually will be different than the sum of the corresponding zones.

Note: generating the **Component Load Summary** report will cause simulations to take longer.

Standard 62.1 Summary

The **Standard 62.1 Summary** produces a report that is consistent with many of the outputs needed when doing calculations consistent with ASHRAE Standard 62.1-2010. The report is generated when sizing calculations are specified. The abbreviations used in the report are consistent with the abbreviations used in Appendix A4 of the Standard. The following tables are part of the report:

- **System Ventilation Requirements for Cooling** containing: Sum of Zone Primary Air Flow - Vpz-sum, System Population - Ps, Sum of Zone Population - Pz-sum, Occupant Diversity - D, Uncorrected Outdoor Air Intake Airflow - Vou, System Primary Airflow - Vps, Average Outdoor Air Fraction - Xs, System Ventilation Efficiency - Ev, Outdoor Air Intake Flow - Vot, Percent Outdoor Air - %OA.
- **System Ventilation Requirements for Heating** containing: Sum of Zone Primary Air Flow - Vpz-sum, System Population - Ps, Sum of Zone Population - Pz-sum, Occupant Diversity - D, Uncorrected Outdoor Air Intake Airflow - Vou, System Primary Airflow - Vps, Average Outdoor Air Fraction - Xs, System Ventilation Efficiency - Ev, Outdoor Air Intake Flow - Vot, Percent Outdoor Air - %OA.
- **Zone Ventilation Parameters** containing: AirLoop Name, People Outdoor Air Rate - Rp, Zone Population - Pz, Area Outdoor Air Rate - Ra, Zone Floor Area - Az, Breathing Zone Outdoor Airflow - Vbz, Cooling Zone Air Distribution Effectiveness - Ez-clg, Cooling Zone Outdoor Airflow - Voz-clg, Heating Zone Air Distribution Effectiveness - Ez-htg, Heating Zone Outdoor Airflow - Voz-htg.
- **System Ventilation Parameters** containing: People Outdoor Air Rate - Rp, Sum of Zone Population - Pz-sum, Area Outdoor Air Rate - Ra, Sum of Zone Floor Area - Az-sum, Breathing Zone Outdoor Airflow - Vbz, Cooling Zone Outdoor Airflow - Voz-clg, Heating Zone Outdoor Airflow - Voz-htg.
- **Zone Ventilation Calculations** for Cooling Design containing: AirLoop Name, Box Type, Zone Primary Airflow - Vpz, Zone Discharge Airflow - Vdz, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Primary Outdoor Air Fraction - Zpz, Primary Air Fraction - Ep, Secondary Recirculation Fraction- Er, Supply Air Fraction- Fa, Mixed Air Fraction - Fb, Outdoor Air Fraction - Fc, Zone Ventilation Efficiency - Evz.
- **System Ventilation Calculations** for Cooling Design containing: Sum of Zone Primary Airflow - Vpz-sum, System Primary Airflow - Vps, Sum of Zone Discharge Airflow - Vdz-sum, Minimum Zone

Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Zone Ventilation Efficiency - Evz-min.

- **Zone Ventilation Calculations for Heating Design** containing: AirLoop Name, Box Type, Zone Primary Airflow – Vpz, Zone Discharge Airflow – Vdz, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Primary Outdoor Air Fraction – Zpz, Primary Air Fraction – Ep, Secondary Recirculation Fraction- Er, Supply Air Fraction- Fa, Mixed Air Fraction – Fb, Outdoor Air Fraction – Fc, Zone Ventilation Efficiency – Evz.
- **System Ventilation Calculations for Heating Design** containing: Sum of Zone Primary Airflow - Vpz-sum, System Primary Airflow – Vps, Sum of Zone Discharge Airflow - Vdz-sum, Minimum Zone Primary Airflow - Vpz-min, Zone Outdoor Airflow Cooling - Voz-clg, Zone Ventilation Efficiency - Evz-min.

Note: The **Standard ASHRAE 62.1 Summary** report is available for generation when using Detailed HVAC or Simple HVAC with the default **1-EnergyPlus** [Simple HVAC autosizing method](#).

Energy Meters Summary

The **Energy Meters Summary** (which is a slight misnomer as some meters may not be strictly energy) provides the annual period (runperiod) results for each meter (reference the meter data dictionary file (.mdd) and/or the meter details file (.mtd). The results are broken out by fuel type (resource type) in this report.

Initialization Summary

The **Initialization Summary** provides the information shown in the initialization output file (.eio) but in a tabular format. Because of this, some of the outputs may be difficult to understand without referencing the documentation on the .eio file that is located in the Output Details and Examples document.

LEED Summary

The **LEED Summary** report provides many of the simulation results required for certification of Energy and Atmosphere Credit 1 Optimized Energy Performance according to the LEED Green Building Rating System™.

Tariff Report

The **Tariff Report** provides the results of the [Tariff Analysis](#) simulation. The report consists of a summary table, categories, charges, ratchets, qualifies, native variables, other variables and computation.

Economic Result Summary

The **Economic Result Summary** provides a summary of the Tariff Reports. The report consists of annual costs and costs for each tariff considered and why it was or was not selected.

Component Cost Economics Summary

The **Component Cost Economics Summary** provides the construction cost estimate summary for the project. The costs are broken into eight categories and the reference building costs are provided as a comparison. A second table is also produced that provides line item details with one line for every line item object.

Life Cycle Cost Report

The **Life Cycle Cost Report** provides a summary of the information from the Life Cycle Cost calculations. This report shows the costs and the timing of costs, often called “cash flows,” along with the present value in several different tables. The tabular results show the present value of all current and future costs.

Heat Emissions Report

The **Heat Emissions Report** provides a summary of the building heat emission to the ambient air. This report shows the heat emissions from buildings by surface heat transfer, zone exhaust, zone relief, HVAC relief, HVAC heat rejection, as well as the total emissions.

Thermal Resilience Summary

The **Thermal Resilience Summary** provides the indoor thermal resilience metrics unmet hours and occupant-hours summary report. This includes the Heat Index Hours and OccupantHours, Humidex Hours and OccupantHours, Heating SET Hours and Cooling SET Hours reports. The Heating SET Hours and Cooling SET Hours are reported only when the Pierce model are chosen for thermal comfort calculations. To activate the Pierce SET calculation in EnergyPlus, users need to define the People and chose Pierce as the thermal comfort method.

CO2 Resilience Summary

The **CO2 Resilience Summary** provides the indoor CO2 level unmet hours and occupant-hours summary report for resilience. This includes the CO2 Level Hours and OccupantHours reports. To activate the CO2 concentration calculation in EnergyPlus, the [Simulate CO2 concentration](#) site level option must be selected and a schedule of outdoor air CO2 levels selected.

Visual Resilience Summary

The **Visual Resilience Summary** provides the indoor illuminance level unmet hours and occupant-hours summary report for resilience. This includes the Illuminance Level Hours and OccupantHours reports. To activate the indoor illuminance calculation in EnergyPlus, it is necessary to select the "Lighting control" on page 461 option.

Summary Monthly Reports

Select from a range of summary monthly reports:

Summary Monthly Reports

- ☐ All Monthly
- ☐ Zone Cooling Summary Monthly
- ☐ Zone Heating Summary Monthly
- ☐ Zone Electric Summary Monthly
- ☐ Space Gains Monthly
- ☐ Peak Space Gains Monthly
- ☐ Space Gain Components At Cooling Peak Monthly
- ☐ Energy Consumption Electricity Natural Gas Monthly
- ☐ Energy Consumption Electricity Generated Propane Monthly
- ☐ Energy Consumption Diesel Fuel Oil Monthly
- ☐ Energy Consumption District Heating Cooling Monthly
- ☐ Energy Consumption Coal Gasoline Monthly
- ☐ End Use Energy Consumption Electricity Monthly
- ☐ End Use Energy Consumption Natural Gas Monthly
- ☐ End Use Energy Consumption Diesel Monthly
- ☐ End Use Energy Consumption FuelOil Monthly
- ☐ End Use Energy Consumption Coal Monthly
- ☐ End Use Energy Consumption Propane Monthly
- ☐ End Use Energy Consumption Gasoline Monthly
- ☒ End Use Energy Consumption Other Fuels Monthly
- ☐ Peak Energy End Use Electricity Part1 Monthly
- ☐ Peak Energy End Use Electricity Part2 Monthly
- ☐ Electric Components Of Peak Demand Monthly
- ☐ Peak Energy End Use Natural Gas Monthly
- ☐ Peak Energy End Use Diesel Monthly
- ☐ Peak Energy End Use Fuel Oil Monthly
- ☐ Peak Energy End Use Coal Monthly
- ☐ Peak Energy End Use Propane Monthly
- ☐ Peak Energy End Use Gasoline Monthly
- ☒ Peak Energy End Use Other Fuels Monthly
- ☐ Setpoints Not Met With Temperatures Monthly
- ☐ Comfort Report Simple 55 Monthly
- ☐ Unglazed Transpired Solar Collector Summary Monthly
- ☐ Occupant Comfort Data Summary Monthly
- ☐ Chiller Report Monthly
- ☐ Tower Report Monthly
- ☐ Boiler Report Monthly
- ☐ DX Report Monthly
- ☐ Window Report Monthly
- ☐ Window Energy Report Monthly
- ☐ Window Zone Summary Monthly
- ☐ Window Energy Zone Summary Monthly
- ☐ Average Outdoor Conditions Monthly
- ☐ Outdoor Conditions Maximum Dry Bulb Monthly
- ☐ Outdoor Conditions Minimum Dry Bulb Monthly
- ☐ Outdoor Conditions Maximum Wet Bulb Monthly
- ☐ Outdoor Conditions Maximum Dew Point Monthly
- ☐ Outdoor Ground Conditions Monthly
- ☐ Window AC Report Monthly
- ☐ Water Heater Report Monthly
- ☐ Generator Report Monthly
- ☐ Daylighting Report Monthly
- ☐ Coil Report Monthly
- ☐ Plant Loop Demand Report Monthly
- ☐ Fan Report Monthly
- ☐ Pump Report Monthly
- ☐ Cond Loop Demand Report Monthly
- ☐ Zone Temperature Oscillation Report Monthly
- ☐ Air Loop System Energy And Water Use Monthly
- ☐ Air Loop System Component Loads Monthly
- ☐ Air Loop System Component Energy Use Monthly
- ☐ Mechanical Ventilation Loads Monthly

Information about the various monthly summary reporting options can be found in the [EnergyPlus Output Details And Examples](#) documentation.

Miscellaneous Outputs

HVAC System temperatures

Check this option if you would like DesignBuilder to generate temperature output for each air and water node in the HVAC system. This output data cannot be displayed directly in DesignBuilder but can be viewed in the separate [Results Viewer](#) application.

HVAC System mass flow rates

Check this option if you would like DesignBuilder to generate mass flow output for each air and water node in the HVAC system. This output data cannot be displayed directly in DesignBuilder but can be viewed in the separate [Results Viewer](#) application.

HVAC System humidity ratios

Check this option if you would like DesignBuilder to generate humidity ratio output for each air node in the HVAC system. This output data cannot be displayed directly in DesignBuilder but can be viewed in the separate [Results Viewer](#) application.

SQLite output

Output from EnergyPlus can be written to SQLite files for manipulation outside of DesignBuilder. When this option is selected all of the predefined database tables as well as time series related data is included in the SQLite output file [eplusout.sql](#). The description for SQLite outputs is described fully in the EnergyPlus Output Details document.

Tip: SQLite outputs can be processed by the DesignBuilder [db_eplusout_reader](#) Python library for generating custom reports and optimisation KPIs.

DXF model output

This option causes EnergyPlus to generate a DXF file summarising the building model. The file is called [eplusout.dxf](#) and it is stored in the [EnergyPlus folder](#). This option can be useful for checking that the correct geometry has been generated.

Construction and surface details

This option causes EnergyPlus to generate a comma separated value file summarising the building model. The file is called [eplusout.eio](#) and it is stored in the [EnergyPlus folder](#).

RDD/EDD File

Select this option if you would like RDD and EDD files to be generated by the simulation for use in future simulations. The RDD file provides a list of all the EnergyPlus output reports that could be selected for the current model, while the EDD files contains a list of all the available actuators.

Time Setpoints Not Met Tolerances

This section of input allows more control over the reporting of “Time setpoint not met” hours.

Tolerance for time heating setpoint not met

This data defines the maximum difference from the heating setpoint when reporting the time heating setpoints are not met. If the zone temperature is below the heating setpoint by more than this value, the following report variables will increment as appropriate:

- Time Heating Setpoint Not Met
- Time Heating Setpoint Not Met While Occupied

It also impacts table report "Annual Building Utility Performance Summary" sub-table "Comfort and Setpoint Not Met Summary".

Tolerance for time cooling setpoint not met

This data defines the maximum difference from the cooling setpoint when reporting the time cooling setpoints are not met. If the zone temperature is above the cooling setpoint by more than this value, the following report variables will increment as appropriate:

- Time Cooling Setpoint Not Met
- Time Cooling Setpoint Not Met While Occupied

It also impacts table report "Annual Building Utility Performance Summary" sub-table "Comfort and Setpoint Not Met Summary".

Simulation Manager

The DesignBuilder Simulation Manager (SM) allows you to run and control multiple parallel simulations and access the results. Some advantages of using the Simulation Manager are:

- It allows you to run multiple simulations in parallel on your local machine, on a local network machine or on the cloud using the [JESS online](#) system.
- You can continue to work on your model or close the DesignBuilder application down while simulations run.
- A single user interface is used for displaying and managing all of your simulations and see their progress and status.
- Access the model and continue with your DesignBuilder work while the simulations run.
- It provides a convenient way to store, access and compare simulation results, either via the [Results Viewer](#) or by loading results into the main DesignBuilder application.

Note: [Design optimisation](#), [ASHRAE 90.1 Appendix G PRM](#) and UK DSM simulations are always run using the Simulation Manager to take advantage of the parallel simulation capability, giving you the fastest possible simulation times.

See also the [Simulation Manager and Results viewer](#) tutorial

Use Simulation Manager

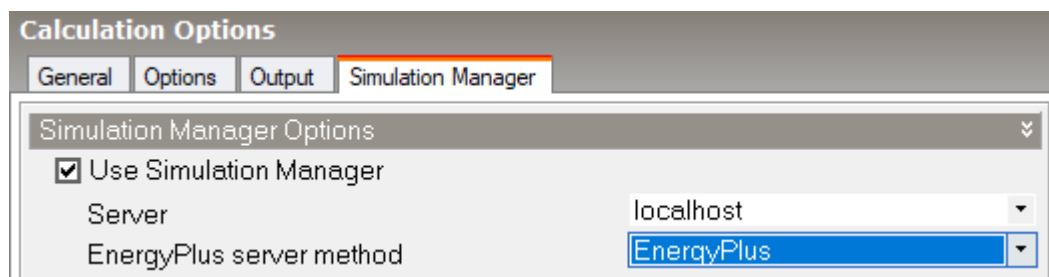
To run an EnergyPlus simulation using the Simulation Manager go to the **Simulation Manager** tab on the Simulation [Calculation options](#) dialog and check the **Use Simulation Manager** checkbox. This shows further data items described below.

The Simulation Manager settings can also be accessed from the Calculation options dialog for Parametric Analysis, Optimisation and UA/SA.

Note: Use of the Simulation Manager is not optional for Parametric Analysis, Optimisation and UA/SA simulations.

Server

By default the **localhost** server is selected, which is the option to use the Simulation Manager installed on the local machine. Choosing this allows you to run and control simulations from DesignBuilder, continue to work on the model or another model or even close DesignBuilder down and the simulations will continue to run unaffected. If the local machine is put into hibernate or sleep then the simulations will be suspended too and will resume when the machine is switched on again. If the machine is shut down while simulations are running then those simulations will be lost but should start again (from the beginning) when the machine is started up again.



Another option is to select a server on a different machine on the network from the drop down list. To enable other servers to appear on the list for selection you should first follow the steps on the [Install Simulation Server](#) page. When running simulations on a server, as well as the advantages already mentioned above for the localhost, you can also:

- Close your machine down and the simulations will continue to run unaffected. You must of course be connected to the server over the network to start the simulation and to retrieve results, but the connection can be disconnected while the simulation runs.
- Frees the machine running DesignBuilder giving it more processor and memory resources for the input file generation process, which further reduces the overall wait time for results.

Note: If the selected server is not available for any reason, DesignBuilder will revert to using the first server on the list of active servers, typically **localhost**. This can happen when, for example, a simulation model set up by modeller A to run on a dedicated simulation server is sent to a modeller B who does not have access to that server.

Tip: With a good internet connection at both ends and a VPN service you can run simulations on an office network server from your laptop when away from the office. Note that in this case there will be a transmission overhead as input files are uploaded to the server and results are downloaded. This is of course more of an issue for larger models, especially when large datasets are generated.

Dedicated Simulation Servers

Some advantages of using a dedicated simulation server for your EnergyPlus simulations are:

- You can close the computer running DesignBuilder down once the job is submitted and the simulation(s) will continue to run. This can be especially useful for long simulations. Results can be retrieved later by opening the Simulation Manager dialog.
- If you choose a powerful multi-core server with plenty of memory you can run many simulations simultaneously.
- The processor of the computer running DesignBuilder doesn't get overloaded while you are working on the model.

Tip: You can obtain a high-performance simulation server configured and ready to run from [Ensims](#).

EnergyPlus server method

There are 2 options:

- **EnergyPlus** - the default option where the Simulation Manager runs EnergyPlus directly. When this option is selected you will see a progress bar for each simulation in the SM dialog.
- **JEPlus** - select this option if you have an account on the [JESS online service](#) and would like the Simulation Manager to run simulations via JEPlus and the JESS server. When this option is selected, an "indeterminate" progress bar is used for each simulation in the SM dialog because the SM does not know the actual percentage progress of the simulation on the JESS server until the simulation is complete.

Accelerate

When the **JEPlus** EnergyPlus server method is selected, you can choose to speed up annual simulations using a special build of EnergyPlus where all 12 months are simulated in parallel at the same time. Running "split simulations" in this way can save a lot of time for large models. Time savings can be up to 10X for large models, but are usually smaller than this due to the overheads involved in each simulation (preconditioning, autosizing, copying files etc). For small models you are unlikely to see any worthwhile benefits in using this option.

Important Warnings

1. The accelerated version of EnergyPlus will produce slightly different results compared with simulations run without the Accelerate option selected. This is because each month is simulated separately and therefore the building thermal mass does not include an accurate history of heat stored from the previous month. **When the Accelerate option is not selected the results produced by JEPlus match exactly with those generated when using the EnergyPlus server method.**
2. Particular cases where the Accelerate option should not be used include:
 - a. Buildings with high levels of thermal mass.
 - b. Models that include [ground heat exchangers](#) where the evolution of ground temperatures must be retained from previous months in the simulation.
 - c. Models that use ground domain objects, for similar reasons to above.
 - d. Simulations where adaptive comfort outputs are required may be affected.
3. A reduced selection of summary outputs are generated for the accelerated version and are formatted differently.
4. [TM52](#) and [TM59](#) calculations are not supported by the accelerated option.

Version Of EnergyPlus

The version of EnergyPlus to be used in all simulations is set on the [EnergyPlus tab](#) of the Program options dialog. The Simulation Manager will choose the appropriate version of EnergyPlus given the version number selected.

See also:

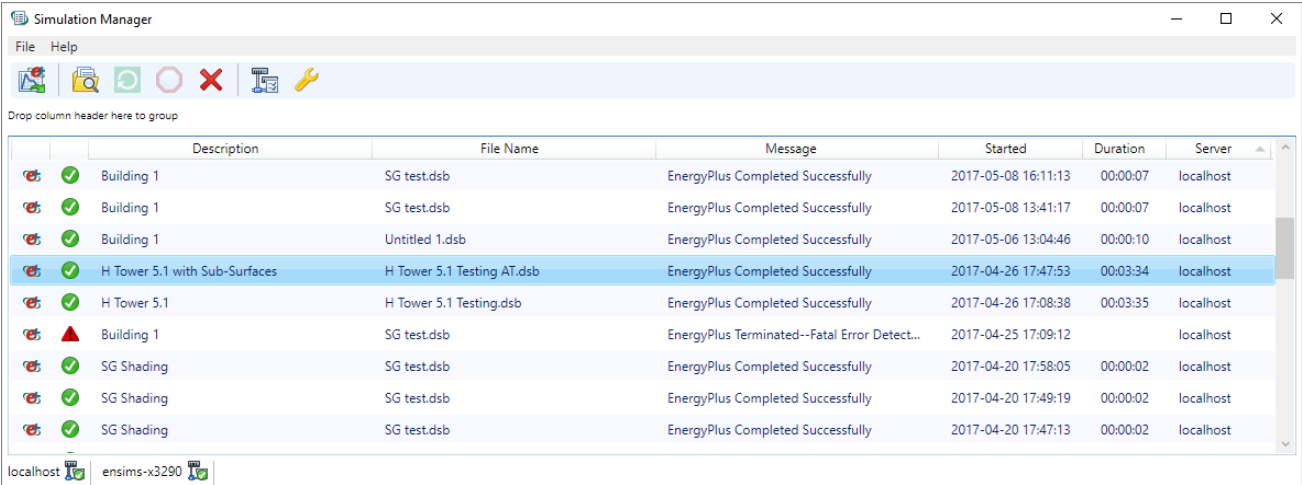
- "Simulation Manager Dialog" on page 1612
- "Installing the Simulation Manager on a Server" on page 1616

Simulation Manager Dialog

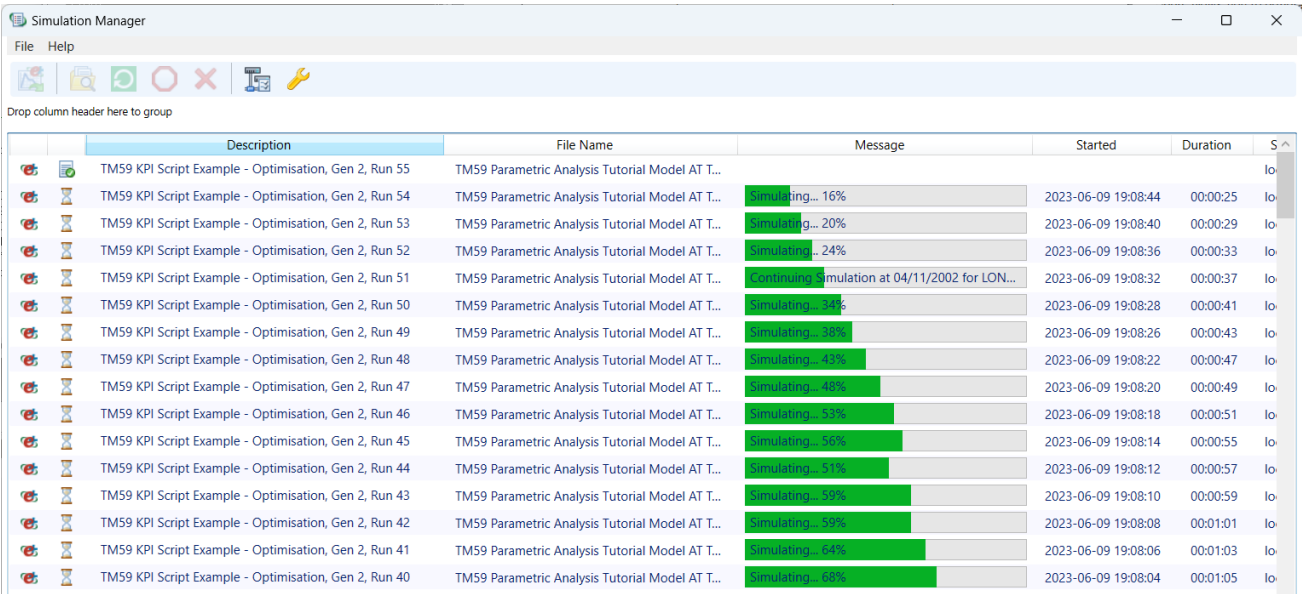
The Simulation Manager (SM) dialog provides access to simulations and information on their progress as they proceed as well as to results that have previously been generated. You can use it load results either to the standalone [Results Viewer](#) or to the inbuilt DesignBuilder results display screen.

The SM dialog can be accessed from within DesignBuilder on the Simulation screen using the  toolbar icon. Alternatively you can open the Simulation Manager dialog as a standalone application by selecting the **Start > DesignBuilder > Simulation Manager** option. This allows simulation results to be accessed independently of the main DesignBuilder application which can be especially useful when running parametric studies such as optimisations.

The Simulation Manager dialog looks something like the image below.



Or when a batch of parametric simulations is running:



Behaviour is different depending on whether the application is loaded from within DesignBuilder and if so whether you are at building level or below. In the latter case, only the simulation results for the current building are displayed. Alternatively if you start the SM dialog from outside DesignBuilder or from site level with a model open or when no model is open from the Opening screen then all simulations (sometimes referred to as "jobs") are displayed.

Tip: You can close the Simulation Manager dialog while simulations are running without affecting the simulations. To continue to work on your model or run other calculations while simulations proceed simply close the SM dialog. You can even close DesignBuilder and the simulations will still continue to run. Returning to the Simulation screen, you can re-open the dialog to check progress and results using the

Open Simulation Manager dialog toolbar icon .



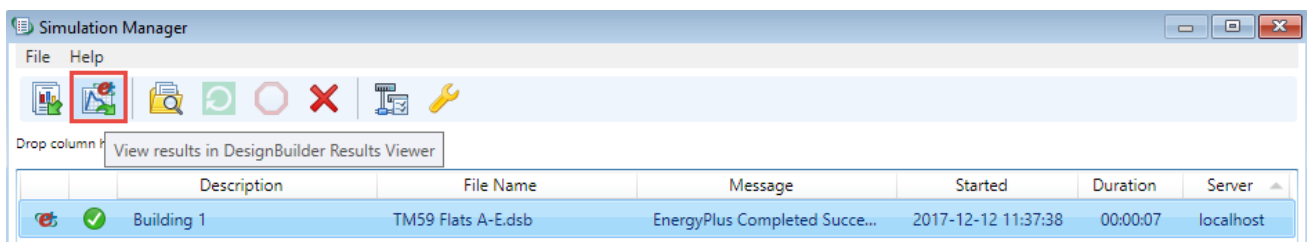
Load results to DesignBuilder

If you opened the SM dialog from DesignBuilder and you are at building level or below in the model then the **Load results to DesignBuilder** icon will be available. Clicking on this causes the results for the currently selected simulation to be loaded to the DesignBuilder Simulation screen. You must go the Simulation screen to actually view the results.



View results in DesignBuilder Results viewer

Clicking on this icon opens the results for the selected simulation to the [Results Viewer](#) application including data stored in both the EnergyPlus tabular summary html and eso output files.

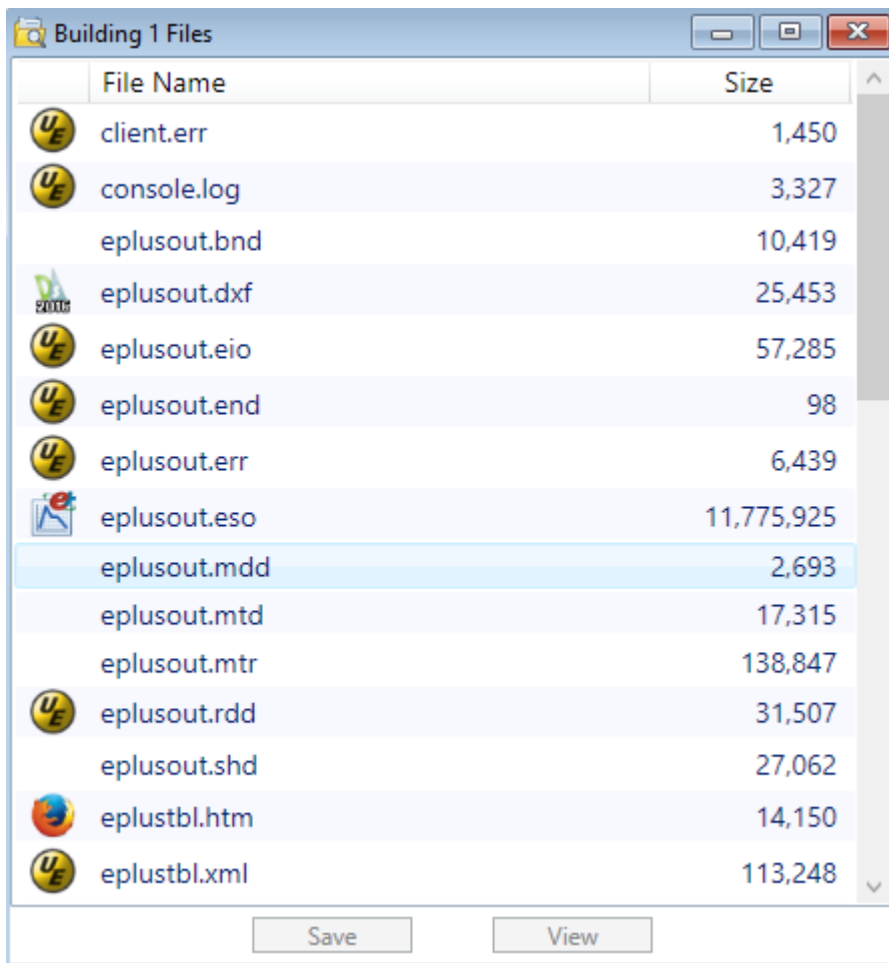


Tip: To load multiple result sets to the same Results Viewer session you should first select multiple simulation jobs before clicking on the **View results in DesignBuilder Results viewer** toolbar icon. This allows you to view results from a number of simulations on the same graph.



View files associated with current simulation

Selecting this command opens a list of all of the files associated with the simulation currently selected including both input and outputs files.



File Name	Size
 client.err	1,450
 console.log	3,327
eplusout.bnd	10,419
 eplusout.dxf	25,453
 eplusout.eio	57,285
 eplusout.end	98
 eplusout.err	6,439
 eplusout.eso	11,775,925
eplusout.mdd	2,693
eplusout.mtd	17,315
eplusout.mtr	138,847
 eplusout.rdd	31,507
eplusout.shd	27,062
 eplustbl.htm	14,150
 eplustbl.xml	113,248

Save View

From here you can save a file to disk or open it using the application currently associated with the file extension.

This option is only available for simulations that have completed.



Stop current simulation

If the currently selected simulation is still running then it can be terminated using this command. In this case simulation results will not be available.



Delete selected simulations

To delete one or more simulation result sets select the jobs for deletion using <Ctrl> mouse click and press the **Delete selected simulations** toolbar icon to remove all data for the selected simulation jobs including any input and output files.



Settings

The Settings command provides access to the Simulation Server Settings dialog which allows you to view and control some of the settings used by the Simulation Manager on local and network machines.

Edit Description

The [Calculation Description](#) that you provide on the General tab of the Simulation Calculation Options dialog is used to identify the simulation in the in SM grid. In some cases it will be important to describe the simulation accurately especially as it is used to identify datasets when loading data to the Results Viewer. If you need to change the Description after the simulation has been submitted you can do that by right clicking with the mouse on the description that you would like to change. When you have finished typing click on the green tick or press the <Enter> key. To cancel any changes made click on the red cross.

	Description	File Name
 	Base case	  Optimization_v7.2_MP.dsb

Simulation Servers Dialog

Simulation Servers

Simulation Server	Service	Parallel Simulations		Use
		Max	Current	
 localhost	Running	7	6	☒
 ensims-x3290	Running	31	6	☒

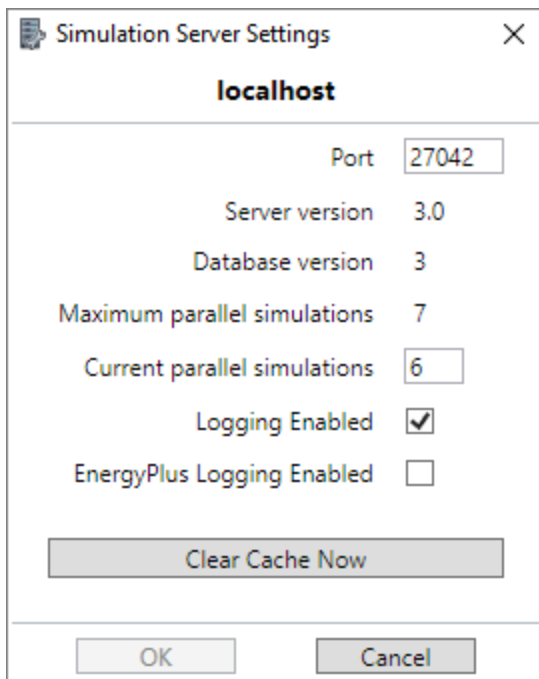
You can view the status of each of the connected servers as well as the machine-dependent maximum possible number of parallel simulations and also the maximum number of parallel simulations. There is also a checkbox to allow servers to be selected/deselected. When a server on the list is deselected then its jobs do not appear in the main SM window.

You can access all settings for the server by clicking on the **Simulation Server Settings** toolbar icon to open a dialog. Depending on your Windows privileges you may or may not be able to edit the following settings:

- **Current parallel simulations** - the maximum number of parallel simulations that can be run. This can be any number up to the limit on the server.
- **Logging enabled** - tick this to enable logging to help track down any issues. You may be asked to use this option by the Support Desk if you run into any problems with the SM.
- **EnergyPlus logging enabled** - tick this to enable logging of EnergyPlus simulation messages to help track down any issues. You may be asked to use this option by the Support Desk if you run into any problems with the SM.

Simulation Server Settings		×
localhost		
Port	27042	
Server version	3.0	
Database version	3	
Maximum parallel simulations	7	
Current parallel simulations	6	
Logging Enabled	☒	
EnergyPlus Logging Enabled	☐	

Simulation Server Settings (without Administrator rights)



Simulation Server Settings (with Administrator rights)

Depending on what else the machine is being used for you should be careful not to allow the simulations to use all of the processing capacity of the machine. The maximum value allowed is 1 less than the number of processors on the machine. For powerful machines the number of processors will typically be equal to the number of physical cores x 2, so for a 4 core machine with 8 virtual processors you cannot enter a value here greater than 7 which would allow you to run up to 7 parallel simulations. If the machine is being used for other applications as well as DesignBuilder you should leave at least 2 processors. For quad core machines with 8 virtual processors a good upper limit will be between 4 and 6.

You can also use this dialog to stop and restart the **localhost** server.

Clearing Cached Versions of EnergyPlus

If you have been informed by the Support Desk about an update or if there was some problem with the download initially then you can ensure that the programs will be downloaded again by clicking on the **Clear Cache Now** button before running new simulations using the Simulation Manager.

You may notice that simulations take a little longer to carry out the first time as EnergyPlus needs to be downloaded. After this EnergyPlus is cached and the download is not necessary.

The first time you request a simulation in the Simulation Manager for a particular version of EnergyPlus (or after clearing the cache) you must have an Internet connection to allow EnergyPlus to be downloaded.

Installing The Simulation Manager On A Server

As well as running on the local DesignBuilder client machine (as it does by default) the Simulation Manager can also run on a separate Windows server machine on the network. This page explains how to set up a simulation server and how to connect to it from DesignBuilder.

Tip: You can read about some of the advantages of using a dedicated simulation server this on the main [Simulation Manager](#) page.

Install The Simulation Manager Server

1. Install the Simulation Manager Server

- Download the **Simulation Manager (JobServer.msi)** from the **Download > Software > [Server Software](#)** area of the website.
- On the dedicated server run the **JobServer.msi** installer and this should be enough to set the Windows service running

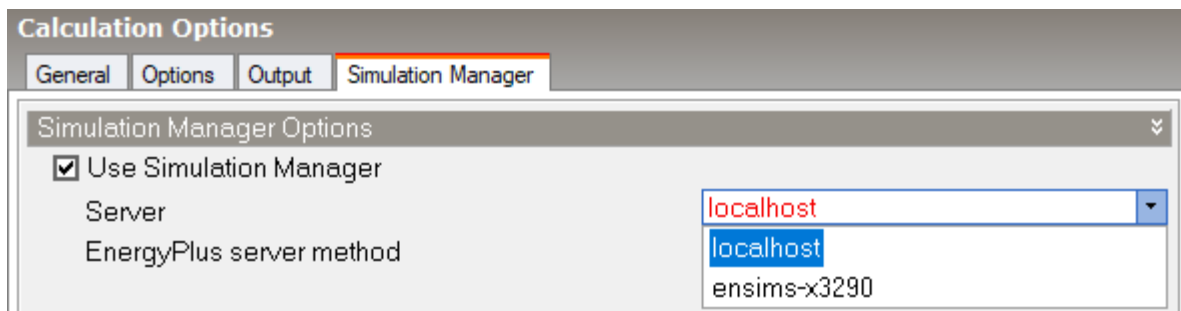
Note: The maximum number of parallel simulations for each server can be edited in the Simulation Manager user interface from the [Simulation Servers](#) dialog.

2. Ensure incoming connections to the server are allowed and not blocked by a firewall. To do this you should create an exception for the executable, **DBJobServer.exe** (not a particular port).

Note: The .NET Framework v4.6 or later must be installed on the server for the service to run correctly.

Connect The Simulation Manager On The Client Machine Running DesignBuilder

The Simulation Manager Server can be selected from the Server list on the Simulation Manager tab of the Simulation Options dialog:



The client machine needs to be configured to run jobs on the dedicated server. This is done by editing **JobServers.txt** on each user computer at C:\ProgramData\DesignBuilder\Simulation Manager

The above list was generated by the following contents in **JobServers.txt**:

```
localhost,27042,User,123456,True
ensims-X3290,27042,User,123456,True
```

Each line in **JobServers.txt** represents a server that can be accessed from the client machine. The format of each line is:

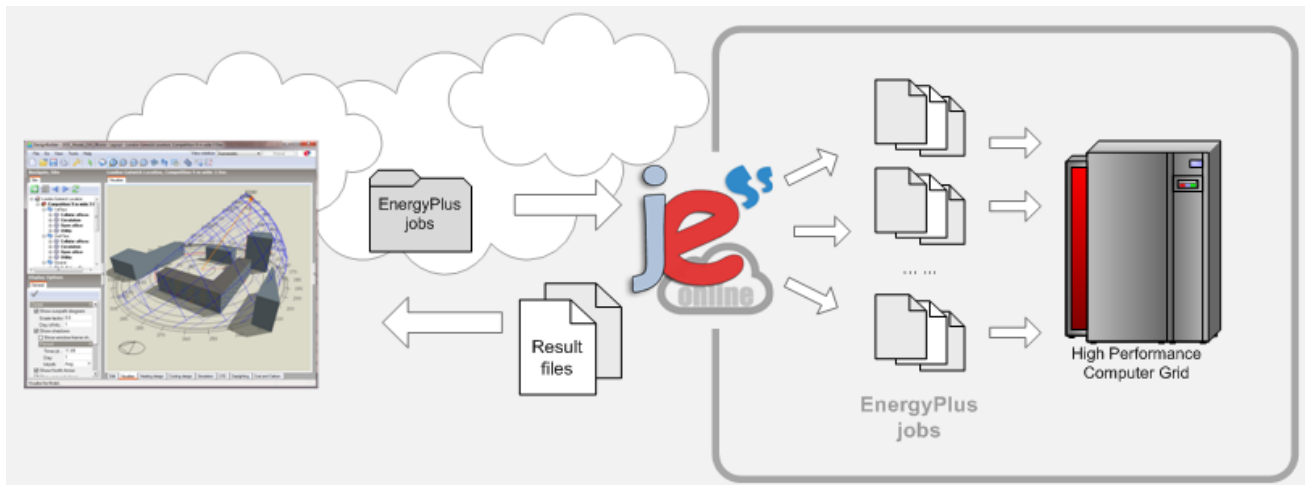
host (or IP address),Port,Username,Password,True

DesignBuilder comes with a pre-installed pre-defined host called **localhost**.which is the first line in the above file. The second line points to a simulation server running on the network on host ensims-X3290 on port 27042 with username "User" and password "123456".

Note: In current versions you must only change the server IP address and port and keep the default username and password ("User" and 123456 respectively).

If you edit JobServers.txt you must reopen the Simulation Calculation Options dialog or the Simulation Manager to see the effects.

JESS Online

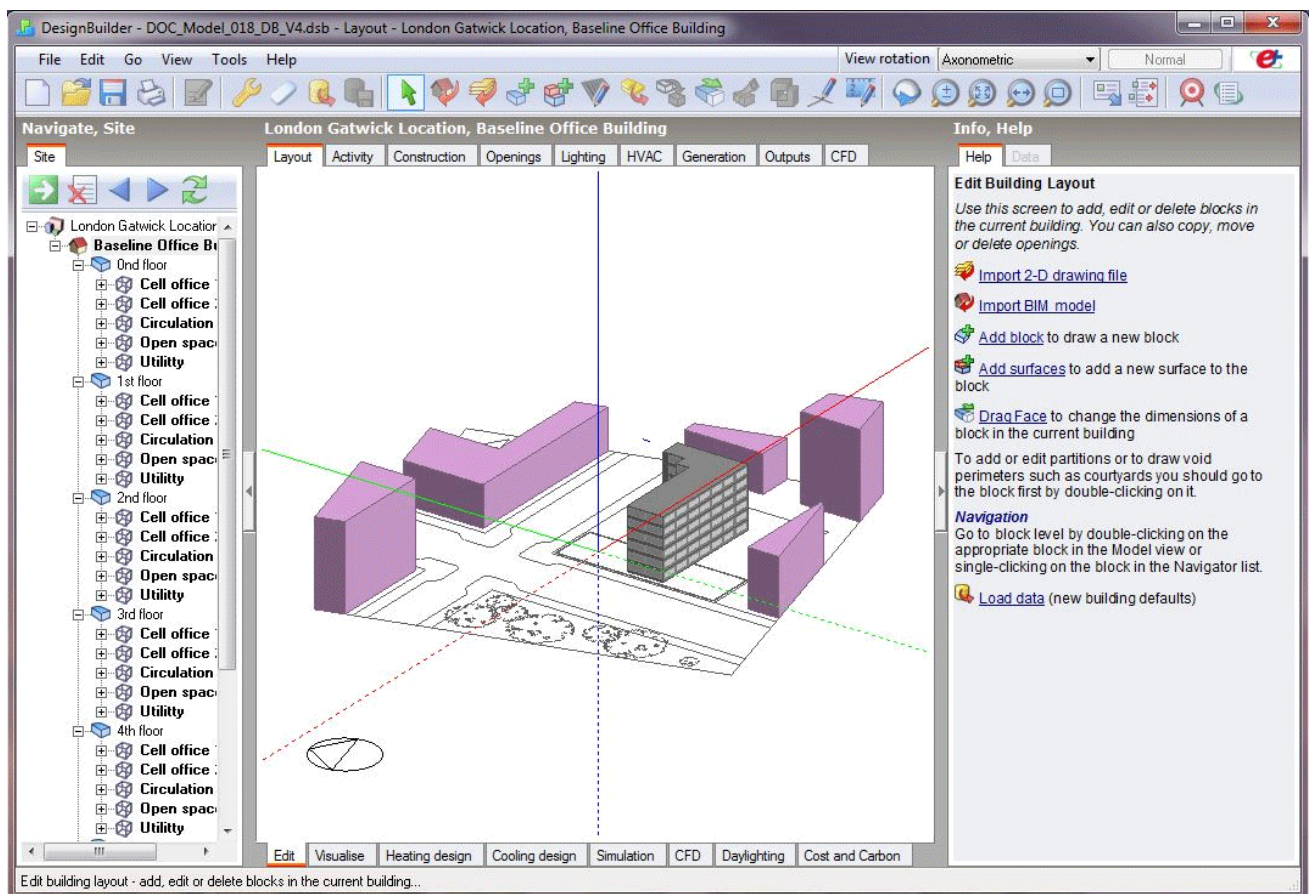


JESS (Java EnergyPlus Simulation Server) is a software system developed by Ensims to provide remote simulation solutions to EnergyPlus users. [JESS Online](#) is a service which runs time-consuming simulation jobs on Ensims' high-performance computers.

Advantages of using JESS include:

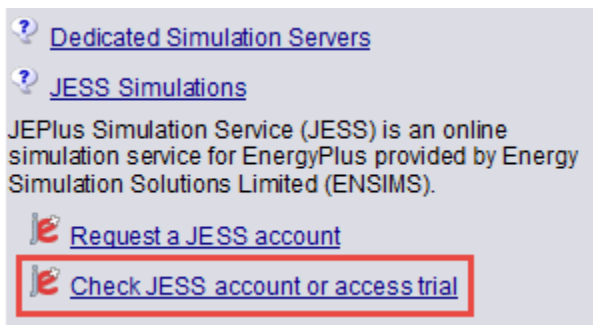
- Free up your local computer for other tasks while you are waiting for simulations to complete.
- Simulations complete quicker in most cases, especially for larger models.
- Large models which may fail to simulate on your local machine due to lack of memory may run on the JESS service.
- Option to use the JESS [Accelerate](#) method, which can speed simulations by 5 to 10 times.

The animation below summarises the method:



How To Access JESS

1. [Request a JESS account](#) on the Ensims website.
2. Log on with your JESS username and password from the dialog in DesignBuilder. To access this dialog, go to the [Simulation Manager](#) tab of the Simulation Calculation Options dialog and click on the **Check JESS account or access trial** Info panel link:



Enter your credentials on the dialog. Confirm that you agree with the JESS conditions and click on **Log on** button:

Login

Logon is required for: [Create Account](#)

User Email:

Password:

☐ I confirm that I have read, understood and agree to the JESS service terms.

[Open JESS Service Terms](#) [Log on](#)

3. On the Simulation Manager select the [localhost Server](#) and the [JEPlus EnergyPlus Server Method](#).
4. Select the [Accelerate](#) option if you have checked and understood the implications.
5. Continue to use the Simulation Manager as you would for any other **localhost** session.

Simulation Display Options

The General tab on the Simulation Display Options Panel provides options for controlling the data displayed in the Simulation data display screen. General Display Options are grouped under three headers:

- Data - control the data displayed.
- Y-axis - for customising the layout of the Y-axis.
- Appearance - for customising the style and appearance of the graphics.

The [Detailed](#) tab on the Display Options panel provides controls which allow you to select exactly the data items you wish to display.

Data Options

Interval

Select an interval from the list:

- **1-Annual** - show data for the entire simulation.
- **2-Monthly** - show data broken down by month.
- **3-Daily** - show data broken down by day.
- **4-Hourly** - show hourly data. Useful for checking operation of equipment and comfort conditions for particular design periods.
- **5-Sub-hourly** - show data recorded at time step intervals (2 and 6 intervals per hour depending on the 'Time steps per hour' calculation option). Useful for detailed checking of building performance.
- **6-Distribution** - [temperature distribution curves](#) showing 'hours at', 'hours below' and 'hours above' temperature levels in 1°C temperature intervals during the occupied period. Note that 'Data' (below) should be set to **3-Comfort** to get temperature distribution curves.

Note: be aware that requesting Hourly and Sub-hourly results in large buildings and/or long simulation periods can result in very large data sets which take a long time to read in after the simulation.

See also:

[Hourly and Sub-Hourly Results Times](#)

Data

Select an option to indicate the data you wish to display:

- **1-All** - fabric and ventilation heat gains/losses, internal gains (not Heating Design), temperatures and outside dry-bulb air temperature.
- **2-Site data** - all site data.
- **3-Comfort** - inside air, the radiant and comfort temperatures, relative humidity, ASHRAE 55 and various comfort indices. If you set Interval to **6-Distribution** you get temperature distribution curves.
- **4-Internal gains** - internal gains including equipment, lighting, occupancy, solar and HVAC heating/cooling delivery.
- **5-Fabric and ventilation** - heat gains to the space from the surface element (walls, floors, ceilings etc.) and ventilation. Negative values indicate heat loss from the space.
- **6-Fuel breakdown** - fuel consumption broken down by system category (**building level only**).
- **7-Fuel totals** - fuel consumption broken down by fuel (**building level only**).
- **8-CO2 production** - CO2 production by weight (**building level only**).
- **9-System loads** - energy loads for various categories of system component (**building level only**).
- **99-Custom** - a custom template is used to define the data to be displayed. In this case, the data to be displayed can be selected from the **Detailed** tab of the Display options panel. When the data to be displayed has been defined using the checkboxes, press the **Apply changes** Display options toolbar icon to save the selection set as a template. This can then be selected in future data analysis sessions using the **Custom** browse setting immediately below. Note that custom display templates are stored only for the current project.

[Click here](#) for information on the meaning of the data.

Tip: to update the data using different calculation options, click Update (Ctrl-U).

Custom

When Data is set to **99-Custom** any previously created display templates can be selected here.

Show as

You can display the data in one of four ways:

- **1-Graph** - displays the data on the screen in graphical format. The default option.
- **2-Grid** - you should select this option to [export](#) data in csv spreadsheet format. This [export format](#) can be loaded to Results Viewer for further analysis and comparison.
- **3-Graph and table** - the same as **1-Graph** but also includes a basic tabular display below the graph.
- **4-Table** - the same basic table format displayed for the **3-Graph and table** option, but without the graph.

Days per Page (Simulation only)

DesignBuilder displays all simulation results together, but you can select how many days are displayed at any one time using The 'Days displayed per page' option. A value of 365 means you will show all simulation data on the same screen and a value of 1 means you see each day on a separate screen.

Note: you can use the horizontal scroll bar at the bottom of the graph to see simulation results not currently displayed.

Normalise by floor area

Check this option if you wish building, block and zone data to be displayed 'per floor area', i.e. /m² or /ft², and surface and opening data to be displayed 'per surface or opening area'.

At building, block or zone level you can select the floor area to use as the denominator from the options:

- **1-All areas** - Total floor area is used,
- **2-Occupied area** - just occupied floor areas are used.

Note: you can only select **1-All areas** if the building and block data *includes* occupied areas as set in [Include unoccupied zones in block and building totals and averages](#) option in Simulation [Output options](#). Likewise you can only select **2-Occupied area** if the building and block data *excludes* occupied areas.

Y-Axis Options

These options give you control over how the Y-axis is displayed.

Lock min/max

This option locks the extent of the Y-axes so that they keep their current maximum and minimum values. This allows consistent comparison with the future results display. The Y-axis keeps its current settings until you unlock it again.

You can achieve the same effect by clicking on the 'Lock Y-axis min/max values' toolbar command.



When you lock the Y-axis the current settings are stored and displayed for each of the available Y-axes. You can edit any of these to get a more precise setting. Remember to click on the Apply button when you are ready to register a change to any numeric display option data.

Overlap multiple axes

Option allows you to save space on the Y-axis allowing different axes to overlap.

Separate axes

Check this option to add a vertical space between each of the displayed axes.

Appearance

You can control the details of the presentation of the graphical output by making selections on the Display Options panel under the [Appearance](#) header.

EnergyPlus IDF Input Data

When the [Store IDF files](#) Program option is selected, you can view the IDF data used to generate simulation results by selecting the **Tools | Display simulation input script** menu option or the **View the input data** Info panel link (in [Learning mode](#)).

If you do not have the EnergyPlus IDF editor installed, then you should associate the [.idf file extension](#) with a text editor to make this option work.

Note: this shows a snapshot of IDF data just before the simulation *not* the [_Ref-1140576427in.idf](#) file currently stored in the [EnergyPlus folder](#) which is generally from the most recent EnergyPlus simulation.

Simulation Detailed Results

The main [Display options](#) are in the bottom left of the screen. They allow you to control the content of the Detailed data and also the Display style.

The 'Analysis' tab has detailed simulation results (displayed as a line graph by default).

See also the [Analysing results in DesignBuilder's interface](#) and [Simulation Manager and Results viewer](#) tutorials

For more on how the output is calculated see: [Calculation of DesignBuilder Output from EnergyPlus Report Variables](#).

Environmental/Comfort Output

- **Air Temperature** - the calculated average temperature of the zone air.
- **Radiant Temperature** - the (area * emissivity) weighted average of the zone inside surface temperatures.
- **Operative Temperature** - the mean of the zone air and radiant temperatures.
- **Outside Dry-Bulb Temperature.**
- **Relative Humidity** - the calculated average relative humidity of the air.
- **Fanger PMV** - Fanger Predicted Mean Vote calculated according to ISO 7730.
- **Pierce PMV ET** - the Predicted Mean Vote (PMV) calculated using the effective temperature and the Pierce two-node thermal comfort model.
- **Pierce PMV SET** - the Predicted Mean Vote (PMV) calculated using the 'Standard' effective temperature and the Pierce two-node thermal comfort model.
- **Pierce Discomfort Index (DISC)** - the Discomfort index calculated using the Pierce two-node thermal comfort model.
- **Pierce Thermal Sens. Index (TSENS)** - the Thermal Sensation Index (PMV) calculated using the Pierce two-node thermal comfort model.
- **Kansas Uni TSV** - the Thermal Sensation Vote (TSV) calculated using the KSU two-node thermal comfort model.

- **Discomfort hrs (summer clothing)** - the time when the zone is occupied that the combination of humidity ratio and operative temperature is not in the [ASHRAE 55-2004](#) **summer** clothes region.
- **Discomfort hrs (winter clothing)** - the time when the zone is occupied that the combination of humidity ratio and operative temperature is not in the [ASHRAE 55-2004](#) **winter** clothes region.
- **Discomfort hrs (all clothing)** - the time when the zone is occupied that the combination of humidity ratio and operative temperature is not in the [ASHRAE 55-2004](#) **summer or winter** clothes region.

More on [Comfort Analysis](#)...

Fabric and ventilation

- **Glazing** - the total heat flow to the zone from the glazing, frame and divider of exterior glazing through conduction (i.e. excluding transmitted short-wave solar radiation which is accounted for in **Solar Gains Exterior Windows** below).

For windows **without an interior shading device** this heat flow is equal to:

- + [Convective heat flow to the zone from the zone side of the glazing]
- + [Net IR heat flow to the zone from zone side of the glazing]
- [Short-wave radiation from zone transmitted back out the window]
- + [Conduction to zone from window frame and divider, if present]

Here, short-wave radiation is that from lights and diffuse interior solar radiation.

For windows **with an interior shading device** this heat flow is equal to:

- [Convective heat flow to the zone from the air flowing through the gap between glazing and shading device]
- + [Convective heat flow to the zone from the zone side of the shading device]
- + [Net IR heat flow to the zone from the zone side of the glazing]
- + [Net IR heat flow to the zone from the zone side of the shading device]
- [Short-wave radiation from zone transmitted back out the window]
- + [Conduction to zone from window frame and divider, if present]

Note: The **Glazing** conduction gain is only displayed when the [Allow custom outputs](#) option is selected, or when generating opening level outputs using the [Store surface output and Store opening](#) outputs along with the **Glazing Heat Gain** and **Solar Transmitted** opening outputs. It is not generated when **Allow custom outputs** is switched off.

- **Walls** - Sum of heat gains to the zone from external wall inner surfaces plus all heat gains through [linear thermal bridges](#).
- **Roofs** - Sum of heat gains to the zone from external roof inner surfaces.
- **Ceilings** - Sum of heat gains to the zone from ceiling inner surfaces.
- **Floors** - Sum of heat gains to the zone from internal floor inner surfaces.
- **Floors (ext)** - Sum of heat gains to the zone from external floor inner surfaces (e.g. floor in cantilevered space, roof eaves etc, not ground floors).
- **Ground floors** - Sum of heat gains to the zone from ground floor inner surfaces.
- **Partitions** - Sum of heat gains to the zone from internal partition inner surfaces.
- **Doors and Vents** - Sum of heat gains to the zone from door and vent inner surfaces.
- **External Infiltration** - heat gain through air infiltration (non-unintentional air entry through cracks and holes in building fabric) when using [Scheduled natural ventilation](#) option.
- **External Natural Vent** - heat gain due to the entry of outside air through natural ventilation, as defined on the HVAC tab, when using [Simple natural ventilation](#) option.
- **Internal Natural Vent** - for Calculated nat vent, this output includes heat gain from other zones due to air exchange through open internal windows, doors, vents, holes and virtual partitions. For Schedule nat vent, it includes the thermal impact of any [Interzone Airflow](#) paths.
- **External Mechanical Vent** - heat gain due to the entry of outside air through the air distribution system. You can exclude mechanical ventilation from the cooling design calculations by unchecking the 'Air Distribution On' Model Data.

- **External Air** - heat gain due to the entry of outside air through external windows, vents, doors, holes and cracks when using [Calculated natural ventilation](#) option.
- **Mixing Air** - heat gain due to the entry of inside mixing air through internal windows, vents, doors and holes when using [Scheduled natural ventilation](#) option.

Note: Surface conduction data for Walls, Roofs, Ceilings, Floors, Partitions and Doors and Vents represents the heat conduction flow *just below the surface* of the construction and so includes all surface heat transfer mechanisms (convection, long and short-wave radiation).

Airflow

- **Mech Vent + Nat Vent + Infiltration** - The sum of **outside air** (in ac/h) flowing into the zone through:
 - The HVAC air distribution system +
 - Infiltration +
 - Natural ventilation +
 - Airflow through earth tubes (mechanically or naturally driven)

Note: You will usually see a small difference in air changes per hour (ac/h) between the sum of mechanical and natural ventilation and infiltration values you entered on in model data and the values reported in simulation results. This is caused by the method used to calculate and report ac/h in EnergyPlus. ac/h is reported including consideration of the air density and the exact values displayed will depend on the density of the outside air being introduced into the building relative to the density of the zone air. Cooler air is of course more dense than warmer air. More specifically, the design volume flow rate entered in ac/h is converted to mass flow (kg/s) during the simulation using the current outdoor air density. The simulated mass flow rate is then reported back out in indoor air density (for infiltration and natural ventilation) or EnergyPlus standard air density for HVAC system outside air, neither of which will ever (or extremely rarely) match the outdoor air density. So, the ac/h outputs will never exactly match the inputs. The difference between input and reported ac/h will normally be less than +/- 10%. At times when the outside air is cooler than inside there will be an underestimate of ac/h and when it is warmer there will be an overestimate. The difference between input and output values is in proportion to the difference in inside and outside air temperature.

Internal Gains

- **Task Lighting** - heat gain due to task lighting.
- **General Lighting** - heat gain due to general lighting.
- **Miscellaneous** - heat gain due to miscellaneous equipment.
- **Process** - heat gain due to process equipment.
- **Catering** - heat gain due to cooking.
- **Computer and Equipment** - heat gain due to computer and other IT-related equipment.
- **Occupancy** - sensible gain due to occupants. Please note that this can vary depending on the internal conditions. With very high temperatures the sensible gain can drop to zero with all cooling effects taking place through latent heat transfer.
- **Solar Gains Exterior Windows** - Short-wave solar radiation transmission through all *external* windows. For a bare window, this transmitted radiation consists of solar radiation passing through the glass and diffuse radiation from solar reflected from the outside window reveal, if present. For windows with a shade, this transmitted radiation is totally diffuse (shades are assumed to be perfect diffusers). For windows with a blind, this transmitted radiation consists of beam + diffuse short-wave radiation that passes between the slats and diffuse radiation from beam-to-diffuse reflection from the slats. The heating effect of solar radiation on opaque roofs and walls is accounted for in the Roofs and Walls fabric heat conduction data. *Any solar radiation re-reflected back out of the external window or transmitted through interior windows is not subtracted.*
- **Solar Gains Interior Windows** - Total beam + diffuse solar radiation transmission through *interior* windows. Requires the [3-Full interior and exterior](#) solar model option to be set.
- **Zone Sensible Cooling** - is the overall sensible cooling effect on the zone of any air introduced into the zone through the HVAC system. It includes any 'free cooling' due to introduction of relatively cool

outside air and the heating effect of any fans present. Cooling always shows as a negative heat gain in the results. *It is therefore not the same as a cooling coil energy delivery* when mechanical ventilation is involved. **It is best thought of as the overall HVAC cooling contribution to the zone heat balance.** [More on this below.](#)

- **Zone Sensible Heating** - is the overall sensible heating effect of any air introduced into the zone through the HVAC system including any 'free heating' due to introduction of relatively warm outside air and the heating effects of fans. *It is therefore not the same as a heating coil energy delivery* when mechanical ventilation is involved. **It is best thought of as the overall HVAC heating contribution to the zone heat balance.** [More on this below.](#)

Surface heat gain data refers to heat transfer from the inside surface of the building elements to the zone.

Note: taken together the **Internal gains** and the **Fabric and ventilation** data above give an *approximate Zone Heat Balance*, i.e. the data represents all of the heat flows into and out of the zone.

Heat Balance Technical Note

It is important to understand when looking at the **Zone heat balance** data that the values do not necessarily add to exactly to zero. Factors which have a bearing on this are:

1. Solar radiation values reported by EnergyPlus are gross values and do not account for the radiation re-reflected back out of internal and external windows. EnergyPlus tracks solar radiation in full detail but does not report all of the calculated values.
2. Also bear in mind that solar and some internal gains are mostly radiative gains and find their way into the fabric whereas the cooling is to the air. In thermal modelling terms it is not possible to make a simple heat balance of heat which is added to different parts of the thermal network. More specifically, heat transfer gains between the zone and the surfaces cannot be added directly to ventilation gains because they are to different 'points' in the zone heat balance (surface heat transfer is to the 'rad-air' point and ventilation is to the 'air' point).
3. Minor EnergyPlus inaccuracy in reporting of conduction through window frames causes a minor error (~1%) in Glazing heat gain reported values. This does not affect the rest of the simulation, it is just in the Glazing heat gain report.
4. There are issues such as heat conduction through walls, floors, roofs etc and diurnal storage of heat in thermal mass, especially in thermally massive buildings to take into account. When doing heat balances make sure to view data over a reasonable period (> week) to avoid such thermal storage issues.

System Heat Flows

These heat flows are plotted on the System Energy axes:

- **Zone Heating** - energy supplied by local room heaters and reheat coils to maintain room internal heating temperature setpoint temperature when using Detailed HVAC data.
- **Radiant + Convective Heating** - energy provided by heated floors to the floor construction. Check the **Ground Floors** and **Internal Floors** zone heat balance outputs to see the heat transfer from the floor to the zone itself.
- **Total Cooling** is the rate at which total energy (sensible and latent) is removed from the mixed outside and return air stream in order to bring the mixed air stream to the [specified temperature and humidity ratio](#) of the supply air stream. Total Cooling also includes any cooling energy provided by radiant cooling systems.
- **Sensible Cooling** is the rate at which sensible energy is removed from the mixed outside and return air stream in order to lower its temperature to the [specified temperature](#) of the supply air stream. Any energy needed for moisture addition or removal is ignored.

What is the difference between Zone Sensible Heating/Cooling (on the Heat Balance graph) and Zone Heating, Sensible/Total Cooling (on the System Energy graph)?

The Zone Sensible Heating is the heating effect of the HVAC system action on the zone heat balance, or in other words, the heating effect of introducing air that is warmer than the zone air. Likewise Zone Sensible Cooling is the cooling effect of the HVAC system on the zone. Note that these are not always directly related to heating and cooling coil energy delivery, especially because of the effect of free cooling from outside air. So for example even if there is no cooling coil operational at a particular time, the Zone Sensible

Cooling output on the Heat Balance graph can be high due to introduction of relatively cooler outside air into the space through mechanical ventilation. These Zone Sensible Heating/Cooling outputs will also include a component due to fans (if operational) which will tend to warm air that moves through it. The Zone Heating data in the System energy graph is the energy provided by zone heating equipment such as reheat coils and radiators/baseboard units. Likewise Sensible and Total Cooling report the sensible and total energy transferred by cooling coils to the air stream.

- **Preheat** - energy supplied by preheat coils to temper the outside air before it enters the outside air mixing box when using Detailed HVAC data.
- **AHU Heating** - energy supplied by the AHU heating coil when using Detailed HVAC data.
- **Chiller Load** - cooling energy supplied by the chiller.
- **Total Cooling** - at *building level*: sensible + latent cooling transfer to the supply air from the AHU cooling coil + any single zone unitary and fan coil units in the building. At *zone level*: sensible + latent cooling transfer to the supply air from a single zone unitary or fan coil unit. Note that any differences between **Total Cooling** and **Chiller Load** outputs are typically caused by the thermal inertia of the water in the chilled water loops. These differences will be most noticeable for data at hourly and sub-hourly intervals.
- **Sensible Cooling** - sensible only cooling transfer from the cooling coil to the supply air. Latent coil heat can be calculated as the difference: **Total Cooling - Sensible Cooling**.
- **Heat Recovery Sensible Cooling** - the *sensible* cooling rate of the supply air by the heat exchanger. This rate is determined using the supply air mass flow rate through the heat exchanger unit, the supply air inlet and outlet conditions, and the specific heat of the inlet supply air. A positive value is reported if the supply air is cooled by the heat exchanger, else the rate is set to zero.
- **Heat Recovery Total Cooling** - the *total* cooling rate of the supply air by the heat exchanger. This rate is determined using the supply air mass flow rate through the heat exchanger unit, and the enthalpy of the supply air entering and leaving the unit. A positive value is reported if the enthalpy of the supply air is decreased by the heat exchanger, else the rate is set to zero.
- **Heat Recovery Sensible Heating** - the *sensible* heating rate of the supply air by the heat exchanger. This rate is determined using the supply air mass flow rate through the heat exchanger unit, the supply air inlet and outlet conditions, and the specific heat of the inlet supply air. A positive value is reported if the supply air is heated by the heat exchanger, else the rate is set to zero.
- **Heat Recovery Total Heating** - the *total* heating energy added to the supply air by the heat exchanger.

Building Level Only

Fuel breakdown

The data for fuel consumption for building level broken down by end-use:

- **Heating** - total fuel consumption due to operation of heat generators such as boilers and heat pumps.
- **Cooling** - total chiller fuel consumption plus any fuel consumption due to DX cooling coils, air-cooled chiller condenser fans and evaporative coolers.
- **Heat rejection** - cooling tower fan electric consumption.
- **Fans** - total electrical energy consumed by HVAC fans (only available in Simple HVAC when using the Separate fans and pumps model option).
- **Pumps** - total electrical energy consumed by HVAC pumps (only available in Simple HVAC when using the Separate fans and pumps model option).
- **Auxiliary Energy** - total electrical energy consumed by HVAC fans and pumps (Simple HVAC only and only when using the NCM model option).
- **Preheat** - heating energy transferred from main plant preheat coils to HVAC system air when using Detailed HVAC data.
- **Lighting** - electricity consumed by general and task/display lights.
- **Room Electricity** - electricity consumed by room equipment other than lights (computers, equipment, process etc).
- **Room Gas** - gas consumed by room equipment (miscellaneous, process and catering)
- **Room Oil** - oil consumed by room equipment (miscellaneous, process and catering)
- **Room Solid** - solid fuel (such as coal) consumed by room equipment (miscellaneous, process and catering)
- **Room Bottled gas** - bottled gas (propane) consumed by room equipment (miscellaneous, process and catering)
- **Room Other** - other fuel consumed by room equipment (miscellaneous, process and catering)

Fuel totals

Total fuel consumption for building, data available at building level only:

- **Electricity** - total building electricity consumption.
- **Gas** - fuel consumption total building gas consumption.
- **Oil** - total building oil consumption.
- **Solid** - total building solid fuel consumption.
- **Bottled gas** - total building gas consumption.
- **Other fuel consumption** - total consumption of all other fuels.

CO2 emissions

Total carbon dioxide emission for building, data available at building level only:

- **CO2 emissions** - total CO2 emissions mass. This is calculated from fuel consumption totals (above) and the fuel emission factors for the region.

Site Weather data

Weather data stored at the site level and derived from the hourly weather file:

- **Outside Dry-Bulb Temperature**
- **Outside Dew-Point Temperature**
- **Direct Normal Solar**
- **Diffuse Horizontal Solar**
- **Wind Speed**
- **Wind Direction**
- **Atmospheric Pressure**
- **Solar Altitude**
- **Solar Azimuth**

Note: Although Site weather data is stored at the Site level, it can be displayed at Building, Block, Zone, Surface and Opening levels too. When at Building level or lower the site weather reported by EnergyPlus is displayed, but at site level, the data stored in the epw file itself is displayed. You will therefore notice small differences in site weather data when viewing at site level compared with other levels.

Note: If the data in the hourly weather file itself is changed then the corresponding simulation site weather within DesignBuilder must be manually updated for correct display at site level. You can do this at site level by pressing the Update button or by pressing <Ctrl>I-U.

Surface And Opening Levels

- **Inside Surface Temperature** - temperature at the inside surface of the surface or opening.
- **Outside Surface Temperature** - temperature at the outside surface of the surface or opening.
- **Surface Heat Gain** - heat gain from the inside surface of the main opaque part of the surface into the zone, excluding any effects from openings.
- **Glazing Heat Gain** - total conductive heat gain into the zone, minus any solar gains for openings.
- **Solar Incident** - total long-wave and short-wave incident solar gain received on the outside surface of a surface or opening.
- **Solar Transmitted** - total long-wave and short-wave solar gain transmitted through a window opening.
- **External Sunlit Fraction** - The average fraction of a surface or opening that receives direct solar gain.
- **Internal Convective Heat Transfer Coefficient** - the average internal convective heat transfer coefficient over the interval for surfaces and openings.
- **External Convective Heat Transfer Coefficient** - the average external convective heat transfer coefficient over the interval for surfaces and openings.
- **Airflow** - data is available at both surface and opening level. At surface level the data includes only infiltration through the opaque part of the parent surface and does any flow through the openings

contained on the surface. At opening level the flow includes both natural ventilation through the main opening and also infiltration through gaps. Window infiltration is defined by the selected crack template. Positive flow is flow into the zone and negative airflow is flow out. This is true for both red and blue **Airflow** data.

Hourly And Sub-Hourly Results Times

The times on the x-axis for hourly and sub-hourly results refer to the time *at the end of the period*. So for example hourly results data for 10am refers to the period 9am to 10am.

Note also that the x-axis times in the results refer to "standard" time without any daylight saving correction applied and not the "clock" time used in schedules. This means that when viewing gains results (for example) in the summer periods with an hour of daylight saving time applied you will see the gains starting an hour earlier than in the winter.

Time values that appear in schedules refer to clock time and, while the gains would be simulated as starting at the same (clock) times as defined in the schedules in summer and winter, you may see what at first sight seem like discrepancies between scheduled start/end times and outputs until you consider the points above.

Comfort Analysis

DesignBuilder EnergyPlus simulations can generate extensive data on environmental conditions within the building and resultant occupant comfort levels. The main static and adaptive comfort reports and metrics available are:

- [TM59](#) and [TM52](#), the latest domestic and non-domestic adaptive methods from CIBSE.
- [Part-O](#) Overheating Mitigation Calculations UK Building Regulations for homes.
- ASHRAE Standard 55 and CEN Standard 15251 Adaptive methods.
- [ASHRAE 55-2004](#) static graphical method.
- Fanger PMV/PPD static method.
- [Temperature distribution](#) over the simulation period, provides hours over 28°C etc.

The following comfort-related outputs can be generated:

- **Internal air temperature** - the calculated average temperature of the air.
- **Internal radiant temperature** - the average Mean Radiant Temperature (MRT) of the zone, calculated assuming that the person is in the centre of the zone, with no weighting for any particular surface.
- **Internal operative temperature** - The mean of the internal air and radiant temperatures.
- **Outside dry-bulb temperature** - site data.
- **Relative Humidity** - the calculated average relative humidity of the air.
- **Fanger PMV** - Fanger Predicted Mean Vote calculated according to ISO 7730.
- **Pierce PMV ET** - the Predicted Mean Vote (PMV) calculated using the effective temperature and the Pierce two-node thermal comfort model.
- **Pierce PMV SET** - the Predicted Mean Vote (PMV) calculated using the 'Standard' effective temperature and the Pierce two-node thermal comfort model.
- **Pierce Discomfort Index (DISC)** - the Discomfort index calculated using the Pierce two-node thermal comfort model.
- **Pierce Thermal Sens. Index (TSENS)** - the Thermal Sensation Index (PMV) calculated using the Pierce two-node thermal comfort model.
- **Kansas Uni TSV** - the Thermal Sensation Vote (TSV) calculated using the KSU two-node thermal comfort model.
- **Discomfort hrs (summer clothing)** - the time when the combination of zone humidity ratio and operative temperature is not in the ASHRAE 55-2004 **summer** clothes region.
- **Discomfort hrs (winter clothing)** - the time when the combination of zone humidity ratio and operative temperature is not in the ASHRAE 55-2004 **winter** clothes region.
- **Discomfort hrs (all clothing)** - the time when the combination of zone humidity ratio and operative temperature is not in the ASHRAE 55-2004 **summer or winter** clothes region.
- **Mech Vent + Nat Vent + Infiltration** - The sum of **outside air** (in ac/h) flowing into the zone through:

- The HVAC air distribution system +
- Infiltration +
- Natural ventilation

The above data can be displayed at all time increments. In addition, air, radiant and operative temperatures can be displayed as distribution curves (below).

You can find more detailed information about Fanger, Pierce and Kansas State University [Thermal comfort prediction](#) algorithms in the [EnergyPlus Background](#) Section.

The **Mech Vent + Nat Vent + Infiltration** data is the sum of fresh air delivered through the HVAC system, infiltration and natural ventilation outside airflow in air changes per hour (ac/h). When used together with the other environmental output it can be very useful in checking for occupant discomfort.

Note 1: It is possible to display comfort data for occupied periods only by selecting **2-Just occupied periods** as **Zone environmental and comfort reports** on the Output tab of the simulation options dialog. This option applies to all data described in this Comfort Analysis help section.

Note 2: Comfort data are not generated when the [Gains data](#) model option is set to **Lumped**. This is because in EnergyPlus the request for comfort calculations is made in the 'PEOPLE' statement and this is not used with Lumped gains.

Block and Building level data

The comfort data listed above is calculated for blocks using *floor area weighted averages* of the zones in the block. For example:

$$\text{Block average Air temperature (T}_a\text{)} = (A_1 \times T_{az1} + A_1 \times T_{az2} + A_3 \times T_{az3} + \dots) / (A_1 + A_2 + A_3 + \dots)$$

Where:

A_n is floor area of zone n ,

T_{azn} is the air temperature of zone n

There is one exception, **Mech Vent + Nat Vent + Infiltration**, which is calculated in a similar way but using *volume-weighted averages*, i.e. using zone volumes instead of floor areas in the above equation.

Building data is calculated in the same way but including all blocks.

if the **Include unoccupied zones in block and building totals and averages** option is not set then unoccupied zones are not included in the above equation.

Note: if all zones in the block/building are unoccupied and the option is not set the denominator is zero and an error message is shown and results are deleted.

Temperature Distribution

Summary description

To view temperature distribution graphs follow this procedure:

1. Select the [Temperature distribution](#) option on the Output tab of the Simulation options dialog *before* running the simulation.
2. If, as is common, you need temperature distribution data for occupied periods only, then you should select the **2-Just occupied periods** option for **Zone environmental and comfort reports** on the Output tab of the Simulation options dialog, again *before* the simulation. Note that if you do this and the zone is always unoccupied then you won't see any output.

3. Run the simulation for the period of interest. There is no need to select sub-hourly, hourly, daily or monthly output data if temperature distribution data is all you require.
4. Navigate to the zone of interest (you *must* be at zone level to view temperature distribution data).
5. Set the Interval option in the Display Options panel to **Distribution** and the Data option to **Comfort**
6. View the temperature distribution chart.

Detailed description

If you requested [Temperature distribution](#) data prior to the simulation, you can generate temperature distribution curves showing 'hours at', 'hours below' and 'hours above' temperature levels in 1°C temperature intervals during the occupied period. Bins include data for the labelled temperature through to that temperature + 1°C, so for example, the bin labelled '12°C' holds data for all temperatures in the range 12°C ≤ T < 13°C.

The temperature distribution result set is calculated using sub-hourly data generated during the simulation regardless of whether sub-hourly or hourly results were requested for output. You should therefore expect to see slightly different distribution results between runs if you change the timesteps per hour setting.

To view temperature distribution data set the Interval option in the Display Options panel to **Distribution** and the Data option to **Comfort**. This data can be useful for getting an overview of the levels of comfort (or otherwise) provided by the building over time. The results can be compared with benchmark comfort criteria.

For example, CIBSE provides recommended benchmark summertime peak temperatures and overheating criteria as follows:

Offices, schools, dwellings (living areas) should have no more than 1% of the annual occupied hours over operative temperature of 28°C. Bedrooms in dwellings should have no more than 1% of the annual occupied hours over operative temperature of 26°C.

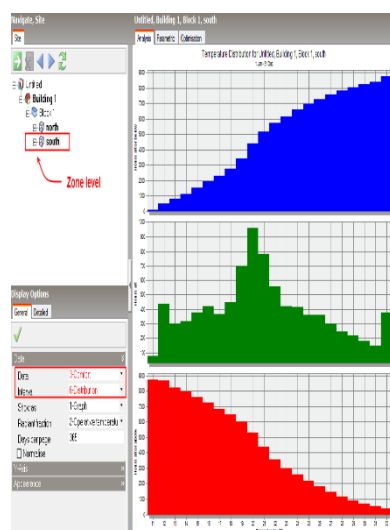
UK DfES Building Bulletin BB87 recommends an allowable overheating criteria of 80 occupied hours in a year over and air temperature of 28°C.

Note 1: Temperature distribution data is only available at **zone level**.

Note 2: This data is not available when using [Lumped](#) gains model option as the data is tied in with the EnergyPlus 'People' data set and this is not generated for Lumped gains.

Note 3: This data is not available if the [1 time step per hour](#) option is selected

Note 4: Temperature distribution data is currently only available for display in °C, even when IP units have been selected. You can however find tables of temperature distribution results in °F in the [eplustbl.htm](#) output file and plot these in a spreadsheet.



Example Temperature Distribution Output

Simplified ASHRAE 55-2004 Graph Related Outputs

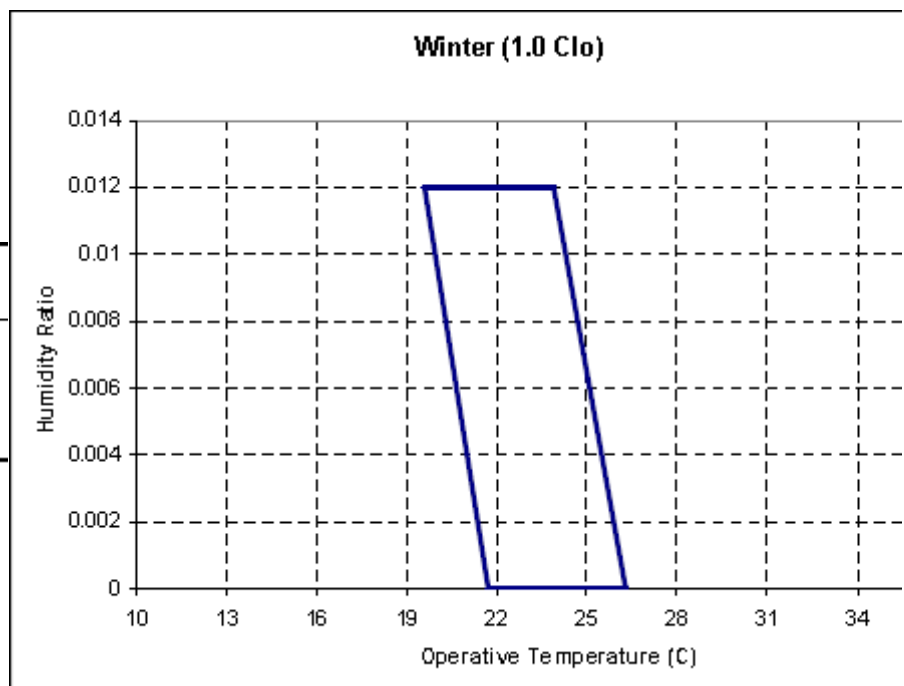
The [Discomfort hours](#) data is based on whether the humidity ratio and the operative temperature is within the region shown in ASHRAE Standard 55-2004 in Figure 5.2.1.1. For these outputs the operative temperature is simplified to be the average of the air temperature and the mean radiant temperature. For summer, the 0.5 Clo level is used and, for winter, the 1.0 Clo level is used. The graphs below are based on the following tables which extend the ASHRAE values to zero humidity ratio.

In some cases you may find a large number of discomfort hours for a particular zone even though the air temperature in the space is within normal ranges. If this happens you should check:

1. Operative temperatures which can be very different in highly glazed spaces in the summer or poorly insulated spaces in the winter.
2. Humidity levels are within range - you may find very high or very low humidity caused by inadequate ventilation. Note that the minimum comfort temperature with winter clothes when the air is very dry is 21.7°C / 71°F which may be lower than some heating temperature setpoints. Likewise in the summer when the air is very humid operative temperatures above 26.8°C / 80.2°F are considered too hot with summer clothes.

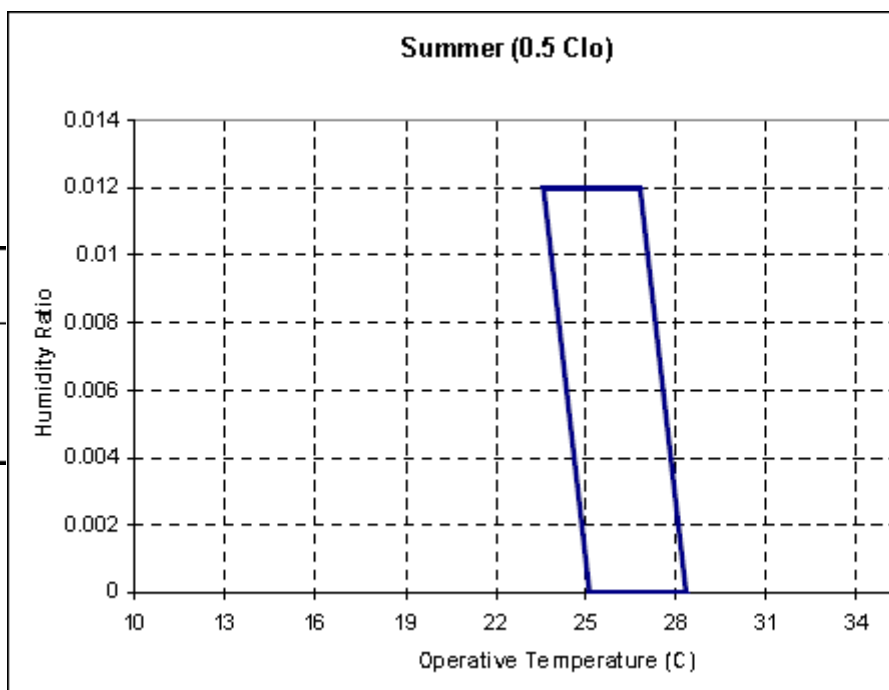
Winter Clothes

Air Temp(degC)	Humidity Ratio
19.6	0.012
23.9	0.012
26.3	0.000
21.7	0.000



Summer Clothes

Air Temp(deg C)	Humidity Ratio
23.6	0.012
26.8	0.012
28.3	0.000
25.1	0.000



Note: in ASHRAE 55 comfort assessments the criteria can be exceeded, regardless of the temperature, if the relative humidity is too high.

You can also determine **distributed comfort levels** using [CFD Comfort Calculations](#).

Note on discomfort hours calculations

The **Discomfort hrs** data displayed by DesignBuilder is similar to the "Time Not Comfortable Based on Simple ASHRAE 55-2004" data displayed in the [Summary report](#), but EnergyPlus uses a different method to DesignBuilder for calculating building level discomfort hours. "Time not comfortable Based on Simple ASHRAE 55-2004" is the summation of the number of hours over the year when one or more zones in the building fail the ASHRAE 55 comfort criteria. DesignBuilder on the other hand calculates discomfort hours at building level in the same way as other [building average values](#) as the floor area-weighted average of all zones.

CIBSE TM52 Overheating Assessments

To generate CIBSE TM52 overheating reports use the corresponding selection on the Output tab of the Simulation options dialog. When this option is selected DesignBuilder uses the EnergyPlus EMS system to provides custom TM52 outputs in the eso file for review in the [Results Viewer](#). You can see some example TM52 output at the [bottom of this page](#).

- **Criterion 1:** The first criterion sets a limit for the number of hours that the operative temperature can exceed the threshold comfort temperature (upper limit of the range of comfort temperature) by 1 K or more during the occupied hours of a typical non-heating season (1 May to 30 September).
- **Criterion 2:** The second criterion deals with the severity of overheating within any one day, which can be as important as its frequency, the level of which is a function of both temperature rise and its duration. This criterion sets a daily limit for acceptability.
- **Criterion 3:** The third criterion sets an absolute maximum daily temperature for a room, beyond which the level of overheating is unacceptable.

A room or building that fails any two of the three criteria is classed as overheating. Further information on these criteria can be found in TM52: 2013, Section 6.1.2.

Important Note: CIBSE TM52 outputs are only generated for zones without active cooling.

DesignBuilder automatically excludes results from unoccupied periods and from periods in the heating season (October - March).

Procedure:

1. Select appropriate [DSY hourly weather data](#).
2. Select the **CIBSE TM52** output option.
3. Select the [Building category](#) I-III, using guidance from the table below.

Building Category	Explanation	Suggested Acceptable Range (K)
I	High level of expectation only used for spaces occupied by very sensitive and fragile persons	2
II	Normal expectation (for new buildings and renovations) - CIBSE recommendation	3
III	A moderate expectation (used for existing buildings)	4

4. Run the simulation for at least the period 1 May - 30 September.
5. Open either the .eso or .drb output file using the Results Viewer.
6. Access summary results on the CIBSE TM52 tab as explained in [Example TM52 Outputs](#) below.

For example:

Criterion 1 is passed in Block1:Zone1 if RunPeriod output **Block1:Zone1 Criterion 1 CIBSE_TM52** has a value of 0.03 or less (i.e. it must be $\leq 3\%$)

Criterion 2 is passed in Block1:Zone1 if no Daily output **Block1:Zone1 Criterion 2 CIBSE_TM52** has a value of greater than 6.

Criterion 3 is passed in Block1:Zone1 if RunPeriod output **Block1:Zone1 Criterion 3 CIBSE_TM52** has a value of 0. A value of zero indicates that the Delta T does not exceed 4K.

Note: 2 out of the 3 criteria must be passed for an overall pass for the zone.

Tip: You can generate T_{Max} adaptive comfort and the deltaT intermediate values used in the TM52 calculations by selecting timestep outputs as well as TM52 outputs. These outputs can be viewed in the Results Viewer along with the other TM52 outputs. **TMax CIBSE_TM52** is the name of the report for the maximum adaptive comfort temperature.

Example TM52 Outputs

When you open an eso file in the Results Viewer that includes TM52 results, an extra **CIBSE TM52** tab is displayed. Clicking on this tab reveals the summary TM52 results grid as shown below.

Daily	Monthly	RunPeriod	CIBSE TM52	TM52 Example Output	
CIBSE TM52					Export CSV
Block	Zone	Criterion 1 (%)	Criterion 2 (K.hr)	Criterion 3 (hr)	Pass/Fail
Floor1	Coridor	0.00	0.00	0.00	Pass
Floor1	EOffice	0.00	0.00	0.00	Pass
Floor1	NWOffice	0.20	2.00	0.00	Pass
Floor1	Reception	0.32	4.00	0.00	Pass
Floor1	SWOffice	0.32	3.50	0.00	Pass
Floor1	WC	0.00	1.00	0.00	Pass
Floor2	Coridor	0.28	3.50	0.00	Pass
Floor2	EOffice	0.44	4.50	0.00	Pass
Floor2	NWOffice	0.71	4.50	0.00	Pass
Floor2	Reception	0.83	7.00	0.00	Pass
Floor2	SWOffice	0.87	7.50	0.00	Pass
Floor2	WC	0.32	3.00	0.00	Pass
GroundFloor	Coridor	0.00	0.00	0.00	Pass
GroundFloor	EOffice	0.00	0.00	0.00	Pass
GroundFloor	NWOffice	0.16	1.50	0.00	Pass
GroundFloor	Reception	0.08	1.00	0.00	Pass
GroundFloor	SWOffice	0.24	3.00	0.00	Pass
GroundFloor	WC	0.00	0.50	0.00	Pass

Weather data

DSY weather data for the nearest location should be used for TM52 simulations. The guidance does not specify which DSY files should be used.

Tip: You can obtain CIBSE Current and Future weather files suitable for use in TM52 analyses from the [DesignBuilder CIBSE weather file page](#).

The DSY weather data files required for the simulations can be imported using the approach described on the [Add new hourly weather data](#) help page.

CIBSE TM59 - Modelling And Analysis Guide

CIBSE TM59 is a standardised approach to predicting overheating risk for residential building designs (new-build or major refurbishment) using dynamic thermal analysis. The testing of the methodology has focused on flats, as they tend to represent a higher overheating risk than houses. However, the methodology should also be applicable to houses.

There are two versions of CIBSE TM59, and the details of how to use these two versions are described in separate sections:

- [TM59:2017](#) - the original TM59 methodology introduced in 2017.
- [TM59:2026](#) - the 2026 revision first implemented in DesignBuilder v2025.1.

Differences between TM59 2017 and 2026

The 2026 revision introduces several important changes shaped by industry feedback, practical experience and recent research. While it retains the overall dynamic thermal modelling approach and the naturally

ventilated / mechanically ventilated split, it changes several of the pass criteria and adds an elevated air speed provision.

New bedroom assessment criteria

One of the most notable changes in TM59:2026 relates to how overheating is assessed in bedrooms. Under the previous methodology, bedroom overheating was assessed based on the number of hours the room exceeded a specified temperature during sleeping hours. TM59:2026 introduces a new approach based on mean night-time temperatures, reflecting the latest research into sleep quality and the impact of heat on occupants overnight. This change is intended to provide a more realistic assessment of sleeping comfort and how people experience overheating during the night.

New 3-stage assessment process

TM59:2026 introduces a three-stage process for assessing overheating risk:

- Stage 1 – Considers passive design measures only
- Stage 2 – Takes account of local constraints, such as noise, security and air quality restrictions
- Stage 3 – Allows for mechanical cooling where required

This structure reinforces the principle that passive measures should be considered first, with mechanical cooling used only where it is necessary.

Updated weather files

TM59:2026 adopts a dedicated weather file framework based on the latest climate projections. This provides a more robust basis for overheating assessments by considering future climate conditions as well as current weather patterns.

More detailed real-world modelling

TM59:2026 provides expanded guidance on practical issues that can affect overheating risk in real homes, including:

- Noise restrictions
- Security concerns
- Air quality limitations
- Window opening constraints

These factors can have a significant influence on ventilation strategies and how occupants are able to use their homes during periods of hot weather. By recognising these real-world constraints, the updated methodology supports a more practical and realistic assessment of overheating risk.

Recognition of passive cooling measures

TM59:2026 provides clearer recognition to measures that can improve thermal comfort without significantly increasing energy use. This includes updated guidance on the use of ceiling fans, alongside continued emphasis on passive cooling measures such as solar shading and natural ventilation.

The table below summarises the changes that affect how a model is set up and assessed.

Topic	TM59 (2017)	TM59 (2026)
Criteria set	Same four tests present but not lettered: a nat-vent adaptive test, a bedroom night test, a fixed-temperature test for mechanically ventilated homes, and an advisory corridor check.	The same tests are now given explicit letters A to D: A living rooms/kitchens/bedrooms adaptive exceedance; B bedroom night mean; C mechanically ventilated or cooled fixed temperature; D communal / circulation. C and D existed in 2017 but were not lettered.

Topic	TM59 (2017)	TM59 (2026)
Bedroom night criterion (B)	Operative temperature 22:00 to 07:00 must not exceed 26 °C for more than 1% of annual hours (32 h; 33 h or more is a fail).	Mean operative temperature during sleeping hours must not exceed the night threshold T _n on more than 4 nights. T _n = 27 °C (Cat II) or 26 °C (Cat I, vulnerable).
Sleeping window	22:00 to 07:00.	23:00 to 08:00.
Assessment period	Criterion A over May to September; the night and mechanically ventilated tests read on an annual-hours basis.	All four criteria assessed over May to September.
Home office activity	No home-working profile. Living spaces modelled with evening-weighted occupancy.	New home office activity for daytime home-working occupancy. Assessed under Criterion A only; Criterion B (the night test) is not applied to it.
Mechanically ventilated / cooled homes	Assessed against the fixed-temperature test (Criterion C) only.	Must pass both Criterion B (bedroom night mean) and Criterion C (fixed temperature). Bedrooms in these homes are no longer exempt from the night test.
Elevated air speed / ceiling fans	Credited by adjusting the operative temperature: the assessed temperature was reduced according to fan speed, then compared against the fixed comfort limit.	Credited by lifting the criterion instead: operative temperature is left unchanged and the comfort limit is raised by the air-speed uplift. Naturally ventilated adaptive (A only): +0.2 °C per 0.1 m/s, capped at +1.2 °C, applied above 25 °C. Mechanically ventilated / cooled (C): +0.35 °C per 0.1 m/s, capped at +2.1 °C.
Vulnerable occupants	Type I occupancy assumed for predominantly naturally ventilated care homes and similar.	Category I applied (a 1 K reduction on the adaptive threshold) for criteria A and B.
Communal / circulation (D)	Corridors flagged as a significant risk if operative temperature exceeds 28 °C for more than 3% of annual hours. Advisory only.	Criterion D on a 28 °C basis, assessed over May to September. Common circulation (TM59_CommonCirculationAreas) is a pass / fail; in-dwelling circulation (TM59_CirculationAreas) reports a significant-risk flag.

CIBSE TM59:2017 - Modelling And Analysis Guide

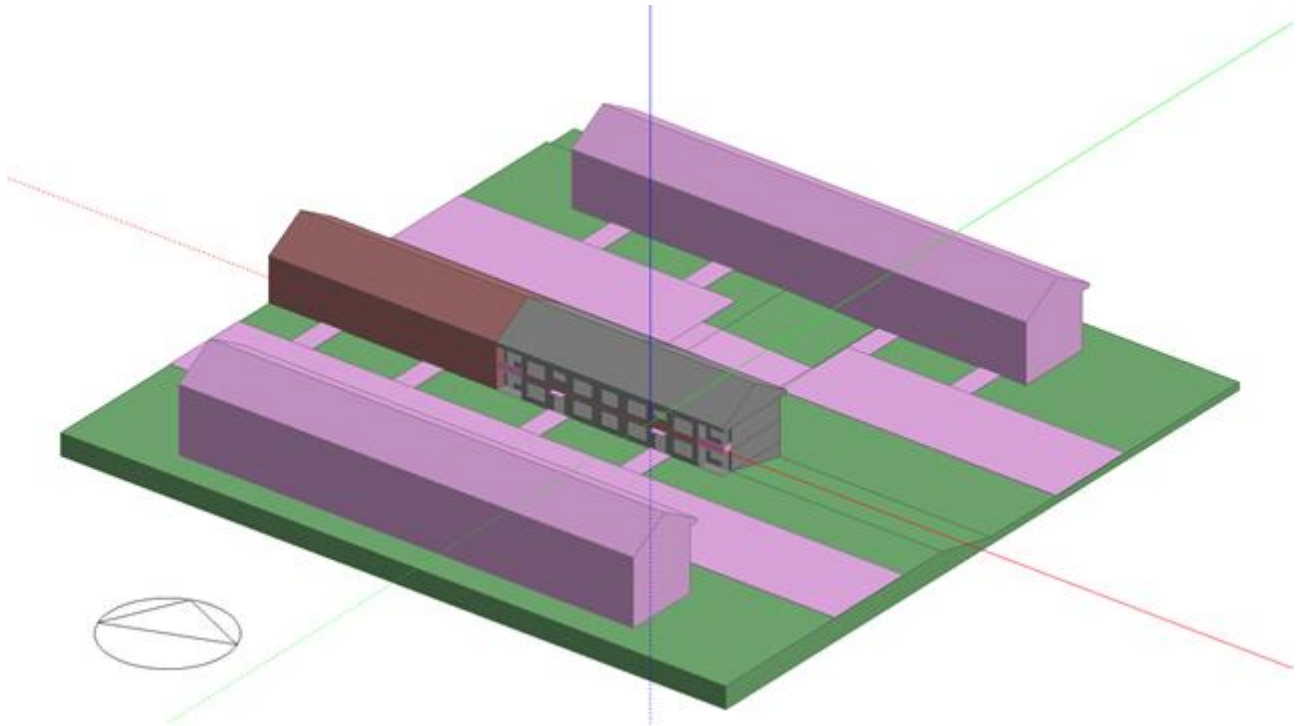
TM59:2017 is a standardised approach to predicting overheating risk for residential building designs (new-build or major refurbishment) using dynamic thermal analysis. The testing of the methodology has focused on flats, as they tend to represent a higher overheating risk than houses. However, the methodology is also applicable to houses.

Overview

This guide summarises the key points in the CIBSE TM59:2017 methodology and provides a commented example of the DesignBuilder TM59 analysis workflow. It is divided into the following sections:

- TM59:2017 Modelling process
 - [Weather data](#)
 - [Setting up Model options](#)
 - [Working with TM59 templates](#)
 - [Ventilation strategy](#)
 - [Part-O Overheating Mitigation Building Regulations](#)
- [Running the TM59 analysis](#)
- [TM59 outputs](#)

The modelling example starts after the model geometry has been completed (including construction and opening assignment).



Note: this guide does not include all of the details required to build, simulate and assess a model according to TM59 2017 and it's still necessary to refer to the TM59 Design methodology for the assessment of overheating risk in homes standard itself.

TM59 Modelling Process

Weather data

The Design Summer Year (DSY) file to use is prescribed per location in CIBSE TM59 Weather file requirements (CIBSE, 2026a), section 1.3. Use the DSY it specifies for the building's location, import it through [Add new hourly weather data](#), and check the correct file is loaded before running. CIBSE will update the files and guidance as new weather projections emerge, so you should check you are using the correct data for each project.

Since 2017, the *minimum* requirement for TM59 simulations is to use DSY1 Future weather data for the nearest location and the 2020s, High emissions, 50th % percentile scenario.

Note: Although 2020 is clearly a previous year it is still considered to be "future" weather data in TM59 since this aspect of the standard was conceived back in 2017.

Tip: You can obtain CIBSE Future 2016 weather files suitable for use in TM59 analyses from the [DesignBuilder CIBSE weather file page](#).

The DSY weather data files required for the simulations can be imported using the approach described on the [Add new hourly weather data](#) help page.

Exposure type

This setting affects the calculated natural ventilation flow rates and should reflect the real site. The appropriate site exposure type can be set at site level on the [Location](#) tab > Site Details > [Exposure to wind](#).

Air speed

According to **TM59 chapter 3.6**, the modelled air speed should be set to 0.1 m/s. This is the value used where no elevated air movement is present. To meet this requirement, an appropriate [Air velocity schedule](#) can be set at building level on the Activity tab to define air speed in all zones.

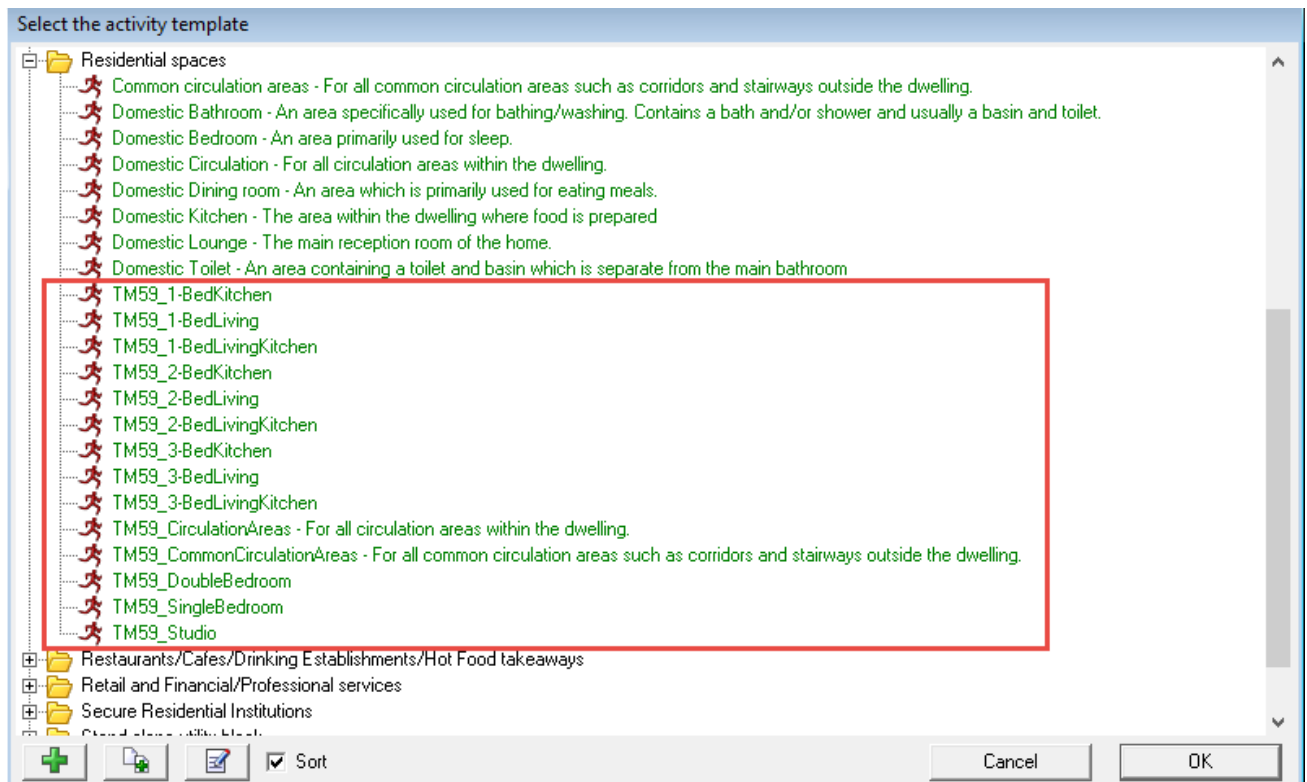
Setting up Model options

Adjust the default model options so the pre-defined TM59 activity templates behave correctly:

Setting	Value	Model Options tab	Header	Explanation
Gains data	Early	Data	Gains Data	Aligns gain profiles with the TM59 templates.
Occupancy method	3-Number of people	Data	Gains Data	Uses fixed people counts per the templates.
Occupancy latent gains	2-Fixed fraction	Data	Gains Data	According to TM59 chapter 5.1 : “A maximum sensible heat 75 W/person and a maximum latent heat gain of 55 W/person are assumed in living spaces”. Setting the Occupancy latent gains to 2-Fixed fraction ensures that the simulated occupancy gains will be aligned with the standard, i.e. the 75/55W ratio will be always used.
Equipment gain units	2-Absolute zone power	Data	Gains Data	Absolute equipment gains per zone.
Lighting gain units	1-Power density	Data	Gains Data	Power-density lighting loads (TM59 ch. 5.2).
Natural ventilation	Calculated	Data	Natural Ventilation and Infiltration	Required for airflow-network window control.
Lump similar windows on surface	Unchecked	Advanced	Simplification	Keeps per-window control settings distinct. Similar windows on the same surface should not be lumped to allow each window to have its own settings.

Working with TM59 templates

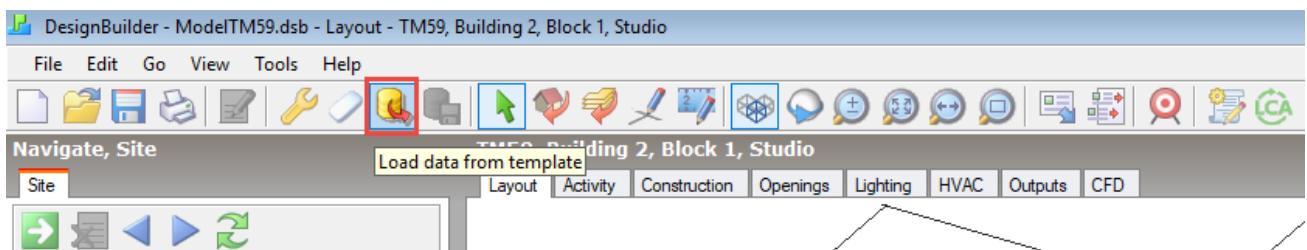
To reduce the amount of work necessary to set up the occupant and equipment gain profiles, a set of TM59 Activity templates are provided. These can be found in the “Residential spaces” activity folder.



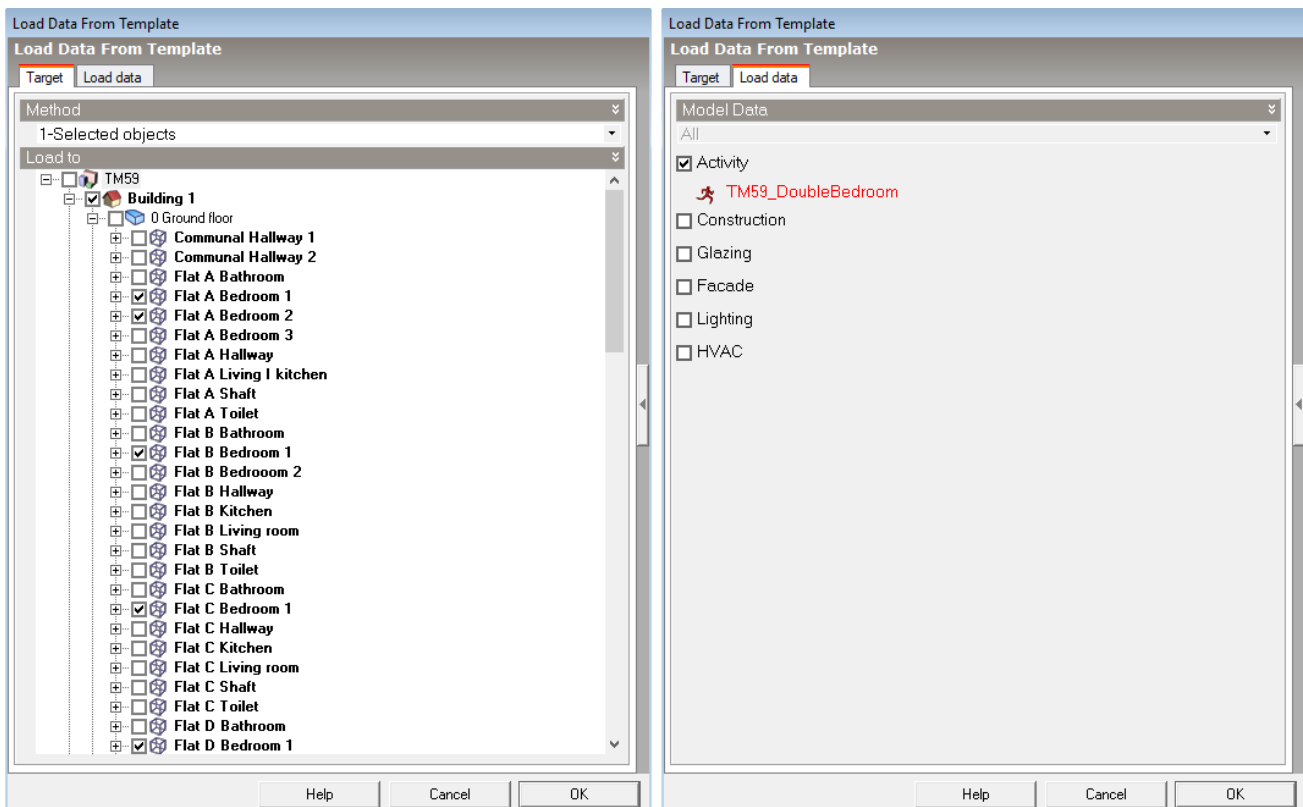
Selecting the right Activity template ensures that the correct TM59 reports are generated for each zone and avoids unnecessary outputs from being generated.

The "TM59 Lighting" Lighting template is also provided to allow appropriate lighting loads to be set, according to **TM59 chapter 5.2**. This can be selected on the Lighting tab.

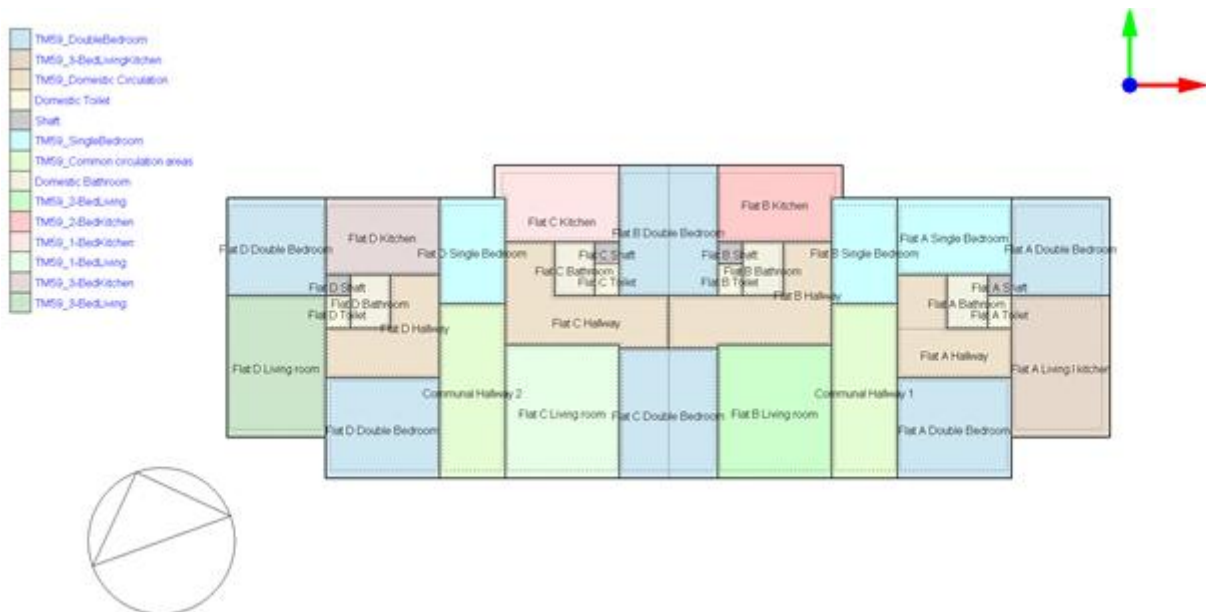
Tip: The [Load data from template](#) tool can help speed up the Activity and Lighting template assignment process. See screenshot below.



1. Open the **Load data** tab and choose the Activity to be applied.
2. Open the **Target** tab and select all the relevant zones (be aware that when selecting Building or Block level, the inheritance process still works as usual).



An example of a fully assigned floor could look something like:



Tip: When naming a large number of zones, it can be helpful to use the [Double-click to navigate site hierarchy](#) Program option. This allows you to quickly and easily change the name of zones at Block level by clicking and editing the zone name in the Navigator without unnecessarily navigating to the Zone level. This leaves you ready to change the name of the next zone without having to navigate back up to block level

Internal Environment Control - Ventilation strategy

Choosing an appropriate ventilation strategy is an important part of the TM59 assessment process and therefore all the input data should be carefully selected and results should be analysed to assure that the window opening control works as designed.

When using the **Calculated Natural Ventilation** model option, the input is defined in the following sections of the Model data.

Activity Tab

- [Ventilation setpoint temperature](#) set at Activity tab > Environmental control > Ventilation Setpoint Temperatures.

Environmental Control	
Heating Setpoint Temperatures	
Heating (°C)	18.0
Heating set back (°C)	12.0
Cooling Setpoint Temperatures	
Cooling (°C)	25.0
Cooling set back (°C)	28.0
Humidity Control	
Ventilation Setpoint Temperatures	
Natural Ventilation	
☒ Indoor min temperature control	
Min temperature definition	1-By value
Min temperature (°C)	22.0
☐ Indoor max temperature control	

HVAC Tab

- Set the appropriate HVAC template, ensuring that natural and mechanical ventilation are activated as appropriate.

Note 1: TM59 outputs will only be reported for zones without mechanical cooling.

Note 2: Natural ventilation must be activated on the HVAC tab before it is possible to access to the corresponding data for windows and vents on the Openings tab.

- [Wind factor](#) – if carrying out a conservative design assessment, it might be appropriate to reduce the effect of wind. To do this use the HVAC tab > Natural Ventilation > Options > Wind factor setting.
- [Control mode](#) – when **2-Temperature** mode is selected, opening control will automatically aim to maintain the minimum ventilation setpoint temperature defined on the Activity tab.

Natural Ventilation	
☒ On	
Outside air definition method	1-By zone
Outside air (ac/h)	5.000
Operation	
Schedule	TM59_DoubleBed_Occ
Outdoor Temperature Limits	
Delta T Limits	
Delta T and Wind Speed Coefficients	
Options	
Wind factor	1.00
Control mode	2-Temperature
☐ Modulate opening areas	
Mixed Mode Zone Equipment	
☐ Mixed mode on	

Implementation note

The TM59 standards states that:

“Windows in each room should be controlled separately and modelled as open when both the internal dry bulb temperature exceeds 22 °C and the room is occupied”.

To avoid windows opening when zones are not occupied and temperature is below 22° C, pre-defined TM59 Activity templates have [Natural ventilation setpoint temperature](#) set to 22° C and use default TM59 occupancy schedule.

Note that the [2-TemperatureControl mode](#) provides idealised control in that when the outdoor air conditions are not favourable, openings are closed. i.e. openings are closed when $T_{\text{outdoor}} > T_{\text{indoor}}$ to prevent introduction of warm air into the controlled zone. This may not be compliant with the TM59 standard because this type of an advanced control mechanism is not described in the standard which states simply that windows should be opened when $T_{\text{indoor}} > 22^{\circ}\text{C}$ and the room is occupied.

In summary TM59 requires that windows are open when the room is occupied and:

$$T_{\text{indoor}} > 22^{\circ}\text{C}$$

Whereas the default DesignBuilder rule opens windows when:

$$T_{\text{outdoor}} < T_{\text{indoor}} > 22^{\circ}\text{C}$$

Because the default DesignBuilder natural ventilation control rules do not comply exactly with the TM59 requirements, a [2-TM59 rules](#) natural ventilation control override option is provided to guarantee that the openings stay open even when the outdoor conditions are not favourable. The TM59 natural ventilation rules override can be applied from the “Edit calculation Options” dialog > “Graphable outputs” > “Comfort and Environmental” > “CIBSE TM59” > “Natural ventilation rules”.

Note that by default, the override will be applied only to zones using TM59 Activity template (excluding corridors). Note also that the override only applies when using the Calculated natural ventilation model option.

Temperature control

Default TM59 activity temperature set points and schedules are based on the UK NCM data. Heating set point schedules have been modified to use heating setback temperature instead of set point in a period from May to September.

The environmental controls will typically be configured based on the building design.

To reduce the chance of simultaneous heating and window opening, a control mechanism is implemented through the [Apply TM59 natural ventilation rules](#) calculation option. See discussion above under [HVAC tab](#).

Important note: The TM59 requirement to open the windows when the internal temperature rises above 22°C can easily lead to simultaneous heating and window opening. This is more likely to occur when:

- [Heating setpoint temperature](#) is close to the [Ventilation setpoint temperature](#)- Large openable areas are present in the zone
- Lower “Time steps per hour” is used for simulation

In general, to avoid simultaneous heating and venting, the heating setpoint temperatures should not be greater than 21°C when modelling for TM59 overheating assessments.

Openings Tab

- [Operation](#) – when using default templates, this input field should use the [Occupancy](#) schedule (so window opening is allowed only when the room is occupied). Set at Openings tab > External Windows > Operation.
- [Free Aperture](#) – the window openable area can be defined using the [% Glazing area opens](#) setting. This input will typically have a significant impact on the natural ventilation fresh air flow rates. Set at Openings tab > External Windows > Free Aperture.

Operation		<<
Operation schedule	TM59_DoubleBed_Occ	
Free Aperture		<<
Opening position	1-Top	▼
% Glazing area opens	10.0	
Discharge coefficient	0.650	

Note: Blinds and shading devices should be modelled according to [TM59 chapter 3.7](#).

Internal openings

According to the TM59 standard:

“Internal doors can be included and left open in the model in the daytime, but should be assumed to be closed when the occupants are sleeping.”

To improve the flow of air through the airflow network, it will normally be beneficial to include internal openings such as doors, set to open based on the TM59 rules.

For any other details, guidance or restrictions regarding window and door opening refer to [TM59 chapter 3.3](#).

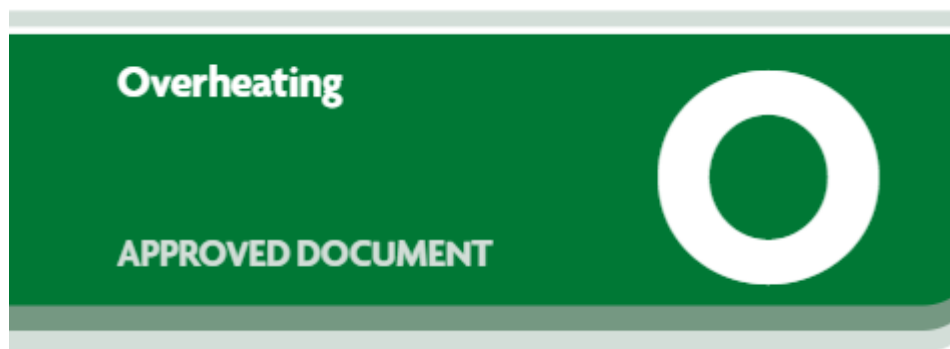
Pipework, HIU and heat maintenance tape heat loss

Heat gains from pipework, heat interface units and heat maintenance tape, need to be manually added to relevant spaces using internal gains Activity model data. More information can be found in [TM59 chapter 3.8](#).

Part-O Overheating Mitigation Calculations Building Regulations

The [Approved Document O: Overheating](#), Section 2: Dynamic thermal modelling assessment is based on TM59, but with some limitations as explained below under Ventilation Rules.

Ventilation Rules – Approved Document O (2021 Edition)



CIBSE's TM59 method requires the modeller to make choices. The dynamic thermal modelling method Part-O applies limits to these choices, which are detailed in paragraph 2.6 below. These limits should be applied when following the guidance in CIBSE's TM59.

All of the following limits on CIBSE's TM59, section 3.3, apply.

- a. When a room is occupied during the day (8am to 11pm), openings should be modelled to do all of the following.
 - i. Start to open when the internal temperature exceeds 22°C.
 - ii. Be fully open when the internal temperature exceeds 26°C.
 - iii. Start to close when the internal temperature falls below 26°C.
 - iv. Be fully closed when the internal temperature falls below 22°C.
- b. At night (11pm to 8am), openings should be modelled as fully open if both of the following apply.
 - i. The opening is on the first floor or above and not easily accessible.
 - ii. The internal temperature exceeds 23°C at 11pm.
- c. When a ground floor or easily accessible room is unoccupied, both of the following apply.

- i. In the day, windows, patio doors and balcony doors should be modelled as open, if this can be done securely, following the guidance below.
- ii. At night, windows, patio doors and balcony doors should be modelled as closed.
- d. An entrance door should be included, which should be shut all the time.

Modelling Notes

- Custom controls cover points a. and b. above in the Ventilation Rules when the **3-Approved Document O rules** [TM59 Output option](#) is selected.

Output Data

- ☒ Building and block output of zone data
- ☐ Include unoccupied zones in block and building totals and averages
- ☐ Allow custom outputs

Graphable Outputs

Energy

- ☐ Surface heat transfer
- ☒ Internal gains including solar
- ☒ Energy, HVAC etc
- ☒ Latent loads

Comfort and Environmental

Reporting period: 1-All periods

- ☒ Environmental
- ☒ Fresh air supply
- ☐ Simple ASHRAE Standard 55
- ☐ Adaptive ASHRAE Standard 55
- ☐ Adaptive CEN Standard 15251
- ☐ CIBSE TM52
- ☒ CIBSE TM59
 - ☐ Vulnerable occupants
 - ☐ Corridors
- Natural ventilation rules: 3-Approved Document O rules
- ☐ Fanger
- ☐ Pierce two-node
- ☐ Kansas State University two-node
- ☐ Temperature distribution

- The Ventilation setpoint is always modelled as 22°C, regardless of the setting made on the Activity tab.
- To apply Approved Document O control, openings must have their [Control mode](#) set to **2-Temperature**.

Layout Openings HVAC Miscellaneous CFD

Natural Ventilation

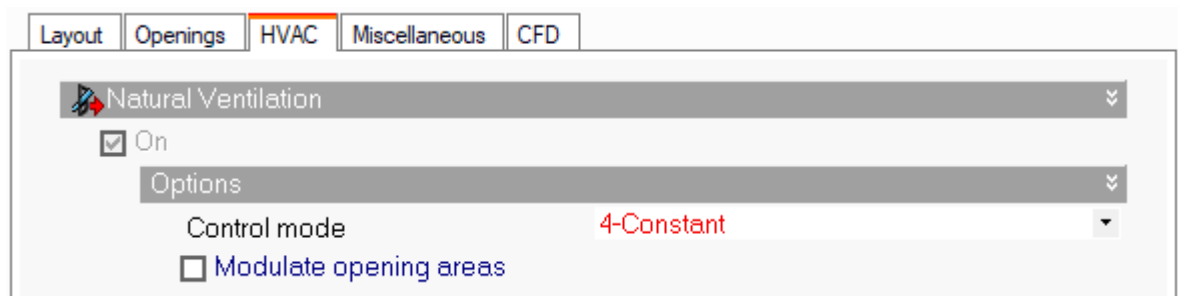
- ☒ On

Options

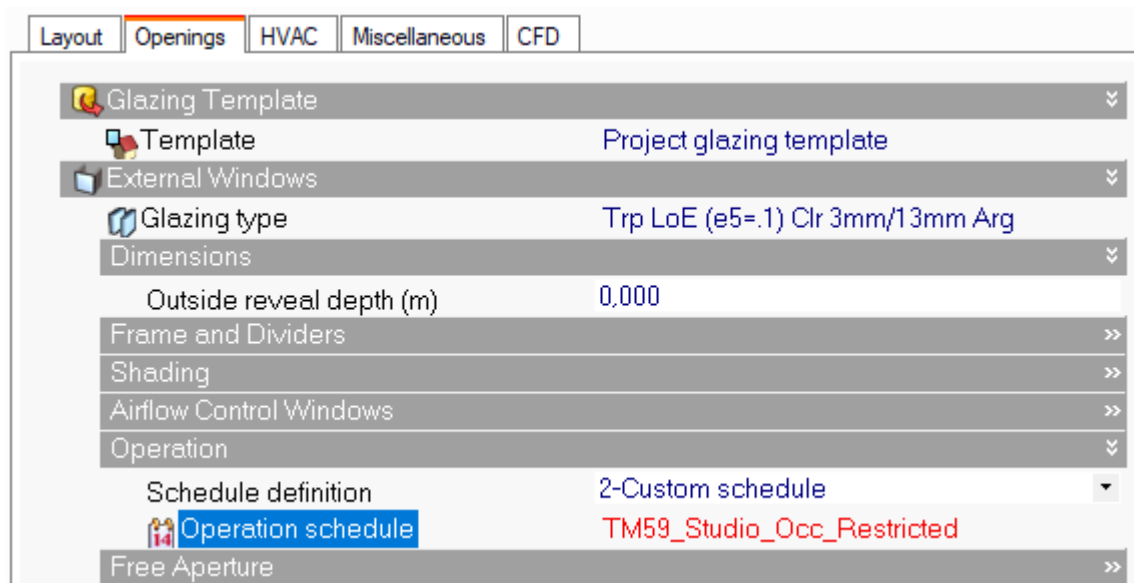
Control mode: 2-Temperature

- ☐ Modulate opening areas

- To model simple open / closed operation without modulation, you can set [Control mode](#) to **4-Constant**. This control can be used to provide purge ventilation. In this case, if internal temperature > 22°C the windows are fully opened. However, please note that this operation mode does not comply with Approved Document O, its purpose is informational only. Other control modes are not valid.



- Opening restriction during nighttime can be applied using a custom operation schedule, for example:



```
SCHEDULE:COMPACT,
TM59_Studio_Occ_Restricted,
Fraction,
Through: 31 Dec,
For: WinterDesignDay,
Until: 24:00, 0,
For: AllOtherDays,
Until: 08:00, 0,
Until: 23:00, 1,
Until: 24:00, 0;
```

Running The TM59 Analysis

Calculation options - General

To obtain correct results it's necessary to run a full annual analysis (1st Jan – 31st Dec). However, it is also important to run short term simulations, checking hourly / sub-hourly results to ensure that the ventilation / environmental control works as intended.

Bear in mind that the purpose of a TM59 analysis is to assess the risk of overheating and that because of the specific ventilation requirements of TM59, the same model might not generate accurate energy consumption results. To help avoid any such issues, the **1-Standard EnergyPlus controls** option can be selected from the [Natural ventilation rules](#) calculation option drop list when running simulations where energy consumption is an important output.

Tip: Once you are happy that your model is running well and you are ready to generate TM59 results it is usually best to uncheck the [hourly and sub-hourly output interval](#) checkboxes on the Simulation Calculation options dialog to help keep result sets to manageable sizes.

Calculation options - Options

When using Calculated natural ventilation, you should generally use a simulation timestep of at least 6 timesteps per hour. While this will increase the simulation time it does provide more accurate results, especially for window opening control.

Calculation options - Simulation Manager

Using the Simulation Manager to run your TM59 simulations will enable you to continue working on the model while the simulation is running, and if necessary to run more than one simulation in parallel and compare the results side by side in the Results Viewer.

Note: To continue working in DesignBuilder or to run another simulation, the Simulation Manager dialog must first be closed.

Calculation options - Output

Results for the TM59 analysis can be obtained by selecting [CIBSE TM59](#) output on the Output tab > Graphable Outputs. Other settings (Mechanical ventilation, vulnerable occupants) are discussed below.

Edit Calculation Options

Calculation Options

General Options **Output** Simulation Manager

Output Data

- ☒ Building and block output of zone data
- ☐ Include unoccupied zones in block and building totals and averages
- ☐ Allow custom outputs

Graphable Outputs

Energy

- ☐ Surface heat transfer
- ☒ Internal gains including solar
- ☒ Energy, HVAC etc
- ☒ Latent loads

Comfort and Environmental

Reporting period 1-All periods

- ☒ Environmental
- ☒ Fresh air supply
- ☒ Simple ASHRAE Standard 55
- ☐ Adaptive ASHRAE Standard 55
- ☐ Adaptive CEN Standard 15251
- ☐ CIBSE TM52
- ☒ CIBSE TM59
 - Version 2-CIBSE TM59:2026
 - ☐ Vulnerable occupants
 - Natural ventilation rules 2-TM59 rules
 - ☒ Restrict venting factor by schedule

Basic CIBSE TM59 output

Selecting the [CIBSE TM59](#) output option will generate results according to the requirements of **TM59 chapter 4.2** as described below.

☒ CIBSE TM59

☐ Vulnerable occupants
☐ Corridors

Natural ventilation rules
2-TM59 rules
▼

Criteria for homes predominantly naturally ventilated

Drawing from TM52 and CIBSE Guide A, compliance is based on passing both Criteria 1 and 2 as follows.

a) For living rooms, kitchens and bedrooms:

- **CIBSE TM52 Criterion 1:** Hours of exceedance ($dT \geq 1K$ shall not be more than 3% of occupied periods during the period May to September) must be passed.
This criterion is reported for occupied, non-cooled zones using the following Activity templates:
 - TM59_1/2/3-BedKitchen,
 - TM59_1/2/3-BedLiving,
 - TM59_1/2/3-BedLivingKitchen,
 - TM59_Studio,
 - TM59_Single_Bedroom,
 - TM59_DoubleBedroom.

b) For bedrooms only:

- **CIBSE TM59 chapter 4.2:** Operative temperature in the bedroom from 10 pm to 7 am shall not exceed 26 °C for more than 1% of annual hours. Note that 1% of the annual hours between 22:00 and 07:00 for bedrooms is 32 hours, so 33 or more hours above 26 °C will be considered as a fail. This criterion is reported for occupied, non-cooled zones using following “Activity” templates:
 - TM59_Studio,
 - TM59_Single_Bedroom,
 - TM59_DoubleBedroom.

Criteria 2 and 3 of CIBSE TM52 may fail to be met, but both (a) and (b) above must both be passed for all relevant rooms. In particular, bedrooms must pass both Criteria 1 and 2.

To obtain results for Criteria 2 and 3 of CIBSE TM52 it is necessary to select TM52 outputs (results for TM52 and TM59 can be selected simultaneously).

By default, in TM59 calculations, the building category (based on CIBSE TM52 definitions) is Cat. II.

CIBSE TM59 Mechanical ventilation

DesignBuilder determines whether a building is predominantly mechanically ventilated through the settings on the HVAC tab and will add results according to **TM59 chapter 4.3** as described below.

Criteria for homes predominantly mechanically ventilated

All occupied rooms should not exceed an operative temperature of 26 °C for more than 3% of the annual occupied annual hours (CIBSE Guide A (2015a)).

This criterion is only reported for occupied, non-cooled zones using the following Activity templates:

- TM59_1/2/3-BedKitchen,
- TM59_1/2/3-BedLiving,
- TM59_1/2/3-BedLivingKitchen,
- TM59_Studio,
- TM59_Single_Bedroom,

- TM59_DoubleBedroom.

CIBSE TM59 Vulnerable Occupants

☒ CIBSE TM59

☒ Vulnerable occupants
 ☐ Corridors

Natural ventilation rules

2-TM59 rules

Selecting this output will change the building category for predominantly naturally ventilated buildings according to **TM59 chapter 4.2** as described below.

Adjustments for homes with vulnerable occupants

Care homes and accommodation for vulnerable occupants, which are predominantly naturally ventilated should assume Type I occupancy (see CIBSE TM52 (2013) for description).

CIBSE TM59 Corridors

☒ CIBSE TM59

☐ Vulnerable occupants
 ☒ Corridors

Natural ventilation rules

2-TM59 rules

Selecting this output will produce results according to **TM59 chapter 4.5** as described below.

Corridors: assessment criteria

The overheating test for corridors should be based on the number of annual hours for which an operative temperature of 28 °C is exceeded. Whilst there is no mandatory target, if an operative temperature of 28 °C is exceeded for more than 3% of total annual hours, this should be flagged as a significant risk within the report.

This criterion is reported zones using the following Activity templates:

- TM59_CommonCirculationAreas
- TM59_CirculationAreas.

TM59:2017 Outputs

TM59:2017 results must be viewed in the DesignBuilder Results Viewer which can be accessed either from within DesignBuilder or from the Simulation Manager.

DesignBuilder Results Viewer

The [Results Viewer](#) is a separate application which can be used to view EnergyPlus results stored in one or more .eso files. It can be downloaded from the main [Downloads > Software](#) area of the DesignBuilder website. When installed, the application allows you to view any results contained within EnergyPlus .eso and .htm results files.

Standard Simulation (without Simulation Manager)

Results from the most recent simulation are stored in the **eplusout.eso** file in the EnergyPlus folder.

It is possible to use either of the following access routes implemented in the DB interface to open the file:

1. Navigating to the EnergyPlus folder (File > Folders > EnergyPlus folder) and double-clicking on the eso file to open it in the Results Viewer.
2. Using the toolbar icon on the Simulation screen.

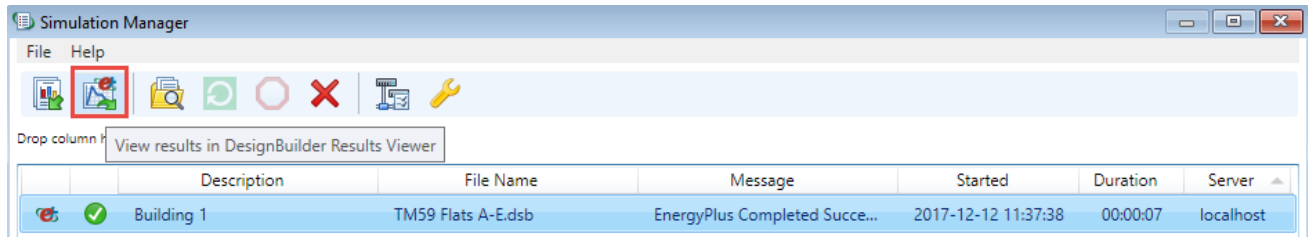


Other ways to open .eso results files are:

- From within the Results Viewer use the **File > Open eso/data set** menu command
- By making the appropriate selection on the [Program options](#) dialog, DesignBuilder will offer to open the .eso results file at the end of the simulation.

Simulations using the Simulation Manager

The TM59 results can be accessed directly from the [Simulation Manager](#) interface by clicking on the View results in the DesignBuilder Results Viewer toolbar icon.



Understanding results displayed in Results Viewer

When you open an eso file in the Results Viewer that includes TM59 results, an extra **CIBSE TM59** tab is displayed. Clicking on this tab reveals the summary TM59 results grid as shown below.

[Daily](#)
[Monthly](#)
[RunPeriod](#)
[CIBSE TM59](#)
[8% glazing opening](#)

CIBSE TM59 [Export CSV](#)

[Naturally ventilated](#)
[Corridors](#)

Criteria for predominantly naturally ventilated homes

Block	Zone	Criterion A (%)	Criterion B (hr)	Pass/Fail
TypicalFlat	Bedroom1	2.40	8.00	Pass
TypicalFlat	Bedroom2	2.86	4.00	Pass
TypicalFlat	LivingRoomXKitchen	3.08	N/A	Fail

[Daily](#)
[Monthly](#)
[RunPeriod](#)
[CIBSE TM59](#)
[8% glazing opening](#)

CIBSE TM59 [Export CSV](#)

[Naturally ventilated](#)
[Corridors](#)

Criteria for corridors

Block	Zone	% Hours Exceeded	Pass/Fail
▶ TypicalFlat	CommunalCorridor	0.07	Pass
TypicalFlat	Corridor	0.27	Pass

Criteria for homes predominantly ventilated

Compliance is based on passing both of the following two criteria:

(a) For living rooms, kitchens and bedrooms:

- **CIBSE TM52 Criterion 1:** Hours of exceedance ($dT \geq 1K$ shall not be more that 3% of occupied periods during the period May to September) must be passed.

Criterion A is passed if the value is equal or less than 3%.

In the example output above Zone TypicalFlat:LivingRoomXKitchen fails to pass Criterion A because **Hours of exceedance** are greater than 3%.

(b) For bedrooms only:

- Operative temperature in the bedroom from 10 pm to 7 am shall not exceed 26 °C for more than 1% of annual hours. (Note: 1% of the annual hours between 22:00 and 07:00 for bedrooms is 32 hours, so 33 or more hours above 26 °C will be recorded as a fail).

Criterion B is passed if the value is equal or less than 32 hours.

In the above example both bedrooms pass Criterion B because the operative temperature is exceeded for less than 32 hours.

Criteria for home predominantly mechanically ventilated

All occupied rooms should not exceed an operative temperature of 26 °C for more than 3% of the annual occupied annual hours (CIBSE Guide A (2015a)).

Corridors: Assessment criteria

The overheating test for corridors should be based on the number of annual hours for which an operative temperature of 28 °C is exceeded. Whilst there is no mandatory target, if an operative temperature of 28 °C is exceeded for more than 3% of total annual hours, this should be flagged as a significant risk within the report.

In the above example both corridor zones pass the criteria with less than 3% exceedance.

CIBSE TM59:2026 - Modelling And Analysis Guide

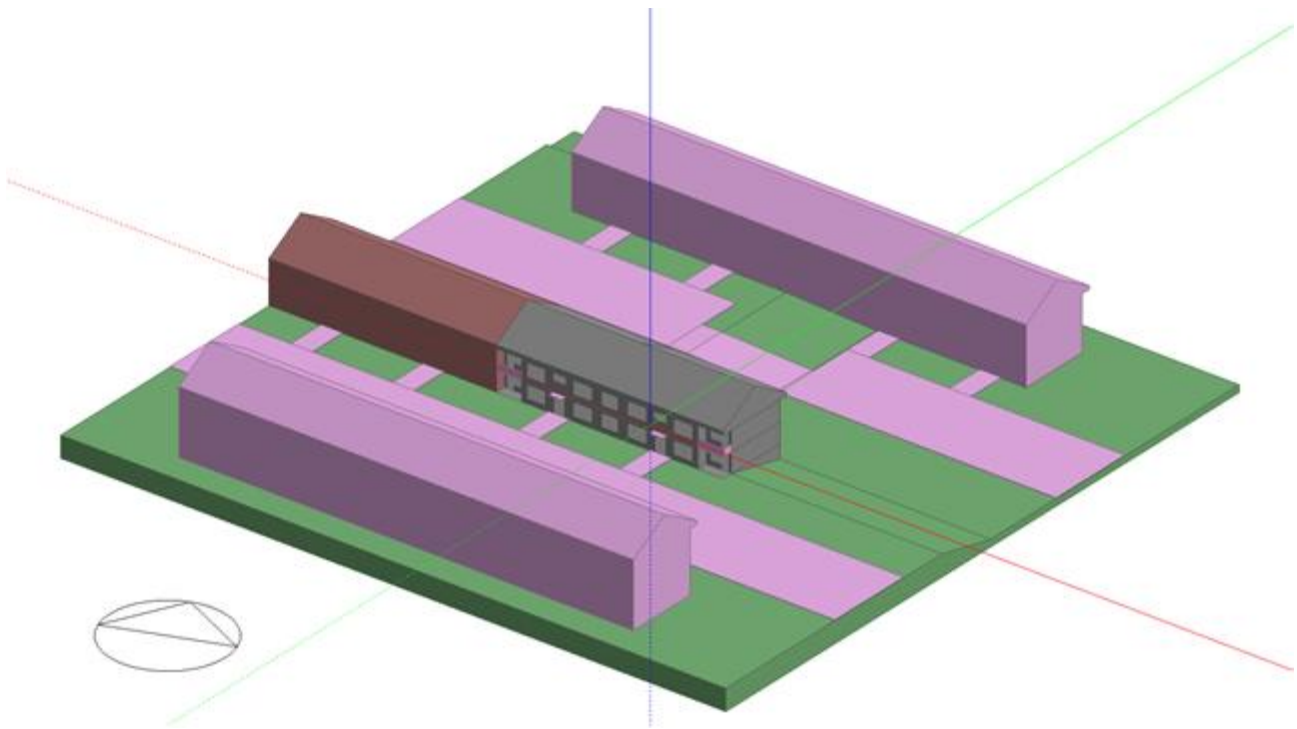
TM59:2026 is a standardised approach to assessing overheating risk for residential building designs (new-build or major refurbishment) using dynamic thermal simulation. The testing of the methodology has focused on flats, as they tend to represent a higher overheating risk than houses. However, the methodology is also applicable to houses.

Overview

This guide summarises the key points in the CIBSE TM59:2026 methodology and provides a commented example of the DesignBuilder TM59 analysis workflow. It is divided into the following sections:

- [TM59:2026 Modelling Process](#)
- [Assessment Criteria \(A to D\)](#)
- [Running the TM59 Analysis](#)
- [TM59:2026 Output Report](#)

The modelling example starts after the model geometry has been completed (including construction and opening assignment).



Note: this guide does not include all of the details required to build, simulate and assess a model according to TM59:2026, and it's still necessary to refer to the TM59 Design methodology for the assessment of overheating risk in homes standard itself.

TM59:2026 Modelling Process

Weather data

The Design Summer Year (DSY) file to use is prescribed based on the project location in CIBSE TM59 Weather file requirements (CIBSE, 2026a), section 1.3. Use the DSY it specifies for the building's location and check the correct file is loaded before running. CIBSE will update the files and guidance as new weather projections emerge, so you should check you are using the current set for each assessment.

The DSY weather data files required for the simulations can be imported using the method described on the [Add new hourly weather data](#) help page.

Exposure type

This setting affects the calculated natural ventilation flow rates and should reflect the real site. The appropriate site exposure type can be set at site level on the [Location](#) tab > Site Details > [Exposure to wind](#).

Air speed

The modelled air speed should be set to 0.1 m/s where no elevated air movement is present. To meet this requirement, an appropriate [Air velocity schedule](#) can be set at building level on the Activity tab to define air speed in all zones.

Where ceiling fans or other means provide elevated local air movement, the 2026 revision credits the resulting adaptive comfort uplift. The local air speed drives the uplift applied to the comfort threshold in criteria A and C (see Section 3). The uplift is only applied where the operative temperature is at or above 25 °C, matching the fan activation gate in the standard.

Ceiling fans are automatically introduced to elevate the zone air velocity when appropriate. Schedule values between 0.2 - 0.6 linearly increase the threshold up to 1.2C for criterion A and 2.1C for criterion C.

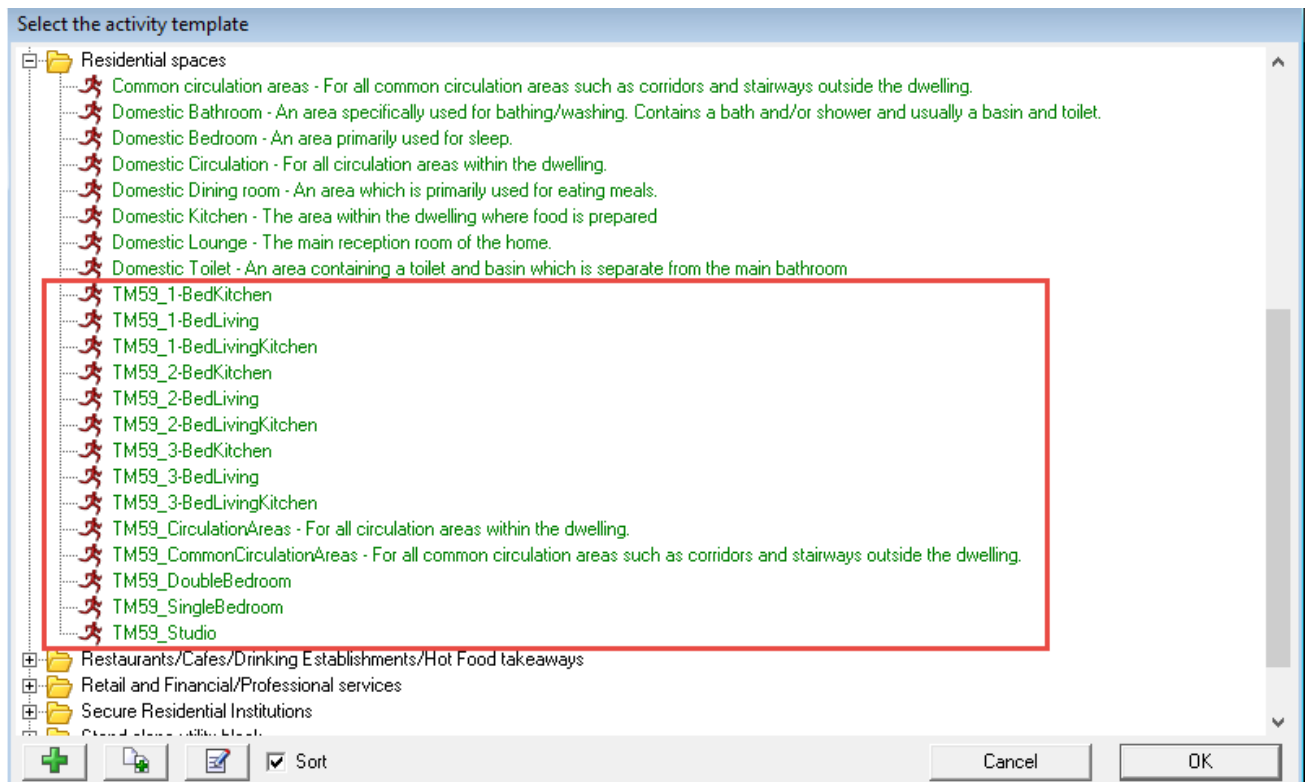
Setting up Model options

Adjust the default model options so the pre-defined TM59 activity templates behave correctly:

Setting	Value	Model Options tab	Header	Explanation
Gains data	Early	Data	Gains Data	Aligns gain profiles with the TM59 templates.
Occupancy method	3-Number of people	Data	Gains Data	Uses fixed people counts per the templates.
Occupancy latent gains	2-Fixed fraction	Data	Gains Data	According to TM59 chapter 5.1 : “A maximum sensible heat 75 W/person and a maximum latent heat gain of 55 W/person are assumed in living spaces”. Setting the Occupancy latent gains to 2-Fixed fraction ensures that the simulated occupancy gains will be aligned with the standard, i.e. the 75/55W ratio will be always used.
Equipment gain units	2-Absolute zone power	Data	Gains Data	Absolute equipment gains per zone.
Lighting gain units	1-Power density	Data	Gains Data	Power-density lighting loads (TM59 ch. 5.2).
Natural ventilation	Calculated	Data	Natural Ventilation and Infiltration	Required for airflow-network window control.
Lump similar windows on surface	Unchecked	Advanced	Simplification	Keeps per-window control settings distinct. Similar windows on the same surface should not be lumped to allow each window to have its own settings.

Working with TM59 templates

To reduce the work necessary to set up the occupant and equipment gain profiles, a set of TM59 Activity templates are provided. These can be found in the “Residential spaces” activity folder.



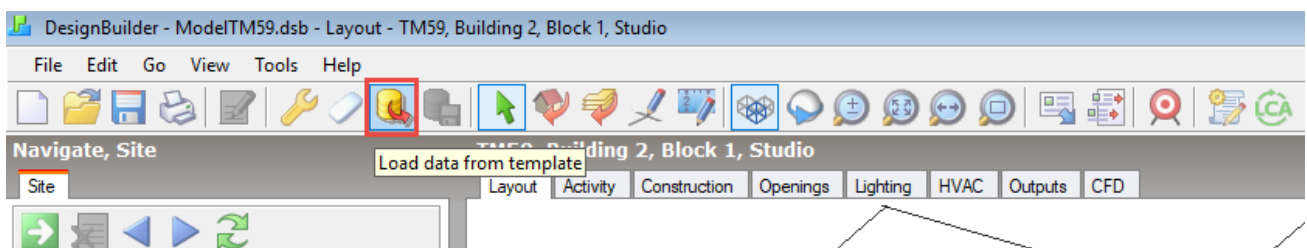
Activity template

Selecting the appropriate Activity template ensures that the correct criterion is reported for each zone, so assigning the right template to each room is essential. Use the TM59 Activity templates in the **Residential spaces** folder.

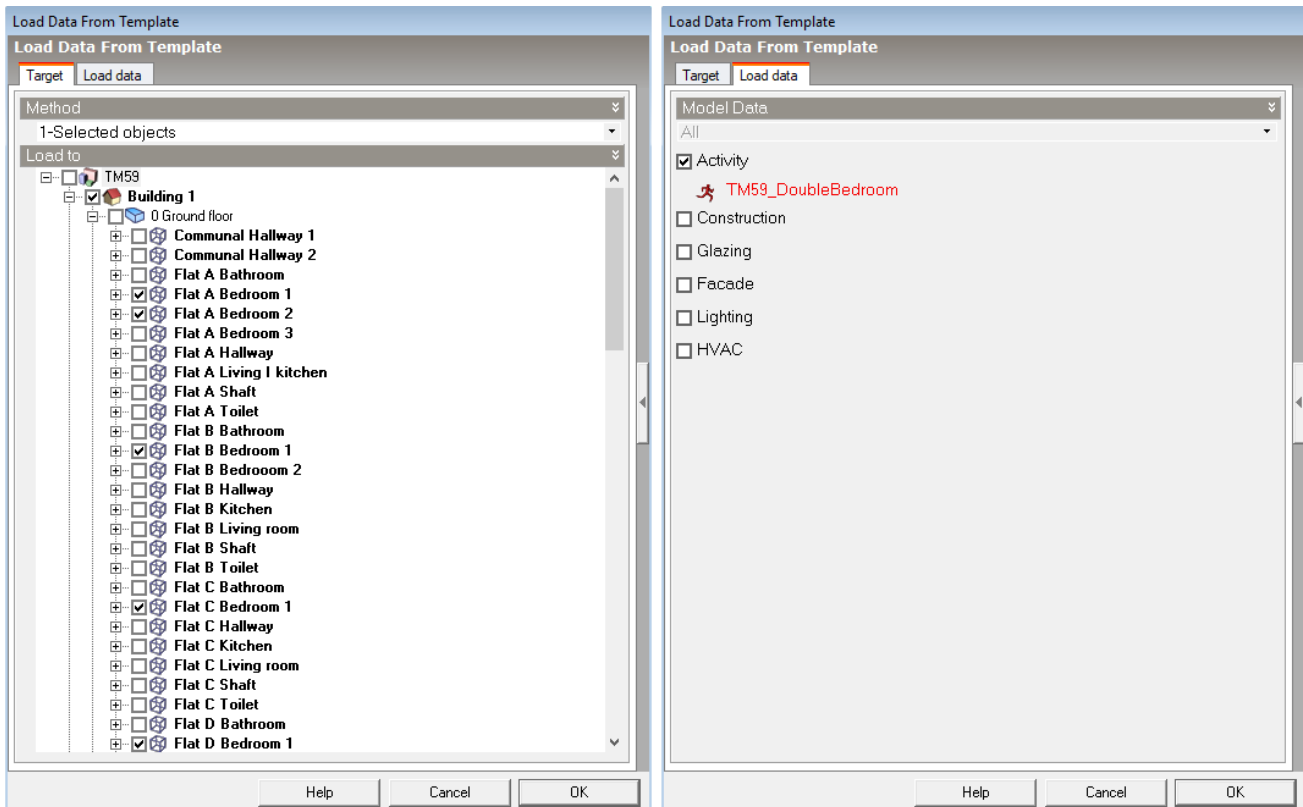
Lighting template

Use the "TM59 Lighting" Lighting template in the <System> category to set appropriate lighting loads, according to **TM59 chapter 3.7.1**. This can be selected on the Lighting tab.

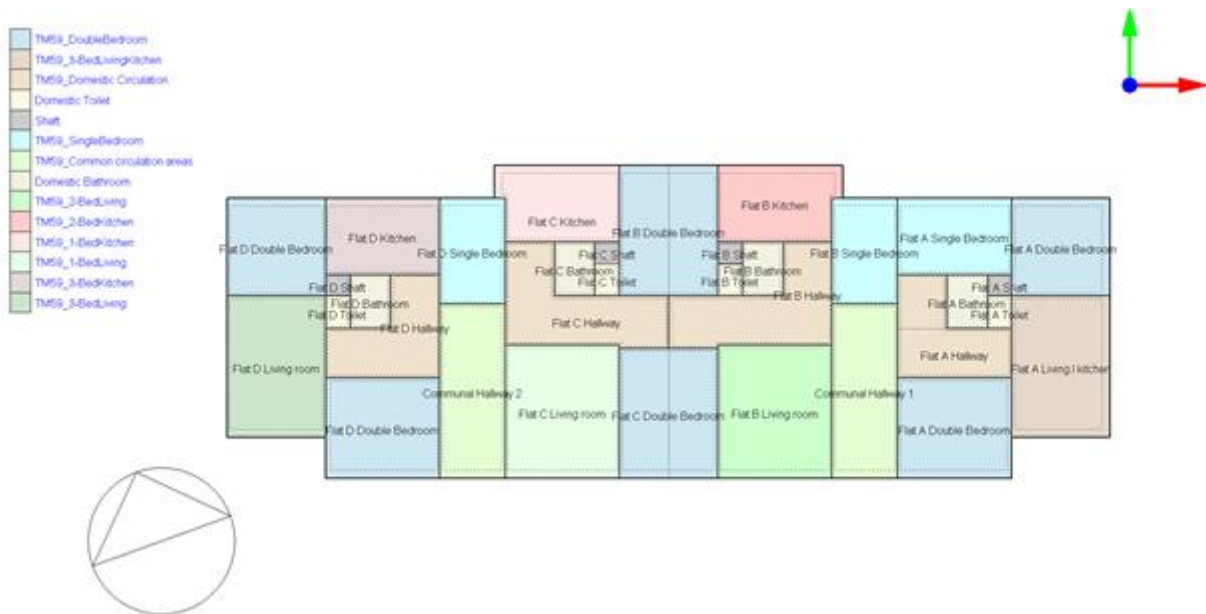
Tip: The [Load data from template](#) tool can help speed up the Activity and Lighting template assignment process. See screenshot below.



1. Open the **Load data** tab and choose the Activity to be applied.
2. Open the **Target** tab and select all the relevant zones (be aware that when selecting Building or Block level, the inheritance process still works as usual).



An example of a fully assigned floor could look something like:



Tip: When naming a large number of zones, it can be helpful to use the [Double-click to navigate site hierarchy](#) Program option. This allows you to quickly and easily change the name of zones at Block level by clicking and editing the zone name in the Navigator without unnecessarily navigating to the Zone level. This leaves you ready to change the name of the next zone without having to navigate back up to block level

Internal Environment Control - Ventilation strategy

Choosing an appropriate ventilation strategy is an important part of the TM59 assessment process and therefore all the input data should be carefully selected and hourly or sub-hourly airflow rates should be analysed to confirm the window opening control works as expected.

When using the [Calculated Natural Ventilation](#) model option, the input is defined in the following sections of the Model data.

Window opening rule

TM59:2026 specifies the window opening control directly. In DesignBuilder it is set through the [Natural ventilation rules](#) setting, which offers four options:

- **1-Standard EnergyPlus controls:** DesignBuilder's default control, which closes openings when outdoor air is warmer than indoor ($T_{out} > T_{in}$) - behaviour the standard does not describe.
- **2-TM59 rules:** while the space is occupied, openings are held closed below 22 °C operative temperature, open linearly between 22 °C and 26 °C, and are fully open at 26 °C or above (see Openings modulation profile below; schedule clarifications are in section 3.4, Natural ventilation, windows and door openings, of CIBSE TM59:2026).
- **3-Approved Document O rules:** the Part-O window rules set out below in [Part-O Overheating Mitigation](#).
- **4-TM59 Scotland** (ADO rules for Scotland): applies the Approved Document O rules adapted for the Scottish standard, setting the discharge coefficient to 0.62 and the % Glazing area opens to 100%.

Openings modulation profile

The 2026 revision modulates the openable area with temperature rather than switching it fully on at the setpoint. While a space is occupied, openings are held closed when the internal operative temperature is below 22 °C, and they are fully open at 26 °C or above. Between 22 °C and 26 °C the openable area increases linearly with rising operative temperature.

Heating and setback

The TM59 templates use a heating setback rather than a setpoint from May to September. To avoid simultaneous heating and window opening, keep heating setpoints no higher than 21 °C for the overheating run. The risk is greatest when heating and ventilation setpoints are close, openable areas are large, or the timestep is coarse.

Openings and internal doors

- **Operation:** use the Occupancy schedule so opening is allowed only when the room is occupied (Openings tab, External Windows, Operation).
- **Free Aperture:** set the openable area with the % Glazing area opens field. This strongly affects fresh air flow rates.
- **Internal doors:** may be open in the daytime but should be modelled closed while occupants are sleeping. Including internal openings improves airflow through the network.
- **Blinds, shading, and pipework / HIU / heat maintenance gains:** model per TM59 chapter 3.7

Restricting venting by schedule

DesignBuilder v2025 adds a [Restrict venting factor by schedule](#) option that allows you to vary an opening's free aperture through the day, typically to hold windows more closed overnight than during the day for security and/or noise reasons. When this option is selected, the [Opening operation schedule](#) defines a fraction that multiplies the window's free aperture, so the effective opening at any hour is % Glazing area opens × schedule fraction.

Note: The [Restrict venting factor by schedule](#) option is only available for the **2-TM59 Rules** and **3-Approved Document O Rules** natural ventilation rules outputs. It can be selected from the Output tab of the Simulation Calculation Options dialog. It is not available for the other TM59 rules options. Also, this functionality does not apply to non-TM59 natural ventilation simulations, where the natural ventilation operations schedule only defines availability, in which case, schedule values of 0 mean the window is closed, and values > 0 mean that it is fully open.

Take a window that should open 50% by day and 10% at night, with the night period set to 23:00–08:00 to match the TM59 sleeping period. There are two ways to obtain that split.

Method 1 – schedule carries the fractions directly. Set [% Glazing area opens](#) to 100% and let the schedule define the day and night open fractions themselves: 0.5 by day, 0.1 at night.

```
Schedule:Compact,  
DayNightSchedule,  
Fraction,  
Through: 12/31,  
For: AllDays,  
Until: 08:00, 0.1,  
Until: 23:00, 0.5,  
Until: 24:00, 0.1;
```

Method 2 – schedule scales a fixed aperture. Set [% Glazing area opens](#) to the daytime value, 50%, and use the schedule purely as a restriction factor: 1.0 by day (no restriction) and 0.2 at night, since $0.2 \times 50\% = 10\%$.

```
Schedule:Compact,  
DayNightSchedule,  
Fraction,  
Through: 12/31,  
For: AllDays,  
Until: 08:00, 0.2,  
Until: 23:00, 1.0,  
Until: 24:00, 0.2;
```

Both give the same result — 50% by day, 10% overnight — but Method 2 keeps [% Glazing area opens](#) meaningful as the aperture the window can actually reach and uses the schedule only to express the restriction. This is easier to audit and to reuse across windows that share a night profile but have different daytime apertures.

Note: The worked example above illustrates only how the venting-factor schedule is applied. It does not cover the free area and discharge coefficient calculation.

Activity Tab

- [Ventilation setpoint temperature](#) set at Activity tab > Environmental control > Ventilation Setpoint Temperatures.

The screenshot shows the 'Environmental Control' tab in the EnergyPlus GUI. Under 'Ventilation Setpoint Temperatures', the 'Natural Ventilation' section is expanded and highlighted with a red box. It contains the following settings:

- ☒ Indoor min temperature control
 - Min temperature definition: 1-By value
 - Min temperature (°C): 22.0
- ☐ Indoor max temperature control

HVAC Tab

- Set the appropriate HVAC template, ensuring that natural and mechanical ventilation are activated as appropriate.

Note 1: TM59 outputs will only be reported for zones without mechanical cooling.

Note 2: Natural ventilation must be activated on the HVAC tab before it is possible to access to the corresponding data for windows and vents on the Openings tab.

- [Wind factor](#) – if carrying out a conservative design assessment, it might be appropriate to reduce the effect of wind. To do this use the HVAC tab > Natural Ventilation > Options > Wind factor setting.
- [Control mode](#) – when **2-Temperature** mode is selected, opening control will automatically aim to maintain the minimum ventilation setpoint temperature defined on the Activity tab.

Implementation note

The TM59 standard requires *windows in each room to be controlled separately and modelled as open when both the internal dry bulb temperature exceeds 22 °C and the room is occupied*”.

To avoid windows opening when zones are not occupied and temperature is below 22° C, pre-defined TM59 Activity templates have [Natural ventilation setpoint temperature](#) set to 22° C and use the default TM59 occupancy schedule.

Note that the **2-Temperature Control mode** provides idealised control in that when the outdoor air conditions are not favourable, openings are closed. i.e. openings are closed when $T_{\text{outdoor}} > T_{\text{indoor}}$ to prevent introduction of warm air into the controlled zone. This may not be compliant with the TM59 standard because this type of an advanced control mechanism is not described in the standard which states simply that windows should be opened when $T_{\text{indoor}} > 22^{\circ}\text{C}$ and the room is occupied.

In summary TM59 requires that windows are open when the room is occupied and:

$$T_{\text{indoor}} > 22^{\circ}\text{C}$$

Whereas the default DesignBuilder rule opens windows when:

$$T_{\text{outdoor}} < T_{\text{indoor}} > 22^{\circ}\text{C}$$

Because the default DesignBuilder natural ventilation control rules do not comply exactly with the TM59 requirements, a **2-TM59 rules** natural ventilation control override option is provided to guarantee that the openings stay open even when the outdoor conditions are not favourable. The TM59 natural ventilation rules override can be applied from the CIBSE TM59 [Natural ventilation rules](#) setting. See screenshot below.

☒ CIBSE TM59

Version 2-CIBSE TM59:2026

☐ Vulnerable occupants

Natural ventilation rules 2-TM59 rules

☒ Restrict venting factor by schedule

Note that by default, this override will be applied only to zones using TM59 Activity templates (excluding corridors).

Temperature control

Default TM59 activity temperature set points and schedules are based on the UK NCM data. Heating set point schedules have been modified to use heating setback temperature instead of set point in a period from May to September.

The environmental controls will typically be configured based on the building design.

To reduce the chance of simultaneous heating and window opening, a control mechanism is implemented through the [Apply TM59 natural ventilation rules](#) calculation option. See discussion above under [HVAC tab](#).

Important note: The TM59 requirement to open the windows when the internal temperature rises above 22°C can easily lead to simultaneous heating and window opening. This is more likely to occur when:

- [Heating setpoint temperature](#) is close to the [Ventilation setpoint temperature](#)- Large openable areas are present in the zone
- Lower “Time steps per hour” is used for simulation

In general, to avoid simultaneous heating and venting, the heating setpoint temperatures should not be greater than 21°C when modelling for TM59 overheating assessments.

Openings Tab

- [Operation](#) – when using default templates, this input field should typically use the [Occupancy](#) schedule (so window opening is allowed only when the room is occupied). Set at Openings tab > External Windows > Operation. However, see also the guidance for [Restricting venting by schedule](#) above where fractional values for the Operation Schedule are used to define time-varying window opening fractions.
- [Free Aperture](#) – the window openable area can be defined using the [% Glazing area opens](#) setting. This input will typically have a significant impact on the natural ventilation fresh air flow rates. Set at Openings tab > External Windows > Free Aperture.

Operation	
Operation schedule	TM59_DoubleBed_Occ
Free Aperture	
Opening position	1-Top
% Glazing area opens	10.0
Discharge coefficient	0.650

Note: Blinds and shading devices should be modelled according to [TM59 chapter 3.3](#).

Internal openings

According to the TM59-2026 standard:

Internal doors have an opening regimen that differs for bedrooms and other spaces.

- Internal bedroom doors can be modelled as open or closed during the day. At night, i.e. between 23:00 and 08:00, bedroom doors should be modelled as closed.
- Internal doors to rooms other than bedrooms may be modelled as open or closed at any time.
- If the opening of internal doors would contravene fire safety requirements or other constraints, such as privacy, the affected internal doors should be modelled as closed.

To improve the flow of air through the airflow network, it will normally be beneficial to include internal openings such as doors, set to open based on the TM59 rules.

For any other details, guidance or restrictions regarding window and door opening refer to [TM59 chapter 3.4](#).

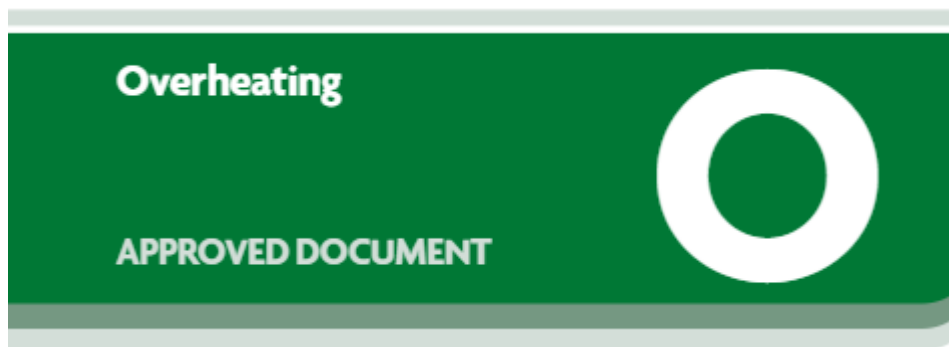
DHW cylinders, pipework, HIU and heat maintenance tape heat gains

Heat gains from DHW cylinders, heat interface units, distribution pipework and heat maintenance tape, need to be manually added to relevant spaces using internal gains Activity model data. More information can be found in [TM59 chapter 3.7.2](#).

Part-O Overheating Mitigation

The [Approved Document O: Overheating](#), Section 2: Dynamic thermal modelling assessment is based on TM59, but with some limitations as explained below under Ventilation Rules.

Ventilation Rules – Approved Document O (2021 Edition)



Approved Document O (2021) is based on TM59:2026 but it constrains some choices. CIBSE's TM59 method requires the modeller to make choices. The dynamic thermal modelling method Part-O applies limits to some of these choices, which are detailed in paragraph below. These limits should be applied when following the guidance in CIBSE's TM59.

All of the following limits on TM59:2026 apply.

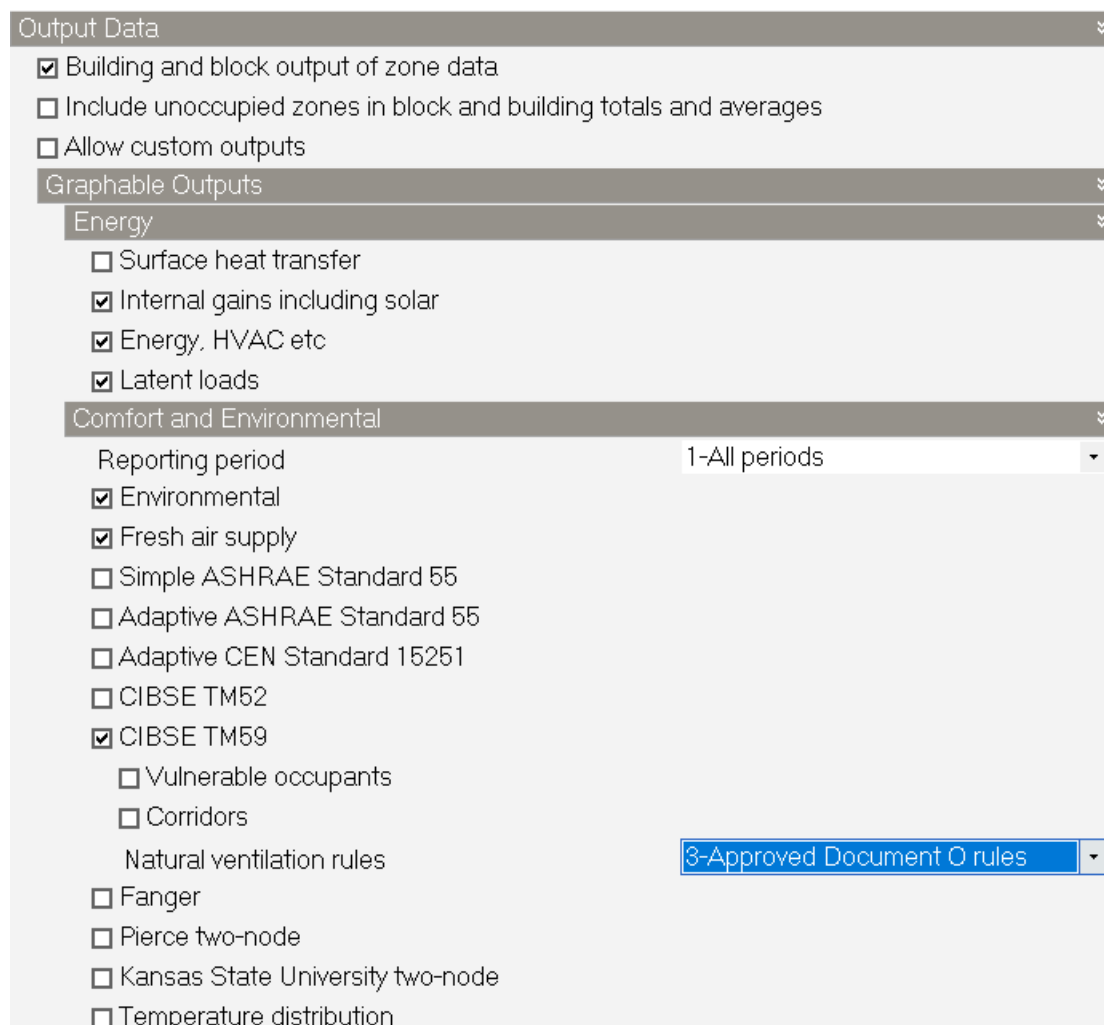
- When a room is occupied during the day (08:00 to 23:00), openings should be modelled to do all of the following.
 - Start to open when the internal temperature exceeds 22°C.
 - Be fully open when the internal temperature is at 26°C or higher.
 - Start to close when the internal temperature falls below 26°C.
 - Be fully closed when the internal temperature falls below 22°C.
- At night (23:00 to 08:00), openings should be modelled as fully open if both of the following apply.
 - The opening is on the first floor or above and not easily accessible.
 - The internal temperature exceeds 23°C at 23:00.
- When a ground floor or easily accessible room is unoccupied, both of the following apply.
 - In the day, windows, patio doors and balcony doors should be modelled as open, if this can be done securely, following the guidance in paragraph 3.7 below.
 - At night, windows, patio doors and balcony doors should be modelled as closed.
- An entrance door should be included, which should be shut all the time.
- The ventilation setpoint is forced to 22 °C regardless of the settings on the Activity tab, and openings must use the [2-Temperature Control mode](#).

TM59 Scotland

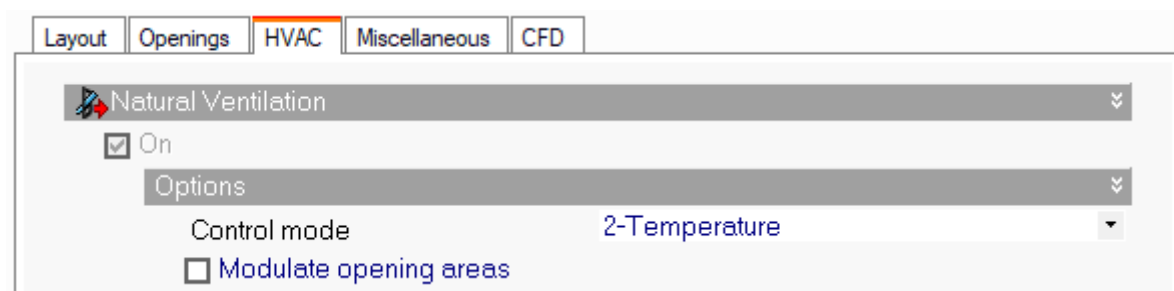
The **4-TM59 Scotland rules** [TM59 Output option](#) applies the same Approved Document O controls, but adapted for the Scottish standard, additionally setting the discharge coefficient to 0.62 and the [% Glazing area opens](#) to 100%. DesignBuilder applies these rules automatically.

Modelling Notes

- Custom controls cover points a. and b. above in the Ventilation Rules when the **3-Approved Document O rules** [TM59 Output option](#) is selected.

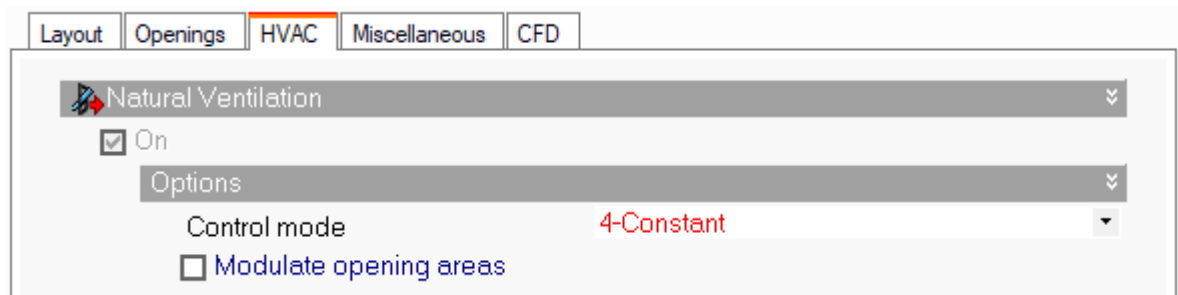


- The Ventilation setpoint is always modelled as 22°C, regardless of the setting made on the Activity tab.
- To apply Approved Document O control, openings must have their [Control mode](#) set to **2-Temperature**.

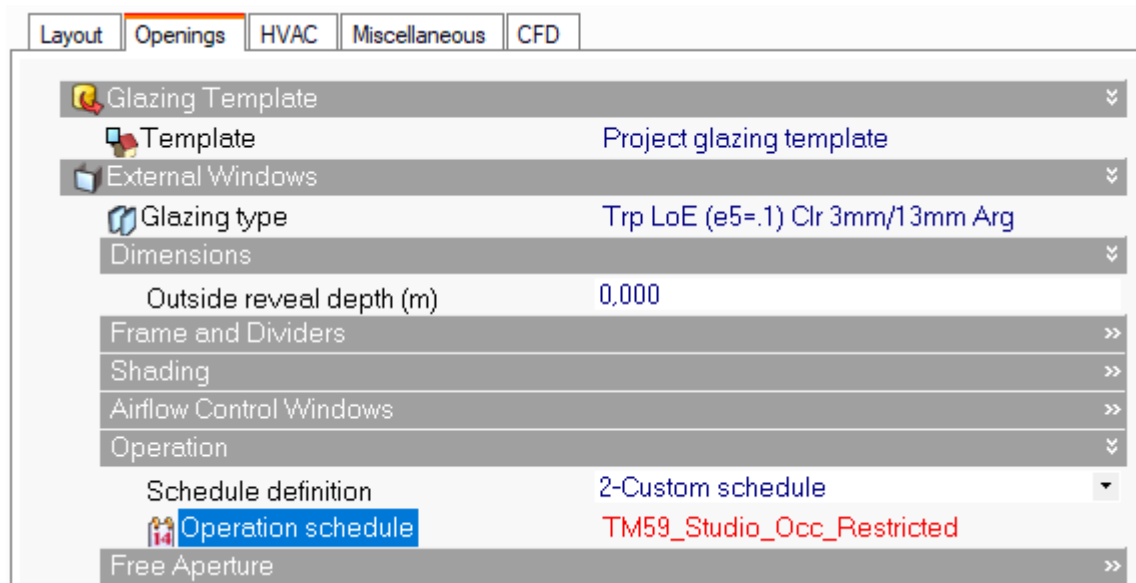


- To model simple open / closed operation without modulation, you can set [Control mode](#) to **4-Constant**. This control can be used to provide purge ventilation. In this case, if internal temperature > 22°C the windows are fully opened. However, please note that this operation mode does not comply

with Approved Document O, its purpose is informational only. Other control modes are not valid.



- Opening restriction during nighttime can be applied using a custom operation schedule, for example:



```
SCHEDULE:COMPACT,
TM59_Studio_Occ_Restricted,
Fraction,
Through: 31 Dec,
For: WinterDesignDay,
Until: 24:00, 0,
For: AllOtherDays,
Until: 08:00, 0,
Until: 23:00, 1,
Until: 24:00, 0;
```

Assessment Criteria (A To D)

Each zone is assessed against the criteria that apply to its Activity template. The reported criterion depends on whether the home is predominantly naturally ventilated or mechanically ventilated, and whether the zone is occupied and free-running (uncooled). In mechanically ventilated or cooled homes, bedrooms must pass both Criterion B and Criterion C. All four criteria are assessed over the period May to September.

By default the building comfort category is Category II, but the [Vulnerable occupants](#) option shifts the relevant criteria to Category I.

Criterion	Applies to	Basis	Pass condition
A	Living rooms, kitchens, bedrooms (nat-vent)	Adaptive exceedance (TM52 Criterion 1)	Hours where $\Delta T \geq 1$ K not more than 3% of occupied hours, May to Sep.
B	Bedrooms (all homes, inc.)	Night mean operative temperature	Mean sleeping-hours temperature exceeds T_n on not more than 4 nights.

Criterion	Applies to	Basis	Pass condition
	mech-vent / cooled)		
C	All occupied rooms (mech-vent / cooled)	Fixed temperature (CIBSE Guide A)	Operative temperature above 26 °C for not more than 3% of occupied hours. Bedrooms must also pass B.
D	Communal / circulation areas	Fixed temperature (28 °C)	Common circulation (TM59_CommonCirculationAreas): pass / fail. In-dwelling circulation (TM59_CirculationAreas): significant-risk flag.

3.1 Criterion A: living rooms, kitchens and bedrooms

Drawn from TM52 Criterion 1. For each occupied, non-cooled zone, the operative temperature is compared against the adaptive comfort limit T_{max} derived from the running mean outdoor temperature (EN 15251 Category II upper limit). The exceedance ΔT is the amount by which operative temperature exceeds T_{max} , rounded to the nearest whole degree per the TM52 convention.

The zone passes if the number of occupied hours with $\Delta T \geq 1$ K is not more than 3% of occupied hours over May to September.

New in 2026: where elevated local air speed is present, the comfort limit is raised by the air-speed uplift (+0.2 °C per 0.1 m/s, capped at +1.2 °C) before ΔT is computed, applied only at operative temperatures of 25 °C or above. For [vulnerable occupants](#) the limit is reduced by 1 K (Category I).

Reported for occupied, non-cooled zones on the TM59 bedroom, studio, and living/kitchen activity templates.

3.2 Criterion B: bedrooms at night

For bedrooms, the mean operative temperature over the sleeping window 23:00 to 08:00 is evaluated each night. A night counts as an exceedance when that mean exceeds the night threshold T_n .

- $T_n = 27$ °C for **Category II** (standard occupancy).
- $T_n = 26$ °C for **Category I** (vulnerable occupants).

The zone passes if the number of exceedance nights across May to September is not more than 4. Four nights is a pass; five or more is a fail. This replaces the 2017 annual-hours test (26 °C for no more than 1% of annual hours). The air-speed uplift does not apply to this criterion: the nightly mean is compared against T_n directly.

Reported for bedrooms on the TM59 bedroom and studio activity templates. In naturally ventilated homes bedrooms must pass both Criterion A and Criterion B. In mechanically ventilated or cooled homes bedrooms must pass both Criterion B and Criterion C: the night test is no longer skipped for these homes.

3.3 Criterion C: predominantly mechanically ventilated or cooled homes

Where the home is predominantly mechanically ventilated (determined from the HVAC tab), the fixed-temperature test from CIBSE Guide A applies. All occupied rooms pass if the operative temperature does not exceed 26 °C for more than 3% of occupied hours, assessed over May to September.

New in 2026: bedrooms in these homes must also pass Criterion B (the bedroom night mean). In 2017 the mechanically ventilated route relied on the fixed-temperature test alone; the night test now applies alongside it.

Also new: where elevated local air speed is present, the effective threshold is raised by +0.35 °C per 0.1 m/s, capped at +2.1 °C. This is a larger uplift than the adaptive credit used in Criterion A. Criterion B receives no uplift.

3.4 Criterion D: communal and circulation areas

Criterion D covers circulation and communal spaces in which the operative temperature shall not exceed 28 °C between 1st May and 30th September for more than 3% of occupied hours during this period.

Two activity templates drive it, and they are reported differently:

- **TM59_CommonCirculationAreas:** communal circulation shared between dwellings. Assessed as a pass / fail against Criterion D.
- **TM59_CirculationAreas:** circulation within a dwelling. Not a pass / fail; reported as a significant overheating risk flag where the 28 °C basis is exceeded.

Assign the correct template to each zone: the template determines whether the space returns a hard pass / fail or an advisory risk flag.

Running The TM59 Analysis

Calculation options - General

To obtain correct results it's necessary to run a full annual analysis (1st Jan – 31st Dec). However, before running whole year simulations, it is important to first run shorter simulations, checking hourly / sub-hourly results to ensure that the ventilation / environmental control works as intended.

Bear in mind that the purpose of a TM59 analysis is to assess the risk of overheating and that because of the specific ventilation requirements of TM59, the same model might not generate accurate energy consumption results. To help avoid any such issues, the **1-Standard EnergyPlus controls** option can be selected from the [Natural ventilation rules](#) calculation option drop list when running simulations where energy consumption is an important output.

Tip: Once you are happy that your model is running well and you are ready to generate TM59 results it is usually best to uncheck the [hourly and sub-hourly output interval](#) checkboxes on the Simulation Calculation options dialog to help keep result sets to manageable sizes.

Calculation options - Options

When using Calculated natural ventilation, you should use a simulation timestep of at least 6 timesteps per hour for accurate window opening control.

Calculation options - Simulation Manager

Using the Simulation Manager to run your TM59 simulations will enable you to continue working on the model while the simulation is running, and if necessary to run more than one simulation in parallel and compare the results side by side in the Results Viewer.

Note: To continue working in DesignBuilder or to run another simulation, the Simulation Manager dialog must first be closed.

Calculation options - Output

Results for the TM59 analysis can be obtained by selecting [CIBSE TM59](#) output on the Output tab > Graphable Outputs. Other settings (Mechanical ventilation, vulnerable occupants) are discussed below.

Edit Calculation Options

Calculation Options

General Options **Output** Simulation Manager

Output Data

☒ Building and block output of zone data

☐ Include unoccupied zones in block and building totals and averages

☐ Allow custom outputs

Graphable Outputs

Energy

☐ Surface heat transfer

☒ Internal gains including solar

☒ Energy, HVAC etc

☒ Latent loads

Comfort and Environmental

Reporting period 1-All periods

☒ Environmental

☒ Fresh air supply

☒ Simple ASHRAE Standard 55

☐ Adaptive ASHRAE Standard 55

☐ Adaptive CEN Standard 15251

☐ CIBSE TM52

☒ CIBSE TM59

Version 2-CIBSE TM59:2026

☐ Vulnerable occupants

Natural ventilation rules 2-TM59 rules

☒ Restrict venting factor by schedule

Note: In [TM59:2017](#) DesignBuilder exposed separate **Include cooled zones** and **Corridors** output options. These are no longer needed in TM59:2026 as cooled zones and corridors are always reported.

Basic CIBSE TM59 output

Select [CIBSE TM59](#) on the Output tab (Graphable Outputs) to generate the assessment outputs. Set the Mechanical ventilation and Vulnerable occupants options to match the scheme so the correct criteria and category are reported.

CIBSE TM59 Mechanical ventilation

DesignBuilder determines whether a building is predominantly mechanically ventilated through the settings on the HVAC tab and will add results according to [TM59 chapter 4.3](#) as described below.

CIBSE TM59 Vulnerable Occupants

Select this output option when modelling a dwelling intended for thermally sensitive or fragile occupants - including care homes, sheltered accommodation, and any building that provides an element of health care. In these cases the assessment should use Category I rather than the standard Category II.

TM59:2026 Output Report

While [TM59:2017 results](#) are viewed in the DesignBuilder Results Viewer, for TM59:2026 a dedicated plugin must be installed to view the results, and the Results Viewer **CIBSE TM59** tab used in earlier versions no longer applies.

With the plugin installed, the summary grid displayed in the report shows the pass / fail status for each zone against each applicable criterion.

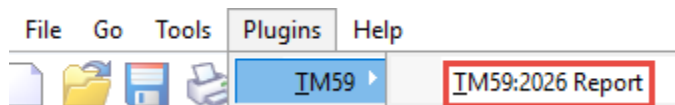
Install the plugin

Install the plugin by downloading it from the [Download > Scripts and Plugs](#) section of the DesignBuilder website.

Then follow the instructions in our [TM59:2026 Report KB article](#).

Generate reports

With the plugin installed, you can generate a report after the simulation has been run from the **Plugins > TM59 > TM59: 2026 Report** menu option.



Selecting this menu command opens the **TM59:2026 Report** dialog shown below.

TM59:2026 Report

×

Generates a CIBSE TM59:2026 overheating compliance report (PDF) by combining the current model with TM59:2026 simulation outputs (eplusout.eso).

Project

Project title

Model example

Phase

1

Project team

Modeller

Name

Company

Role

Reviewer / checked by

Name

Company

Role

Results file

☒ EnergyPlus folder

C:\Users\AndyT\AppData\Local\DesignBuilder\EnergyPlus\eplusout.eso

...

☐ Simulation Manager results

Refresh jobs

Model	Description	Started	Results
Model for TM59 Parametric analysis with...	London 90ori uninsulated	16-07-2026 10:37:05	Found
Model for TM59 Parametric analysis with...	London 90ori uninsulated	15-07-2026 16:35:41	Found
Model for TM59 Parametric analysis with...	London 90ori uninsulated	14-07-2026 19:02:39	Found

Output (PDF)

C:\Users\YourName\OneDrive - Designbuilder Software Ltd\Documents\TM59_2026_Report.pdf

...

☒ Open when done

Zone evaluation

Standard zones; communal circulation areas - corridors, stairwells, lift lobbies etc. - are always assessed under Criterion D: mandatory pass/fail for TM59_CommonCirculationAreas, advisory risk flag for TM59_CirculationAreas.

Zone	Activity template	Mech. vent.	Active cooling	Evaluation
00 Ground Floor:Bedroom1	TM59_SingleBedroom	No	No	Natural (Criteria A + ... ▾)
00 Ground Floor:Bedroom2	TM59_SingleBedroom	No	No	Natural (Criteria A + ... ▾)
00 Ground Floor:LivingKitchen	TM59_2-BedLivingKitchen	No	No	Natural (Criteria A + ... ▾)

Note: the model may have changed since the simulation run (model last saved 27-02-2024 15:15; results produced 16-07-2026 10:42). Zone areas, activity templates and other model-sourced supporting information may no longer match the simulated case; overheating criteria and pass/fail results come from the simulation and are unaffected.

Generate report

Close

The dialog consists of several sections:

1. **Project** section: enter the Project title and the Phase.
2. **Project team** section: enter the Name, Company, and Role of the Modeller, and the same for the Reviewer.
3. **Results file** section: select whether the results will be loaded from the EnergyPlus folder or the Simulation Manager.
4. **Output (PDF)** section: select the name of the PDF file you want to create and whether you want it to open after the report has been generated.

5. **Zone evaluation** section: select whether the evaluation is based on natural or mechanical ventilation for each zone listed in the grid.

Once all the data has been entered, click the **Generate Report** button, and the PDF file will be created. If **Open when done** was selected, the report will automatically open in the application assigned to open PDF files on your system.

An example of the summary page is displayed below.

TM59:2026 Report

Overheating risk assessment
May to September

Model example

Model example



Summary

CIBSE TM59:2026 assesses predominantly naturally ventilated zones against Criteria (a) and (b), and predominantly mechanically ventilated or cooled zones against Criteria (b) and (c). Threshold temperatures below reflect the selected **Category II** occupant category:

Criterion (a) — for living rooms, kitchens and bedrooms, the number of occupied hours during which ΔT (operative temperature minus the adaptive comfort threshold T_{max}) is greater than or equal to 1K over May to September shall not exceed 3% of occupied hours. Living rooms, kitchens and home offices are assumed occupied from 09:00 to 22:00 (13 hours a day, 1989 occupied hours over May to September), so the maximum permitted exceedance is **59 hours**; bedrooms may be occupied at any time of day or night (3672 hours), so their maximum is **110 hours**. Elevated air-speed (ceiling fan) adjustments to T_{max} are applied in the simulation.

Criterion (b) — for bedrooms, the number of nights on which the average operative temperature between 23:00 and 08:00 exceeds 27°C shall not exceed 4 nights.

Criterion (c) — for predominantly mechanically ventilated or cooled zones, the operative temperature shall not exceed 26°C for more than 3% of occupied hours — the same fixed occupied-hours basis as criterion (a), giving maximum permitted exceedances of **59 hours** (living rooms, kitchens, home offices) and **110 hours** (bedrooms).

Criterion (d) — for communal areas outside the living areas (circulation spaces, corridors, stairwells, lift lobbies and similar), whether naturally or mechanically ventilated and/or cooled, the number of assessment-period hours during which the operative temperature exceeds 28°C shall not exceed 3% of occupied hours. Communal areas may be occupied at any time of day or night (3672 hours), so the maximum permitted exceedance is **110 hours**. For **common circulation areas** (*TM59_CommonCirculationAreas*) this is a mandatory pass/fail requirement that counts towards the overall verdict; for other circulation areas (*TM59_CirculationAreas*) it is reported as an advisory significant-risk flag only.

Zone name	Ventilation	Criterion (a) naturally ventilated	Criterion (b) bedrooms	Criterion (c) mech. vent. / cooled	Criterion (d) communal areas
00 Ground Floor:Bedroom1	Natural	Pass	Pass	—	—
00 Ground Floor:Bedroom2	Natural	Pass	Pass	—	—
00 Ground Floor:LivingKitchen	Natural	Fail	—	—	—
00 Ground Floor:Corridor 1	Natural	—	—	—	Pass

Simulation Summary Output

You can display a summary of results generated by EnergyPlus in tabular form on the Summary tab of the Simulation screen.

End Uses By Subcategory Table

DesignBuilder uses its own electric equipment end use codes to link gains reported in EnergyPlus outputs to the zones in which the equipment is installed. The screenshot below illustrates how the codes link to the various types of equipment defined.

End Uses By Subcategory

	Subcategory	Electricity [W]	Natural Gas [W]	Propane [W]	District Cooling [W]	District Heating [W]	Water [m3/s]
Heating	General	0.00	0.00	0.00	0.00	69153.07	0.00
Cooling	Lighting	0.00	0.00	0.00	142094.62	0.00	0.00
Interior Lighting	ELECTRIC EQUIPMENT#1FFirstFloorEast:OpenOffice#GeneralLights	3354.10	0.00	0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorWest:OpenOffice#GeneralLights	3354.10	0.00	0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorWest:OpenOffice#GeneralLights	3354.05	0.00	0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorEast:OpenOffice#GeneralLights	3354.05	0.00	0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#CoreCirculation:Circulation#GeneralLights	507.83	0.00	0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#AtriumReception:Reception#GeneralLights	447.59	0.00	0.00	0.00	0.00	0.00
Exterior Lighting	General	0.00	0.00	0.00	0.00	0.00	0.00
Interior Equipment	ELECTRIC EQUIPMENT#1FFirstFloorEast:OpenOffice#04	3.00	#04 Catering gains		0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorEast:OpenOffice#02	2.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorEast:OpenOffice#05	5.00	#05 Office equipment		0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorEast:OpenOffice#03	4.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorWest:OpenOffice#05	6.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorWest:OpenOffice#04	3.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorWest:OpenOffice#02	2.00	#02 Miscellaneous gains		0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorWest:OpenOffice#03	4.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#1FFirstFloorWest:OpenOffice#07	958.31		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorWest:OpenOffice#05	6.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorWest:OpenOffice#04	3.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorWest:OpenOffice#02	2.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorWest:OpenOffice#03	4.00	#03 Process gains		0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorWest:OpenOffice#07	718.73		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorEast:OpenOffice#05	6.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorEast:OpenOffice#04	3.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorEast:OpenOffice#02	2.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorEast:OpenOffice#03	4.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#0FGroundFloorEast:OpenOffice#07	239.58	#07 Aux energy. eg. Ventilation and pumps		0.00	0.00	0.00
	ELECTRIC EQUIPMENT#CoreCirculation:Circulation#05	5.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#AtriumReception:Reception#05	5.00		0.00	0.00	0.00	0.00
	ELECTRIC EQUIPMENT#AtriumReception:Reception#07	223.79		0.00	0.00	0.00	0.00
Exterior Equipment	General	0.00		0.00	0.00	0.00	0.00
Fans	General	0.00		0.00	0.00	0.00	0.00
Pumps	General	0.00		0.00	0.00	0.00	0.00

Parametric Analysis

DesignBuilder can automatically run multiple simulations automatically adjusting up to 2 variables to create design curve outputs. Design curves can be very useful as part of the design process to understand how building performance is affected by variations in the building configuration and operation. DesignBuilder allows you to generate curves to show how performance parameters like energy use, cost and comfort vary as key elements of the building design are adjusted. Up to 2 design variables can be selected. For example you might want to plot the cooling demand for the base design with a range of window to wall % and glazing types variations. The design variables to be used can be selected on the [Variables](#) tab of the "Parametric Analysis - Parametric Settings Dialog" on page 1670 dialog.

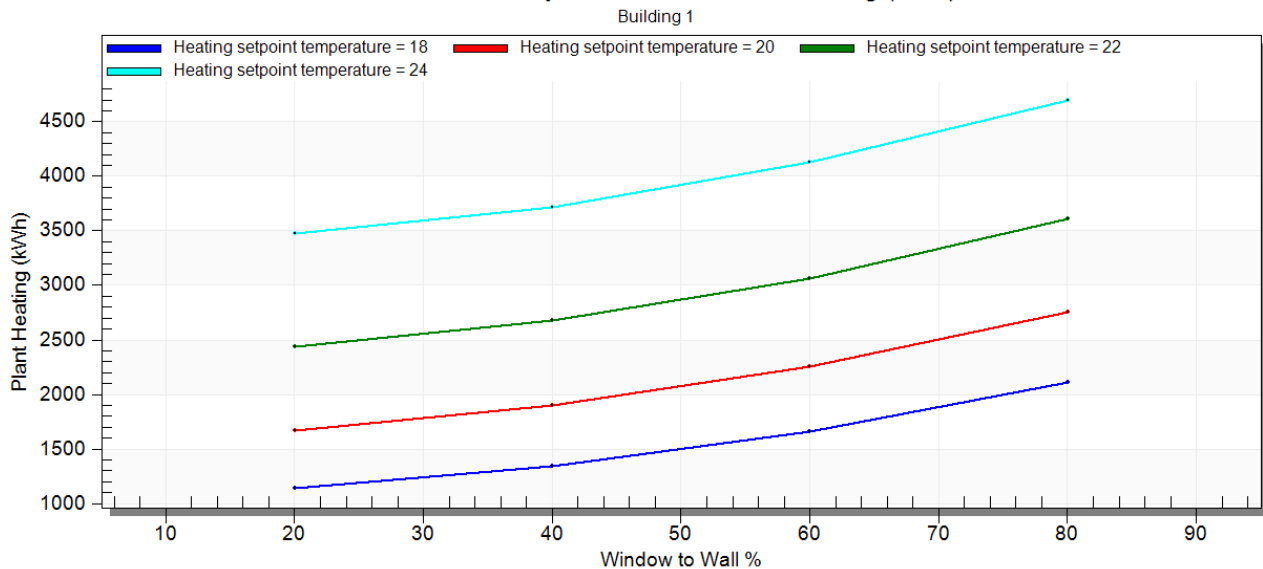
Click on the **Parametric** tab on the Simulation screen to view parametric results and carry out any simulations required.

The design curves are generated by automatically running multiple simulations of copies of the base design (i.e. the model defined on the edit screen) with the design variations applied.

Note: It is necessary to run the base simulation with monthly/annual outputs before opening the Parametric tab. The parametric simulations will all be run using the base calculation and model options. **The simulation period and all other options used in the parametric analysis is the same as that defined on the [Simulation calculation options](#) dialog.**

How many simulations are required? The number of design variants that are tested for 2 variable tests is the number of variants for the first variable multiplied by the number for the second variable. The variations in the example below would be different combinations of Heating setpoint temperature and Window to wall %. In all, $6 \times 5 = 30$ simulations would be required to generate the design curves in this example.

Parametric Analysis Results - Plant Heating (kWh)



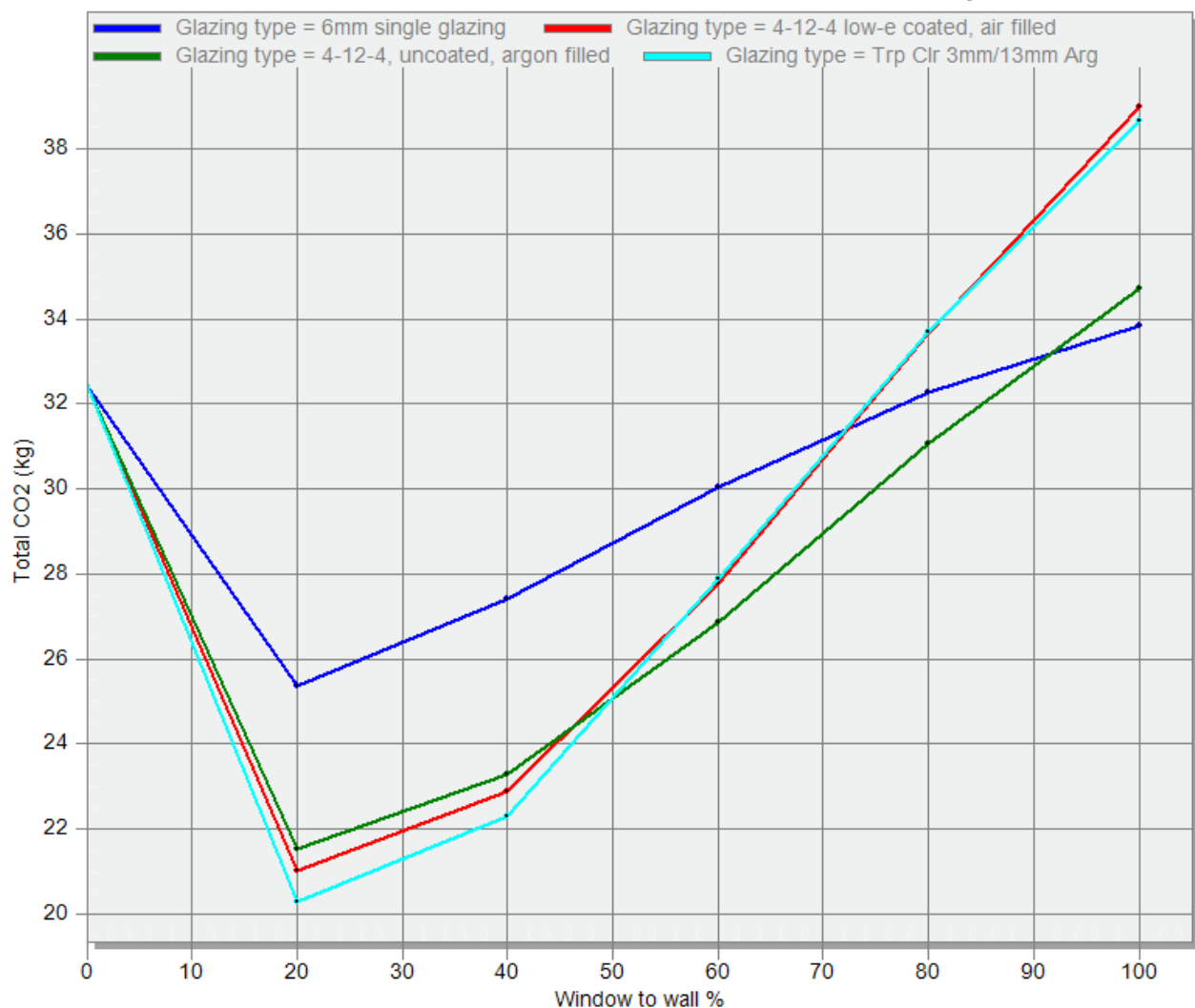
Impact of varying WWR and Heating setpoint temperature on Annual heating load.

Total CO2 (kg) - Untitled, Building 1

EnergyPlus Output

20 Mar - 26 Mar, Parametric Analysis

Licensed



The image above shows a parametric analysis of total annual carbon emissions where glazing percentage is varied for 4 different glazing types. The building has lighting control, heating and cooling and results illustrate how in some cases optimum configurations can be identified using the parametric analysis approach.

Typical applications for parametric simulation are:

- Generating design curves to assist with decision making at early design stages.
- As a teaching aid to help students understand the impact of variations in the design on building performance.
- To help communicate concepts with non-experts in building simulation, clients, architects etc. Architects often appreciate the visual nature of parametric data displayed in this way.
- Design optimisation.

Parametric Analysis Contents

- "Parametric Analysis - Parametric Settings Dialog" on page 1670
- "Parametric Analysis Calculation Options" on page 1675
- "Parametric Analysis Results" on page 1677
- "Parametric Analysis Limitations" on page 1679
- "Parametric Analysis Tutorials" on page 1679

Parametric Analysis - Parametric Settings Dialog

The **Parametric Settings** dialog can be accessed by clicking on the **Parametric settings** icon  on the toolbar.

You can use this dialog to set up the parametric analysis problem by entering data on the tabs described below:

- [Analysis Type](#) - set the Analysis type
- [Outputs](#) - Add as many outputs as required to show how various aspects of building performance are affected by the parametric changes to the model.
- [Design Variables](#) - the elements of the model that are to be allowed to vary during the parametric analysis and the new values that these elements may take. When used for Parametric Analysis only the top 2 variables on the list are used as parametric analysis only supports a maximum of 2 variables.

Analysis Type Tab

There are 3 types of analysis and these can be set from the first tab on this dialog

Analysis

The analysis to select for parametric analysis is **1-Parametric analysis**.

Outputs Tab

The outputs tab allows you to define the outputs to be generated for each iterative design variation simulated. This can help in the final analysis of results to gain a clearer understanding of how variations in the design will impact on the building performance.

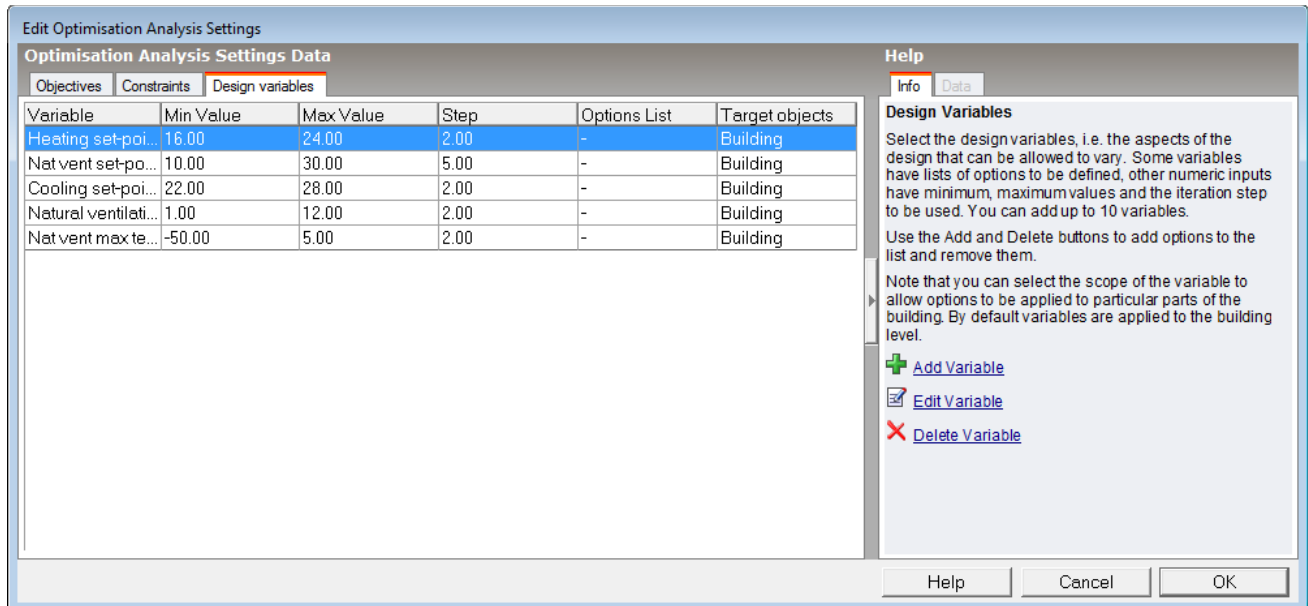
Any number of outputs can be added.

Controls on the Info panel are:

- **Add Output:** Creates a new output (using the [Edit Analysis Outputs](#) dialog) where you can enter information on that output (name and KPI).
- **Edit Output:** Edits the selected existing output from the table (using the [Edit Analysis Outputs](#) dialog).
- **Delete Output:** Deletes the selected existing output from the table.

Variables Tab

The variables tab allows you to define the elements of the building design that can be allowed to vary as well as the allowed values that the variable can take on during the analysis. You can select from a wide range of variables such as **Window to wall %**, **Heating setpoint temperature**, **HVAC Template**, **External wall construction** etc.



List data types such as constructions, glazing or templates require a set of options to be selected to define the allowed variable options.

Controls on the Info panel are:

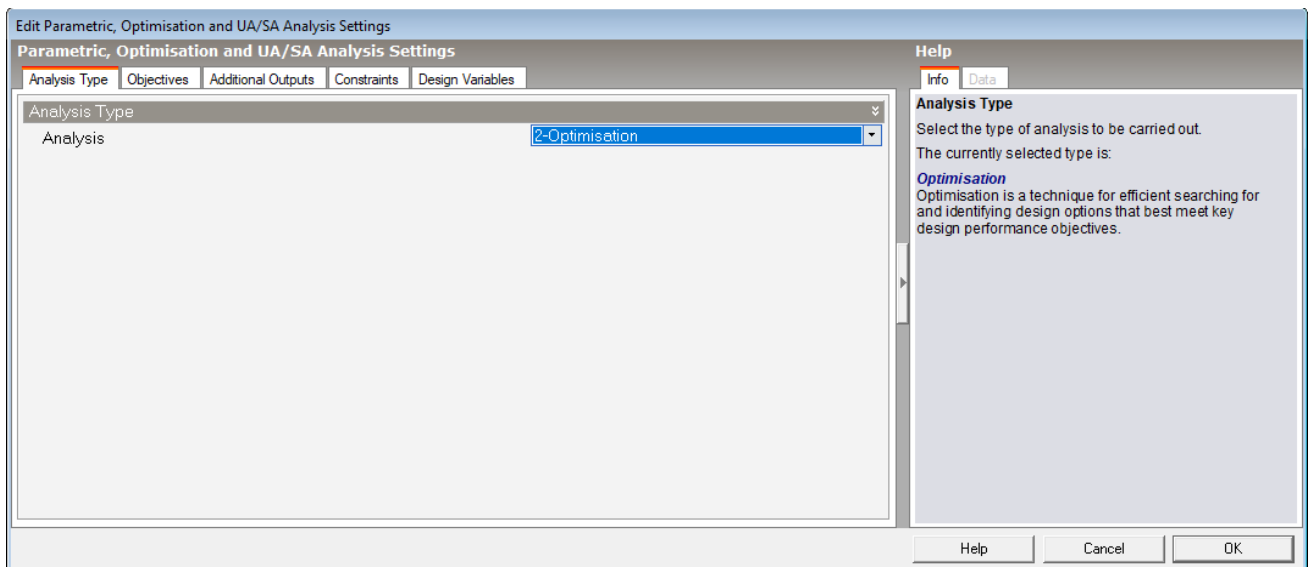
- **Add Variable:** Creates a new Design Variable (using [Edit Analysis Variable](#) dialog).
- **Edit Variable:** Edits the selected existing Design Variable from the table (using [Edit Analysis Variable](#) dialog).
- **Delete Variable:** Deletes the selected existing Design Variable from the table.

Parametric Analysis - Analysis Type

Accessed from [Parametric Settings](#) dialog

Analysis Type

Data on the **Analysis Type** tab allows you to define the type of analysis.



Analysis

The following options are available:

- **1-Parametric analysis** - to run a parametric analysis with up to 2 [design variables](#) and any number of [Output KPIs](#) on the Y-axis.
- **2-Optimisation** - to run an optimisation with 1 or 2 design objectives and any number of [design variables](#), [constraints](#) and [additional outputs](#).
- **3-Uncertainty/Sensitivity analysis** - to run an uncertainty and sensitivity analysis (UA/SA) with any number of [outputs](#) and [design variables](#). See the specific [section on UA/SA](#) for more details on this option.

Note: Optimisation and UA/SA results and simulation tools are accessed from the **Optimisation + UA/SA** tab on the Simulation screen, while Parametric analysis results and simulation tools are accessed from the **Parametric** tab.

Parametric Analysis - Outputs Dialog

Accessed from [Parametric Settings](#) dialog

Outputs selected for parametric analysis can be displayed on the Y-axis in subsequent analysis to quantify how various aspects of building performance are affected by the parametric changes to the model.

Name

Your name for the output for your reference, e.g. "Total Lighting".

Output KPI

Select a predefined or custom [Key Performance Indicator \(KPI\)](#) from the browse list.

Parametric Analysis - Design Variables Dialog

Accessed from [Parametric Settings](#) dialog

Design Variable

Variable type

Type of input Design Variable, to be selected from the browse list in the right-hand info panel.

Tip: See the "Optimisation - Design Variables Dialog" on page 1710 help page for a full list of the available [Variable types](#).

Note: All variables are listed for selection regardless of the state of the model so care must be taken to select variables that are appropriate. For example, if the Scheduled natural ventilation model option is used then changes in the **% External window opens** variable (which only applies to Calculated natural ventilation) won't affect results.

Min value

The lowest value that this numeric design variable can take on. This option is only visible when a numeric variable type has been selected (above).

Max value

The highest value that this numeric design variable can take on. This option is only visible when a numeric variable type has been selected (above).

Step (parametric)

Enter the step between possible variable options. This option is only visible when a numeric variable type has been selected (above).

The possible values for the variables are Min value, Min value + Step, Min value + 2 x Step, ... MaxValue. For example if Min value is 16, Step (optimisation) is 2 and Max value is 24, then possible values for this numeric design variable are: 16, 18, 20, 22 and 24.

Step (optimisation)

Not used for parametric analysis.

Options list

Click on the browse button to open a list of components or templates for selection. Items can be selected by checking the appropriate checkboxes. Each item selected will be an option that can be applied during the optimisation process.

When a template is selected as the variable type, one of the selected templates from the list are loaded in each iteration of the optimisation analysis.

This option is only visible when a list variable type has been selected (above).

After the list option selection process has been completed the **Variable Option Order** dialog is displayed. See below.

Important Note: While any number of variables can be added, only the top 2 in the list will be used in the parametric analysis.

List Variable Option Ordering

During the process of using the [Options list](#) browse button for selecting discrete design variables, a **Variable Option Order** dialog appears after selecting the list options to use and closing **Select design options** dialog. In this **Variable Option Order** dialog, you can define the order in which the variable options are listed. Depending on the case it may or may not be important to list the options in the right order.

If the list of components or templates represents some sort of numerical sequence then it is generally important to set the order of those options in an ascending or descending order, otherwise it is not normally necessary.

- **Example where ordering is important:** A case where a glazing type variable options being selected consist of incrementally varying U-value, such as a list of 4 glazing systems with U-values of 1.0, 1.5, 2.0 and 2.5 W/m²-K. In this case, ordering the selection (in ascending or descending order) will help ensure that parametric analysis results are displayed with variable options in a controlled order.
- **Example where ordering is NOT important:** A case where HVAC System Template options being selected have completely different HVAC systems such as a list of 4 HVAC templates, Fan coil units, Hot water radiators with nat vent, VAV and reversible chilled/heated floor. In this case there is no clear sequence, therefore ordering the list may not be important for parametric analysis.

Note: In some projects, it may be necessary to set up a design variable that changes an attribute that is also changed by another template type variable. In such cases you should ensure that the template variable is defined first, i.e. it appears higher in the variables list. For example, if the Design variables list includes parametric changes for both Construction templates and Infiltration, the Construction template variable should be higher on the list than the Infiltration variable for the infiltration variable to have any effect on the analysis.

Target

Target objects

This option allows you to select one or more object(s) to which the variations are to be applied. The object(s) you select here are the places in the model where the variations will actually be applied. Normal model data inheritance rules apply so if you set the building as the target then the change will set for the building but will also inherit down to block, zone, surface and opening levels where appropriate. In this case any hard set data at block or lower levels will prevent inheritance in the normal way.

Tip: For models imported via [gbXML](#) any construction or glazing assignments are set in the DesignBuilder model at surface level and so the normal inheritance paths used for construction and glazing will be overridden. Therefore in this case, if you need to parametrically adjust construction or glazing surface properties you may need to apply the changes to each surface individually by selecting the appropriate surfaces as targets.

Key

Variable key

This data is available for variables of the **Custom IDF** and **Custom Script** categories. Enter the name of the variable to be used in one or more corresponding DB Etl / EMS [<Var, VarName, DefaultValue>](#) statements. The variable key entered here corresponds with the **VarName** value in the DB Etl script.

Parametric Analysis Calculation Options

To start a parametric analysis go to the [Simulation screen](#) and click on the **Parametric** tab. Calculations are updated using the Update toolbar icon in the usual way. The data on the Parametric Analysis Calculation Option dialog is used to provide options to control the analysis as well as outputs to help you to understand the progress being made as the calculations continue.

Note: It is necessary to run the base simulation with monthly/annual outputs before opening the **Parametric** tab. The parametric simulations will all be run using the base calculation and model options. **So the simulation period and all other options used in the analysis are the same as those defined on the [Simulation calculation options](#) dialog.**

There are 2 tabs on the Dialog:

- "Parametric Analysis Calculation Options - General Tab" on page 1675
- "Parametric Analysis Calculation Options - Simulation Manager Tab" on page 1677

Parametric Analysis Calculation Options - General Tab

Accessed from [Parametric Analysis Calculation Options Dialog](#)

The parametric analysis management algorithms used by DesignBuilder require a number of settings as described below. These and other settings are made on the General tab of the Calculation Options dialog as described below.

Calculation Description

Calculation description

Enter a description for the calculation. This text will be used to identify the results on the graph and any other related outputs.

From

The **Start date** set up for the base simulation on the analysis screen is displayed but cannot be edited here.

To

The **End date** set up for the base simulation on the analysis screen is displayed but cannot be edited here.

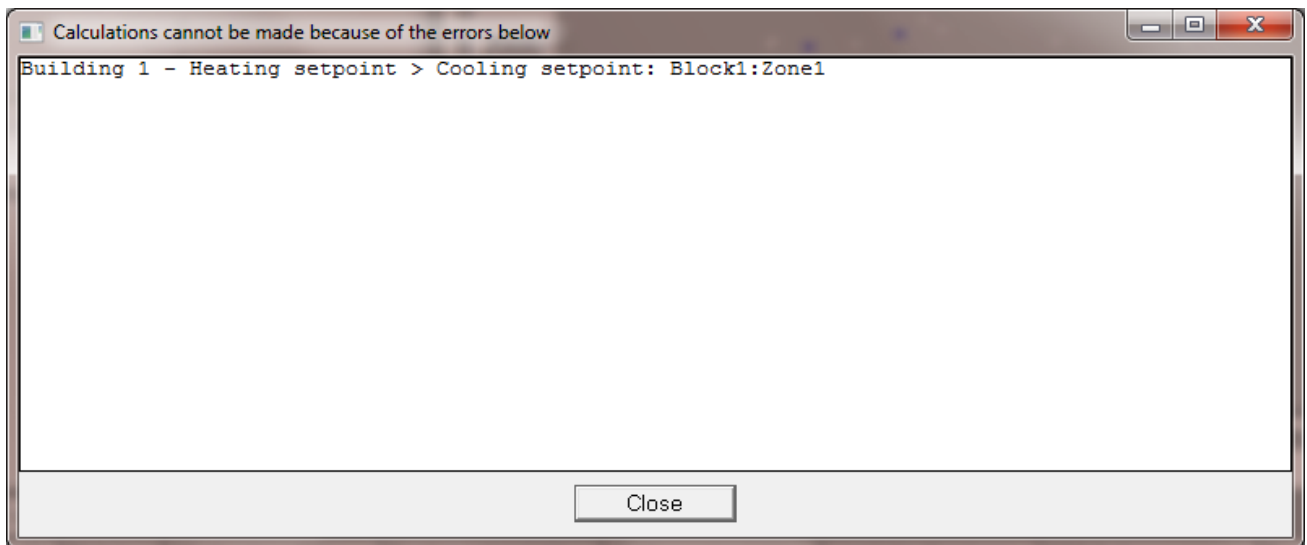
Parametric Options

Parametric output

Select the output to be displayed in the graph on the right. The list of available outputs will match the list of output KPIs previously set up on the [Outputs](#) tab of the Parametric Settings dialog. This selection does not affect the results that are generated.

Pause on errors

It is important to understand the cause of any errors that occur during a parametric analysis. The **Pause on errors** option, when checked, causes a message to be displayed and the calculations are paused. For example if there is an overlap in heating and cooling setpoint temperatures, in configurations where the heating setpoint is higher than the cooling setpoint the following error message is displayed:



Once you are clear about the cause of any errors you may want to cancel the analysis or you may decide to continue but to stop seeing reports and allow the calculations to continue. To do this simply uncheck the **Pause on errors** check box. Simulations with errors are displayed in red in the grid as shown below.

Iteration	Generati...	Status	Window to wall %	Cooling set-point te...	Heating set-point te...	Local shading type	CO2 (kg)	Discomfort ASHRA...
72	5		43,000	25,300	23,400	1.0m Overhang	22915,129	665,5000
71	5		23,000	25,600	16,700	0.5m Overhang	19739,116	1800,5000
70	5		42,000	25,900	17,100	2.0m Overhang	19846,498	1927,0000
69	5		60,000	24,800	21,400	2.0m Overhang	22878,883	1312,0000
68	5		41,000	26,400	23,300	2.0m Overhang	21801,074	900,5000
67	5	Calculation error	38,000	22,400	22,600	1.5m Overhang	1000000000,000	8761,0000
66	5	Optimal Designs	41,000	27,400	22,600	2.0m Overhang	20992,442	1214,0000
65	5		49,000	24,400	20,900	2.0m Overhang	22200,984	1297,5000
64	5		62,000	27,400	20,000	1.0m Overhang	21769,323	2240,5000

Common causes of errors are:

- Invalid combinations of variables such as heating setpoint > cooling setpoint. These errors are trapped by DesignBuilder before a simulation is run.
- Failure of a simulation due to errors in input that were not trapped by DesignBuilder.
- A severe EnergyPlus error such as simulation instability.
- A failure in the simulation infrastructure, e.g. a network failure when using a network simulation server.

Quietly retry if server is unavailable

Normally, if DesignBuilder is unable to access the requested Simulation Server, it will ask you whether you would like to **Abort**, **Retry** or **Ignore** the missing server. Selecting **Abort** cancels the whole analysis. Selecting **Retry** or **Ignore** instructs the software to keep trying. The latter option would be selected if the server became temporarily unavailable, perhaps due to a network error or because the server was down for a while.

However, if you are running a long analysis and you don't want to be prompted in this way due to a temporary glitch in the availability of a remote simulation server then you should consider checking this option. The effect is equivalent to pre-selecting the **Retry** or **Ignore** options in advance.

Advanced

Generation population size

The number of solutions to be evaluated in each iteration. More design variables requires larger population size in general. The choice is also influenced by the number of processor cores you have for running these simulations.

Retain Simulation Manager results

This setting allows advanced users to store the results for all parametric simulations on the Simulation Manager JobServer. Note that large amounts of data can be generated and stored and this will quickly fill up your disk if too many simulations are allowed to run. **Use this option with care!**

Tip: You must use the [Simulation Manager](#) to clear out all results once you have finished with them.

Parametric Analysis Graph

The results for the parametric analysis are displayed on the graph once the simulations are all complete and all simulation outputs have been loaded.

Parametric Analysis Calculation Options - Simulation Manager Tab

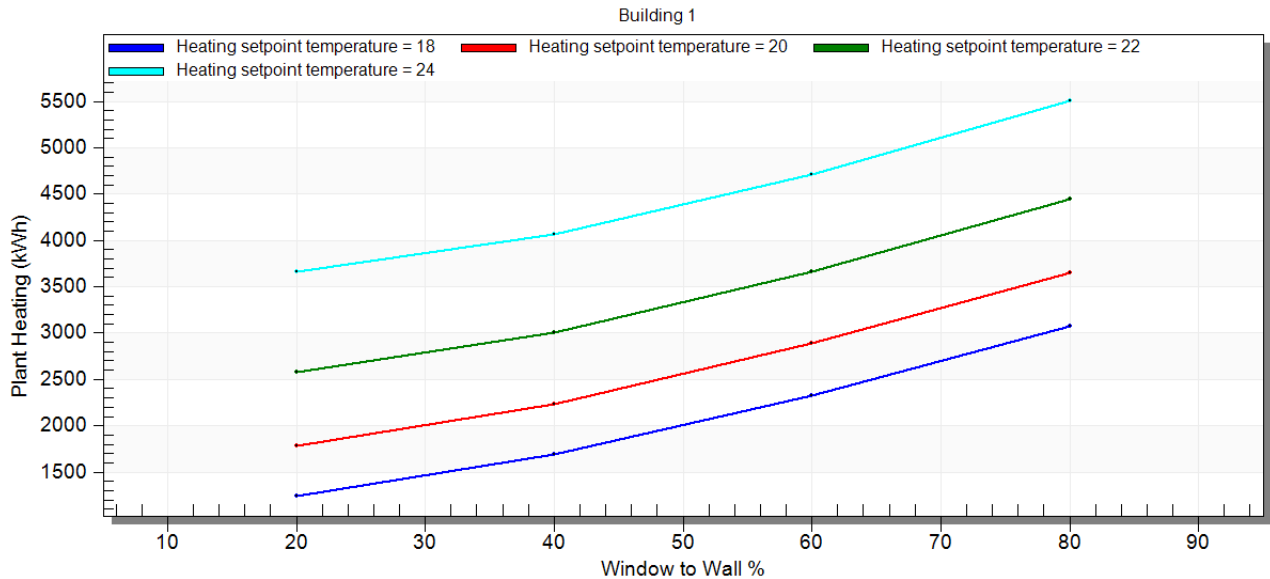
Accessed from [Parametric Analysis Calculation Options Dialog](#)

See "Optimisation Calculation Options - Simulation Manager Tab" on page 1729 for details on how to configure the Simulation Manager settings for parametric analysis.

Parametric Analysis Results

The Parametric analysis results allow you to view the outputs of the analysis in the form of both design curve graphics and a full listing of all results for each design iteration tested.

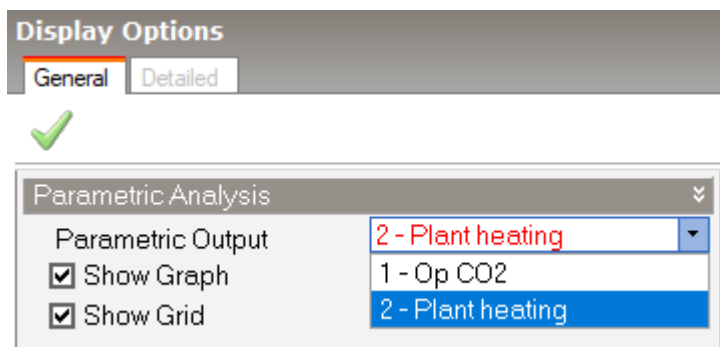
Parametric Analysis Results - Plant Heating (kWh)



Iteration	CO2 (kg)	Plant Heating (kWh)	Window to Wall %	Heating setpoint temperature (°C)
1	8453.906	5505.788	80.000	24.000
2	7886.734	4716.211	60.000	24.000
11	7310.698	4070.565	40.000	24.000
15	6753.802	3663.880	20.000	24.000
3	8192.881	4449.802	80.000	22.000
10	7641.004	3667.230	60.000	22.000
13	7072.177	3009.674	40.000	22.000
5	6527.576	2580.178	20.000	22.000
9	7977.524	3654.048	80.000	20.000
12	7435.545	2896.073	60.000	20.000
6	6877.030	2233.428	40.000	20.000
7	6337.176	1785.570	20.000	20.000
14	7816.607	3070.946	80.000	18.000
4	7286.651	2328.581	60.000	18.000

Parametric Analysis Display Options

Once the parametric analysis simulations have completed and the Calculation Options dialog has been closed the results are loaded to the main screen for review. There are some simple settings to allow you control the results to be viewed as described below.



Parametric output

Select the output to be displayed in the graph on the right. The list of available outputs will match the list of output KPIs previously set up on the [Outputs](#) tab of the Parametric Settings dialog.

Show graph

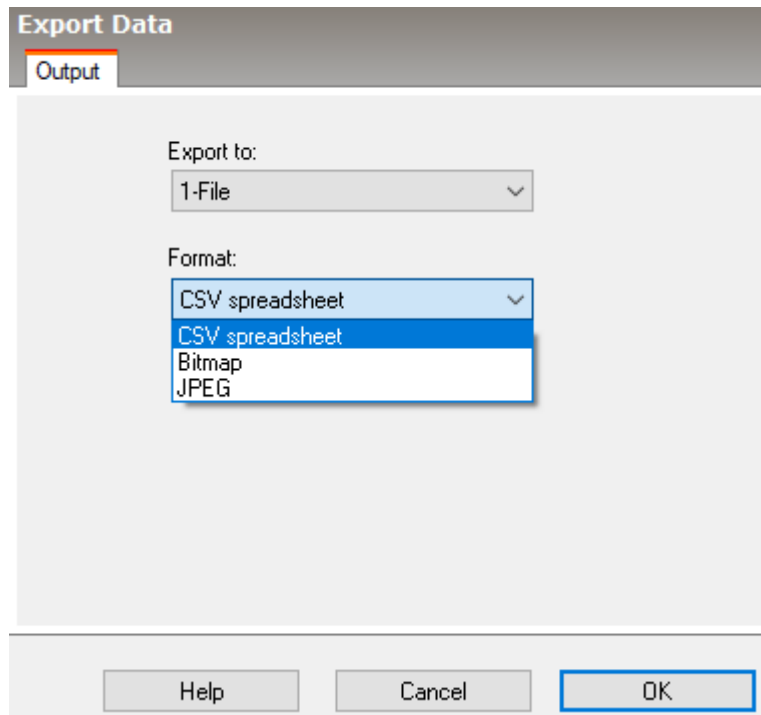
Check this checkbox to show the design curve graphic on the main screen to the right.

Show grid

Check this checkbox to show the grid with all the results on the main screen to the right.

Export Results

You can export the parametric analysis results by clicking on the Export  toolbar icon which opens the [Export dialog](#).



Here you can select whether to export the design curve graphic in Bitmap or JPEG form, or alternatively to export the underlying data itself as a csv file for further analysis in a spreadsheet.

Parametric Analysis Limitations

Current versions of DesignBuilder have these limitations related to Parametric analysis:

1. Parametric Analysis only allows a maximum of 2 design variables to be used. However you can workaround this limitation if you need to include more than 2 design variables. See the [Settings to Run all Possible Combinations](#) in the Optimisation help section.
2. You must run a simulation with monthly/annual output for the model (on the Analysis tab) before clicking on the Parametric tab. If you don't have any monthly/annual results on this tab then you must re-run the base simulation before updating the Parametric analysis.
3. It isn't possible to store multiple parametric results sets within the DesignBuilder model.

Parametric Analysis Tutorials

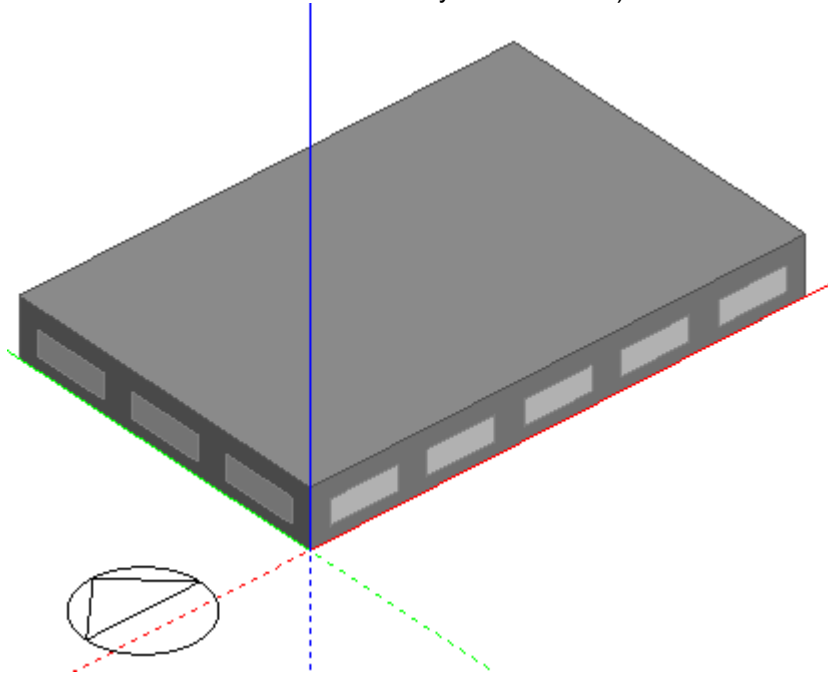
- "Parametric Analysis - Basic Tutorial" on page 1679
- "Parametric Analysis Tutorial - Future Climate Study" on page 1684
- "Parametric Analysis Tutorial - Custom Script KPI - TM59 Comfort" on page 1690
- "Parametric Analysis Tutorial - Custom Script Variable - Advanced Parametric Facade" on page 1698

Parametric Analysis - Basic Tutorial

This tutorial will help you get up to speed with running a parametric analysis study in DesignBuilder in a few easy steps.

Step 1 - Create a base model

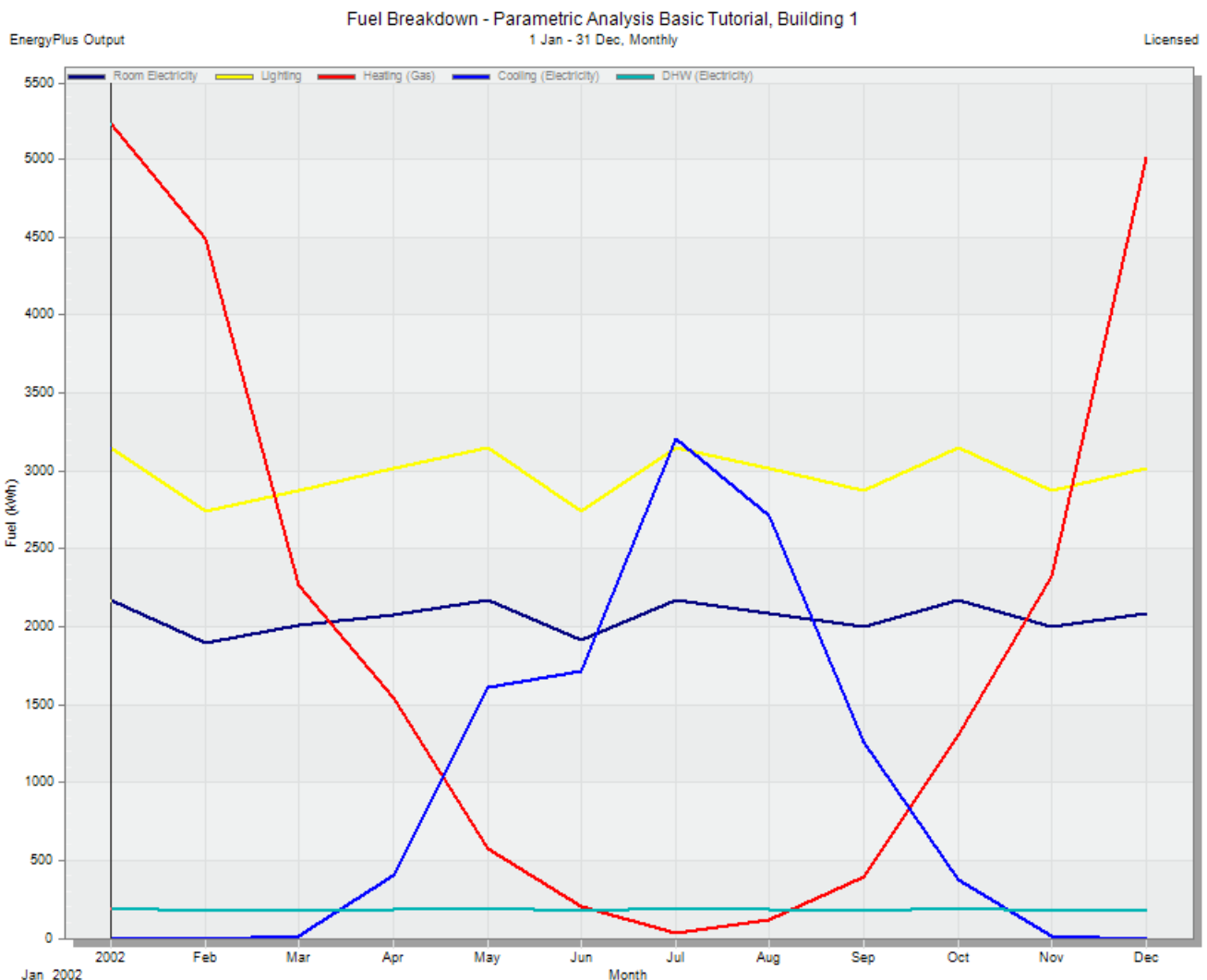
Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m as shown below. Use default Model options and template settings. DesignBuilder as supplied will use a Simple HVAC Fan Coil Unit system as the default. For this example you should make sure that you have heating and cooling selected and no natural ventilation (this will be the case for a new model with a FCU HVAC system selected).



Step 2 - Run a standard simulation

Click on the **Simulation** tab and run a base annual simulation. Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation.

Check the hourly results for the simulation period and make sure that the model is behaving as expected, including temperatures within the building, operations periods etc. If not fix the model and repeat this step until you are happy with the base model hourly results. Monthly results should look something like the screenshot below.



Step 3 - Define optimisation problem on Optimisation Analysis Settings dialog

Once you have a good understanding of how the base model operates, you are ready to start the parametric analysis stage.

For this simple example we will investigate the effect of changing the Window to Wall Ratio (WWR) and overhang length on building annual heating and cooling loads .

To do that follow these steps.

Step 3.a - Open the [Parametric Settings](#) dialog

Step 3.b - On the **Analysis type** tab select the **1-Parametric analysis** option.

Step 3.c - On the **Outputs** tab add the Output results for the parametric analysis study - annual heating and cooling load:

- Click on the **Add Output** Info panel link.
- On the Analysis Outputs dialog select the **Cooling load** Output KPI and enter the name as "Cooling Load". Press OK.
- Click on the **Add Output** Info panel link again.
- On the Analysis Outputs dialog select the **Heating load** Output KPI and enter the name as "Heating Load". Press OK.

The Outputs tab should now look like this:

Parametric, Optimisation and UA/SA Analysis Settings	
Analysis Type	Outputs
Name	Output KPI
Cooling Load	Cooling load
Heating Load	Heating load

Step 3.d - On to the **Design Variables** tab.

- Window to wall ratio will be already set up by default on a new model and we need to keep this variable for this study. Delete any other variables though. In a default model these will normally be for Cooling and heating setpoint temperatures. Select each of these in the grid and press the **Delete variable** Info panel link.
- You should now have only Window to wall ratio in the Design Variables list.
- Click on the **Add variable** Info panel link.
- On the **Edit Analysis Variables** dialog select the **Local shading type** Variable type (it can be found under the **Glazing/shading** category heading). By default this includes 5 levels of overhang length from 0m to 2m. You can keep these default options. Press OK.

Step 3.e - On to the **Variable Option Order** dialog

The **Variable Option Order** dialog allows you to reorder the shading options so that they are displayed in an appropriate order on the Parametric Analysis graphs. By default, the options should already be in the correct order, so press OK.

Variable Option Order


 If the list of options selected represents some sort of numerical sequence, then please re-arrange the list in an increasing or decreasing order to make the optimisation process more efficient. Alternatively, select 'OK' to continue. See 'Help' for more details.

No shading

0.5m Overhang

1.0m Overhang

1.5m Overhang

2.0m Overhang

OK

Help

Up

Down

Step 3.f - Review the **Design Variables** tab which should now look like this:

Parametric, Optimisation and UA/SA Analysis Settings							
Analysis Type	Outputs	Design Variables					
Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisation)	Options list	Target objects
Window to Wall %	Window to wall %	20.00	80.00	20.00	2.000	-	Building
Local shading type	Local shading type	0.00	0.00	0.00	0.000	5 options	Building

Variations are made at building level for both variables in this case as shown by the **Target** objects column.

Press OK to confirm changes and close the Parametric, Optimisation and UA/SA Settings dialog.

Step 4 - Open Optimisation Calculation Options dialog

Click on the **Parametric** tab of the Simulation screen and click on the **Update** toolbar icon to open the **Calculation options** dialog.

Select the Output you would like to see displayed as simulation results are received back from the Simulation Manager.

Step 5 - Start the Parametric analysis

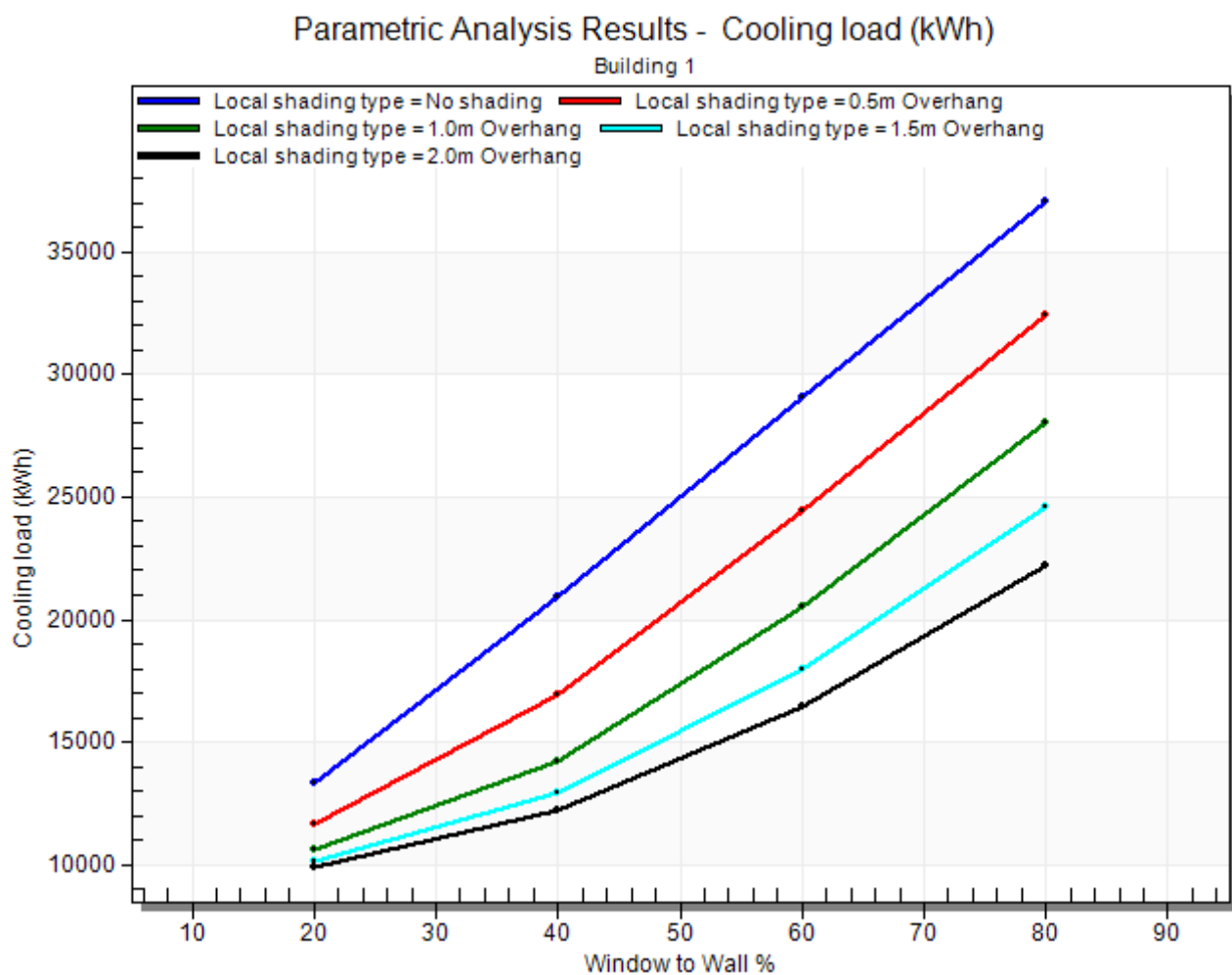
Once you have completed your review of the options press the **Start** button in the bottom right of the screen. The process will involve running several simulations (number of options for design variable 1 x number of options for design variable 2).

Simulations are run in parallel using the [Simulation Manager](#).

Once simulations are complete, close the Calculation options dialog.

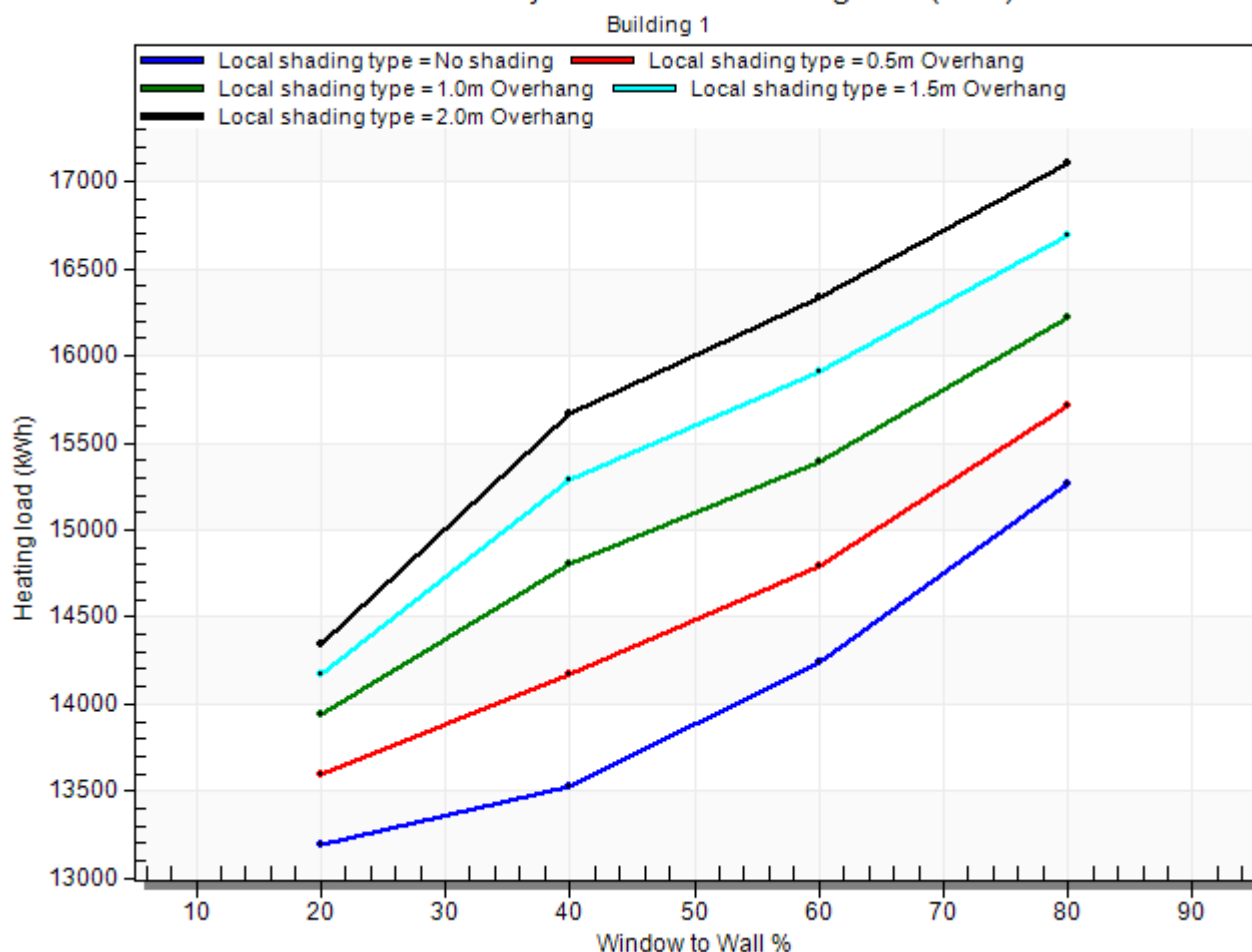
Step 6 - Analyse results

Choose the Parametric output to display on the **Display options** panel to the bottom left of the main screen.



Effect on Cooling load of changing WWR and Overhang length

Parametric Analysis Results - Heating load (kWh)



Effect on Heating load of changing WWR and Overhang length

Perhaps not surprisingly the overhang length has a much greater impact on Cooling load than on Heating load.

Tip: See: "Insights Tutorial - Basic Parametric Analysis " on page 2090 to learn about how to export results from this tutorial into Insights for more detailed analysis and high-quality graphical outputs.

Parametric Analysis Tutorial - Future Climate Study

Building designers are increasingly being asked to assess how their buildings will perform in future climate conditions. To help with future building performance assessments, organisations such as CIBSE and CSIRO have issued sets of 'future weather years' for building simulation that take into account the impact of various projected climate change scenarios. These are based on a series of standard assumptions related to human economic and population growth and the effectiveness of the political and technical solutions applied. Future weather files can be used instead of the usual historical weather data in building models to assess future building performance.

Because future weather files are provided for a range of scenarios and future periods, it makes sense to run simulations for the range of scenarios all in one go and to display results for all files on a single page. DesignBuilder's parametric analysis tools are ideal for this. This tutorial explains all of the steps involved in generating parametric outputs for future weather files.

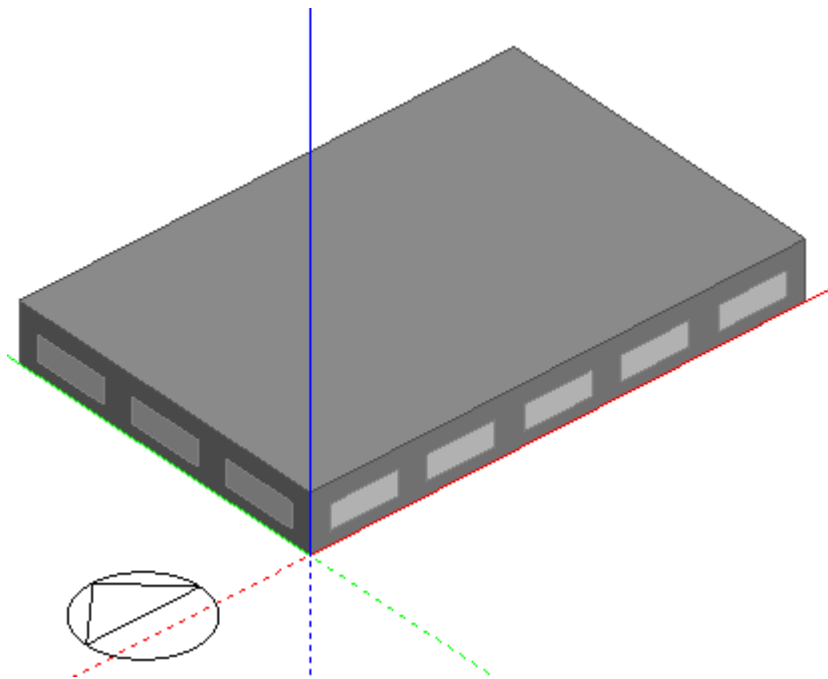
Tip: A similar approach can be taken for running batch simulations for any controlled set of variations in weather files, such as for a range of actual weather years to ensure that *all* historic weather conditions are covered (rather than using 1 "typical" weather year).

Step 1 - Create a base model

Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m as shown below.

Navigate to building level and make the following changes to the base model to use a simple heating system with natural ventilation and to minimise internal and solar gains to levels that are perhaps more realistic in future buildings:

1. On the HVAC tab select the **Radiator heating, Boiler HW, Mech vent Supply + Extract** template.
2. Still on the HVAC tab set the Natural Ventilation, Outside air flow rate to 10 ac/h.
3. On the Opening tab switch on Local shading and select 1m overhangs.
4. On the Lighting tab select the **LED with linear control** Lighting template.
5. On the Activity tab set the power density for Office equipment to be 8 W/m².
6. On the Constructions tab select the **Best practice heavyweight** template.



Step 2 - Run a standard simulation

Click on the **Simulation** tab and run a base annual simulation.

On the calculation options dialog, first select the **2-Operative temperature** control method to help the comfort assessment methods avoid reporting underheating in winter.

Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation.

Then run the simulation.

Check the hourly results for the simulation period and make sure that the model is behaving as expected, including temperatures within the building, comfort levels, operations periods etc. If necessary fix the model and repeat this step until you are happy with the base model hourly results.

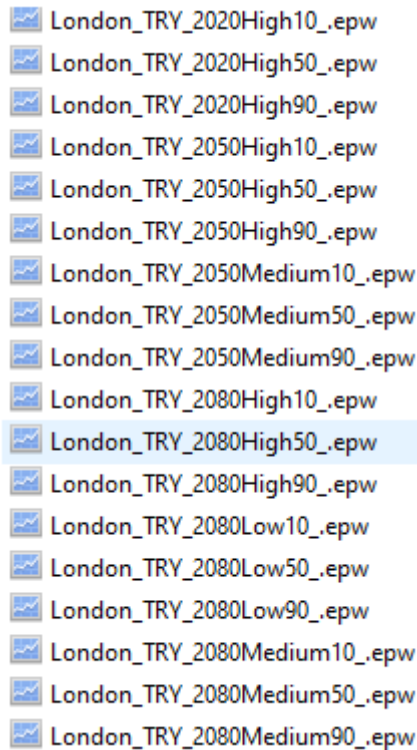
Now run a final simulation without hourly and daily results but with **Monthly and Run period** results to prepare the model for the parametric analysis.

Step 3 - Obtain hourly weather files

To follow the exact steps in this example you will need access to the CIBSE weather files for London. However you can use any future files suitable for your location, but do make sure that the weather files are for more or less the same location as the rest of the location data set at site level.

You can obtain the CIBSE weather files from the [DesignBuilder website](#).

Once you have downloaded the weather files, you must extract the data from the zip file and copy all of the required epw files to the DesignBuilder Weather folder. You can open Windows Explorer at this location through the **File > Folders > Weather Data** menu command. You should end up with these files in the Weather Data folder.



London_TRY_2020High10_.epw
London_TRY_2020High50_.epw
London_TRY_2020High90_.epw
London_TRY_2050High10_.epw
London_TRY_2050High50_.epw
London_TRY_2050High90_.epw
London_TRY_2050Medium10_.epw
London_TRY_2050Medium50_.epw
London_TRY_2050Medium90_.epw
London_TRY_2080High10_.epw
London_TRY_2080High50_.epw
London_TRY_2080High90_.epw
London_TRY_2080Low10_.epw
London_TRY_2080Low50_.epw
London_TRY_2080Low90_.epw
London_TRY_2080Medium10_.epw
London_TRY_2080Medium50_.epw
London_TRY_2080Medium90_.epw

Step 4 - Create Hourly Weather Components

The next step is to create a new [Hourly weather data component](#) for each of the 18 weather files to be analysed. This can be done by following the steps explained in "Add New hourly weather data" on page 1420. To summarise:

1. Open the **Select Hourly weather data** browse dialog.
2. Take a copy of the Default London Hourly weather data component by selecting it and pressing the **Create copy of highlighted item** button.
3. Edit the copy by selecting it and pressing the **Edit selected data** icon.
4. On the Edit hourly weather dialog start by selecting the first file in the list, **London_TRY_2020High10_.epw**. Copy the file name minus the .epw extension and use the text as the name of the Hourly weather component by pasting the copied text into the Name field. You might also like to enter the Source as "CIBSE" but that is optional. The dialog should now look something like this:

Hourly weather	
General	
Name	London_TRY_2020High10
Source	CIBSE
Country	UNITED KINGDOM
Filename	London_TRY_2020High10_
Details	
Latitude (°)	51.48
Longitude (°)	-0.45
WMO station identifier	3772
ASHRAE climate zone	Unknown

5. Now repeat the process for the remaining 17 weather files. Be sure to follow the process carefully in a systematic way to avoid errors.

When you've finished the list of Hourly weather data should look like this:

Select the hourly weather

- London_TRY_2020High10
- London_TRY_2020High90
- London_TRY_2050High10
- London_TRY_2050High50
- London_TRY_2050High90
- London_TRY_2050Medium10
- London_TRY_2050Medium50
- London_TRY_2050Medium90
- London_TRY_2080High10
- London_TRY_2080High50
- London_TRY_2080High90
- London_TRY_2080Low10
- London_TRY_2080Low50
- London_TRY_2080Low90
- London_TRY_2080Medium10
- London_TRY_2080Medium50
- London_TRY_2080Medium90

☐ Add ☐ Delete ☐ Select ☒ Sort

Step 5 - Edit Parametric Settings dialog

Once you have a good understanding of how the base model operates, especially during warm summer periods, you are ready to start the parametric analysis stage.

For this simple example we will investigate the effect of changing the hourly weather file on building discomfort.

To do that follow these steps.

Step 5.a - Open the [Parametric Settings](#) dialog

Step 5.b - On the **Analysis type** tab select the **1-Parametric analysis** option.

Step 5.c - On the **Outputs** tab add the Output results for the parametric analysis study - Discomfort Summer ASHRAE 55 Adaptive 80% and 90% Acceptability:

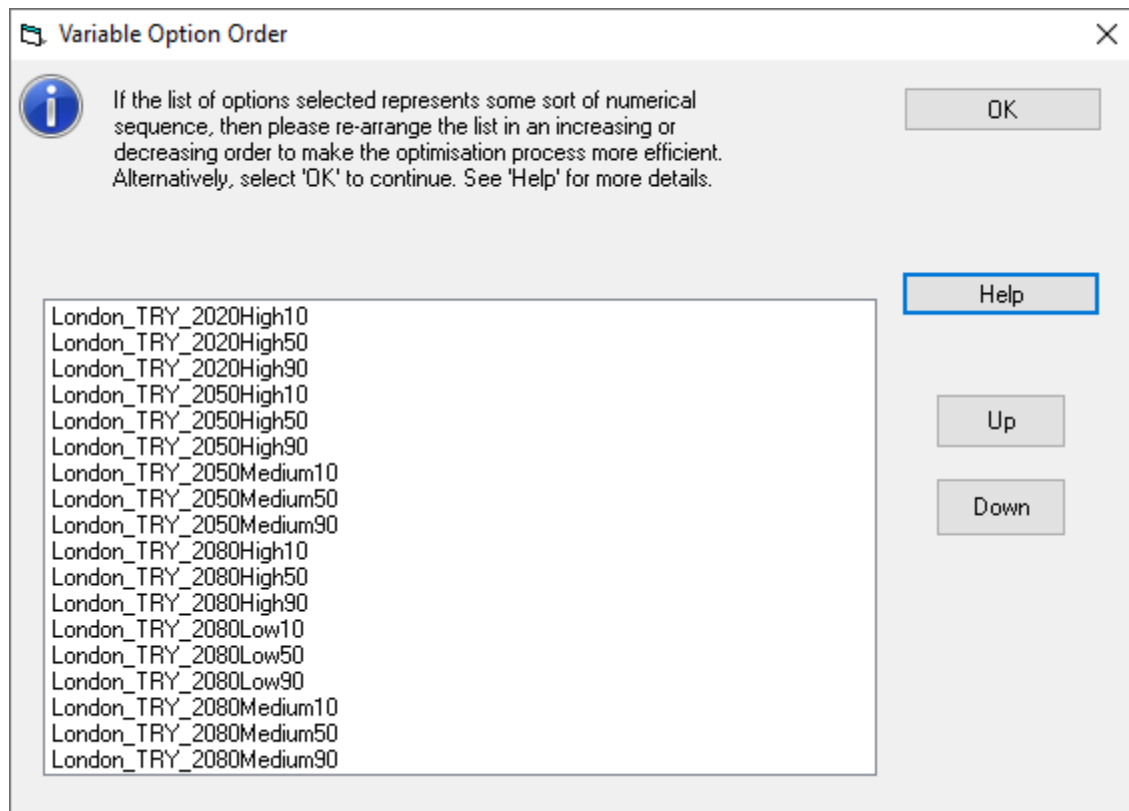
- a. Click on the **Add Output** Info panel link.

- b. On the Analysis Outputs dialog select the **Discomfort Summer ASHRAE 55 Adaptive 80% Acceptability** Output KPI and enter the name as "Discomfort Summer ASHRAE 55 Adaptive 80% Acceptability". Press OK.
- c. On the Analysis Outputs dialog select the **Discomfort Summer ASHRAE 55 Adaptive 90% Acceptability** Output KPI and enter the name as "Discomfort Summer ASHRAE 55 Adaptive 90% Acceptability". Press OK.

Step 5.d - On to the **Design Variables** tab.

- a. Delete all but one variable from the list by selecting each in the grid in turn and pressing the **Delete variable** Info panel link.
- b. You should now have only 1 item Design Variables list.
- c. Select it and click on the **Edit variable** Info panel link.
- d. On the **Edit Analysis Variables** dialog select the **Hourly weather data** Variable type (it can be found under the **Location/weather/orientation** category heading). Press OK.
- e. Click on the Options list dialog option and then on the browse button to display a list of the available Hourly weather components. The 18 custom components should appear in red (since they were added as custom components). Select these 18 components by checking the checkbox to the left of each item and ensure that no other items are checked. Press OK.

When you press OK, a dialog appears to help you order the options list. In this example you may like to order them as follows:



Once you are happy with the order, press OK to accept and return to the Variables dialog.

- f. Click on the Target objects dialog option and press the browse button to the right. Ensure that the Building is selected.
- g. Select the Name field on the dialog and enter the name of the variable as "Climate Scenario". Press OK to close the Analysis variables dialog and return to the Variables tab of the Parametric Settings dialog, which should now look like this:

Parametric, Optimisation and UA/SA Analysis Settings							
Analysis Type	Outputs	Design Variables					
Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisation)	Options list	Target objects
Climate Scenario	Hourly weather data	0.00	0.00	0.00	0.000	18 options	1 Target Selected

Press OK to confirm changes and close the Parametric, Optimisation and UA/SA Settings dialog.

Step 6 - Open Optimisation Calculation Options dialog

Go to the Simulation screen and if the results are no longer available re-run an annual simulation. When the simulation is complete, click on the **Parametric** tab and click on the **Update** toolbar icon to open the **Calculation options** dialog.

Select the Output you would like to see displayed as simulation results are received back from the Simulation Manager (optional).

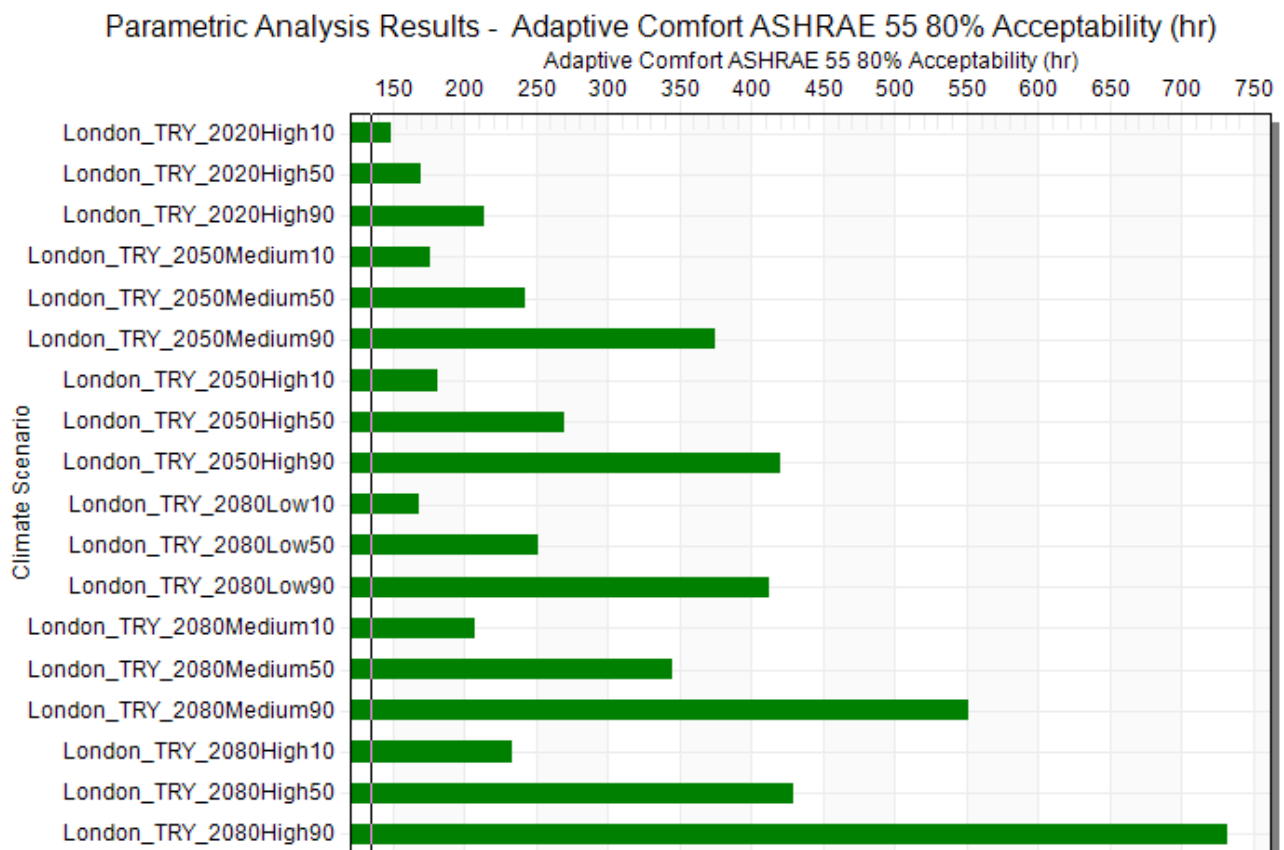
Step 7 - Start the Parametric analysis

Once you have completed your review of the options press the **Start** button in the bottom right of the screen. The process will involve running 18 simulations which are run in parallel using the [Simulation Manager](#). Once simulations are complete, close the Calculation options dialog.

Step 8 - Analyse results

Choose the Parametric output to display on the **Display options** panel to the bottom left of the main screen. You can choose to display either of the 80% or 90% acceptability criteria options.

Right click on the output graph and set the Plotting method to **Bar**. You may instead prefer the **Horizontal Bar** option to display the bars horizontally to ensure that the scenarios are written horizontally instead of sloped as they are when displaying vertical bars.



The results show the extent of discomfort in the model for each of the 18 future climate scenarios.

Parametric Analysis Tutorial - Custom Script KPI - TM59 Comfort

This example shows how to run a parametric analysis with a custom KPI to evaluate the percentage area of discomfort in the building based on the [CIBSE TM59 method](#). DesignBuilder doesn't include a built-in KPI for TM59 comfort, so this is added through a Python script.

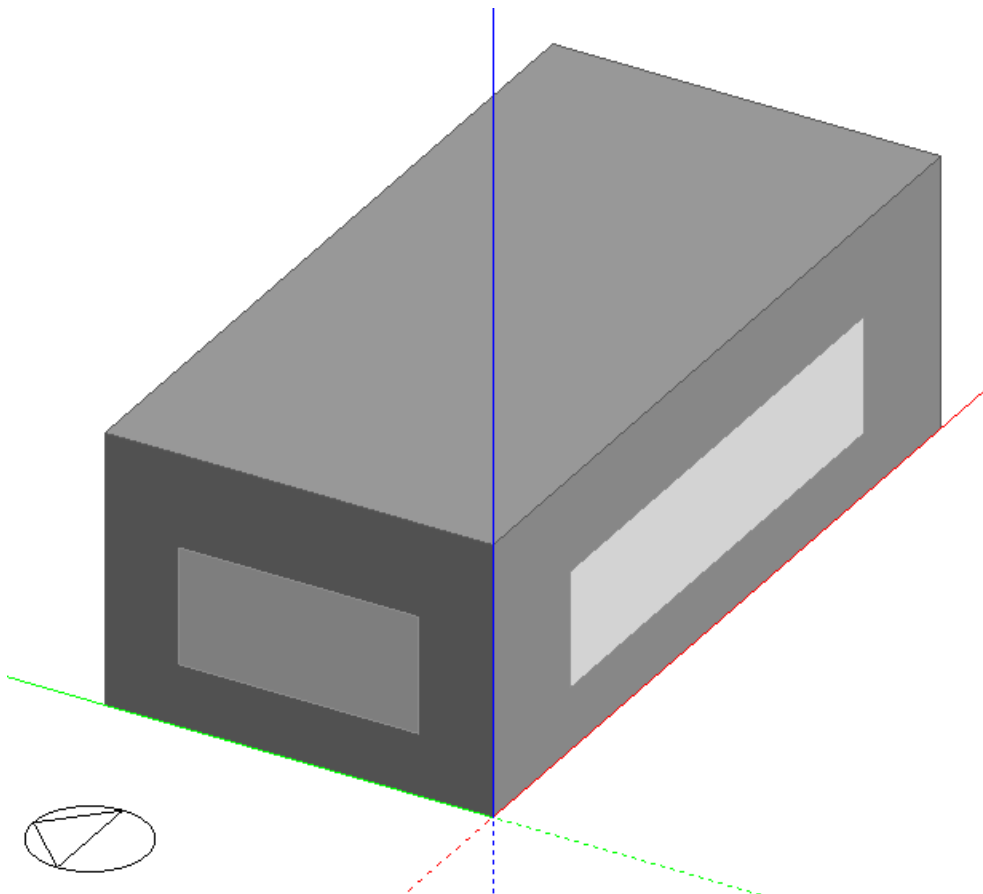
A detailed set of instructions is provided to set up a 2-bedroom apartment model for TM59 parametric analysis. However if you prefer you can instead work with your own TM59 model and make appropriate adjustments to the tasks below. Of course in this case, your results will be different from the example results presented in Step 7.

Step 1 - Create A Base Model

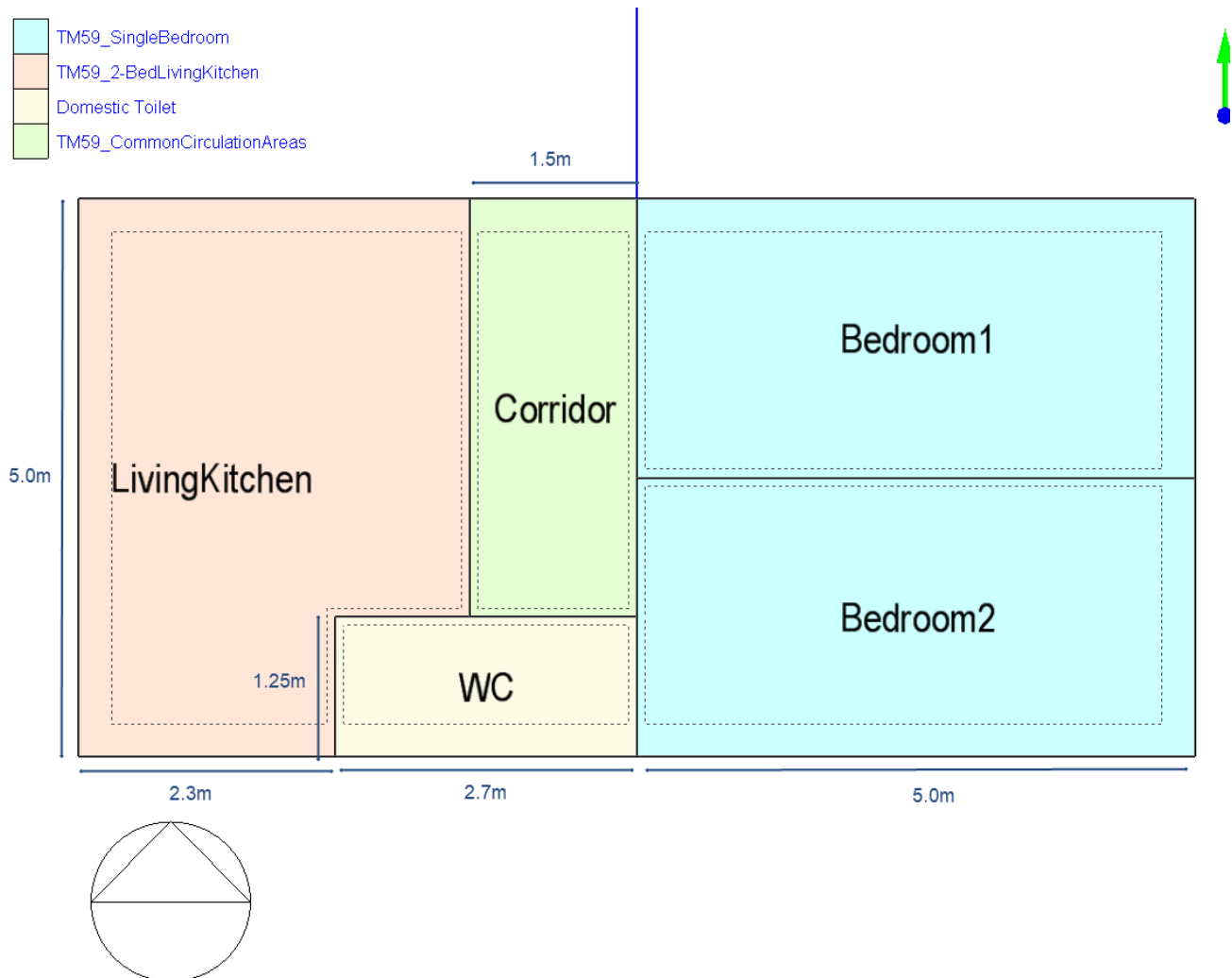
1. Create a new model located at London Gatwick.

Note: For this example, default London Gatwick weather data is used, however, for TM59 analysis, the minimum requirement for TM59 simulations is to use DSY1 weather data for the nearest location and the 2020s, High emissions, 50th % percentile scenario. You can obtain CIBSE Future 2016 weather files suitable for use in TM59 analyses from the DesignBuilder [CIBSE weather data webpage](#).

2. Add a building using default settings and draw a rectangular block measuring 10m x 5m. It should look like the image below.



3. Subdivide the block into 5 zones as follows: 2 single bedrooms, 1 living/kitchen and 1 corridor. The dimensions for each zone are provided in the image below. Enter appropriate names for each zone.



The image above also shows zone colours due to activity assignment , see point 9 below.

4. Define the external windows and doors. At building Level:

- Delete the large south-facing window in Bedroom2.
- Delete all north-facing windows.
- Delete the south-facing window in the WC zone.

5. Navigate to the Living/Kitchen west-facing surface and:

- On the Openings tab add a new External door by checking the **Doors > External > Auto generate** checkbox.
- Set the **Preferred width** to 1.0m.
- Set the **Preferred height** to 2.1m.

6. Navigate to the WC south-facing surface and add a 1m x 1m custom window. Note that because it is a custom window its size will not be affected by parametric changes to the **Window to wall %**.

7. Open the Model options dialog. The following Model option settings should be applied:

- **Gains data:** Early
- **Gain definition level:** 1- Globally in model options
- **Occupancy method:** 3- Number of People
- **Occupancy latent gains:** 2- Fixed Fraction
- **Equipment gain units:** 2-Absolute Zone Power
- **Lighting gain units:** 1-Power density
- **Natural ventilation:** Calculated

- On the [Advanced](#) tab, uncheck the [Lump similar windows on the surface](#) option.
- On the [Simulation](#) tab set the [Time steps per hour](#) to 6. This change is not essential to follow the learning points in this tutorial but it is good practice for accurate Calculated natural ventilation simulations, especially when accurate overheating results are required.

8. Switch on TM59 Outputs:

- Still on the Model options dialog, on the Simulation tab, under the **Output, Graphable** and **Comfort and Environmental** headers, check the [CIBSE TM59](#) checkbox to ensure that DesignBuilder generates standard TM59 outputs. These standard outputs will need further processing to create a single discomfort value for the whole building and we'll write a Python script to do that in Step 3.
- Still on the Simulation tab, under the **Output** and **Miscellaneous Outputs** headers, select the [SQLite output](#) option. This will ensure that EnergyPlus generates the results in a form that can be read by our TM59 script.

These settings are discussed in the [TM59 Analysis Guide](#).

Press OK to close the Model options dialog.

9. On the Activity Tab, define the activities for each zone. You can find the TM59 activities under the **Residential spaces** category:

- Bedrooms: **TM59_SingleBedroom**
- LivingKitchen: **TM59_2-BedLivingKitchen**
- Corridor: **TM59_CommonCirculationAreas**
- WC: **Domestic Toilet**

10. On the HVAC tab at building level:

- Select the Radiator heating, **Boiler HW, Nat Vent** HVAC Template. Note that the key changes that selecting this HVAC template has on the model for the purpose of this tutorial are:
 - Switches off [Cooling](#)
 - Switches on [Natural Ventilation](#)

Note : TM59 outputs will only be reported for zones without mechanical cooling.

11. On the Openings tab at building level:

- Under the **External Windows > Operation** headers, change the [Schedule definition](#) to **2-Custom schedule**
- Change the [Operation schedule](#) to **TM59_Unrestricted_Openings**

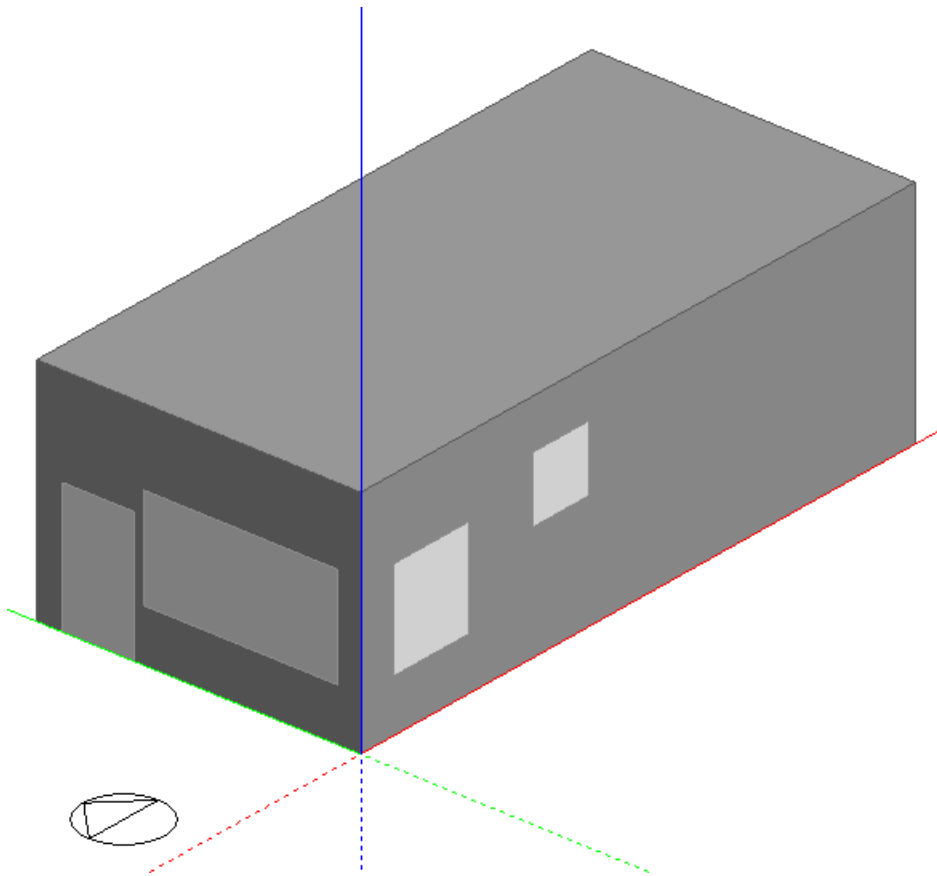
Note: Approved Document O states that for buildings at ground level opening restrictions should be applied due to privacy and security. However, for this example, all openings are unrestricted.

- Under the **Doors > External > Operation** header:
 - Set the [% Time door is open](#) to 0.
- Under the **Doors > Internal > Operation** header:
 - Set the [Schedule definition](#) to **2-Custom schedule**.
 - Change the [Operation Schedule](#) to **TM59_Unrestricted_Openings**.
 - Set the [% Area door opens](#) and [% Time door is open](#) to 100 for internal doors.
 - Check **Auto generate** to create doors in all internal surfaces by default.
 - Set **Preferred width** to 0.9m and **Preferred height** to 2.0m.
 - Remove the internal door between the 2 bedrooms by navigating to the adjoining partition and unchecking the **Auto generate** checkbox.
 - Similarly, remove the internal door between the LivingKitchen and WC zones.

12. On the Lighting tab at building level:

- Select the **TM59_Lighting** Lighting template

The model should now look like the image below.



Step 2 – Run A Simulation To Check The Base Model

Click on the Simulation tab and run a set of simulations to check the operation of the nat vent system to make sure you understand how it works in practice. You should generate hourly or sub-hourly results to gain the best understanding on model behaviour.

Are comfort conditions adequate during the summer? Is the air change rate excessive at any point? Do you see any simultaneous heating and venting? Make any adjustments to the model and repeat this step until you are satisfied with the base model hourly results.

Step 3 – Adding Parametric Design Variable

In this step we will be setting up 2 design variables based on these standard variable types:

- % External window opens
- Window to wall %

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon.

2. On the **Analysis Type** tab, set the Analysis type to **1-Parametric Analysis**

3. Go to the **Design Variables** tab and delete all default variables in the list. We will be adding the 2 new variables here.

Tip: The suggested order is to add **% External window opens** first, then **Window to wall %**. This ensures that **% External window opens** appears on the X-axis and there will be a set of curves, one for each **Window to wall %** option.

3.a. Add a new Variable and select the **% External window opens** Variable type. You can find it in the **Natural Ventilation/Infiltration** category.

3.b. Change the **Min value** to 10 and the **Max value** to 80 to allow the window opening area to vary between 10 and 80%.

3.c Set the **Step (Parametric)** to 10.

3.d Keep the default **Building** target so that the % opening area varies similarly in all external windows and press OK to complete the definition of the **% External window opens** variable.

4.a Add a new **Window to Wall %** variable and edit **Min value** to 20 and **Max value** to 50 to allow the WWR to vary between 20 and 50% for all wall surfaces with windows.

4.b Set the **Step (Parametric)** to 10.

4.c. Keep the default target of **Building** and press OK to complete the definition of the **Window to Wall %** variable.

After Step 3 is complete the **Design Variables** tab should look like this:

Parametric, Optimisation and UA/SA Analysis Settings							
Analysis Type	Outputs	Design Variables					
Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisation)	Options list	Target objects
% External window...	% External window...	10.00	80.00	10.00	1.000	-	Building
WWR	Window to wall %	20.00	50.00	10.00	2.000	-	Building

You won't be able to save the settings and close the Analysis Settings dialog until the parametric analysis Output has been set up which brings us to the next task.

Step 4 – Set Up Parametric Outputs

Click on the **Outputs** tab. The list of outputs will be empty if you started from a new model.

1. Add a new output and on the Analysis Outputs dialog browse the list of available Output KPIs.

2. DesignBuilder does not include a TM59 output so we need to create it. To create the TM59 comfort output, a new Summary Output (aka KPI) must first be created. Click on the **Output KPI** to open the list of existing Summary Outputs in the Info panel. Our new custom KPI won't be listed there at first, so create it now by clicking on the existing **Custom Script** Summary Output on the list of KPIs, click on the **Create copy of highlighted item** Info toolbar icon and edit the copy.

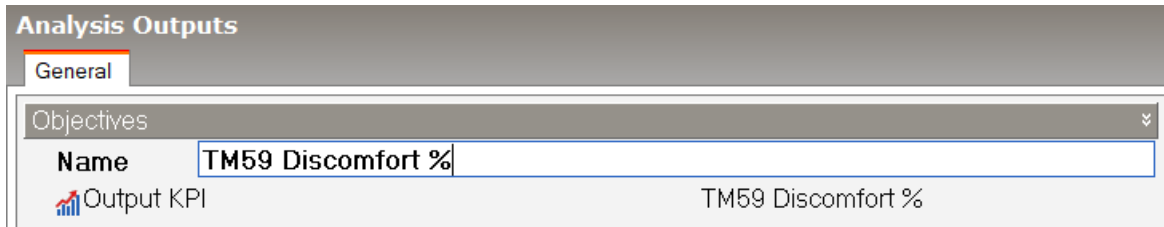
3. Make the following settings on the Summary Output dialog.

The screenshot shows the 'Summary Output' dialog box with the 'General' tab active. The 'Name' field is populated with 'TM59 Discomfort %' and the 'Category' dropdown menu is set to 'Custom script'.

Important Note: The name, "TM59 Discomfort %", entered here on the Summary Outputs dialog must match the text written to field 0 of the record added to the "ParamResultsTmp" table in the [script](#). You may

like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

4. Save the newly created KPI, select it on the **Output Variables** tab and press OK to return to the main Analysis Settings dialog.



You can now press OK on the Analysis Settings dialog to confirm the parametric analysis settings.

Step 5 – Write Python Script For The Custom Script KPI

The next task is to write the script that will read the base TM59 simulation outputs from the EnergyPlus output file, process them and pass back to DesignBuilder as a Custom KPI. The steps are as follows.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Click on the **Python-Script** category and press the **Add new item** Info toolbar icon to open the Edit script dialog.
4. Check the **Enable program** option.
5. Name the script "TM59 Discomfort".
6. Copy and paste the script below into the script window.

The script reads the TM59 Criteria 1, 2, and 3 outputs and checks against the corresponding criteria based on [TM59 rules](#). Any zone passing the overall annual TM59 criteria has its area excluded from the total area failing. The TM59 Discomfort % value for the simulation is then added to the special "ParamResultsTmp" table together with the name of the Objective to be picked up by DesignBuilder when it processes the simulation results.

The "TM59 Discomfort %" Summary output is added as a record to the "ParamResultsTmp" table. More custom KPIs could be added to the table in a more sophisticated analysis.

Python example: TM59 KPI for Optimisation using db_eplusout_reader

```
"""
Use CIBSE TM59 results as optimisation KPI.
Required model inputs:
- Enable TM59 outputs
- Add a "Custom Script" KPI, use "TM59 Discomfort %" name
- Run a "Standard" simulation to make sure that TM59 outputs are present
The script calculates a percentage of failing TM59 relevant zones.
"""

import ctypes
import os

from db_eplusout_reader import Variable, get_results
from db_eplusout_reader.constants import *
from db_eplusout_reader.exceptions import NoResults
```

```

THRESHOLD_A = 3
THRESHOLD_B = 32

def show_message(title, text):
    ctypes.windll.user32.MessageBoxW(0, text, title, 0)

def zone_fails_tm59(zone_name):

    """Verify if given zone meets tm59 criteria."""

    outputs_path = os.path.join(api_environment.EnergyPlusFolder, "eplusout.sql")
    variable1 = Variable("EMS", "CIBSE TM59 Criterion A {}".format(zone_name), "%")
    variable2 = Variable("EMS", "CIBSE TM59 Criterion B {}".format(zone_name), "H")

    criterion_a_results = get_results(outputs_path, variable1, RP)
    criterion_a = criterion_a_results.scalar
    fail = criterion_a > THRESHOLD_A

    try:
        criterion_b_results = get_results(outputs_path, variable2, RP)
        criterion_b = criterion_b_results.scalar
        fail = fail or criterion_b > THRESHOLD_B
    except NoResults:
        pass

    return fail

def after_energy_simulation():
    site = api_environment.Site
    fail_area = 0
    total_area = 0 # area only includes TM59 zones

    for block in active_building.BuildingBlocks:
        for zone in block.Zones:
            zone_name = zone.GetAttribute("SSEPObjectNameInOP")
            try:
                if zone_fails_tm59(zone_name):
                    fail_area += zone.FloorArea
                    total_area += zone.FloorArea
            except NoResults:
                pass

    try:
        percentage_area_fail = (fail_area / total_area) * 100
    except ZeroDivisionError:
        raise NoResults("No TM59 results found!")

    # show_message("Results TM59", "{}% fails, total_area {}m2, fail_area:
    {}m2".format(percentage_area_fail, total_area, fail_area))

    table = site.GetTable("ParamResultsTmp")
    record = table.AddRecord()
    record[0] = "TM59 Discomfort %"
    record[1] = str(percentage_area_fail)

```

Note: The script above is shown with the `show_message` line commented out, but the message can be enabled by removing the "#" character to display TM59 compliance results. Once you are confident that the TM59 script is working well you can delete or comment out the `show_message` line to avoid seeing messages after each simulation during the optimisation.

Tip: The [DesignBuilderSoftware \ db-scripts](#) Github repository includes the latest versions of the above script as well a variation where Criterion A and Criterion B are reported separately to give a more in-depth analysis of the results.

Step 6 - Re-Run A Standard Simulation

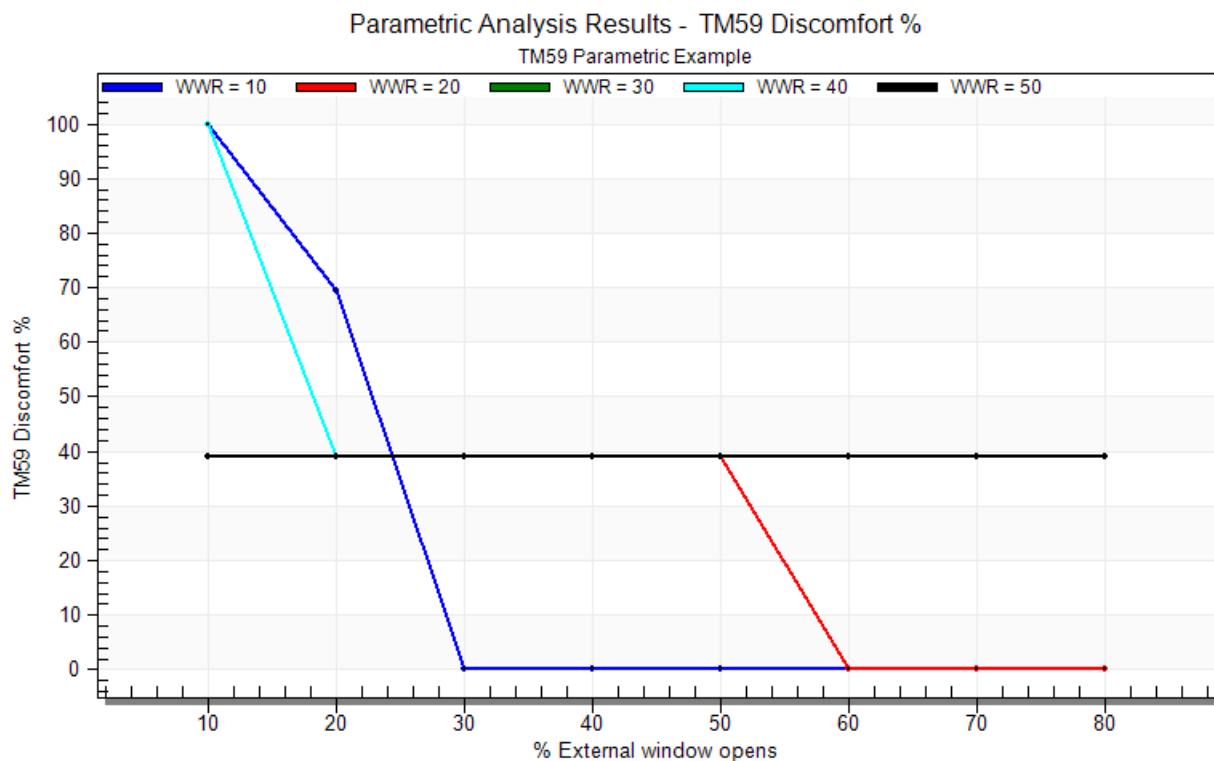
Click on the Simulation tab and press update to re-run a base annual simulation with the **Monthly and RunPeriod** results option checked.

Important Note: Make sure to switch off **Daily, Hourly** and **Sub-hourly** results for this simulation to ensure that the upcoming optimisation simulations and results processing are done efficiently.

Step 7 - Run The Parametric Simulation

Navigate to the Parametric tab on the Simulation screen, ensure you are at building level and press the **Update** toolbar icon to open the Optimisation Calculation Options and press the Start button to run the parametric simulation.

After the simulation has completed, you should see similar results to those below on the main screen:



Some points to note are:

- The parametric results show that the **% External window opens** variable has a significant impact on the TM59 Discomfort % with larger opening areas being needed to avoid discomfort due to overheating.
- The parametric analysis **TM59 Discomfort %** graph only includes a limited number of result values (0, 38.9, 69.5 and 100) as there are only 4 permutations of pass/fail zones in this small model.
- The results suggest that 20% WWR with 60% window opening area could be a viable solution. However if more glazing or smaller opening area is required then some window shading solution (e.g. overhangs or solar control glazing) may be required to ensure that the building meets the TM59 overheating requirements.

Parametric Analysis Tutorial - Custom Script Variable - Advanced Parametric Facade

This tutorial explains the process involved in setting up a Parametric analysis using [Custom Script variables](#) to modify the shading configuration in a model by loading various façade configurations from a series of XML files. Each shading configuration to be tested in the parametric simulations is set up in the model by adding component blocks and saving them in an external XML file. A simple Python script loads the appropriate XML file to the model for each parametric custom variable configuration.

This approach to parametric modelling is ideal for investigating and optimising shading systems that have more complex or non-standard shapes than can be modelled using the [Local shading](#) mechanism,

The main learning points are:

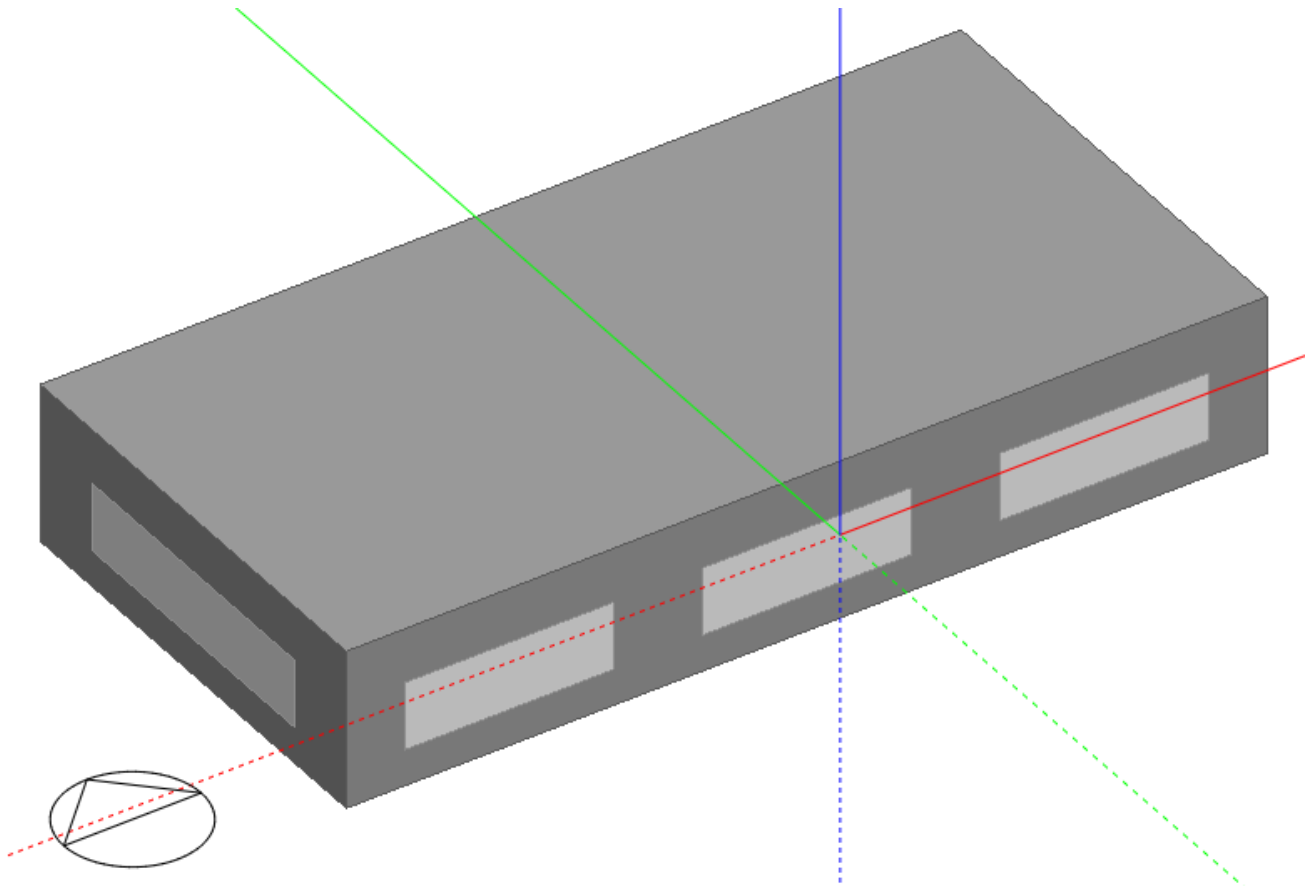
1. How to save a series of component block files in XML format to represent each shading configuration to be tested.
2. Writing a simple Python script to load the XML files as required during the parametric simulations.

Secondary learning points re-enforce points from the more basic [optimisation custom script variable tutorial](#):

1. [Design objectives](#) for minimising occupant discomfort and annual carbon emissions are set up in the usual way.
2. [Design variables](#) for heating and cooling setpoint temperatures are set up using the **Custom script** variable type category including the min, max and interval values to be used.
3. A corresponding script for the **on_design_variable_changed** hook point is written to respond to changes in the model.

Step 1 - Create a base model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 20m by 10m with block height of 3.5m as shown below.



3. Delete the windows on the north, east and west façades.

4. Load the **Generic Office Area** Activity template at building level on the Activity Tab (if it is not already selected).

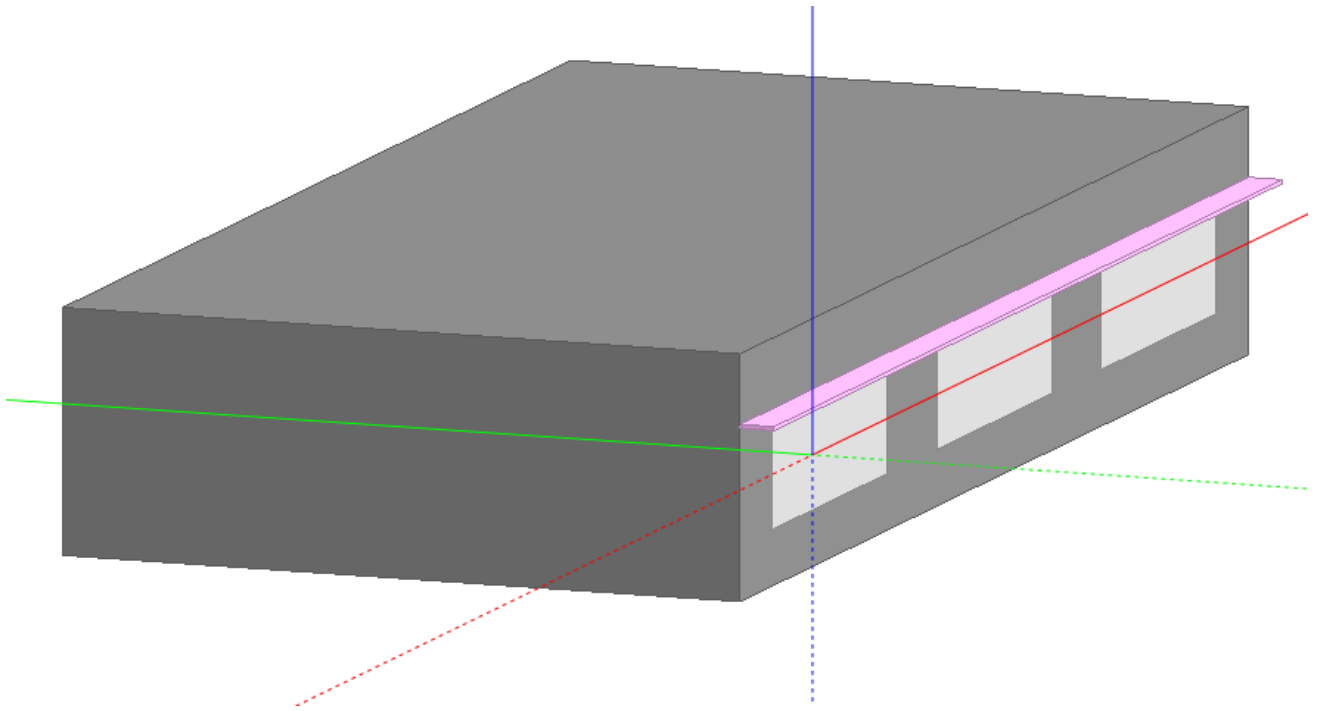
Step 2 - Set up the Parametric XML Files

In this step we will create 3 XML files to represent 3 of the 4 anticipated variable options:

- No overhang (no file to be loaded)
- 0.5m overhang
- 1.0m overhang
- 1.5m overhang

To create the 3 files follow these steps. All these steps should be carried out at building level.

1. [Draw a component block](#) above the windows on the south façade to represent the 0.5m overhang case. The model should look like this.



2. Select the component block and save it as an XML file using the **File > Export > Export selected component blocks as XML** menu command. Use the name "Overhang 0.5m.xml".
3. Now increase the depth of the overhang to 1.0m. You can use the [Drag face](#) command to do this.
4. As before, select the component block and save it as an XML file. Use the name "Overhang 1.0m.xml".
5. Now increase the depth of the overhang to 1.5m and follow the same procedure as above to save the XML file to represent the 1.5m overhang.

You should now have a list of 3 XML files:

- Overhang 0.5m.xml
- Overhang 1.0m.xml
- Overhang 1.5m.xml

6. It would be good practice to import the XML files manually to check that everything is in order before the parametric runs. Use the **File > Import > Import component blocks from XML** menu command to do this. You will need to delete the original component block first to allow the component blocks to be imported from file. Intersection checks will prevent the parametric XML imports if the component block is left there.

Make a note of where you have saved these files and be sure to set this folder name in the XML_DIR variable in the script that we will write in the next step.

7. The final and essential task is to delete the overhang component block to allow the XML files to be imported during the parametric runs. As mentioned above, intersection checks will prevent the parametric XML imports if the component block is left there.

Step 3 - Edit Parametric Settings dialog

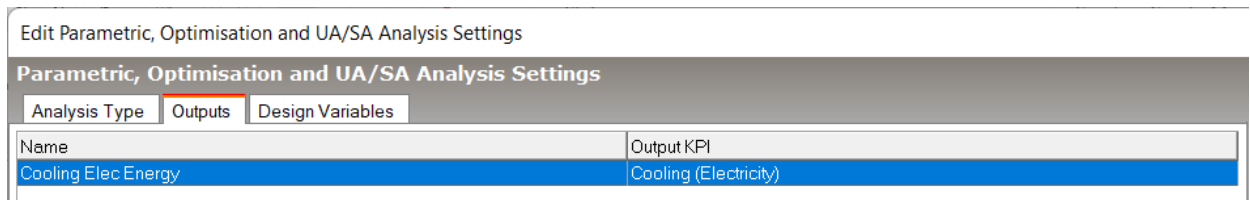
To define the parametric analysis follow these steps.

Step 3.a - Open the **Parametric Settings** dialog by clicking on the  toolbar icon.

Step 3.b - On the **Analysis type** tab set the [Analysis type](#) to **1-Parametric analysis**.

Step 3.c - On the **Outputs** tab add the Output results for the parametric analysis study. We will be using cooling electric energy as the KPI for this analysis:

- a. Click on the **Add Output** Info panel link.
- b. On the Analysis Outputs dialog select the **Cooling Electricity** Output KPI under the "Energy and Loads" category and enter the name as "Cooling Elec Energy". Press OK. The Outputs tab should now look like this:



Step 3.d - On to the **Design Variables** tab.

- a. Delete all but one variable from the list by selecting each in the grid in turn and pressing the **Delete variable** Info panel link.
- b. You should now have only 1 item Design Variables list.
- c. Select it and click on the **Edit variable** Info panel link.
- d. On the **Edit Analysis Variables** dialog select the "Custom Script" Variable type under the "Custom script" category heading. Press OK.
- e. Under the **Range** header set the **Min value** to 0 and the **Max value** to 1.5. This will allow us to test overhangs of between 0 and 1.5m length.
- f. Under the **Increment Step** header set **Step (parametric)** to 0.5 to ensure that we test the overhangs in 0.5m steps, i.e. overhang values of 0, 0.5, 1.0 and 1.5m will be tested. You could set **Step (optimisation)** to 0.5 as well although we don't plan to run an optimisation study in this example.
- g. The **Variable key** is not used in this analysis and can be left blank.
- h. Set the **Name** of the design variable to "Overhang".

Important Note: The name, "Overhang", entered here on the **Edit Analysis Variables** dialog must match the corresponding name used in the custom script described below. You may like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

- i. Press OK to close the Analysis variables dialog and return to the **Design Variables** tab of the Parametric Settings dialog, which should now look like this:

Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisation)	Options list	Target objects
Overhang	Custom Script	0.00	1.50	0.50	0.500	-	Script

Step 4 - Write Script for Custom Script Variable

Follow the steps below to set up the Python script for the analysis.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Click on the **Python-Script** category and press the Add  Info panel icon to create a new Python script.
4. Change the name of the script to "Overhang Custom Variable". This is for your convenience and does not need to match with any other settings.
5. Edit the script to apply the overhang settings to the model as follows:

```
import os
import ctypes
```

```

def show_message(title, text):
    ctypes.windll.user32.MessageBoxW(0, text, title, 0)

XML_DIR = r"C:\Users\<USERNAME>\Documents\DesignBuilder Data"

def on_design_variable_changed(opt_var_id, variable_current_value):

    site = api_environment.Site
    table = site.GetTable("OptimisationVariables")
    record = table.Records.GetRecordFromHandle(opt_var_id)
    opt_variable_name = record["Name"]

    if opt_variable_name == 'Overhang' and float(variable_current_value) > 0:
        overhang_length = "{:.1f}".format(float(variable_current_value))
        xml_name = "Overhang {}.xml".format(overhang_length)
        xml_path = os.path.join(XML_DIR, xml_name)
        if os.path.exists(xml_path):
            active_building.ImportComponentBlocksFromXMLFile(xml_path)
        else:
            show_message("Error", "Cannot find xml file: '{}'.format(xml_path))

```

Tip: To save time you can copy the above script and paste it into the dialog. Note that Python requires the indentation as part of its syntax and when you paste the above script the indents may become lost. In this case you must insert the indents manually after the paste. The easiest way to do that is by using the <Tab> key.

Note: If you copy the above script you will need to change XML_DIR to define the path on your machine where the XML files have been saved.

The **on_design_variable_changed** hook point is called once for each variable when the model is being prepared for the simulation. The value of **opt_variable_name**, passed in via **opt_var_id**, defines the variable being handled and the value of **variable_current_value** also passed in provides the value of this variable for this simulation. The script simply loads the appropriate XML file depending on the **variable_current_value**.

active_building in this case is a reference to the parametric clone copied from the base model for each simulation and so it is safe to apply any parametric changes to it without affecting the base model.

6. Check the **Enable program** checkbox to ensure that the script will be executed at runtime.

7. Press OK to confirm the changes to the script.

Step 5 - Run a Standard Simulation

Click on the Simulation tab and press update to run a base annual simulation as a necessary step before running the optimisation. Before pressing OK, You should make the following settings:

1. Ensure you are running simulations for the whole year by clicking on the **Annual simulation** Info panel link.
2. Ensure you are generating only Annual/Monthly results by checking the **Monthly and Run period** interval and that all other outputs are switched off.

Leave all other settings in their default state.

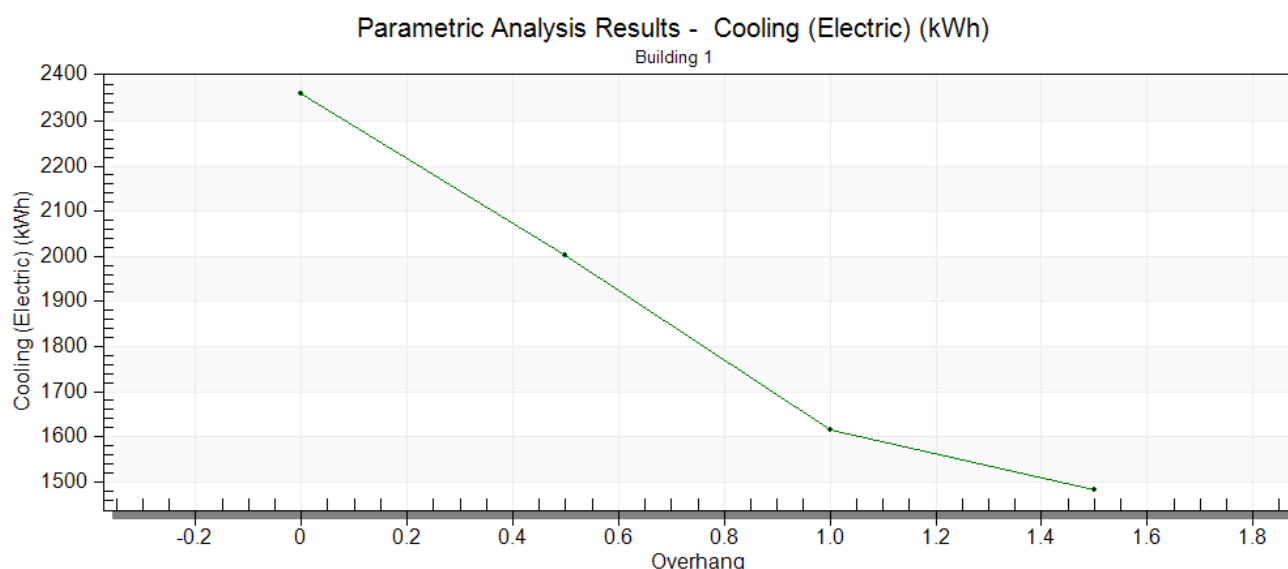
Step 6 - Run Parametric Analysis

Navigate to the **Parametric** tab and press the **Update** toolbar icon to open the Calculation Options dialog and run the parametric analysis simulations. Accept default settings and press the **Start** button.

The 4 simulations should run through quickly and when they have finished press the **Close** button to return to the main screen.

Step 7 - Analyse Results

After the parametric analysis simulations have completed and you have closed the Calculation Options dialog, the main screen should look similar to the screenshot below.



Results show diminishing returns as the overhang extends from 1.0m to 1.5m.

Note: The tutorial demonstrates how to test the impact of various rectangular shading overhangs on building cooling energy. To help focus on the main learning points the shading case being modelled is a very simple one that could actually have been implemented more easily using the [Local shading overhang](#) mechanism. However note that much more geometrically complex shading configurations can also be tested through a process very similar to the one demonstrated in this tutorial.

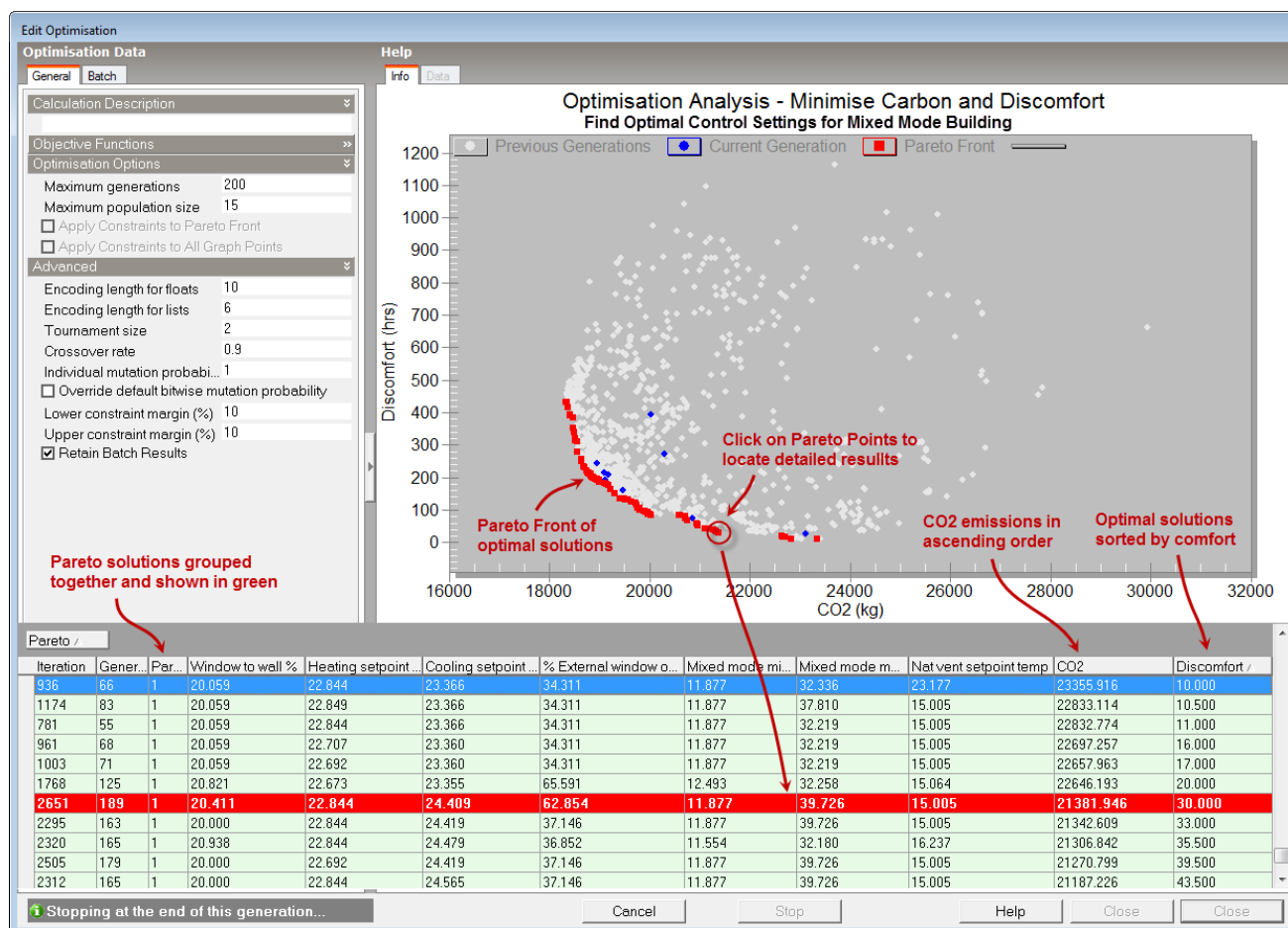
Optimisation

Optimisation is a technique for efficient searching for and identifying design options that best meet key design performance objectives. It is similar in many ways to [Parametric analysis](#), a more well-known technique for analysing how design performance varies with changes in the building configuration through use of design curves. A parametric analysis would usually consist of 1, 2 or 3 design variables being adjusted in a systematic way to illustrate trends and find designs with the most favourable characteristics (e.g. low energy consumption, best comfort etc). With parametric analysis, a maximum of 3 variables is normally used because a) the results of more than 3 dimensions to a design problem are difficult to visualize and b) the large number of simulations required with 4 or more design variables would take too long to complete. For example a designer might want to investigate the impact on carbon emissions of variable levels and types of glazing. The results would be displayed as a series of [parametric design curves](#). This can be a very useful way to visualise simple comparisons over a limited range of design options, but is of less use for wider studies and for optimisation as only a few variables and one key performance indicator can practically be included per analysis.

In DesignBuilder Optimisation, [Genetic Algorithms \(GA\)](#) (aka Evolutionary Algorithms or EA) are used to search for optimal design solutions, much more efficiently than is possible with parametric analysis when more variables are involved. In DesignBuilder, any number of [design variables](#) can be included in the analysis in combination with up to 2 [objectives](#), such as "minimise carbon emissions" and "minimising construction costs". Cost and carbon are a frequently used pair of objectives in building design optimisation analyses because they allow a study of the trade off between cost and carbon impacts for a large range of

designs. For example, an optimisation study might involve a base design which is to be optimised for cost and carbon emissions with building orientation, wall and roof construction, glazing amount and type, degrees of shading and HVAC system type being allowed to vary. The results might be displayed graphically with operational carbon emissions on one axis and investment cost on the other and the performance of each design option that is tested as part of the procedure plotted on the graph. The designs with lowest combinations of cost and carbon form a "Pareto front" of optimal designs along the bottom-left edge of the data point "cloud".

Likewise "minimise carbon emissions" and "minimise discomfort" are often used to analyse the trade off between carbon emissions and the degree of comfort provided by the design. In the example output below the control parameters of a changeover mixed mode natural ventilation model is being optimised based on carbon emissions and discomfort. The outputs indicate the control options resulting in minimal discomfort hours while at the same having the lowest environmental impact.



Tip: A good starting point to learn more about how to use the design optimisation tools is to follow through the [Optimisation Basic Tutorial](#).

To get started with an optimisation analysis go to the [Simulation screen](#) and click on the Optimisation tab. From the Simulation Optimisation screen you can:

- Edit the [Optimisation Analysis Settings](#) (objectives, constraints and variables). This is a required first step before running the first analysis.
- Update the Optimisation Analysis which opens the [Optimisation Calculation Options Dialog](#).
- [View Optimisation results](#).

Optimisation Contents

- [Optimisation Parametric Settings](#)

- [Optimisation Calculation Options Dialog](#)
- [Optimisation Results](#)
- [Optimisation Limitations](#)
- [Optimisation Tips and Troubleshooting](#)
- [Optimisation Background](#)
- "Optimisation Tutorials" on page 1736

Optimisation - Parametric Settings Dialog

The **Parametric Settings** dialog can be accessed by clicking on the **Parametric settings** icon  on the toolbar.

You can use this dialog to set up the optimisation problem by entering data on the tabs described below:

- [Analysis Type](#) - select the Analysis type.
- [Objectives](#) - set up 1 or 2 objectives for the analysis to define what constitutes a "good design".
- [Additional Outputs](#) - optional extra outputs can be used in later analysis to quantify how various aspects of building performance are affected by the parametric changes to the model.
- [Constraints](#) - optional limits on performance to be imposed.
- [Design Variables](#) - the elements of the model that are to be allowed to vary during the optimisation analysis and the new values that these elements may take.

These tabs contain respectively tables of the objectives, additional outputs, constraints and design variables to define the optimisation problem.

Analysis Type Tab

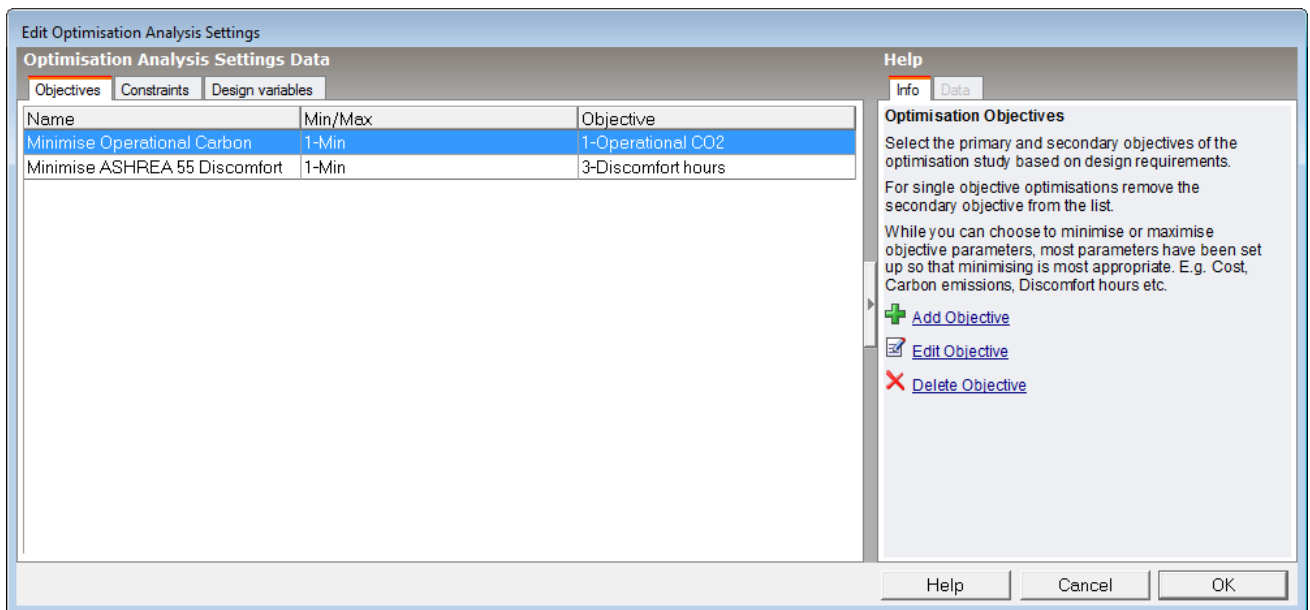
There are 3 types of analysis and these can be set from the first tab on this dialog.

Analysis

The analysis to select for optimisation is **2-Optimisation**.

Objectives Tab

The Objectives tab allows you to define how the "success" of a particular design is be measured. This is done by defining either 1 or 2 objectives for the analysis. Typical settings here might include 2 objectives to investigate the trade off between 2 conflicting objects such as minimising both construction cost and energy consumption. See screenshot below.



Controls on the Info panel are:

- **Add Objective:** Creates a new objective (using the [Edit Optimisation Objective](#) dialog). This option is only available if there are less than 2 objectives already defined.
- **Edit Objective:** Edits the selected existing objective from the table (using the [Edit Optimisation Objective](#) dialog)
- **Delete Objective:** Deletes the selected existing output from the table. This option is only available if there are 2 objectives listed.

It is only possible to have 1 or 2 objectives and the number of objectives (1 or 2) radically affects the nature of the analysis. With a single objective optimisation there is no trade off between 2 objectives, but simply a requirement to select designs that give highest or lowest values of the objective KPI.

Additional Outputs Tab

The additional outputs tab allows you to define some extra outputs to be included in the table of results for each iterative design variation simulated. This can help in the final analysis of results to gain a clearer understanding of how variations in the design will impact on the building performance.

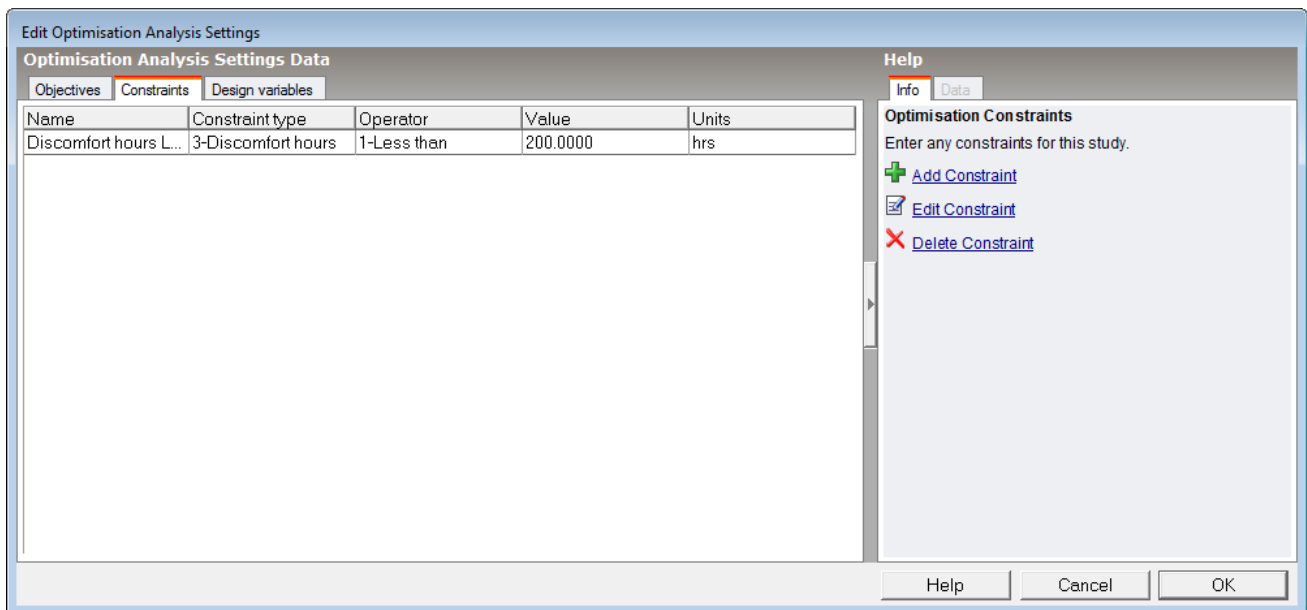
Any number of Additional outputs can be added.

Controls on the Info panel are:

- **Add Output:** Creates a new output (using the [Edit Analysis Outputs](#) dialog)
- **Edit Output:** Edits the selected existing output from the table (using the [Edit Analysis Outputs](#) dialog)
- **Delete Output:** Deletes the selected existing output from the table

Constraints Tab

The Constraints tab allows you to define limits on the performance of the building. Examples of constraints that might be applied to an optimisation analysis are "Discomfort hours must be less than 200" or "Construction cost must be greater than \$6m".



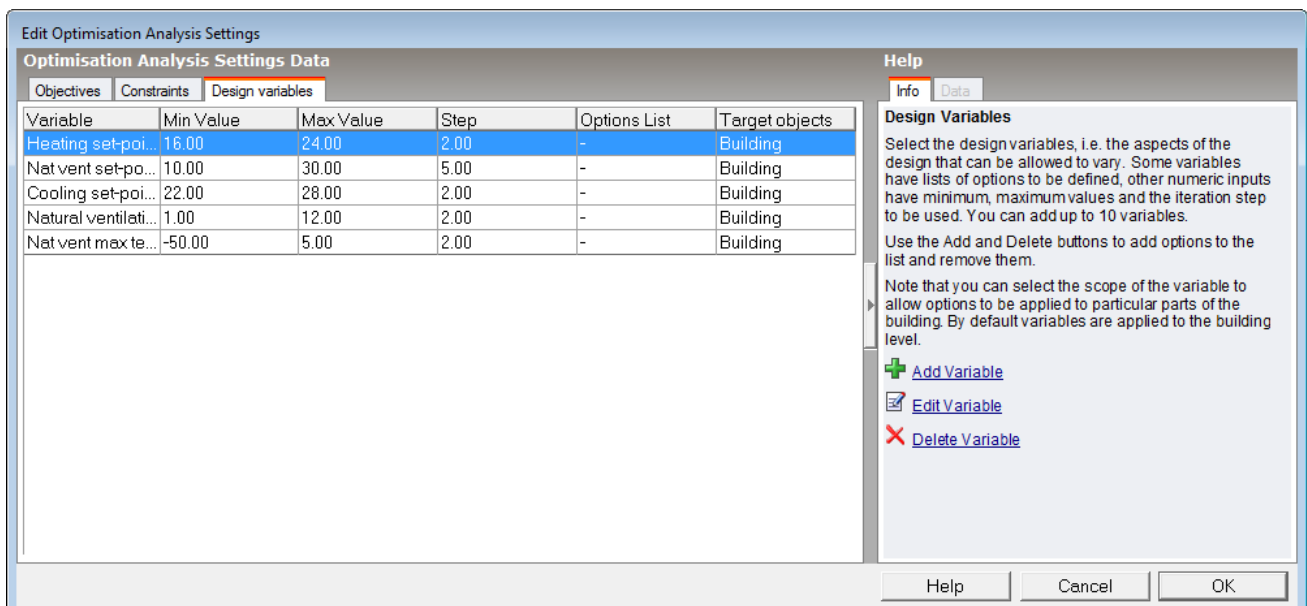
Any number of constraints can be added, or the analysis can be run without constraints.

Controls on the Info panel are:

- **Add Constraint:** Creates a new objective (using the [Edit Optimisation Constraints](#) dialog).
- **Edit Constraint:** Edits the selected existing constraint selected in the grid (using the [Edit Optimisation Constraints](#) dialog)
- **Delete Constraint:** Deletes the selected existing output from the table.

Variables Tab

The variables tab allows you to define the elements of the building design that can be allowed to vary as well as the allowed values that the variable can take on during the analysis. You can select from a wide range of variables such as **Window to wall %**, **Heating setpoint temperature**, **HVAC Template**, **External wall construction** etc.



Note: the data in the Min and Max Value columns are always displayed in SI units even when IP units are being used.

List data types such as constructions, glazing or templates require a set of options to be selected to define the allowed variable options.

Any number of variables can be added.

Controls on the Info panel are:

- **Add Variable:** Creates a new Design Variable (using [Edit Analysis Variable](#) dialog).
- **Edit Variable:** Edits the selected existing Design Variable from the table (using [Edit Analysis Variable](#) dialog).
- **Delete Variable:** Deletes the selected existing Design Variable from the table.

Tip: If you aren't sure whether some variables should be included in an optimisation or not, you can include them and then use [Sensitivity Analysis tools to make a statistical assessment](#) of whether the variable has a meaningful impact on the aspects of building performance that are under consideration. If not then the variable should be removed from the list and so excluded from future analyses.

Design variable definition order

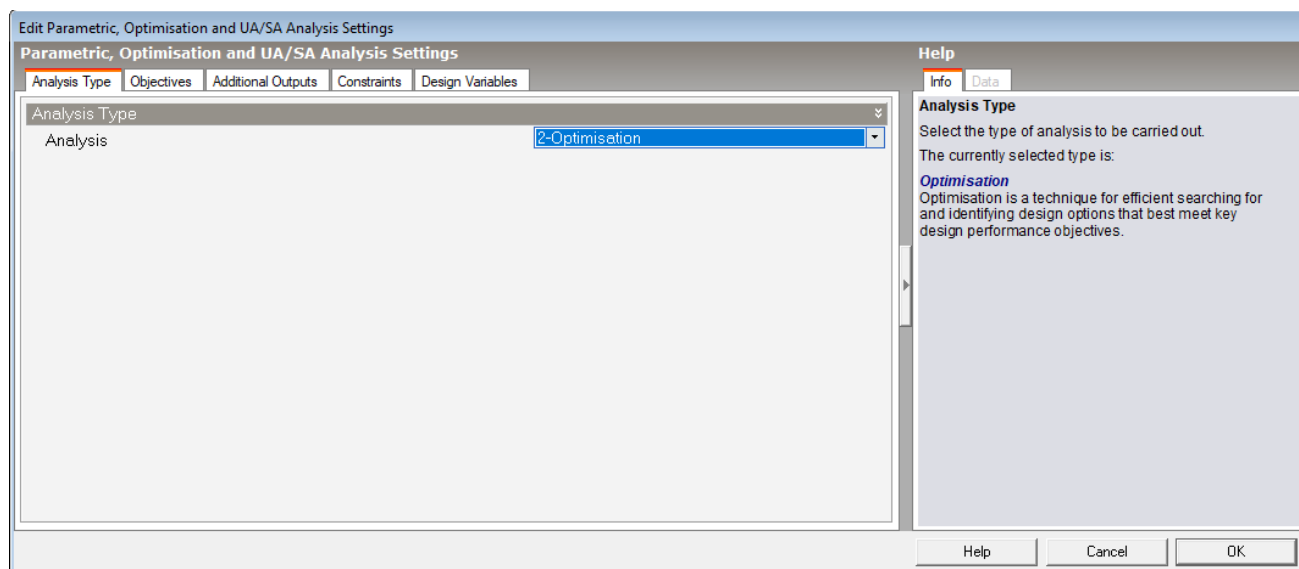
When using templates as Design variables in Parametric, Optimisation, or UA/SA Analysis, the order in which the variables are added to the list can affect the results. This happens when a template and one of its attributes are both included as Design variables, in which case the variable appearing lower in the list determines the value used in the analysis.

For example, if 2 variables to be included in the list of variables are **Miscellaneous power density** and **Activity template**, the effect of variations in **Miscellaneous power density** will only have an impact on results if the corresponding item in the list appears below the Activity template item. This is because the variables are applied to the model in the order they are defined in the list, and, because the Activity template also loads **Miscellaneous power density** values, the value loaded by the template will be overridden by the value defined by the **Miscellaneous power density** variable.

Optimisation - Analysis Type

Accessed from [Parametric Settings](#) dialog

Data on the **Analysis Type** tab allows you to define the type of analysis.



Analysis

The following options are available:

- **1-Parametric analysis** - to run a parametric analysis with up to 2 [design variables](#) and any number of [Output KPIs](#) on the Y-axis.
- **2-Optimisation** - to run an optimisation with 1 or 2 design objectives and any number of [design variables](#), [constraints](#) and [additional outputs](#).
- **3-Uncertainty/Sensitivity analysis** - to run an uncertainty and sensitivity analysis (UA/SA) with any number of [outputs](#) and [design variables](#). See the specific [section on UA/SA](#) for more details on this option.

The analysis to select for optimisation is **2-Optimisation**.

Optimisation - Objectives Dialog

Accessed from [Parametric Settings](#) dialog

Name

Your name for the objective for your reference, e.g. "Minimise ASHRAE 55 discomfort hours"

Objective KPI

Select a predefined or custom objective [Summary Output Key Performance Indicator \(KPI\)](#) from the list.

Minimise or Maximise

Select whether you wish to maximise or minimise the selected objective. By convention objectives are usually set up in such a way that the objective is minimised. For example for a comfort-related objective it is usual to minimise discomfort rather than maximise comfort. Minimising both objectives means that the Pareto Front will form the lower left portion of the scatter plot showing all results which is the conventional way to display optimisation results.

Optimisation - Additional Outputs Dialog

Accessed from [Parametric Settings](#) dialog

Name

Your name for the output for your reference, e.g. "Total Lighting".

Output KPI

Select a predefined or custom Summary Output [Key Performance Indicator \(KPI\)](#) from the browse list.

Units

The units of predefined KPIs are defined internally by the selected KPI itself. For custom KPIs the units are defined by the script that you write and in those cases it is usual to include the units within the name of the KPI. For example in the [Custom Script KPI Tutorial](#) the custom KPI created is given the name "EUI (kWh/m2)".

Optimisation - Constraints Dialog

Accessed from [Parametric Settings](#) dialog

Optimisation Constraints allow you to impose upper or lower limits on the performance of the building. Examples of constraints that might be applied to an optimisation analysis are "Discomfort hours must be less than 200" or "Construction cost must be greater than \$6m".

Name

Your name for the constraint (e.g. "ASHRAE 55 discomfort hours < 200")

Constraint KPI

Select a predefined or custom constraint [Summary Output Key Performance Indicator \(KPI\)](#) from the browse list.

Less than or Greater than

Select whether the criterion refers to an upper or lower limit.

- **1-Less than** - the KPI must have a value less than that entered in the next text box.
- **2-Greater than** - the KPI must have a value greater than that entered in the next text box.

Value

Enter the upper or lower limit value for the KPI.

Units

The units of predefined KPIs are defined internally by the selected KPI itself. For custom KPIs the units are defined by the script that you write and in those cases it is usual to include the units within the name of the KPI. For example in the [Custom Script KPI Tutorial](#) the custom KPI created is given the name "EUI (kWh/m2)".

Optimisation - Design Variables Dialog

Accessed from [Parametric Settings](#) dialog

Variable type

Select a predefined or custom variable from the [variables](#) browse list.

Important Note: All variable options are listed for selection regardless of the state of the model so care must be taken to select variables that are appropriate. For example, if the [Scheduled](#) natural ventilation model option is used then changes in the [% External window opens](#) variable (which only applies to [Calculated](#) natural ventilation) won't affect results.

Min value

The lowest value that this numeric design variable can take on. This option is only visible when a numeric variable type has been selected (above).

Max value

The highest value that this numeric design variable can take on. This option is only visible when a numeric variable type has been selected (above).

Note: the data in the **Min** and **Max Value** columns is always displayed in SI units even when IP units are being used.

Step (parametric)

Not used for optimisation.

Step (optimisation)

Enter the step between possible variable options. This option is only visible when a numeric variable type has been selected (above).

The possible values for the variables are Min value, Min value + Step, Min value + 2 x Step, ... MaxValue. For example if Min value is 16, Step (optimisation) is 2 and Max value is 24, then possible values for this numeric design variable are: 16, 18, 20, 22 and 24.

Options list

Click on the browse button to open a list of components or templates for selection. Items can be selected by checking the appropriate checkboxes. Each item selected will be an option that can be applied during the optimisation process.

When a template is selected as the variable type, one of the selected templates from the list are loaded in each iteration of the optimisation analysis.

This option is only visible when a list variable type has been selected (above).

After the list option selection process has been completed the **Variable Option Order** dialog is displayed. See below.

List Variable Option Ordering

During the process of using the [Options list](#) browse button for selecting discrete design variables, a **Variable Option Order** dialog appears after selecting the list options to use and closing **Select design options** dialog. In this **Variable Option Order** dialog, you can define the order in which the variable options are listed. Depending on the case it may or may not be important to list the options in the right order.

If the list of components or templates represents some sort of numerical sequence then it is generally important to set the order of those options in an ascending or descending order, otherwise it is not normally necessary.

- **Example where ordering is important:** A case where a glazing type variable options being selected consist of incrementally varying U-value, such as a list of 4 glazing systems with U-values of 1.0, 1.5, 2.0 and 2.5 W/m²-K. In this case, ordering the selection (in ascending or descending order) will help the optimiser to work more efficiently.
- **Example where ordering is NOT important:** A case where HVAC System Template options being selected have completely different HVAC systems such as a list of 4 HVAC templates, Fan coil units, Hot water radiators with nat vent, VAV and reversible chilled/heated floor. In this case there is no clear sequence, therefore ordering the list is not important for efficient optimisation analysis.

Target

Target objects

This option allows you to select one or more object(s) to which the variations are to be applied. The object(s) you select here are the places in the model where the variations will actually be applied. Normal model data inheritance rules apply so if you set the building as the target then the change will set for the building but will also inherit down to block, zone, surface and opening levels where appropriate. In this case any hard set data at block or lower levels will prevent inheritance in the normal way.

Tip: For models imported via [gbXML](#) any construction or glazing assignments are set in the DesignBuilder model at surface level and so the normal inheritance paths used for construction and glazing will be overridden. Therefore in this case, if you need to parametrically adjust construction or glazing surface

properties you may need to apply the changes to each surface individually by selecting the appropriate surfaces as targets.

Key

Variable key

This data is available for variables of the **Custom IDF** and **Custom Script** categories. Enter the name of the variable to be used in one or more corresponding DB Erl / EMS [<Var, VarName, DefaultValue>](#) statements. The variable key entered here corresponds with the **VarName** value in the DB Erl script.

List Of Variables

When creating a [Design variable](#) it is necessary to select a [variable type](#). This topic summarises the variable types available for selection in the browse list.

Variables List

Natural ventilation/infiltration

- **Natural ventilation rate** - Used only when the [Natural ventilation](#) model option is set to **Scheduled**.
- **Nat vent set-point temp** - Requires a Natural ventilation system to have been included in the base model.
- **Nat vent max temp difference** - Requires a Natural ventilation system to have been included in the base model.
- **Infiltration** - Scheduled natural ventilation only. When this variable is applied infiltration is automatically activated in the optimisation simulations even if the [Model infiltration](#) model option is switched off in the base model.
- **Airtightness** - Value range between 0 and 4 to cover the 5 Airtightness slider settings when using Calculated natural ventilation. When this variable is applied infiltration is automatically activated in the optimisation simulations even if the [Model infiltration](#) model option is switched off in the base model.
- **Crack template** - Used only when the [Natural ventilation](#) model option is set to **Calculated**. When this variable is applied infiltration is automatically activated in the optimisation simulations even if the [Model infiltration](#) model option is switched off in the base model.
- **% External window opens** - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
- **% Roof window opens** - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
- **Vent area** - the area of the auto-generated external vents (does not affect custom drawn vents) - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
- **Internal vent area** - the area of the auto-generated internal vents (does not affect custom drawn vents) - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
- **Roof vent area** - the area of the auto-generated roof vents (does not affect custom drawn vents) - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
- **Mixed mode switch off when raining** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
- **Mixed mode max wind speed** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
- **Mixed mode min temperature** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the

- Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
- **Mixed mode max temperature** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
 - **Mixed mode min enthalpy** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
 - **Mixed mode max enthalpy** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
 - **Mixed mode min dew point temperature** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
 - **Mixed mode max dew point temperature** - When this variable is applied the Mixed mode natural ventilation option is automatically activated in the optimisation simulations even if the [Mixed Mode Zone Equipment > Mixed mode on](#) model option is switched off in the base model. Note however that the **Natural Ventilation > On** model option must still be manually set in the base model for mixed mode setting to be applied in the simulation.
 - **Nat vent indoor min temp schedule** -
 - **External window operation schedule** - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
 - **Internal window operation schedule** - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
 - **Roof window operation schedule** - Used only when the [Natural ventilation](#) model option is set to **Calculated**.
 - **Earth tube ventilation rate** - When this variable is applied the earthtube option is automatically activated in the optimisation simulations even if the [Earth Tube > Include earthtube](#) option is switched off in the base model.
 - **Limit value of opening nat vent modulation factor** -
 - **Nat vent modulation lower value of Tin-Tout** -
 - **Nat vent modulation upper value of Tin-Tout** -

Glazing/shading

- **Glazing template** - Select a list of glazing templates to be evaluated. As each is template is assessed its data is loaded to the model.
- **Glazing type** - Select a list of glazing components to be evaluated.
- **Facade type** - Select a list of façade types to be evaluated.
- **Window to wall %** - Select the range of window to wall ratios to be used in the analysis. For guidance on this setting and how it interacts with custom facades, see "Window to Wall %, Custom Facades and Parametric Tools " on page 409
- **Window frame type** - Select a list of constructions to be used to define the window frames in the analysis.
- **Window blind type** - When this variable is applied window blinds on external walls are automatically activated in the optimisation simulations even if the [Window shading](#) model option is switched off in the base model.
- **Window shading control schedule** - When this variable is applied window blinds on external walls are automatically activated in the optimisation simulations even if the [Window shading](#) model option is switched off in the base model.
- **Local shading type** - When this variable is applied local shading on external walls are automatically activated in the optimisation simulations even if the [Local shading](#) model option is switched off in the base model.
- **Internal glazing type** - Select a list of glazing components to be evaluated for internal windows.

- **Internal facade type** - Select a list of façade types to be evaluated.
- **% Internal glazing** - Select the range of % internal glazing (window to wall ratios for partitions) to be used in the analysis.
- **Internal window frame type** - Select a list of constructions to be used to define the frames for internal windows in the analysis.
- **% Internal window opens** -
- **Roof glazing type** - Select a list of roof glazing components to be evaluated.
- **Roof facade type** -
- **Roof window frame type** - Select a list of constructions to be used to define the roof window frames in the analysis.
- **Roof shading coefficient** -
- **Roof window blind type** - When this variable is applied window blinds on roof surfaces are automatically activated in the optimisation simulations even if the [Roof Window shading](#) model option is switched off in the base model.
- **Roof window shading control schedule** - When this variable is applied window blinds on roof surfaces are automatically activated in the optimisation simulations even if the [Roof Window shading](#) model option is switched off in the base model.

Activity/occupancy/comfort setpoints

- **Activity template** - Select a list of activity templates to be evaluated. As each is template is assessed its data is loaded to the model.
- **Occupancy density** -
- **Occupancy (Days/week)** -
- **Heating set-point temperature** -
- **Heating set-back temperature** - Note that heating setback temperatures only apply when the [Heating operation schedule](#) includes [values of 0.5](#) to indicate times of setback operation.
- **Cooling set-point temperature** -
- **Cooling set-back temperature** - Note that heating setback temperatures only apply when the [Cooling operation schedule](#) includes [values of 0.5](#) to indicate times of setback operation.
- **Heating set-point PMV** - Only applies when using the **4-Fanger comfort** option for the [Temperature control](#) option for a Detailed HVAC simulation.
- **Heating set-back PMV** - Only applies when using the **4-Fanger comfort** option for the [Temperature control](#) option for a Detailed HVAC simulation.
- **Cooling set-point PMV** - Only applies when using the **4-Fanger comfort** option for the [Temperature control](#) option for a Detailed HVAC simulation.
- **Cooling set-back PMV** - Only applies when using the **4-Fanger comfort** option for the [Temperature control](#) option for a Detailed HVAC simulation.
- **Mech vent set-point temp** -
- **Mech vent max temp difference** -
- **Mech vent rate per person** -
- **Mech vent rate per area** -

Constructions

- **Construction template** - Select a list of construction templates to be evaluated. As each is template is assessed its data is loaded to the model. This option only applies when the (default) [General](#) construction and glazing data model option is set.
- **Insulation** - Like the Construction template variable above but with a limited set of options based on the "Early Design" Constructions templates and intended for use together with the Thermal mass variable described below. This variable only applies when the [Pre-design](#) construction and glazing data model option is set. The variable is numeric and parametric values are mapped as follows: 0=Uninsulated, 1=Typical ref, 2=Energy code, 3=Best practice. You should normally accept the default range 0-3. You can read about how the Pre-design construction templates are set up and controlled on the "Legislative Region Templates - Standard Insulation" on page 1510 help page.
- **Thermal mass** - See above for the Insulation variable. This variable only applies when the [Pre-design](#) construction and glazing data model option is set. The variable is numeric and parametric values are mapped as follows: 0=Light, 1=Medium, 2=Heavy. You should normally accept the default range 0-2.
- **External wall construction** - Select a list of [external wall](#) constructions.
- **Flat roof construction** - Select a list of [flat roof constructions](#).
- **Pitched roof construction (occ)** -

- **Pitched roof construction (unocc)** -
- **Semi-exposed ceiling construction** - Select a list of [semi-exposed ceiling constructions](#) between occupied and semi-exterior unconditioned zones above (e.g. ceilings between occupied zones and roofspaces).
- **Semi-exposed floor construction** - Select a list of [semi-exposed floor constructions](#) between occupied and semi-exterior unconditioned zones below (e.g. floors between occupied zones and basements).
- **Ground floor construction** - Select a list of [ground floor](#) constructions.
- **External floor construction** - Select a list of [external floor constructions](#).
- **Internal floor construction** - Select a list of [internal floor constructions](#).
- **Wall sub-surface construction** -
- **Partition sub-surface construction** -
- **Roof sub-surface construction** -
- **External door construction** -
- **Internal door construction** -
- **Internal thermal mass construction** - Select a list of Internal thermal mass constructions to be tested. This is the Construction used for hanging partitions and when a non-zero value has been entered for the zone internal thermal mass exposed area on the Constructions tab under the [Internal Thermal Mass](#) header.
- **Component block material** -
- **Wall below grade construction** -
- **Partition construction** -
- **Basement ground floor construction** -

Internal gains/lighting

- **Lighting template** - Select a list of lighting templates to be evaluated. As each is template is assessed its data is loaded to the model.
- **Computer power density** - Used only when the [Equipment gain units](#) model option is set to **1-Power density**. When this variable is applied computer gains are automatically activated in the optimisation simulations even if the [Computer > On](#) model option is switched off in the base model.
- **Equipment power density** - Used only when the [Equipment gain units](#) model option is set to **1-Power density**. When this variable is applied Office equipment gains are automatically activated in the optimisation simulations even if the [Office Equipment > On](#) model option is switched off in the base model.
- **Miscellaneous power density** - Used only when the [Equipment gain units](#) model option is set to **1-Power density**. When this variable is applied Miscellaneous gains are automatically activated in the optimisation simulations even if the [Miscellaneous > On](#) model option is switched off in the base model.
- **Catering power density** - Used only when the [Equipment gain units](#) model option is set to **1-Power density**. When this variable is applied Catering gains are automatically activated in the optimisation simulations even if the [Catering > On](#) model option is switched off in the base model.
- **Process power density** - Used only when the [Equipment gain units](#) model option is set to **1-Power density**. When this variable is applied Process gains are automatically activated in the optimisation simulations even if the [Process > On](#) model option is switched off in the base model.
- **General lighting power density** - Used only when the [Lighting gain units](#) model option is set to **1-Power density**. When this variable is applied general lighting gains are automatically activated in the optimisation simulations even if the [General Lighting > On](#) model option is switched off in the base model.
- **General lighting normalised power density** - Used only when the [Lighting gain units](#) model option is set to **2-Normalised Power density**. When this variable is applied general lighting gains are automatically activated in the optimisation simulations even if the [General Lighting > On](#) model option is switched off in the base model.
- **Task lighting power density** - When this variable is applied task lighting gains are automatically activated in the optimisation simulations even if the [Task and Display Lighting > On](#) model option is switched off in the base model.
- **External lighting** -
- **All gains** - Used only when the [Gains data](#) model option is set to **Lumped**.

HVAC systems

- **Simple HVAC template (Simple HVAC)** - Select a list of HVAC templates to be evaluated. As each is template is assessed its data is loaded to the model. This option applies only when using the [Simple HVAC](#) model option.
- **HVAC template (Detailed HVAC)** - Select a list of Detailed HVAC templates to be evaluated. As each is template is assessed its data is loaded to the model. This option applies only when using the [Detailed HVAC](#) model option. Note that Detailed HVAC templates only support templates with a single HVAC zone group. The way that zones are allocated to the zone group is as follows. If the building is the [target](#) then all zones that don't use the **<None>** Simple HVAC template are included in the HVAC zone. Otherwise if individual zones are set as targets then only those zones are loaded to the HVAC zone group. In this case zones that are not included as targets will not be assigned an HVAC system and so will be unconditioned.
- **Mechanical ventilation rate** -
- **Auxiliary energy** - Simple HVAC only
- **Pump etc energy** - Simple HVAC only
- **Heating system seasonal CoP** - Simple HVAC only
- **Cooling system seasonal CoP** - Simple HVAC only
- **Heating operation schedule** -
- **Cooling operation schedule** -

Location/weather/orientation

- **Location template** - Select a list of Location templates to be evaluated. As each is template is assessed its data is loaded to the model at site level.
- **Hourly weather data** - Select a list of Hourly weather data to be evaluated. As each is Hourly weather component is assessed its data is loaded to the model at site level.
- **Site orientation** - adjusts the orientation of all objects on the site. See also **Building rotation** in the Other category.

Note: Although the above site-related settings are actually made at site level behind the scenes, the Target should nevertheless be set to Building, not Site.

Renewables

- **Load centre 1** - the load centre variables can be useful for varying the number of PV panels in a study. Set up and select a number of load centres each one having a different number of PV panels.
- **Load centre 2** - see Load centre 1
- **Load centre 3** - see Load centre 1
- **Load centre 4** - see Load centre 1
- **Load centre 5** - see Load centre 1

Other

- **Building rotation** - adjusts the orientation of the building relative to the site. Building objects are rotated about the geometric centre point of all building objects. **Care should be taken to ensure that building objects would not intersect with any site objects for all possible building rotations.** Note that component blocks that are part of the wider site are not rotated with the building, but component blocks that are part of the building are. See under the help for [Level](#) setting under Component block settings. See also [Site orientation](#) in the Location/weather/orientation category.
- **Zone capacitance multiplier** -

Custom EMS

- **Custom EMS** - a special variable type which acts on [EMS variables](#) within EMS scripts. For example a line of EMS script such as:

SET HeatingSetpoint = <Var,SPTemp,20>,
could be written to indicate that HeatSetpoint can vary and if a Custom EMS variable with **Variable key** of SPTemp was included in the optimisation, then for each iteration of the simulation, the code in

angle brackets would be automatically replaced, behind the scenes, with the setpoint temperature currently being tested, e.g.
 SET HeatingSetpoint = 20.7,

The process involved in setting up Custom EMS variables is explained in detail in the "Optimisation Tutorial - Custom EMS Variable" on page 1739

Custom IDF

- **Custom IDF** - a special variable type which adds a parametric variation to IDF files through an **Output:PreprocessorMessage** statement. This statement can be processed using [CS-Script](#) code together with the [EpNet](#) library for example.

Extra IDF is added to main IDF in the form:

```
Output:PreprocessorMessage,
EpNet Script,
Information,
Variable key,
CurrentValue;
```

Where [Variable key](#) is the setting made on the dialog, matching up with corresponding code in the scripts and CurrentValue would be automatically replaced with the value of the variable for the current iteration.

Important Note: You must write extra script to pre-process the IDF **Output:PreprocessorMessage** IDF statement to ensure that variables of type **Custom IDF** have an effect on the simulations. This might typically be done in the **BeforeEnergySimulation** callback script which can be used to process the Output:PreprocessorMessage IDF data generated by the Custom IDF variable and insert the setting for the CurrentValue for each simulation.

Custom Script

- **Custom Script** - where you can write a script which applies changes to the model via the DesignBuilder API in response to the variable value passed in. A hook function is called in all active [CS-Scripts](#) and plugins to allow you to define your own custom variants of the model based on current variable values. This is an extremely flexible way to set up design variables as it allows you to control the exact impact on the model of each variable change.

The process involved in setting up Custom Script variables is explained in detail in the [Custom Script Variable Tutorial](#).

Tip: The [Extensibility Tools User Guide](#) is the main source of help for scripts and the API.

Summary Outputs (KPIs)

When creating an [Objective](#), [Additional Output](#) or [Constraint](#) it is necessary to select a **Summary output** (aka KPI) to define the output involved. This topic summarises the Summary outputs available for selection.

Note: In general, outputs should be selected in the Simulation output options to allow them to be reported as results in Optimisation. For example, The [Lighting energy](#) KPI is only reported in parametric simulations when [Internal gains including solar](#) are requested as "graphable outputs" in [Simulation output options](#).

List Of KPIs

You can find more detailed descriptions of many of these outputs on the "Simulation Detailed Results" on page 1622 page.

Comfort

All comfort outputs are for the whole building using an area-weighted average of discomfort calculated for each zone. Note that the comfort-related outputs are termed "discomfort" because by convention DesignBuilder aims to minimise objectives and in the context of standardising optimisation outputs it is better to minimise discomfort than to maximise comfort.

- **Discomfort ASHRAE 55 (sum clo)** - total discomfort hours when summer clothes are worn (see comments below for Discomfort (all clo))
- **Discomfort ASHRAE 55 (win clo)** - total discomfort hours when winter clothes are worn (see comments below for Discomfort (all clo))
- **Discomfort ASHRAE 55 (all clo)** - total discomfort hours when winter or summer clothes are worn based on [ASHRAE 55](#) method. Note that this objective is not the same as the building level aggregation method used by EnergyPlus in the Summary output html data which adds up hours in which *any* zone in the building is uncomfortable. If you would like to only consider [occupied hours](#) in your comfort analysis then make sure this option is set in the main Simulation options dialog.
- **Discomfort Summer ASHRAE 55 Adaptive 80% Acceptability** - Reports whether the operative temperature falls into the 80% acceptability limits of the adaptive comfort in ASHRAE 55-2010 during summer occupied hours. A value of 1 means within (inclusive) the limits, i.e. the zone operative temperature is no more than 2.5°C from than the ASHRAE 55 Adaptive model temperature. A value of 0 means outside the limits (i.e. uncomfortable), and a value of -1 means not applicable, i.e. the running average outdoor air temperature is below 10°C.
- **Discomfort Summer ASHRAE 55 Adaptive 90% Acceptability** - Reports whether the operative temperature falls into the 90% acceptability limits of the adaptive comfort in ASHRAE 55-2010 during summer occupied hours. A value of 1 means within (inclusive) the limits, i.e. the zone operative temperature is no more than 3.5°C from than the ASHRAE 55 Adaptive model temperature. A value of 0 means outside the limits (i.e. uncomfortable).
- **Discomfort Summer CEN 15251 Adaptive Category I** - reports whether the operative temperature falls into the Category I (90% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007 during occupied hours when the running mean outdoor air temperature is higher than 10°C. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits.
- **Discomfort Summer CEN 15251 Adaptive Category II** - reports whether the operative temperature falls into the Category II (80% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007 during occupied hours when the running mean outdoor air temperature is higher than 10°C. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits.
- **Discomfort Summer CEN 15251 Adaptive Category III** - reports whether the operative temperature falls into the Category III (65% acceptability) limits of the adaptive comfort in the European Standard EN15251-2007 during occupied hours when the running mean outdoor air temperature is higher than 10°C. A value of 1 means within (inclusive) the limits, a value of 0 means outside the limits.

Cost

- **Capital cost (Building Construction Capex)** - The basic cost of the building and site construction . This is the only construction cost option available when the [cost model](#) is set to **1-Basic**.
- **Capital cost (Building + Design Capex)** - The cost of the building + site construction plus the design, overhead and contractor profit costs. This option should only be used when the [cost model](#) is set to **2-UNIFORMAT II** or **3-NRM1**.
- **Capital cost (Total Capex)** - The cost of the building + site construction plus the design, overhead and contractor profit costs + cost of risk and inflation etc. This option should only be used when the [cost model](#) is set to **2-UNIFORMAT II** or **3-NRM1**.

- **Operational cost (Opex)** - annual cost of the fuel used to operate the building. To use this KPI you must set up tariff data on the [Economics](#) model data tab and have the [Detailed HVAC](#) model option set.
- **LCC total present value** - The [life-cycle](#) cost calculated by EnergyPlus including the construction cost based on the selected [cost model](#). To use the LCC KPI you must set up tariff data as well as construction cost data in the model. This KPI requires settings on the Economics tab and the [Detailed HVAC](#) model option. For more details see notes below under expanding header.

Important Advice for LCC and Opex KPIs

The **Operational cost (Opex)** and **LCC total present value** KPIs both require the [Detailed HVAC](#) model option to be used as they use settings on the [Economics](#) model data tab which is only available for Detailed HVAC.

The **LCC total present value** values calculated in optimisation and other parametric simulations will only correspond with the value calculated directly by EnergyPlus if the basis for calculation is the same. In practice this means that the following 2 requirements must be met:

- Any additional [Adjustment](#) costs on the [Component cost](#) dialog should be removed. Set all costs to zero and the [Regional adjustment factor](#) to 1.
- The **1-BeginningOfYear** [Discounting convention](#) on the Life cycle cost dialog must be used.

This should be the case for models created in v7.

Tip: You can carry out powerful analyses to simultaneously optimise whole life cycle cost and whole life cycle environmental impact performance using DesignBuilder's optimisation tools. Choose the LCC and [LCA](#) KPIs for this sort of analysis.

Daylight

No daylight-specific KPIs are available in current versions of DesignBuilder.

Energy and Loads

- **Net site energy** is the net energy consumed on site (total site fuel consumption minus any on-site generation). For Simple HVAC, this value is affected by the COP values set on the HVAC tab and is calculated as the sum of the fuel consumption minus any generation. In Detailed HVAC it is the value reported in the Summary ABUPS report, i.e. the sum of the fuel consumption minus generation. You must select either the [ABUPS Summary](#) or the [All Summary](#) annual report to enable this KPI.
- **Net primary energy** is the net primary energy due to energy consumed on site (total minus any on-site generation). This output should only be selected when using Detailed HVAC.
- **Total site energy** is the total gross energy consumed on site. For Simple HVAC, this value is affected by the COP values set on the HVAC tab and is calculated as the sum of the fuel outputs. In Detailed HVAC it is the value reported in the Summary ABUPS report which is also the same as the sum of the fuel consumption reported in the DesignBuilder results. You must select either the [ABUPS Summary](#) or the [All Summary](#) annual report to enable this KPI.
- **Total primary energy** is the total primary energy due to energy consumed on site. This output should only be selected when using Detailed HVAC.
- **Heating (Electricity)** - electricity consumption used in heating
- **Heating (Gas)** - gas consumption used in heating
- **Heating (Oil)** - oil consumption used in heating
- **Heating (Solid fuel)** - solid fuel consumption used in heating
- **Heating (Bottled Gas)** - bottled gas (e.g. propane) consumption used in heating
- **Heating (Other fuels)** - Other fuels (e.g. biofuel) consumption used in heating
- **Heating load** - same as **Zone sensible heating** below.
- **Cooling (Electricity)** - electricity consumption used in cooling
- **Cooling (Gas)** - gas consumption used in cooling
- **Cooling (Oil)** - oil consumption used in cooling
- **Cooling (Solid fuel)** - solid fuel consumption used in cooling
- **Cooling (Bottled Gas)** - bottled gas (e.g. propane) consumption used in cooling

- **Cooling (Other fuels)** - other fuel (e.g. biofuel) consumption used in cooling
- **Cooling energy** - sensible + latent cooling transfer to the supply air from the AHU cooling coil + any single zone unitary and fan coil units in the building.
- **Cooling load** - same as **Cooling energy**, above.
- **DHW (Electricity)** - electricity consumption used in DHW
- **DHW (Gas)** - gas consumption used in DHW
- **DHW (Oil)** - oil consumption used in DHW
- **DHW (Solid fuel)** - solid fuel consumption used in DHW
- **DHW (Bottled gas)** - bottled gas (e.g. propane) consumption used in DHW
- **DHW (Other fuels)** - other fuel (e.g. biofuel) consumption used in DHW
- **AHU heating energy** - energy supplied by the AHU heating coil when using Detailed HVAC data.
- **Fans** - fan energy consumption when using Simple HVAC.
- **Pumps** - pump energy consumption when using Simple HVAC.
- **Preheat energy** - energy supplied by preheat coils to temper the outside air before it enters the outside air mixing box when using Detailed HVAC data.
- **Reheat energy** - total reheat energy
- **Radiant heater energy** - total radiant heater energy
- **DHW** - total DHW energy
- **Process energy** - total process energy to zones
- **Catering energy** - total catering gains to zones over the simulation period
- **Lighting Energy** - total lighting energy consumption over the simulation period
- **System Fans** - System fan energy consumption to zones over the simulation period when using Detailed HVAC.
- **System Pumps** - System pump energy consumption when using Detailed HVAC.
- **System Misc** - System miscellaneous energy consumption when using Detailed HVAC.
- **Electricity** - total annual electricity consumption
- **Gas** - total annual gas consumption
- **Oil** - total annual oil consumption
- **Solid** - total annual solid fuel (typically coal) consumption
- **Bot Gas** - total annual bottled gas fuel consumption
- **Other** - total annual other fuel consumption
- **Heat Rejection** - heat rejection energy by the chillers, e.g. through cooling towers, ground heat exchangers, fluid coolers.
- **Generated Electricity** - total electricity generated by solar PV and wind.
- **Sys sens heating** - total sensible heating provided by HVAC components to the air stream and zones.
- **Sys sens cooling** - total sensible cooling provided by HVAC components to the air stream and zones.
- **Sys lat heating** - total latent heating provided by HVAC components to the air stream and zones.
- **Sys lat cooling** - total latent cooling provided by HVAC components to the air stream and zones.
- **Heat rec sens heat** - total sensible heat recovered by heat recovery systems.
- **Heat rec total heat** - total sensible + latent heat recovered by heat recovery systems.
- **Heat rec sens cool** - total sensible cooling energy recovered by heat recovery systems.
- **Heat rec total cool** - total cooling energy recovered by heat recovery systems.
- **Chiller load** - Net annual chiller load is calculated as Chiller load – Chiller evaporator cooling rate [kWh] when using Detailed HVAC. Not available in Simple HVAC.
- **Room Electricity** - Electric energy consumed by zone equipment such as computers, office equipment etc.
- **Room Gas** - Gas energy consumed by Miscellaneous, Catering and Process zone equipment.
- **Room Oil** - Oil energy consumed by zone equipment.
- **Room Solid** - Solid fuel energy consumed by zone equipment.
- **Room Bottled Gas** - Bottled gas energy consumed by zone equipment.
- **Room Other** - Other energy consumed by zone equipment.
- **Zone sensible cooling** - equivalent to the [Zone Sensible Cooling](#) output, the overall sensible cooling effect on the zone of any air introduced into the zone through the HVAC system. It includes any 'free cooling' due to introduction of relatively cool outside air and the heating effect of any fans present. Note that this output doesn't include the cooling effect of surface radiant cooling systems such as chilled ceilings whose cooling effect is accounted for in the ceiling, roof etc surface heat gains output.
- **Zone sensible heating** - equivalent to the [Zone Sensible Heating](#) output, the overall sensible heating effect of any air introduced into the zone through the HVAC system including any 'free heating' due to introduction of relatively warm outside air and the heating effects of fans. Note that this output

doesn't include the heating effect of surface radiant heating systems such as heated floors whose heat is accounted for in the floor etc surface heat gains output.

Environmental impact

- **Operational CO2 emissions** - annual operational carbon emissions due to fuel consumption in the building. Fuel consumption is calculated through an EnergyPlus simulation. Carbon emissions are calculated from fuel consumption using the [Carbon emission factors](#) on the [Legislative regions](#) dialog.
- **LCA (Simple)** - Life Cycle Analysis giving the net total lifetime carbon emissions. The sum of the total [embodied carbon](#) + operational carbon emissions due to burning of fuel minus beneficial impact of on-site energy generation calculated over the assumed lifetime of the building. Some settings for this including the LCA time of study can be found on the [Carbon](#) tab of the Model data dialog. **LCA (Simple)** is calculated simply as **Embodied CO2** + [Years of study](#) x Annual **Operational CO2 emissions**.
- **Embodied CO2** - total [embodied carbon](#) as reported on the [Cost and Carbon tab](#).

Heat gains

- **Display lighting gains** - total task + display lighting gains to zones over the simulation period
- **General lighting gains** - total general lighting gains to zones over the simulation period
- **Misc gains** - miscellaneous gains to zones over the simulation period
- **Mach gains** - process gains to zones over the simulation period
- **Cooking gains** - catering gains to zones over the simulation period
- **Computer gains** - computer gains to zones over the simulation period
- **Occupancy gains** - occupancy gains to zones over the simulation period
- **Solar gains internal Windows** - solar gains to zones through interior windows over the simulation period
- **Solar gains external Windows** - solar gains to zones through exterior windows over the simulation period
- **Glazing gain** - thermal conduction gains through windows
- **Walls gain** - thermal conduction gains through walls, including any heating/cooling effects from radiant surface HVAC systems.
- **Ceilings gain** - thermal conduction gains ceilings, including any cooling effects from chilled ceilings.
- **Floors gain** - thermal conduction gains through floors, including any heating effects from heated floors.
- **Solid floors gain** - thermal conduction gains through ground floors
- **Partitions gain** - thermal conduction gains through partitions
- **Roofs gain** - thermal conduction gains through roofs, including any cooling effects from chilled ceilings.
- **Rooflights gain** - thermal conduction gains through rooflights
- **Ext floors gain** - thermal conduction gains through exterior floors, including any heating effects from heated floors.
- **Sens cooling energy** - sensible only cooling transfer from the cooling coil to the supply air.
- **Ext airflow gain** - sensible gain due to all external airflow including natural ventilation through openings and infiltration through cracks when using Calculated natural ventilation.
- **Ext infiltration gain** - sensible gain due to infiltration air when using scheduled natural ventilation.
- **Ext nat vent gain** - sensible gain due to natural ventilation air when using scheduled natural ventilation.
- **Ext mech vent gain** - sensible gain due to mechanical ventilation air when using scheduled natural ventilation and [Room ventilation](#) model option.

Unmet loads

The [unmet load](#) hours KPIs described below are the same values as those reported in the EnergyPlus [Summary](#) report. The Unmet load hours are described in more detail in the "ASHRAE 90.1 Appendix G PRM Reference" on page 2135 topic and also in the [DesignBuilder LEED and ASHRAE 90.1 Appendix G User Guide](#).

- **Unmet load hours all** - number of hours where the heating or cooling setpoints were not met over the simulation period for the whole building.
- **Unmet load hours heating** - as above but for cooling only.

- **Unmet load hours cooling** - as above but for heating only.

Custom script

You can add your own custom script KPIs which allow you to read and, if necessary post-process, simulation results from any EnergyPlus output file or other source and write it to a specific database table using the API. You can use either CS-Script or Python scripts to do this. Note that the name you give to your Custom script KPI is important as it will be used to reference it in the script that you write.

The Custom script mechanism works as follows. For each simulation:

1. In the **AfterEnergySimulation** hook that is called immediately after the simulation has been completed and results have been loaded to DesignBuilder, the script must create a new site level table called "ParamResultsTmp". It is necessary to first remove any existing table of the same name.
2. The "ParamResultsTmp" table has 2 fields.
3. Records should be added to the table, one for each custom script KPI used in the simulation.
4. The first field in each record (index 0) should match the name of the KPI, the second field (index 1) should be the value to be read for this KPI for the current simulation.
5. DesignBuilder reads the table after the script has finished executing and results are loaded to the parametric (optimisation, UA/SA or parametric analysis) system.

Note: While all simulations will call the **AfterEnergySimulation** hook, only parametric simulations actually read the ParamResultsTmp table created in the script.

Custom Script KPI Tutorials

The easiest way to understand how custom script KPIs works is to work through the Custom Script KPI tutorials which explain the steps involved:

- "Optimisation Tutorial - Custom Script KPI Example 1 - db_eplusout_reader Python Libraries" on page 1752
- "Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760
- "Optimisation Tutorial - Custom Script KPI Example 3 - Using EMS + Python Scripts to Optimise Controls for Comfort and Heating" on page 1770
- "Optimisation Tutorial - Custom Script KPI Example 4 - Adding Python Script for TM52 Comfort to Example 3" on page 1778

User defined

No user-defined options are currently available.

Optimisation Calculation Options Dialog

To start an optimisation analysis go to the [Simulation screen](#) and click on the **Optimisation + UA/SA** tab. Calculations are updated using the Update toolbar icon in the usual way. The data on the Optimisation Calculation Options dialog is used to provide options to control the optimisation analysis as well as outputs to help you to understand the progress being made as the calculations continue.

Note: It is necessary to run the base simulation with monthly/annual outputs before opening the **Optimisation + UA/SA** tab. The optimisation simulations will all be run using the base calculation and model options. **So the simulation period and all other options used in the optimisation are the same as those defined on the [Simulation calculation options](#) dialog.**

There are 2 tabs on the Dialog:

- [Optimisation Calculation Options Dialog - General](#)
- [Optimisation Calculation Options Dialog - Simulation Manager](#)

You can learn about the outputs generated on the graph on this dialog during the optimisation:

[Optimisation Outputs](#)

Optimisation Calculation Options Dialog - General Tab

Accessed from [Optimisation Calculation Options Dialog](#)

The genetic algorithms (GA) used by DesignBuilder require a number of settings to control the way that the solutions evolve. These and other settings are made on the General tab of the Calculation Options dialog as described below.

The engine used in for optimisation is JEA - a powerful optimisation engine developed by EnSims.

Calculation Description

Calculation description

Enter a description for the calculation. This text will be used to identify the results on the graph and any other related outputs.

From

The **Start date** set up for the base simulation on the analysis screen is displayed but cannot be edited here.

To

The **End date** set up for the base simulation on the analysis screen is displayed but cannot be edited here.

Optimisation Options

Generation population size

The number of solutions to be evaluated in each generation. The first-generation simulations are selected at random but form the basis of subsequent optimisation generations. More design variables require a larger population size in general. The choice may also be influenced by the number of processor cores you have for running these simulations. The default value is 15.

Tip: The Generation population size will typically be between 10 and 50 depending on the size of the analysis. For 3 design variables with say 10 options each then choose an initial population size of between 10 and 20. For 10 variables, each with 10 options then choose a value between 20 and 50. Like other settings on this dialog, the best values to use for the fastest and most reliable convergence will come from experience. In the meantime, a value of 15 will work well for most typical applications.

Note: All simulations for a generation must complete before the next generation is started so if you have many parallel cores at your disposal and the time taken to run the simulations is the main bottleneck in calculations (as opposed to IDF generation) then a higher number here can be helpful.

Generations for convergence

This option is used to decide when to stop the optimisation simulations. As the optimisation process progresses and most optimum solutions have been identified, fewer and fewer new optimal solutions will be found and there comes a point when it's time to stop. At the end of each generation, DesignBuilder checks whether any new optimal solutions were found in the most recent **Generations for convergence** generations. If so, the simulations continue, otherwise the optimisation process is considered to be complete, and results are displayed. It isn't possible to be sure that no new optimum solutions wouldn't be found if the optimisation process continued, but this setting offers a simple way to stop simulations once it seems that most have been identified.

The default value is 5 generations. Enter a higher number to give you the best chance of finding all possible solutions, or enter a smaller number if you are less concerned about finding every possible optimum solution. Alternatively you can enter a very high number (e.g. same as **Maximum generations**) if you would like to control convergence manually by using the **Stop** button.

This setting works in conjunction with the [Max wall time](#) and [Max evaluations](#) settings - whichever limit is reached first will cause the optimisation to stop.

Apply constraints to Pareto front

This option allows you to control the way that constrained optimisation result sets are displayed but does not affect the optimisation process itself. It is only available when one or more [constraints](#) are applied. The options are:

- **1-Show all points** where all points (even those failing constraints) can appear on the Pareto front.
- **2-Apply constraints to Pareto front**, the default and recommended option where only feasible solutions are included on the Pareto front and infeasible solutions are shown in yellow.
- **3-Apply constraints to all graph points** where infeasible solutions failing constraints are not displayed at all.

The **Apply constraints to Pareto front** selection can be changed during the simulation.

Run all combinations

In some cases it can be useful to run simulations for all possible combinations of design variables without actually trying to optimise the results. This is in fact more like [Parametric Analysis](#), but because Parametric Analysis only allows a maximum of 2 design variables to be used, a different approach is required if you need to include more than 2 design variables.

To simulate all possible combinations, it is necessary to disable the optimisation process and systematically run all simulations.

Once all the simulations are finished you can [export](#) the full set of results to a spreadsheet for detailed analysis and for plotting as a parametric analysis.

Note: Because this option completely disables optimisation, if you stop the run before all combinations are simulated, results may not be suitable for interpreting either as parametric analysis or optimisation.

Advanced Options

Maximum generations

The maximum number of generations to be used will determine the time and computing resources required to complete the analysis. The value entered here will usually reflect the size and complexity of the analysis. Typical values are in the range 50-500 depending on the size of the problem and the population size.

Tip 1: Setting the right number of Maximum generations is not usually a crucial setting provided you use a large number such as 100 or more.

Tip 2: When seeking accurate optimal outputs and hence using a large value for Maximum generations it can be worth experimenting with higher mutation rates to help avoid early convergence.

Maximum population size

The number of solutions to be included in each generation may increase as the Pareto archive builds up. This is the maximum number of solutions that can be included in any one generation. This parameter specifies an upper limit and if exceeded, some existing Pareto solutions will be dropped.

Mutation rate

This setting defines how often random changes are applied to new solutions. A lower value is preferred for fast convergence and to avoid the algorithm behaving like a random trial and error. On the other hand it is important to avoid "premature convergence" where the optimisation settles on a Pareto front which is incomplete or inaccurate. Mutation helps to maintain diversity in the population to ensure that the parameter space is fully explored which in turn helps to avoid premature convergence and missing the full set of optimal solutions.

Low values of mutation rate (e.g. 0.2) can be used to avoid wasting time exploring non-optimal parts of the parameter space which leads to fast convergence; however the results may not be as accurate as when using higher values. DesignBuilder uses a balanced default of 0.4.

Crossover rate

Crossover is a genetic operator used to combine the genetic information of two parents to generate new offspring. It is one way to stochastically generate new solutions from an existing population, and analogous to the crossover that happens during sexual reproduction in biology. JEA uses a "hybrid integer" crossover method. The crossover rate defines how often the new solutions are created by merging features of existing solutions. A high value of 1.0 is normally used.

Tournament size

Tournament selection involves running several "tournaments" among a few individuals chosen at random from the population. The winner of each tournament (the one with the best fitness) is selected for crossover. With larger tournament sizes, weak individuals have a smaller chance to be selected, because, if a weak individual is selected to be in a tournament, there is a higher probability that a stronger individual is also in that tournament. In other words, the larger the tournament size, the harder the algorithm pushes towards the desired objectives. A tournament size of 1 is equivalent to random selection. A tournament size of 2 is normally a good balance.

Objective bias

This setting controls the level of the push for feasibility, i.e. meeting any constraints applied. It controls the ranking bias between Pareto ranking and constraint infeasibility scores. 0 means meeting constraints first. 1 means improving objectives first. 0.5 specifies a balanced strategy.

Maximum evaluations

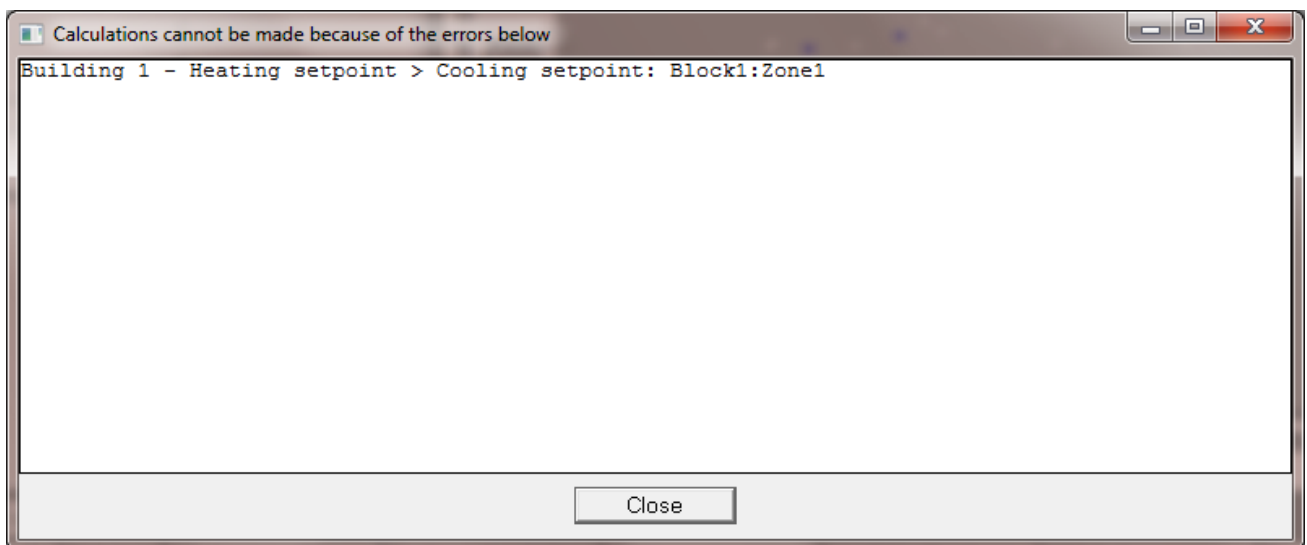
Enter the maximum number of iterations you want the optimisation to run for. If the optimisations are still running after the number of iterations entered here they will be considered to be completed. This limit works in conjunction with the [Generations for convergence](#) and [Max wall time](#) settings - whichever limit is reached first will cause the optimisation to stop. If you don't want optimisations to be limited by the number of iterations then you should enter a large number here such as 50,000.

Maximum wall time

Enter the time limit in hours for the optimisation to run. If the optimisations are still running after the duration entered here they will be considered to be completed. This limit works in conjunction with the [Generations for convergence](#) and [Max evaluations](#) settings - whichever limit is reached first will cause the optimisation to stop. If you don't want optimisations to be limited by time then you should enter a large number here, such as 240 (10 days).

Pause on errors

It is important to understand the cause of any errors that occur during an optimisation. The **Pause on errors** option, when checked, causes a message to be displayed and the optimisation is paused. For example if there is an overlap in heating and cooling setpoint temperatures, in configurations where the heating setpoint is higher than the cooling setpoint the following error message is displayed:



This sort of error does not cause a problem for the overall optimisation study. If a particular design variant has an invalid configuration then it is assigned a penalty using the maximum objective score (assuming objectives are being minimised). This applies evolutionary pressure to discourage similar invalid design variants from appearing in future generations.

Once you are clear about the cause of any errors you will normally want to stop seeing reports and allow the optimisation to continue. To do this simply uncheck the **Pause on errors** check box. Simulations with errors are displayed in red in the grid as shown below.

Iteration	Generati...	Status	Window to wall %	Cooling set-point te...	Heating set-point te...	Local shading type	CO2 (kg)	Discomfort ASHRA...
72	5		43,000	25,300	23,400	1.0m Overhang	22915,129	665,5000
71	5		23,000	25,600	16,700	0.5m Overhang	19739,116	1800,5000
70	5		42,000	25,900	17,100	2.0m Overhang	19846,498	1927,0000
69	5		60,000	24,800	21,400	2.0m Overhang	22878,883	1312,0000
68	5		41,000	26,400	23,300	2.0m Overhang	21801,074	900,5000
67	5	Calculation error	38,000	22,400	22,600	1.5m Overhang	1000000000,000	8761,0000
66	5	Optimal Designs	41,000	27,400	22,600	2.0m Overhang	20992,442	1214,0000
65	5		49,000	24,400	20,900	2.0m Overhang	22200,984	1297,5000
64	5		62,000	27,400	20,000	1.0m Overhang	21769,323	2240,5000

Common causes of errors are:

- Invalid combinations of variables such as heating setpoint > cooling setpoint. These errors are trapped by DesignBuilder before a simulation is run.
- Failure of a simulation due to errors in input that were not trapped by DesignBuilder.
- A severe EnergyPlus error such as simulation instability.
- A failure in the simulation infrastructure, e.g. a network failure when using a network simulation server.

Quietly retry if server is unavailable

Normally, if DesignBuilder is unable to access the requested Simulation Server, it will ask you whether you would like to **Abort**, **Retry** or **Ignore** the missing server. Selecting **Abort** cancels the whole analysis. Selecting **Retry** or **Ignore** instructs the software to keep trying. The latter option would be selected if the server became temporarily unavailable, perhaps due to a network error or because the server was down for a while.

However, if you are running a long analysis and you don't want to be prompted in this way due to a temporary glitch in the availability of your simulation server then you can check this option. This is similar to pre-selecting the retry or ignore options in advance.

Retain simulation manager results

Normally results for each simulation are held within the Simulation Manager system only for as long as required and are deleted once the key results have been loaded into DesignBuilder. However, if you require results for each simulation to be retained within the Simulation Manager system then check this checkbox.

Backup results table each generation

If you are concerned about the stability of the optimisation process (e.g. due to possible memory issues) you can select this option to ensure that summary results are stored in an external file at the end of each generation. This allows you to later load partially completed results into a spreadsheet for post-processing outside DesignBuilder.

Optimisation backup results are stored in the [DesignBuilder Data\Backup\Opt Results](#) folder and have the filename format: **OPTResults_dd_mm_yyyy_hh_mm_ss.txt** where the date and time is defined at the time of the first generation processed.

The file uses the "#" character as a field delimiter which is non-standard for external tools. To load the file into Excel, you should follow these steps:

1. Open Excel and select the Open file command and the Browse option.
2. Select the relevant .txt file in the [DesignBuilder Data\Backup\Opt Results](#) folder.
3. On the Excel "Text Import Wizard", use the following settings:

Text Import Wizard - Step 1 of 3

The Text Wizard has determined that your data is Fixed Width.

If this is correct, choose Next, or choose the data type that best describes your data.

Original data type

Choose the file type that best describes your data:

☒ Delimited - Characters such as commas or tabs separate each field.

☐ Fixed width - Fields are aligned in columns with spaces between each field.

Start import at row: 1 File origin: 437 : OEM United States

☐ My data has headers.

Preview of file C:\Users\AndyT1\OneDrive - Designbuild... \OPTResults__12_11_2025_15_05_26.txt.

1									
2	#UNKNOWN	#UNKNOWN	#UNKNOWN	#UNKNOWN	#UNKNOWN	#UNKNOWN	#UNKNOWN	#UNKNOWN	#
3	#0	#0	#32	#25	#23	#0	#0	#0	#
4	#1	#0	#54	#24	#21	#0	#0	#0	#
5	#2	#0	#66	#25	#21	#0	#0	#0	#

Cancel < Back Next > Finish

Press **Next**

Text Import Wizard - Step 2 of 3

This screen lets you set the delimiters your data contains. You can see how your text is affected in the preview below.

Delimiters

☐ Tab

☐ Semicolon

☐ Comma

☐ Space

☒ Other: #

☐ Treat consecutive delimiters as one

Text qualifier: *

Data preview

	UNKNOWN	UNKNOWN	UNKNOWN	UNKNOWN	UNKNOWN	UNKNOWN	UNKNOWN	UNKNOWN	
0	0	32	25	23	0	0	0		
1	0	54	24	21	0	0	0		
2	0	66	25	21	0	0	0		

Cancel < Back Next > Finish

Press **Finish**

The data should now be ready for processing within Excel.

Note: This process doesn't recover the names of each column, however the natural column order is retained, so it should generally be possible to identify the column headings by cross-referencing with the results grid in DesignBuilder.

Discussion On Selecting Settings

An important and often overlooked consideration is the impact of the time taken by DesignBuilder to generate the IDF input file for each design. The longer this is relative to simulation time the greater the model generation bottleneck and the less useful the large number of parallel simulations. On the other hand, the larger and more complex the model and the more detailed the simulations (nat vent, detailed HVAC, more time steps etc) the more time is taken in simulation and relatively less in IDF generation. For an extremely large/complex model, all IDF inputs will have been generated before the first results are in and the large number of parallel simulations will speed progress. For a smaller model, the IDF generation bottleneck is often more significant and first simulation results will be in before the 3rd or 4th IDF input has been generated and in this case the multiple cores will not be used. This is often the case for simple single zone models. To test and understand how this works you may find it useful to watch the standalone [Simulation Manager](#) dialog while the optimisation takes place and . You will see new jobs being submitted, queued and simulated.

Multi-Objective Pareto Scatter Graph

When 2 objective functions are defined, the trade off for the range of design variables considered is displayed graphically on the scatter graph. The Pareto optimal solutions are displayed in red, current generation are shown in blue and previous generations are shown in dark grey.

Single Objective Graph

When a single objective is used the graphical output shows the evolution of the objective function for each iteration. Because there is no trade off with single objective optimisation, only the single most optimal solution is highlighted in green in the grid.

Optimisation Calculation Options - Simulation Manager Tab

Accessed from [Optimisation Calculation Options Dialog](#)

Optimisation studies require many hundreds or even thousands of simulations to be carried out in order to achieve accurate results. DesignBuilder is provided with a Simulation Manager to help run these simulations as quickly and efficiently as possible. To use the Simulation Manager tick the **Use Simulation Manager** checkbox on the Simulation Manager tab and select the appropriate options below. If you don't choose this option then simulations will be carried out sequentially, which will generally take a lot longer.

Tip: If you are planning to run an optimisation unsupervised for a long period, e.g. overnight, you should make sure that the machine power settings are such that the machine won't go into sleep or hibernation mode during the process. If the machine does go into a sleep/hibernate state during an optimisation it can be continued simply by waking the machine, but of course progress won't have been made while the machine was inactive.

Server

Select the server from the drop list. DesignBuilder comes with a server called "localhost" and if you haven't installed a [simulation server](#) on your network then this will be the only option available.

Contact [DesignBuilder support](#) for information on how to connect to a server running on a Local Area Network.

EnergyPlus server method

Two EnergyPlus server method options are available when the **localhost** server is selected:

- **EnergyPlus** - the normal option to choose which runs the standard version of EnergyPlus per your select on the EnergyPlus tab of the Program options dialog.
- **JEPlus** - if you have an account with [JESS Online](#) then select this option which will allow you to run your simulation using [JEPlus](#). The first time you attempt to run JEPlus through this option you will be prompted for your account details. This option is not available for simulation servers (i.e. when **localhost** is not selected as the server).

Override default number of jobs per batch

If you would like to override the default number of jobs per batch (e.g. if you would like to see results for each generation to be displayed as they are calculated) then you can override the setting by selecting this checkbox and entering a smaller value in the **Number of jobs per batch** text box below.

Number of jobs per batch

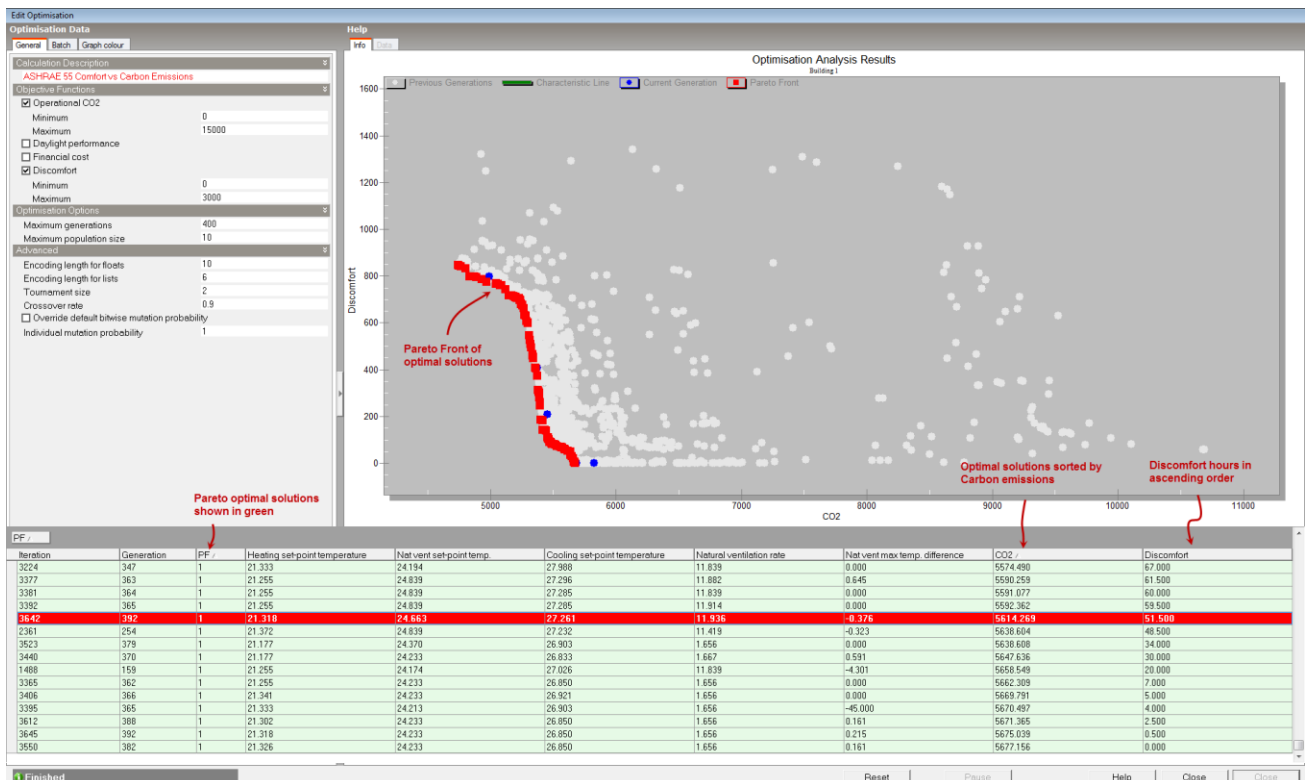
This option allows you to control the number of simulations to be run before results are displayed and can be used to see results more frequently when testing a large model. Enter a low value (e.g. 2) to see results displayed as they are generated. This option is only available for multi-objective optimisation.

Tip: To make the most of parallel computing facilities you should either uncheck the **Override default number of jobs per batch** option above and allow DesignBuilder to apply a high number of jobs per batch internally or enter a high value here and not lower than the [Initial population size](#) (or [Maximum population size](#) if you are using a Pareto archive).

See also "Simulation Manager" on page 1609

Optimisation Results

The screenshot below illustrates some of the main outputs of the Optimisation analysis.



Optimisation Limitations

Current versions of DesignBuilder have these limitations related to Optimisation:

1. You must run a simulation with monthly/annual output for the model (on the Analysis tab) before clicking on the optimisation tab. If you don't have any monthly/annual results on this tab then you must re-run the base simulation before updating the optimisation analysis.
2. It isn't possible to store multiple optimisation results sets within the DesignBuilder model.

See also [Optimisation Tips and Tricks](#)

Optimisation Tips And Troubleshooting

Tips

Speeding up Optimisation

Optimisation studies require hundreds or even thousands of simulations to be carried out and so can be extremely time/calculation intensive. To help speed simulations, you could consider using one or more of these techniques:

1. Analyse a [single representative zone](#) in the building and apply findings to whole building.
2. Use a representative simulation period instead of the whole annual simulation. For example, in many climates using the first or second 6 months of the year would be a good approximation to use of the full 12 month simulation. Alternatively if the analysis is mainly related to summer overheating then you might choose a shorter representative summer period for the simulations.
3. In most cases, the more variables that are defined in the analysis, the more data points that will be required before convergence of the solution is achieved. Instead of trying to include all possible optimisation variables in one analysis, you might consider splitting the problem into 2 or more sub problems. For example it can be easiest to carry out HVAC controls optimisation studies independently of component selection. Start with the most fundamental design decisions (e.g. window size and

position) and as the design becomes more established and study details such optimal selection of system types and control options later in separate analyses.

4. Use [JEPlus](#) or a network [simulation server](#) to maximise the number of simulations that are run in parallel at one time. If you have a simulation server with many cores at your disposal then you might consider increasing both the population size and [Number of job per batch](#) which allows more cases to be analysed at once.
5. Minimise the number of output variables requested on the Analysis tab before running an optimisation, UA/SA or parametric analysis. In particular deselect all outputs not required for the simulations, remove requests for daily, hourly and sub-hourly results, as well as any requests for summary annual and monthly results and comfort outputs not required.

See also [Working with Large Models](#).

Ensure Inheritance Path

It is important to ensure that the variations defined by the optimisation [Design Variables](#) will have the intended effect. Optimisation works by applying variations to a copy of the base model and these changes are applied to the object defined in the Variable. It is important to take into account how this data will be inherited to down to any child blocks, zones etc to ensure you are testing the design variants you think you are. For example if the Window to wall % was hard set at block level then changes in this data at building level would not influence the surfaces of the design variants in that block and the results will indicate that the Window to wall % does not influence results.

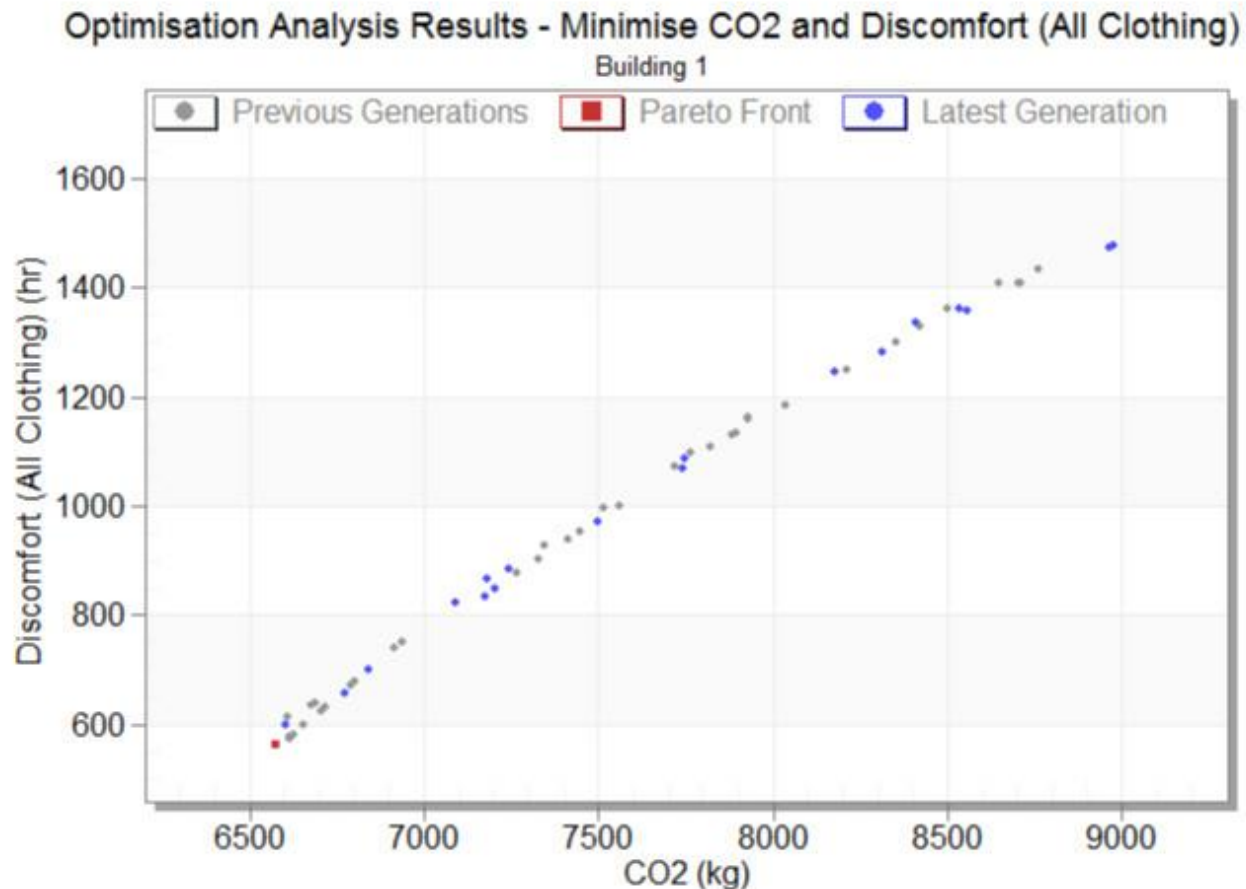
Switching on Shading when Shading System Type is a Variable

Optimisation variables such as Local shading type and Window shading will only take effect if the corresponding checkbox activating these options is checked. A common mistake for new users is to request these variables without switching on the option.

Troubleshooting

No Pareto Front

Sometimes when running a multi-object optimisation study, when looking at the scatter plot of results, instead of a nice clean Pareto front, you may see a line formed where the Y-values increase as X-values increase, something like the plot below.



This case indicates that the 2 design objectives do not conflict with each other and an optimisation study is not needed because there is a clear optimum case and no trade off is involved. For example in the above case the 2 objectives were Operational CO2 emissions and Discomfort hours. The 2 design variables were Window to Wall % and Building orientation. Here the Window to Wall % dominates over orientation and both objectives can be met simply by minimising the Window to Wall %.

Optimisation Technical Background

How It Works

DesignBuilder uses a Genetic Algorithm (GA) (aka Evolutionary Algorithm or EA) based on the NSGA2 method (aka NSGA-II) Deb et al. 2002. This is widely used as a "fast and elitist multi-objective" method providing a good trade off between a well converged and a well distributed solution set. NSGA2 uses the non-dominated sorting method, which is proven to be highly effective in ranking competing objectives. One deficiency of the original NSGA2 is that it does not provide a way for handling constraints efficiently. This is addressed in JEA using the stochastic ranking method. It works as follows:

1. First, the population is randomly initialised.
2. Chromosomes (design variants) are sorted and put into fronts based on Pareto non-dominated sets. Within a Pareto front, the chromosomes are ranked based on Euclidean distances between solutions or l-dist (the term used in NSGA2). Generally, solutions which are far away (not crowded) from other solutions are given a higher preference in the selection process to help create a diverse solution set and avoid crowding.
3. The best designs are picked from the current population and put into a mating pool.
4. In the mating pool, tournament selection, crossover and mating is carried out.
5. The mating pool and current population is combined. The resulting set is sorted, and the best chromosomes are passed into the new population.
6. Go to step 2, unless convergence has been reached.
7. The solution set is the highest ranked Pareto non-dominated set from all populations.

Constraints

Constraint implementation in JEA

Constraint handling is a topic that attracts lots of attention from algorithm designers. The efficiency of constraint handling is measured by not only how quickly feasible solutions are found, but also the quality of those feasible solutions. If a strategy pushes too hard for meeting the constraints, it may hamper exploration for better objective values. On the other hand, if it is too lenient, too much time may be wasted on infeasible solutions. The balance depends on the nature of the problem, so a perfect strategy may not exist. JEA uses one of the best strategies in terms of robustness of adaptability to different problems called Stochastic Ranking (Runarsson and Yao, 2000).

Stochastic Ranking is a probabilistic strategy to rank solutions with the different objective and constraint values. Its original form is designed for one objective against one constraint. In order to make it work with NSGA-II, the following steps are taken:

1. All constraints are scaled (normalized) and then aggregated so that the infeasibility (constraint violations) is measured as a value in $[0, 1]$.
2. Infeasibility is used as an additional objective and sorted with Non-dominated Sorting with all other objectives. This produces an initial ranking order of all solutions.
3. Stochastic Ranking is then applied to the initial rank of each solution (treated as its objective value) and its aggregated infeasibility value. This produces the final ranking of the solutions.

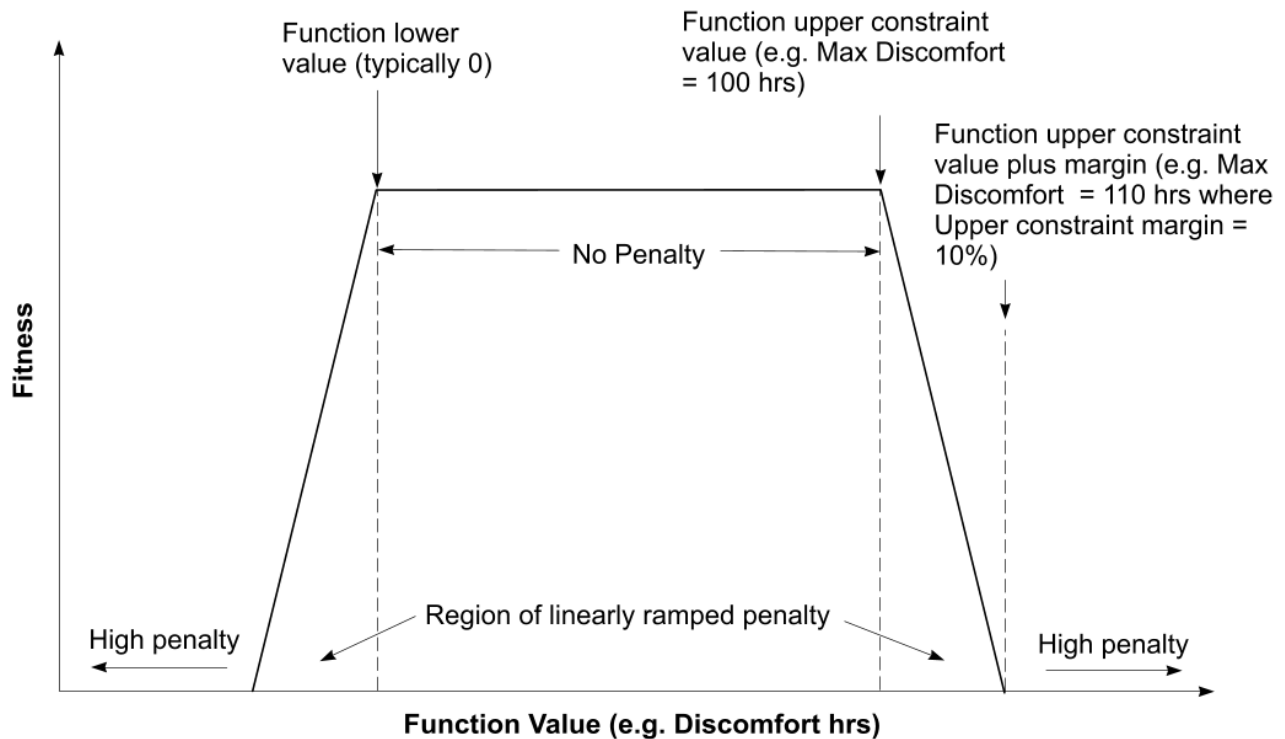
The benefit of this slightly complex arrangement is that it works well with problems with any number of objectives (including single objective) and constraints, and unconstrained problems. The user can use a single Objective bias parameter to control the level of the push for feasibility.

Two configuration parameters directly affect the behaviour of NSGA2 with Stochastic Ranking. **Objective bias** controls the ranking bias between Pareto ranking and infeasibility scores. A value of zero means solutions with lower infeasibility score is better, regardless of the Pareto ranking result. **Elitism tolerance** controls if infeasible solutions can be selected as elites (see below) or not. If a positive value is assigned, e.g. 0.1, it means solutions with infeasibility scores up to 0.1 may be selected as elites.

Constraint implementation in Open Beagle

The lower and upper constraint margins are used as part of the constraint handling method used by DesignBuilder, which is based on a generic penalty function approach. During a constrained optimisation each design alternative is tested against any constraints applied and a penalty is generated based on whether the design is feasible or not, i.e. whether it meets constraint requirements. If the function value is within the feasible range then no penalty is applied and conversely if it is outside the feasible range then a penalty is applied. There is a margin between the feasible and the infeasible regions where a linearly ramped penalty is applied. The upper and lower margins set on the Calculation options dialog are used to define these regions as shown on the graph below.

DesignBuilder Constraint Handling



Pareto Archived Elitism

Evolutionary algorithms are stochastic in nature. The user has little control over which direction or what solutions to explore next. Quite often promising solutions may appear in one generation and then disappear for good in subsequent ones. Elitism is a method for preserving good solutions by selecting cases from a pool of known optimum solutions and inserting them back into the working population.

In Pareto Archived elitism, all known solutions on the global Pareto front are stored. They form the pool from which the elitism operator picks “elites”. In JEA, the maximum number of elites and whether they can include infeasible solutions can be controlled. **Global elitism** controls the pool from which the elite solutions are selected. If set to true (default), selection will be made in the global archive; otherwise it is limited to the current population. **Elitism tolerance** (explained above) also affects how elitism works.

Bibliography

1. Introduction to Genetic Algorithms:

- https://en.wikipedia.org/wiki/Genetic_algorithm
 - <https://www.obitko.com/tutorials/genetic-algorithms/>
2. Sean Luke, 2009, Essentials of Metaheuristics, Lulu, available for free at <https://cs.gmu.edu/~sean/book/metaheuristics/>
 3. A.E. Eiben and J.E. Smith, Introduction to Evolutionary Computing, Springer, Natural Computing Series, Corr. 2nd printing, 2007, ISBN: 978-3-540-40184-1 <https://www.cs.vu.nl/~gusz/ecbook/Eiben-Smith-Intro2EC-Ch2.pdf>
 4. Deb, K., Pratap, A., Agarwal, S., and Meyarivan, T. (2002). A fast and elitist multi-objective genetic algorithm: NSGA-II. IEEE Transaction on Evolutionary Computation, 6(2), 181-197.

Optimisation Tutorials

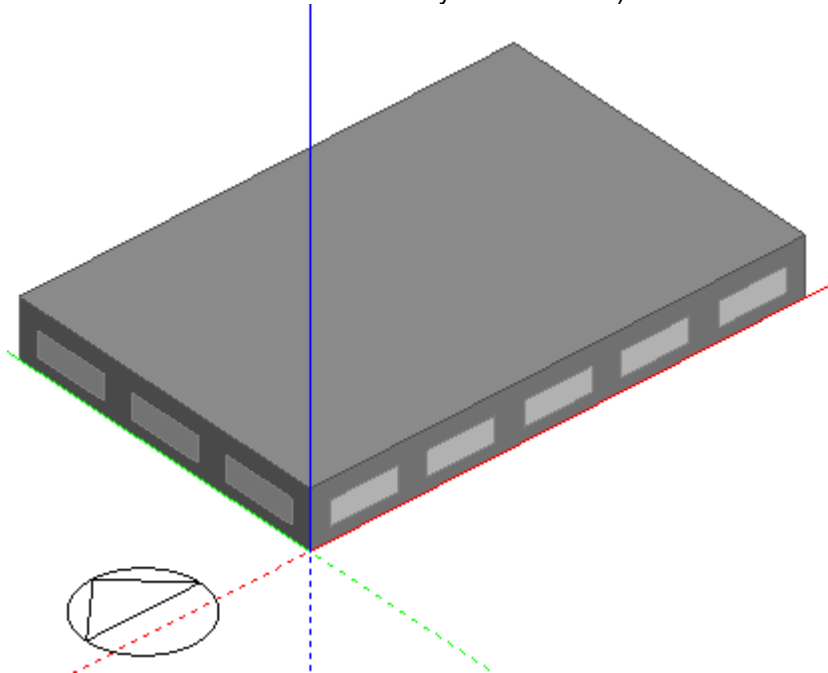
- "Optimisation Basic Tutorial" on page 1736
- "Optimisation Tutorial - Custom EMS Variable" on page 1739
- "Optimisation Tutorial - Custom Script Variable" on page 1745
- "Optimisation Tutorial - Custom Script KPI Example 1 - db_eplusout_reader Python Libraries" on page 1752
- "Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760
- "Optimisation Tutorial - Custom Script KPI Example 3 - Using EMS + Python Scripts to Optimise Controls for Comfort and Heating" on page 1770
- "Optimisation Tutorial - Custom Script KPI Example 4 - Adding Python Script for TM52 Comfort to Example 3" on page 1778

Optimisation Basic Tutorial

This tutorial will help you get up to speed with running an optimisation study in DesignBuilder in a few easy steps.

Step 1 - Create a base model

Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m as shown below. Use default Model options and template settings. DesignBuilder as supplied will use a Simple HVAC Fan Coil Unit system as the default. For this example you should make sure that you have heating and cooling selected and no natural ventilation (this will be the case for a new model with a FCU HVAC system selected).

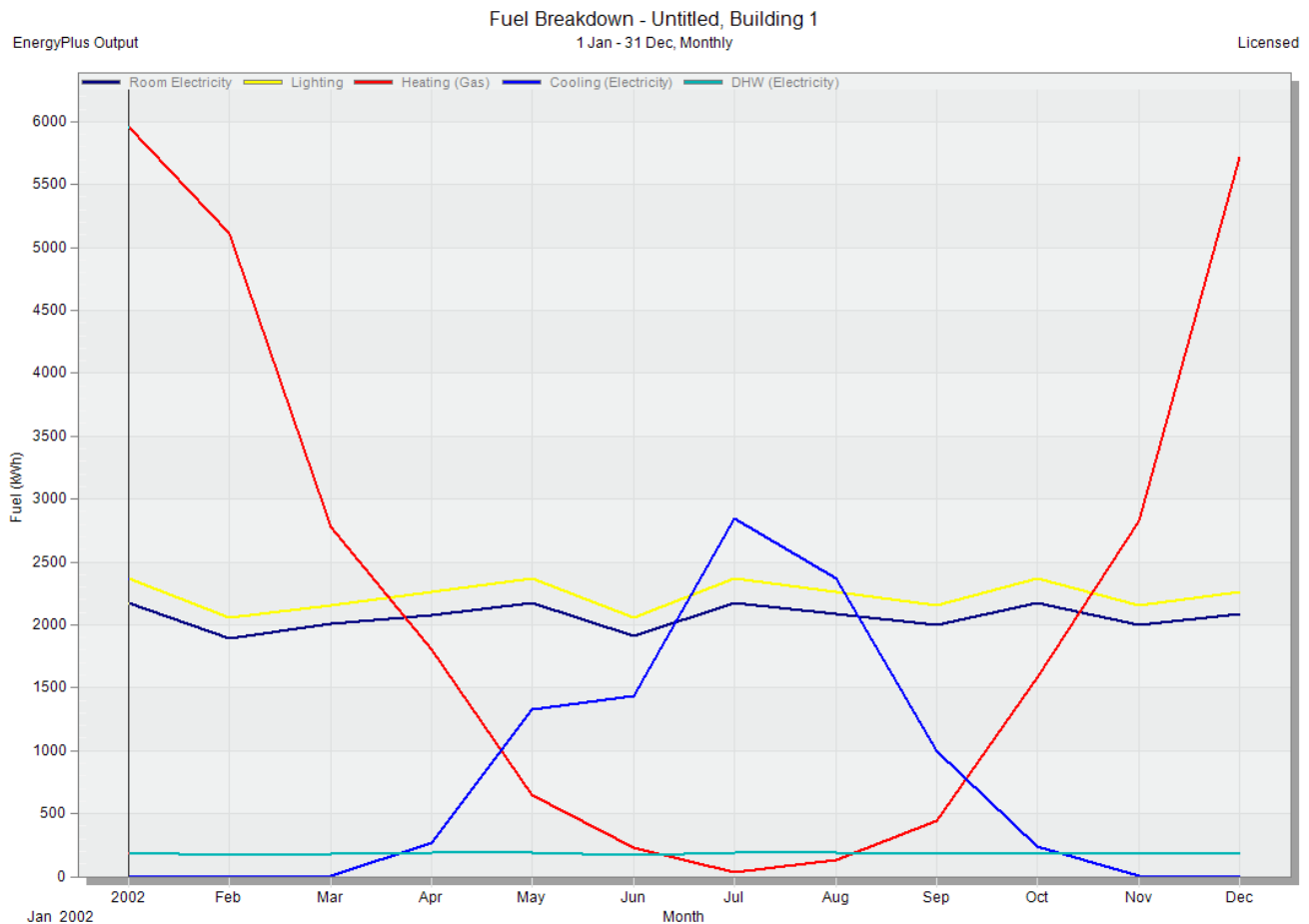


Step 2 - Run a standard simulation

Click on the Simulation tab and run a base annual simulation.

Optional: Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation.

Check the hourly results for the simulation period and make sure that the model is behaving as expected, including temperatures within the building, operations periods etc. If not fix the model and repeat this step until you are happy with the base model hourly results. Monthly results should look something like the screenshot below.



Now run a final simulation without hourly and daily results but with **Monthly and Run period** results to prepare the model for the optimisation.

Step 3 - Define optimisation problem on Optimisation Analysis Settings dialog

Once you have a good understanding of how the base model operates, you are ready to start the optimisation stage. To do this click on the **Optimisation + UA/SA** tab on the Simulation screen. Because you don't have any results yet the [Optimisation Analysis Settings](#) dialog is displayed. This allows you to define the optimisation problem, i.e. what it is that you want to achieve from the optimisation study. Use the default settings for this simple example which request an optimum set of solutions to minimise both operational construction emissions and discomfort allowing for variations in window to wall ratio and heating and cooling setpoint temperatures.

Objectives:

1. Minimise "Operational CO2 emissions"
2. Minimise "Discomfort ASHRAE 55 (all clo)"

Constraints:

None

Design variables:

Allow WWR and cooling and heating setpoints to vary as shown in the screenshot below. Variations are made at building level for all variables in this case as shown by the Target objects column.

Name	Variable type	Min Value	Max Value	Step	Options List	Target objects
Window to Wall %	Window to wall %	20.00	80.00	20.00	-	Building
Cooling setpoint temperature	Cooling set-point temperature	23.10	27.50	2.00	-	Building
Heating setpoint temperature	Heating set-point temperature	18.00	23.00	2.00	-	Building

Only minor changes may be needed as most settings described so far are defaults for a new model.

Step 4 - Open Optimisation Calculation Options dialog

Having confirmed the Optimisation analysis options the next dialog to open will be the Optimisation calculation options. Change the Initial population size to 10 for this small analysis and otherwise use default settings.

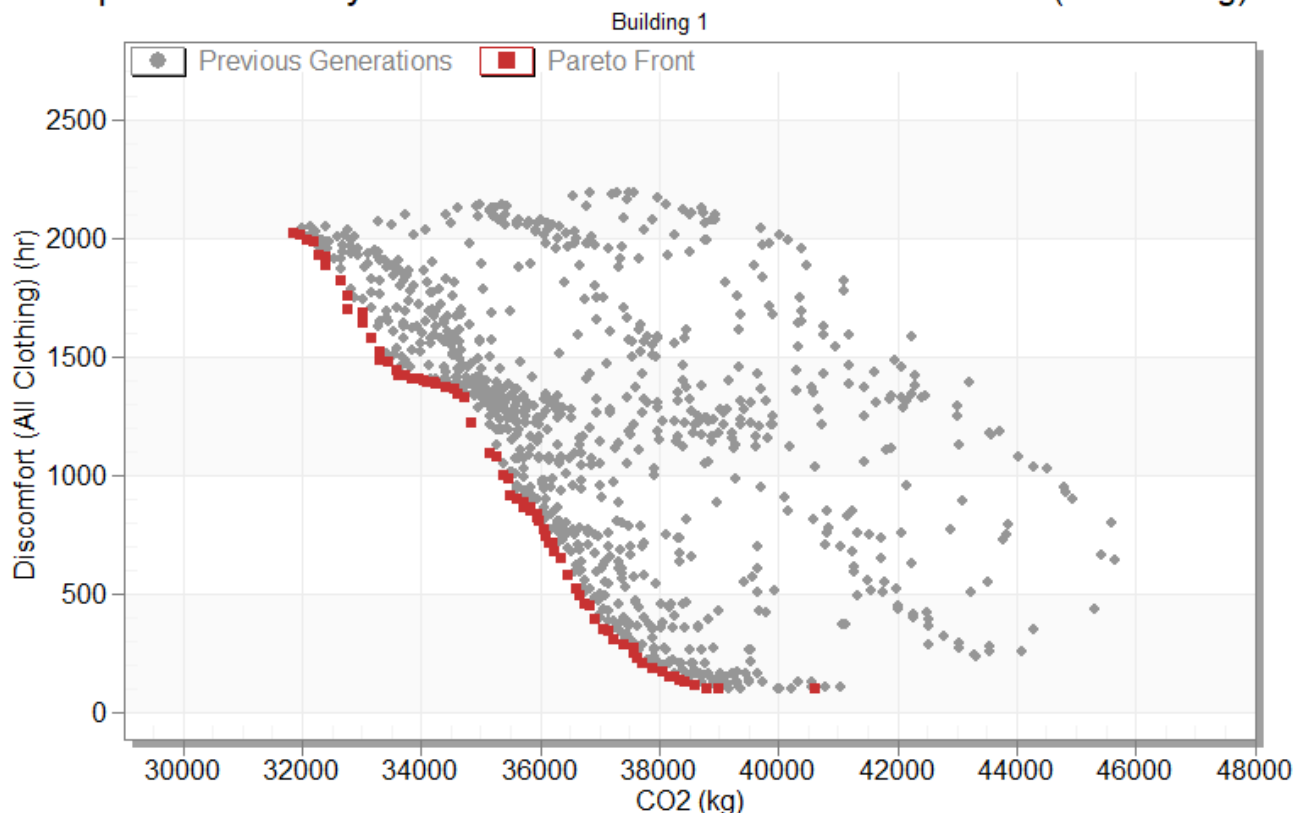
Step 5 - Start optimisation analysis

Once you have completed your review of the options press the Start button in the bottom right of the screen. The optimisation process will involve running a lot of simulations. With the recommended settings there will be 100 generations with a population size of at least 10 in each, growing as Pareto solutions are added. That means that at least $100 \times 10 = 1000$ simulations will be run! For our simple model and when using the Simulation manager to run these in parallel this shouldn't take too long. However if time is an issue it is usually worth keeping an eye on the solutions as they come in to check whether convergence has been achieved.

While preparing this tutorial we made the judgement that the solution was effectively converged after 77 generations and pressed the Stop button at that point. If the optimisation had been left to continue the red Pareto front line would probably have been filled in further however this would not provide much extra information for the purposes of our example test. After pressing Stop the current generation continues to simulate and once that is finished we see a graph like that below.

Step 6 - Analyse results

Optimisation Analysis Results - Minimise CO2 and Discomfort (All Clothing)



Try using some of DesignBuilder's analysis tools to investigate the best settings in this case.

1. Drag the Status header into the dark grey area above the grid where it says "Drag a column header here to group by that column"
2. Sort the grid data by Discomfort by clicking on the "Discomfort" header. Note steps 1 and 2 are automatically carried out for the graph on the main screen.
3. Scroll down to the bottom of the grid where you will find the optimal "Pareto Front" options coloured green. These are the design configurations which most successfully achieve the design objectives (minimise carbon and discomfort in this case). Because you sorted by discomfort you will see results either in ascending or descending order depending on how many times you clicked on the header. You will notice that when the discomfort results are in ascending order the carbon emission results in the next column are in descending order and vice-versa.
4. All points on the Pareto front have 20% (or a little more) glazing. This is because it was the lowest allowed value in the analysis and higher values increase both discomfort and cooling loads (hence carbon emissions).
5. Heating setpoint temperatures range from 18°C (lowest carbon emissions) to 23°C (lowest discomfort).
6. Cooling setpoint temperatures range from 24°C (lowest discomfort) to 27.5°C (lowest carbon emissions).
7. The easiest way to identify a balanced result is to click on a point on the Pareto Front which gives to right degree of emphasis to the 2 objectives. The corresponding item is highlighted in the grid giving the corresponding design settings.

Next steps to learning. Try adding more design variables. Glazing type, shading systems etc. Try using some other variable types.

Tip: See: [Insights Tutorial - Basic Optimisation](#) to learn about how to export results from this tutorial into Insights for more detailed analysis and high-quality graphical outputs.

Optimisation Tutorial - Custom EMS Variable

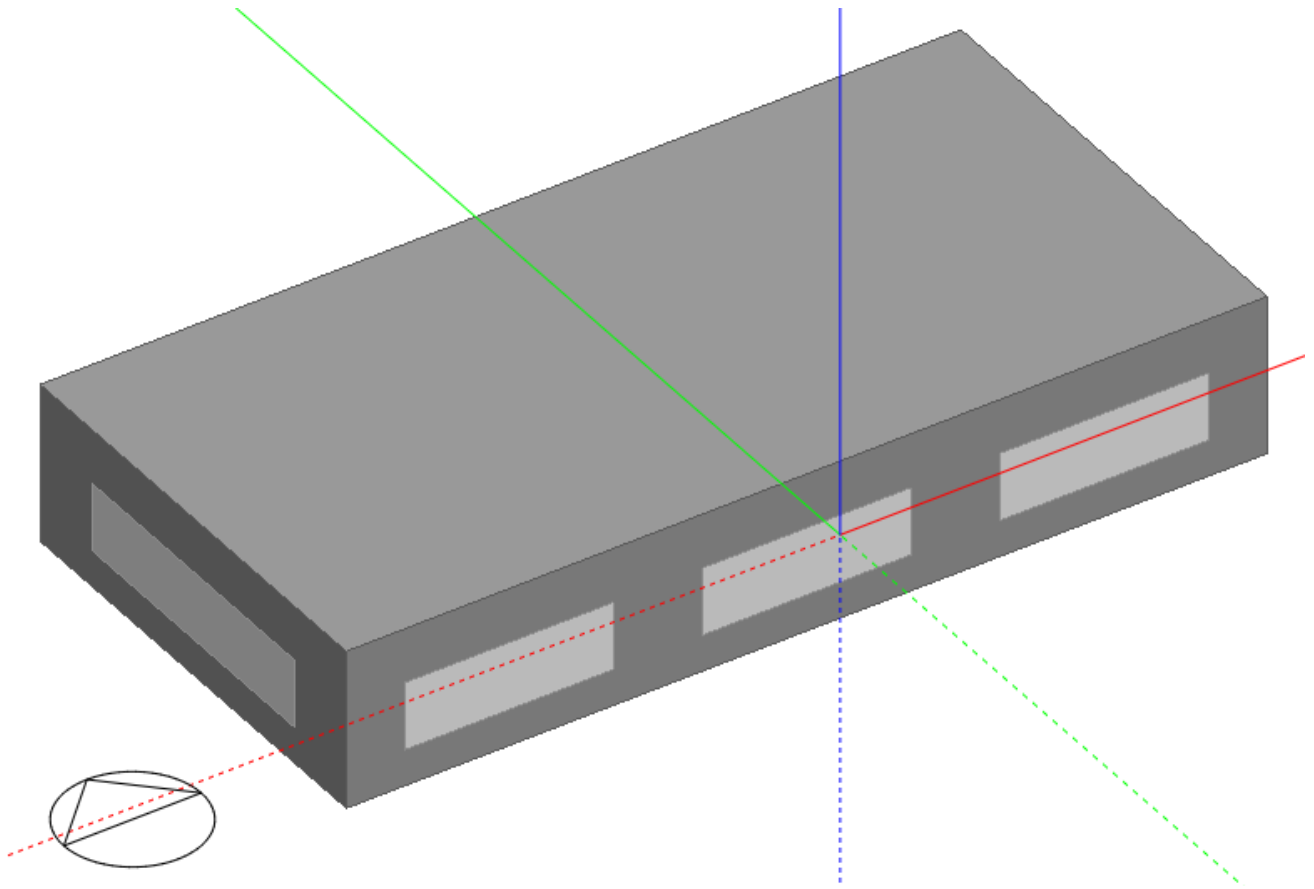
This tutorial shows you how to set up an optimisation analysis to help select a single key control parameter within an EMS script-based optimum start controller. The control parameter setting in question affects both heating energy consumption and occupancy comfort and the design curve resulting from the analysis is used to help make an informed decision on the most appropriate start up time that trades off occupant comfort on the coldest mornings of the year to help save heating energy over the whole year. A similar process can be used to help analyse the impact of one or more settings within *any* EMS script.

A custom EMS variable can be used in parametric processes (optimisation, parametric analysis or UA/SA) when an EMS script is used as part of the simulation model and a value in the script is to take a different value for each simulation as controlled by the parametric processor. In the example used in this tutorial, an EMS script is used to define a simple optimum start heating controller which sets the heating system morning start up time based on the outdoor temperature at 1am. The algorithm uses a simple linear control based on the earliest possible start up time and the corresponding outdoor temperature, along with the latest possible start time and its corresponding outdoor temperature. The earliest possible start up time is defined as a variable in the script with a default value of 4am. This allows the script to be used for standard simulations, in which case the optimum start controller uses an earliest possible start time of 4am. Otherwise, for optimisations, the earliest start time selected for each simulation is set by the optimisation process controller.

The tutorial model uses a default Simple HVAC fan coil unit system, though the same process would work equally well using Detailed HVAC.

Step 1 - Create A Base Model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 20m by 10m with block height of 3.5m as shown below.



3. Load the **Generic Office Area** Activity template at building level on the Activity Tab.

Step 2 - Set Up The Optimum Start Control Script

The script to be used in this example is based on the "Optimum start heating control (non-domestic)" example EMS script provided with the software. Follow the steps below to set it up for the analysis.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Open the **EMS** category and select the "Optimum start heating control (non-domestic)" script. Then copy that script to create a new custom script that can be edited, select it and click on the Edit Info toolbar icon to edit the copied script.
4. Change the name of the script to "Optimum start heating control with variable earliest start time"
5. Edit the script to set up the variable start time by changing the line below:

```
! earliest start time
SET TimeTosMin = 5,
```

to:

```
! earliest start time
SET TimeTosMin = <Var,EarliestStartTime,4>,
```

This Erl statement says that for standard simulations the variable is initialised to 4am, otherwise it is variable using the "EarliestStartTime" Variable key (see Step 5, point 5 below) and initialised by the optimisation controller.

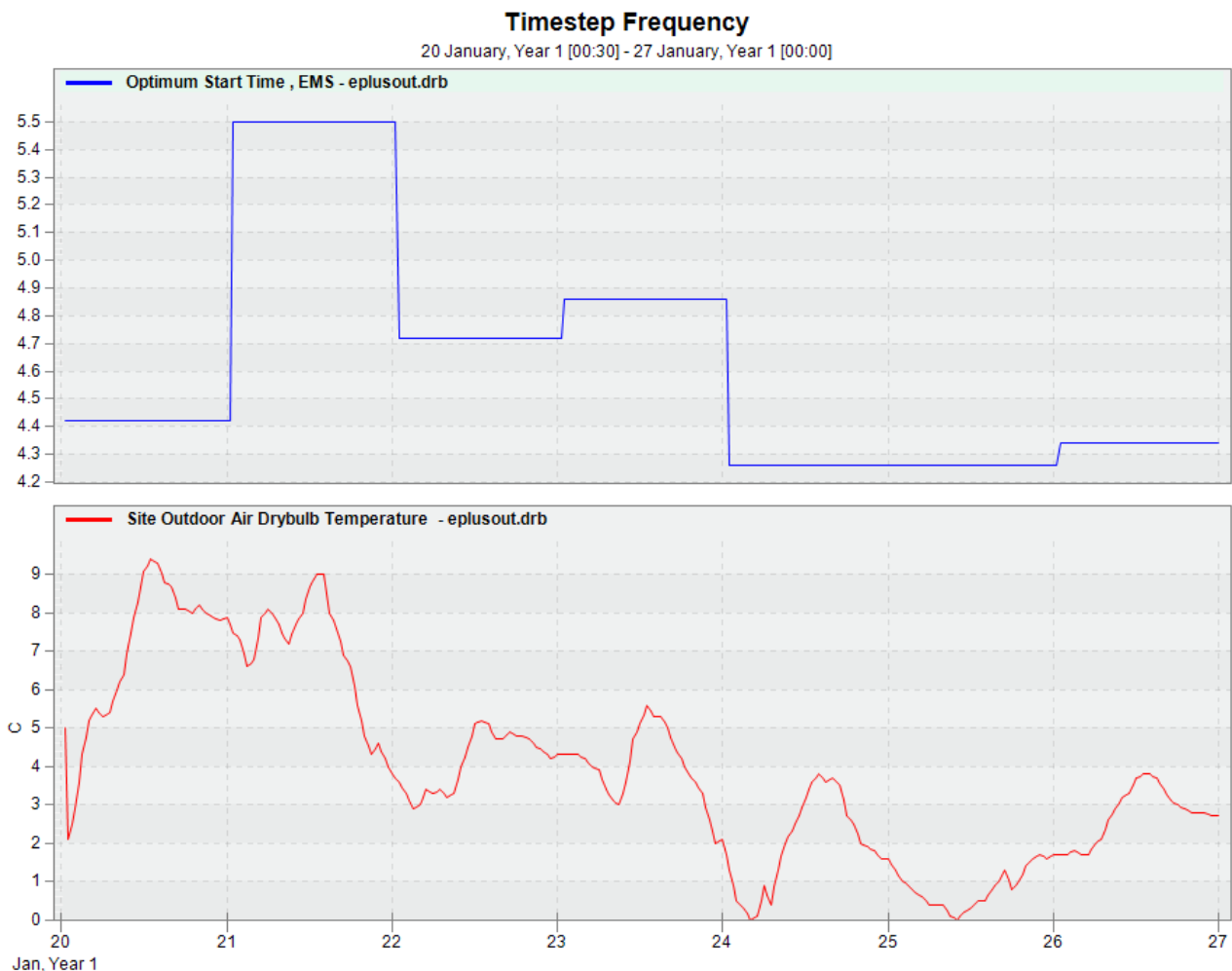
6. Check the **Enable program** checkbox to ensure that the script will be executed at runtime.

7. Press OK to confirm the changes to the script.

Step 3 - Run Standard Simulation

1. Run a standard simulation for the **Winter typical week** with hourly results and open the results in [Results Viewer](#) by clicking on the **View EnergyPlus results**  toolbar icon.

2. View hourly results for **Optimum start time** (a custom output generated by the EMS script) and **Site Outdoor Air Drybulb Temperature** on different graphs. You should see something like the screenshot below.



The **Optimum start time** output shows the time the heating system starts up for each day of the simulation vs the outdoor temperature. You can clearly see that the colder the outdoor air temperatures at 1am, the earlier the heating system starts. This confirms that our EMS controller is working as intended.

Step 4 - Make Optimisation Objective Settings

Now to find the best setting for the controller earliest start time (`TimeToSMin` in the EMS script) make the following settings on the **Parametric Settings** dialog.

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon and set the [Analysis type](#) on the Analysis tab to **2-Optimisation**.

2. Go to the [Objectives](#) tab and add 2 new objectives to minimise **Heating load** and **Discomfort ASHRAE 55 (all clo)**. You can either edit the existing objectives or first delete those and create 2 new objectives.

With both objectives defined, the Objectives tab should now be set up as follows:

Parametric, Optimisation and UA/SA Analysis Settings		
Analysis Type	Objectives	Additional Outputs
Constraints	Design Variables	
Name	Min/Max	Objective KPI
Minimise Heating Load	1-Min	Heating load
Minimise Discomfort (ASHREA 55)	1-Min	Discomfort ASHRAE 55 (all clo)

Step 5 - Make Optimisation Variable Settings

Now set up a single design variable to vary the controller earliest start time.

1. Delete all but one of the existing variables and edit the last remaining variable
2. On the **Edit Analysis Variables** dialog, start by setting the Variable type to **Custom EMS**.
3. Set the Name to "Earliest Heating Start Time".
4. Set the **Min value** to 1, the **Max value** to 7 and the Step (optimisation) to 0.2. This will allow us to explore the impact of start times between 1am and 7am on heating load and discomfort.
5. Set **Variable key** to "EarliestStartTime" to match up with the `<Var, EarliestStartTime, 4>`, setting in the script.

The dialog should now look like this:

Analysis Variables

General

Design Variables

Name	Earliest Heating Start Time
Variable type	Custom EMS

Range

Min Value	1.00
Max Value	7.00

Increment Step

Step (parametric)	1.00
Step (optimisation)	0.200

Target

Target objects	EMS
----------------	-----

Key

Variable key	EarliestStartTime
--------------	-------------------

5. Press OK to confirm.

Step 6 - Re-Run A Standard Simulation

Click on the Simulation tab and press update to re-run a base annual simulation. Before pressing OK, You should make the following settings:

1. Ensure you are running simulations for the whole year by clicking on the **Annual simulation** Info panel link.
2. Ensure you are generating only Annual/Monthly results by checking the **Monthly and Run period** interval and ensuring that all other outputs are switched off.

3. Select operative temperature control by clicking on the Options tab and selecting the **2-Operative temperature** Temperature control option. This setting is advised because the ASHRAE 55 comfort assessment is based on operative temperature and it is important to consider the radiant effects especially when looking at morning time comfort which will be significantly impacted by cool wintertime internal surfaces in the mornings. If you select air temperature control instead you will see very different results.

Step 7 - Run Optimisation

1. Navigate to the **Optimisation + UA/SA** tab and press the **Update** toolbar icon to open the Optimisation Calculation Options dialog.
2. Check the [Run all combinations](#) checkbox since there are not many options to evaluate and optimisation is not required.
3. Set the [Generation population size](#) to a value greater than the number of combinations for the fastest evaluation . 40 will work well here.

The dialog should look similar to the image below.

Calculation Options

General | Simulation Manager

Calculation Description

From

Start date 1 Jan

To

End date 31 Dec

Optimisation Options

Generation population size 40

Generations for convergence 5

☒ Run all combinations

Advanced

Maximum generations 20

Maximum population size 10

Mutation rate 0.40

Crossover rate 1.00

Tournament size 2

Objective bias 0

Maximum evaluations 10000

Maximum wall time 240

☐ Pause on errors

☐ Quietly retry if server becomes unavailable

☐ Retain Simulation Manager results

☐ Backup results table each generation

Press the **Start** button to run the optimisation simulations.

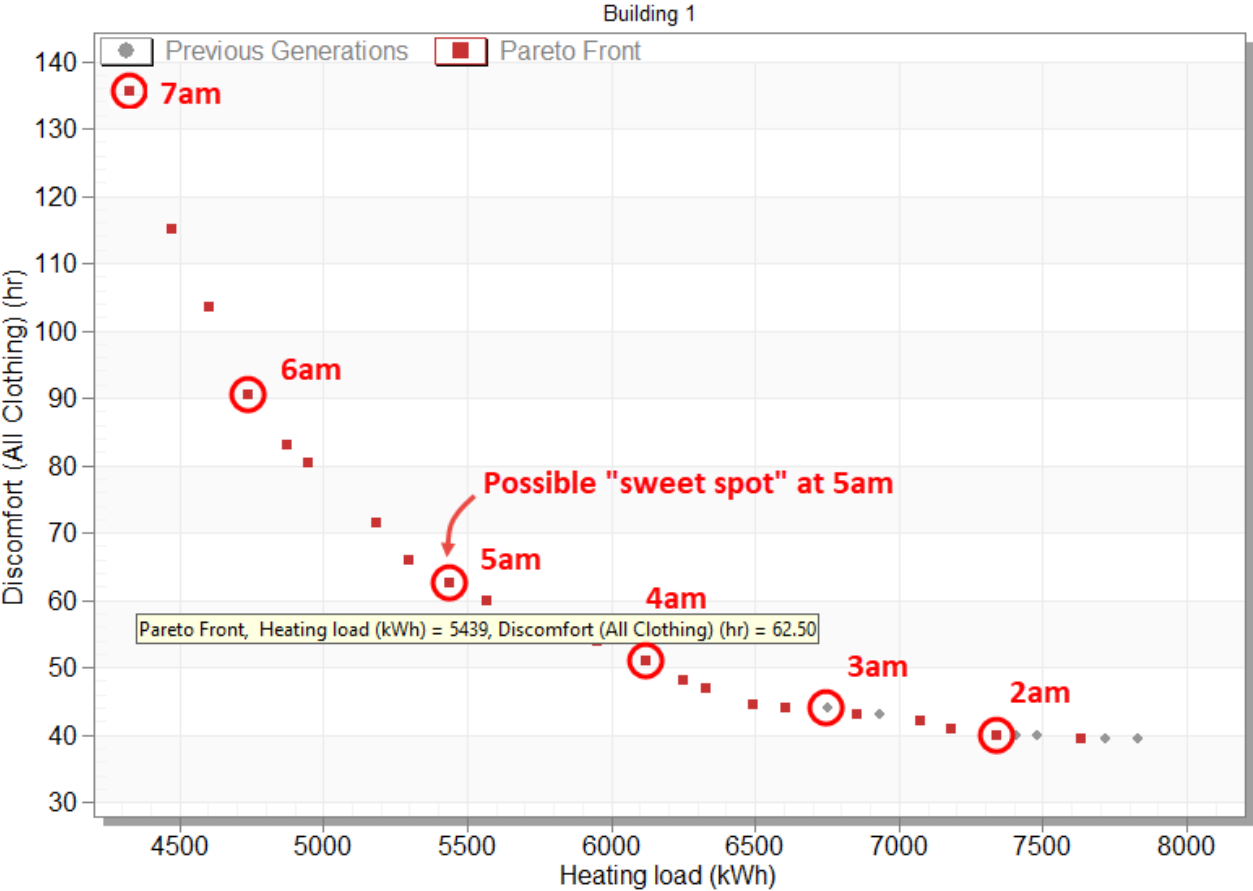
Step 7 - Analyse Results

After the optimisation has completed and you have closed the Calculation Options dialog, the main screen should look similar to the screenshot below (the annotations in red were added manually). The graph shows how variations in the "earliest start time" affects heating load and discomfort. As expected, as the earliest

allowed start time is reduced from 7am to 1am, the levels of discomfort decrease at the expense of higher heating energy.

The results also show diminishing returns in terms of comfort when reducing the earliest start time much below 4am. On the other hand small extra preheat time between 6am and 7am cause big improvements in comfort. The annotations propose a possible "sweet spot" selection at around 5am which gives rise to 5439 kWh annual heating load and 62.5 discomfort hours.

Optimisation Analysis Results - Minimise Heating load and Discomfort (All Clothing)



Status ▾					
Iteration	Generation	Status ▾	Heating load (kWh)	Discomfort (All Clothing) (hr)	Earliest Heating Start Time /
16	0	Optimal Desi...	7077.08	42.0000	2.400
19	0	Optimal Desi...	6856.29	43.0000	2.800
28	0	Optimal Desi...	6605.76	44.0000	3.200
27	0	Optimal Desi...	6494.66	44.5000	3.400
26	0	Optimal Desi...	6330.76	47.0000	3.600
25	0	Optimal Desi...	6252.98	48.0000	3.800
24	0	Optimal Desi...	6122.65	51.0000	4.000
23	0	Optimal Desi...	5950.91	54.0000	4.200
21	0	Optimal Desi...	5842.10	55.0000	4.400
20	0	Optimal Desi...	5682.28	57.0000	4.600
18	0	Optimal Desi...	5570.61	60.0000	4.800
9	0	Optimal Desi...	5438.75	62.5000	5.000

Tip: Although the analysis described in this example uses the Optimisation analysis type, because there is only one variable, the results actually look more like those from a parametric analysis, but with 2 outputs one on the x-axis, the other on the y-axis. Another way to do a similar analysis would be to use the [Parametric analysis](#) analysis type instead of Optimisation, adding Heating load and Discomfort hours as [Outputs](#) instead of Objectives.

Optimisation Tutorial - Custom Script Variable

This tutorial shows how to set up an optimisation analysis using [Custom Script variables](#) to modify the model. You will learn how to use scripts to set up cooling and heating setpoint temperature variables and to identify optimal settings that simultaneously minimise occupant discomfort and annual carbon emissions.

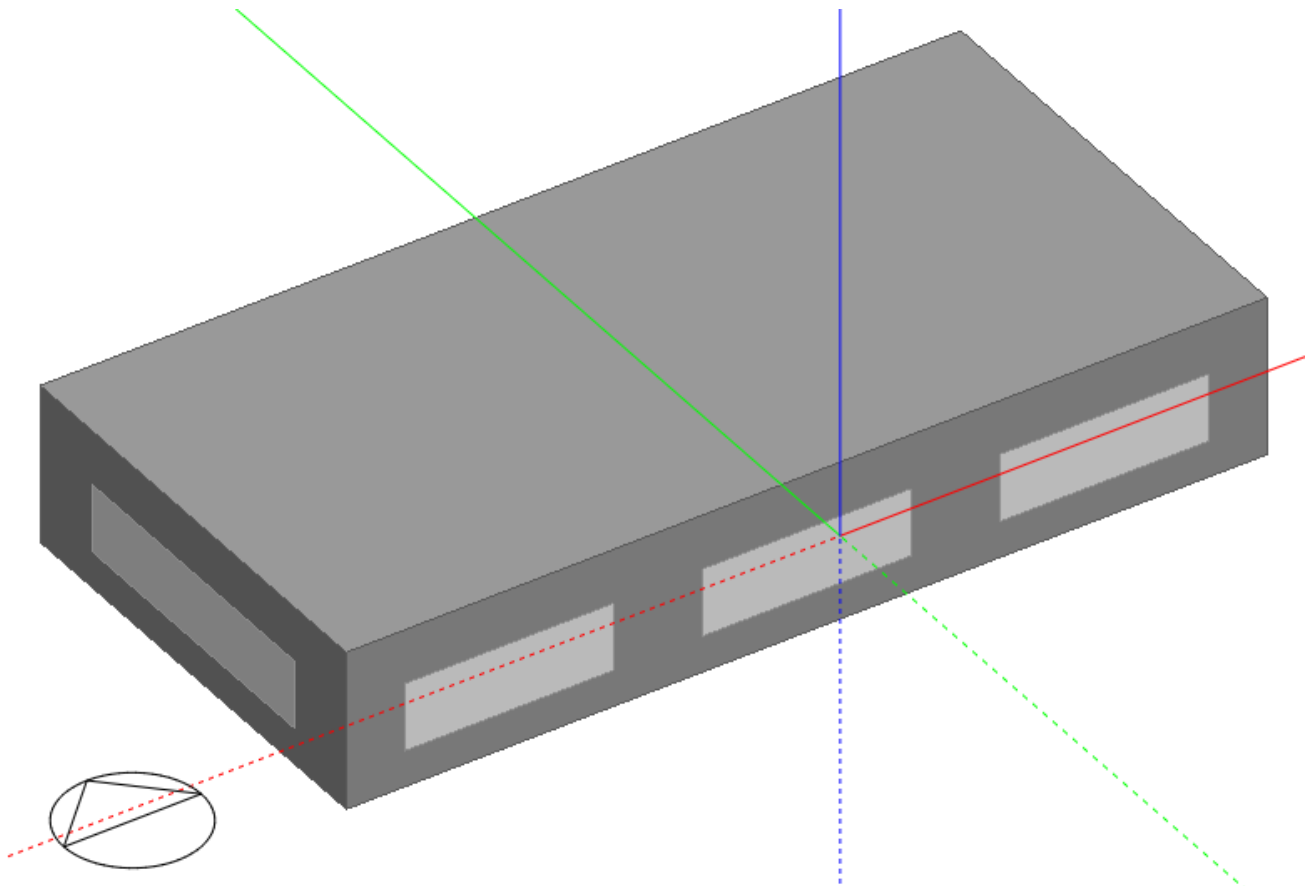
The tutorial model uses a default Simple HVAC fan coil unit system, though the same process would work equally well using Detailed HVAC.

The main learning points are:

1. [Design objectives](#) for minimising occupant discomfort and annual carbon emissions are set up in the usual way.
2. [Design variables](#) for heating and cooling setpoint temperatures are set up using the **Custom script** variable type category including the min, max and interval values to be used.
3. A corresponding script for the **OnDesignVariableChanged** hook point is written to respond to changes in the model. The script applies the setpoint requested by the optimiser to each zone in the model.

Step 1 - Create A Base Model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 20m by 10m with block height of 3.5m as shown below.



3. Load the **Generic Office Area** Activity template at building level on the Activity Tab.

Step 2 - Make Optimisation Objective Settings

First set up the design objectives for the study by making the following settings on the **Parametric Settings** dialog.

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon and set the **Analysis type** on the Analysis tab to **2-Optimisation**.
2. Go to the **Objectives** tab and add 2 new objectives to minimise **Operational CO2 Emissions** and **Discomfort ASHRAE 55 (all clo)**. You can either edit the existing objectives or first delete those and create 2 new objectives.

With both objectives defined, the Objectives tab should now be set up as follows:

Parametric, Optimisation and UA/SA Analysis Settings		
Analysis Type	Objectives	Additional Outputs
Constraints	Design Variables	
Name	Min/Max	Objective KPI
Minimise Operational Carbon	1-Min	Operational CO2 emissions
Minimise Discomfort (ASHRAE 55)	1-Min	Discomfort ASHRAE 55 (all clo)

Step 3 - Make Optimisation Variable Settings

Now set up 2 design variables to vary the heating and cooling setpoint temperatures.

1. Delete all but 2 of the existing default variables and edit the first remaining variable.
2. On the **Edit Analysis Variables** dialog, start by setting the Variable type to **Custom Script**.
3. Set the Name to "Heating Setpoint".

Important Note: The name entered here on the **Edit Analysis Variables** dialog must match the corresponding name used in the custom script described below. You may like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

4. Set the **Min value** to 18, the **Max value** to 24 and the Step (optimisation) to 0.2.
5. The **Variable key** is not used in this analysis and can be left blank.

The dialog should now look like this:

Analysis Variables

General

Design Variables

Name **Heating Setpoint**

Variable type **Custom Script**

Range

Min Value **18**

Max Value **24**

Increment Step

Step (parametric) **1.00**

Step (optimisation) **0.2**

Target

Target objects **Script**

Key

Variable key

5. Press OK to confirm.

6. Now follow a similar procedure to set up the Cooling Setpoint variable. The dialog should look like this after you have finished defining the second variable:

Analysis Variables

General

Design Variables

Name **Cooling Setpoint**

Variable type **Custom Script**

Range

Min Value **22**

Max Value **28**

Increment Step

Step (parametric) **1.00**

Step (optimisation) **0.2**

Target

Target objects **Script**

Key

Variable key

Step 4 - Write Script For Custom Script Variables

You can choose whether to use Python or CS-Script for this next step.

Step 4a - CS-Script Method

The script to be used in this example is based on the "CS example: Custom Hook" example EMS script provided with the software. Follow the steps below to set it up for the analysis.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.

3. Open the **CS-Script** category and select the "CS example: Custom Hook" script. Then copy that script to create a new custom script that can be edited, select it and click on the Edit Info toolbar icon to edit the copied script.
4. Change the name of the script to "Heating and Cooling Setpoints"
5. Edit the script to apply the heating and cooling setpoints proposed by the optimiser to the model as follows:

```
using System.Collections.Generic;
using System.Linq;
using System.Windows.Forms;
using DB.Api;
using DB.Extensibility.Contracts;
namespace DB.Extensibility.Scripts
{
    public class RunCustomScript : ScriptBase, IScript
    {
        public override void OnDesignVariableChanged(int optVarId, string
variableCurrentValue)
        {
            Site site = ApiEnvironment.Site;
            Table table = site.GetTable("OptimisationVariables");
            Record record = table.Records.GetRecordFromHandle(optVarId);
            string optVariableName = record["Name"];
            // match names given to Custom Script type variables in "Optimisation
Analysis / Variables" dialog
            if (optVariableName == "Heating Setpoint")
            {
                // Actions for first custom script variable
                // Set Heating Setpoint to Variable Current Value
                foreach (BuildingBlock block in ActiveBuilding.BuildingBlocks)
                {
                    foreach (Zone zone in block.Zones)
                    {
                        double heatingSetpoint;
                        heatingSetpoint = double.Parse(variableCurrentValue);
                        zone.SetAttribute("HeatingDesignSetPointTemperature",
heatingSetpoint.ToString());
                    }
                }
                // match names given to Custom Script type variables in "Optimisation
Analysis / Variables" dialog
                else if (optVariableName == "Cooling Setpoint")
                {
                    // Set Cooling Setpoint to Variable Current Value
                    foreach (BuildingBlock block in ActiveBuilding.BuildingBlocks)
                    {
                        foreach (Zone zone in block.Zones)
                        {
                            double coolingSetpoint;
                            coolingSetpoint = double.Parse(variableCurrentValue);
                            zone.SetAttribute("ActiveCoolingDesignSetPointTemperature",
coolingSetpoint.ToString());
                        }
                    }
                }
            }
        }
    }
}
```

The **OnDesignVariableChanged** hook point is called once for each Custom script variable when the model is being prepared for the simulation. The value of **optVariableName** passed in via **optVarId** defines the variable being handled and the value of **variableCurrentValue** also passed in provides the value of this variable for this simulation. The script simply loops through the blocks and the zones and copies the setpoint requested in **variableCurrentValue** to the zone attribute.

ActiveBuilding in this case is a reference to the parametric clone copied from the base model for each simulation and so it is safe to apply any parametric changes to it without affecting the base model.

Step 4b - Python Method

Follow the steps below to set up the Python script for the analysis.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Click on the **Python-Script** category and press the Add  Info panel icon to create a new Python script.
4. Change the name of the script to "Heating and Cooling Setpoints"
5. Edit the script to apply the heating and cooling setpoints proposed by the optimiser to the model as follows:

```
def on_design_variable_changed(opt_var_id, variable_current_value):

    site = api_environment.Site
    table = site.GetTable("OptimisationVariables")
    record = table.Records.GetRecordFromHandle(opt_var_id)
    opt_variable_name = record["Name"]

    if opt_variable_name == "Heating Setpoint":
        for block in active_building.BuildingBlocks:
            for zone in block.Zones:
                heatingSetpoint = str(variable_current_value)
                zone.SetAttribute("HeatingDesignSetPointTemperature",
heatingSetpoint)
    elif opt_variable_name == "Cooling Setpoint":
        for block in active_building.BuildingBlocks:
            for zone in block.Zones:
                coolingSetpoint = str(variable_current_value)
                zone.SetAttribute("ActiveCoolingDesignSetPointTemperature",
coolingSetpoint)
```

Tip: To save time you can copy the above script and paste it into the dialog. Note that Python requires the indentation as part of its syntax and when you paste the above script the indents may become lost. In this case you must insert the indents manually after the paste. The easiest way to do that is by using the <Tab> key.

The **on_design_variable_changed** hook point is called once for each variable when the model is being prepared for the simulation. The value of **opt_variable_name**, passed in via **opt_var_id**, defines the variable being handled and the value of **variable_current_value** also passed in provides the value of this variable for this simulation. The script simply loops through the blocks and the zones and copies the setpoint requested in **variable_current_value** to the zone attribute.

active_building in this case is a reference to the parametric clone copied from the base model for each simulation and so it is safe to apply any parametric changes to it without affecting the base model.

Note: Comparing the scripts in 4a and 4b you can clearly see how the Python version is more compact and clearer to read than the CS-Script equivalent. For this reason Python will generally be the preferred scripting method for non C# developers.

6. Check the **Enable program** checkbox to ensure that the script will be executed at runtime.

7. Press OK to confirm the changes to the script.

Step 5 - Run A Standard Simulation

Click on the Simulation tab and press update to run a base annual simulation as a necessary step before running the optimisation. Before pressing OK, You should make the following settings:

1. Ensure you are running simulations for the whole year by clicking on the **Annual simulation** Info panel link.
2. Ensure you are generating only Annual/Monthly results by checking the **Monthly and Run period** interval and ensuring that all other outputs are switched off.

Leave all other settings in their default state.

Step 6 - Run Optimisation

Navigate to the **Optimisation + UA/SA** tab and press the **Update** toolbar icon to open the Optimisation Calculation Options dialog and make the following settings. These should be the same or similar to the defaults.

Calculation Options

General | **Simulation Manager**

Calculation Description

From

Start date 1 Jan

To

End date 31 Dec

Optimisation Options

Maximum generations 100

Generations for convergence 5

Initial population size 20

☐ Pause on errors

☐ Quietly retry if server becomes unavailable

Advanced

Optimisation engine 2-JEA

Generation population size 20

Max population size 500

Mutation rate 0.40

Crossover rate 1.00

Tournament size 2

Objective bias 0

Max evaluations 10000

Max wall time 240

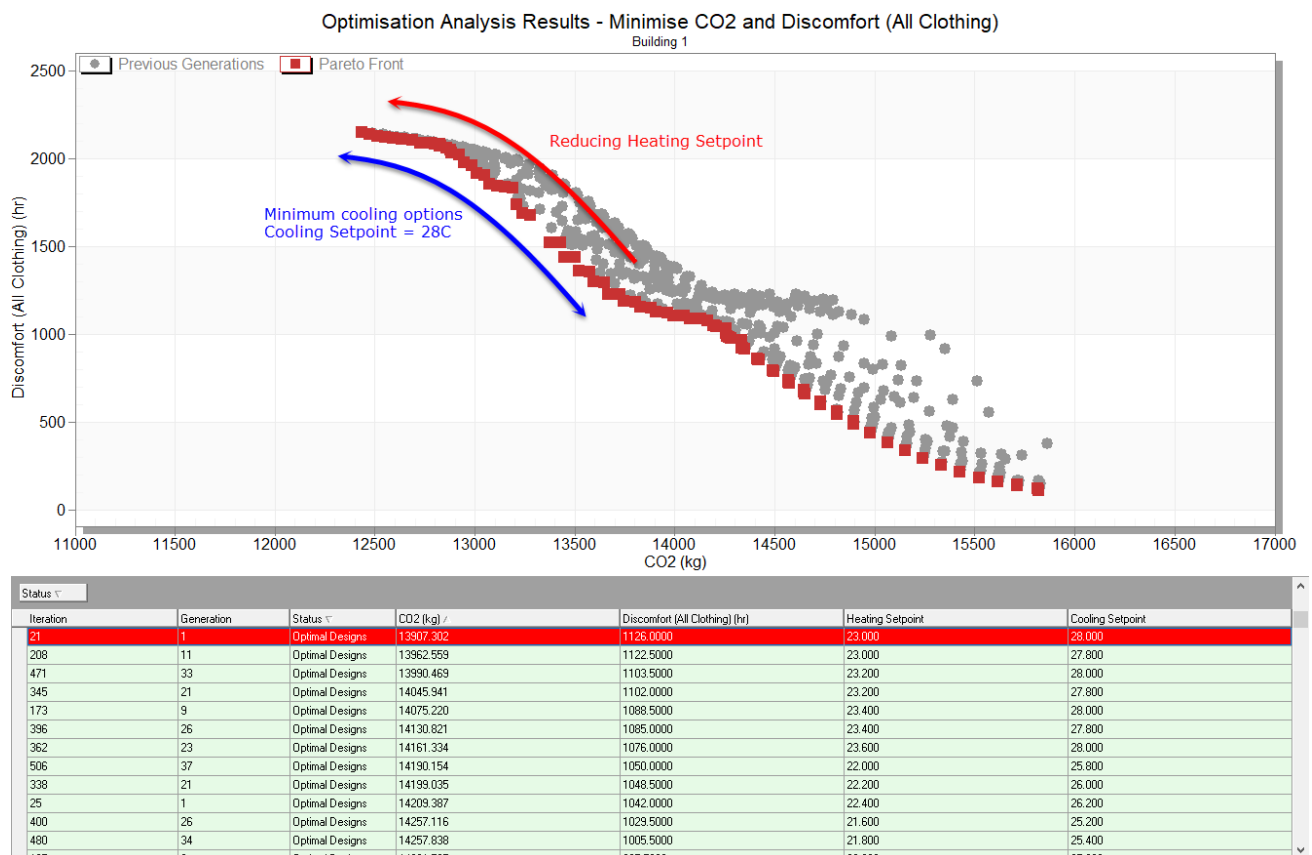
☐ Retain Simulation Manager results

☐ Backup results table each generation

Press the **Start** button to run the optimisation simulations.

Step 7 - Analyse Results

After the optimisation has completed and you have closed the Calculation Options dialog, the main screen should look similar to the screenshot below (the annotations were added manually).



The graph shows how carbon emissions tend to be higher for more comfortable optimal solutions, i.e. that there is a "tension" between discomfort and heating and cooling energy use. Depending on how we prioritise occupant comfort over energy use / environmental impact there is a range of optimal heating and cooling setpoint temperatures.

The top left group of Pareto points represent the options with cooling setpoint at its highest allowed value of 28°C.

Note: The logic used to modify the setpoint temperature is applied in a simple way in this tutorial to illustrate how to use Custom script variables to make basic changes to the model. Actually, you could have reproduced exactly the same analysis without writing scripts using the built-in [Heating setpoint temperature](#) and [Cooling setpoint temperature](#) variable types! However in a real project there might be a requirement to make corresponding adjustments to natural ventilation setpoint temperatures or other changes that should respond to changes in the design variable. More complex design variables like this can easily be set up by making the necessary adjustments to a script such as the one used in this tutorial.

Optimisation Tutorial - Custom Script KPI Example 1 - Db_eplusout_reader Python Libraries

This tutorial shows how to create a basic Custom Script KPI for an optimisation analysis with a Python script using the [db_eplusout_reader](#) library provided with DesignBuilder. It is based on an example problem to identify the set of design variations that lead to optimal building performance based on design objectives to minimise "Total system energy" consumption while at the same time minimising construction cost. DesignBuilder doesn't include a built-in "Total system energy" KPI so the tutorial demonstrates how to write a script to generate that output from the EnergyPlus results. The script calculates a project-specific definition of "Total system energy" by accessing 3 energy consumption output variables reported in the EnergyPlus SQLite output file, summing them and feeding the result back to DesignBuilder after each simulation to process as a KPI value.

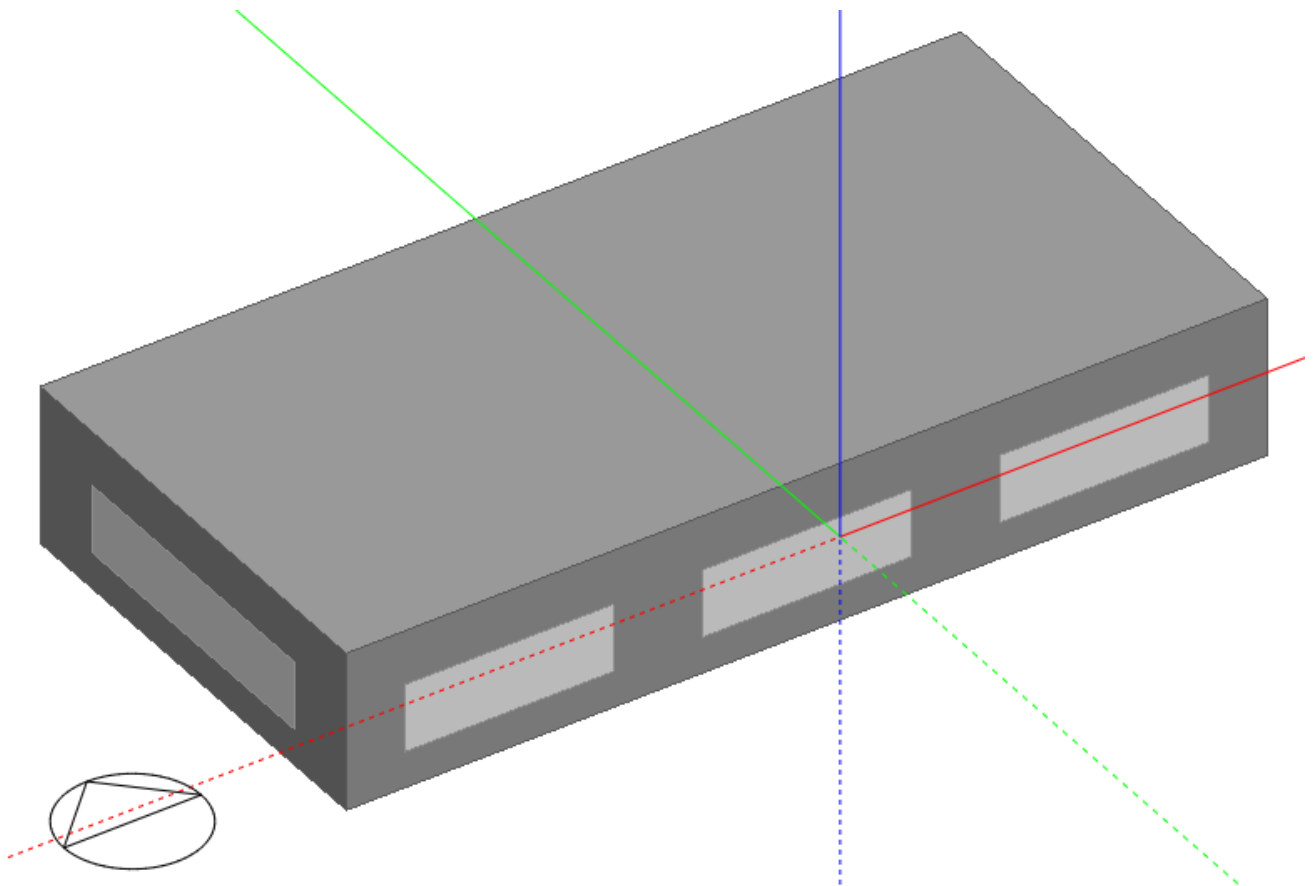
The model is the same base model used in the previous "Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760, but in this case it uses a Unitary Heat Cool HVAC system to allow EnergyPlus to calculate a realistic energy consumption value. Outputs for individual end uses are combined in the script to form the "Total system energy" KPI.

Optimal settings for Window To Wall %, Glazing type and Shading overhang length will be identified for the building that provide the lowest total system energy consumption while keeping construction costs as low as possible. Default building component costs and other settings are used throughout to allow us to focus on the Custom Script KPI.

The steps involved are as follows.

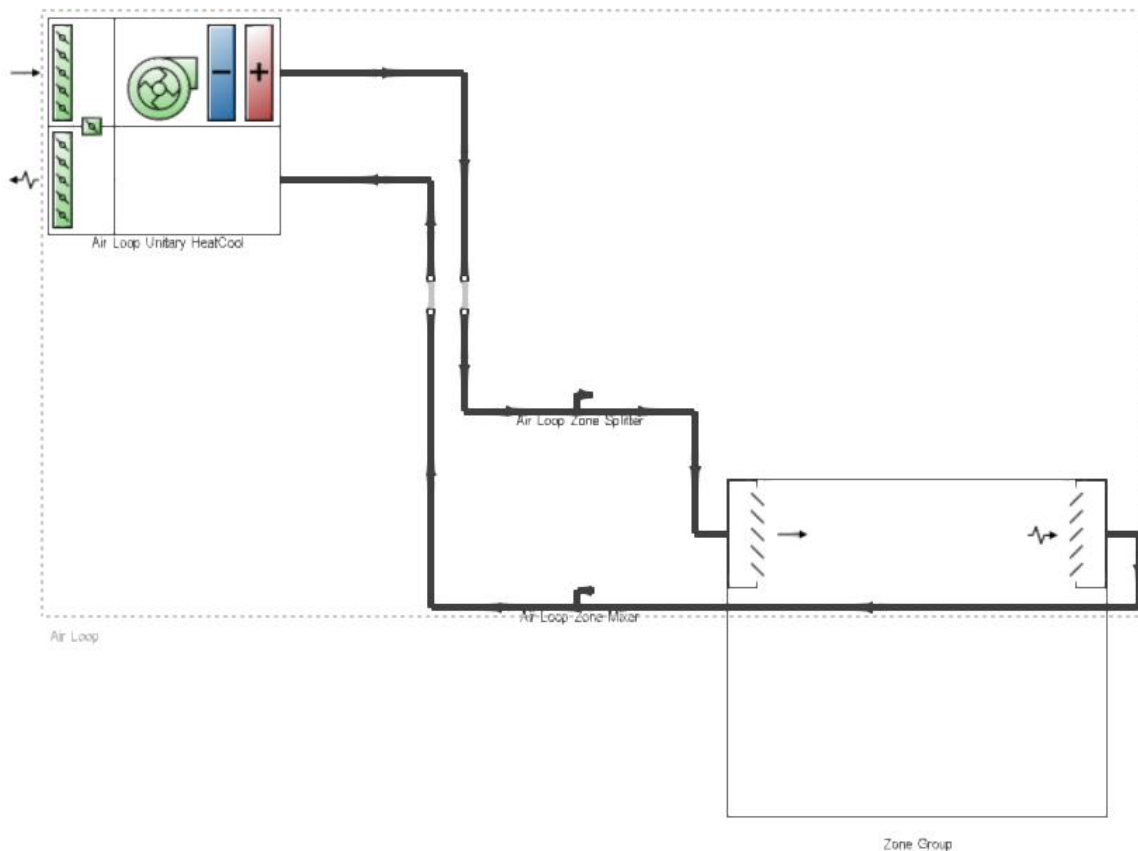
Step 1 - Create A Base Model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 20m by 10m with block height of 3.5m as shown below.



3. Load the **Generic Office Area** Activity template at building level on the Activity Tab.
4. On the Lighting tab at building level switch on Lighting control
4. On the Model options dialog set [Detailed HVAC](#) option.
5. Navigate to the **<HVAC System>** node and click on the [Load HVAC Template](#) toolbar icon. On the dialog select the **Unitary Heat Cool** template and press the Next button on the wizard.
6. Select the zone to be the control zone - in this single zone model it will be Block1:Zone1.
7. Press Finish to load the template.

8. The **Unitary heat cool** system is one of the simplest systems capable of maintaining comfortable conditions by heating and cooling multiple zones from a single AHU.



Step 2 - Run A Standard Simulation

Click on the Simulation tab and run a base annual simulation. Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation process. Check the hourly results for the simulation period to ensure that the model is behaving as expected, including temperatures within the building, operations periods etc. Make any adjustments to the model and repeat this step until you are happy with the base model hourly results.

Step 3 - Make Optimisation Objective Settings

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon and set the [Analysis type](#) on the Analysis tab to **2-Optimisation**.
2. Go to the [Objectives](#) tab and add 2 new objectives for construction capital cost and total system energy. You can either edit the existing objectives or first delete those and create 2 new objectives. The steps involved are described below.

Total System Energy

To create the "Total system energy" objective, a new Summary Output (aka KPI) must be created. Click on the **Objective KPI** to open the list of existing Summary Outputs in the Info panel. Our "Total system energy" KPI won't be listed there at first, so create it now by clicking on the existing **Custom Script** Summary Output on the list of KPIs, click on the **Create copy of highlighted item** Info toolbar icon and edit the copy. Make the following settings on the Summary Output dialog.

Summary Output

General

General

Name **Total system energy (kWh)**

Category Custom script

Important Note: The name, "Total system energy (kWh)", entered here on the Summary Outputs dialog must match the text written to field 0 of the record added to the "ParamResultsTmp" table in the [script](#). You may like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

Save the newly created KPI and select it on the **Optimisation Objectives** dialog.

Name the objective "Total system energy". That name is for your own reference only and unlike the Summary Output name, it does not need to match anything else.

Select the **1-Min** Min/Max option to say that you wish to minimise EUI. The dialog should now look something like the screenshot below.

Optimisation Objectives

General

Objectives

Name **Total system energy**

Min/Max 1-Min

Objective KPI Total system energy (kWh)

Capital Cost

Now create the cost objective. This is straightforward as we can use a standard KPI.

Select the **Capital cost (Total Capex)** KPI and call the objective "Capital Cost".

Select the **1-Min** Min/Max option to say that you wish to minimise EUI. The objectives dialog should now look like the screenshot below.

Optimisation Objectives

General

Objectives

Name **Capital Cost**

Min/Max 1-Min

Objective KPI Capital cost (Total Capex)

With both objectives defined, the Objectives tab should now be set up as follows:

Parametric, Optimisation and UA/SA Analysis Settings

Analysis Type	Objectives	Additional Outputs	Constraints	Design Variables
Name	Min/Max	Objective KPI		
Total system energy	1-Min	Total system energy (kWh)		
Cost	1-Min	Capital cost (Total Capex)		

Step 4 - Make Optimisation Variable Settings

Now set up the design variables. The 3 variables to be included are:

- Window to Wall % - ranging from 20% to 80% with an optimisation step of 1%.
- Glazing type - select the following glazing systems as shown in the **Variable Order** dialog:



Re-arrange the order of the list items to match the position of each option in the sequence.

OK

Help

Up

Down

DbI Clr 3mm/13mm Air

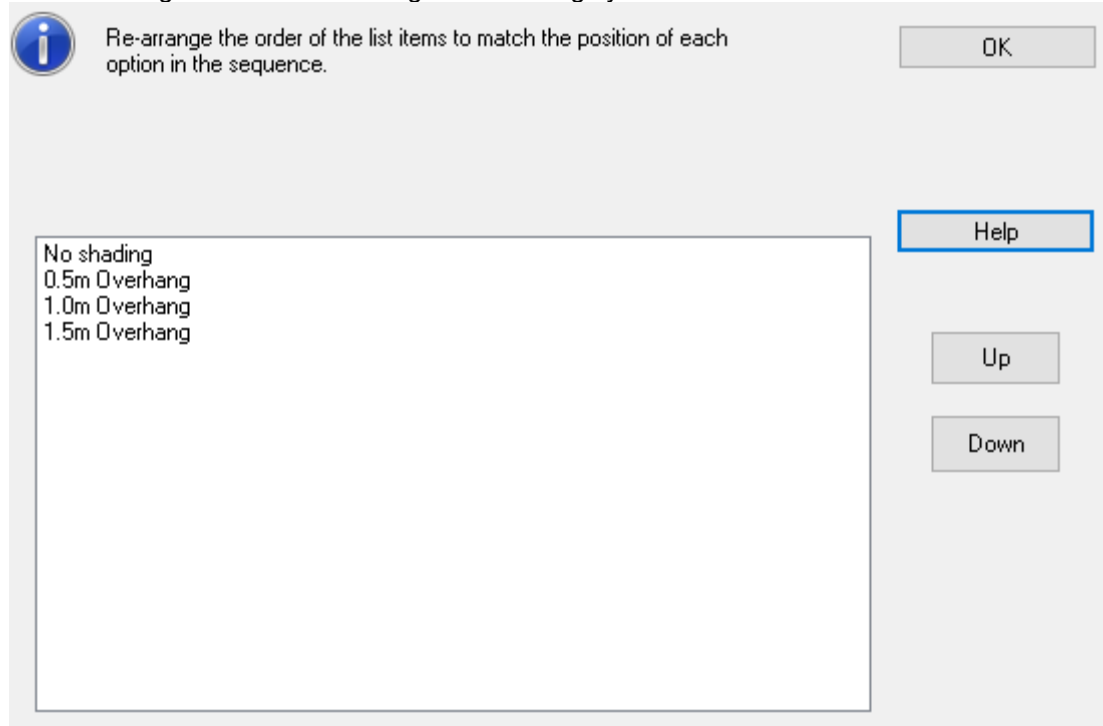
DbI Ref-B-H Tint 6mm/6mm Air

DbI LoE (e2=.1) Clr 3mm/13mm Arg

Trp Clr 3mm/13mm Air

Trp Clr 3mm/13mm Arg

- Local shading - select the following local shading systems as shown in the **Variable Order** dialog:



With all 3 variables defined, the Variables tab should now be set up as follows:

Parametric, Optimisation and UA/SA Analysis Settings							
Analysis Type	Objectives	Additional Outputs	Constraints	Design Variables			
Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisati...	Options list	Target objects
Window to wall %	Window to wall %	20.00	80.00	10.00	1.000	-	Building
Glazing type	Glazing type	0.00	0.00	0.00	0.000	5 options	Building
Local shading ty...	Local shading ty...	0.00	0.00	0.00	0.000	4 options	Building

Step 5 - Write Custom Script

The next task is to write the script that will read the simulation results from the ESO file, process them and pass back to DesignBuilder as a Custom KPI. Steps are as follows.

- Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
- Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
- Click on the **Python-Script** category and press the **Add new item** Info toolbar icon to open the Edit script dialog.
- Copy and paste the script below into the script window. In summary, the script works as follows. The key heating, cooling and system fan energy consumption outputs are loaded from the eplusout.sql simulation output file, summed to create a total and then written to the temporary "ParamResultsTmp" DesignBuilder results table used for the purpose.

Python example: Optimisation using db_eplusout_reader

```
from db_eplusout_reader import Variable, get_results
from db_eplusout_reader.constants import *
from datetime import datetime
```

```
# ensure that SQLite outputs are generated
def before_energy_idf_generation():
    site = api_environment.Site
    for building in site.Buildings:
```

```

        building.SetAttribute("SSSQLiteOP", "1")

def after_energy_simulation():
    file = api_environment.EnergyPlusFolder + r"eplusout.sql"
    variables = [
        Variable("", "Heating Coil Electricity Energy", "J"),
        Variable("", "Cooling Coil Electricity Energy", "J"),
        Variable("", "Fan Electricity Rate", "W"),
    ]
    results = get_results(file, variables=variables, alike=True, frequency=RP)
    convertJ_kWh = 3600000
    convertW_kWh = 8.76

    heating = results.arrays[0][0] / convertJ_kWh
    cooling = results.arrays[1][0] / convertJ_kWh
    fan = results.arrays[2][0] * convertW_kWh

    system_energy_kWh = str(heating + cooling + fan)

    site = api_environment.Site
    table = site.GetTable("ParamResultsTmp")

    record = table.AddRecord()
    record[0] = "Total system energy (kWh)"
    record[1] = str(system_energy_kWh)

```

Note: The heating and cooling coil electric energy variable outputs have been divided by 3,600,000 and fan power multiplied by 8.76 to convert the units from Joules to kWh and Watts to kWh respectively.

Tip: To save time you can copy the above script and paste it into the dialog. Note that Python requires the indentation as part of its syntax and when you paste the script the indents may become lost. In this case you must insert the indents manually after the paste. The easiest way to do that is by using the <Tab> key.

Tip: The [DesignBuilderSoftware \ db-scripts](#) Github repository includes the latest versions of the above script as well as other scripts used in DesignBuilder tutorials and examples.

Step 6 - Re-Run A Standard Simulation

Click on the Simulation tab and press update to re-run a base annual simulation.

Step 7 - Run The Optimisation

Navigate to the **Optimisation + UA/SA** tab on the Simulation screen and press the **Update** toolbar icon to open the Optimisation Calculation Options dialog and make the following settings. These should be the same or similar to the defaults.

Calculation Options

General

Simulation Manager

Calculation Description

From

Start date1 Jan

To

End date31 Dec

Optimisation Options

Generation population size40

Generations for convergence5

☒ Run all combinations

Advanced

Maximum generations20

Maximum population size10

Mutation rate0.40

Crossover rate1.00

Tournament size2

Objective bias0

Maximum evaluations10000

Maximum wall time240

☐ Pause on errors

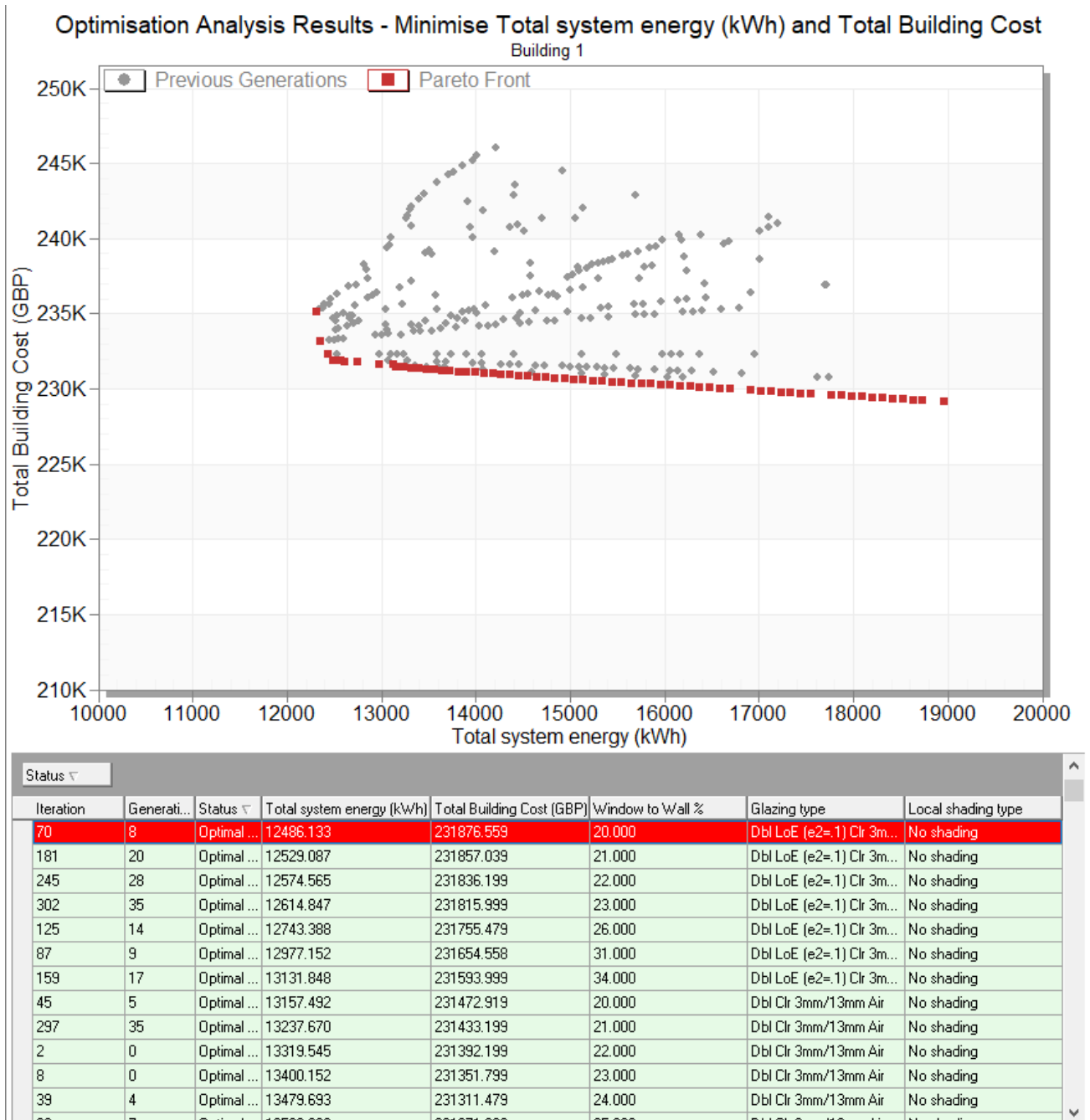
☐ Quietly retry if server becomes unavailable

☐ Retain Simulation Manager results

☐ Backup results table each generation

Press the **Start** button to run the optimisation simulations.

After the optimisation has converged and you have closed the Calculation Options dialog, you should see results similar to those below on the main screen.



The optimisation results showing the trade off between Total energy consumption and capital cost will be dependent on the costs associated with the various building components, especially the glazing, external walls and overhang shading fins. Given the default costs used in this model, we can see that for a balanced energy performance at 12486 kWh, it necessary to spend 231,876 GBP for 20% WWR, double glazed low-e glazing with no overhangs.

Optimisation Tutorial - Custom Script KPI Example 2 - Reading The EUI KPI Using EpNet (C#) And Eppy (Python)

DesignBuilder Optimisation, Parametric Analysis and Uncertainty and Sensitivity Analysis tools provide an extensive list of Key Performance Indicators (KPIs) that can be used to assess the performance of building design variants, but in some cases you may need to create your own specific KPIs to assess a design. This example shows how to write a C# or Python script to create your own KPI that reads any EnergyPlus output and links it to an optimisation objective.

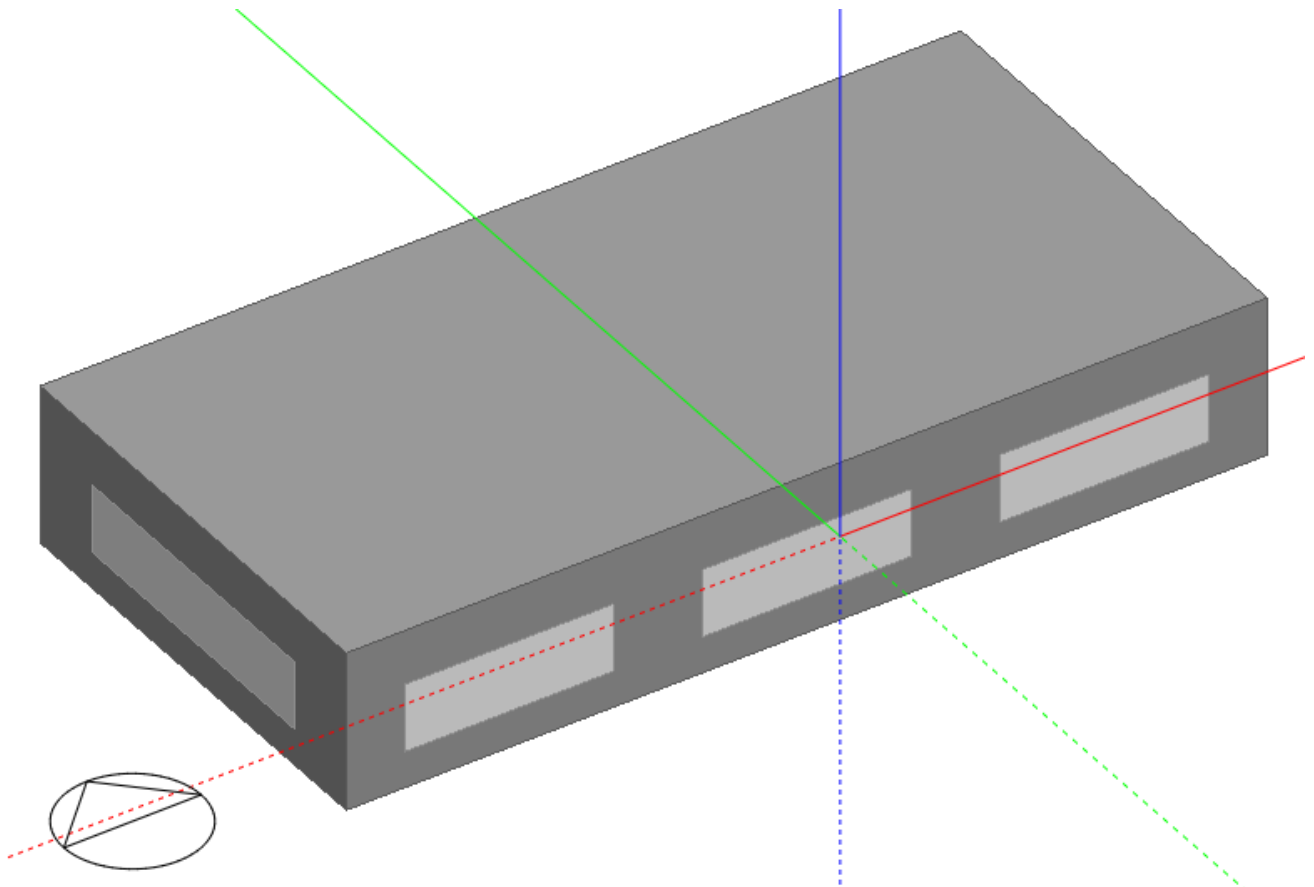
The tutorial is based on an example problem to identify the set of design variations that lead to optimal building performance based on Energy Use Intensity (EUI), while at the same time minimising construction cost. Current versions of DesignBuilder don't include a built-in EUI KPI so the tutorial demonstrates how to write a script to access that output from the EnergyPlus results.

Optimal settings for Window To Wall %, Glazing type and Shading overhang length will be identified for the building that provide the lowest EUI while keeping costs as low as possible. Default building component costs and other settings are used throughout to allow us to focus on the Custom Script KPI.

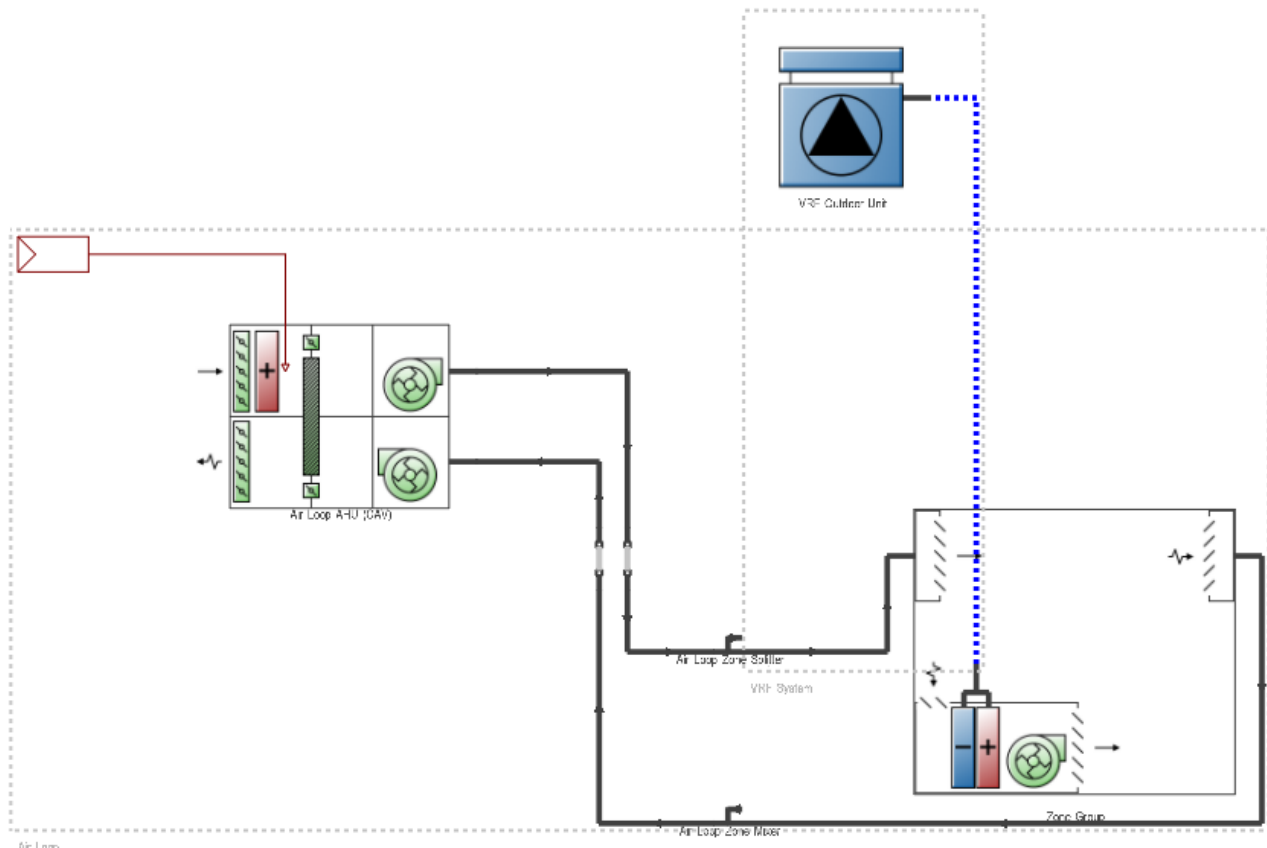
The model has a VRF system and it uses Detailed HVAC to allow EnergyPlus to calculate a realistic EUI value. Simple HVAC does not allow the HVAC energy consumption to be included in the EUI reported by EnergyPlus. The default VRF HVAC template is used.

Step 1 - Create A Base Model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 20m by 10m with block height of 3.5m as shown below.



3. Load the **Generic Office Area** Activity template at building level on the Activity Tab.
4. Go to the Lighting tab and switch on Lighting control. This makes for a complex relationship between the façade configuration and heating, cooling and lighting energy consumption which would make it nigh on impossible to pre-guess the outcome without an optimisation analysis.
5. Set the Detailed HVAC model option and load the default **VRF with HR and DOAS** HVAC template.



Step 2 - Run A Standard Simulation

Click on the Simulation tab and run a base annual simulation. Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation process. Check the hourly results for the simulation period to ensure that the model is behaving as expected, including temperatures within the building, operations periods etc. Make any adjustments to the model and repeat this step until you are happy with the base model hourly results.

Step 3 - Make Optimisation Objective Settings

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon and set the [Analysis type](#) on the Analysis tab to **2-Optimisation**.
2. Go to the [Objectives](#) tab and add 2 new objectives for construction capital cost and EUI. You can either edit the existing objectives or first delete those and create 2 new objectives. The steps involved are described below.

EUI

To create the "EUI" objective, a new Summary Output (aka KPI) must be created. Click on the **Objective KPI** to open the list of existing Summary Outputs in the Info panel. The "EUI" KPI won't be listed there at first, so create it now by clicking on the existing **Custom Script** Summary Output on the list of KPIs, click on the **Create copy of highlighted item** Info toolbar icon and edit the clone. Make the following settings on the Summary Output dialog.

Summary Output

General

General

Name

Category Custom script

Important Note: The name, "EUI (kWh/m2)", entered here on the Summary Outputs dialog must match the text written to field 0 of the record added to the "ParamResultsTmp" table in the [script](#). You may like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

Save the newly created KPI and select it on the **Optimisation Objectives** dialog.

Name the objective "EUI". That name is for your own reference only and unlike the Summary Output name, does not need to match anything else. Select the **1-Min** Min/Max option to say that you wish to minimise EUI. The dialog should now look something like the screenshot below.

Optimisation Objectives

General

Objectives

Name

Min/Max 1-Min

Objective KPI EUI (kWh/m2)

Capital Cost

Now create the cost objective. This is more straightforward as we can use a standard KPI.

Select the **Capital cost (Total Capex)** KPI and call the objective "Capital Cost". Select the **1-Min** Min/Max option to say that you wish to minimise EUI. The objectives dialog should now look like the screenshot below.

Optimisation Objectives

General

Objectives

Name

Min/Max 1-Min

Objective KPI Capital cost (Total Capex)

With both objectives defined, the Objectives tab should now be set up as follows:

Parametric, Optimisation and UA/SA Analysis Settings		
Analysis Type	Objectives	Additional Outputs
Name	Min/Max	Objective KPI
EUI	1-Min	EUI (kWh/m2)
Cost	1-Min	Capital cost (Total Capex)

Step 4 - Make Optimisation Variable Settings

Now set up the design variables. The 3 variables to be included are:

- Window to Wall % - ranging from 20% to 80% with an optimisation step of 1%.

- Glazing type - select the following glazing systems as shown in the **Variable Order** dialog:

 Re-arrange the order of the list items to match the position of each option in the sequence.

OK

Help

Up

Down

Dbl Clr 3mm/13mm Air
 Dbl Ref-B-H Tint 6mm/6mm Air
 Dbl LoE (e2=.1) Clr 3mm/13mm Arg
 Trp Clr 3mm/13mm Air
 Trp Clr 3mm/13mm Arg

- Local shading - select the following local shading systems as shown in the **Variable Order** dialog:

 Re-arrange the order of the list items to match the position of each option in the sequence.

OK

Help

Up

Down

No shading
 0.5m Overhang
 1.0m Overhang
 1.5m Overhang

With all 3 variables defined, the Variables tab should now be set up as follows:

Parametric, Optimisation and UA/SA Analysis Settings							
Analysis Type	Objectives	Additional Outputs	Constraints	Design Variables			
Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisati...	Options list	Target objects
Window to wall %	Window to wall %	20.00	80.00	10.00	1.000	-	Building
Glazing type	Glazing type	0.00	0.00	0.00	0.000	5 options	Building
Local shading ty...	Local shading ty...	0.00	0.00	0.00	0.000	4 options	Building

Step 5 - Write Custom KPI Script

The next task is to write the script that will feed the EUI result into the parametric results reading system. Steps are as follows.



1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Depending on whether your script will be written in C# or Python, click either on the **CS-Script** or **Python-Script** category and press the **Add new item** Info toolbar icon to open the Edit script dialog.
4. Copy and paste the script below into the script window. In summary, the script works as follows. The eplustbl.csv simulation output file is loaded into the [EpNet](#) library (when using CS-Script) or to the [eppy](#) library (when using Python); the LEED "EAp2-17a. Energy Use Intensity - Electricity" table is opened and the EUI sub-total for Electricity loaded. The script then writes that to a temporary table used by DesignBuilder to pick up custom script results for each optimisation simulation. Our example building is entirely powered by electricity, so the Electricity EUI is also the total EUI.

CS-Script

```
using System;
using System.IO;
using System.Collections.Generic;
using System.Linq;
using System.Windows.Forms;
using DB.Api;
using DB.Extensibility.Contracts;
using EpNet;
using EpNet.Results;

namespace DB.Extensibility.Scripts
{
    public class RunCustomScript : ScriptBase, IScript
    {
        public override void AfterEnergySimulation ()
        {
            // called after simulation is finished but before results are
            loaded to DB

            var tablePath = ApiEnvironment.EnergyPlusFolder + @"eplustbl.csv";
            if (File.Exists(tablePath)) {
                var tableReader = new TableReader(tablePath);
                if (tableReader != null) {
                    // open the LEED EUI Electricity results table
                    ResultTable EUITable = tableReader.GetTable("EAp2-17a.
                    Energy Use Intensity - Electricity");
                    // read the sub total for EUI Electricity
                    string EUI = EUITable.Rows[7][1].ToString();

                    // store results in the site level "ParamResultsTmp"
                    table, one record per result

                    Site site = ApiEnvironment.Site;
                    Table table = site.GetTable("ParamResultsTmp");
                    // field 0 is the Name of the corresponding
                    SummaryResults record, field 1 is the result
                    Record record = table.AddRecord();
                    record[0] = "EUI (kWh/m2)";
                    record[1] = EUI;
                } else {
                    MessageBox.Show(@"Error creating TableReader");
                }
            }
        }
    }
}
```

```

        } else {
            MessageBox.Show(@"File does not exist " + tablePath);
        }
    }
}

```

Python-Script

```

import ctypes
from os import path
from eppy.results import fasthtml

def show_message(title, text):
    ctypes.windll.user32.MessageBoxW(0, text, title, 0)

def after_energy_simulation():
    table_path = api_environment.EnergyPlusFolder + "eplustbl.htm"
    if path.exists(table_path):
        with open(table_path, "r") as filehandle:
            eui_table = fasthtml.tablebyname(filehandle, "EAp2-17a. Energy Use
Intensity - Electricity")
            eui_table_content = eui_table[1]
            eui = str(eui_table_content[8][1])

            site = api_environment.Site
            table = site.GetTable("ParamResultsTmp")
            record = table.AddRecord()
            record[0] = "EUI (kWh/m2)"
            record[1] = eui
    else:
        show_message("Information", "File does not exist " + table_path)

```

Tip: The example EUI scripts and associated optimisation KPIs and objectives described above can be easily adapted to report *any* result included in the EnergyPlus .csv (for EpNet) or .htm (for eppy) output files by making the appropriate modifications.

Tip: To save time you can copy the above script and paste it into the dialog.

Tip: The [DesignBuilderSoftware \ db-scripts](#) Github repository includes the latest versions of the above script as well as other scripts used in DesignBuilder tutorials and examples.

Step 6 - Re-Run A Standard Simulation

Click on the Simulation tab and press update to re-run a base annual simulation. Before pressing OK, ensure that you are generating LEED Summary outputs as this data is required by the custom script.

Step 7 - Run The Optimisation

Navigate to the **Optimisation + UA/SA** tab on the Simulation screen and press the **Update** toolbar icon to open the Optimisation Calculation Options dialog and make the following settings. These should be the same or similar to the defaults.

Calculation Options

General

Simulation Manager

Calculation Description

From

Start date1 Jan

To

End date31 Dec

Optimisation Options

Generation population size40

Generations for convergence5

☒ Run all combinations

Advanced

Maximum generations20

Maximum population size10

Mutation rate0.40

Crossover rate1.00

Tournament size2

Objective bias0

Maximum evaluations10000

Maximum wall time240

☐ Pause on errors

☐ Quietly retry if server becomes unavailable

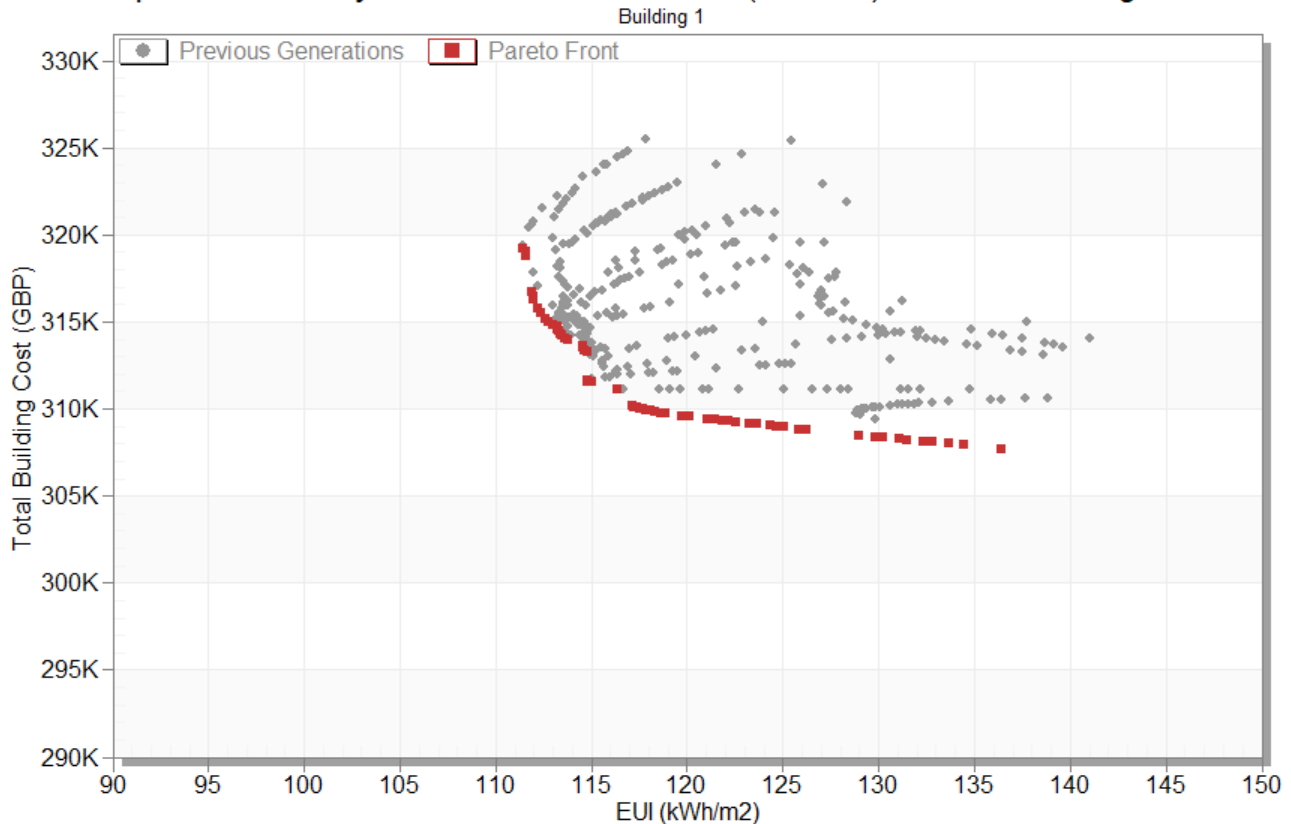
☐ Retain Simulation Manager results

☐ Backup results table each generation

Press the **Start** button to run the optimisation simulations.

After the optimisation has converged and you have closed the Calculation Options dialog, you should see results similar to those below on the main screen.

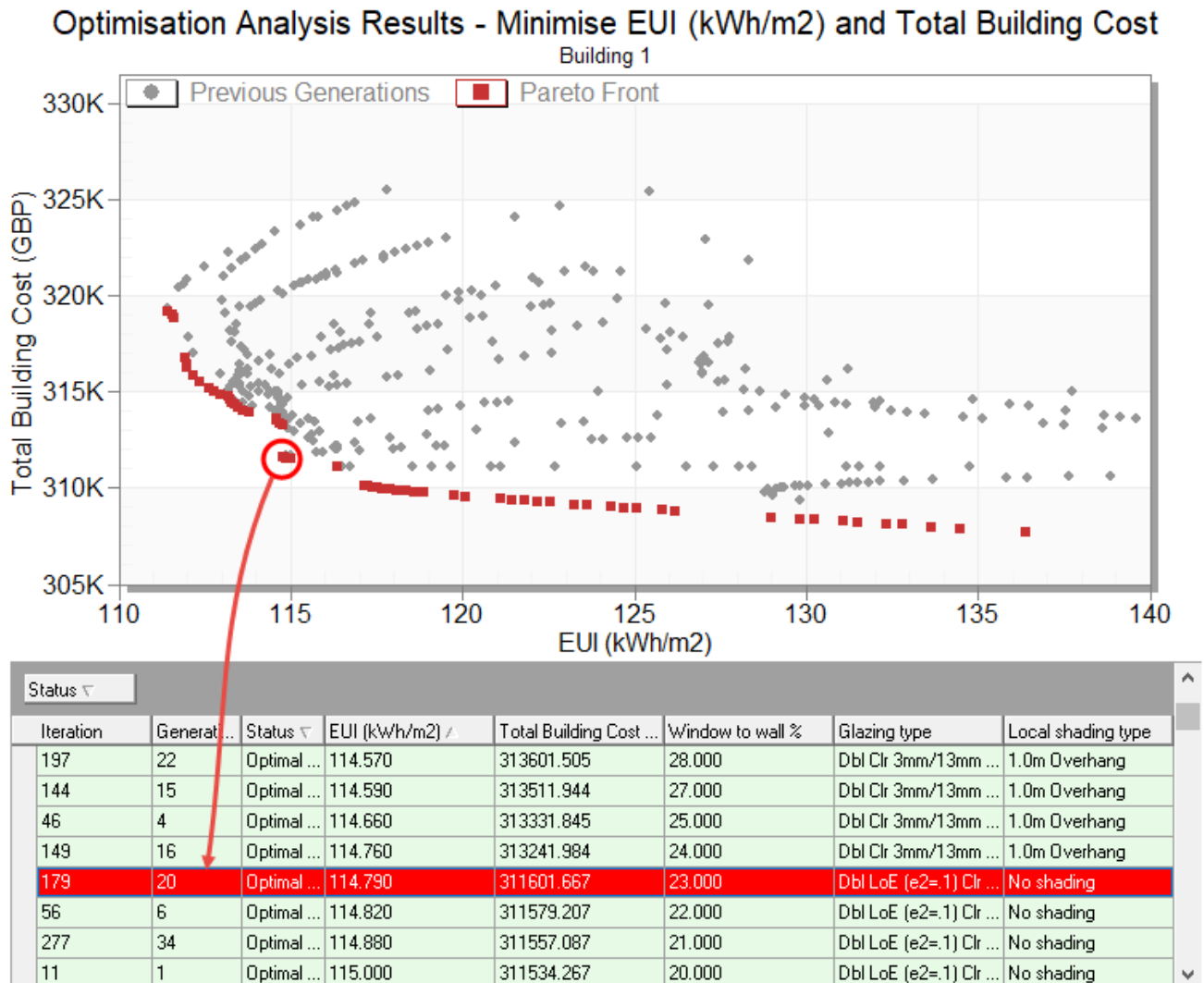
Optimisation Analysis Results - Minimise EUI (kWh/m2) and Total Building Cost



Iteration	Generation	Status ▾	EUI (kWh/m2) /	Total Building Cost (GBP)	Window to wall %	Glazing type	Local shading type
Optimal Designs							
329	42	Optimal D...	111.420	319173.710	45.000	Dbl LoE (e2=.1) Clr 3m...	1.5m Overhang
153	17	Optimal D...	111.550	318994.301	44.000	Dbl LoE (e2=.1) Clr 3m...	1.5m Overhang
52	5	Optimal D...	111.610	318814.531	43.000	Dbl LoE (e2=.1) Clr 3m...	1.5m Overhang
344	44	Optimal D...	111.920	316747.302	36.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
362	47	Optimal D...	111.960	316432.723	34.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
117	12	Optimal D...	111.980	316275.442	33.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
287	35	Optimal D...	112.170	315803.564	30.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
129	14	Optimal D...	112.360	315488.985	28.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
301	38	Optimal D...	112.610	315174.403	26.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
234	28	Optimal D...	112.770	315017.125	25.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang
50	5	Optimal D...	112.980	314859.824	24.000	Dbl LoE (e2=.1) Clr 3m...	1.0m Overhang

The optimisation results showing the trade off between EUI and capital cost will be highly dependent on the costs associated with the various building components, especially the glazing, external walls and overhang shading fins. Given the default costs used in this model, we can see that for the very best energy performance at 111.4 kWh/m2, it necessary to spend 319,173 GBP for 45% WWR, double glazed low-e glazing with 1.5m overhangs.

The Pareto graphs (above and below) clearly show that there are diminishing returns in energy savings as more glazing and shading overhang length are added to reduce EUI. Construction costs can be saved by reducing glazing and overhang lengths with a possible "sweet spot" illustrated in the screenshot below with 23% WWR, double glazed low-e glazing and no shading keeping costs down while still keeping EUI low at 114.8 kWh/m2.



Impact of HVAC Modelling on Fabric Design Optimisation Results

This example is essentially the same as the previous optimisation custom KPI example but it is based on a) a VRF system instead of Unitary Heat Cool and b) the objective is to minimise "EUI" instead of "Total system energy". It is interesting to note that the results in this example are rather more straightforward in that they easily point to a specific configuration whereas the results for the VRF system when using EUI as an objective are more nuanced. Why do you think this might be?

One explanation to consider is that the Unitary Heat Cool system is significantly more energy intensive compared with the VRF system (at least partly because the DesignBuilder **Unitary Heat Cool** system introduces higher mechanical ventilation fresh air change rates than the default **VRF with DOAS** template) and so heating and cooling loads dominate over lighting and other loads. This difference will of course influence the conclusions of the analysis.

To test this idea you might like to try loading a Unitary Heat Cool system to replace the VRF system and see what results you get. Which system leads to the higher EUIs? Are the same set of optimal façade design configurations recommended?

Tip: One takeaway for anyone having completed both this tutorial and the previous one, is that even when you are analysing fabric/facade only design variables, the type of HVAC system used can influence the decision-making process! If you know that the design will use a VRF system then it's important to set that up in the model used for the optimisations. That way the right balance of heating and cooling influence will be included in the overall result. If you only consider zone loads then you miss out on the difference between heating and cooling efficiency. In general you should think carefully about what the results are

telling you and about any assumptions that may be built into them. As always it helps to understand the base model and its results well before running the optimisations.

Optimisation Tutorial - Custom Script KPI Example 3 - Using EMS + Python Scripts To Optimise Controls For Comfort And Heating

This tutorial demonstrates how to find the optimum set of heating and natural ventilation system control set point temperatures that simultaneously minimise Fanger PPD discomfort and heating load across the whole building. Because EnergyPlus does not include a suitable whole building comfort PPD metric a **Custom EMS script** is written to generate a "Percentage Discomfort Area" output for each simulation to characterise the overall degree of discomfort for all zones in the building. When combined with a **Custom Script KPI** it can be used as a design objective in the optimisation.

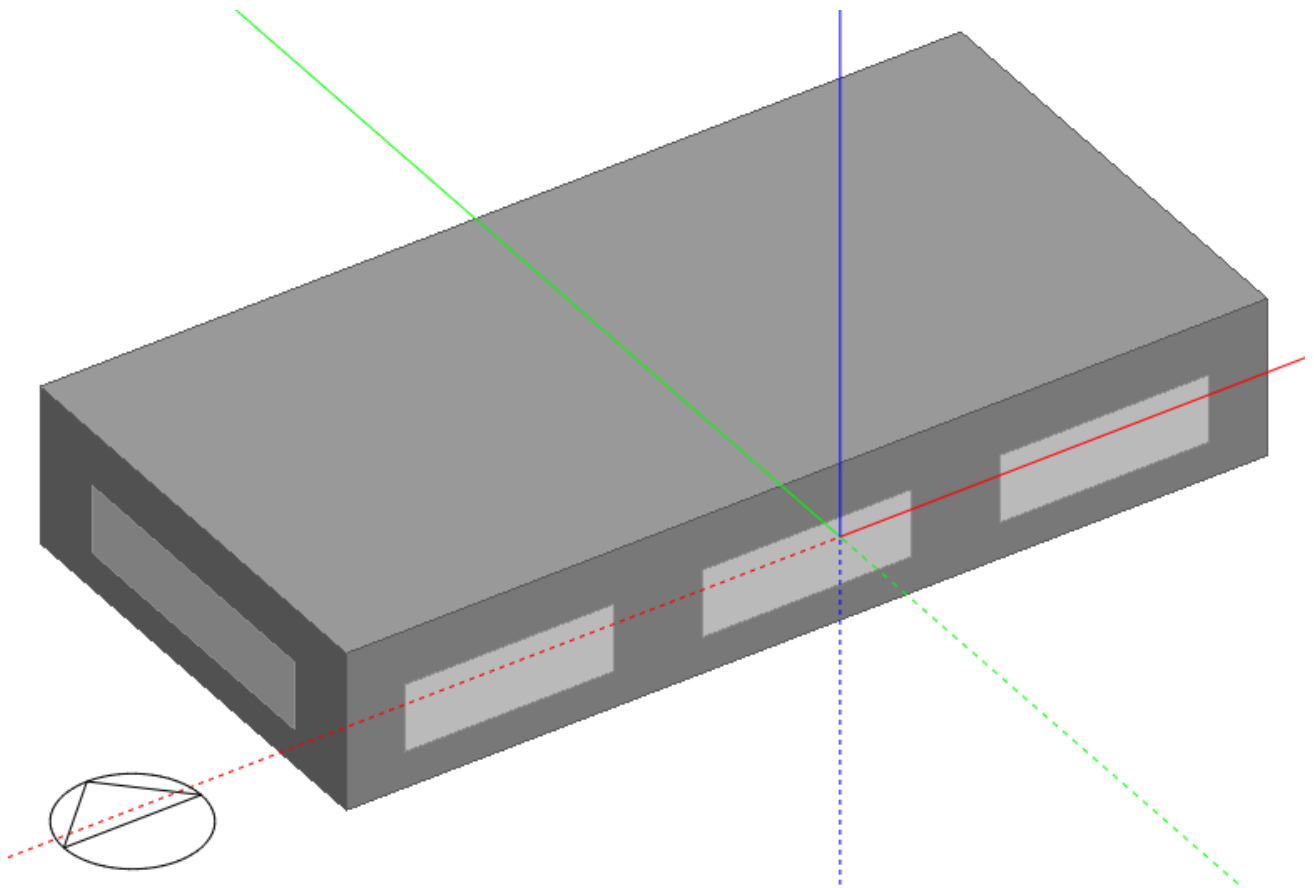
The EMS script calculates the overall building "Percentage Discomfort Area" at each timestep of the simulation (during occupied periods only) by summing the floor area of uncomfortable zones having Fanger PPD < 20% and dividing by the total floor area. The % discomfort area is averaged over the year as a single value for the building which is read and processed in a Custom KPI script written in Python. The Python KPI script uses the [db_eplusout_reader](#) library provided with DesignBuilder to read and process the comfort EMS output from the SQL simulation output file and feeds the result back to DesignBuilder after each simulation to process.

Optimal settings for heating and natural ventilation setpoints are identified that provide the lowest heating loads while also minimising PPD discomfort levels.

The main learning point is to understand the process used to generate a custom output from an EnergyPlus simulation using EMS and use it as a KPI to identify optimum design parameters.

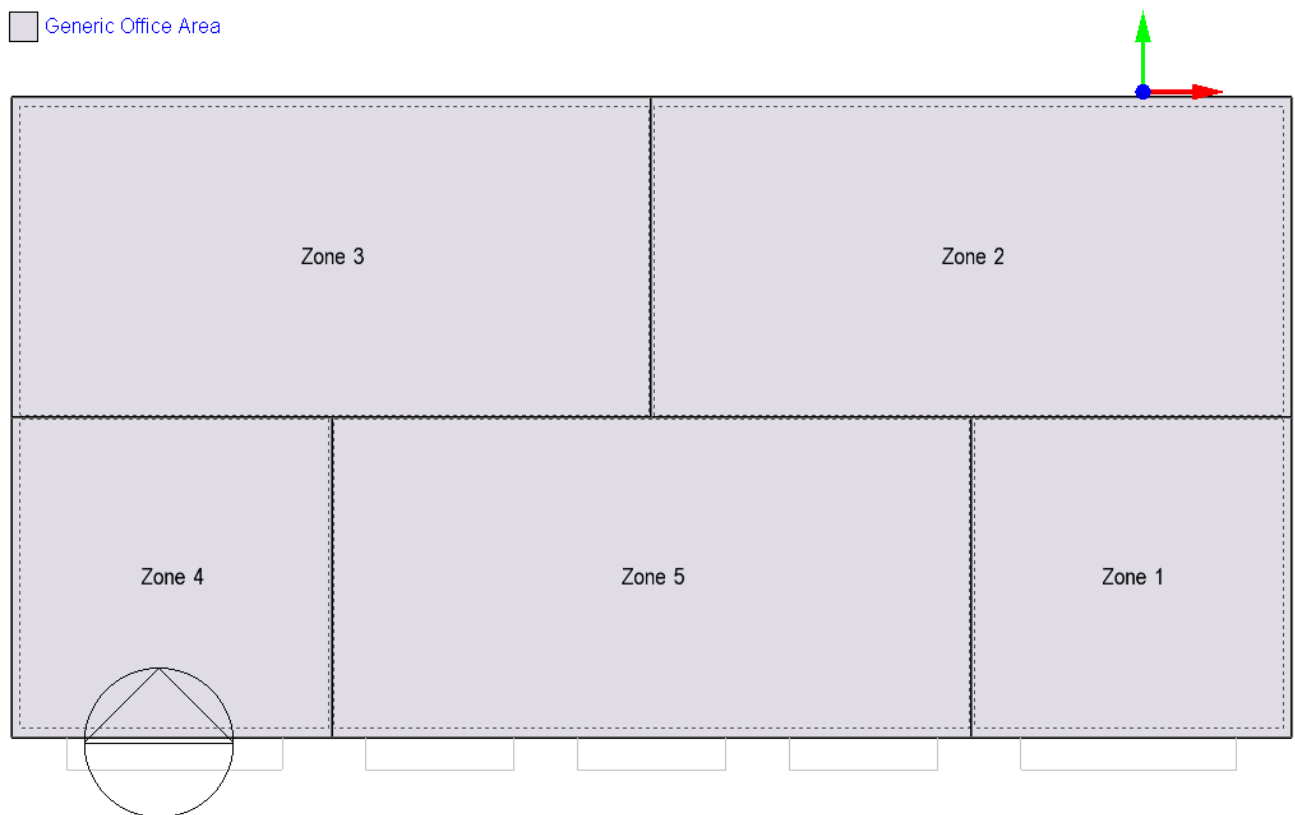
Step 1 - Create A Base Model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 40m by 20m with block height of 3.5m as shown below.



3. Sub-divide the block into several zones as follows. The SE and SW-facing corner zones both have 10 x 10m floor area. For the purposes of this tutorial the objective is to create zones that will overheat to varying degrees over the course of hot days. The exact layout is not particularly important:

 Generic Office Area



4. Open the Model options dialog:

- Select the [Calculated natural ventilation](#) option.
 - Select a (relatively high) 10 [Timesteps per hour](#) option since we are interested in the interaction of the heating and venting setpoint temperatures and it is a Calculated natural ventilation model.
 - Select the [Fanger](#) output option since we will be reading the base PPD results generated by EnergyPlus.
5. On the Activity Tab at building level:
- Load the **Generic Office Area** Activity template (this is usually the default so you probably won't need to follow this step).
 - Select the **8:00 - 18:00 Mon - Fri** occupancy schedule which can be found under in <General> category listing. This is selected because the default office occupancy schedule includes some low level occupancy after the office heating schedule has switched off at the end of each day (to account for cleaning) and this change ensures that we don't test for discomfort during times when the heating is not switched on.
6. On the Openings tab at building level:
- Switch on [Window shading](#) and use the default **Blind with high reflectivity slats** option.
 - Set the [% Glazing area opens](#) to 50%.
8. On the Lighting tab load the **LED with linear control** Lighting template.
9. On the HVAC tab:
- Load the **HW Convectors, Nat Vent** HVAC template.
 - Select the **Winter heating (Northern Hemisphere)** Heating Operation schedule which can be found under in <General> category listing.
 - Switch on the [Modulate opening areas](#) Natural ventilation option to reduce window opening areas during periods requiring heating to reduce the chances of simultaneous heating and venting.

Step 2 - Run A Simulation To Check The Base Model

Click on the Simulation tab and run a set of simulations to check the operation of the nat vent system to make sure you understand how it works in practice. You will probably want to run simulations for at least summer design, summer typical and winter design weeks looking at hourly or sub-hourly results to give the clearest picture of building behaviour. Does it maintain adequate comfort during the summer? Is the air change rate excessive at any point? Do you see any simultaneous heating and venting? Make any adjustments to the model and repeat this step until you are satisfied with the base model hourly results.

Step 3 - Make Optimisation Objective Settings

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon and set the [Analysis type](#) on the Analysis tab to **2-Optimisation**.
2. Go to the [Objectives](#) tab and add 2 new objectives for comfort and heating load. You can either edit the existing objectives or first delete those and create 2 new objectives. The steps involved are described below.

Minimise Discomfort > 20% PPD

To create the "Minimise Discomfort > 20% PPD" objective, a new Summary Output (aka KPI) must first be created. Click on the **Objective KPI** to open the list of existing Summary Outputs in the Info panel. Our custom KPI won't be listed there at first, so create it now by clicking on the existing **Custom Script** Summary Output on the list of KPIs, click on the **Create copy of highlighted item** Info toolbar icon and edit the clone. Make the following settings on the Summary Output dialog.

Summary Output

General

General

Name

Category

Important Note: The name, "PPD Discomfort > 20%", entered here on the Summary Outputs dialog must match the text written to field 0 of the record added to the "ParamResultsTmp" table in the [script](#). You may like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

Save the newly created KPI and select it on the **Optimisation Objectives** dialog.

Name the objective "Minimise Discomfort > 20% PPD". That name is for your own reference only and unlike the Summary Output name, does not need to match anything else. Select the **1-Min** Min/Max option to say that you wish to minimise discomfort. The dialog should now look something like the screenshot below.

Optimisation Objectives

General

Objectives

Name **Minimise Discomfort > 20% PPD**

Min/Max

Objective KPI

Heating load

Now create the Heating load objective. This is straightforward as we can use a standard built-in KPI.

Select the **Heating load** KPI and call the objective "Heating load". Select the **1-Min** Min/Max option to say that you wish to minimise it. The objectives dialog should now look like the screenshot below.

Optimisation Objectives

General

Objectives

Name

Min/Max

Objective KPI

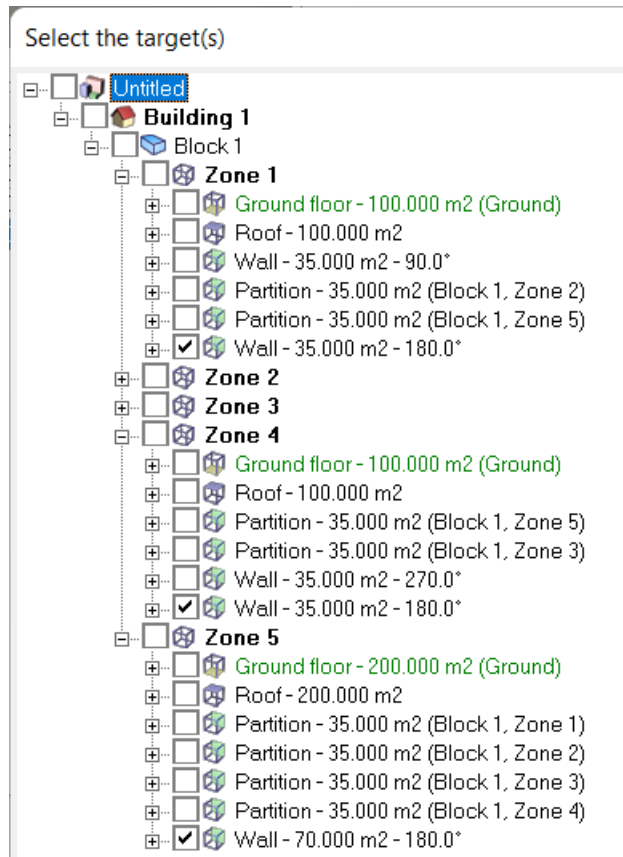
With both objectives defined, the Objectives tab should now be set up like this:

Parametric, Optimisation and UA/SA Analysis Settings		
Analysis Type	Objectives	Additional Outputs
Name	Min/Max	Objective KPI
Minimise Discomfort > 20% PPD	1-Min	PPD Discomfort > 20%
Heating load	1-Min	Heating load

Step 4 - Make Optimisation Variable Settings

Now set up the design variables. The variables to be included are:

- Heating setpoint temperature - ranging from 20 to 22°C with an optimisation step of 0.1°C.
- Nat vent setpoint temperature - ranging from 24 to 28°C with an optimisation step of 0.1°C.
- % External window opens - ranging from 0 to 100% with an optimisation step of 1.
- Local shading with the 5 overhang lengths ranging from 0 to 2m in length. This is the default selection when choosing Local shading as the variable. The overhangs are applied only to the South facing surfaces:



With the above variables defined, the Variables tab should now look like this:

Parametric, Optimisation and UA/SA Analysis Settings							
Analysis Type	Objectives	Additional Outputs	Constraints	Design Variables			
Name	Variable type	Min Value	Max Value	Step (parametric)	Step (optimisation)	Options list	Target objects
% External window op...	% External window op...	0.00	100.00	20.00	1.000	-	Building
Nat vent set-point temp	Nat vent set-point temp	24.00	28.00	5.00	0.100	-	Building
Heating setpoint temp...	Heating set-point temp...	20.00	22.00	2.00	0.100	-	Building
Overhang S Facade	Local shading type	0.00	0.00	0.00	0.000	5 options	3 Targets Selected

Step 5 - Write Custom EMS Script To Calculate Fanger % Discomfort Area

The EMS script to be used in this example is provided below. Follow the steps below to set it up for the analysis.



1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Open the **EMS** category and click on the Add new item toolbar icon to create a new script.

4. Enter the name as "Fanger % Discomfort Area",
5. Enable the program
6. Copy the script below and paste it in the script text area.

Now check you understand how the EMS script works.

The script sets up a sensor for each occupied zone in the model to get a reading for the Fanger PPD. This is checked at the end of each zone timestep during occupied periods and compared against the threshold of 20%. If the % PPD is higher than the comfort threshold the whole zone is considered to be uncomfortable and its floor area is added to the total. The total uncomfortable floor area is divided by the total floor area of all occupied zones to give the floor-area-weighted % discomfort for the building. The timestep % discomfort outputs are averaged over the whole simulation to create a single discomfort metric which is reported as "Percentage Discomfort Area" in the SQL output file.

EMS example: Script to Calculate % Discomfort Area

```
<ForAllOccupiedZones>
EnergyManagementSystem:Sensor,
    <LoopZoneVariableName>OccSchedule_PPD,
    <LoopZoneAttribute OccupancySchedule>,
    Schedule Value;

EnergyManagementSystem:Sensor,
    PPD_<LoopZoneVariableName>,
    PEOPLE <LoopZoneIDFName>,
    Zone Thermal Comfort Fanger Model PPD;

EnergyManagementSystem:InternalVariable,
    Area_<LoopZoneVariableName>,
    <LoopZoneIDFName>,
    Zone Floor Area;
<LoopNextZone>

EnergyManagementSystem:ProgramCallingManager,
    Fanger PPD Discomfort Area,
    EndOfZoneTimestepBeforeZoneReporting,
    FangerDiscomfortArea;

EnergyManagementSystem:GlobalVariable,
    PercentageDiscomfortArea;

EnergyManagementSystem:OutputVariable,
    Percentage Discomfort Area [%],
    PercentageDiscomfortArea,
    Averaged,
    ZoneTimeStep;

EnergyManagementSystem:Program,
    FangerDiscomfortArea,
    SET SumNumerator = 0,
    SET SumDenominator = 0,
<ForAllOccupiedZones>,
    IF <LoopZoneVariableName>OccSchedule_PPD > 0.25,
        IF PPD_<LoopZoneVariableName> > 20,
            SET SumNumerator = SumNumerator + Area_<LoopZoneVariableName>,
        ENDIF,
        SET SumDenominator = SumDenominator + Area_<LoopZoneVariableName>,
    ENDIF,
<LoopNextZone>
    IF SumDenominator > 0,
```

```

    SET PercentageDiscomfortArea = (SumNumerator / SumDenominator)*100,
ENDIF;

```

```

! provide outputs based on DB output settings
<If BuildingAttribute TimesteplyOutput = 1 Then>
Output:Variable, *, Percentage Discomfort Area, timestep;
<Endif>
<If BuildingAttribute HourlyOutput = 1 Then>
Output:Variable, *, Percentage Discomfort Area, hourly;
<Endif>
<If BuildingAttribute DailyOutput = 1 Then>
Output:Variable, *, Percentage Discomfort Area, daily;
<Endif>
<If BuildingAttribute MonthlyOutput = 1 Then>
Output:Variable, *, Percentage Discomfort Area, monthly;
<Endif>
Output:Variable, *, Percentage Discomfort Area, RunPeriod;

```

Step 6 - Write Python Script For The Custom Script KPI

The next task is to write the script that will read the EMS "Percentage Discomfort Area" simulation outputs from the SQL output file, process them and pass back to DesignBuilder as a Custom KPI. Steps are as follows.



1. Click on the Scripts  toolbar icon to open the Script Manager dialog. The **Enable scripts** checkbox should already be checked.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Click on an existing script within the **Python-Script** category and press the **Add new item** Info toolbar icon to open the Edit script dialog.
4. Enter the name as "Python Script: PPD Discomfort > 20%", enable the program and copy and paste the script below into the script text area.

In summary, the script works as follows. After each simulation the SQL output file is opened and the building "Percentage Discomfort Area" EMS output is read for the RunPeriod. This floor-area-weighted % discomfort value indicates the % of time that the building was uncomfortable over the year. This value is converted to hours by dividing by 100 (converting a % to a fraction) and multiplying by 8760 (hours in the year). The value for the simulation is then written to the special "ParamResultsTmp" table together with the name of the Objective to be picked up by DesignBuilder when it processes the simulation results.

Python example: Optimisation for PPD Discomfort > 20% KPI using db_eplusout_reader

```

from db_eplusout_reader import Variable, get_results
from db_eplusout_reader.constants import *
from datetime import datetime

# ensure that SQLite outputs are generated
def before_energy_idf_generation():
    site = api_environment.Site
    for building in site.Buildings:
        building.SetAttribute("SSSQLiteOP", "1")

def after_energy_simulation():
    file = api_environment.EnergyPlusFolder + r"eplusout.sql"
    variables = Variable("EMS", "Percentage Discomfort Area", "")
    results = get_results(file, variables=variables, alike=True, frequency=RP)

    discomfort_hrs = results.scalar
    site = api_environment.Site
    table = site.GetTable("ParamResultsTmp")

```

```
record = table.AddRecord()  
record[0] = "PPD Discomfort > 20%"  
record[1] = str(discomfort_hrs)
```

Tip: To save time you can copy the above script and paste it into the dialog.

Tip: The [DesignBuilderSoftware \ db-scripts](#) Github repository includes the latest versions of the above script as well as other scripts used in DesignBuilder tutorials and examples.

Step 6 - Re-Run A Standard Simulation

Click on the Simulation tab and press update to re-run a base *annual* simulation with the **Monthly and RunPeriod** results option checked.

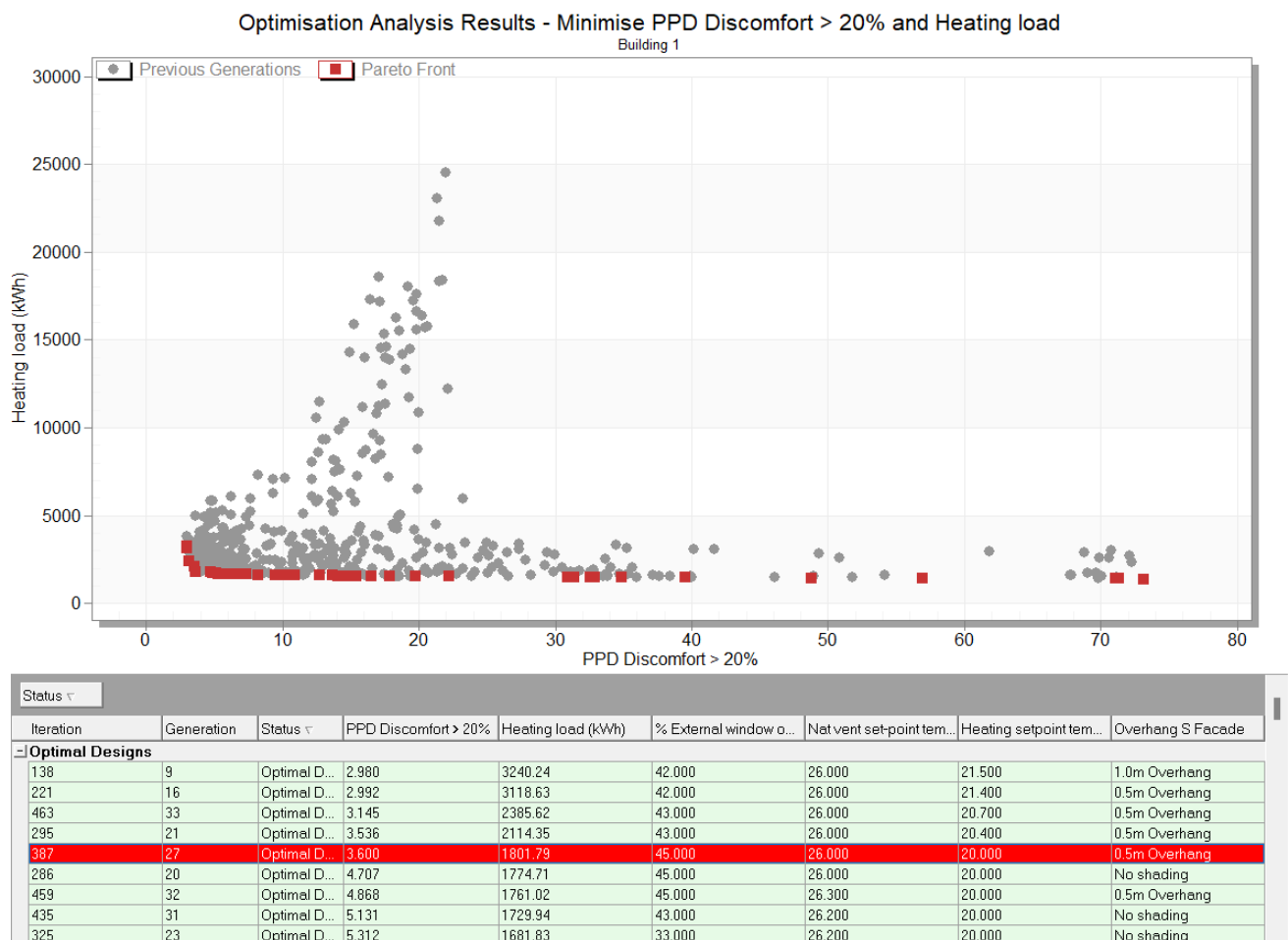
Important Note: Make sure to switch off **Daily, Hourly** and **Sub-hourly** results for this simulation to ensure that the upcoming optimisation simulations and results processing are kept as efficient as possible.

Step 7 - Run The Optimisation

Navigate to the **Optimisation + UA/SA** tab on the Simulation screen and press the **Update** toolbar icon to open the Optimisation Calculation Options dialog. You can accept the default optimisation settings for the purposes of this tutorial.

Press the **Start** button to run the optimisation simulations.

After the optimisation has converged and you have closed the Calculation Options dialog, you should see results similar to those below on the main screen. Depending on the speed of your machine and the number of cores available, it will some time before all the simulations required for the optimisation are completed.



The results show that to minimise both heating energy and PPD discomfort over the year, the following optimal variable settings are indicated:

- Heating setpoint temperature of 20°C.
- Natural ventilation setpoint temperatures of around 26°C. Natural ventilation setpoints below this increase the chance of simultaneous heating and venting and hence increase heating load for no benefit in comfort, while higher setpoints do not provide sufficient venting to maintain comfortable conditions in the summer.
- Window opening % of around 45%. Values below this do not give enough venting to keep occupants cool in summer, while higher values increase the incidence of simultaneous heating and venting.
- Overhang lengths on the South facade of 0 to 1.0m are favoured.

Optimisation Tutorial - Custom Script KPI Example 4 - Adding Python Script For TM52 Comfort To Example 3

This example extends Example 3 by adding a constraint to ensure that [CIBSE TM52](#) summertime comfort criteria are met - in addition to the "Minimise % PPD Discomfort" optimisation objective. DesignBuilder doesn't include a built-in KPI for TM52 comfort so this is added through a Python script.

The application of 2 custom KPIs is demonstrated, one for the PPD discomfort objective and the other for the TM52 constraint.

Step 1 - Switch On TM52 Outputs

Open the model saved from Example 3 and open the Model options dialog. On the Simulation tab, under the Outputs header, check the [CIBSE TM52](#) checkbox to ensure that DesignBuilder generates standard TM52

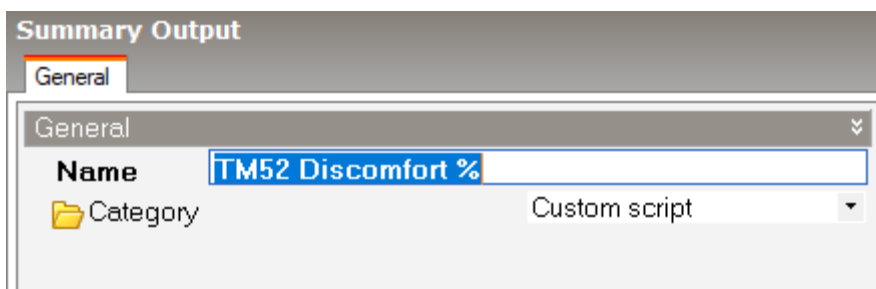
outputs. These standard outputs will need further processing to create a single discomfort value for the whole building and we'll write a Python script to do that in Step 3.

Step 2 - Add Constraint Setting

1. Open the **Parametric Settings** dialog by clicking on the  toolbar icon.
2. Go to the **Constraints** tab and add 1 new constraint for TM52 discomfort. The steps involved are described below.

TM52 Discomfort

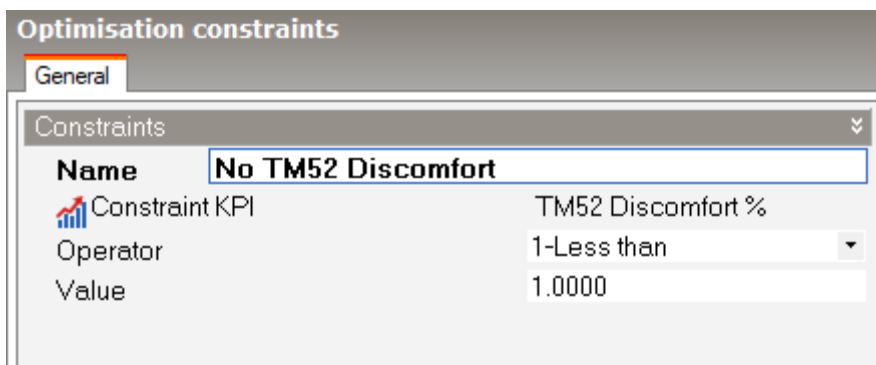
To create the TM52 comfort constraint, a new Summary Output (aka KPI) must first be created. Click on the **Output KPI** to open the list of existing Summary Outputs in the Info panel. Our new custom KPI won't be listed there at first, so create it now by clicking on the existing **Custom Script** Summary Output on the list of KPIs, click on the **Create copy of highlighted item** Info toolbar icon and edit the clone. Make the following settings on the Summary Output dialog.



Important Note: The name, "TM52 Discomfort %", entered here on the Summary Outputs dialog must match the text written to field 0 of the record added to the "ParamResultsTmp" table in the [script](#). You may like to copy the name and paste it somewhere safe where you can easily access it later when writing the script.

Save the newly created KPI and select it on the **Optimisation Constraints** dialog.

Set the maximum value to 1 so the dialog should look as follows:



Step 3 - Write Python Script For The Custom Script KPI

The next task is to write the script that will read the base TM52 simulation outputs from the SQL output file, process them and pass back to DesignBuilder as a Custom KPI. Steps are as follows.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.

3. Click on the **Python-Script** category and press the **Add new item** Info toolbar icon to open the Edit script dialog.
4. Check the **Enable program** option.
5. Name the script "TM52 Discomfort".
6. Copy and paste the script below into the script window.

The script reads the TM52 Criteria 1, 2, and 3 outputs and checks against the corresponding criteria based on [TM52 rules](#). Any zone passing the overall annual TM52 criteria has its area excluded from the total area failing. The TM52 Discomfort % value for the simulation is then added to the special "ParamResultsTmp" table together with the name of the Objective to be picked up by DesignBuilder when it processes the simulation results.

The 2 Summary outputs, "TM52 Discomfort %" and "PPD Discomfort > 20%" are each added as separate records to the "ParamResultsTmp" table. More KPIs could be added to the table in a more sophisticated analysis.

Python example: TM52 KPI for Optimisation using db_eplusout_reader

```

"""
Use CIBSE TM52 results as optimisation KPI.
Required model inputs:
- Enable TM52 outputs
- Add a "Custom Script" KPI, use "TM52 Discomfort %" name
- Run a "Standard" simulation to make sure that TM52 outputs are present
The script calculates a percentage of failing TM52 relevant zones.
"""

import ctypes
import os

from db_eplusout_reader import Variable, get_results
from db_eplusout_reader.constants import *
from db_eplusout_reader.exceptions import NoResults

threshold_crit1 = 3
threshold_crit2 = 6
threshold_crit3 = 0

def show_message(title, text):
    ctypes.windll.user32.MessageBoxW(0, text, title, 0)

def zone_fails_tm52(zone_name):
    """Verify if given zone meets tm52 criteria."""

    outputs_path = os.path.join(api_environment.EnergyPlusFolder, "eplusout.sql")
    variable1 = Variable("EMS", "CIBSE TM52 Criterion 1 {}".format(zone_name), "%")
    variable2 = Variable("EMS", "CIBSE TM52 Criterion 2 {}".format(zone_name), "C")
    variable3 = Variable("EMS", "CIBSE TM52 Criterion 3 {}".format(zone_name), "hr")

    criterion_1_results = get_results(outputs_path, variable1, RP)
    criterion_2_results = get_results(outputs_path, variable2, D)
    criterion_3_results = get_results(outputs_path, variable3, RP)

    criterion_1 = criterion_1_results.scalar
    crit2_series = criterion_2_results.first_array
    criterion_2 = max(crit2_series)
    criterion_3 = criterion_3_results.scalar

    counter = 0

```

```

if criterion_1 > threshold_crit1:
    counter += 1
if criterion_2 > threshold_crit2:
    counter += 1
if criterion_3 > threshold_crit3:
    counter += 1

return counter >= 2

def after_energy_simulation():
    site = api_environment.Site
    fail_area = 0
    total_area = 0 # area only includes TM52 zones
    for block in active_building.BuildingBlocks:
        for zone in block.Zones:
            zone_name = zone.GetAttribute("SSEPObjectNameInOP")
            try:
                if zone_fails_tm52(zone_name):
                    fail_area += zone.FloorArea
                    total_area += zone.FloorArea
            except NoResults:
                pass
        try:
            percentage_area_fail = (fail_area / total_area) * 100
        except ZeroDivisionError:
            raise NoResults("No TM52 results found!")

    # show_message("Results TM52", "{}% fails, total_area {}m2, fail_area: {}m2".format(percentage_area_fail, total_area, fail_area))

    table = site.GetTable("ParamResultsTmp")

    record = table.AddRecord()
    record[0] = "TM52 Discomfort %"
    record[1] = str(percentage_area_fail)

```

Tip: To save time you can copy the above script and paste it into the dialog.

Tip: The [DesignBuilderSoftware \ db-scripts](#) Github repository includes the latest versions of the above script as well as other scripts used in DesignBuilder tutorials and examples.

Step 6 - Re-Run A Standard Simulation

Click on the Simulation tab and press update to re-run a base annual simulation with the **Monthly and RunPeriod** results option checked.

Important Note: Make sure to switch off **Daily**, **Hourly** and **Sub-hourly** results for this simulation to ensure that the upcoming optimisation simulations and results processing are done efficiently.

Note: The script above is shown with the **show_message** line commented out, but the message can be enabled by removing the "#" character to display TM52 compliance results. Once you are confident that the TM52 script is working well you can delete or comment out the **show_message** line to avoid seeing messages after each simulation during the optimisation.

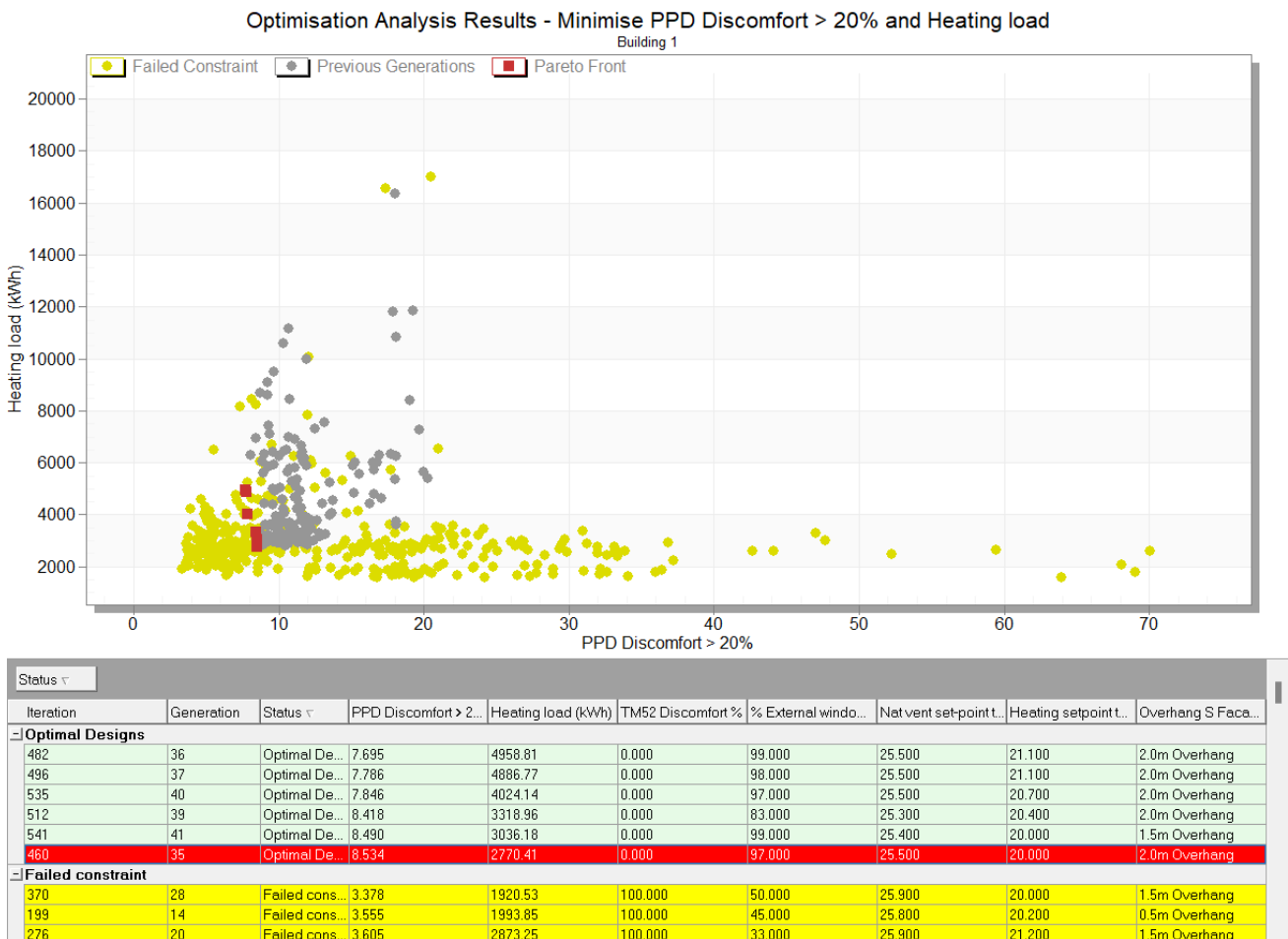
Step 7 - Run The Optimisation

Navigate to the **Optimisation + UA/SA** tab on the Simulation screen and press the **Update** toolbar icon to open the Optimisation Calculation Options dialog. You can accept the default optimisation settings.

Press the **Start** button to run the optimisation simulations.

Tip: When running constrained optimisations it is usually necessary to test more iterations to find optimal solutions that meet the combined requirements of the objectives and the constraint(s). This is because while the fitness of designs is partly assessed based on whether constraints are satisfied or not, a stronger driver for the selection of new generations is the fitness based on the objectives.

After the optimisation has converged and you have closed the Calculation Options dialog, you should see results similar to those below on the main screen.



The results show that in order to simultaneously minimise both PPD discomfort and heating load while also passing the requirements of TM52, the optimal settings are:

- Heating setpoint temperature close to 20°C.
- Natural ventilation setpoint temperatures close to 25.5°C. Natural ventilation setpoints below this cause increased simultaneous heating and venting and hence increase heating load for no benefit in comfort, while setpoints above this range do not provide sufficient venting in the summer to meet the TM52 constraint.
- Window opening % of 100% are required to meet the TM52 constraint. Values below this do not give enough venting to keep occupants cool in summer and pass TM52 summer discomfort requirements.

Comparing these results with equivalent results from Example 3 without the TM52 constraint, we can see that the need to pass TM52 requires additional natural ventilation, both in terms of the natural ventilation setpoint

temperature and window opening area. While 50% window opening area and a 26-27°C natural ventilation setpoint is enough to keep most of the building comfortable most of the time based on PPD alone, adding the "pass TM52 constraint" requires much more ventilation, which in turn involves more simultaneous heating and venting during swing seasons and so heating load is significantly higher.

A more sophisticated set of system control rules would need to be introduced to avoid the simultaneous heating and venting that is occurring in the "optimal" solutions listed above, allowing the heating load to be reduced while still passing TM52. For example adding controls to limit the window % opening area based on the outside and inside temperatures would help reduce simultaneous heating and venting. EMS code could be added to achieve this.

Uncertainty And Sensitivity Analysis

Overview

Building performance simulation is a highly complex process. Some or all of the model inputs are often best available estimates and simplifications that do not necessarily represent reality completely. These uncertainties in input variables can be due to:

1. Lack of knowledge about the building details in various stages of design process.
2. Spread in the manufacture and assembly quality of the building components.
3. Unpredictable behaviour of the building occupants.
4. Uncertainty in weather prediction under changing climate and uncertain weather data information due to the use of synthetic weather data.

This uncertainty in the inputs imposes a limit on our confidence in the output of the model, i.e. results of the simulation model are potentially erroneous. Such errors are quantified in uncertainty analysis which treats the simulation output as "probabilistic" rather than "deterministic".

A related field to uncertainty analysis is sensitivity analysis. While, uncertainty analysis is a method to quantify the variability of a model output due to uncertainty in the one or more input variables, sensitivity analysis is the study of how the uncertainty in each simulation output can be apportioned to various sources of uncertainty in its inputs.

Good modelling practice requires the modeller to evaluate their confidence in the model. This requires, first uncertainty analysis, a quantification of the uncertainty in any model results; and second sensitivity analysis, an evaluation of how much each input is contributing to the output uncertainty. Sensitivity analysis orders by importance, the strength and relevance of the inputs in determining the variation in the output.

The process of recalculating outcomes under alternative model input assumptions to determine the impact of a variable under sensitivity analysis can be useful for a range of purposes including:

1. Testing the robustness of the results in the presence of uncertainty.
2. Increasing understanding of the relationships between input and output variables.
3. Uncertainty reduction, through the identification of model inputs that cause significant uncertainty in the output and should therefore be the focus of attention.
4. Model simplification – fixing model inputs that have no effect on the output or identifying redundant parts of the model structure.
5. Enhancing communication from modellers to decision makers (e.g. by making recommendations more credible, understandable, compelling or persuasive).
6. Supporting a [design optimisation](#) process by narrowing the search space and number of factors to only the most important ones.
7. Calibrating models, a primary sensitivity test can ease the calibration stage by focusing on the most sensitive parameters.

Core Methodology For Uncertainty And Sensitivity Analysis

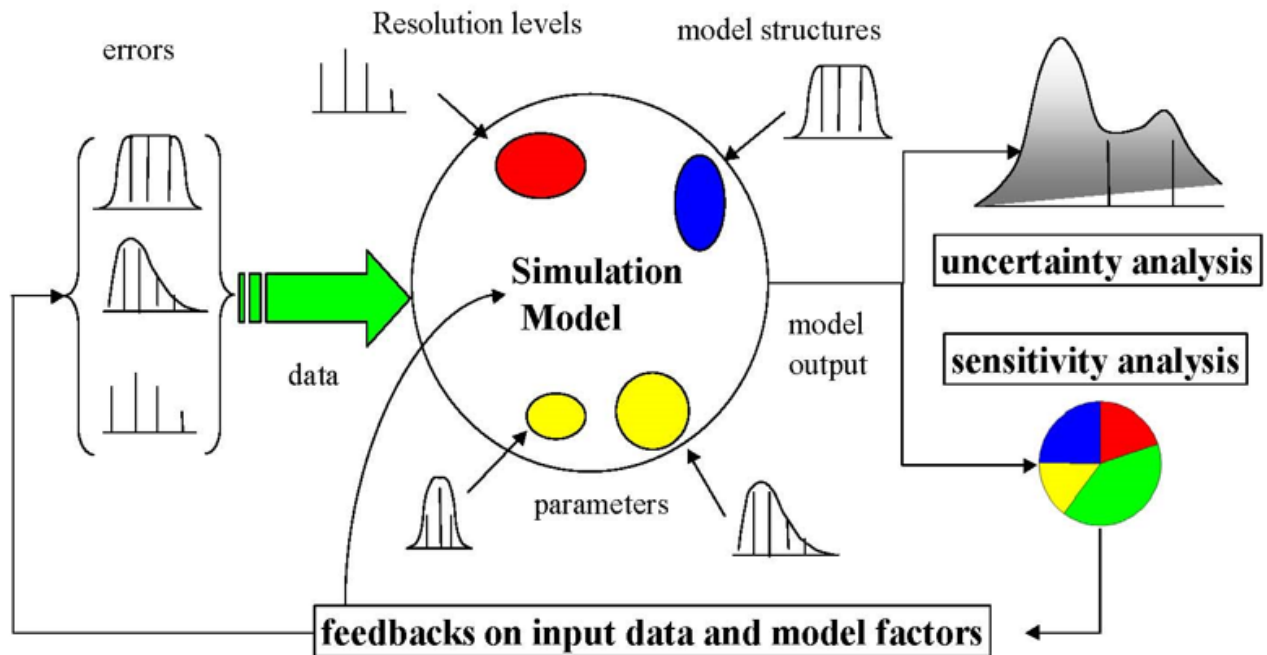
Step 1: Quantify the uncertainty in each input (e.g. ranges, probability distributions). Note that doing this rigorously this can be difficult, and many methods exist to elicit uncertainty distributions from subjective data.

Step 2: Identify the model output to be analysed (the target of interest should ideally have a direct relation to the problem tackled by the model).

Step 3: Run the model several times using appropriate sampling methods and input uncertainties.

Step 4: Using the resulting model outputs to calculate the sensitivity measures of interest.

The figure below shows an ideal scheme of a sampling-based sensitivity analysis. Uncertainty arising from various sources – errors in the data, parameter estimation procedure, alternative model structures – are propagated through the model for uncertainty analysis and their relative importance is quantified via sensitivity analysis.



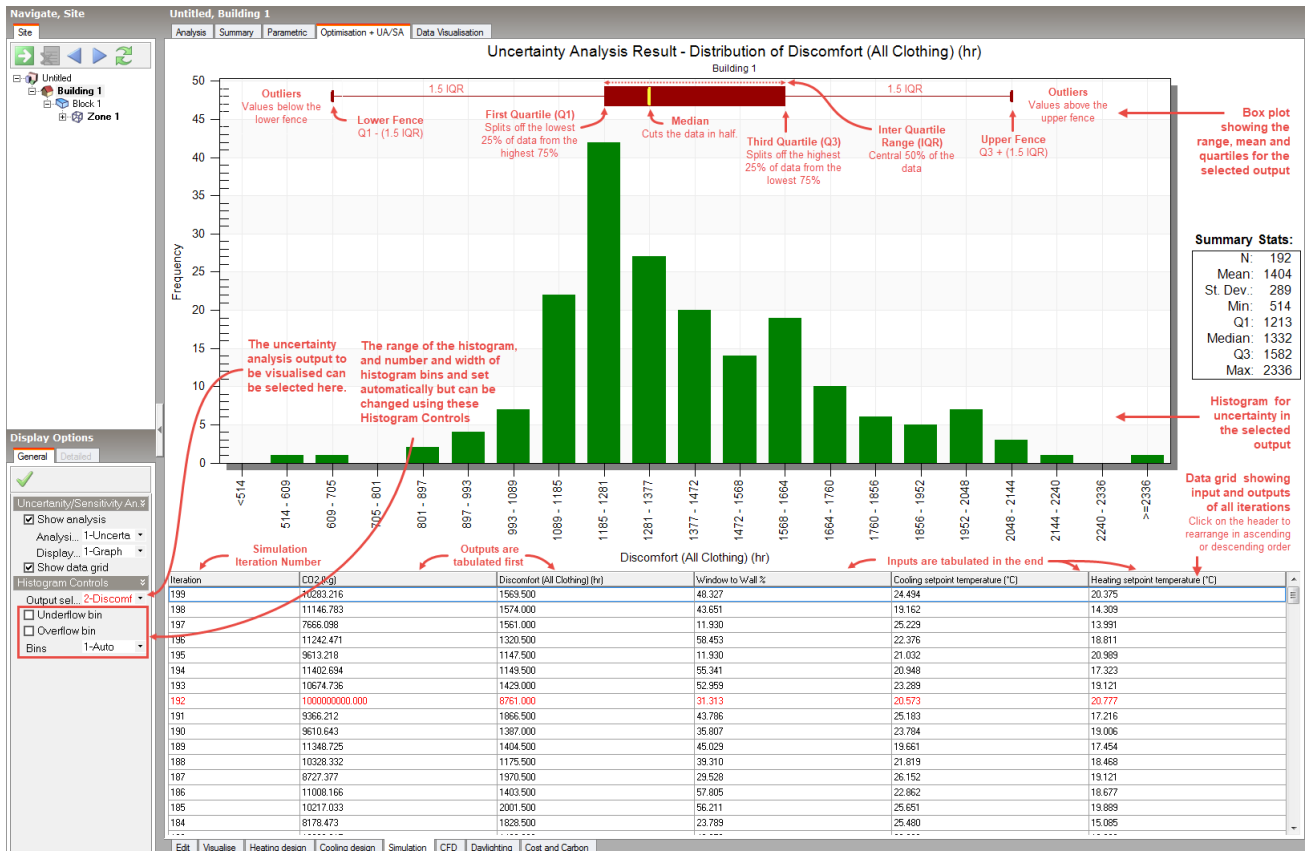
DesignBuilder Implementation

In DesignBuilder Uncertainty and Sensitivity Analysis, various sampling methods and a broad range of probabilistic distributions are supported for the input parameters. Statistics are performed on the simulation results of the sampled inputs to give uncertainty estimations. The Sensitivity Analysis method implemented in DesignBuilder is [regression analysis](#). In DesignBuilder, any number of design variables can be included in the analysis for each of the selected outputs. While we can do uncertainty and sensitivity analysis using multiple inputs and multiple outputs, each output analysis is, in fact, an independent analysis.

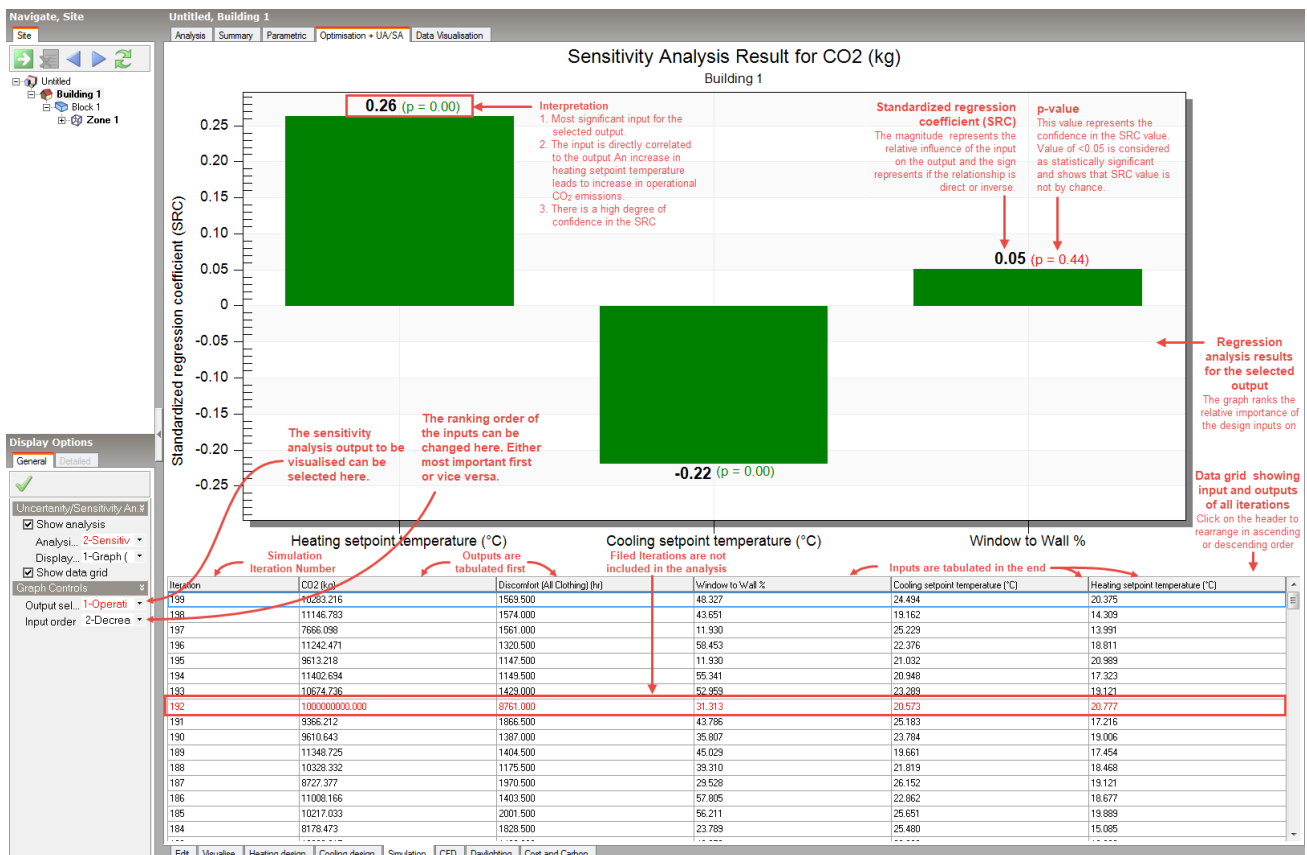
For example, an uncertainty and sensitivity analysis study for total building carbon emissions might involve a base design which is analysed for varying wall and roof construction, glazing amount and type, degrees of shading and HVAC systems. The uncertainty analysis results might be displayed graphically in the form of a histogram showing the variation in total carbon emission. The sensitivity analysis results for the output (total carbon emission) can be represented graphically in the form of a bar chart, in which each bar represents each input variable. The magnitude of the bar tells the relative importance of each variable. The longest bar represents the most important/sensitive input for the selected output. The sign of the bar tells us whether the relationship between the input and the output is direct or indirect. A positive value means that increase in the value of that input variable leads to an increase in the output result and vice-versa.

Tip: A good starting point to learn more about Uncertainty and Sensitivity Analysis is to follow through the Uncertainty and Sensitivity Analysis [Basic](#) and [Advanced](#) Tutorials.

Example Uncertainty Analysis Result:



Example Sensitivity Analysis Result:



To get started with an Uncertainty and Sensitivity Analysis go to the Simulation screen and click on the Optimisation + UA/SA Analysis tab. From the Simulation Optimisation + UA/SA screen, you can:

- Edit the [Uncertainty and Sensitivity Analysis Settings](#) ([Output](#) and [Design Input Variables](#)). This is a required first step before running the first analysis.
- Update the Uncertainty and Sensitivity Analysis which opens the [Uncertainty and Sensitivity Analysis Calculation Options](#) Dialog.
- View [Uncertainty](#) and [Sensitivity Analysis results](#).

Uncertainty And Sensitivity Analysis Contents

- "Uncertainty and Sensitivity Analysis - Parametric Settings Dialog" on page 1786
- "Uncertainty and Sensitivity Analysis - Calculation Options Dialog" on page 1848
- "Uncertainty and Sensitivity Results" on page 1853
- "Uncertainty and Sensitivity Analysis - Tips and Tricks" on page 1867
- "Uncertainty and Sensitivity Analysis - Basic Tutorial" on page 1868
- "Uncertainty and Sensitivity Analysis - Advanced Tutorial" on page 1878

See also Wiki pages:

- [Uncertainties in building design and building energy assessment](#)
- [Sensitivity analysis of an EnergyPlus model](#)
- [Sensitivity analysis](#)

Uncertainty And Sensitivity Analysis - Parametric Settings Dialog

The **Parametric Settings** dialog can be accessed by clicking on the **Parametric settings** icon  on the toolbar.

You can use this dialog to set up the Uncertainty and Sensitivity Analysis by entering data on the tabs described below:

- [Analysis Type](#)
- [Outputs](#)
- [Design Variables](#)

Analysis Type Tab

There are 3 types of analysis and these can be set from the first tab on this dialog.

Analysis

The analysis to select for UA/SA is **3-Uncertainty/Sensitivity Analysis**.

Outputs Tab

Data on the [Outputs Tab](#) is the summary of outputs which are requested for the uncertainty/sensitivity analysis. On this tab uncertainty/sensitivity analysis outputs can be Added, Edited or Deleted by selecting the appropriate option on the right-hand info panel.

The Data shown in the table is:

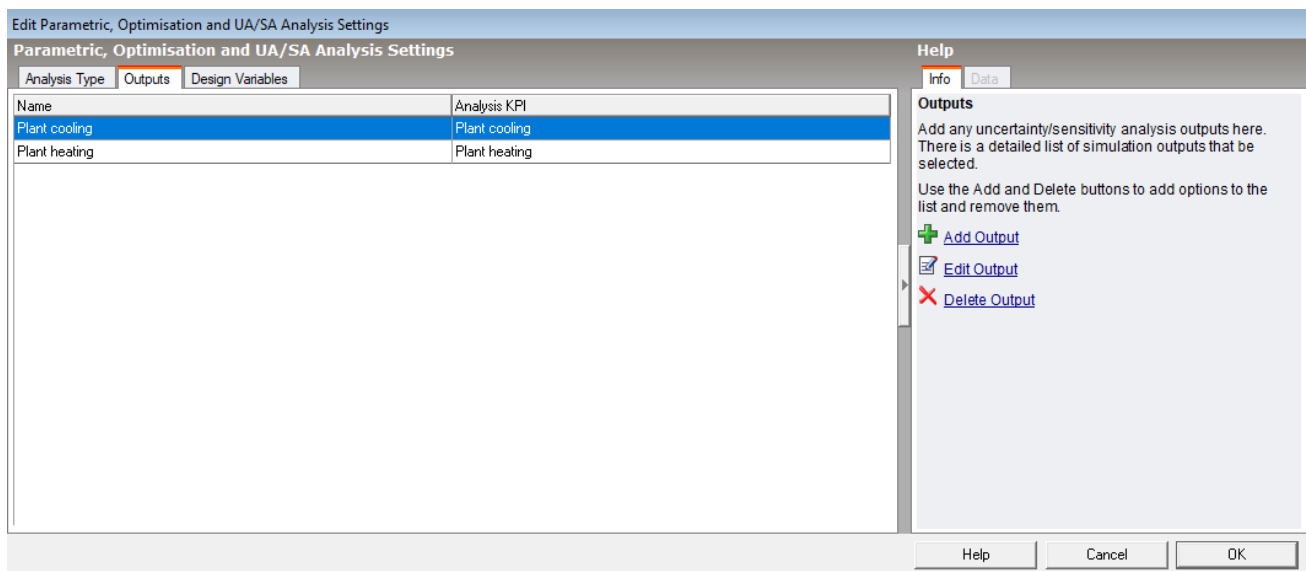
- **Name:** User Defined name for the output.
- **Analysis KPI:** The pre-defined output result option. These outputs are Building Level results.

Note: Each output needs to have a unique name and must have a unique Analysis KPI

Controls on the Info panel are:

- **Add Output:** Creates a new output (using the [Edit Analysis Outputs](#) dialog)
- **Edit Output:** Edits the selected existing output from the table (using the [Edit Analysis Outputs](#) dialog)
- **Delete Output:** Deletes the selected existing output from the table

Tip: The **Add Output/Edit Output** option should be selected with care as it is not possible to cancel or revert to the old settings in the [Edit Analysis Outputs](#) dialog. The settings will either be saved as a new entry if **Add Output** is clicked or overwritten on the selected existing output if **Edit Output** is clicked.



Design Variables Tab

Data on the [Design Variables](#) Tab is the summary of the uncertain (variable) design inputs which are selected for the uncertainty/sensitivity analysis. On this tab uncertainty/sensitivity analysis Design Variables can be Added, Edited or Deleted by selecting the appropriate option on the right-side info panel.

Any number of variables can be added.

The Data shown in the table is:

- **Name:** User defined name for the Design Variable.
- **Variable Type:** Type of input Design Variable (Setpoint, Load, WWR etc.)
- **Distribution Category:** Type of distribution curve selected, Discrete or Continuous
- **Distribution Curve:** Name of distribution (Normal, Uniform, etc.)
- **Distribution Summary:** Details of parameters that control the distribution (Mean, Std Dev etc.)
- **Target Objects:** Scope of the variable suggesting the values are changed for the whole building or selected zones or elements.

Note: Each Design Variable must have a unique name

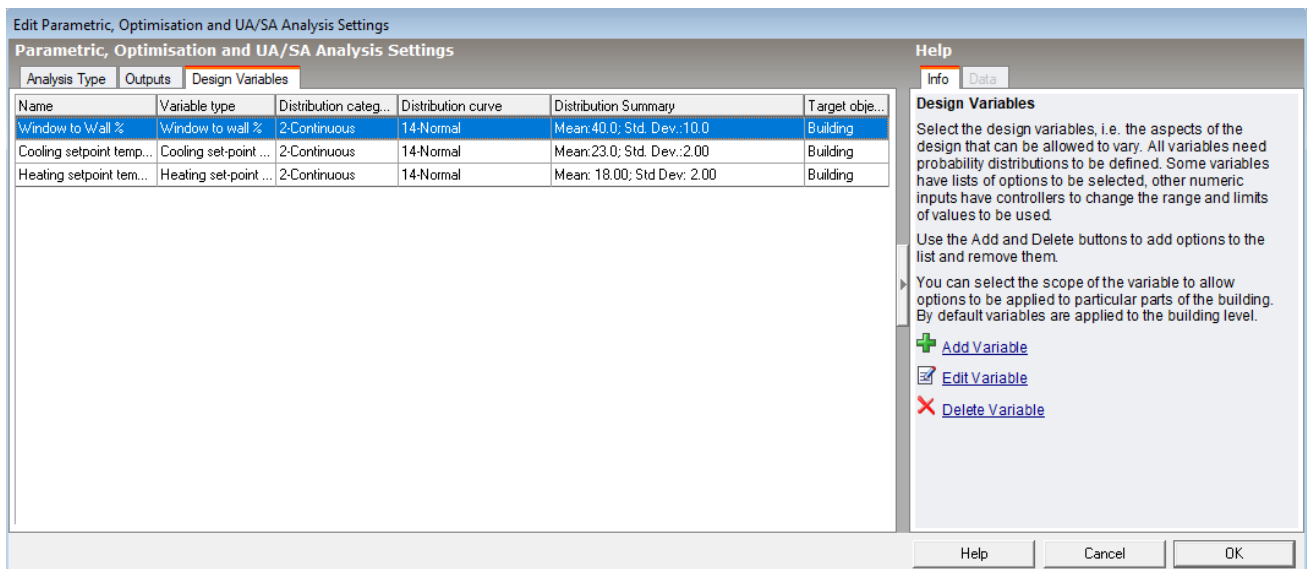
Note: In some projects, it may be necessary to set up a design variable that changes an attribute that is also changed by another template type variable. In such cases you should ensure that the template variable is defined first, i.e. it appears higher in the variables list. For example, if the Design variables list includes parametric changes for both Construction templates and Infiltration, the Construction template

variable should be higher on the list than the Infiltration variable for the infiltration variable to have any effect on the analysis.

Controls on the Info panel are:

- **Add Variable:** Creates a new Design Variable (using [Edit Analysis Variable](#) dialog).
- **Edit Variable:** Edits the selected existing Design Variable from the table (using [Edit Analysis Variable](#) dialog).
- **Delete Variable:** Deletes the selected existing Design Variable from the table.

Tip: The **Add Variable/Edit variable** options should be used with care as it is not possible to cancel or revert to the old settings in the [Edit Analysis Variables](#) dialog. The settings will be saved as a new entry if **Add Variable** is clicked or overwritten on the selected existing Design Variable if **Edit Variable** is clicked.

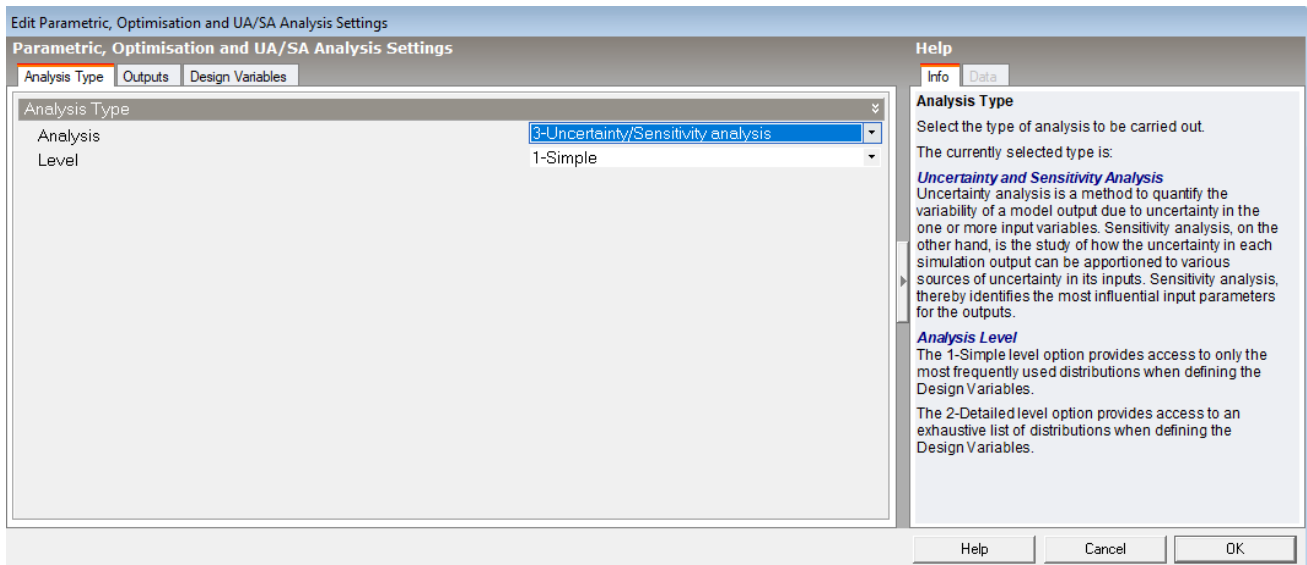


Note: If an Uncertainty/Sensitivity Analysis result already exists, then saving any new changes in the Parametric Settings dialog will delete the existing results.

Uncertainty And Sensitivity Analysis - Analysis Type

Analysis Type

Data on the [Analysis Type](#) tab allows you to define the analysis and its level of complexity.



Analysis

The following options are available:

- **1-Parametric analysis** - to run a parametric analysis with up to 2 [design variables](#) and any number of [Output KPIs](#) on the Y-axis. See the specific section on [Optimisation and Parametric analysis](#) for more details on this option.
- **2-Optimisation** - to run an optimisation with 1 or 2 design objectives and any number of [design variables](#), [constraints](#) and [additional outputs](#). See the specific section on [Optimisation and Parametric analysis](#) for more details on this option.
- **3-Uncertainty/Sensitivity analysis** - to run an uncertainty and sensitivity analysis (UA/SA) with any number of [outputs](#) and [design variables](#).

Note: Optimisation and UA/SA results and simulation tools are accessed from the **Optimisation + UA/SA** tab on the Simulation screen, while Parametric analysis results and simulation tools are accessed from the **Parametric** tab.

Level

The Level setting is only available for Uncertainty and Sensitivity Analysis and can be used to set the level of complexity of the problem definition using one of **1-Simple** or **2-Detailed**. The option selected may depend on the problem at hand and on your level of experience with statistical methods.

1-Simple

The **1-Simple** level option provides access to only the most frequently used variable distributions when defining the [Design Variables](#).

Distribution Category	Distribution Curve
Discrete	• 3-Binomial • 20-Uniform
Continuous	• 12-Log-Normal • 14-Normal • 18-Triangular • 19-Uniform

2-Detailed

The **2-Detailed** level option provides access to an exhaustive list of variable distributions when defining the [Design Variables](#).

Distribution Category	Distribution Curve
Discrete	• 1-Bernoulli* • 3-Binomial • 9-Geometric* • 10-Hyper-Geometric* • 13-Negative Binomial* • 16-Poisson* • 20-Uniform
Continuous	• 2-Beta • 4-Cauchy • 5-Chi-Squared • 6-Exponential • 7-Fischer F • 8-Gamma • 11-Levy • 12-Log-Normal • 14-Normal • 15-Pareto • 17-Student T • 18-Triangular • 19-Uniform • 21-Weibull

* Only available for List Type Variables

Note: The additional distribution curves available with the detailed option require advanced statistical understanding to manage the curves. **The Detailed option should therefore be used by advanced users only.**

Uncertainty And Sensitivity Analysis - Outputs Dialog

Accessed from [Parametric Settings](#) dialog

Outputs

The Outputs dialog can be accessed from the Parametric Settings dialog, when **3-Uncertainty/ Sensitivity Analysis** is selected on the Analysis Type Tab, after pressing **Add Output** or **Edit Output**.

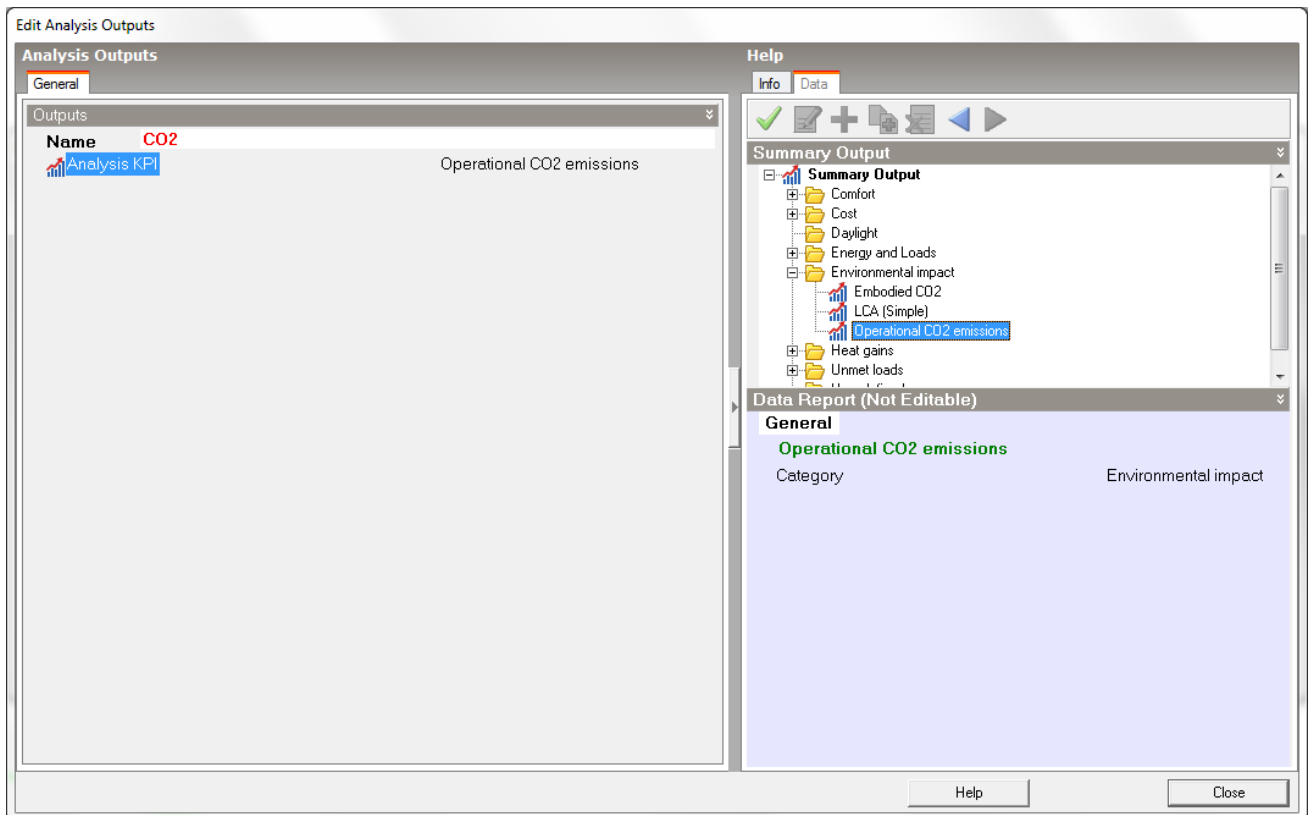
Name

User defined name for the outputs, e.g. "ASHRAE 55 discomfort hours"

Analysis KPI

The pre-defined output result option. These outputs are Building Level results.

Select a predefined or custom [Key Performance Indicator \(KPI\)](#) from the list.



Uncertainty And Sensitivity Analysis - Design Variables Dialog

Accessed from [Parametric Settings](#) dialog

Design Variables

The Design variables dialog can be accessed from the [Parametric Settings](#) dialog, when **3-Uncertainty/ Sensitivity Analysis** is selected on the Analysis Type Tab, after pressing **Add Output** or **Edit Output**.

Name

User defined name for the Design Variable e.g. "Window Wall %"

Variable type

Type of input Design Variable, to be selected from the browse list in the right-hand info panel.

You can view the full list of variable types on the "Optimisation - Design Variables Dialog" on page 1710 section.

Note: All variables are listed for selection regardless of the state of the model so care must be taken to select variables that are appropriate. For example, if the Scheduled natural ventilation model option is used then changes in the **% External window opens** variable (which only applies to Calculated natural ventilation) won't affect results.

Distribution Category

Type of probability distribution. The category can be selected from the drop-down list. The option selected here (along with analysis level on the Analysis Type tab and Variable Type above) determines the distributions curves available. There are two options:

- 1-Discrete: Available for list type variables and numeric variables.
- 2-Continuous: Available for numeric variables only.

Note: The number of options available for distribution are also controlled by Level selected on Analysis Type Tab.

View the full explanation of Discrete and Continuous categories and their availability in simple and detailed settings.

Discrete Distributions: Distribution of the input variable that can take up only finite (or countably infinite) set of values.

Note: When a list type variable is used to modify a single numeric setting within a set of components (e.g. U-value for constructions or glazing) then it is important that the numeric values are evenly spaced in each of the components in the list to ensure meaningful regression sensitivity analysis results. For example if you wish to analyse the effect of changing wall U-value between 0.1 and 0.5 then it is recommended to use a *fixed increment* in U-value such as 0.1 or 0.05.

Continuous Distributions: Distribution of a random variable that may take any value in a finite or infinite set of values.

Note: If the input variable is a list type one, then it will have a discrete probability distribution. e.g. Constructions assembly options for Wall. If the variable is a numeric type one, it generally will be defined as a continuous distribution, e.g. load power densities or system CoP. However, discrete distribution can also be used for the numeric type variables e.g. Temperature setpoints limited to a discrete increment of 0.5 °C.

The table below lists the various discrete and continuous probability distribution curves available for numeric and list type variables for simple and detailed analysis levels.

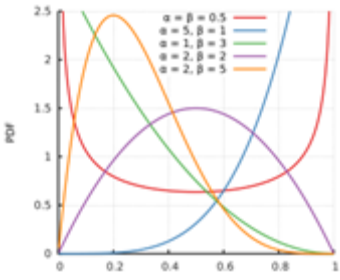
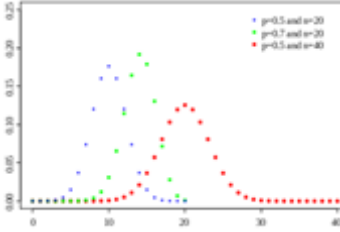
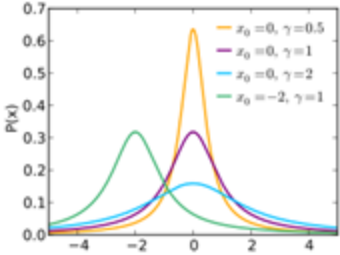
Analysis Level	Variable Type: Numeric	Variable Type: List
Simple	Discrete Distributions: • 3-Binomial • 20-Uniform (discrete)	Discrete Distributions: • 3-Binomial • 20-Uniform (discrete)
	Continuous Distributions: • 12-Log-Normal • 14-Normal • 18-Triangular • 19-Uniform	Continuous Distributions: Not Applicable
Detailed	Discrete Distributions: • 3-Binomial • 20-Uniform	Discrete Distributions: • 1-Bernoulli • 3-Binomial • 9-Geometric • 10-Hyper-Geometric • 13-Negative Binomial • 16-Poisson

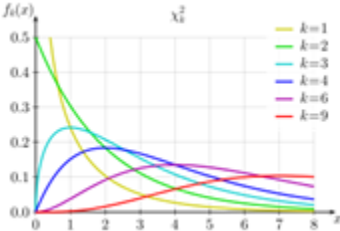
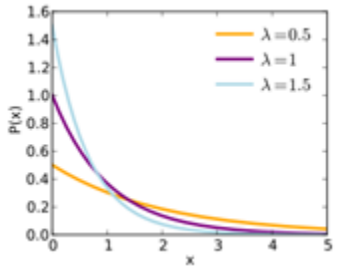
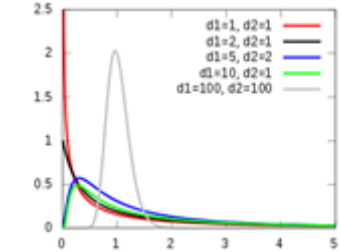
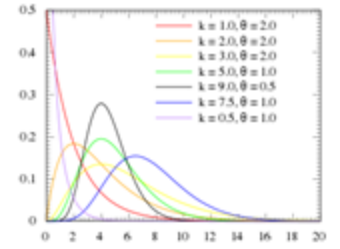
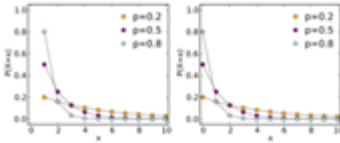
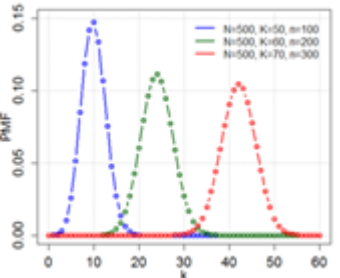
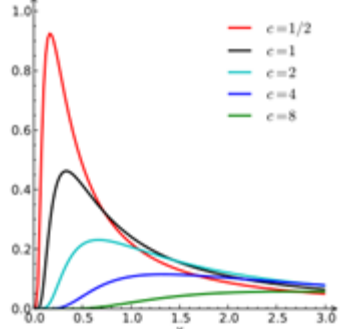
		20-Uniform
	2-Beta 4-Cauchy 5-Chi-Squared 6-Exponential 7-Fischer F 8-Gamma 11-Levy 12-Log-Normal 14-Normal 15-Pareto 17-Student T 18-Triangular 19-Uniform 21-Weibull	Continuous Distributions: Not Applicable

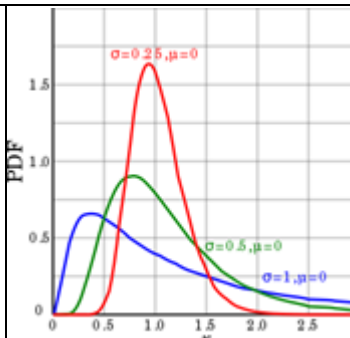
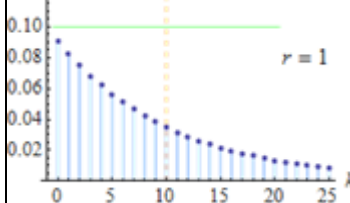
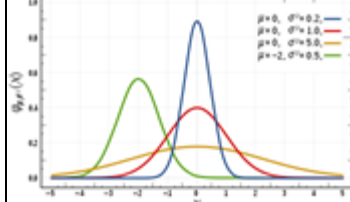
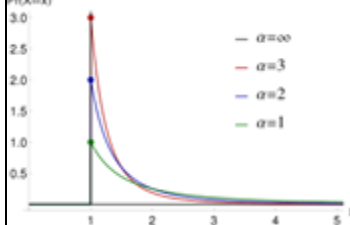
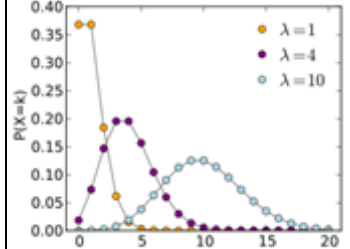
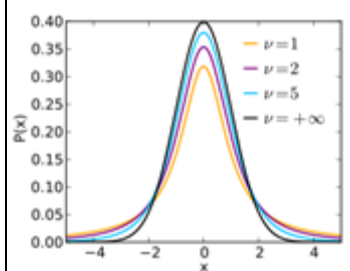
Distribution Curve

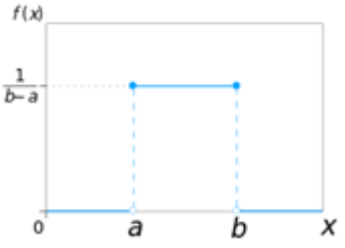
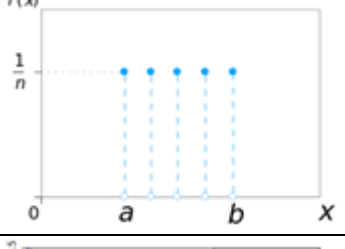
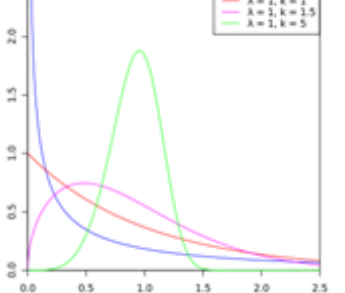
The statistical function that describes the possible values and their likelihoods for the Design Variable. Select the distribution Curve from the drop-down list.

View the full list of all distribution curves: and links to Wikipedia Pages

S.No.	Distribution (hyperlink to Wiki pages)	Category (Discrete / Continuous)	Analysis Level (Simple / Detailed)	Sample Image	Controlling Parameters
1	Bernoulli	Discrete	Detailed	NA	Probability of Second option (Can only have 2 variable value options)
2	Beta	Continuous	Detailed		Left shape parameter (Alpha) Right shape parameter (Beta)
3	Binomial	Discrete	Both		Number of Trials (n) Probability (p)
4	Cauchy	Continuous	Detailed		Scale Parameter Location (Peak Value)

5	Chi Squared	Continuous	Detailed		Degree of Freedom
6	Exponential	Continuous	Detailed		Mean
7	Fisher F	Continuous	Detailed		Degree of Freedom (n) Degree of Freedom(d)
8	Gamma	Continuous	Detailed		Shape parameter Scale parameter
9	Geometric	Discrete	Detailed		Probability (p)
10	Hypergeometric	Discrete	Detailed		Population size Population Number Sample Size
11	Levy	Continuous	Detailed		Min Val (Location Parameter) Scale parameter

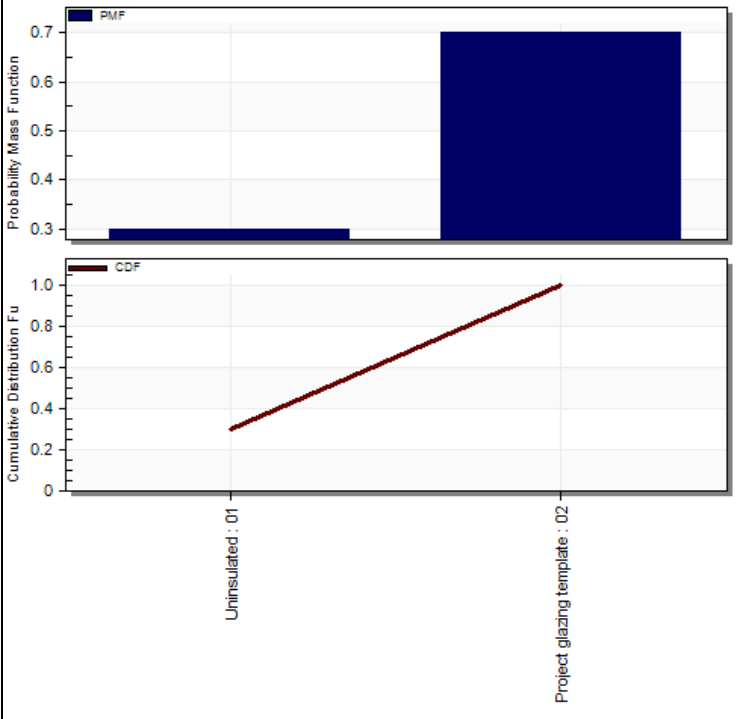
12	Log Normal	Continuous	Both		Mean Standard Deviation
13	Negative Binomial	Discrete	Detailed		Number of success Probability
14	Normal	Continuous	Both		Mean Standard Deviation
15	Pareto	Continuous	Detailed		Shape parameter Scale parameter
16	Poisson	Discrete	Detailed		Mean
17	Student T	Continuous	Detailed		Degree of Freedom (n)
18	Triangular	Continuous	Both		Location (Peak Value) Min Val Max Val

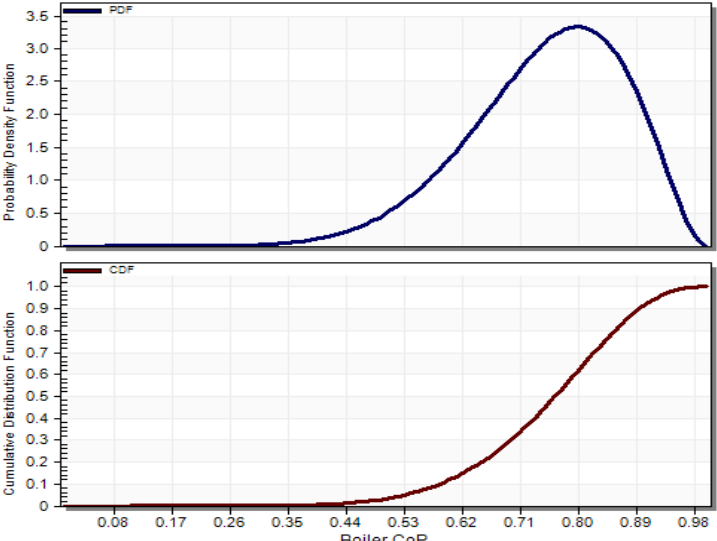
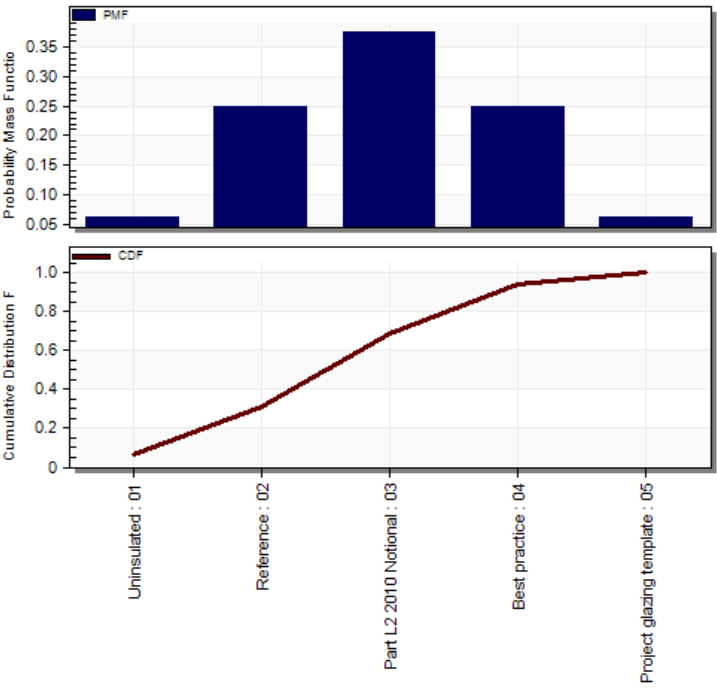
19	Uniform (Continuous)	Continuous	Both		Min Val Max Val
20	Uniform (Discrete)	Discrete	Both		Min Val Step Size Number of Steps
21	Weibull	Continuous	Detailed		Shape parameter Scale parameter

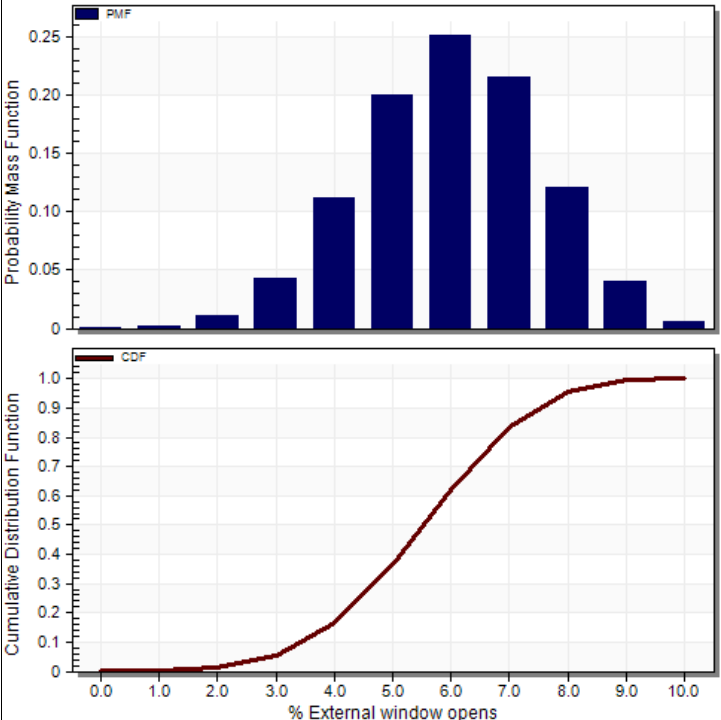
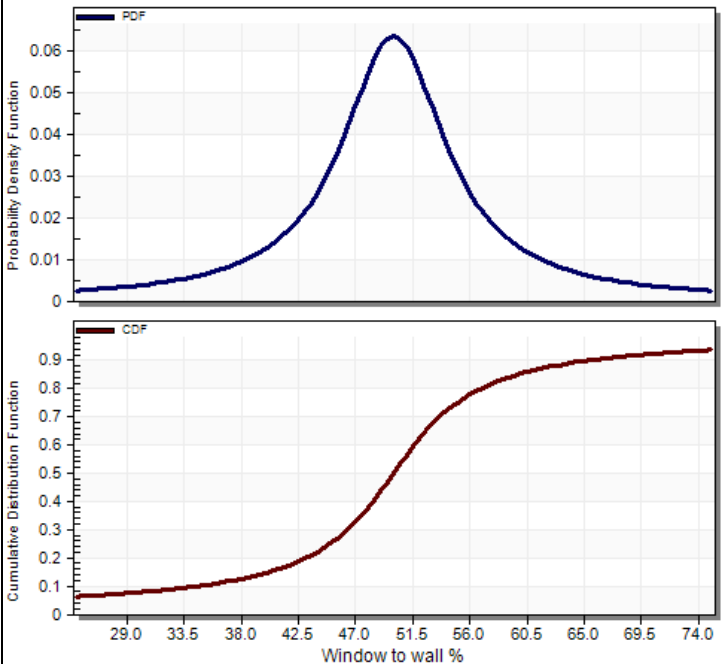
Distribution Parameters

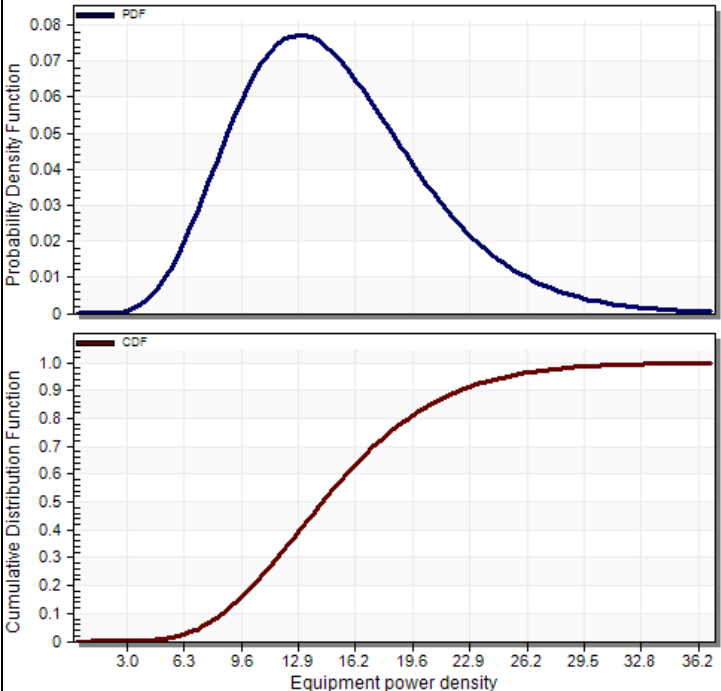
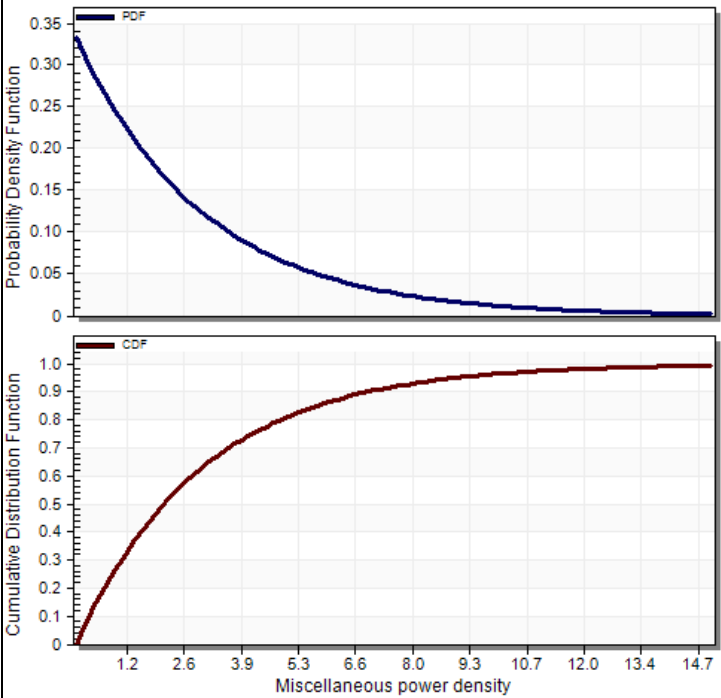
Depending on distribution curve various distribution parameters are available.

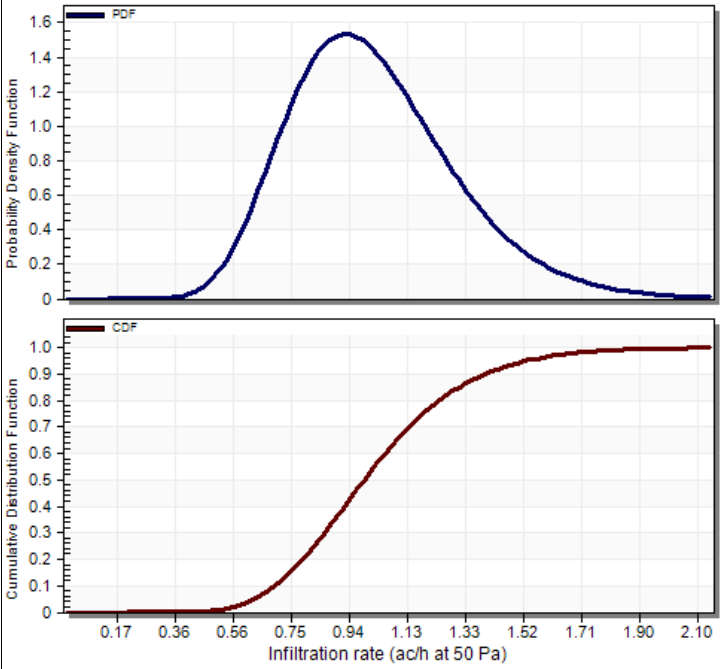
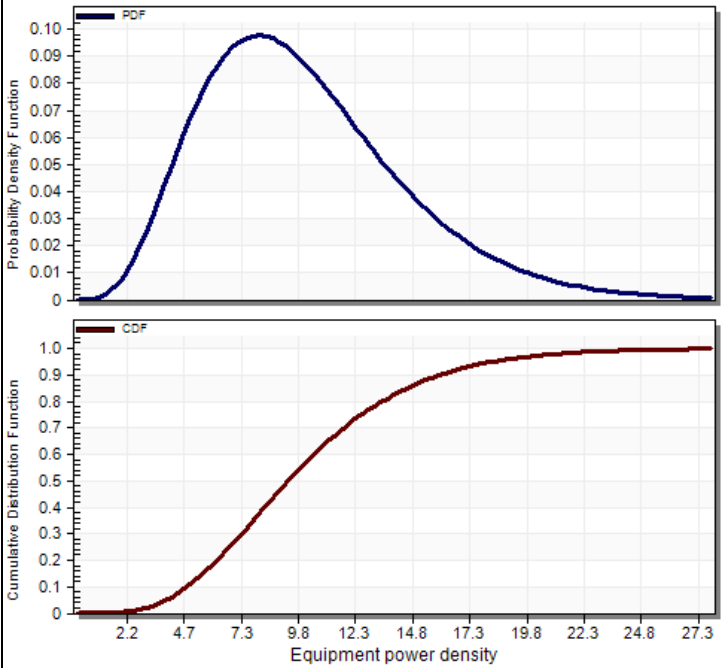
View the full list of all distributions and their controllers

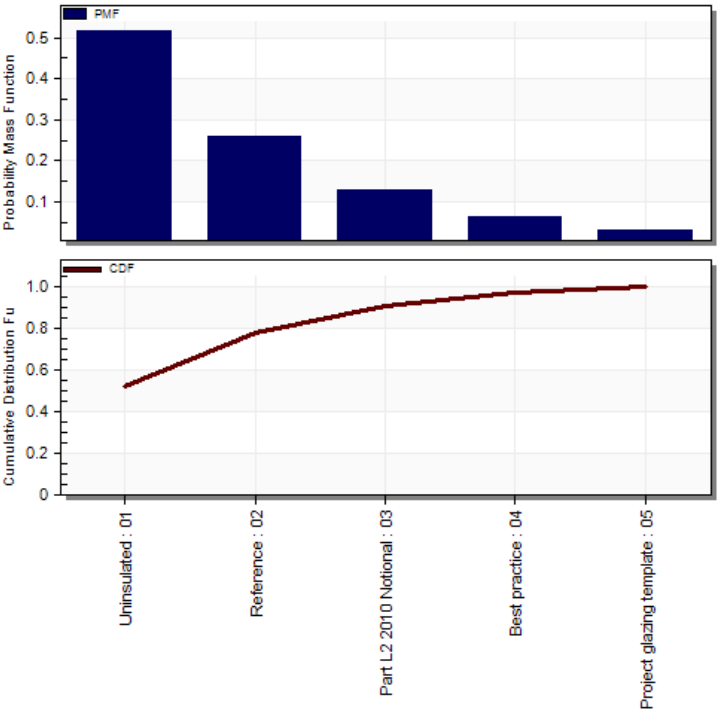
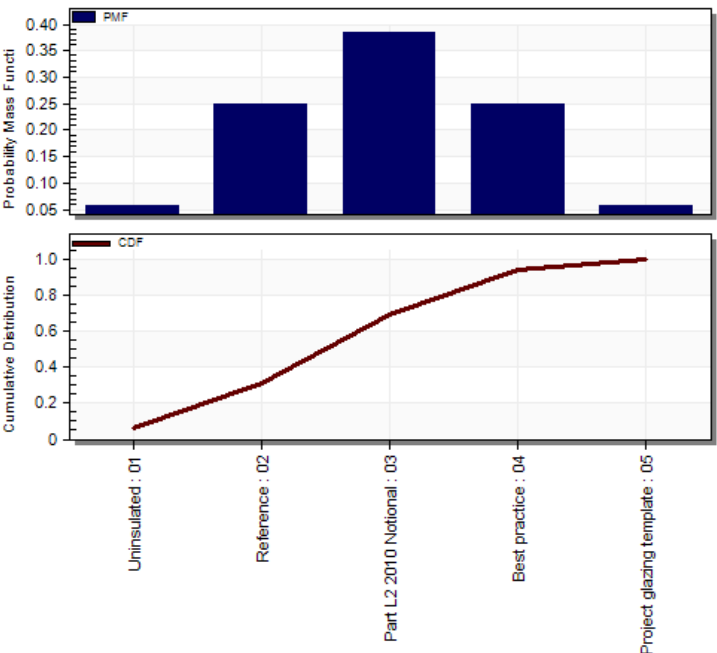
SNo.	Distribution	Parameters	Limitation	Example Case	Example Image
1	Bernoulli	Probability of Second option	Value Between 0 and 1	Variable: Glazing Template Variable options selected: 2 Probability of Second option = 0.7	

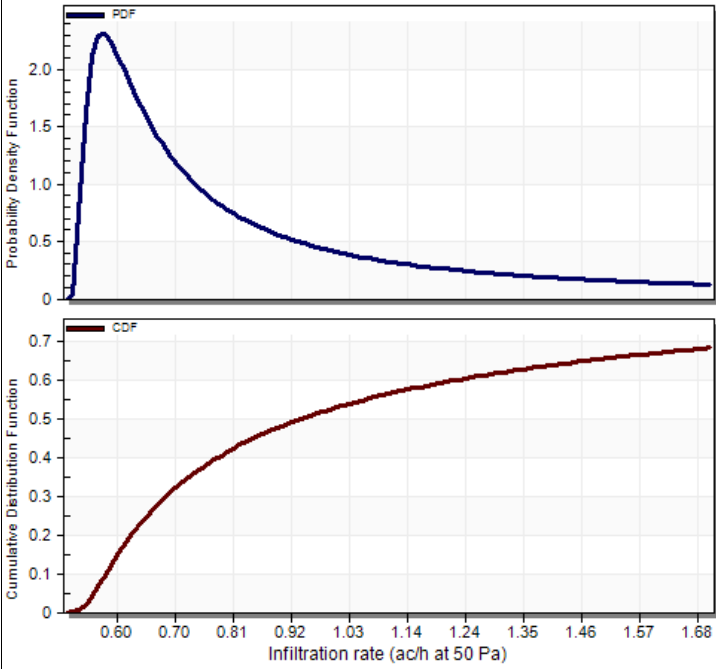
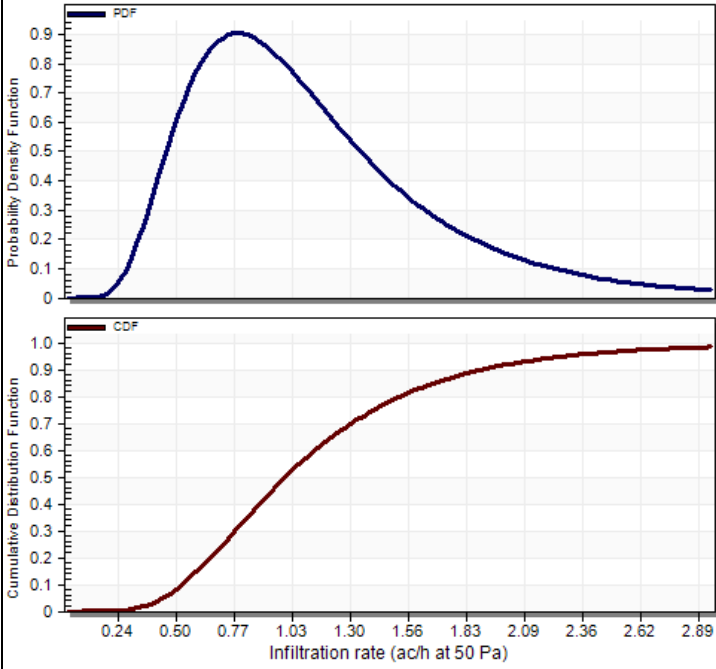
2	Beta	Left shape parameter	>0, Real Number, up to 50	Variable: Boiler CoP Left shape parameter = 9 Right shape parameter = 3	
3 A	Binomial for List type variables	Probability	Value Between 0 and 1		
		Number of Trials	Locked	Variable: Glazing Template Variable options selected: 5 Probability = 0.5	
3 B	Binomial for Numeric type variable	Probability	Value Between 0 and 1	Variable: % External	

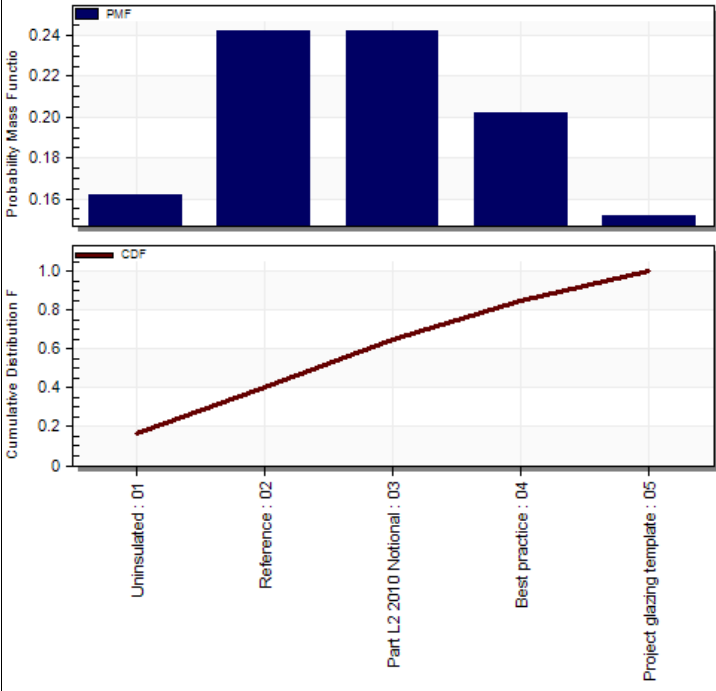
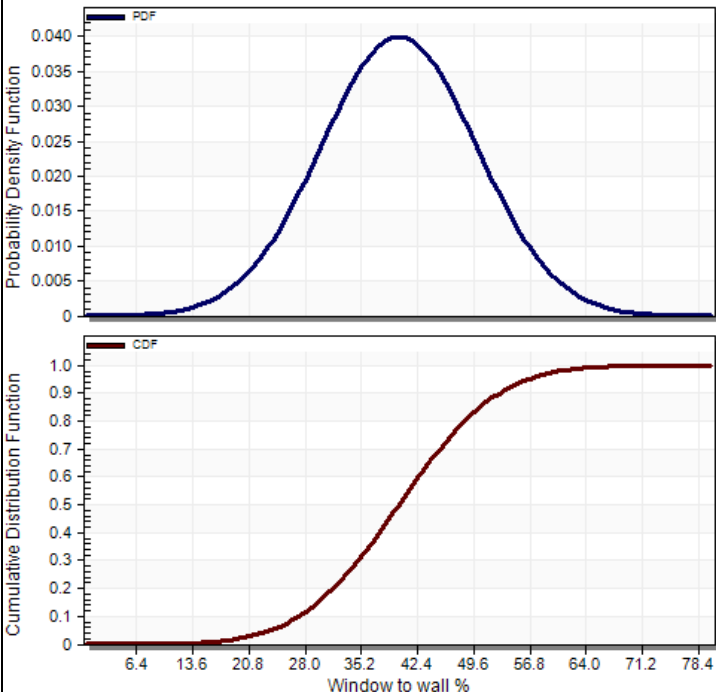
		Number of Trials	Positive Integer	Window Opens Probability = 0.6 Number of trials = 10	
4	Cauchy	Scale Parameter Location (Peak Value)	>0, Real Number up to 10 Real Number	Variable: Window Wall % Scale Parameter = 5 Location = 50	

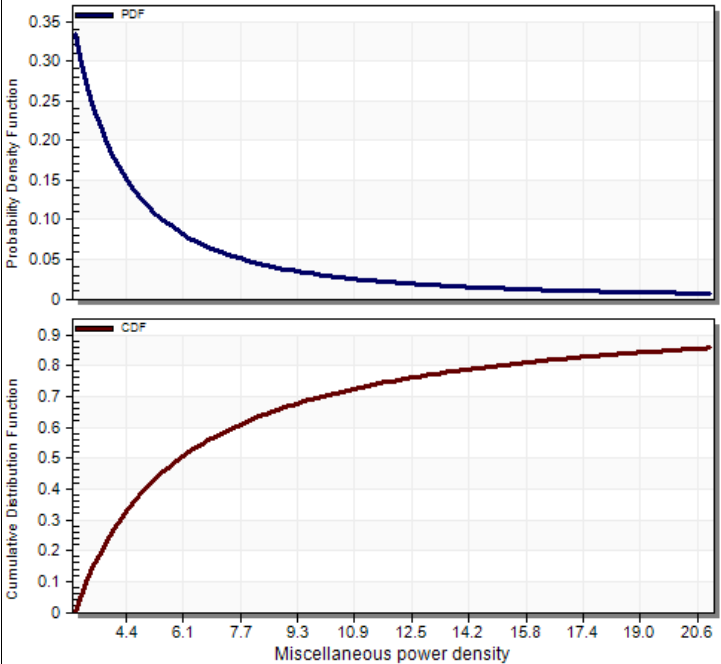
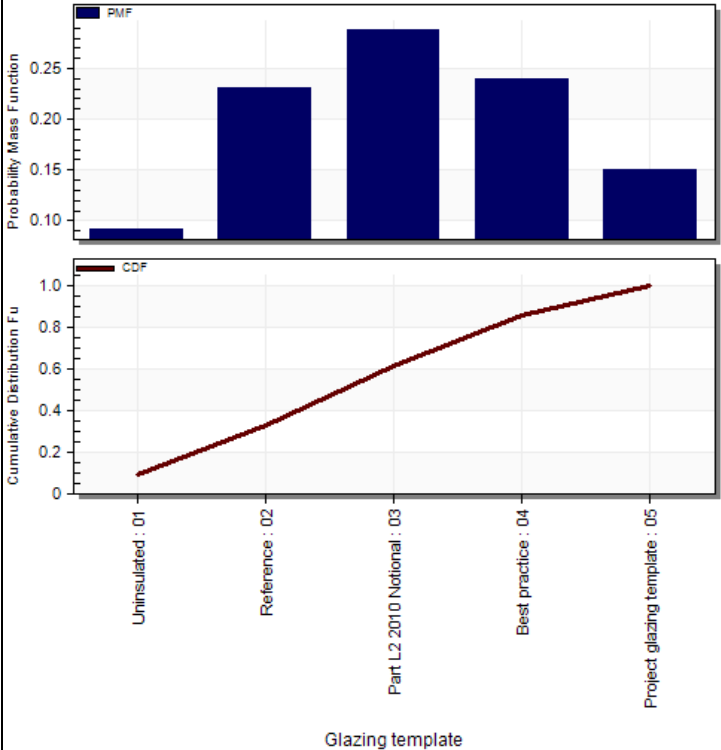
5	Chi Squared	Degree of Freedom (n)	Positive Integer up to 100	Variable: Equipment Power Density Degree of Freedom (n) = 15	
6	Exponential	Scale Parameter	>0, Real Number up to 5	Variable: Miscellaneous Power Density Scale Parameter = 3	
7	Fisher F	Degree of Freedom (n)	>0, Real Number up to 5	Variable: Infiltration rate	

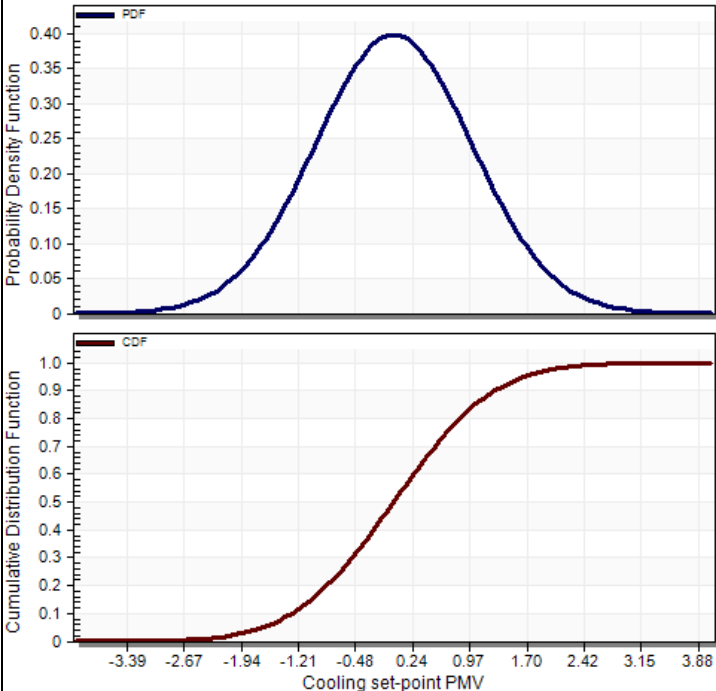
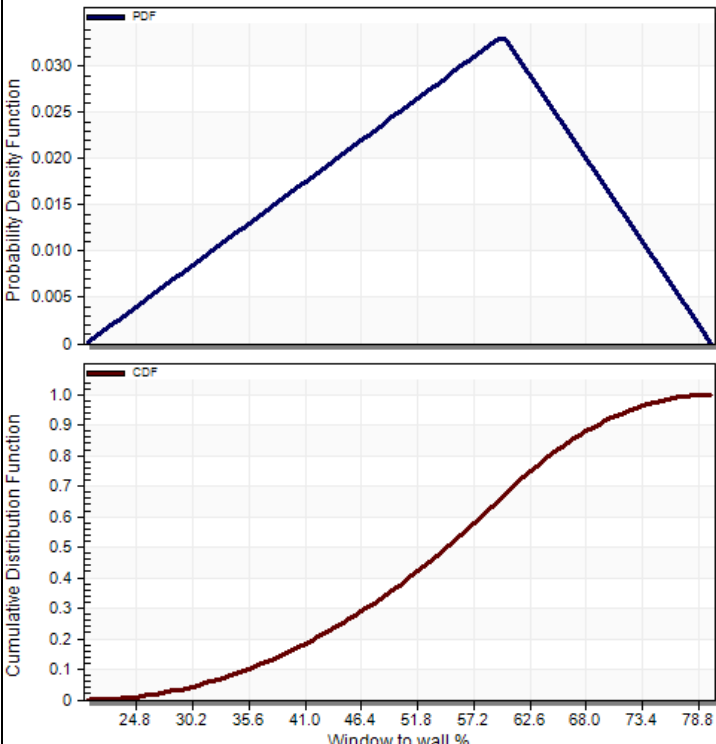
		Degree of Freedom (d)	>0, Real Number up to 5	Degree of Freedom (n) = 40 Degree of Freedom (d) = 90	
8	Gamma	Shape Parameter Scale Parameter	>0, Real Number up to 20 >0, Real Number up to 5	Variable: Equipment Power Density Shape Parameter = 5 Scale Parameter = 2	

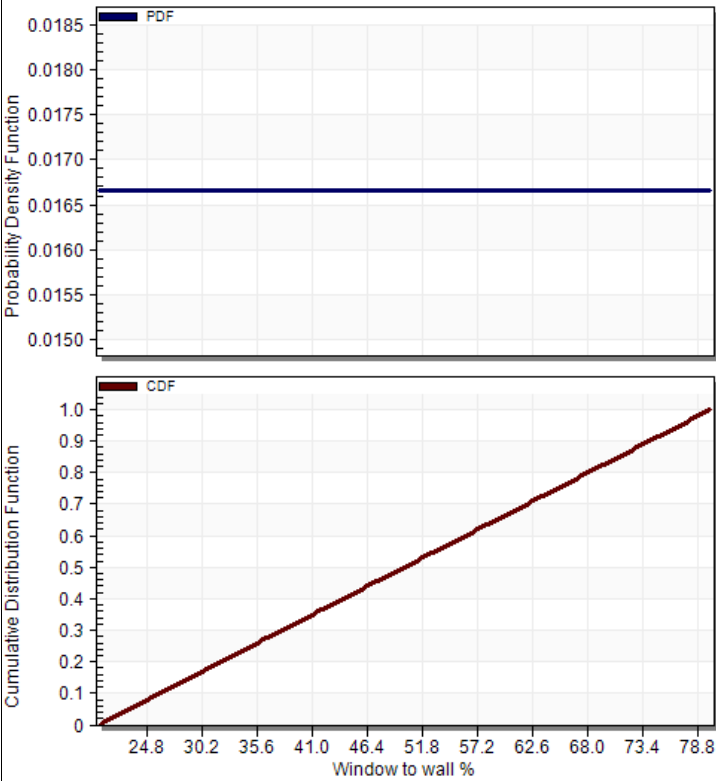
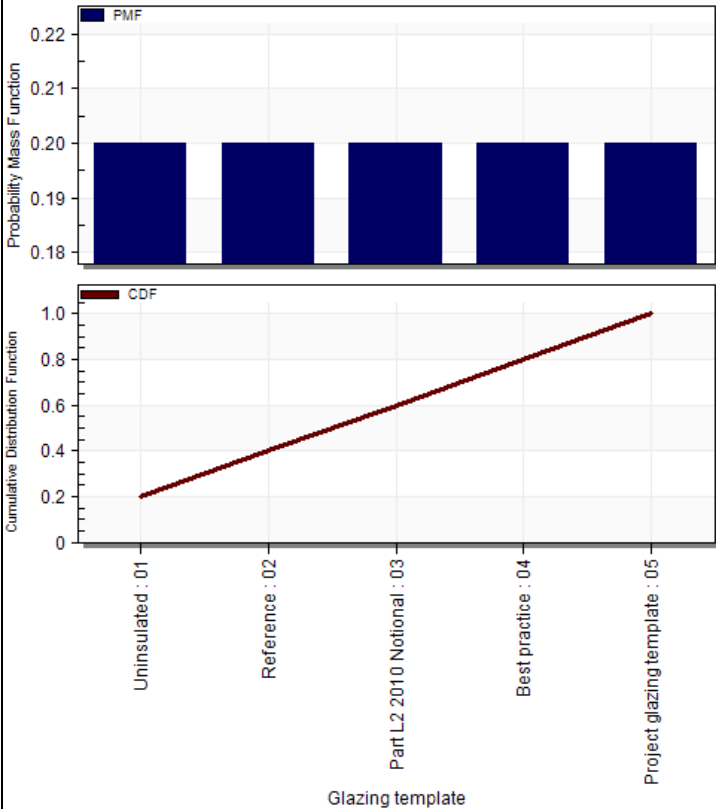
9	Geometric	Probability of first option	Value Between 0 and 1	Variable: Glazing Template Variable options selected: 5 Probability of first option = 0.5	 Glazing template
10	Hypergeometric	Number of trials Size of population No of successes	locked for list type variables locked for list type variables Value between no of trials & (Size of population - number of trials)	Variable: Glazing Template Variable options selected: 5 No of successes = 50	 Glazing template
11	Levy	Scale Parameter	>0, Real Number up to 10	Variable: Infiltration rate	

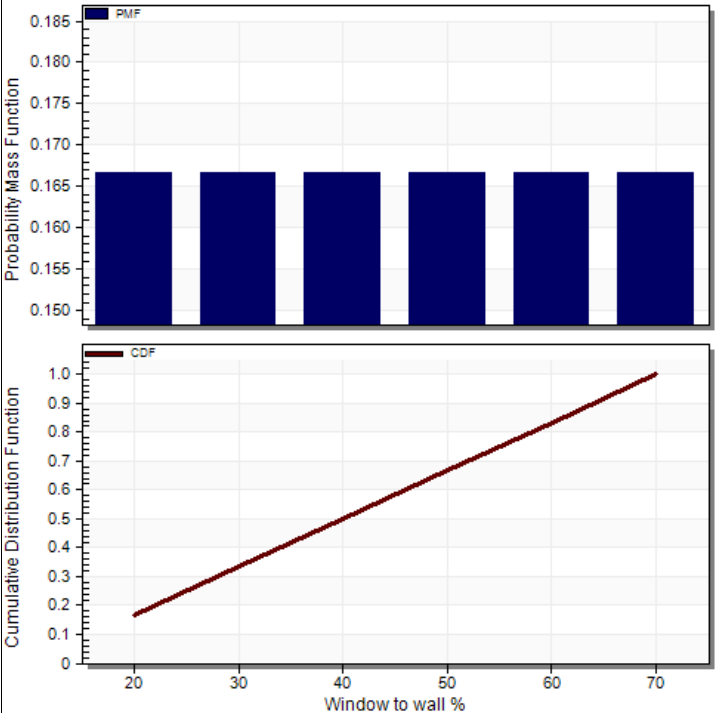
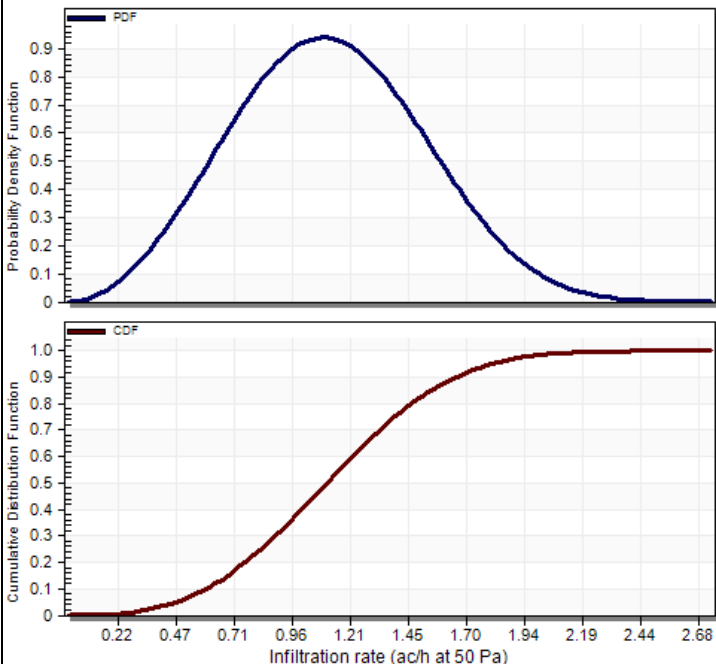
		Min Value	Real Number	Scale Parameter = 0.2 Min Value = 0.5	
12	Log Normal	Mean Standard Deviation	Real Number >0 value	Variable: Infiltration rate Mean = 0 Standard Deviation = 0.5	
13	Negative Binomial	Probability	Value Between 0 and 1	Variable: Glazing Template	

		No. of trials	Positive Integer up to 10	Variable options selected: 5 Probability = 0.5 No of trials = 3	 Glazing template
14	Normal	Mean Standard Deviation	Real Number >0 value	Variable: Window Wall % Mean = 40 Standard Deviation = 10	 Window to wall %
15	Pareto	Shape Parameter	>0, Real Number up to 50	Variable: Miscellaneous	

		Scale Parameter	>0, Real Number up to 50	Power Density Shape Parameter = 1 Scale Parameter = 3	
16	Poisson	Mean	>0, Real Number up to 50	Variable: Glazing Template Variable options selected: 5 Mean = 2.5	

17	Student T	Degree of Freedom (n)	Positive Integer up to 100	Variable: Cooling setpoint PMV Degree of Freedom (n) = 100	
18	Triangular	Location (Peak Value)	Between Min and Max	Variable: Window Wall % Location (Peak Value) = 60	
19	Uniform (Continuous)	Min Value	Real Number less than Max Val	Variable: Window Wall %	

		Max Value	Real Number more than Min Val	Min Value = 20 Max Value = 80	
20 A	Uniform (Discrete) for List type variable	No parameters to control	NA	Variable: Glazing Template Variable options selected: 5	
20 B	Uniform (Discrete) for Numeric type variable	Min Value Max Value Distribution Seep size	Real Number less than Max Val Locked >0, Real Number	Variable: Window Wall % Min Value = 20	

				Distribution Seep size = 10 Number of steps = 6	
21	Weibull	Shape Parameter Scale Parameter	>0, Real Number up to 50 >0, Real Number up to 50	Variable: Infiltration Rate Shape Parameter = 3 Scale Parameter = 1.25	

Transform Distribution

Functions under this header can be used to control the distribution curve value ranges to have more meaningful values to adapt to the variable type. This allows you to use curves whose default values are not necessarily in line with the Input Variable. These are only available for numeric distributions.

Shifting and Scaling: Depending on the operator selected, the value range of the default graph is either shifted or scaled or both.

Truncation: To avoid improbable or conflicting values being sampled, truncation can be used to remove the tales of the distribution.

Shift/Scale Value Range

Operator

There are 4 Options:

- **1-Shift only**: adds Shift value to each untransformed distribution $(x+a)$
- **2-Scale only**: multiplies scale value to each untransformed distribution $(x*b)$
- **3-Shift then Scale**: adds Shift value and then multiplies scale value to each untransformed distribution $(x+a)*b$
- **4-Scale then Shift**: multiplies scale value and then adds Shift value to each untransformed distribution $(x*b)+a$

Shift value

Enter the value which is added to the default distribution's numbers.

Note: When using IP units, for variables with units, the shifting of the distribution is controlled by a shift slider. The values can be increased or decreased by moving the slider right or left respectively. The effect of the shift slider setting on the distribution should be checked by looking at the graph at the bottom of the dialog. See screenshot under Scale value below.

Scale value

Enter the value which is multiplied to the default distribution's numbers.

Note: When using IP units, for variables with units, the scaling of the distribution is controlled by a scale slider. The graph values scale up or down on moving the slider right or left respectively. The effect of the scale slider setting on the distribution should be checked by looking at the graph at the bottom of the dialog.

The screenshot below illustrates a case where cooling setpoint temperature distribution is controlled using the Beta distribution with slider controls and lower and upper truncation when using IP units.

Analysis Variables

General

Design Variables

Name Cooling set-point temperature

Category <All>

Variable type Cooling set-point temperature

Distribution category 2-Continuous

Distribution curve 2-Beta

Distribution Parameters

Left shape parameter 5

Right shape parameter 5

Transform Distribution

☒ Shift/Scale value range

Shift/Scale operator 3-Shift then Scale

Shift Graph

Left Right

Scale Graph

Down Up

☒ Truncate lower side

Lower limit (°F) 75

☒ Truncate upper side

Upper limit (°F) 80

Target

Target objects Building

Graph

PDF

Probability Density Function

CDF

Cumulative Distribution Function

Cooling set-point temperature (°F)

Truncate Lower side

Limit:

This value removes all the variable values below this number. In this case the probability which is removed is redistributed across the remaining curve. The value should be more than the minimum value in the untruncated graph and less than truncation upper side limit.

Truncate Upper side

Limit:

This value removes all the variable values above this number. In this case the probability which is removed is redistributed across the remaining curve. The value should be less than the maximum value in the untruncated graph and more than the truncation lower side limit.

The value should be less than the maximum value in the untruncated graph and more than the truncation lower side limit.

[View the full list of all distributions and transformations available](#)

	Distribution	Transformations available	Example Image
--	--------------	---------------------------	---------------

2

Beta

Shift/Scale

Design Variables

Name **Window to wall %**

Category <All>

Variable type Window to wall %

Distribution category 2-Continuous

Distribution curve 2-Beta

Distribution Parameters

Left shape parameter 9

Right shape parameter 3

Transform Distribution

☒ Shift/Scale value range

Shift/Scale operator 4-Scale then Shift

Addition value (to shift) 10

Multiplier (to scale) 80

☐ Truncate lower side

☐ Truncate upper side

Target

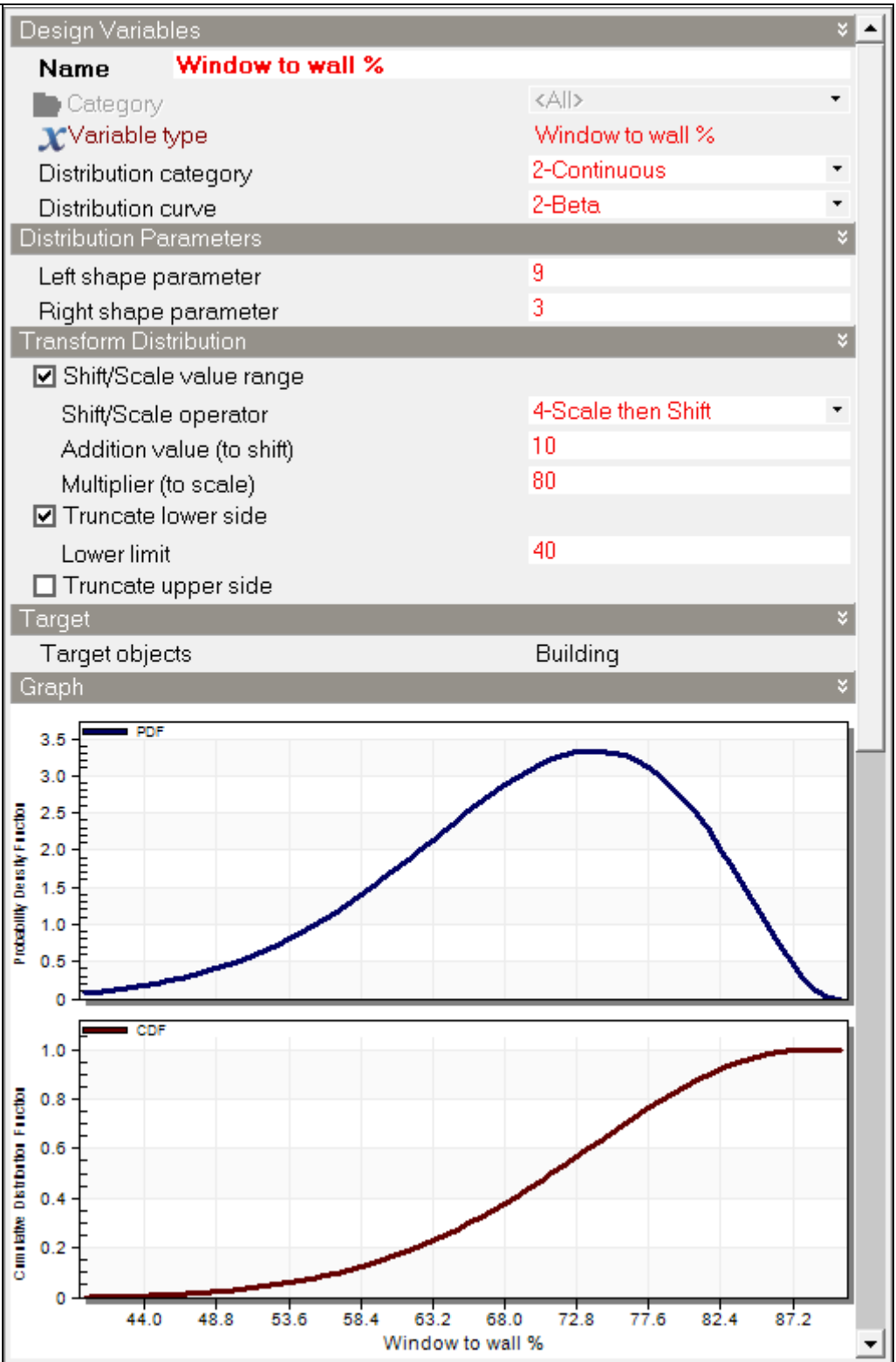
Target objects Building

Graph

The graph displays two plots for the variable 'Window to wall %':

- PDF (Probability Density Function):** The top plot shows a blue curve peaking at approximately 3.4 on the y-axis (ranging from 0 to 3.5) at a value of about 74.0% on the x-axis (ranging from 16.4 to 88.4%).
- CDF (Cumulative Distribution Function):** The bottom plot shows a red S-shaped curve starting near 0 on the y-axis (ranging from 0 to 1.0) at a value of about 16.4% on the x-axis, and reaching 1.0 at approximately 88.4%.

Truncate
Lower Side



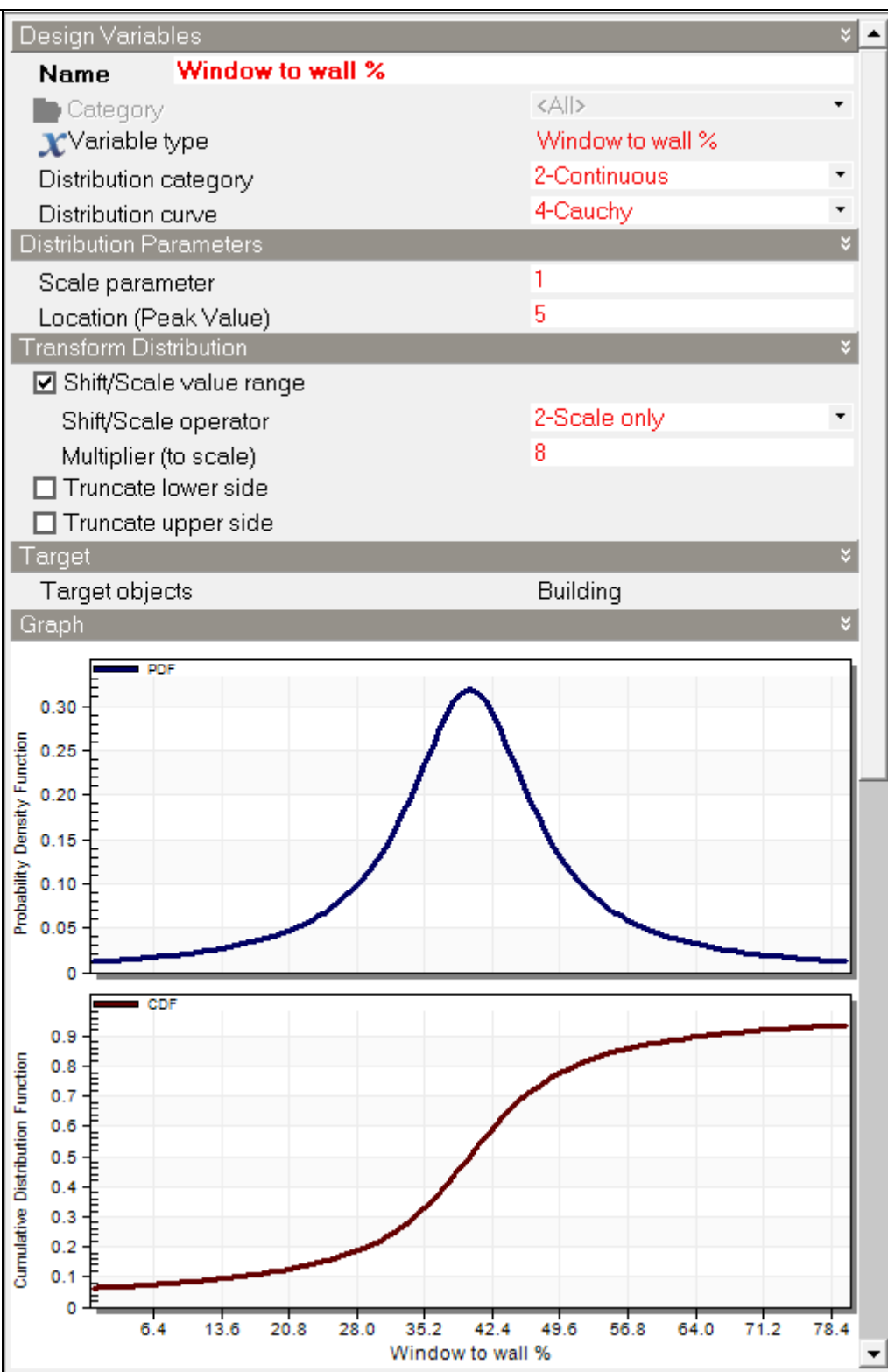
Truncate
Upper Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	2-Beta
Distribution Parameters	
Left shape parameter	9
Right shape parameter	3
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	4-Scale then Shift
Addition value (to shift)	10
Multiplier (to scale)	80
☒ Truncate lower side	
Lower limit	40
☒ Truncate upper side	
Upper limit	80
Target	
Target objects	Building
Graph	
The graph displays two functions for the variable 'Window to wall %'. The top plot is the Probability Density Function (PDF), showing a blue curve that starts near zero at 43.2, rises to a peak of approximately 3.8 at 75.2, and then declines. The bottom plot is the Cumulative Distribution Function (CDF), showing a red curve that starts at 0 and increases monotonically to 1.0 at 79.2. The x-axis for both plots is labeled 'Window to wall %' and ranges from 43.2 to 79.2 with increments of 4.0.	

4

Cauchy

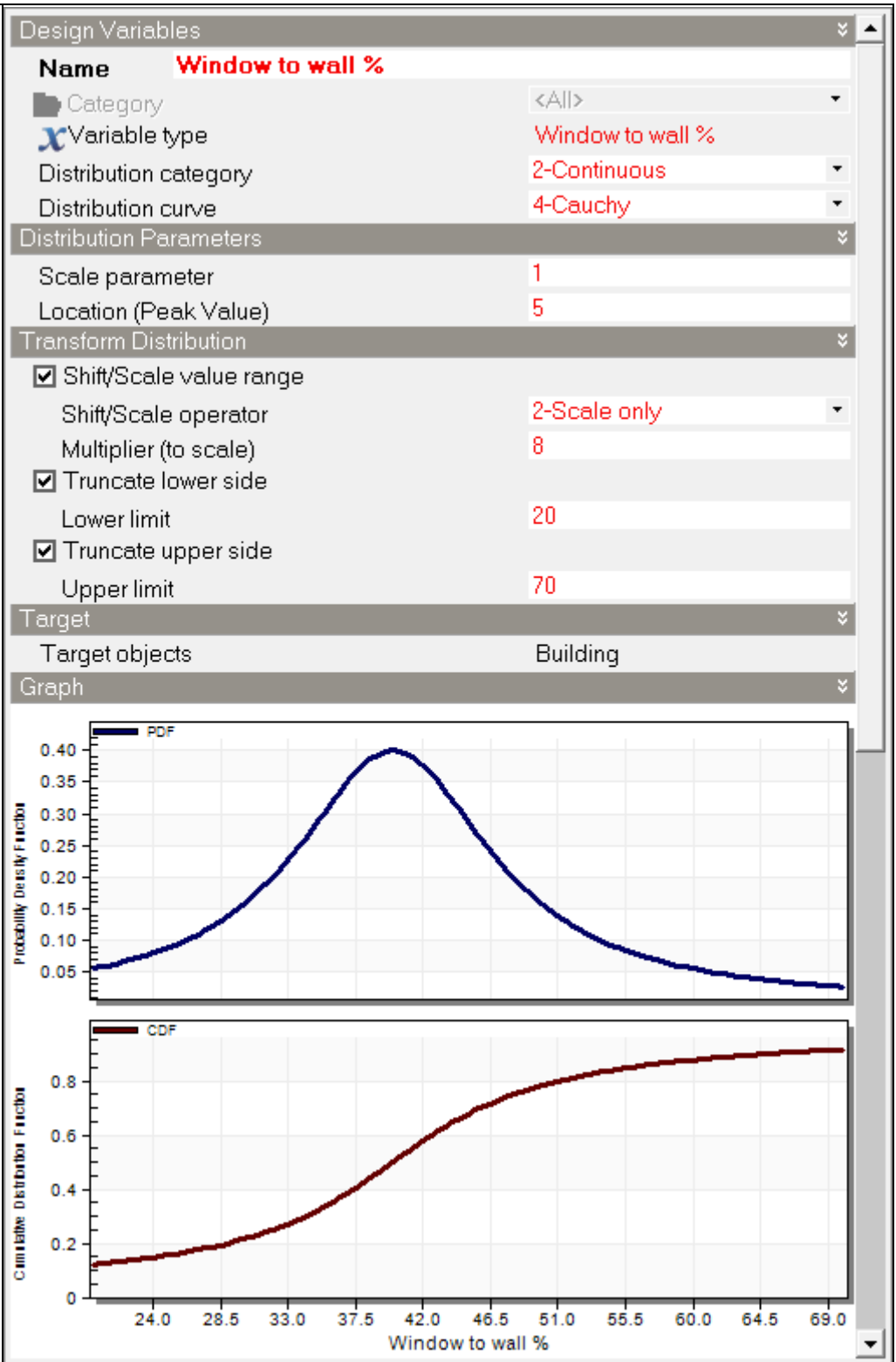
Shift/Scale



Truncate
Lower Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	4-Cauchy
Distribution Parameters	
Scale parameter	1
Location (Peak Value)	5
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	2-Scale only
Multiplier (to scale)	8
☒ Truncate lower side	
Lower limit	20
☐ Truncate upper side	
Target	
Target objects	Building
Graph	

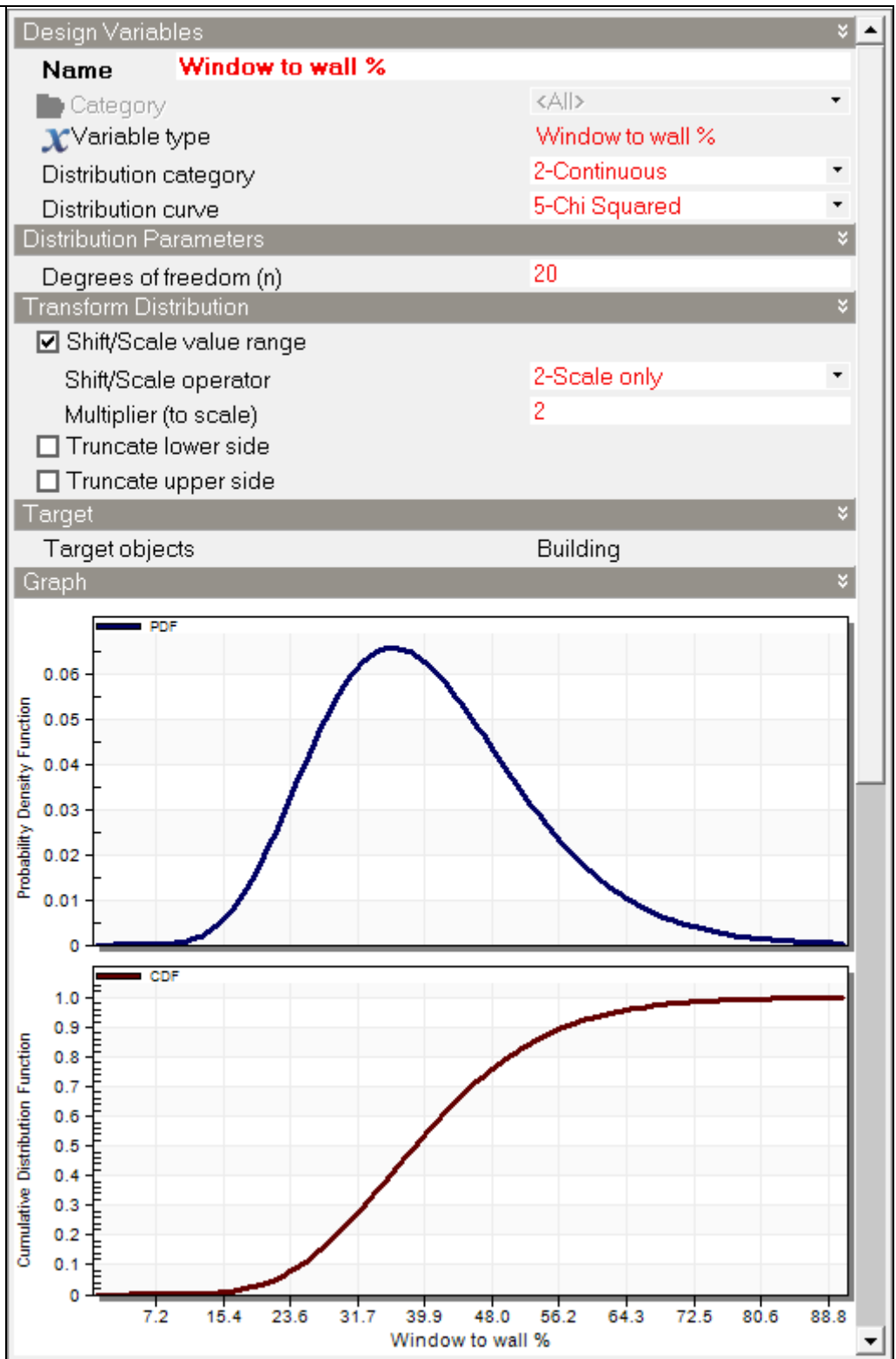
Truncate
Upper Side



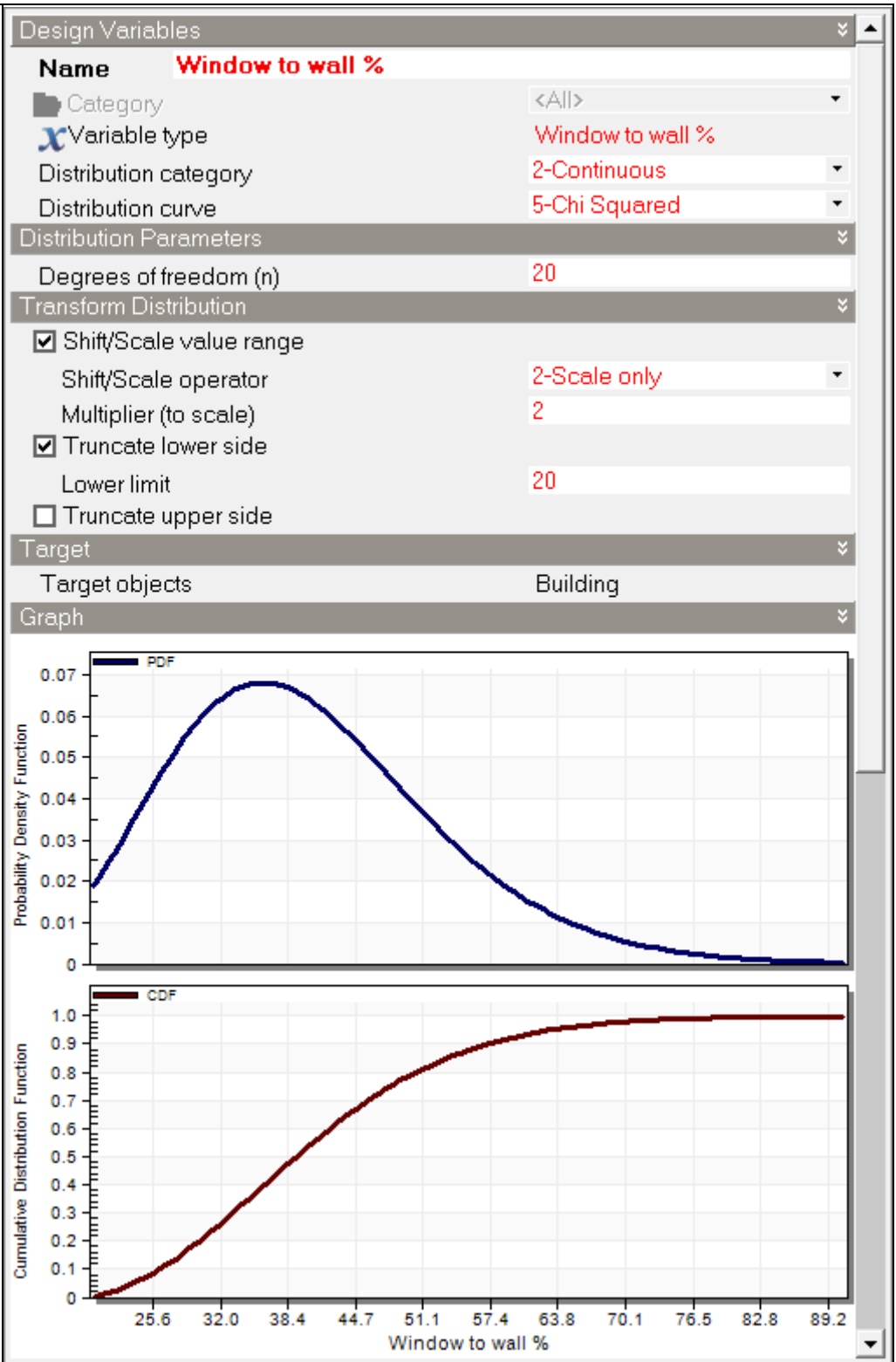
5

Chi Squared

Shift/Scale

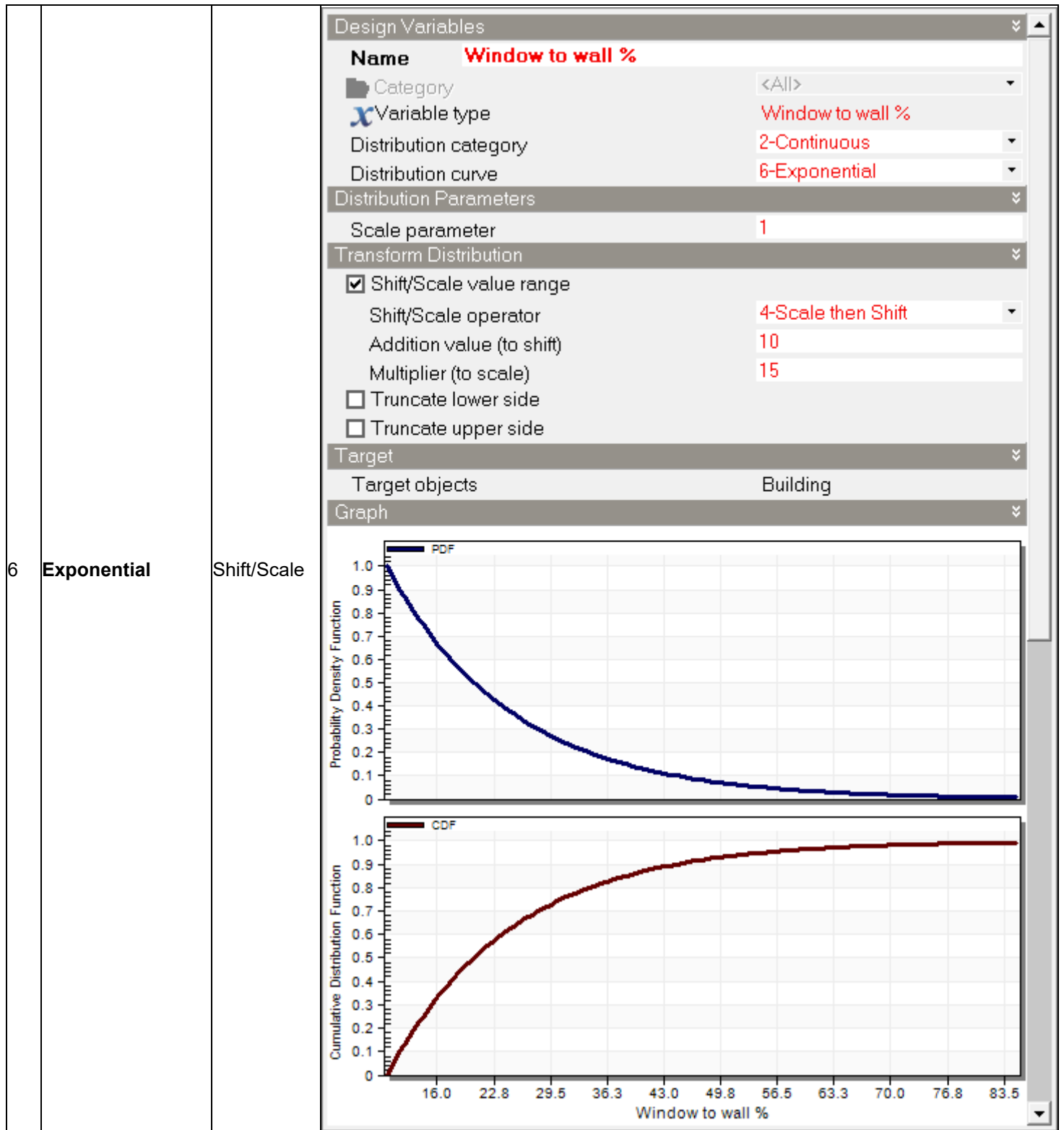


Truncate
Lower Side



Truncate
Upper Side

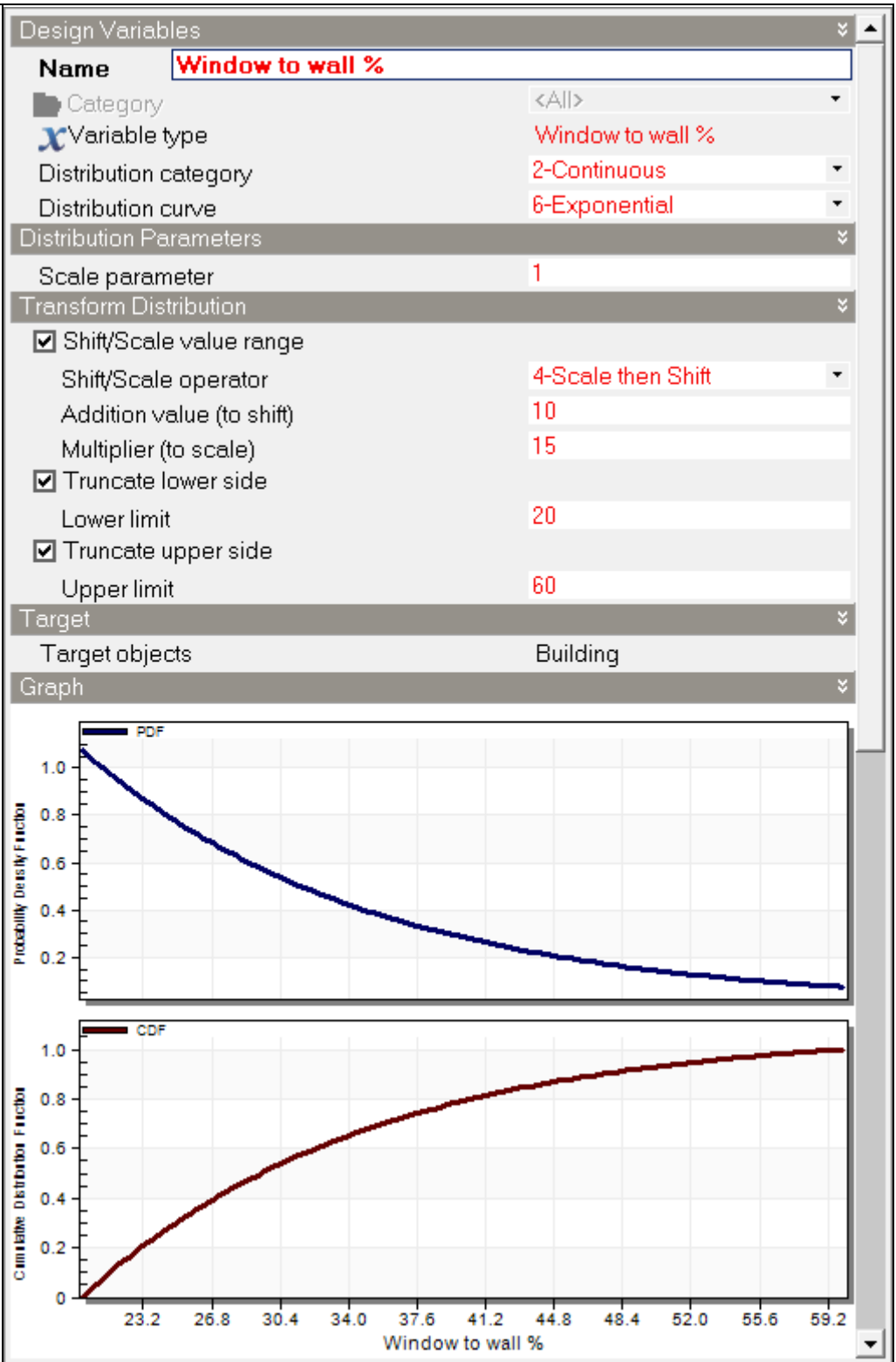
Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	5-Chi Squared
Distribution Parameters	
Degrees of freedom (n)	20
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	2-Scale only
Multiplier (to scale)	2
☒ Truncate lower side	
Lower limit	20
☒ Truncate upper side	
Upper limit	60
Target	
Target objects	Building
Graph	
The graph displays two plots for the variable 'Window to wall %'. The top plot is the Probability Density Function (PDF), showing a bell-shaped curve that peaks at approximately 0.072 around 37.6%. The bottom plot is the Cumulative Distribution Function (CDF), showing a monotonically increasing S-curve from 0 to 1.0 over the range of 23.2% to 59.2%.	



Truncate
Lower Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	6-Exponential
Distribution Parameters	
Scale parameter	1
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	4-Scale then Shift
Addition value (to shift)	10
Multiplier (to scale)	15
☒ Truncate lower side	
Lower limit	20
☐ Truncate upper side	
Target	
Target objects	Building
Graph	
The graph displays two plots for the variable 'Window to wall %'. The top plot is the Probability Density Function (PDF), showing a blue curve that starts at 1.0 and decays exponentially towards zero. The bottom plot is the Cumulative Distribution Function (CDF), showing a red curve that starts at 0 and increases towards 1.0. The x-axis for both plots is labeled 'Window to wall %' and has tick marks at 25.2, 31.1, 36.9, 42.8, 48.6, 54.5, 60.3, 66.2, 72.0, 77.9, and 83.7. The y-axis for the PDF is labeled 'Probability Density Function' and ranges from 0 to 1.0. The y-axis for the CDF is labeled 'Cumulative Distribution Function' and ranges from 0 to 1.0.	

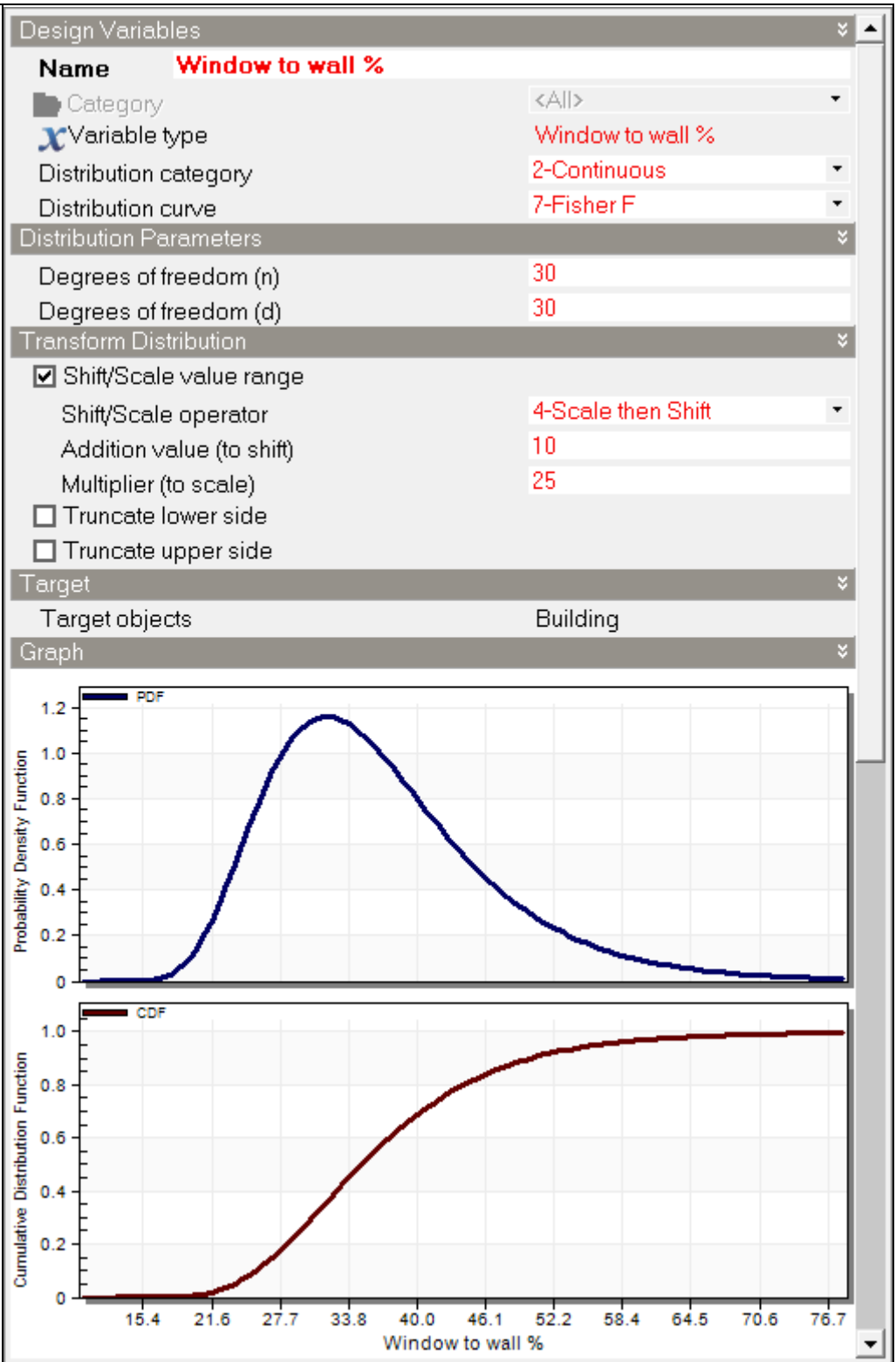
Truncate
Upper Side



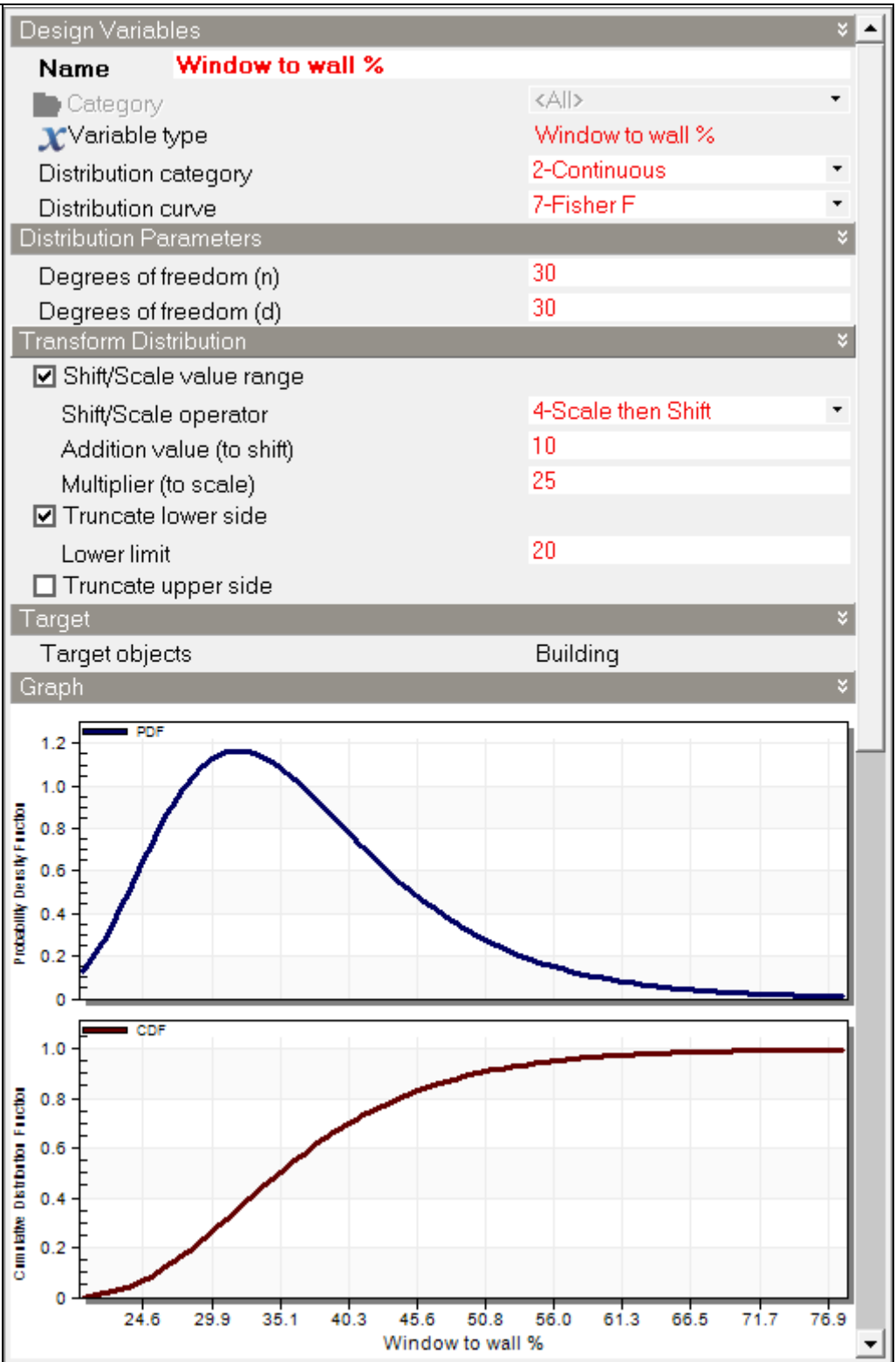
7

Fisher F

Shift/Scale

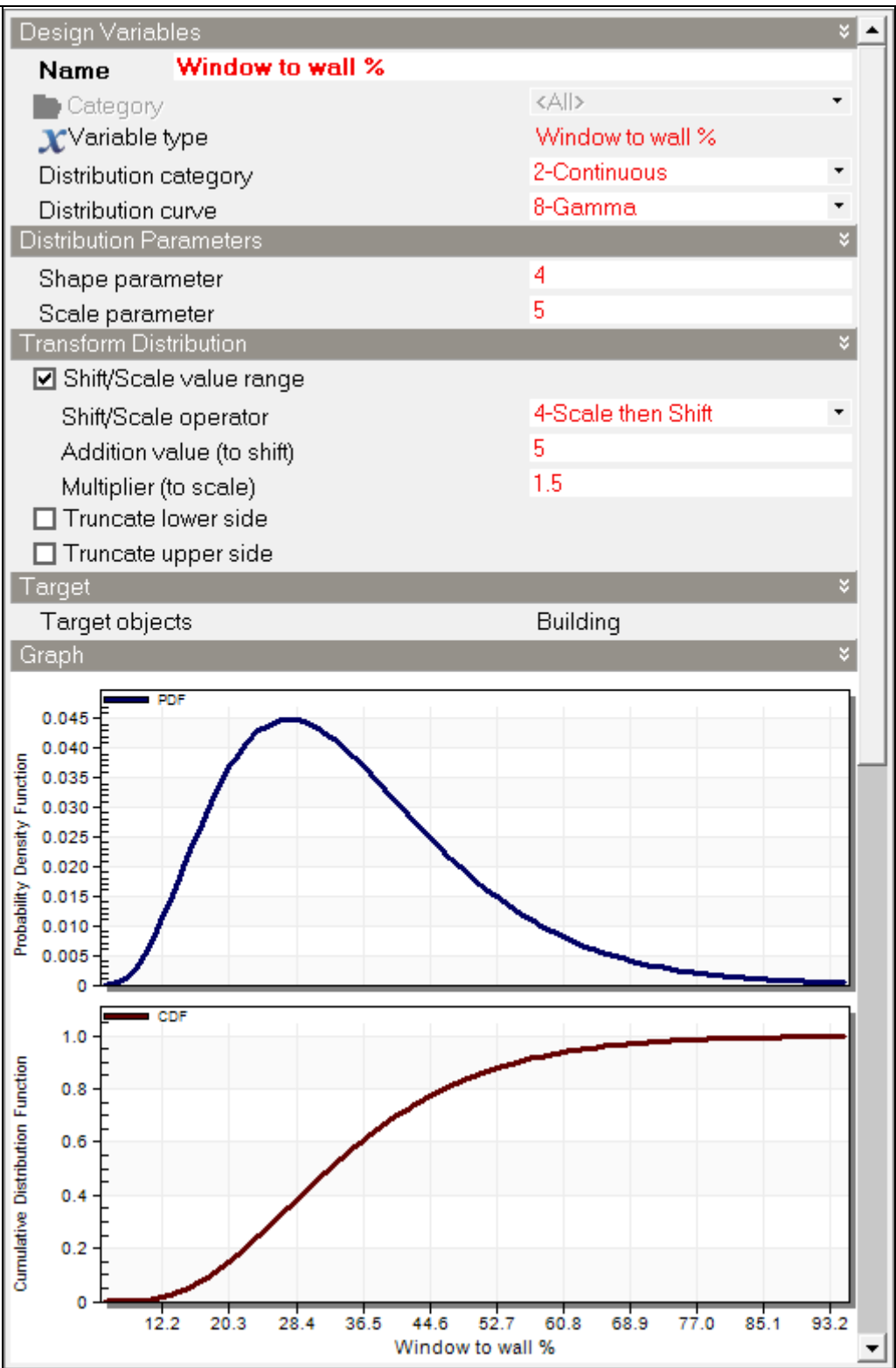


Truncate
Lower Side



Truncate
Upper Side

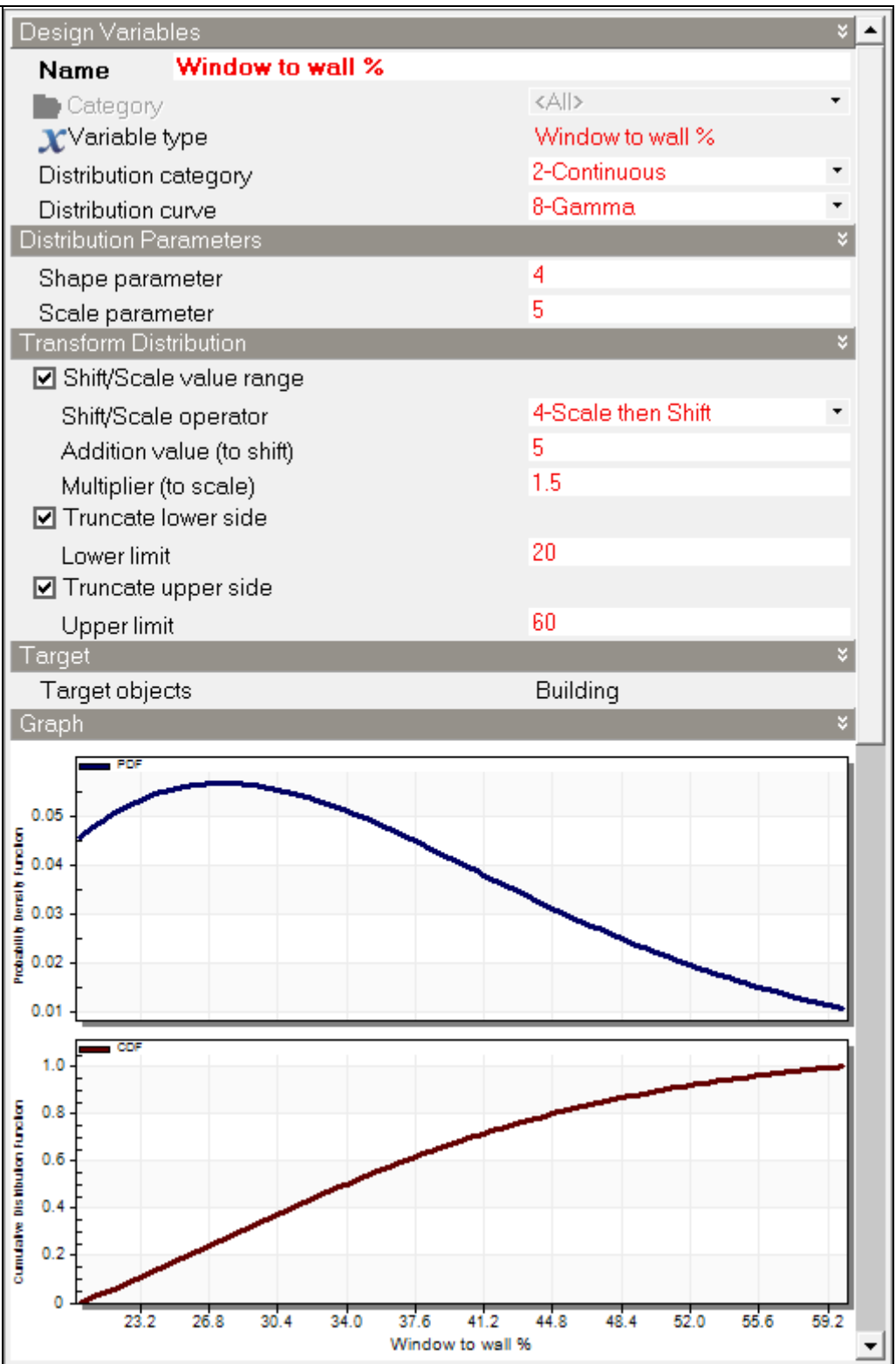
Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	7-Fisher F
Distribution Parameters	
Degrees of freedom (n)	30
Degrees of freedom (d)	30
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	4-Scale then Shift
Addition value (to shift)	10
Multiplier (to scale)	25
☒ Truncate lower side	
Lower limit	20
☒ Truncate upper side	
Upper limit	60
Target	
Target objects	Building
Graph	



Truncate
Lower Side

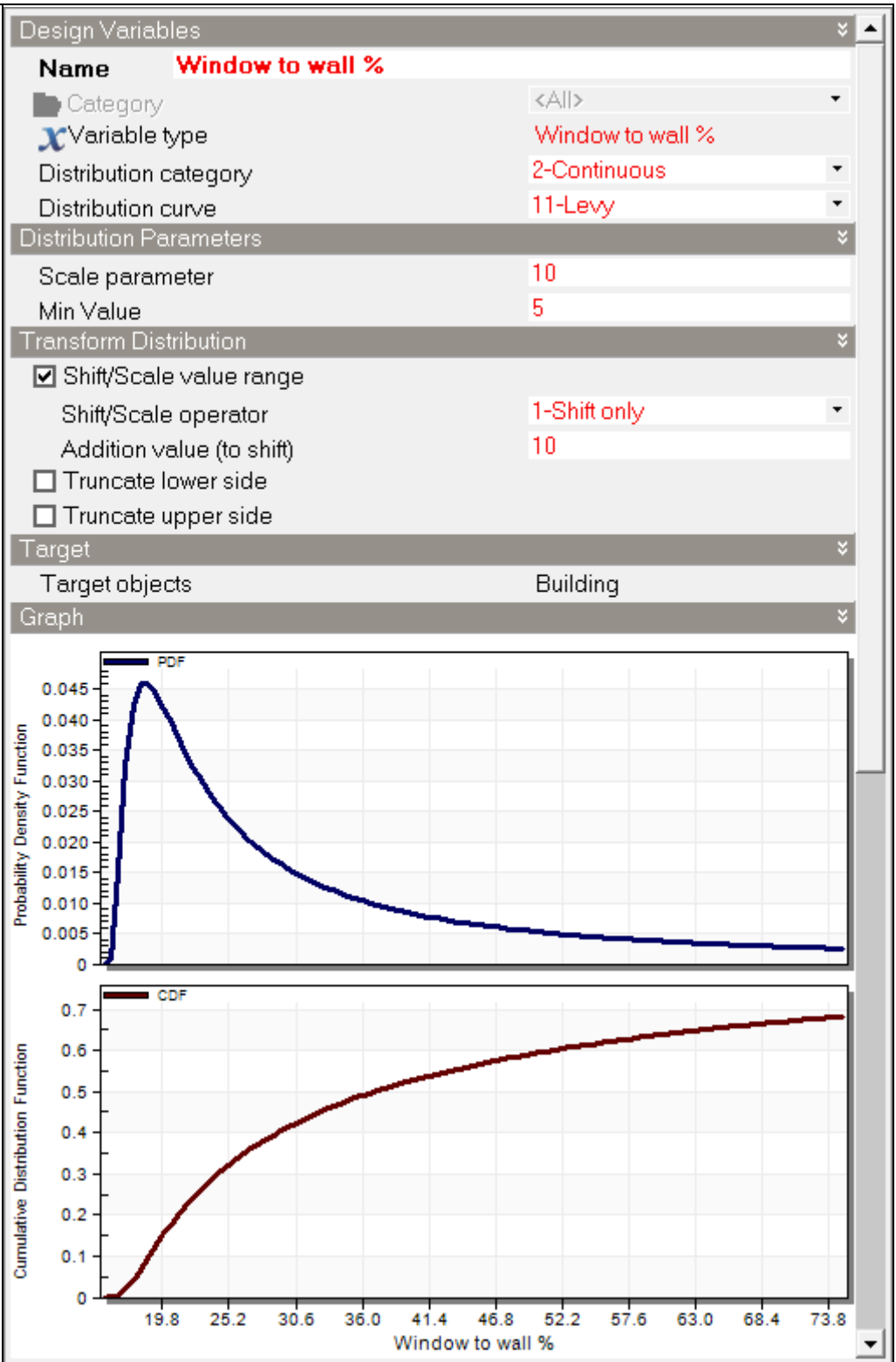
Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	8-Gamma
Distribution Parameters	
Shape parameter	4
Scale parameter	5
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	4-Scale then Shift
Addition value (to shift)	5
Multiplier (to scale)	1.5
☒ Truncate lower side	
Lower limit	20
☐ Truncate upper side	
Target	
Target objects	Building
Graph	

Truncate
Upper Side

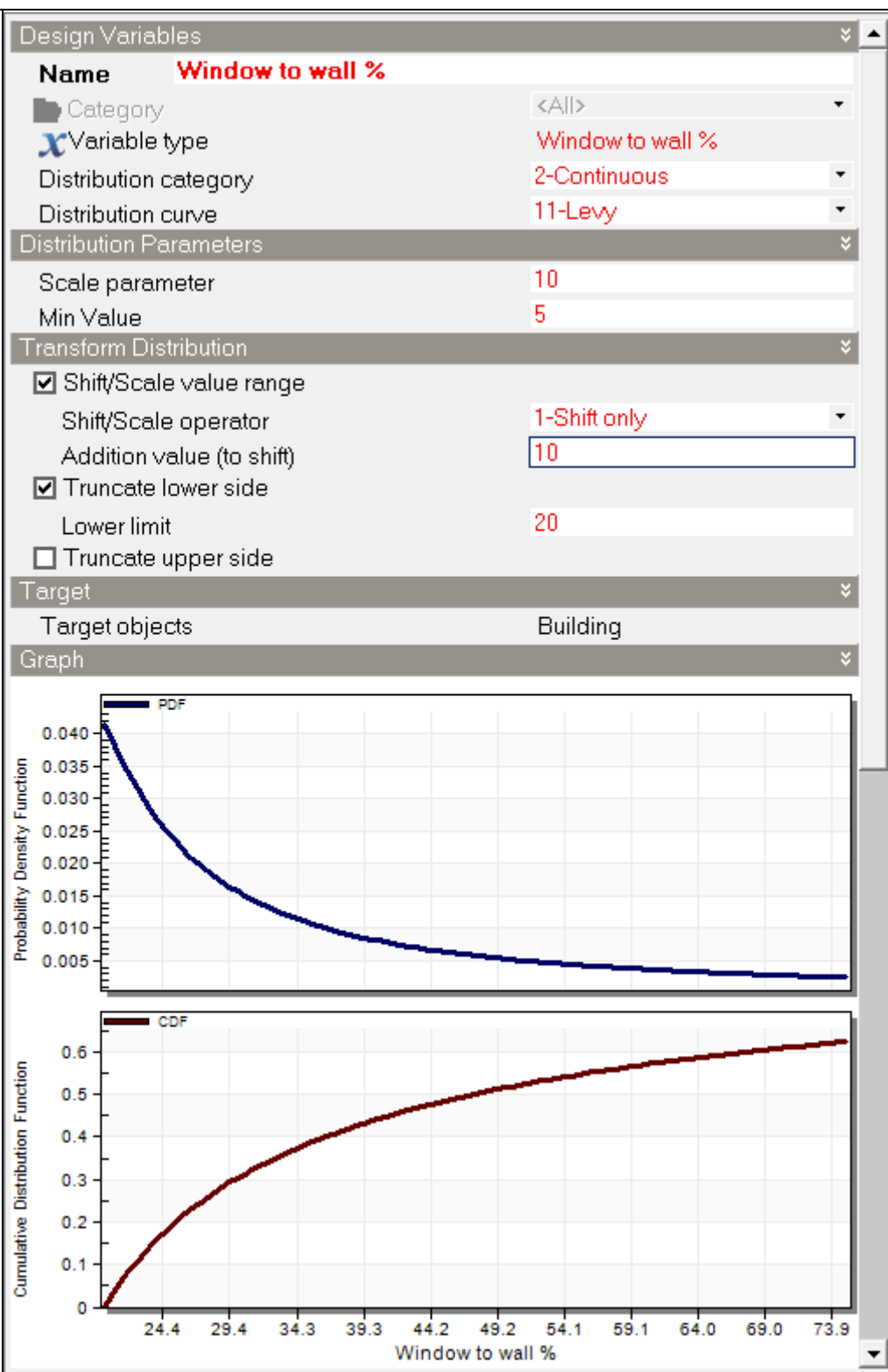


11 Levy

Shift/Scale



Truncate
Lower Side

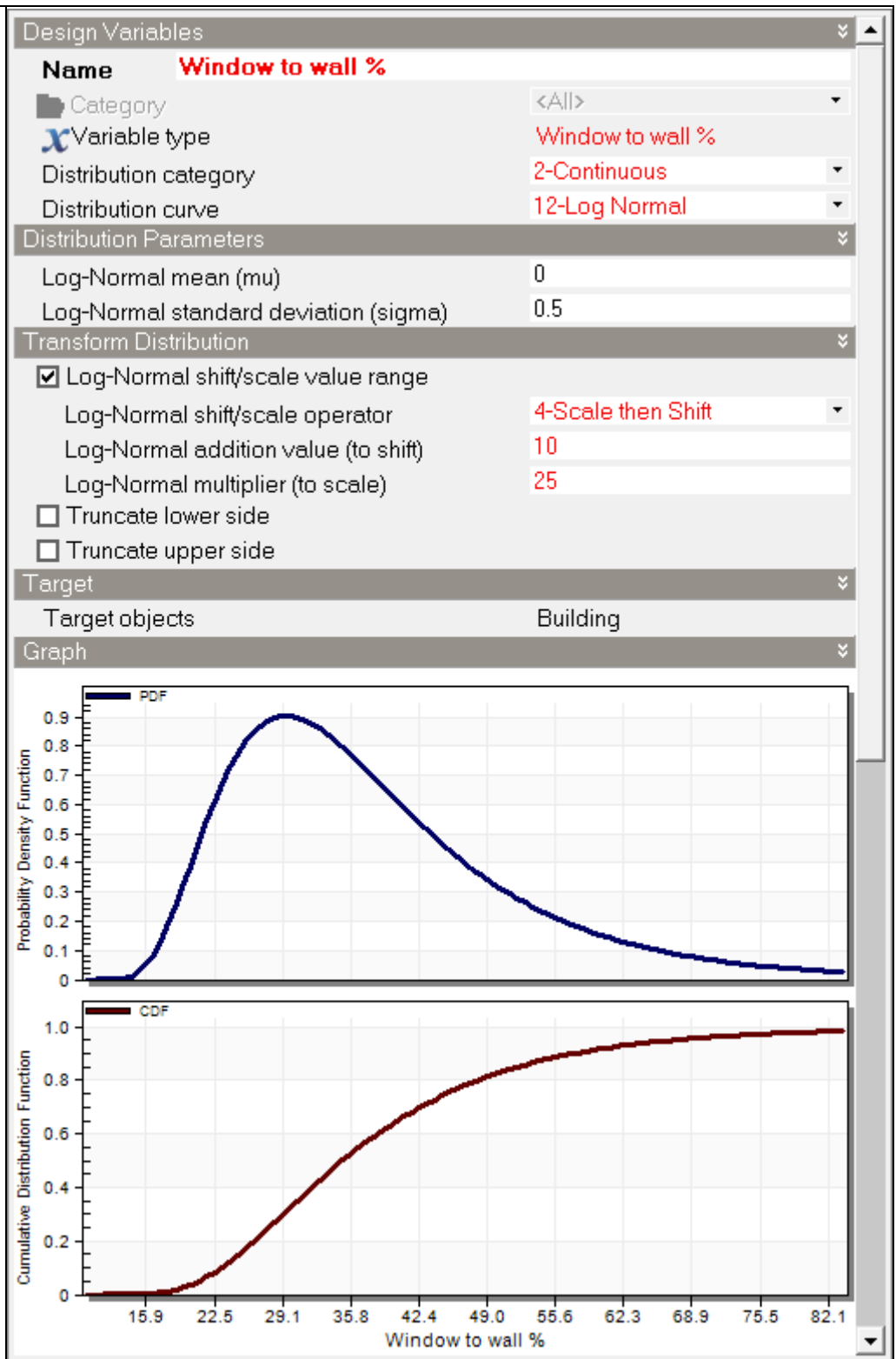


Truncate
Upper Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	11-Levy
Distribution Parameters	
Scale parameter	10
Min Value	5
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	1-Shift only
Addition value (to shift)	10
☒ Truncate lower side	
Lower limit	20
☒ Truncate upper side	
Upper limit	60
Target	
Target objects	Building
Graph	
The graph displays two plots for the variable 'Window to wall %'. The top plot shows the Probability Density Function (PDF) as a blue curve starting at approximately 0.055 at 23.2 and decreasing towards 0.005 at 59.2. The bottom plot shows the Cumulative Distribution Function (CDF) as a red curve starting at 0 at 23.2 and increasing to 1.0 at 59.2. The x-axis for both plots is labeled 'Window to wall %' with values ranging from 23.2 to 59.2. The y-axis for the PDF is labeled 'Probability Density Function' with values from 0.01 to 0.05. The y-axis for the CDF is labeled 'Cumulative Distribution Function' with values from 0 to 1.0.	

12 Log Normal

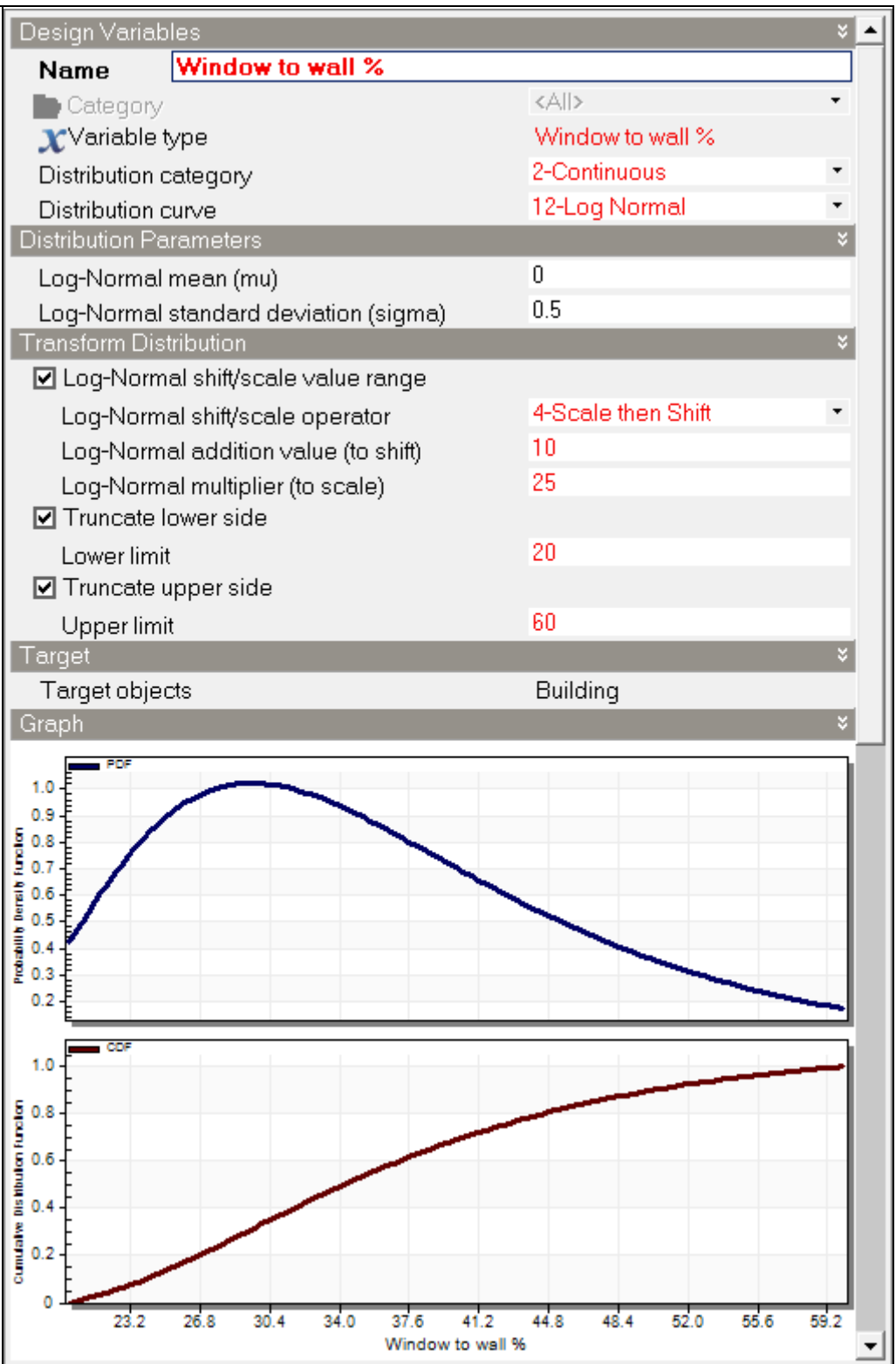
Shift/Scale



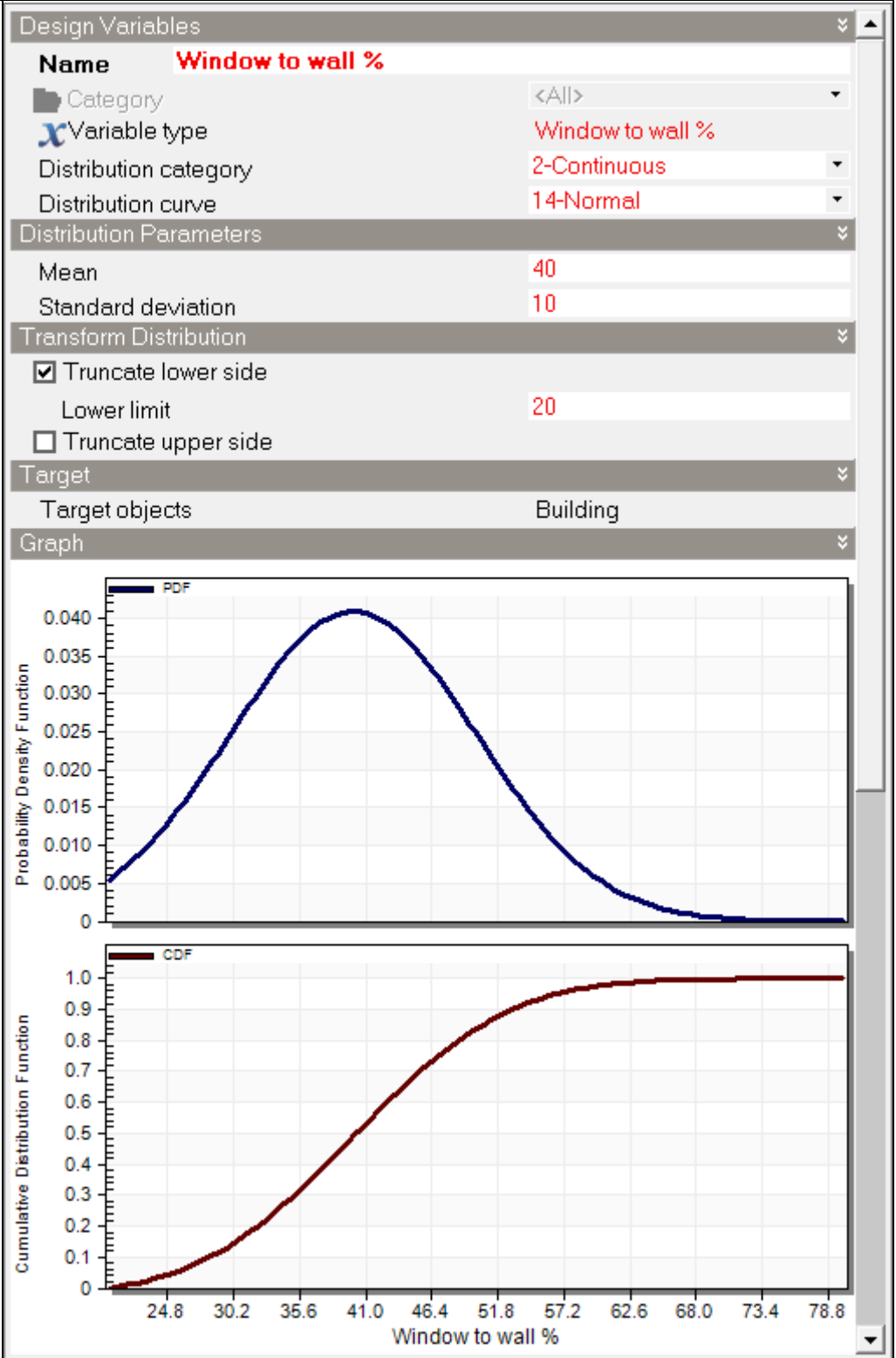
Truncate
Lower Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	12-Log Normal
Distribution Parameters	
Log-Normal mean (μ)	0
Log-Normal standard deviation (σ)	0.5
Transform Distribution	
☒ Log-Normal shift/scale value range	
Log-Normal shift/scale operator	4-Scale then Shift
Log-Normal addition value (to shift)	10
Log-Normal multiplier (to scale)	25
☒ Truncate lower side	
Lower limit	20
☐ Truncate upper side	
Target	
Target objects	Building
Graph	
The graph displays two plots for the variable 'Window to wall %'. The top plot is the Probability Density Function (PDF), showing a blue curve that peaks at approximately 0.95 around 30.8% and then tapers off towards 82.3%. The bottom plot is the Cumulative Distribution Function (CDF), showing a red curve that starts at 0 and levels off at 1.0. The x-axis for both plots is 'Window to wall %' with major ticks at 25.1, 30.8, 36.5, 42.3, 48.0, 53.7, 59.4, 65.2, 70.9, 76.6, and 82.3. The y-axis for the PDF is 'Probability Density Function' ranging from 0 to 0.8, and for the CDF is 'Cumulative Distribution Function' ranging from 0 to 1.0.	

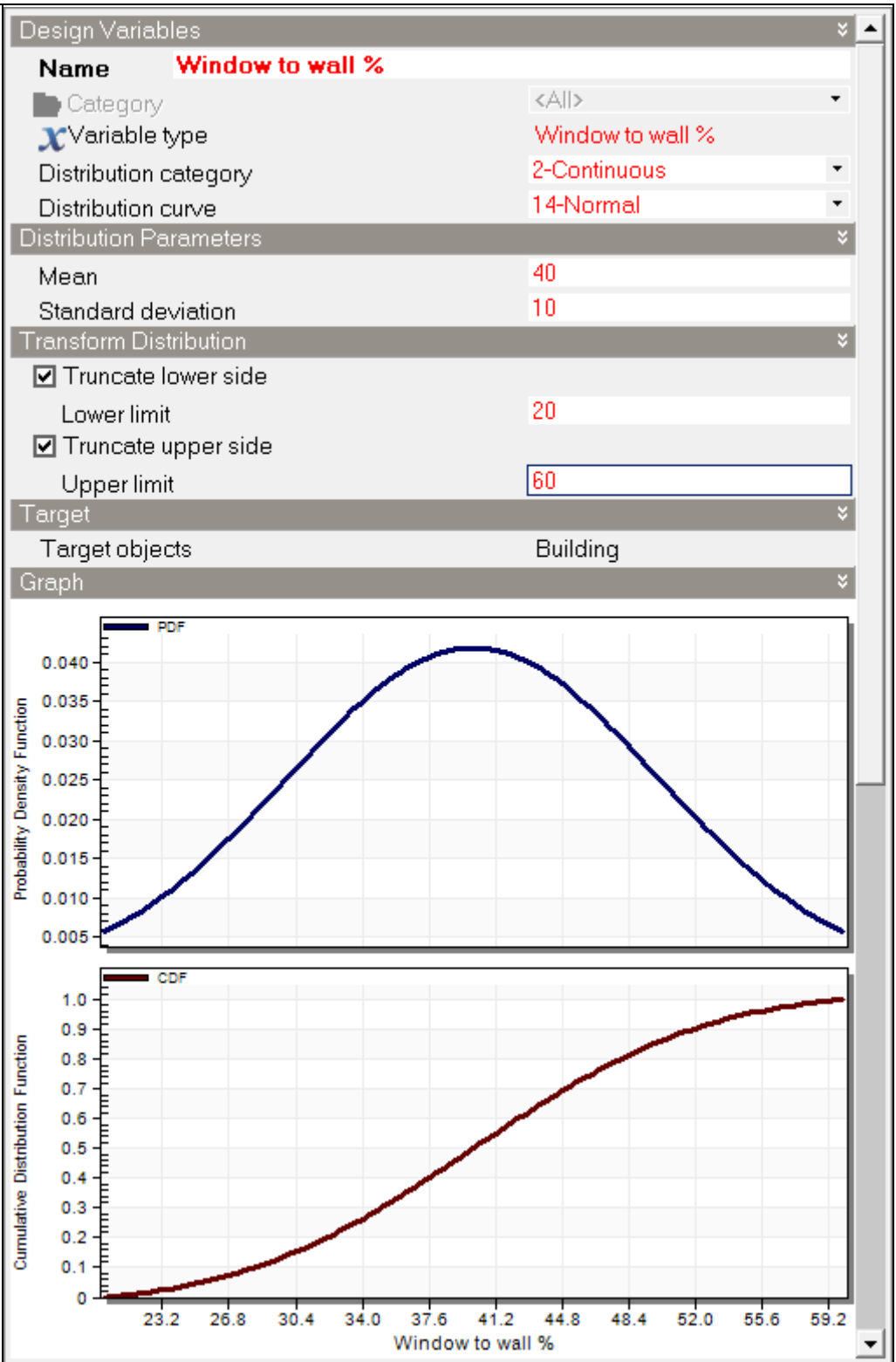
Truncate
Upper Side



14 Normal

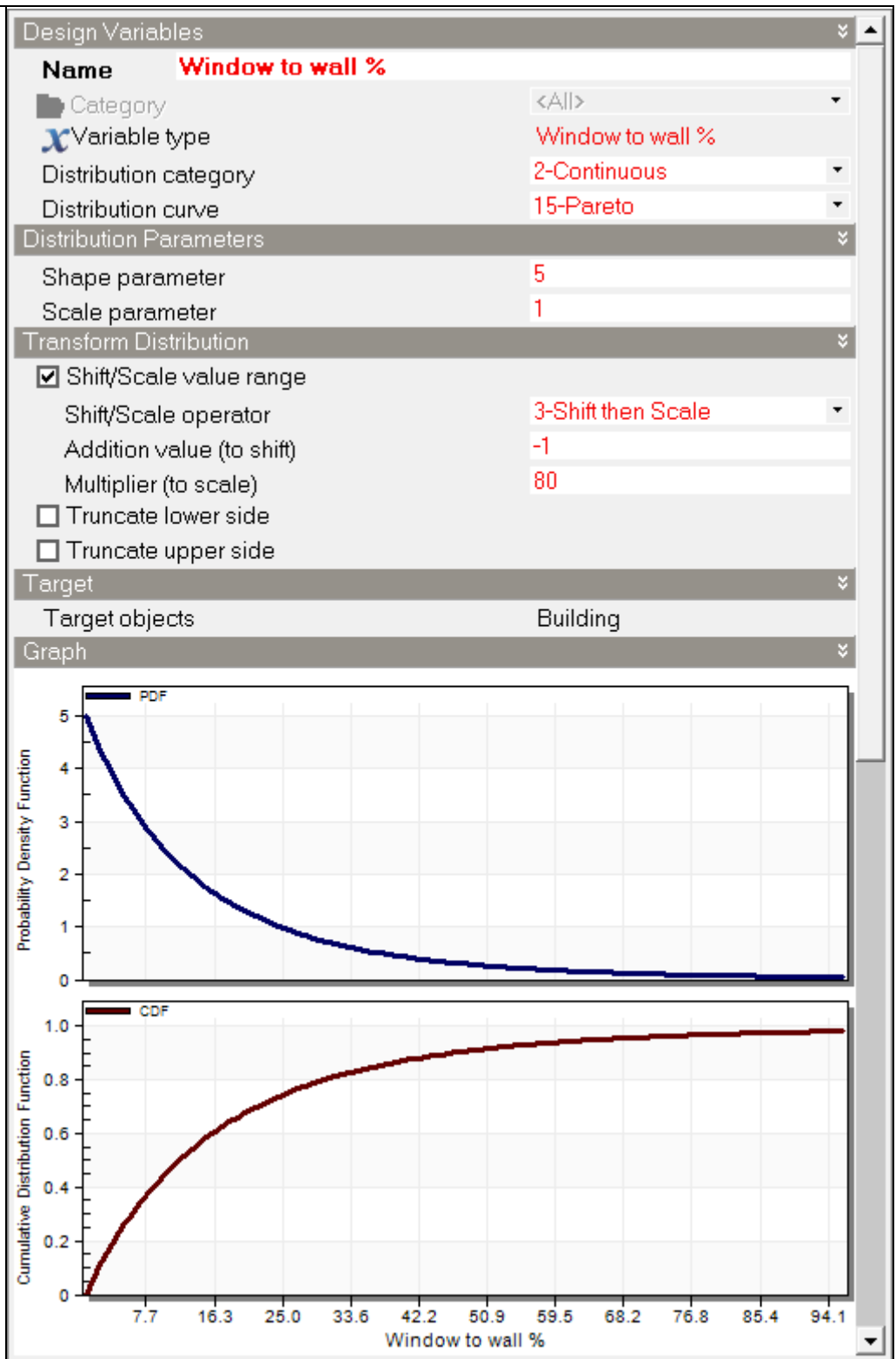
Truncate
Lower Side

Truncate
Upper Side

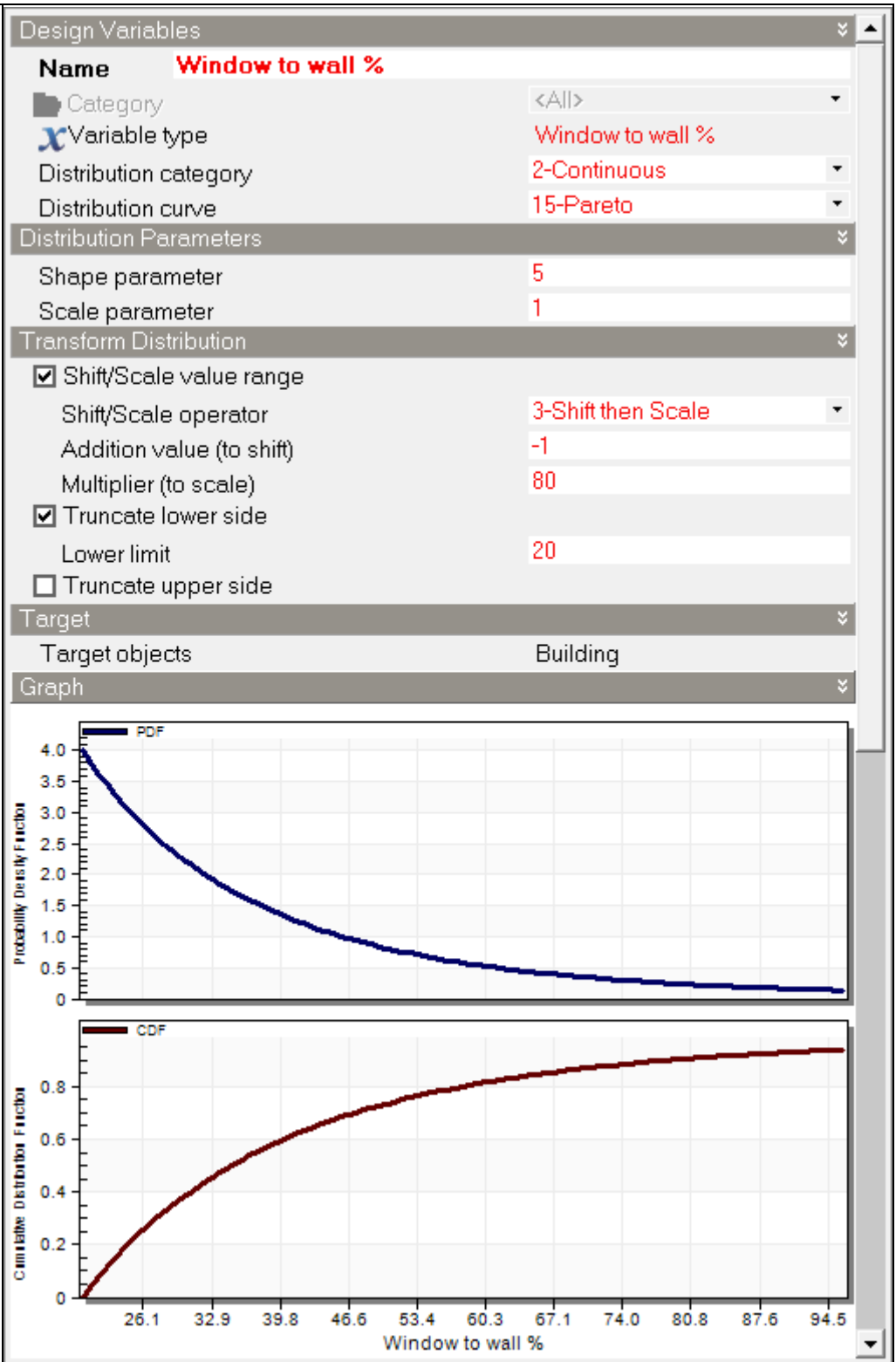


15 Pareto

Shift/Scale



Truncate
Lower Side

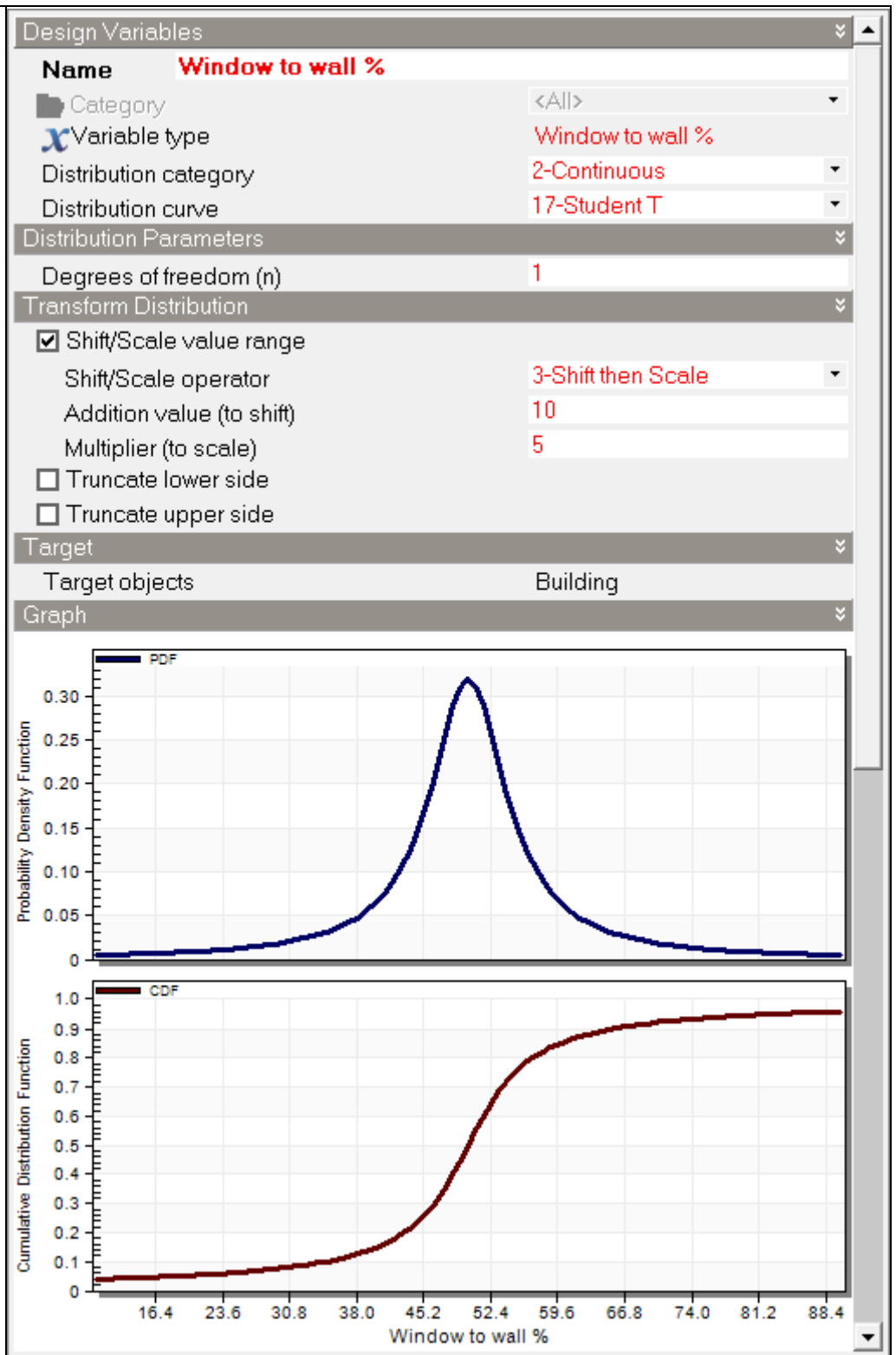


Truncate
Upper Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	15-Pareto
Distribution Parameters	
Shape parameter	5
Scale parameter	1
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	3-Shift then Scale
Addition value (to shift)	-1
Multiplier (to scale)	80
☒ Truncate lower side	
Lower limit	20
☒ Truncate upper side	
Upper limit	60
Target	
Target objects	Building
Graph	
The figure displays two statistical plots for the variable 'Window to wall %'. The top plot is the Probability Density Function (PDF), showing a blue curve that starts at a value of 5.0 on the y-axis and decreases as the x-value increases. The bottom plot is the Cumulative Distribution Function (CDF), showing a red curve that starts at 0 on the y-axis and increases towards 1.0 as the x-value increases. Both plots share the same x-axis, 'Window to wall %', with major tick marks at 23.2, 26.8, 30.4, 34.0, 37.6, 41.2, 44.8, 48.4, 52.0, 55.6, and 59.2.	

17 Student T

Shift/Scale



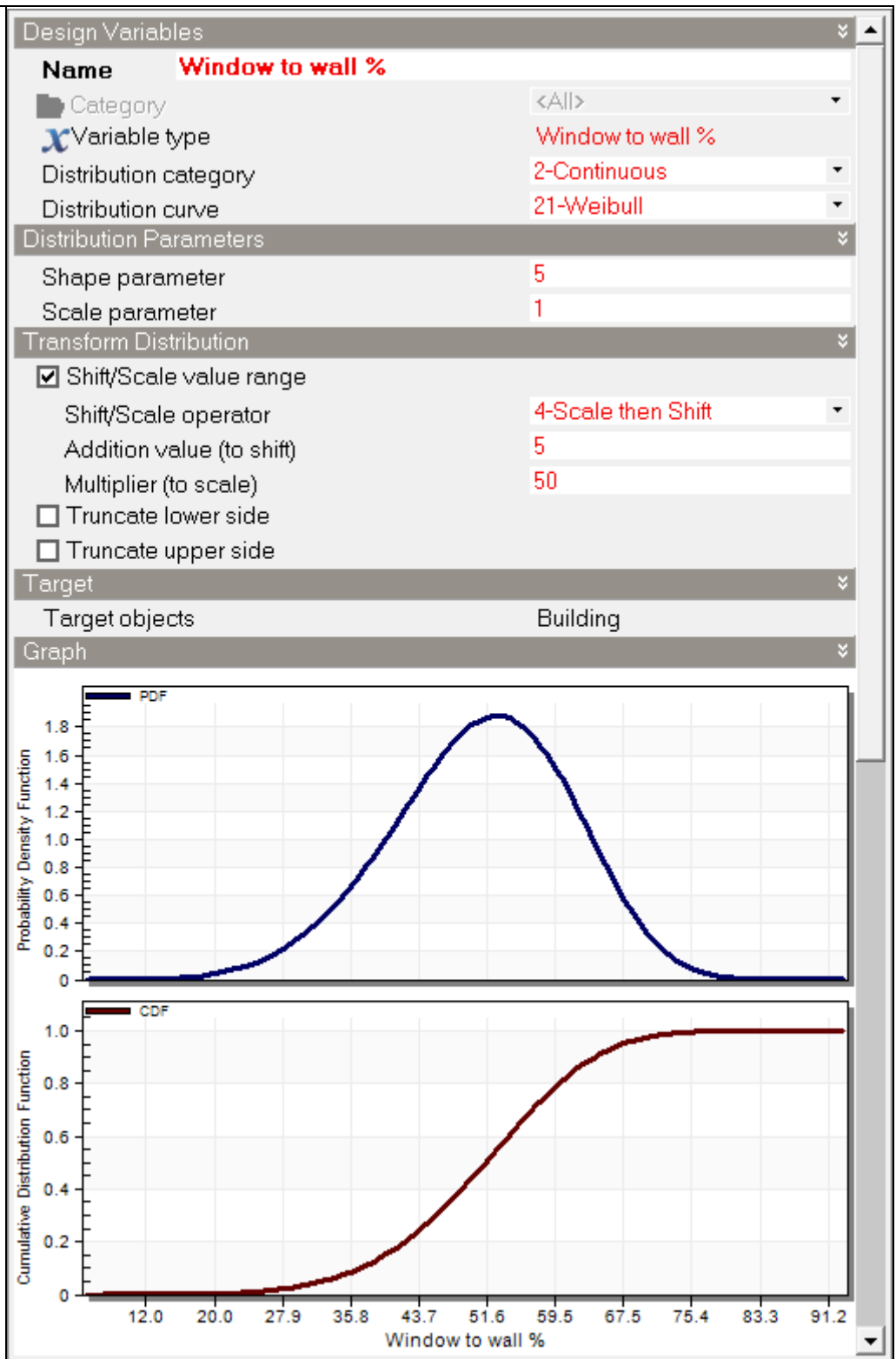
Truncate
Lower Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	17-Student T
Distribution Parameters	
Degrees of freedom (n)	1
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	3-Shift then Scale
Addition value (to shift)	10
Multiplier (to scale)	5
☒ Truncate lower side	
Lower limit	20
☐ Truncate upper side	
Target	
Target objects	Building
Graph	

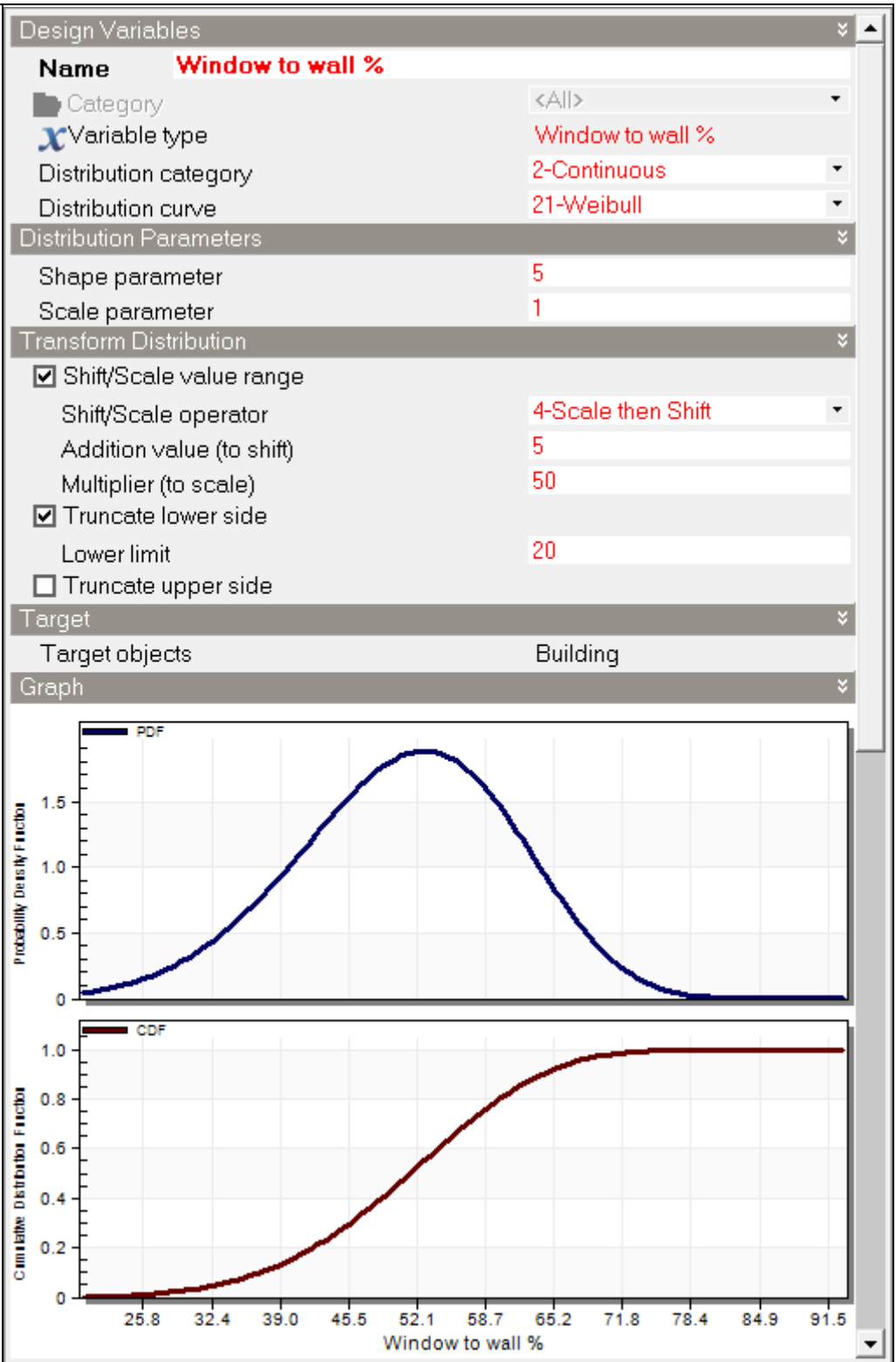
		Truncate Upper Side	Design Variables Name Window to wall % Category <All> Variable type Window to wall % Distribution category 2-Continuous Distribution curve 17-Student T Distribution Parameters Degrees of freedom (n) 1 Transform Distribution ☒ Shift/Scale value range Shift/Scale operator 3-Shift then Scale Addition value (to shift) 10 Multiplier (to scale) 5 ☒ Truncate lower side Lower limit 20 ☒ Truncate upper side Upper limit 60 Target Target objects Building Graph
18	Triangular	N/A	
19	Uniform (Continuous)	N/A	

21 Weibull

Shift/Scale



Truncate
Lower Side



Truncate
Upper Side

Design Variables	
Name	Window to wall %
Category	<All>
Variable type	Window to wall %
Distribution category	2-Continuous
Distribution curve	21-Weibull
Distribution Parameters	
Shape parameter	5
Scale parameter	1
Transform Distribution	
☒ Shift/Scale value range	
Shift/Scale operator	4-Scale then Shift
Addition value (to shift)	5
Multiplier (to scale)	50
☒ Truncate lower side	
Lower limit	20
☒ Truncate upper side	
Upper limit	60
Target	
Target objects	Building
Graph	

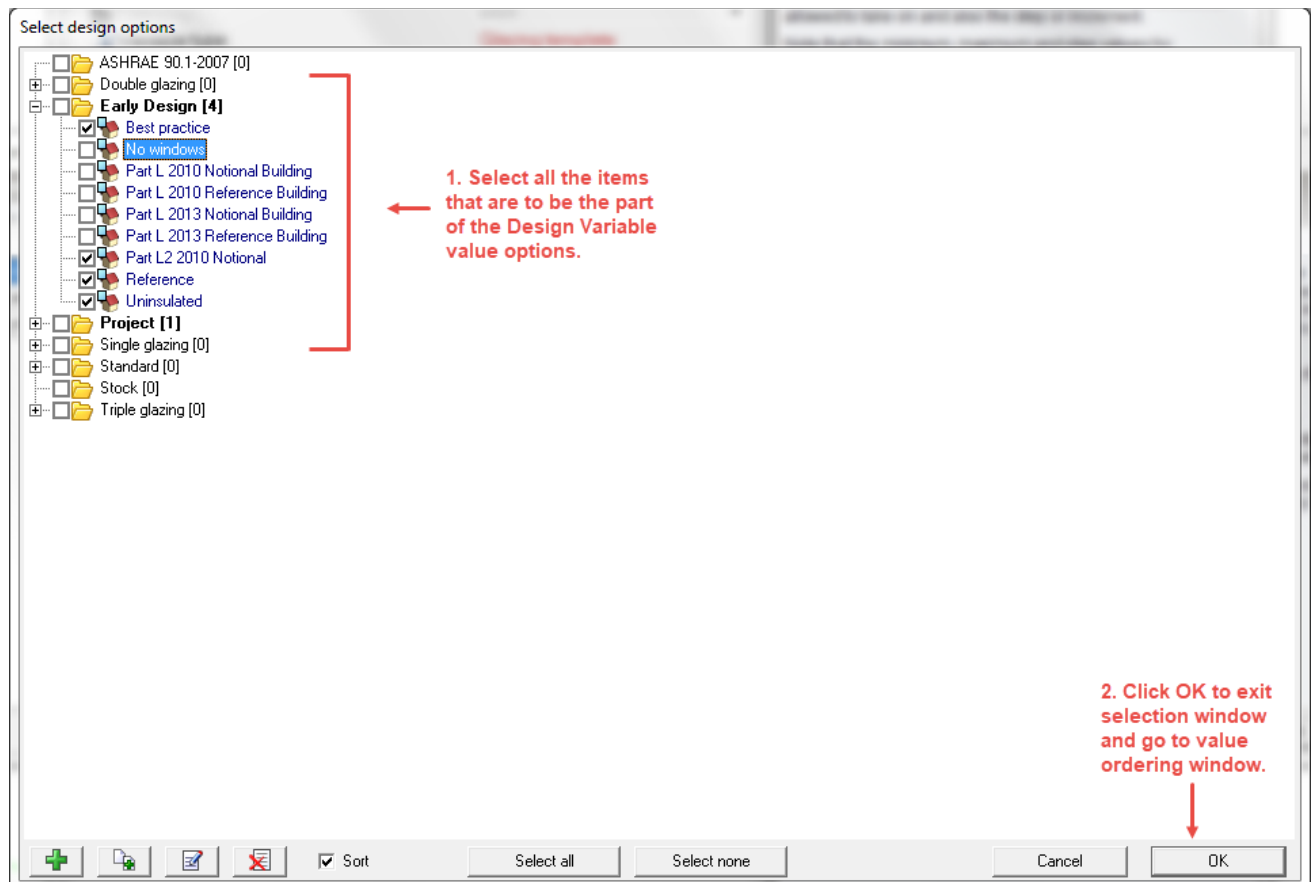
Warning when changing from SI to IP units: If any variables that were defined in SI units are edited, the transformations (shifting/scaling and truncations) will be reset.

Variable Values

Options list

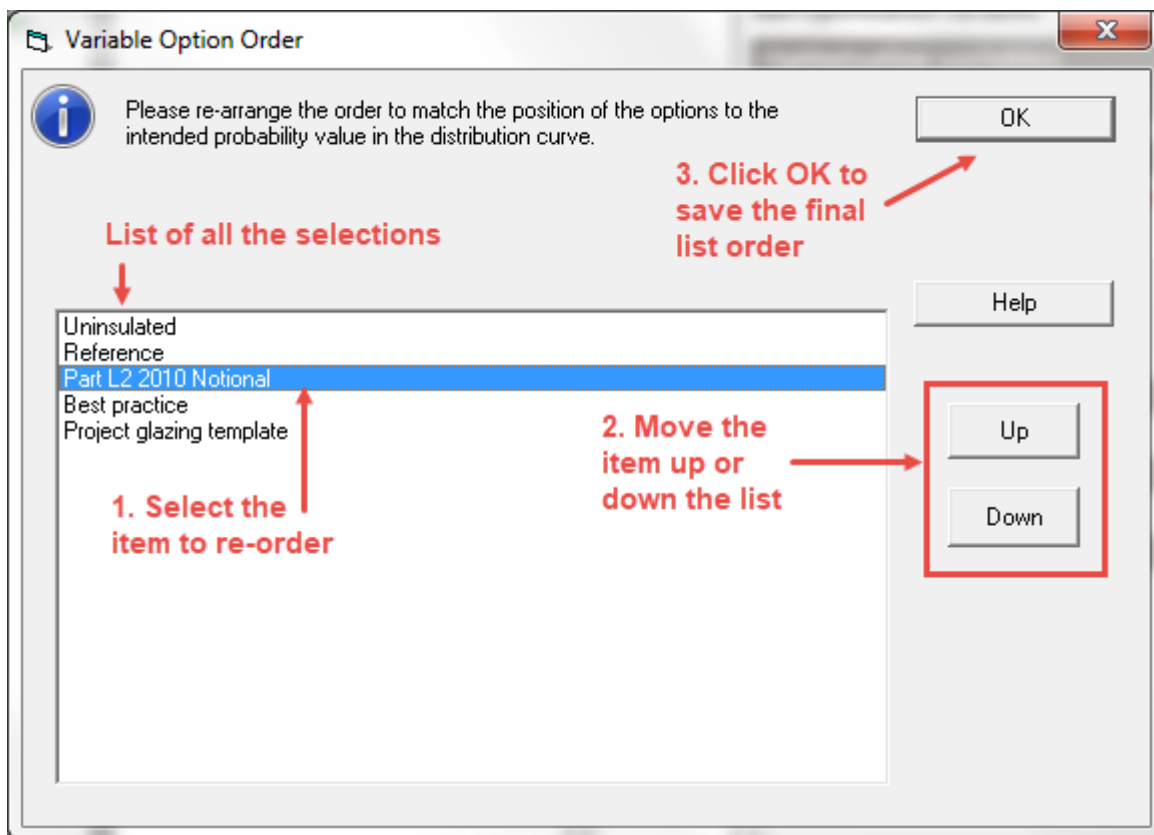
The Options list is only available for list type variables. It allows you to select non-numeric options and create a list of various variables that will be sampled from during the UA/SA.

Click on the browse button to open a list of components or templates for selection. In the new Select design option window, items can be selected by checking the appropriate checkboxes.



The number of items selected from the discrete distribution's items and truncates the distributions selected (if required).

After selecting the options, it is important to order the list, to match the unintended probability values of each of the section. This is done in the **Variable Option Order** dialog which is automatically displayed after adding/editing a list of variable options.



Important Note: The selection order must be set up to ensure that SA can assess the effect of incremental changes to values of the selected options on the main output of interest. For example, if the variable is wall construction with each construction having a different U-value (perhaps by varying insulation thicknesses), and if building heating energy is the SA output, then the constructions should be ordered based on (increasing or decreasing) U-value. This is because U-value is the construction attribute that has the greatest influence on heating energy. If you do not take care to order option lists in this way, then the SA results will not be meaningful. See also: "Running Sensitivity Analysis as a Precursor to Optimisation" on page 1867.

Target

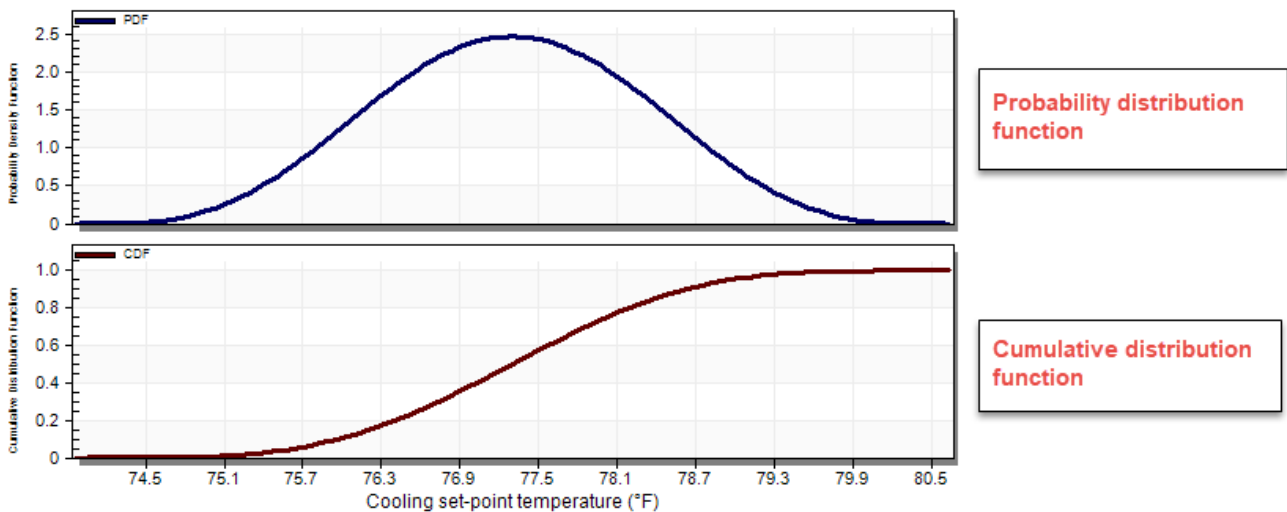
Target objects

This setting allows you to select one or more object(s) to which the variations are to be applied. The object(s) you select here are the places in the model where the variations will be applied. Normal model data inheritance rules apply so if you set the building as the target then the change will set for the building but will also inherit down to block, zone, surface and opening levels where appropriate. In this case any hard-set data at block or lower levels will prevent inheritance in the normal way.

Tip: For models imported via [gbXML](#) any construction or glazing assignments are set in the DesignBuilder model at surface level and so the normal inheritance paths used for construction and glazing will be overridden. Therefore in this case, if you need to parametrically adjust construction or glazing surface properties you may need to apply the changes to each surface individually by selecting the appropriate surfaces as targets.

Graph

This is the visual representation of the probability curve and shows the probability distribution curve/ probability mass curve and cumulative distribution curve. To update the graph to display the current settings click on the **Create/Update Graph** button.



Uncertainty And Sensitivity Analysis - Calculation Options Dialog

To start an Uncertainty and Sensitivity analysis go to the Simulation screen and click on **the Optimisation + UA/SA** tab. From there calculations can be updated using the **Update** toolbar icon in the usual way. The data on the Calculation Options dialog is used to provide options to control the analysis as well as outputs to help you to understand the progress being made as the calculations proceed.

Note: It is necessary to run the base simulation with monthly/annual outputs before opening the optimisation tab. The Uncertainty and Sensitivity analysis simulations will all be run using the base calculation and model options. So, the simulation period and all other options used in the optimisation is the same as that defined on the Simulation calculation options dialog.

There are 2 tabs on the Dialog:

- [General](#)
- [Simulation Manager](#)

A sample uncertainty analysis result is shown within the Calculation Options dialog. Once the analysis has completed, detailed [Uncertainty and Sensitivity Analysis results](#) are best visualised on the **Optimisation + UA/SA** Tab on the Simulation Screen.

You can learn about the results generated on the graph on this dialog and other results during the Uncertainty and Sensitivity analysis from the "Uncertainty and Sensitivity Results" on page 1853 pages.

Uncertainty And Sensitivity Analysis - Calculation Options General Tab

Accessed from [UA/SA Calculation Options Dialog](#)

The UA/SA Sampling Algorithm used by DesignBuilder requires a number of options to be set to control the way that the input variables are sampled. These and other settings are made on the General tab of the Calculation Options dialog as described below.

Calculation Description

Enter a description for the calculation. This text will be used to identify the results on the graph and any other related outputs.

From

The **Start date** set up for the base simulation on the analysis screen is displayed but cannot be edited here.

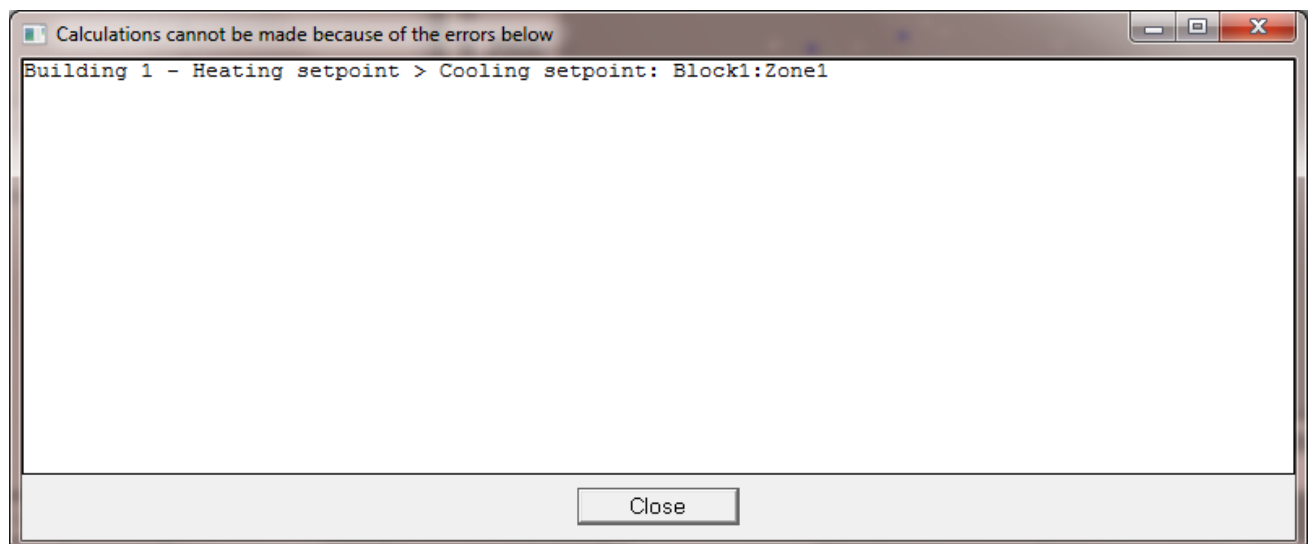
To

The **End date** set up for the base simulation on the analysis screen is displayed but cannot be edited here.

Uncertainty/Sensitivity Analysis Options

Pause on errors

It is important to understand the cause of any errors that occur during an analysis. The Pause on errors option, when checked, causes a message to be displayed and the analysis is paused. For example, if there is an overlap in heating and cooling setpoint temperatures, in configurations where the heating setpoint is higher than the cooling setpoint the following error message is displayed:



This sort of error does not cause a problem for the overall Uncertainty and Sensitivity Analysis study. When calculating Uncertainty and Sensitivity analysis results these erroneous simulations are removed from the calculations.

Once you are clear about the cause of any errors you will normally want to stop seeing reports and allow the optimisation to continue. To do this simply uncheck the **Pause on errors** check box. Simulations with errors are displayed in red in the grid as shown below.

Iteration	CO2 (kg)	Discomfort (All Clothing) (hr)	Window to Wall %	Cooling setpoint temperature (°C)	Heating setpoint temperature (°C)
199	10283.216	1569.500	48.327	24.494	20.375
198	11146.783	1574.000	43.651	19.162	14.309
197	7666.098	1561.000	11.930	25.229	13.991
196	11242.471	1320.500	58.453	22.376	18.811
195	9613.218	1147.500	11.930	21.032	20.989
194	11402.694	1149.500	55.341	20.948	17.323
193	10674.736	1429.000	52.959	23.289	19.121
192	100000000.000	8751.000	31.313	20.573	20.777
191	9366.212	1866.500	43.786	25.183	17.216
190	9610.643	1387.000	35.807	23.784	19.006
189	11348.725	1404.500	45.029	19.661	17.454
188	10328.332	1175.500	39.310	21.819	18.468
187	8727.377	1970.500	29.528	26.152	19.121
186	11008.166	1403.500	57.805	22.862	18.677
185	10217.033	2001.500	56.211	25.651	19.889
184	8178.473	1828.500	23.789	25.480	15.085

Common causes of errors are:

- Invalid combinations of variables such as heating setpoint > cooling setpoint. These errors are trapped by DesignBuilder before a simulation is run.

Tip: Truncation can be used to limit such instances by removing the overlapping tails of the conflicting distributions' inputs.

- Failure of a simulation due to errors in input that were not trapped by DesignBuilder.
- A severe EnergyPlus error such as simulation instability.
- A failure in the simulation infrastructure, e.g. a network failure when using a network simulation server.

Quietly retry if server is unavailable

Normally, if DesignBuilder is unable to access the requested Simulation Server, it will ask you whether you would like to **Abort**, **Retry** or **Ignore** the missing server. Selecting **Abort** cancels the whole analysis. Selecting **Retry** or **Ignore** instructs the software to keep trying. The latter option would be selected if the server became temporarily unavailable, perhaps due to a network error or because the server was down for a while.

However, if you are running a long analysis and you don't want to be prompted in this way due to a temporary glitch in the availability of your simulation server then you can check this option. This is similar to pre-selecting the retry or ignore options in advance.

Uncertainty/Sensitivity Analysis Options

Number of runs

This is the total simulations that are required. The number of runs to be used will determine the time and computing resources required to complete the analysis. The value entered here will usually reflect the size and complexity of the analysis. Typical values are in the range 50-500 depending on size of the problem, number of design variables and the population size.

The two most important factors to consider while selecting the value here are the [Sampling option](#) and the number of design variables. The table below can be used to select the minimum number of runs.

Sampling Option	Number of Runs - Rule of thumb
Random	20 times the number of variables
Random Walk	Not used. Define number of trajectories instead.
LHS	10 times the number of variables
Sobol	15 times the number of variables
Halton	15 times the number of variables

For cases where any of the design variables are discrete and have more than 10 options, then a higher value should be used to ensure that all options are simulated.

Tip: It is important to set number of runs correctly from the outset if a sensitivity analysis is required. In cases when a run is terminated, the sensitivity analysis results will not be shown and uncertainty analysis results may be incomplete. If sensitivity analysis results are not required then you can simply terminate the calculations once you are happy that the distribution curve displayed on the Calculation Options dialog during the analysis is sufficiently well developed and smooth.

Max runs per batch

The value here sets the maximum number of runs that are run in one set and gets plotted on the graph before the next set begins. Default value of results of simulations are plotted in batches is 20.

Note: All simulations for a set must complete before the next set is started so if you have many parallel cores at your disposal and the time taken to run the simulations is the main bottleneck in calculations (as opposed to IDF generation) then a higher number here can be helpful.

Sampling option

DesignBuilder supports Uncertainty and Sensitivity Analysis through implementing various sampling methods. DesignBuilder supports five sampling methods:

- **1-Random:** based on a Pseudo-random sample where a sample is created by shuffling the full list of permutations and selecting the first N cases. This method is only available to problems whose search space size is no greater than 100,000. Unlike the other sampling options this method has the advantage that it allows sensitivity analysis to be carried out even if the analysis is interrupted before all cases have been simulated.
- **2-Random walk:** a simple symmetric method where the number of trajectories (N) determines the number of starting points, which is generated by Latin hypercube sampling. The number of variables selected equals the number of parameters (D) plus 1. So, the total sample size is $N \times (D + 1)$.
- **4-LHS:** Latin hypercube sampling is a highly efficient sampling method. As a rule of thumb, a sample size of 10 times the number of design variables will be sufficient for the population mean to be accurately modelled, i.e. for the mean values of the samples to be close to mean value of requested distribution range. For example if we request a mean temperature of 22°C with standard deviation of 2, and we request 100 runs then the 100 samples generated by LHS sampling method will have a mean value very close to 22°C.
- **5-Sobol:** a quasi-random [low dependency sequence](#).
- **6-Halton:** a quasi-random [low dependency sequence](#).

Sensitivity method

The Sensitivity Analysis method implemented in DesignBuilder is [Regression](#).

Number of trajectories

Only required for the Random Walk sampling method. The value is multiplied with total variables to generate the total runs.

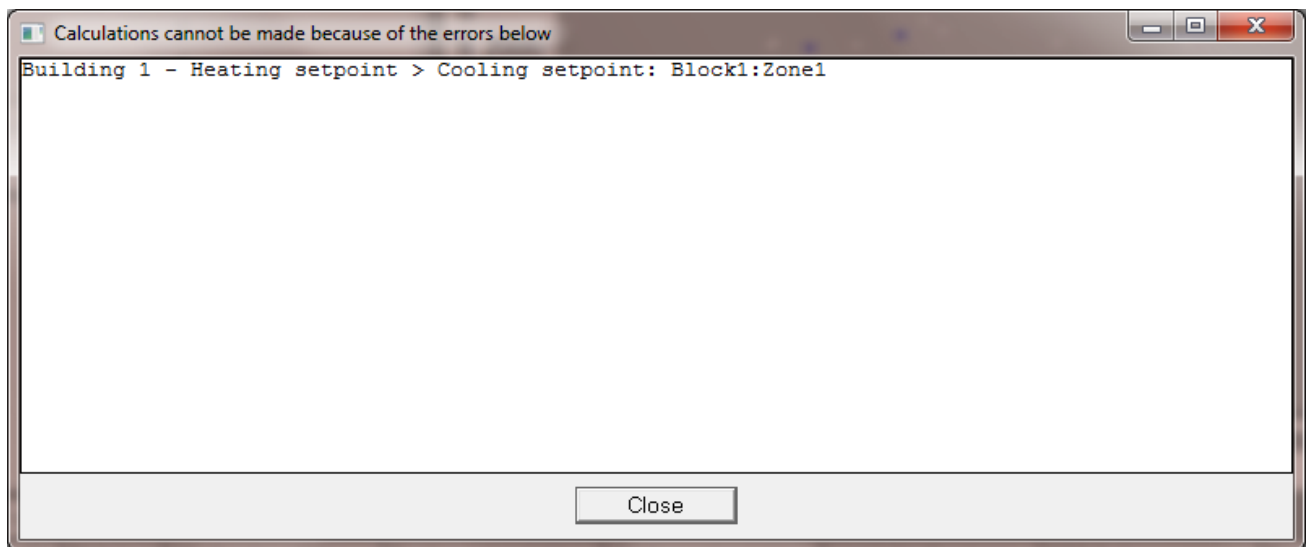
$$\text{Runs} = \text{Trajectories} * (\text{Variables} + 1)$$

The recommended value for Number of trajectories is 20 if there are 10 or less number of design variables. For more than 10 variables this can be reduced in order to meet the computational demands.

Advanced Options

Pause on errors

It is important to understand the cause of any errors that occur during an optimisation. The **Pause on errors** option, when checked, causes a message to be displayed and the optimisation is paused. For example if there is an overlap in heating and cooling setpoint temperatures, in configurations where the heating setpoint is higher than the cooling setpoint the following error message is displayed:



This sort of error does not cause a problem for the overall UA/SA study.

Once you are clear about the cause of any errors you will normally want to stop seeing reports and allow the study to continue. To do this simply uncheck the **Pause on errors** check box. Simulations with errors are displayed in red in the grid as shown below.

Iteration	Generati...	Status	Window to wall %	Cooling set-point te...	Heating set-point te...	Local shading type	CO2 (kg)	Discomfort ASHRA...
72	5		43,000	25,300	23,400	1.0m Overhang	22915,129	665,5000
71	5		23,000	25,600	16,700	0.5m Overhang	19739,116	1800,5000
70	5		42,000	25,900	17,100	2.0m Overhang	19846,498	1927,0000
69	5		60,000	24,800	21,400	2.0m Overhang	22878,883	1312,0000
68	5		41,000	26,400	23,300	2.0m Overhang	21801,074	900,5000
67	5	Calculation error	38,000	22,400	22,600	1.5m Overhang	1000000000,000	8761,0000
66	5	Optimal Designs	41,000	27,400	22,600	2.0m Overhang	20992,442	1214,0000
65	5		49,000	24,400	20,900	2.0m Overhang	22200,984	1297,5000
64	5		62,000	27,400	20,000	1.0m Overhang	21769,323	2240,5000

Common causes of errors are:

- Invalid combinations of variables such as heating setpoint > cooling setpoint. These errors are trapped by DesignBuilder before a simulation is run.
- Failure of a simulation due to errors in input that were not trapped by DesignBuilder.
- A severe EnergyPlus error such as simulation instability.
- A failure in the simulation infrastructure, e.g. a network failure when using a network simulation server.

Quietly retry if server is unavailable

Normally, if DesignBuilder is unable to access the requested Simulation Server, it will ask you whether you would like to **Abort**, **Retry** or **Ignore** the missing server. Selecting **Abort** cancels the whole analysis. Selecting **Retry** or **Ignore** instructs the software to keep trying. The latter option would be selected if the server became temporarily unavailable, perhaps due to a network error or because the server was down for a while.

However, if you are running a long analysis and you don't want to be prompted in this way due to a temporary glitch in the availability of your simulation server then you can check this option. This is similar to pre-selecting the retry or ignore options in advance.

Retain simulation manager results

Normally results for each simulation are held within the Simulation Manager system only for as long as required and are deleted once the key results have been loaded into DesignBuilder. However, if you require results for each simulation to be retained within the Simulation Manager system then check this checkbox.

Backup results table each generation

If you are concerned about the stability of the optimisation process (e.g. due to memory issues) you can select this option to ensure that results are stored at the end of each generation, allowing you to later retrieve partially completed results.

Discussion On Selecting Settings

An important and often overlooked consideration is the impact of the time taken by DesignBuilder to generate the IDF input file for each design. The longer this is relative to simulation time the greater the model generation bottleneck and the less useful the large number of parallel simulations. On the other hand, the larger and more complex the model and the more detailed the simulations (nat vent, detailed HVAC, more time steps etc) the more time is taken in simulation and relatively less in IDF generation. For an extremely large/complex model, all IDF inputs will have been generated before the first results are in and the large number of parallel simulations will speed progress. For a smaller model, the IDF generation bottleneck is more significant and first simulation results will be in before the 3rd or 4th IDF input has been generated and, in this case, the multiple cores will not be needed. This is often the case for simple single zone models. To test and understand this watch the Simulation Manager while the optimisation takes place. You will see new jobs being submitted, queued and simulated.

Uncertainty And Sensitivity Analysis - Calculation Options Simulation Manager Tab

Accessed from [UA/SA Calculation Options Dialog](#)

See "Optimisation Calculation Options - Simulation Manager Tab" on page 1729 for details on how to configure the Simulation Manager settings for UA/SA.

Uncertainty And Sensitivity Results

Uncertainty and Sensitivity analysis (UA and SA) calculations are carried out together. When the analysis is running, the uncertainty results of one of the outputs can be seen in real time. When the analysis has finished and the UA/SA calculation dialog has been closed, the results are plotted in the main simulation screen. Results can be seen either in histogram form or as raw grid data, or both.

Display Options: Uncertainty/Sensitivity Analysis

The data to be displayed is controlled by making settings in the Display options panel on the bottom left of the screen as described below.

Show analysis

Check this checkbox to view a histogram graphic showing uncertainty or sensitivity analysis results.

Analysis type

Here Display options settings can be used to see:

- 1- Uncertainty Analysis results
- 2- Sensitivity Analysis results

Display type

Results can be seen as a graph or report.

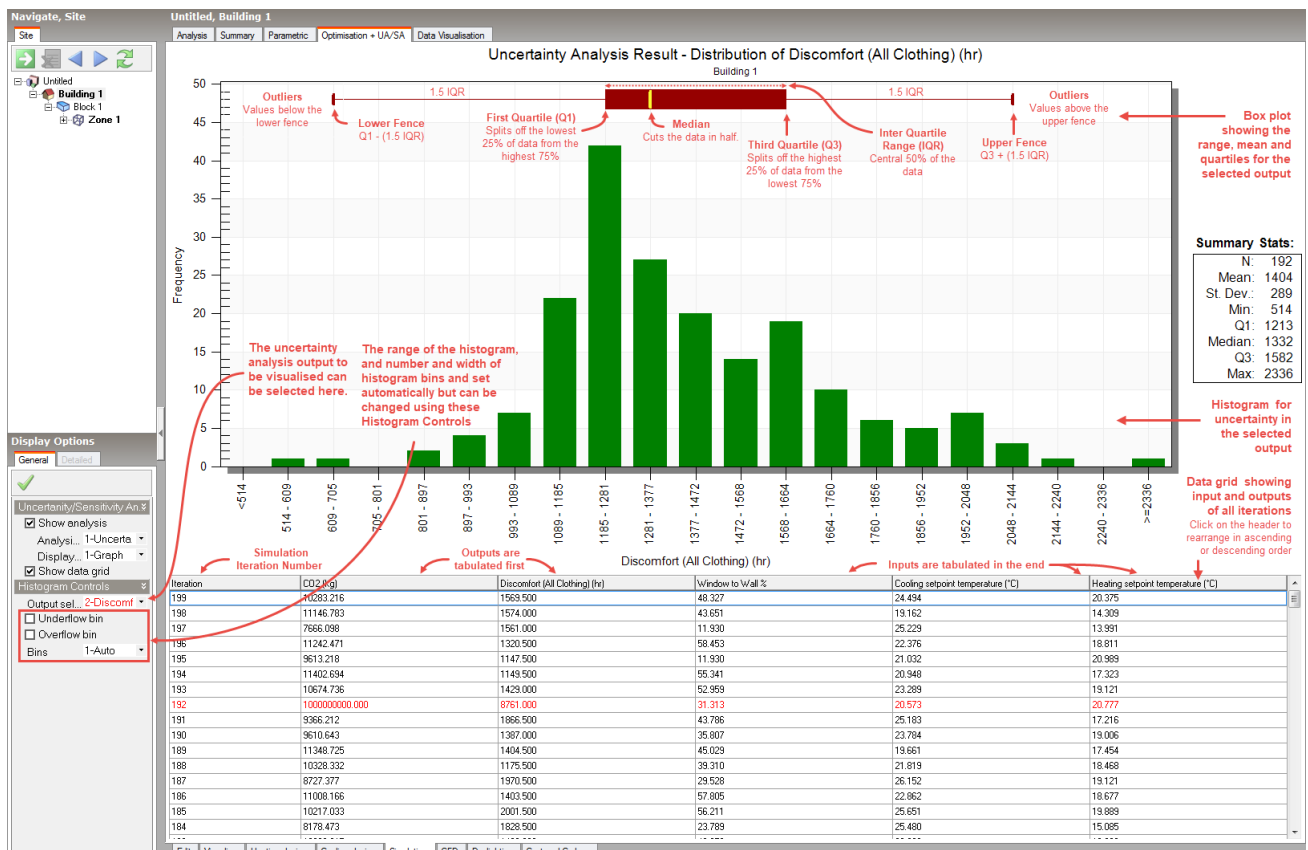
- [1-Uncertainty Analysis Outputs](#)
 - o 1- Graph (Histogram)
 - o 2- Report
- [2-Sensitivity Analysis Outputs](#)
 - o 1- Graph (SRC)
 - o 6- Report

Show Grid Data

Check this option to view raw results for each iteration in a grid.

Uncertainty Analysis Outputs

After finishing the uncertainty and sensitivity analysis, uncertainty analysis results can be accessed from Simulation screen by setting the Analysis type to **1-Uncertainty Analysis** and the Display type to **1-Graph**. The screenshot below shows the main features of the uncertainty analysis graph.



Note: Depending on the number of variations being simulation you may notice that some duplicate cases are reported. This is because uncertainty is based on probability of design permutations happening, so it is necessary to count all the repeated instances to obtain the correct probability of a particular output. If we were to remove all the duplicates then once the entire search space is simulated the results would be

more evenly spread out, which would be incorrect. For more "efficient" sampling you should consider using the LHS option to ensure that samples are more equally distributed.

Histogram Display Options

When the Display type is set to 1-Graph the Histogram plot can be controlled by making settings under the Histogram Controls header on the Display Options panel.

Output selected

The option selected here determines which output is plotted in the histogram. The list contains all available Outputs that were requested as a part of the uncertainty analysis run.

Underflow bin

Select this checkbox to create a bin for all values below or equal to the value in the box to the right.

Value

To change the value, enter a different decimal number in the box. It must be more than the overflow value.

Overflow bin

Select this checkbox to create a bin for all values above the value in the box to the right.

Value

To change the value, enter a different decimal number in the box. It must be less than the underflow value.

Bins

The histogram can be created based on following configuration for the bins:

- **1-Auto:** This is the default setting for histograms. The bin width is calculated to as to have 20 bins in the graph.
- **2-Bin width:** Enter a positive decimal number for the number of data points in each range and the total number of bins are adjusted accordingly.
- **3-Number of Bins:** Enter the number of bins for the histogram (including the overflow and underflow bins) and the bin widths are created accordingly.

Note: Sometimes the number of bins is more than what can be displayed on the screen properly and in such cases a horizontal scroll bar is added to the bottom of the graph to allow all of the data to be seen.

Uncertainty Analysis Report

After finishing the uncertainty and sensitivity analysis, a report detailing the results of the uncertainty analysis can be viewed on the screen by setting Analysis to **1-Uncertainty Analysis** and Display Type to **2-Report**.

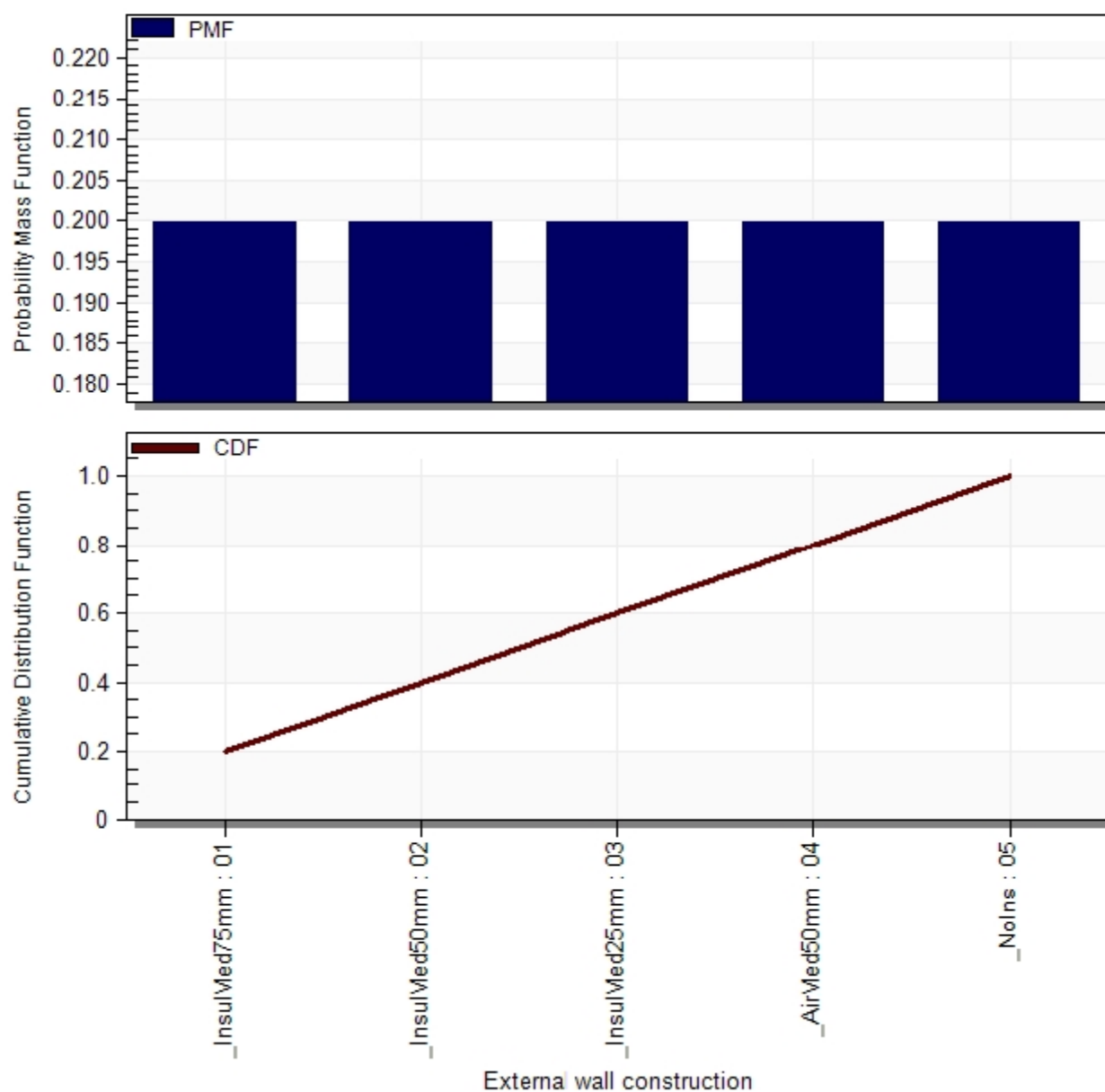
The uncertainty analysis report consists of following sections:

Input Variables: Details of all variables used in the analysis including their distribution curves.

The Input variables report lists the name, units, probability distribution type and options used in the UA and the level at which the variable was applied (e.g. "Building"). The screenshot below shows as example graphic for 1 variable in the analysis.

1.2 Input variable: External wall U-Value

Input Type:	External wall construction
Input Units:	No Units
Probability Distribution:	20-Uniform(Discrete)
Distribution Characteristics:	Prob: 0.200; Options: 5
Level:	Building



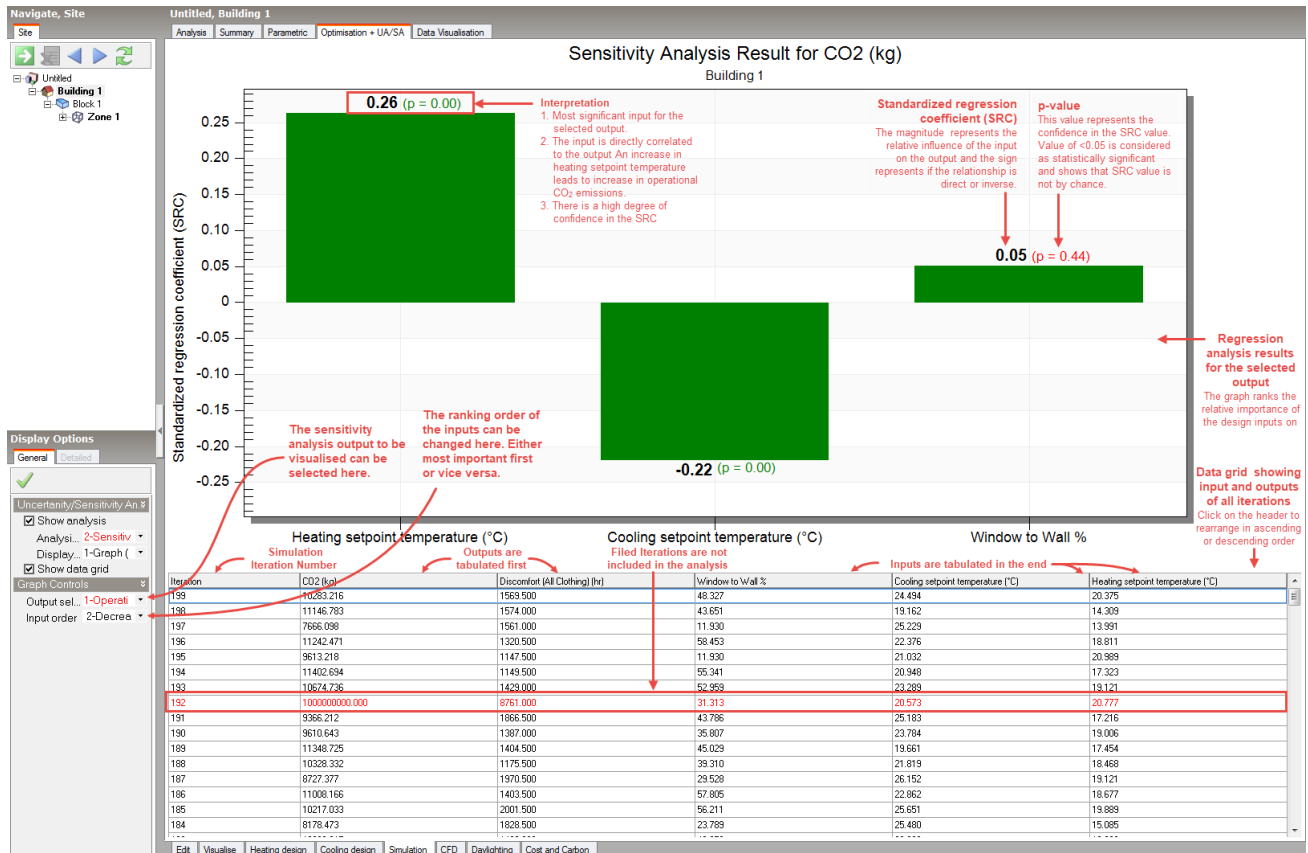
Outputs (KPIs): Details of the outputs analysed.

Analysis Information: Analysis summary, details about the input samples.

Uncertainty Analysis Results: Graphs and statistical details showing the variability in the output.

Sensitivity Analysis Outputs - Standardised Regression Coefficient

After finishing the uncertainty and sensitivity analysis, sensitivity analysis results can be accessed from the Simulation screen, by setting the Analysis type to **1-Sensitivity Analysis** and Display type to **1-Graph (SRC)**. The screenshot below shows the main features of the sensitivity analysis results graph.



Graph Controls

Graph plots one output at a time.

Output Selected

Option selected here determines which output is plotted in the graph. List contains all available outputs that were requested as a part of uncertainty analysis run.

Input Order

This option helps in sorting the most sensitive outputs:

- **1-Auto:** Default order based on the order of inputs generated.
- **2-Decreasing importance:** Reorders the design variables to list the most important ones first.
- **3-Increasing importance:** Reorders the design variables to list the least important ones first.

Sensitivity Analysis Report

After finishing the uncertainty and sensitivity analysis, the Sensitivity Analysis Report can be accessed from the Simulation screen, by setting the Analysis type to **1-Sensitivity Analysis** and the Display type to **6-Report**. The report consists of following sections:

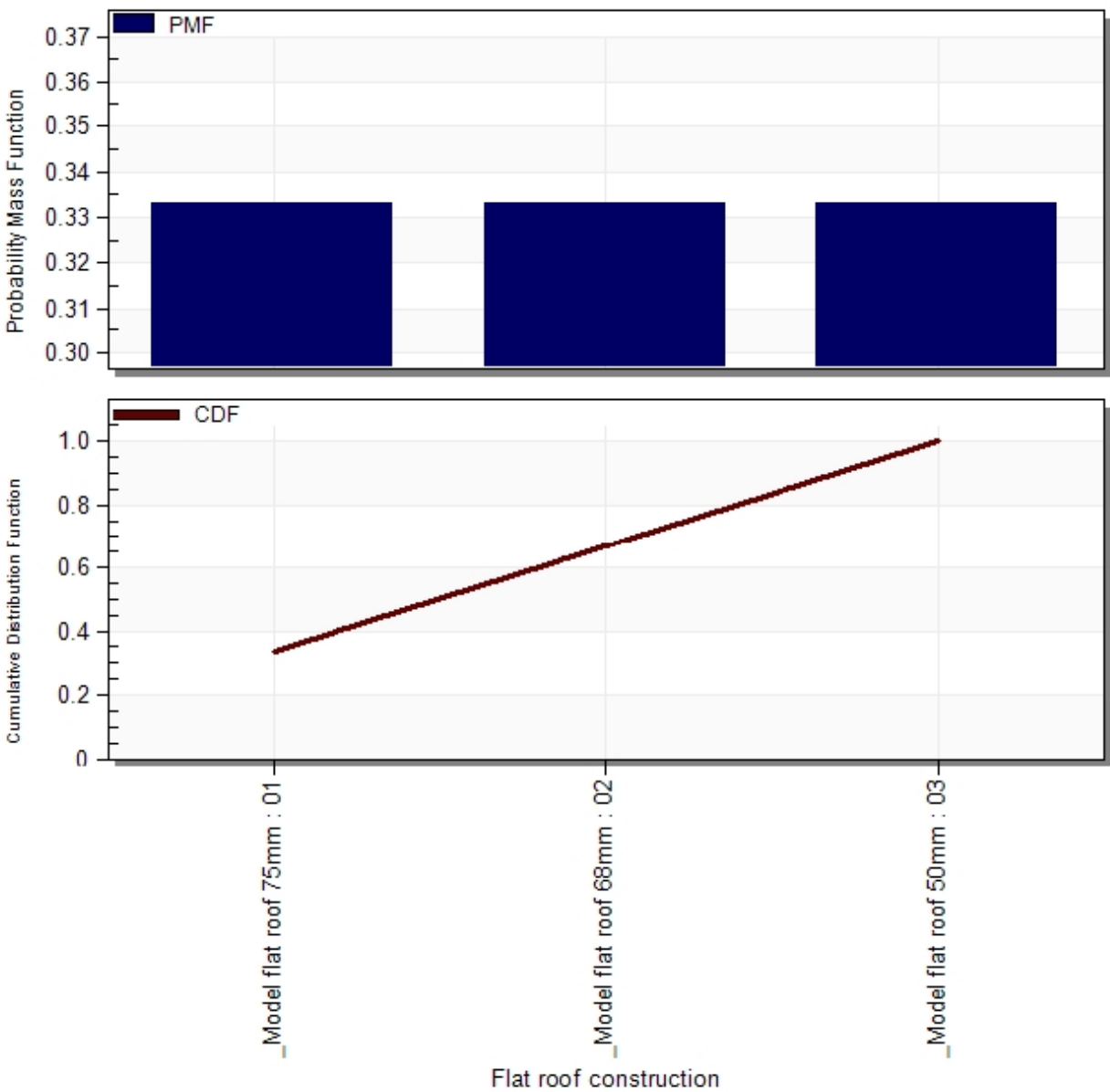
Input Variables: Details of all design variables with their distribution curves

The report shows input details of all design variables with their distribution curves (below). These requested distribution curves were input to the UASA engine. The actual distribution curves for the analysis are shown below under Analysis Information. If there were enough simulations in the analysis then the 2 distribution profiles should roughly match.

Example Input Variable report for a discrete variable:

1.9 Input variable: Roof U-Value

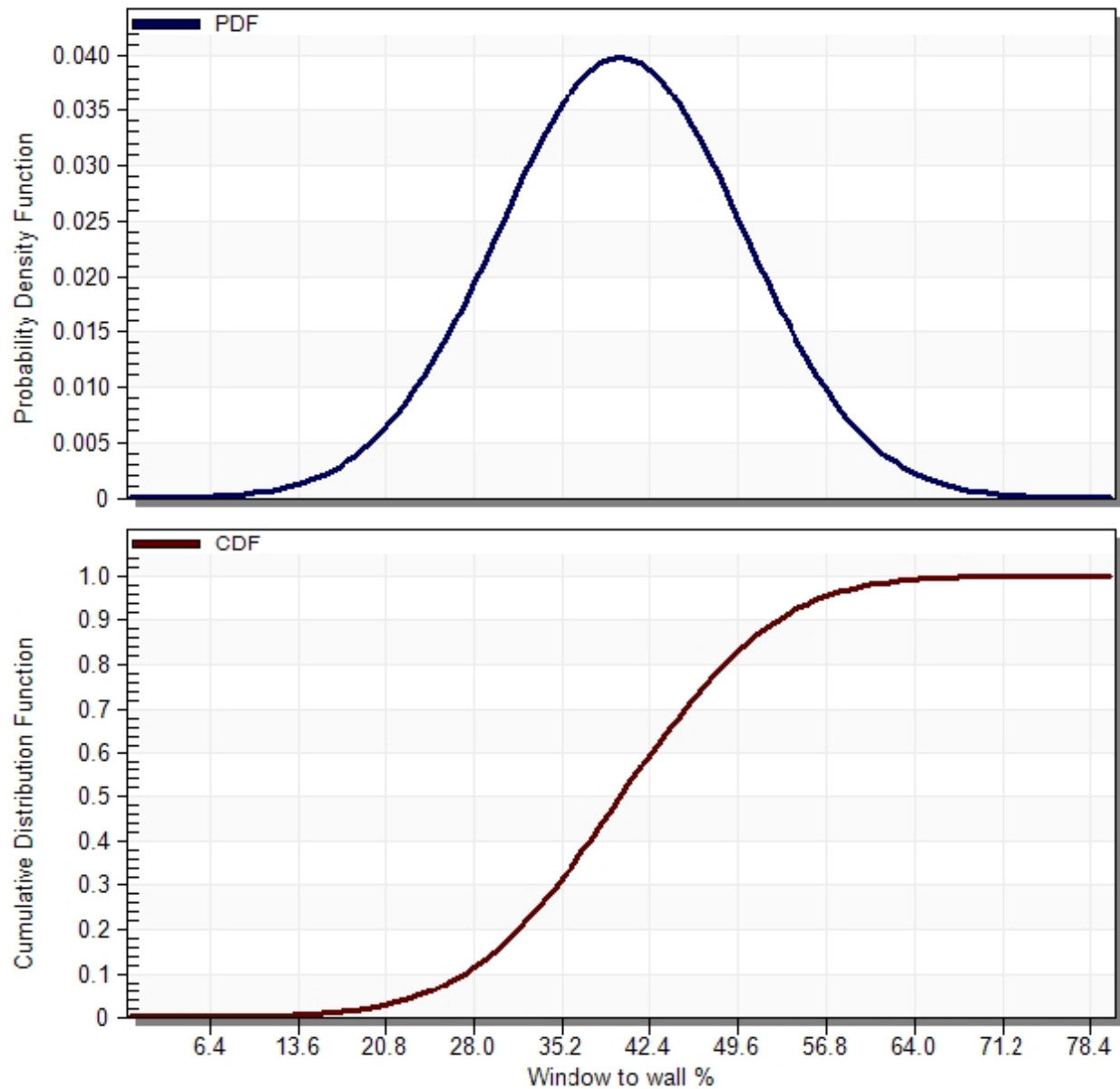
Input Type:	Flat roof construction
Input Units:	No Units
Probability Distribution:	20-Uniform(Discrete)
Distribution Characteristics:	Prob: 0.333; Options: 3
Level:	Building



Example Input Variable report for a continuous variable requesting a normal distribution:

1.1 Input variable: Window to Wall %

Input Type:	Window to wall %
Input Units:	No Units
Probability Distribution:	14-Normal
Distribution Characteristics:	Mean: 40.00; Std. Dev.: 10.00
Level:	Building



Outputs (KPIs): Details of all of the outputs analysed

The next section lists of all of the outputs analysed. In the example below the analysis follows best practice in running a separate analysis for each KPI. This is why there so there is only one output listed.

2. Outputs

1 output KPI was selected. Below are the details of the output.

2.1 Output: Cooling Elec (Cooling (Electric))

Output Type:	Cooling (Electric)
Output Units:	kWh

Then for each output the following sections are included.

Analysis Information: Analysis summary, details about input samples and output uncertainty

The screenshots below shows examples of the input sample details section.

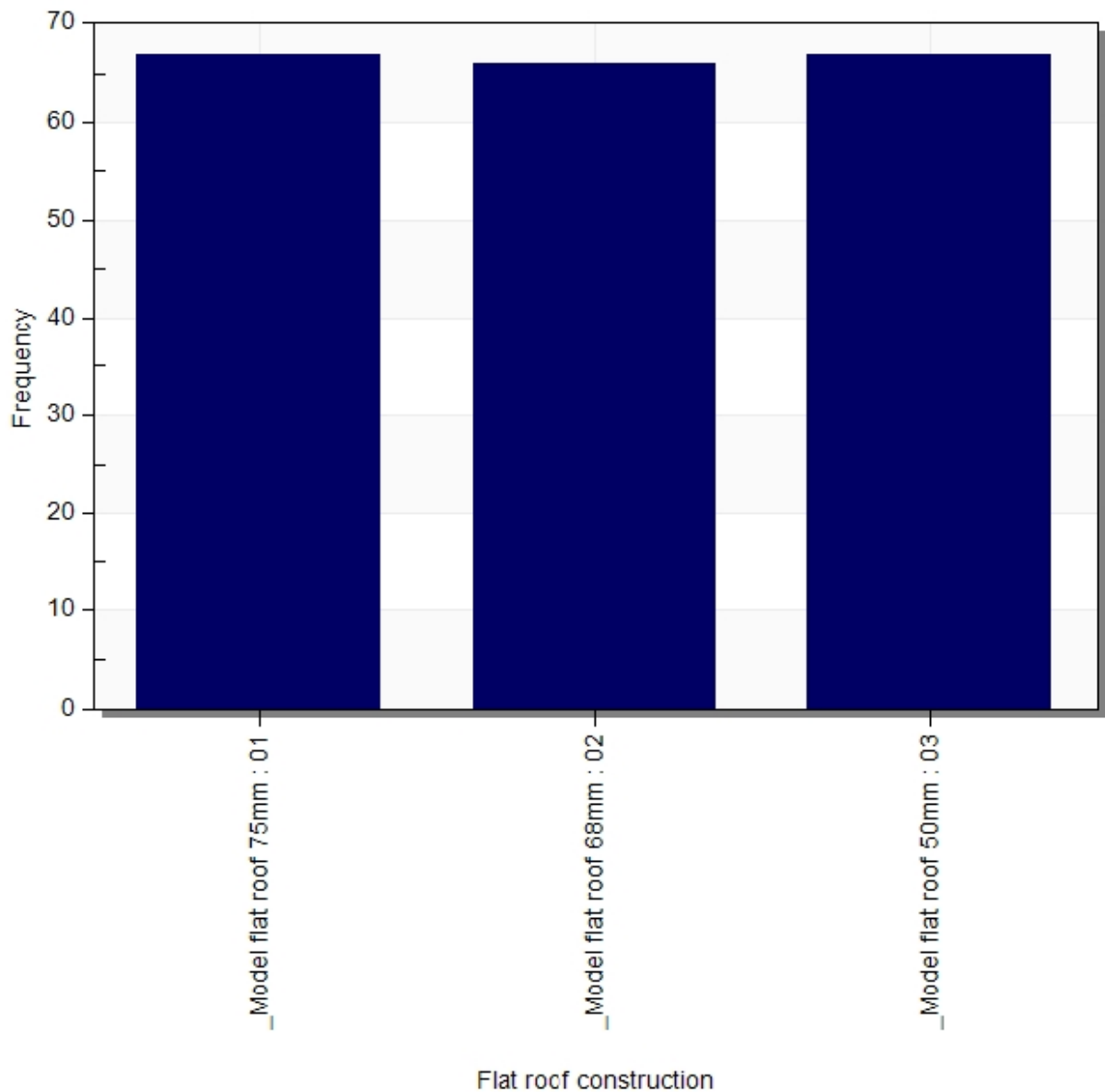
Example sampling details for a discrete variable:

3.2.9 Input variable: Roof U-Value

3.2.9.1 Input Options' Frequency: Roof U-Value

Option Name	Frequency
_Model flat roof 75mm : 01	67
_Model flat roof 68mm : 02	66
_Model flat roof 50mm : 03	67

3.2.9.2 Frequency Distribution Graph: Roof U-Value



This is the simulated distribution frequency for the same "Roof U-value" variable as shown above under Input Variables. Note that the distribution is even as requested in the inputs.

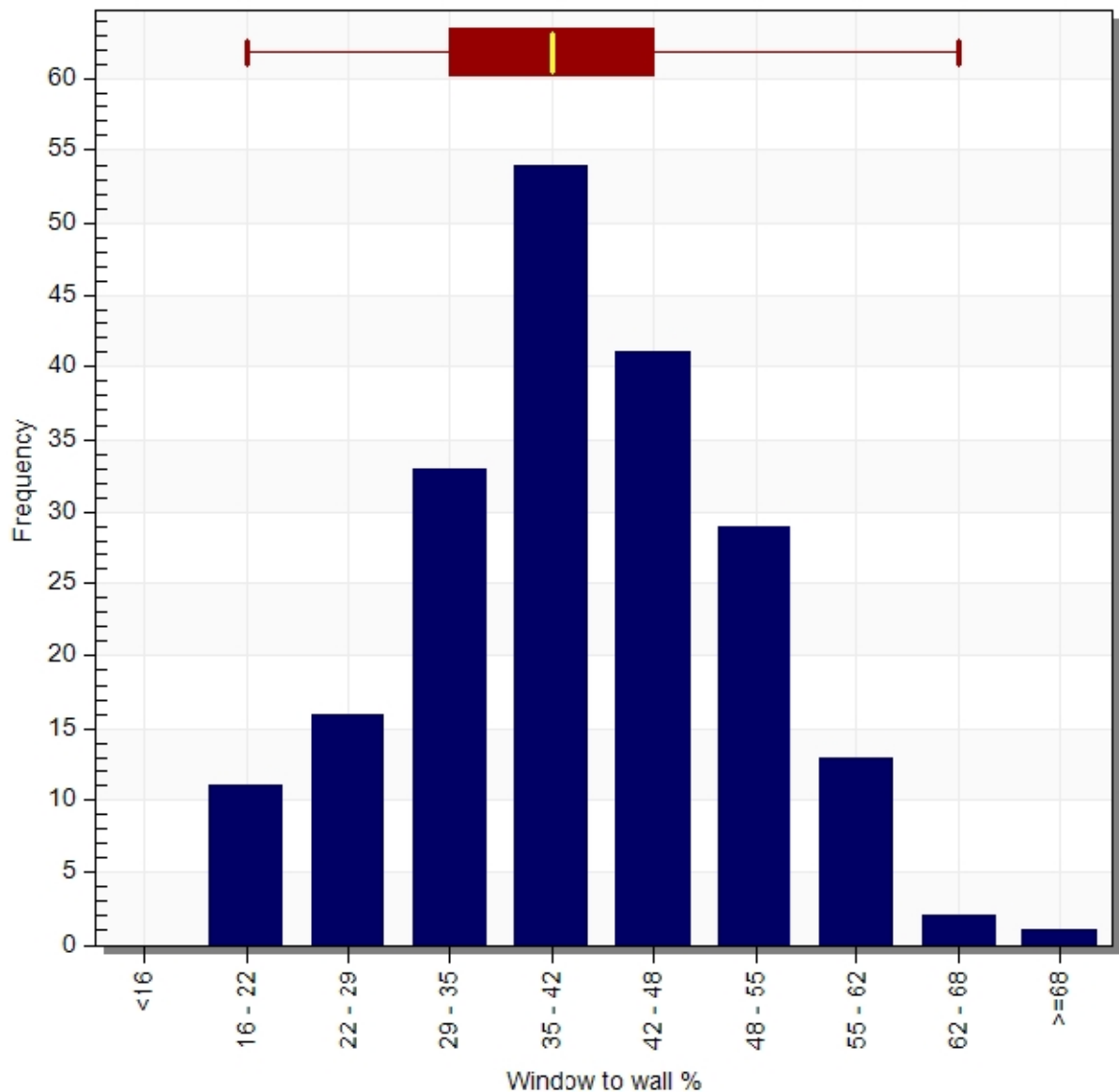
Example sampling details for a continuous variable:

3.2.1 Input variable: Window to Wall %

3.2.1.1 Summary Statistics: Window to Wall %

Mean	SD	Min	Q1	Median	Q3	Max
40	10	16	34	40	47	68

3.2.1.2 Input Distribution Histogram: Window to Wall %



This is the simulated distribution frequency for the same "Window to Wall %" variable as shown above under Input Variables. Note that the distribution is close to "normal" as requested in the inputs.

The Output Uncertainty section of the report presents the uncertainty analysis results for all the outputs requested. Summary statistics and corresponding graphs capture the spread of the outputs due to the variation in the inputs.

3.3 Output Uncertainty

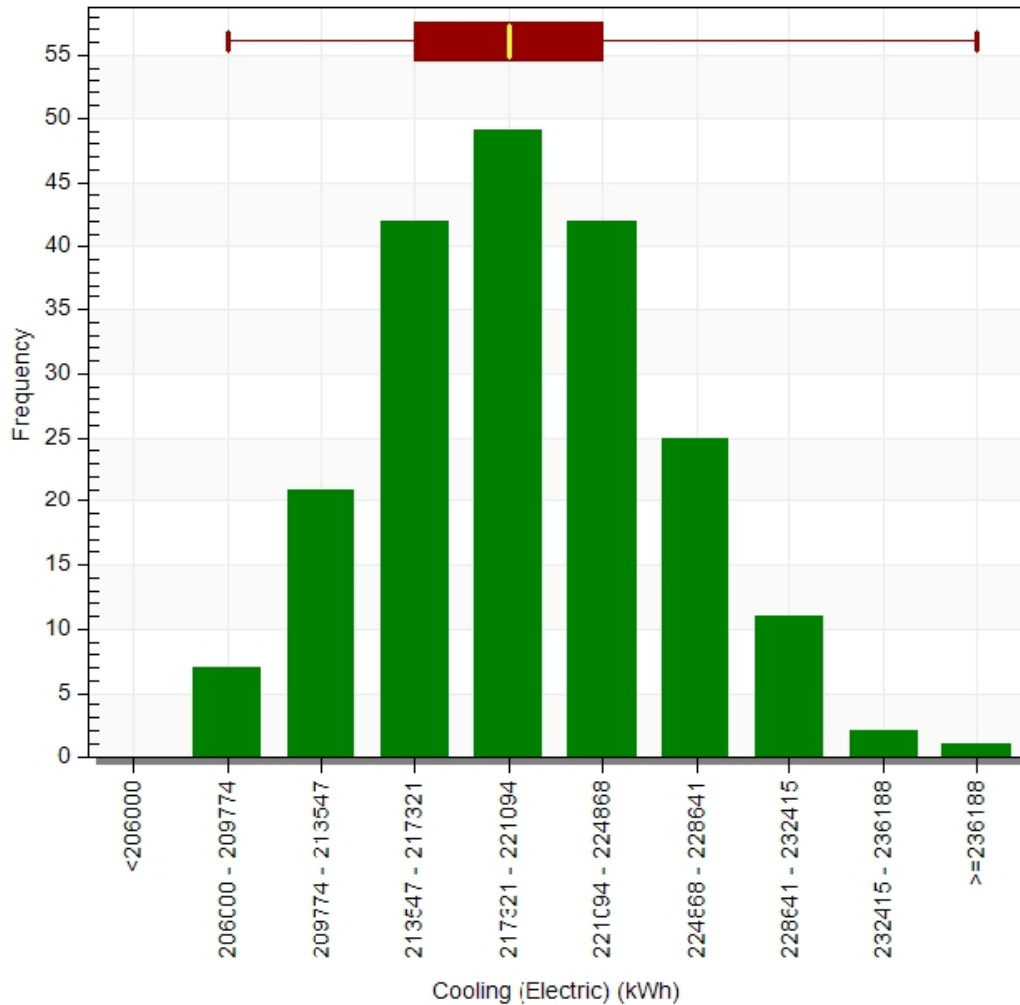
This section of the report presents the uncertainty analysis results for all the outputs requested. Summary statistics and corresponding graphs captures the spread of the outputs due to the variation in the inputs.

3.3.1 Output: Cooling Elec (Cooling (Electric))

3.3.1.1 Summary Statistics:

Mean	SD	Min	Q1	Median	Q3	Max
219755	5801	206000	215612	219568	223866	236188

3.3.1.2 Output Uncertainty Histogram:

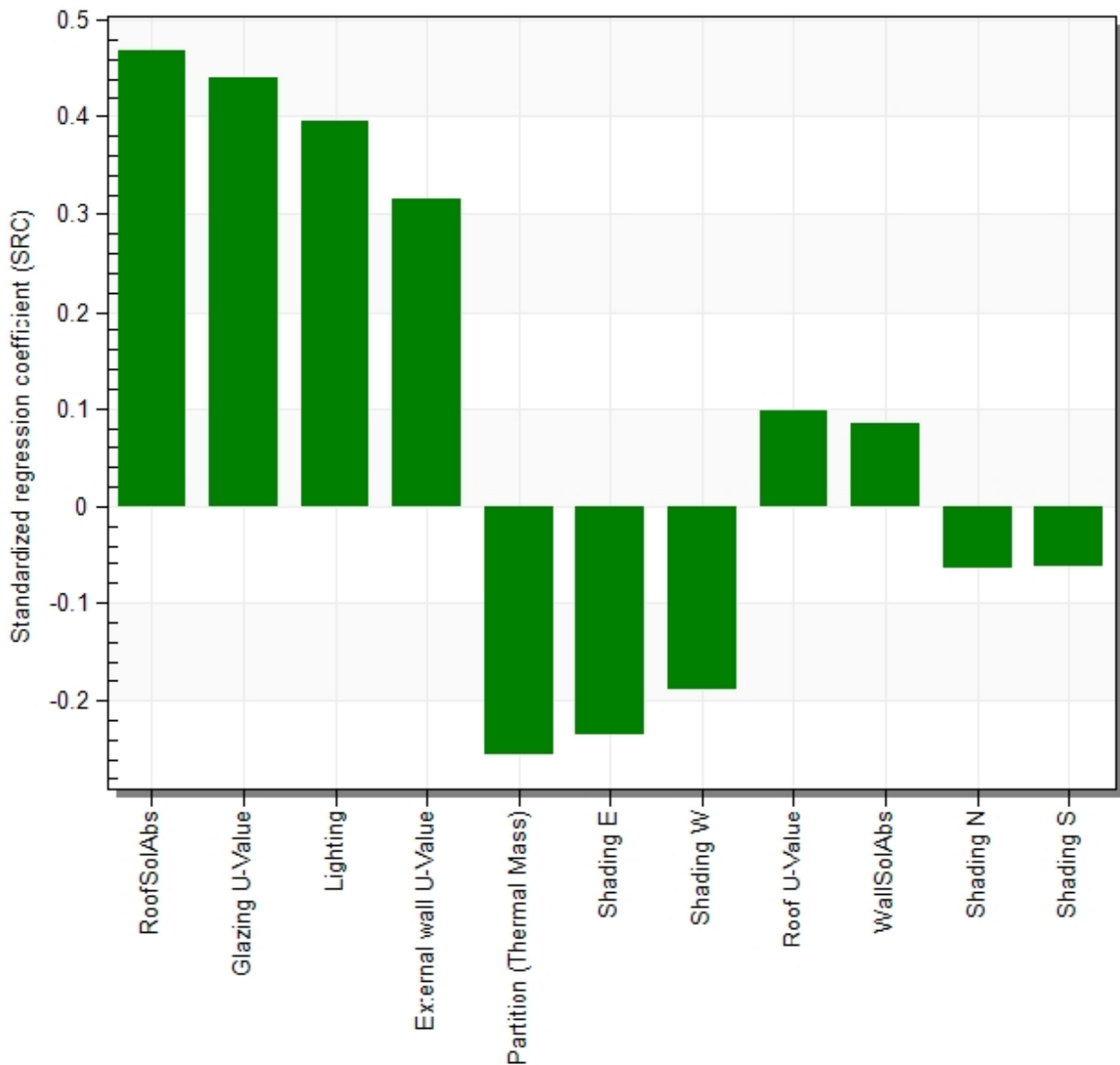


Sensitivity Analysis Results: Interpretation of the results, graphs and statistical details

The SA results are provided in the form of a histogram showing the SRC for each of the variables which are ranked in order of significance. The magnitude of each SRC value signifies the relative influence of the input on the output and the sign represents whether it is a direct or inverse relationship.

4.1 Output: Cooling Elec (Cooling (Electric))

4.1.1 Sensitivity Analysis Graph



The histogram is followed by a section which provides an interpretation of the results in readable text. It summarises the main findings of the analysis, including which variable is most influential in determining the output and those which are less so. The variables are listed in order of significance and a colour code indicates the significance of each one. High importance variables are shown in green, medium importance in yellow and low importance in red.

4.1.2 Result Interpretation

Adjusted R-squared value

It represents goodness of fit of the complete model. It indicates how much variation of the output is explained by the input variables.

For the output: 'Cooling Elec (Cooling (Electric))', the 'adjusted R-squared' value of '0.9622' is high, suggesting that most of the key sensitive input variables have been identified. Only a few input variables might be left that can improve the results. The current results can be usefully considered to identify most and least sensitive input variables.

p-value

This value tells if the input variable has a statistically significant effect on the output.

All the input variables have a p-value less than 0.05. These input variables are significant and there is high level of confidence in their respective regression result values. These regression results can usefully rank the most and the least sensitive input variables.

Standardised regression coefficient (SRC)

This value tells the relative sensitivity of the input variables to the output. Its absolute value ranks the input variables in order of sensitivity the importance and the sign identifies, if relationship to the output is direct or inverse. The list below ranks the variables in decreasing level of importance. (High Importance: **Green**, Medium Importance: **Yellow**, Low Importance: **Red**).

1. RoofSolAbs
2. Glazing U-Value
3. Lighting
4. External wall U-Value
5. Partition (Thermal Mass)
6. Shading E
7. Shading W
8. Roof U-Value
9. WallSolAbs
10. Shading N
11. Shading S

The overall **Adjusted R Squared Value** is also provided along with the variable regression coefficients, standard Errors and P-values.

4.1.3 Summary of Fit

Adjusted R Squared Value	0.9622
--------------------------	--------

4.1.4 Regression Coefficients

Variable	Reg. Coef.	Std. Reg. Coef.	Std. Error	P Value
Intercept	201039.1616	0.0000	922.2606	0.0000
RoofSolAbs (No Units)	16164.9867	0.4685	481.9230	0.0000
Glazing U-Value (No Units)	1114.3297	0.4408	35.6256	0.0000
Lighting (No Units)	2796.0390	0.3955	101.1967	0.0000
External wall U-Value (No Units)	1291.8396	0.3157	57.4932	0.0000
Partition (Thermal Mass) (No Units)	-1807.6920	-0.2547	100.0671	0.0000
Shading E (No Units)	-528.9174	-0.2348	31.5441	0.0000
Shading W (No Units)	-421.9940	-0.1879	31.2442	0.0000
Roof U-Value (No Units)	690.5407	0.0977	99.0171	0.0000
WallSolAbs (No Units)	6984.0096	0.0853	1143.4380	0.0000
Shading N (No Units)	-165.3218	-0.0655	35.5692	0.0000
Shading S (No Units)	-157.2162	-0.0621	35.8169	0.0000

Checking Regression Results

Before analysing the Standardised Regression Coefficient (SRC) results it is important to first check the statistics to ensure that the regression based Sensitivity Analysis was successful and that the results are meaningful:

- The overall **Adjusted R-Squared** value, if greater than 0.7 (ideally, greater than 0.9) indicates that the regression equation can explain the output with high precision. Adjusted R-Squared value less than 0.7 (ideally, lower than 0.4) indicate that the variables selected are bringing high uncertainty to the selected output and the regression cannot determine accurate results. In this case it may be necessary to check whether further variables need to be added to the study or existing variable ranges made more fine-tuned. Please note that low R-squared values are problematic only when you need high precision and in most cases medium to low R² values can still be useful. In such models with weaker correlations you can still ascertain general trends for input variables that are statistically significant (p-values < 0.05), and you can draw important conclusions about the relationships between those inputs including direct or inverse relations between inputs and the output.
- The **P-values** indicates the confidence (statistical significance) in the SRC calculated for each variable and should ideally be lower than 0.05, in which case they are displayed in green on the sensitivity analysis graph. A low p-value is essential for confidence in SRC value of individual variables. However in some result sets there can be scenarios where, Adjusted R-Squared value is medium to high, p-values of all important ranking (high SRC) inputs are <0.05, but some of the low importance ranking (very low SRC) inputs have high p-values (>0.05). In those cases, result interpretation (importance order) can be accepted with reasonable confidence.

If the statistics indicate good confidence in the correlation it is safe to interpret the results as described above. Otherwise, if there are any issues with the SA results, some recommended actions to improve the accuracy are:

- Check that any discrete variables have their selected options listed in order of impact (increasing or decreasing) on the output of interest.
- Check for the dominance of one variable and remove them from analysis. They will still be the most important variable and the subsequent analysis would focus only on the second best. For example if one of the variables is the HVAC system type, and the KPI of interest is cooling energy consumption, then the type of HVAC system selected will typically have a significant impact and lead to domination of the results. In an optimisation study, the same set of variables might lead to clusters of results on the Pareto graph, one cluster for each HVAC system. If one variable dominates then it can help to fix the value for that setting and re-run the study without the dominant variable.
- Remove variables of low importance to reduce unnecessary noise.
- Increasing the number of simulations can improve the results, specifically improve the p-values.
- Check the spread of variable values and their distribution. Sometimes they can be very narrow or wide, either because max and min ranges are incorrect or the distribution for sampling results leads to narrow or very wide ranges.

Uncertainty And Sensitivity Analysis - Tips And Tricks

Speeding Up Uncertainty And Sensitivity Analysis

Uncertainty and sensitivity analysis studies require hundreds or even thousands of simulations to be carried out and so can be extremely time/calculation intensive. To help speed simulations, you could consider using one or more of these techniques:

1. Analyse a [single representative zone](#) in the building and apply findings to whole building.
2. Use a representative simulation period instead of the whole annual simulation. For example, in many climates using the first or second 6 months of the year would be a good approximation to use of the full 12-month simulation. Alternatively, if the analysis is mainly related to summer overheating then you might choose a shorter representative summer period for the simulations.
3. In most cases, the more variables that are defined in the analysis, the more runs will be required to achieve more acceptance results. Therefore, instead of trying a multitude of runs, a preliminary analysis using a more efficient sampling method like [LHS](#) can be used to eliminate the least influential variables and reduce the parameter space before doing a detailed run.
4. Use [JEPlus](#) or a network [simulation server](#) to maximise the number of simulations that are run in parallel at one time. If you have a simulation server with many cores at your disposal then you might consider increasing both the population size and [Number of job per batch](#) which allows more cases to be analysed at once.

See also [Working with Large Models](#).

Running Sensitivity Analysis As A Precursor To Optimisation

Complex [optimisation](#) with a large number of design variables and options makes it difficult to find clearly distinguishable trends. Also, some variables might not have a significant influence on the output can make the assessment of the relative importance of other important variables difficult. Sensitivity Analysis determines which are the most influential Design Variables, and what their relationship to the output is.

Sensitivity Analysis can be used as a precursor to optimisation to identify and remove the less influential variables from the optimisation runs, thereby reducing the complexity and overall optimisation runtime. Also, If the variable is influential for only one of the objectives, its value can be predetermined with SA and removed from the optimisation. **With a focus only on the important variables, the optimisation results are cleaner, quicker and easily understandable.** Another advantage of using SA, is that it helps you to understand any trends associated with design variables and makes it easier to see these in the results. For example, SA can tell us the relationship of each Design variables with the Objectives and whether they are directly or inversely correlated, i.e. whether an increase in the design variable leads to an increase or decrease in the objective's value.

However, there are certain considerations should be kept in mind because unlike optimisation, the idea of Sensitivity Analysis is to understand the relationship between the Design Variables and the Objectives and

not to find the best set of designs. In Sensitivity Analysis two important things need to be done in variable definition which will typically be different from optimisation:

1. The order in which design variable options are set up is very important. The design variable options need to be [ordered](#) in an increasing or decreasing value. For example, in a case where wall constructions with different U-Values is one of the design variables and heating energy and capital cost are the two outputs, the overall U-value of the wall affects heating energy, and the wall construction cost affects building capital cost. Therefore, **if the U-value of a wall is changed by using different materials, then the wall options sequence when ordered in decreasing U-value, might not correspond with the one created that will be created with increasing capital cost. In this case, the Sensitivity Analysis must be done separately for both the outputs.**
2. The design options selected for the variable should not contain options that have the same value (i.e. duplicates) with respect to the target objectives. For example, in the same case as above there might be two wall constructions that have the same U-value or overall cost. **In that case wall construction options that have the same U-value need to be removed from Sensitivity Analysis for heating energy, and wall construction options with the same Cost need to be removed from the Sensitivity Analysis for capital cost.**

Therefore, despite the theoretical possibility of doing Sensitivity Analysis for more than one output at a time, DesignBuilder recommends, as a good practice, that sensitivity analysis of each output is set up separately and as a precursor to Optimisation. **In each output it is important that the variable options in list type variables have a correct sequence so that SA can find the impact of a progressive change of one option to another on the SA output.**

This process is demonstrated in more detail in our [Uncertainty and Sensitivity Analysis: Simplify and Speed up Optimisation Studies webinar](#).

Ensure Inheritance Path

It is important to ensure that the variations defined by the uncertainty and sensitivity analysis [Design Variables](#) will have the intended effect. Uncertainty and sensitivity analysis works by applying variations to a copy of the base model and these changes are applied to the object defined in the Variable. You must take into account how this data will be inherited down to any child blocks, zones etc to ensure you are testing the design variants you think you are. For example, if the Window to wall % was hard set at block level then changes in this data at building level would not influence the surfaces of the design variants in that block and the results will indicate that the Window to wall % does not influence results.

Switching On Shading When Shading System Type Is A Variable

Design variables such as Local shading type and Window shading will only take effect if the corresponding checkbox activating these options is checked. A common mistake for new users is to request these variables without switching on the option.

Uncertainty And Sensitivity Analysis Tutorials

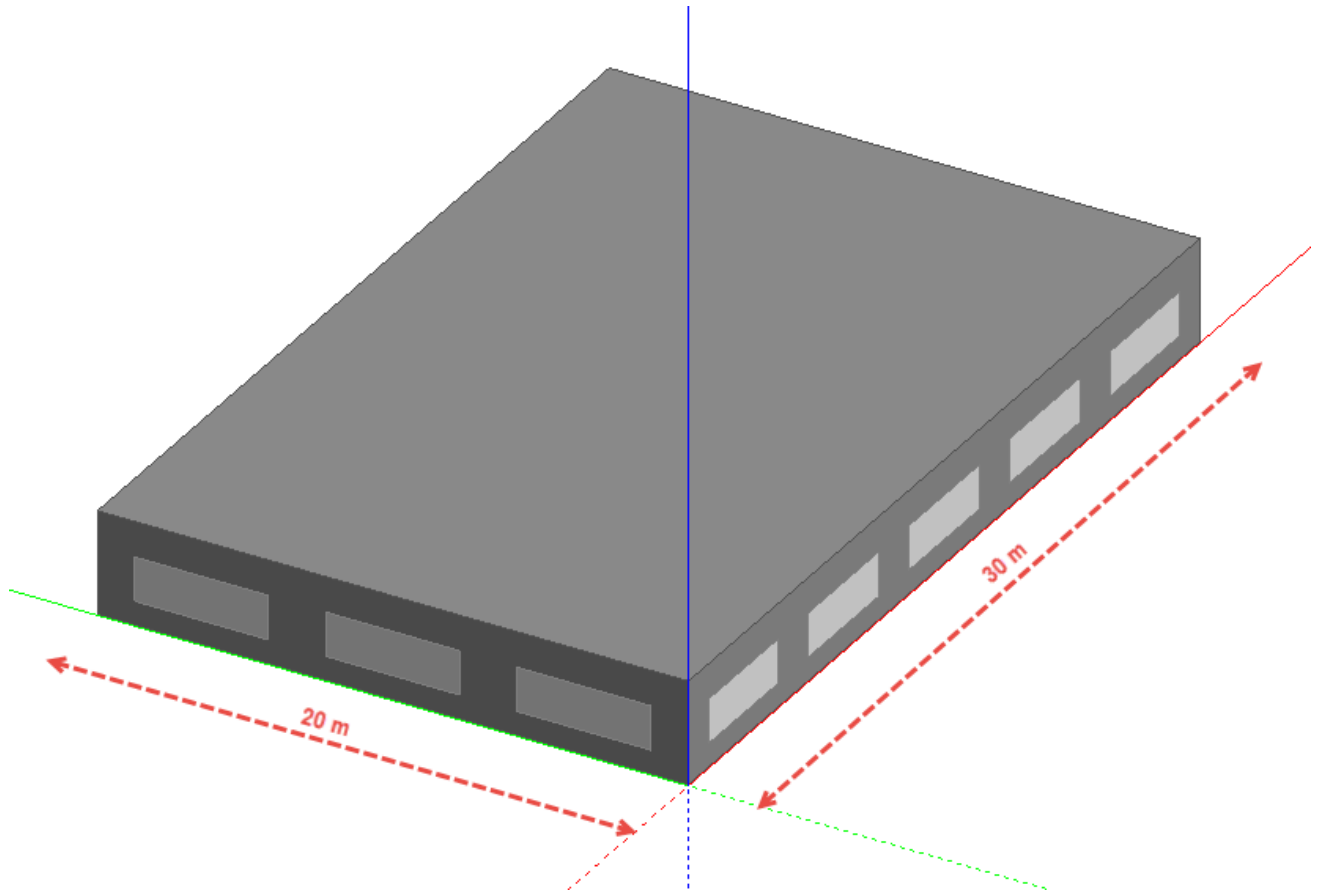
- "Uncertainty and Sensitivity Analysis - Basic Tutorial" on page 1868
- "Uncertainty and Sensitivity Analysis - Advanced Tutorial" on page 1878

Uncertainty And Sensitivity Analysis - Basic Tutorial

This tutorial will help you to learn the basic information needed to run an uncertainty and sensitivity analysis study in DesignBuilder in a few easy steps.

Step 1 - Create A Base Model

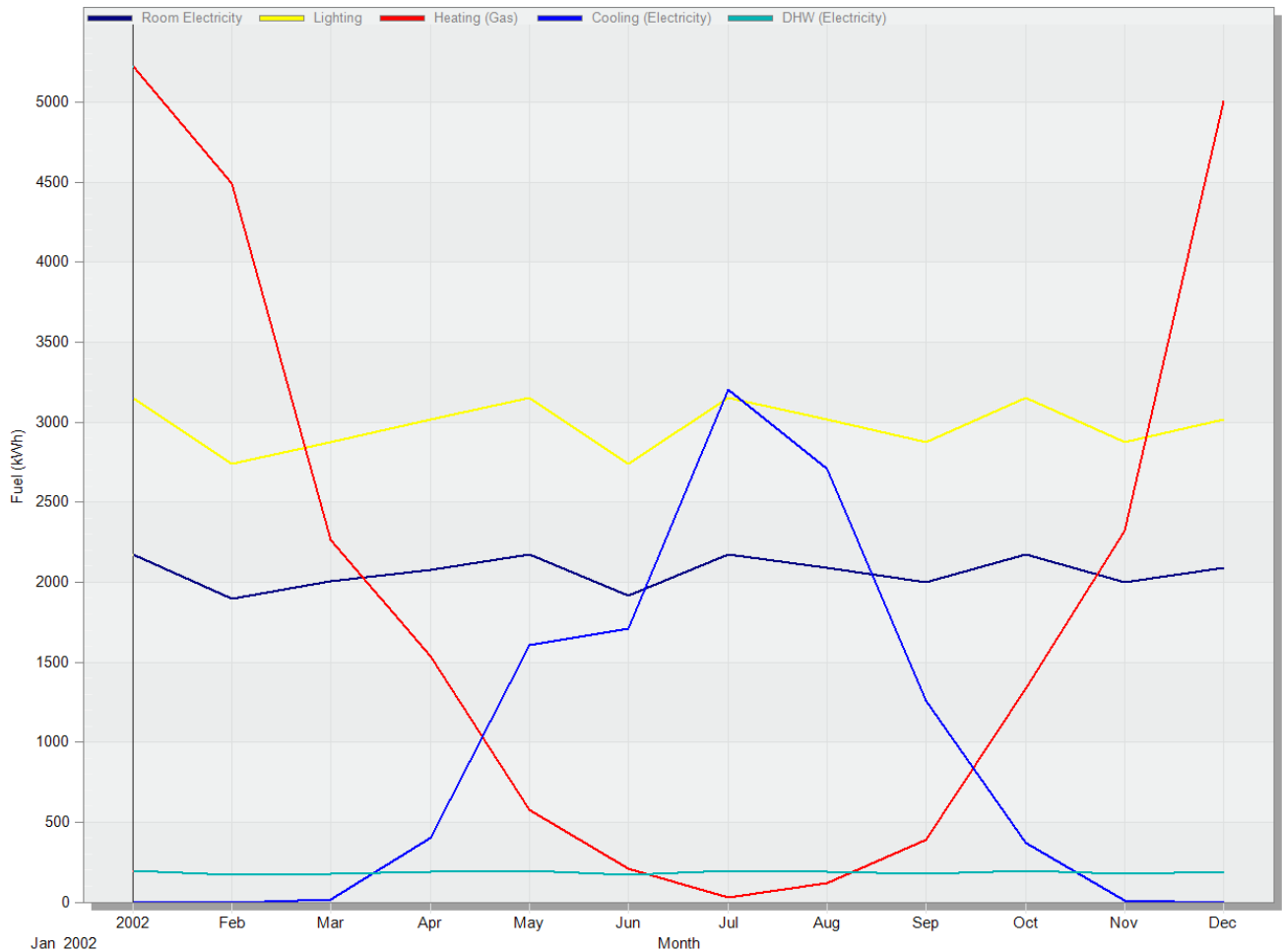
Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m as shown below. Use default Model options and template settings. DesignBuilder as supplied will use a Simple HVAC Fan Coil Unit system as the default. For this example, you should make sure that you have heating and cooling selected and no natural ventilation (this will be the case for a new model with an FCU HVAC system selected).



Step 2 - Run A Standard Simulation

Click on the Simulation tab and run a base annual simulation. Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation.

Check the hourly results for the simulation period and make sure that the model is behaving as expected, including temperatures within the building, operations periods etc. If not fix the model and repeat this step until you are happy with the base model hourly results. Monthly results should look something like the screenshot below.

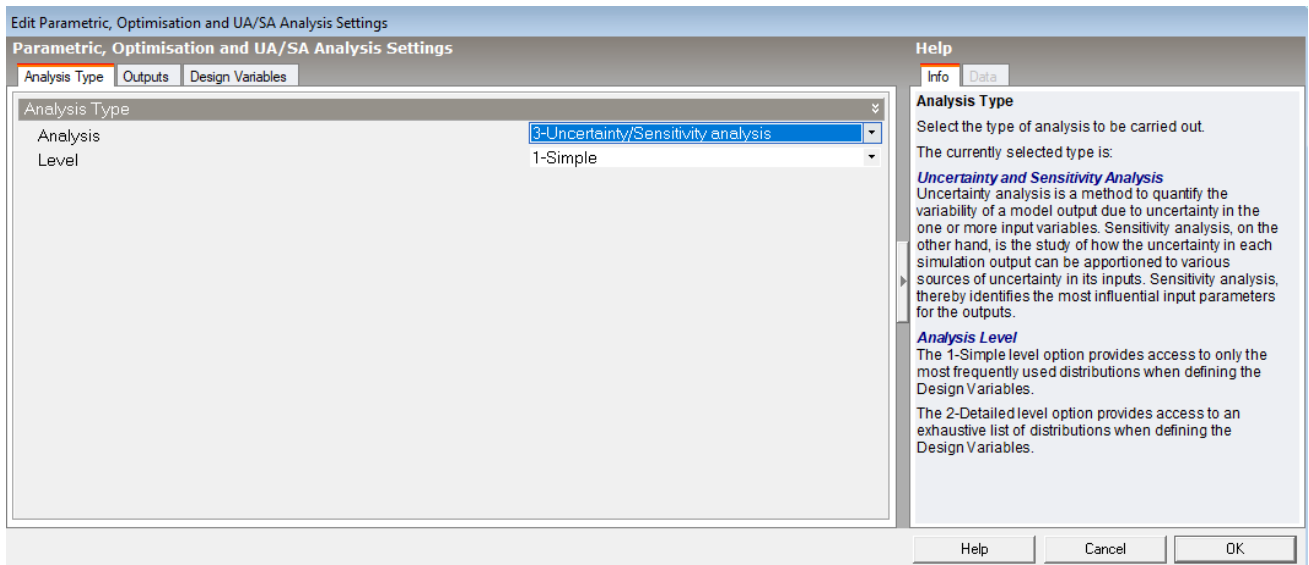


Step 3 - Define Problem On Uncertainty And Sensitivity Analysis Settings Dialog

Once you have a good understanding of how the base model operates, you are ready to start the uncertainty and sensitivity analysis stage. To do this click on the **Optimisation + UA/SA** tab on the Simulation screen. Because you don't have any results yet, the [Parametric Settings](#) dialog is displayed.

Analysis Type

Change the Analysis type to **3-Uncertainty/Sensitivity**.

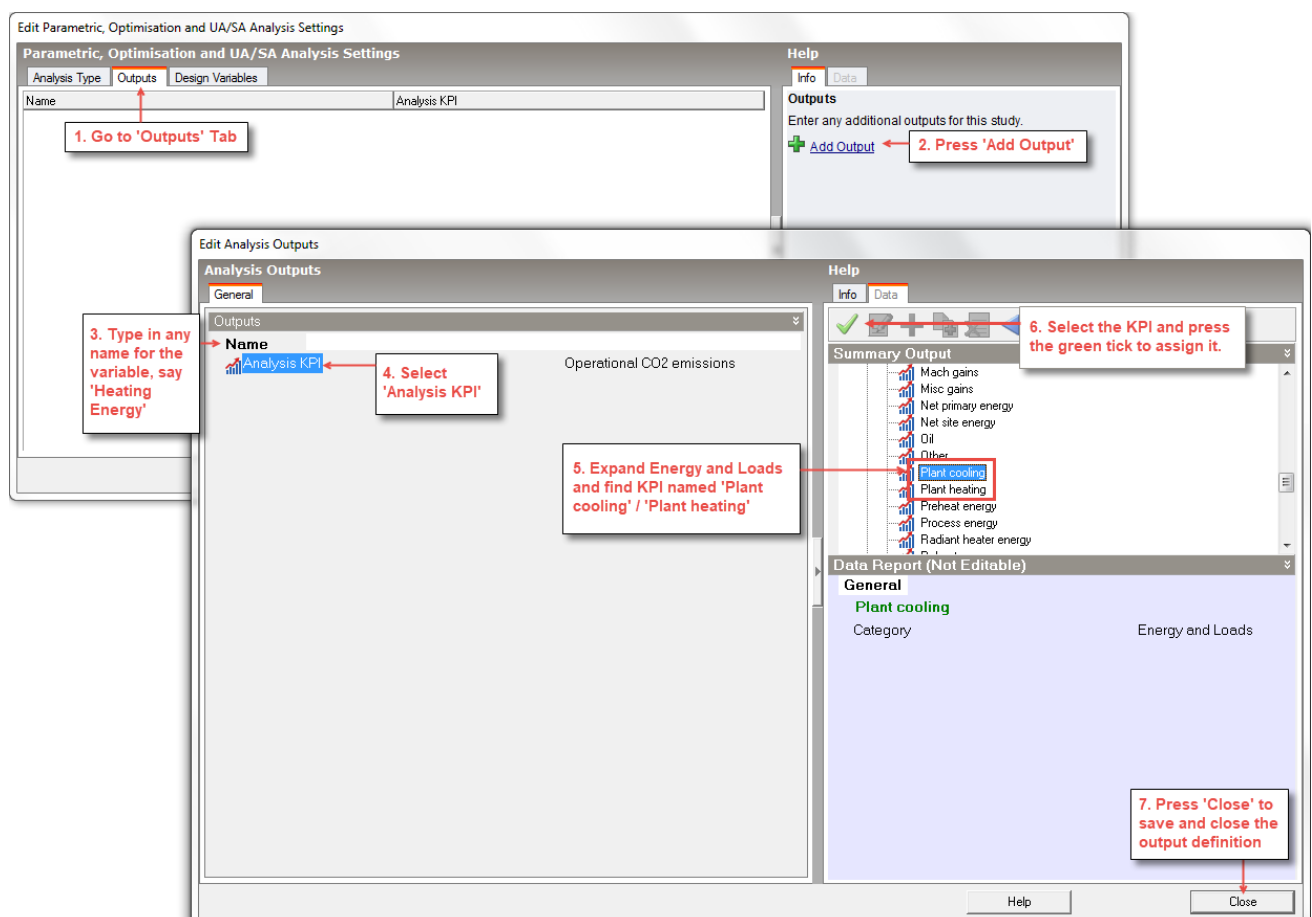


This will set up the rest of the tabs on the dialog and allows you to define the uncertainty and sensitivity analysis problem, i.e. what it is that you want to achieve from the uncertainty and sensitivity analysis study. Use the following settings for this simple example which request to identify:

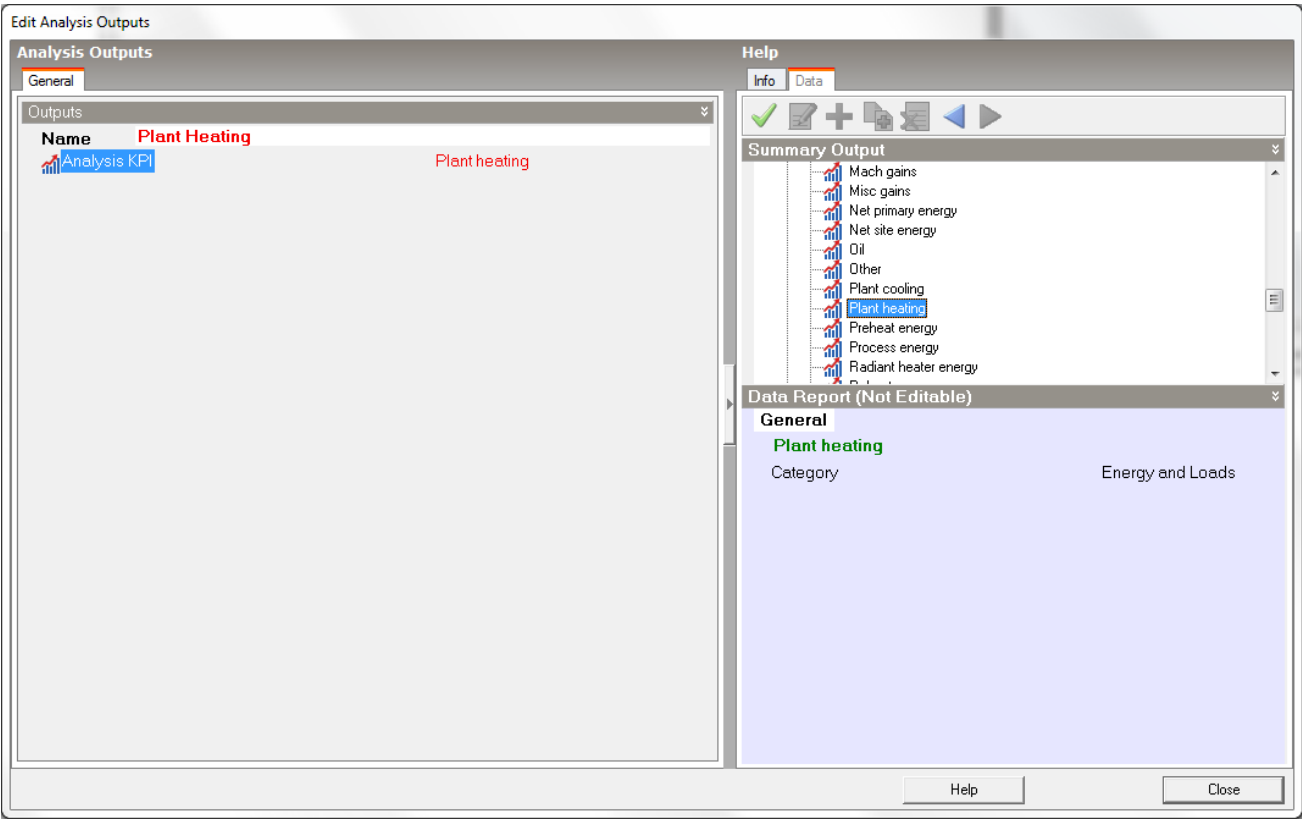
1. The uncertainty in plant heating and plant cooling energy use of the building due to the uncertainty of various inputs
2. The most influential input set of inputs that affect the plant heating and plant cooling energy use.

Output

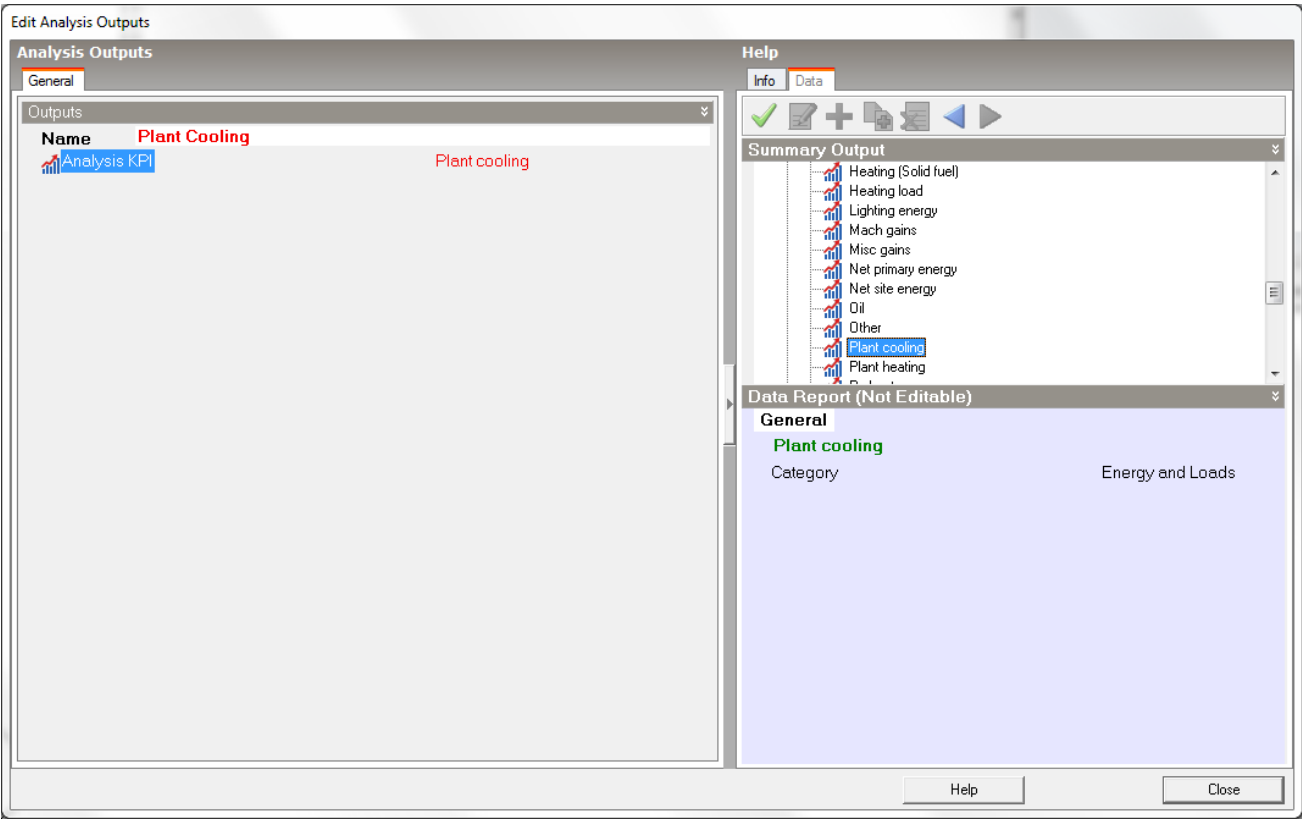
There are no predefined outputs, add new output by using the 'Add Output' tool and accessing the 'Edit Analysis Outputs' dialog. Add 'Plant heating' and 'Plant cooling' output KPIs. The procedure to select the KPIs is shown below.



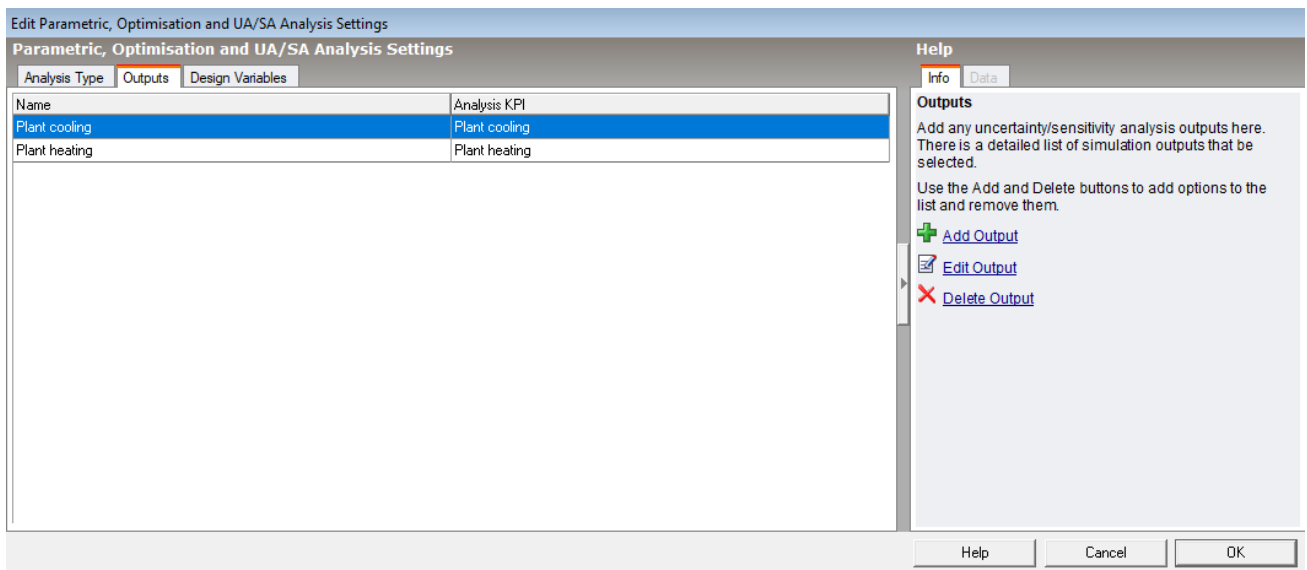
Final Plant heating Analysis Output dialog:



Final Plant cooling Analysis Output dialog:

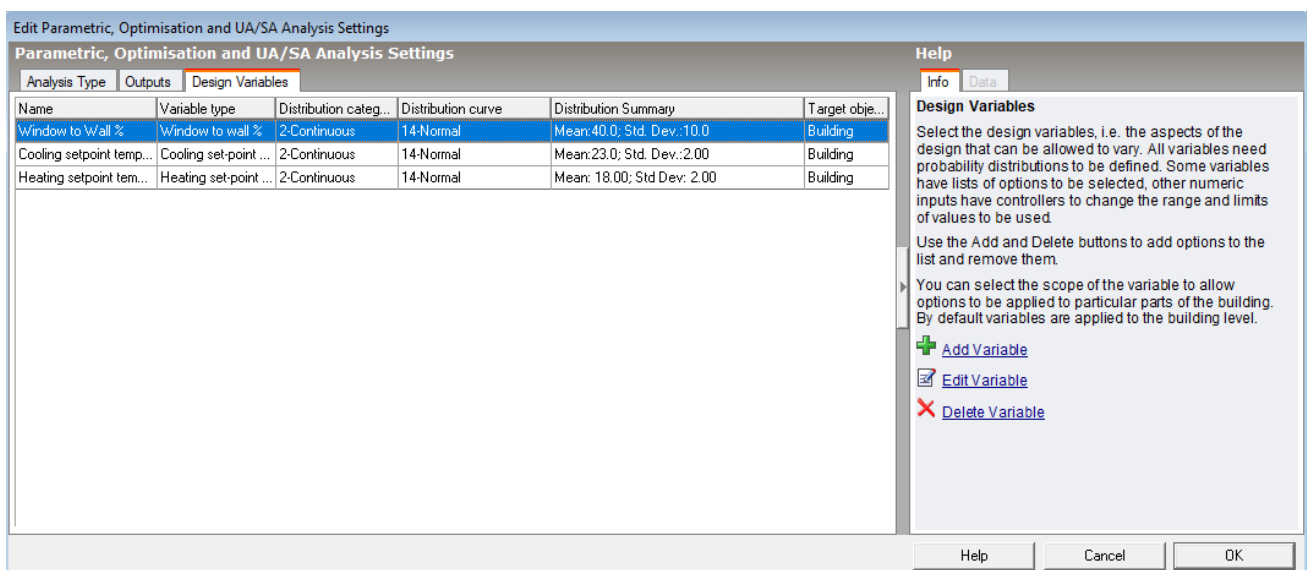


Final outputs table



Design Variables

Go to the Design Variables tab. There are three pre-defined design variables. Allow WWR and cooling and heating setpoints to vary as shown in the screenshot below. Variations are made at building level for all variables in this case as shown by the Target objects column.



Step 4 - Open Calculation Options Dialog

Having confirmed the uncertainty and sensitivity analysis options, the next dialog to open will be the Calculation options. Check the number of runs is set to 200 for this analysis.

Calculation Description		⌵
From		⌵
Start date	1 Jan	
To		⌵
Start date	31 Dec	
☐ Pause on errors ☐ Quietly retry if server becomes unavailable		
Uncertainty/Sensitivity Analysis Options		⌵
Sampling option	1-Random	
Sensitivity method	1-Regression	
Number of runs	200	
Advanced		⌵

Step 5 - Start Uncertainty And Sensitivity Analysis

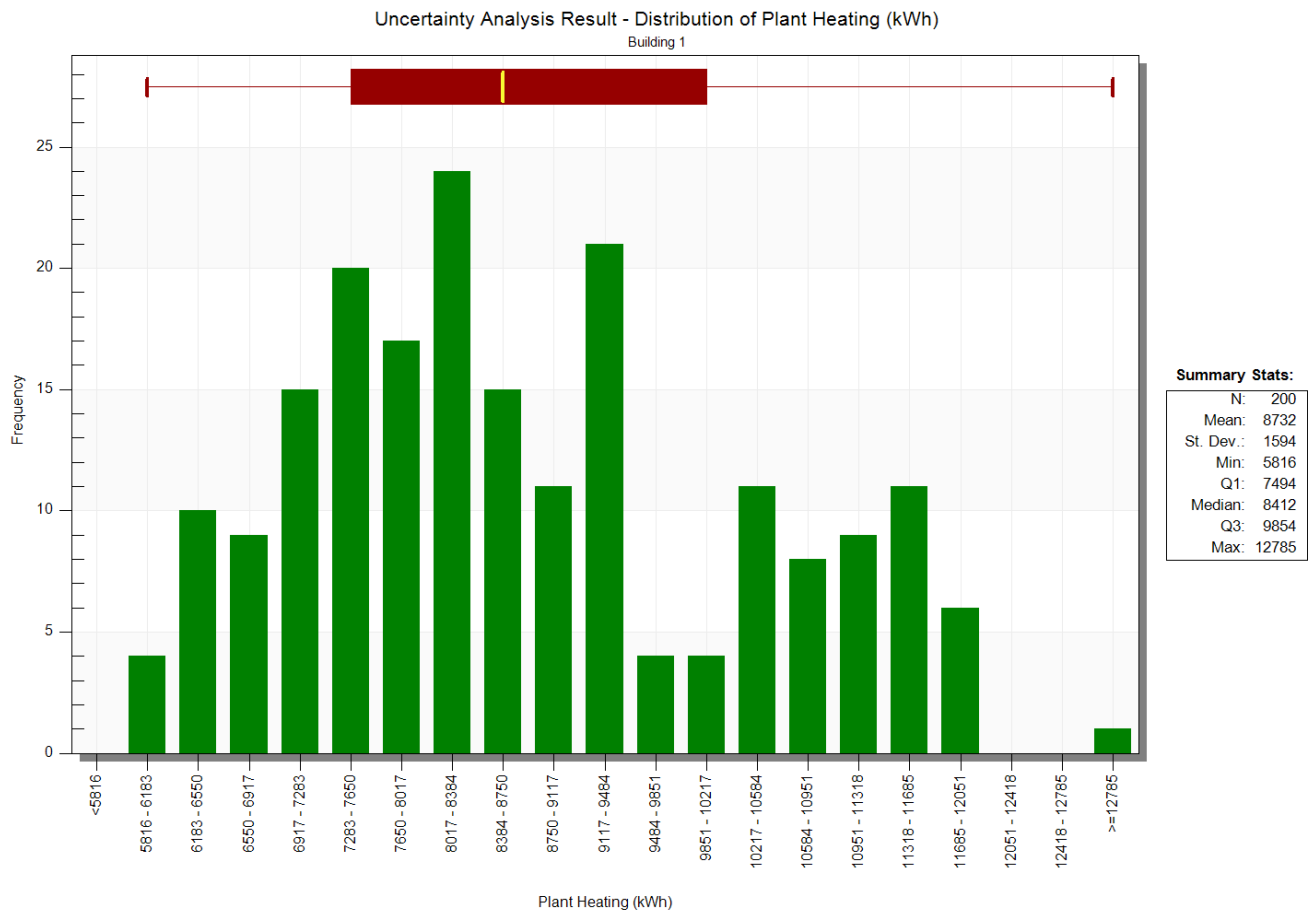
Once you have completed your review of the options press the **Start** button in the bottom right of the dialog. The uncertainty and sensitivity analysis process will involve running a lot of simulations. With the recommended settings there will be 200 runs with 20 runs being plotted in each batch, updating histogram as each batch is finished. That means that 10 batches will be run! For our simple model and when using the Simulation manager to run these in parallel this shouldn't take too long. However, if time is an issue it is usually worth starting with a smaller number of runs.

After all runs have finished an alert is shown. Press OK to dismiss the alert and then Press the **Close** button to return to the **Optimisation +UA/SA** tab on Simulation screen to analyse the results.

Step 6 - Analyse Results

Uncertainty analysis results

Plant Heating results:



To visualise the **Plant Cooling** results, change Output Selected to **2-Plant cooling**.

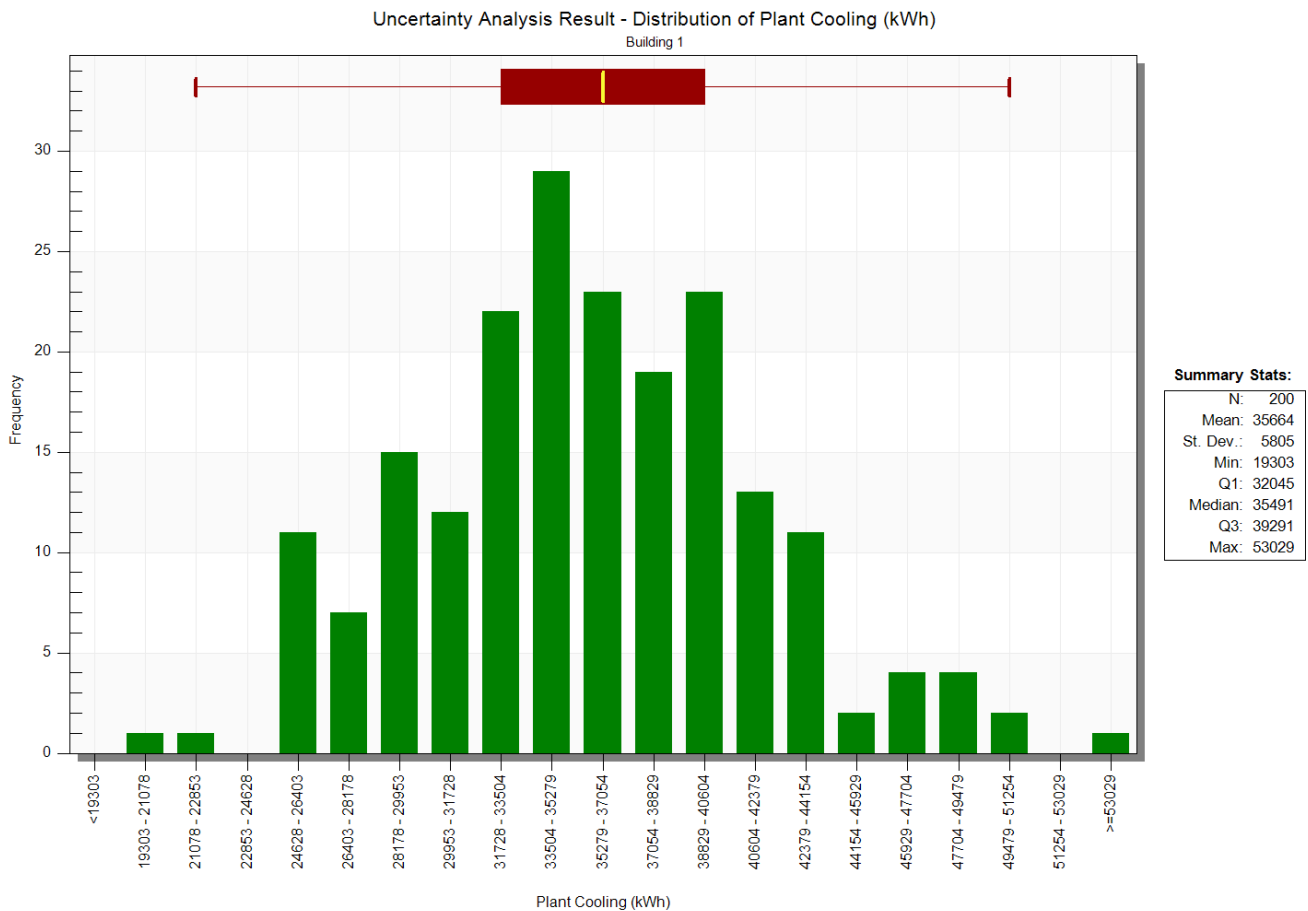
Uncertainty/Sensitivity Analysis

☒ Show analysis
 Analysis type 1-Uncertainty a
 Display type 1-Graph
☐ Show data grid

Histogram Controls

Output selected 2-Plant cooling
☐ Underflow bin
☐ Overflow bin
 Bins 1-Auto

Plant Cooling result:



Keep the default **1-Uncertainty analysis** Analysis type in the Display options panel result to view UA results first. You should be able to see from the histogram and summary statistics in the right hand panel on the graph that:

- The mean value of Plant heating is 8732 kWh with standard deviation of ± 1594 kWh and the mean value of Plant cooling is 35664 kWh with standard deviation of ± 5805 kWh. For these input parameters the total plant heating has a higher degree of variability compared to plant cooling as it has a wider interquartile range.

Sensitivity analysis results

To visualise the sensitivity analysis results, change the Analysis type in the Display options panel to **2-Sensitivity Analysis**:

Uncertainty/Sensitivity Analysis

☒ Show analysis

Analysis type 2-Sensitivity an

Display type 1-Graph (SRC)

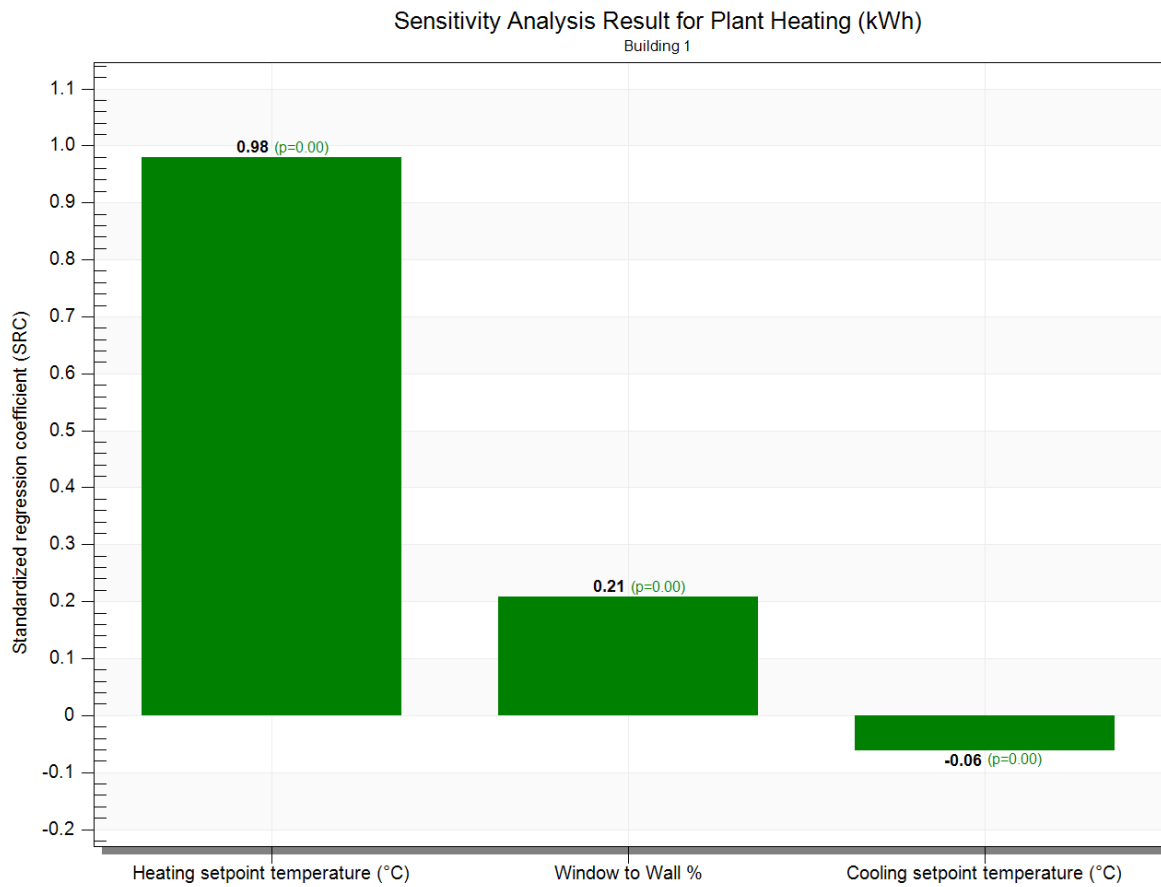
☐ Show data grid

Graph Controls

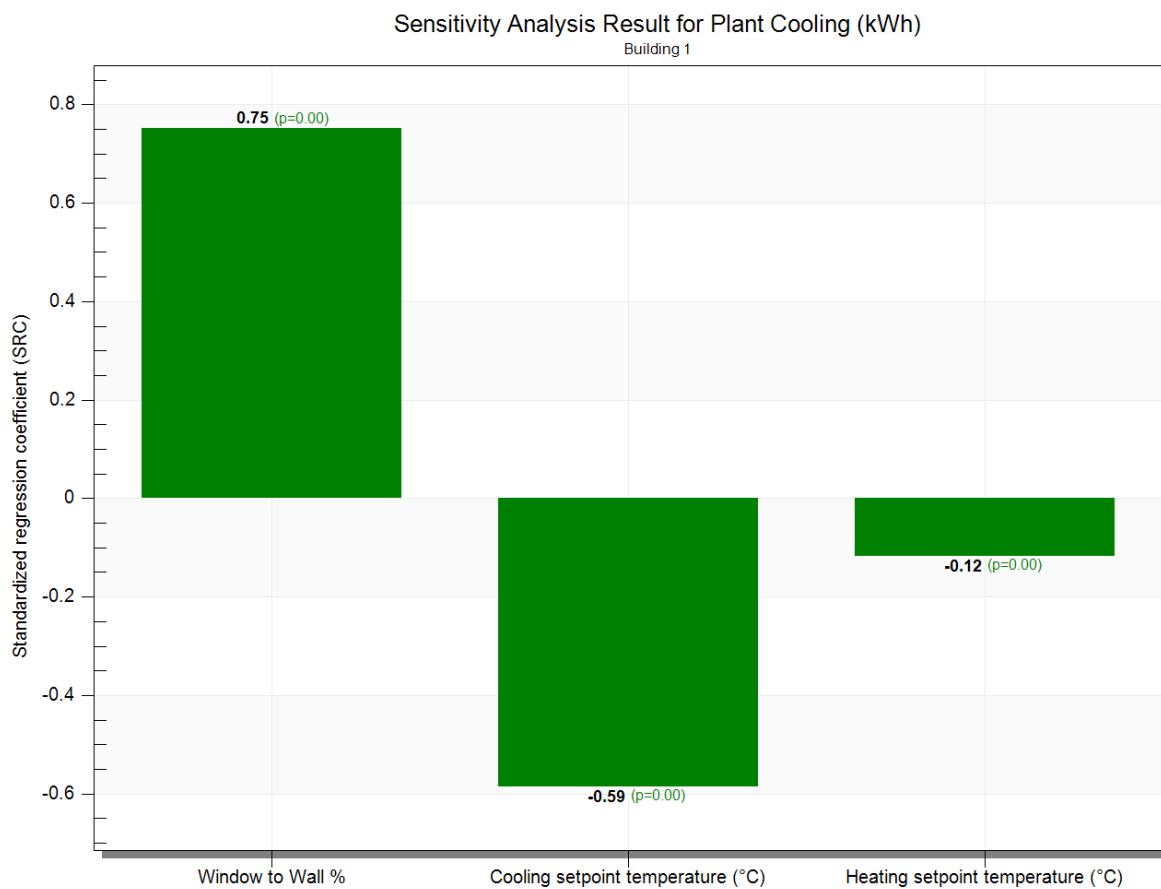
Output selected 1-Plant heating

Input order 2-Decreasing I

Plant Heating result:



Plant Cooling result:



The Sensitivity analysis results are shown by default in decreasing order of importance with the most important variable on the left and the least important on the right. The value printed in black bold above each bar is the standardised regression coefficient (SRC), telling us the relative importance of each input. You should see that the **p-value** (number in green, to the right of the SRC) is <0.05 for all inputs indicating a high level of confidence in the results for each variable.

For **plant heating**, among the three inputs, heating setpoint is the most important and an increase in setpoint increases heating energy use. To achieve better heating energy performance, heating setpoint should be lowered first rather than WWR. This is because increase in WWR on one hand reduces the overall fabric thermal performance (glass having higher U-value than walls) but on the other allows more solar gains, thereby reducing the net negative impact of higher WWR. As expected cooling setpoint has minimal impact on plant heating.

For **plant cooling**, the combined effect of lower fabric performance and high solar gains created because of higher WWR, it becomes the most influential factor that leads to increase in cooling energy use, even more than cooling setpoint. As expected the negative SRC value of cooling setpoint validates its inverse relationship with plant cooling. Additionally, very low SRC value of heating setpoint verifies that it has minimal impact on plant cooling.

Note: Regression-based sensitivity analysis results for the two outputs are separate analyses which consider the relative importance of inputs for one output at a time. They should be analysed separately. The results for the two outputs should not be compared with each other. For example, interpreting that WWR has more influence on plant cooling than on plant heating or lowering WWR will lower the total space conditioning energy use is not appropriate. For that a separate sensitivity analysis of total energy use should be done.

Uncertainty And Sensitivity Analysis - Advanced Tutorial

This tutorial will guide you through a more realistic uncertainty and sensitivity analysis for a post-occupancy application.

Problem: Identify the most influential factors impacting heating energy use in an office building by comparing uncertainty due to occupant behaviour and to construction quality.

Explanation: The example is based on the following assumptions:

- the main uncertainties in occupant behaviour can be analysed by varying heating setpoint and operation of internal equipment, and,
- the uncertainty in construction quality can be represented by varying fabric performance, i.e. variability in U-value of walls, windows and roof and variability in infiltration due to unknown envelope sealing.

The following are the variations that are to be applied for the two categories:

Category	Input	Variability
Occupant behaviour	Heating Setpoint	Target setpoint = 20°C. Uncertainty is represented by normal distribution with a standard deviation of 1°C. Lower Bound = 18°C. Upper Bound = 21°C to maintain a 2°C dead band with cooling setpoint.
	Equipment Power density	Target value = 13W/m ² Uncertainty is represented by triangular distribution Lower Bound = 10W/m ² to represent low IT usage work environment Upper Bound = 20W/m ² to represent a high IT usage work environment
Construction Quality	Infiltration	Target value = 0.25 ach Uncertainty is represented by triangular distribution

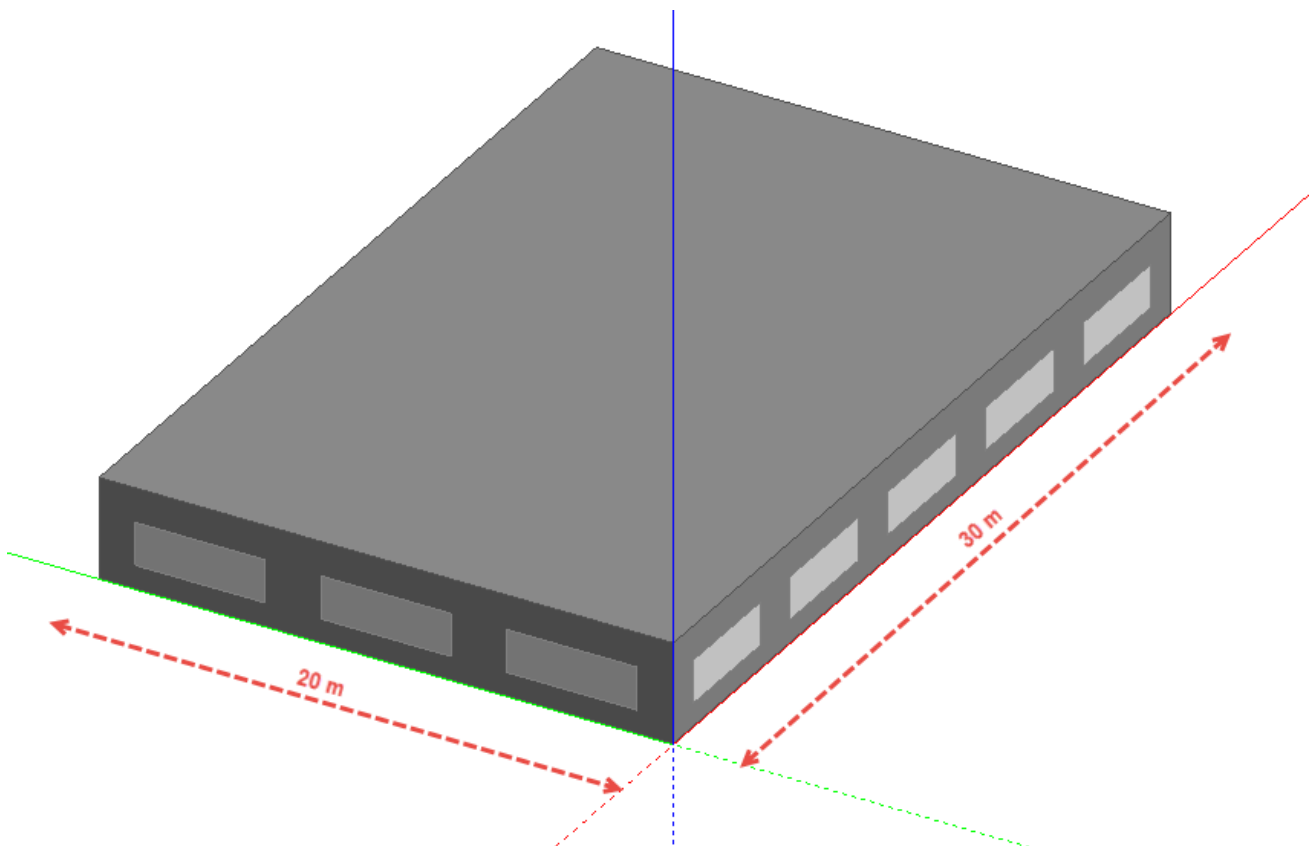
		Lower Bound = 0.22 ach* Upper Bound = 0.35 ach*
	Wall U-value	Target value = 0.30 W/m ² K Uncertainty is represented by Binomial distribution** Lower Bound = 0.25 W/m ² K * Upper Bound = 0.45 W/m ² K *
	Window U-Value	Target value = 2.00 W/m ² K Uncertainty is represented by Binomial distribution** Lower Bound = 1.90 W/m ² K * Upper Bound = 2.40 W/m ² K *
	Roof U-value	Target value = 0.20 W/m ² K Uncertainty is represented by Binomial distribution** Lower Bound = 0.16 W/m ² K * Upper Bound = 0.30 W/m ² K *

* Lower Bound values represent the instances when high degree of care was put on construction quality and targets were exceeded. This is less likely to happen and the over-achievement in normal practice would not be very high. On the other hand, Upper Bound values represent the instances when low care was put on construction quality and targets were not met. This is more likely to happen and in normal practice the deviation could be significant.

** Wall, roof and glazing U-values are not direct inputs in EnergyPlus, therefore the variation in U-value must be represented by creating a sequence of construction components to create fixed incremental increases in U-value. Then discrete distributions can be used to represent the variation.

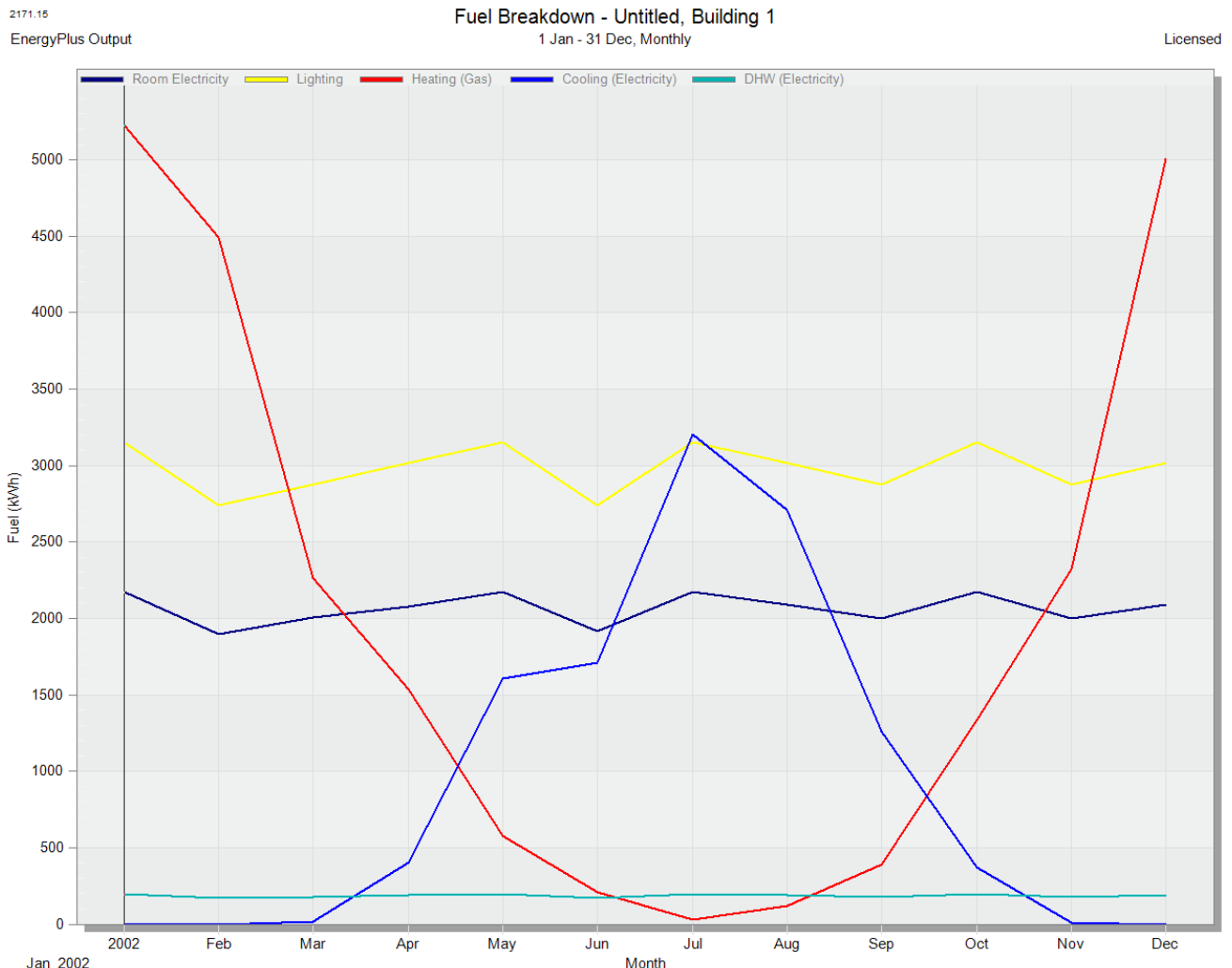
Step 1 - Create A Base Model

Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m as shown below. Use default Model options and template settings. DesignBuilder as supplied will use a Simple HVAC Fan Coil Unit system as the default. For this example, you should make sure that you have heating and cooling selected and no natural ventilation (this will be the case for a new model with an FCU HVAC system selected).



Step 2 - Run A Standard Simulation

Click on the Simulation tab and run a base annual simulation. Because it is a simple model you can select hourly results. Make sure to also choose Monthly results which are required by the Optimisation. Check the hourly results for the simulation period and make sure that the model is behaving as expected, including temperatures within the building, operations periods etc. If not, adjust the model and repeat this step until you are happy with the base model hourly results. Monthly results should look something like the screenshot below.

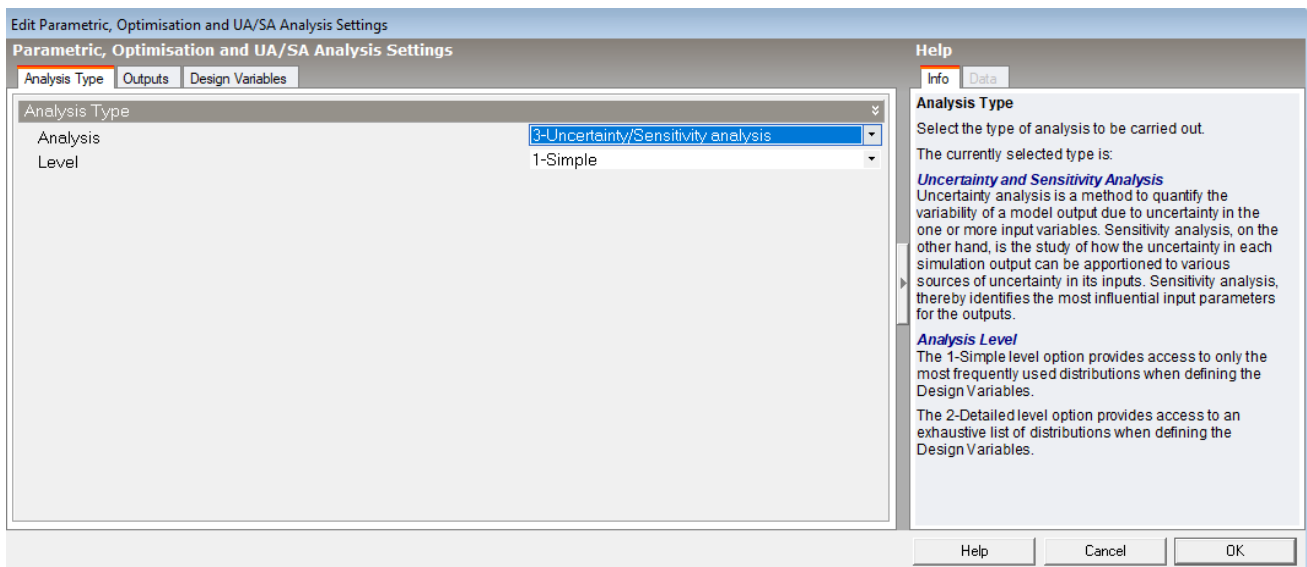


Step 3 - Define Problem On Uncertainty And Sensitivity Analysis Settings Dialog

Once you have a good understanding of how the base model operates, you are ready to start the uncertainty and sensitivity analysis stage. To do this click on the **Optimisation + UA/SA** tab of the Simulation screen. Because you don't have any results yet, the **Parametric Settings** dialog is displayed.

Analysis Type

Change the Analysis type to **3-Uncertainty/Sensitivity** as shown below.



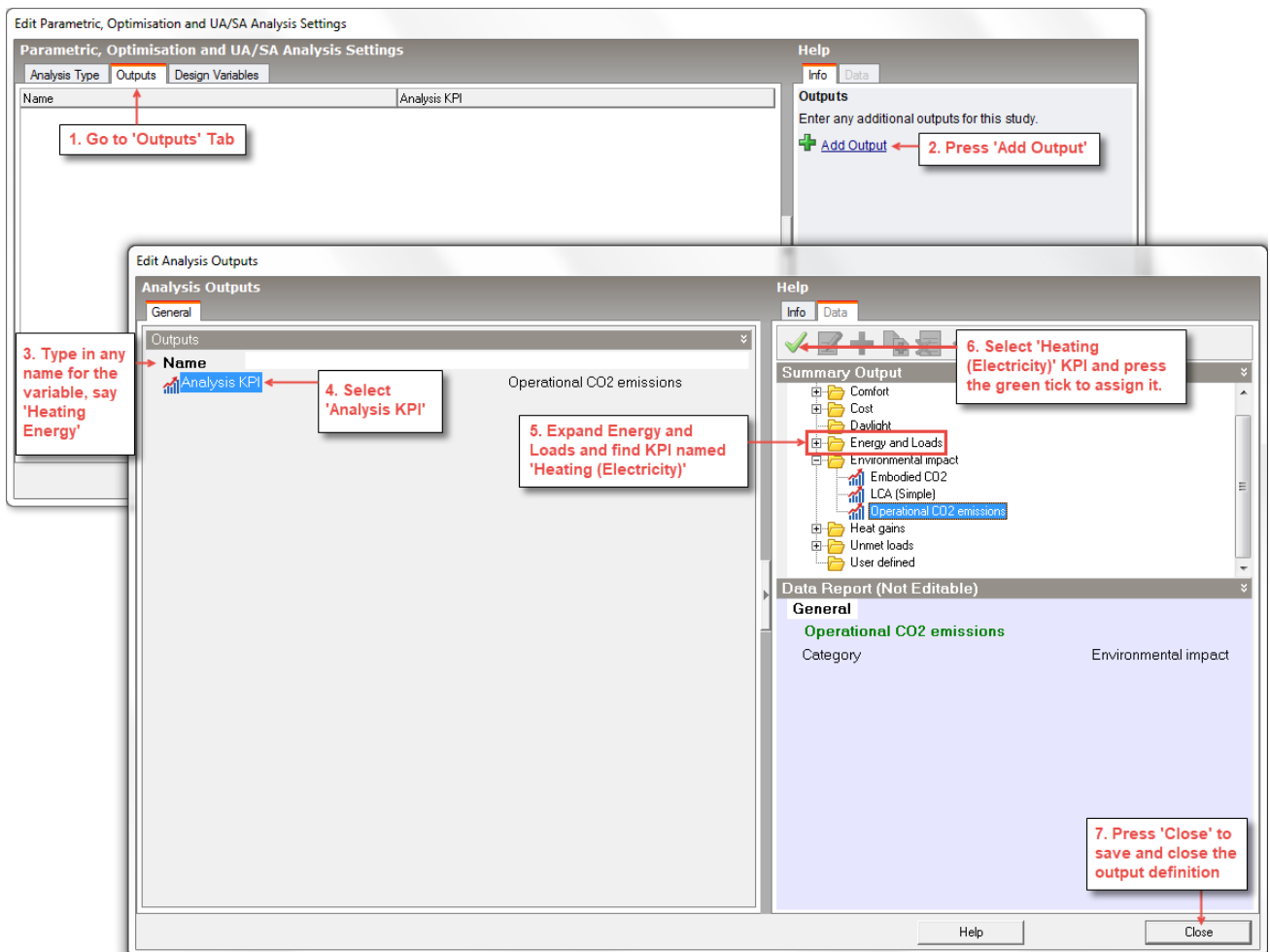
This will set up the rest of the tabs on the dialog and allow you to define the uncertainty and sensitivity analysis problem, i.e. what it is that you want to achieve from the uncertainty and sensitivity analysis study.

Use the following settings for this example to us identify:

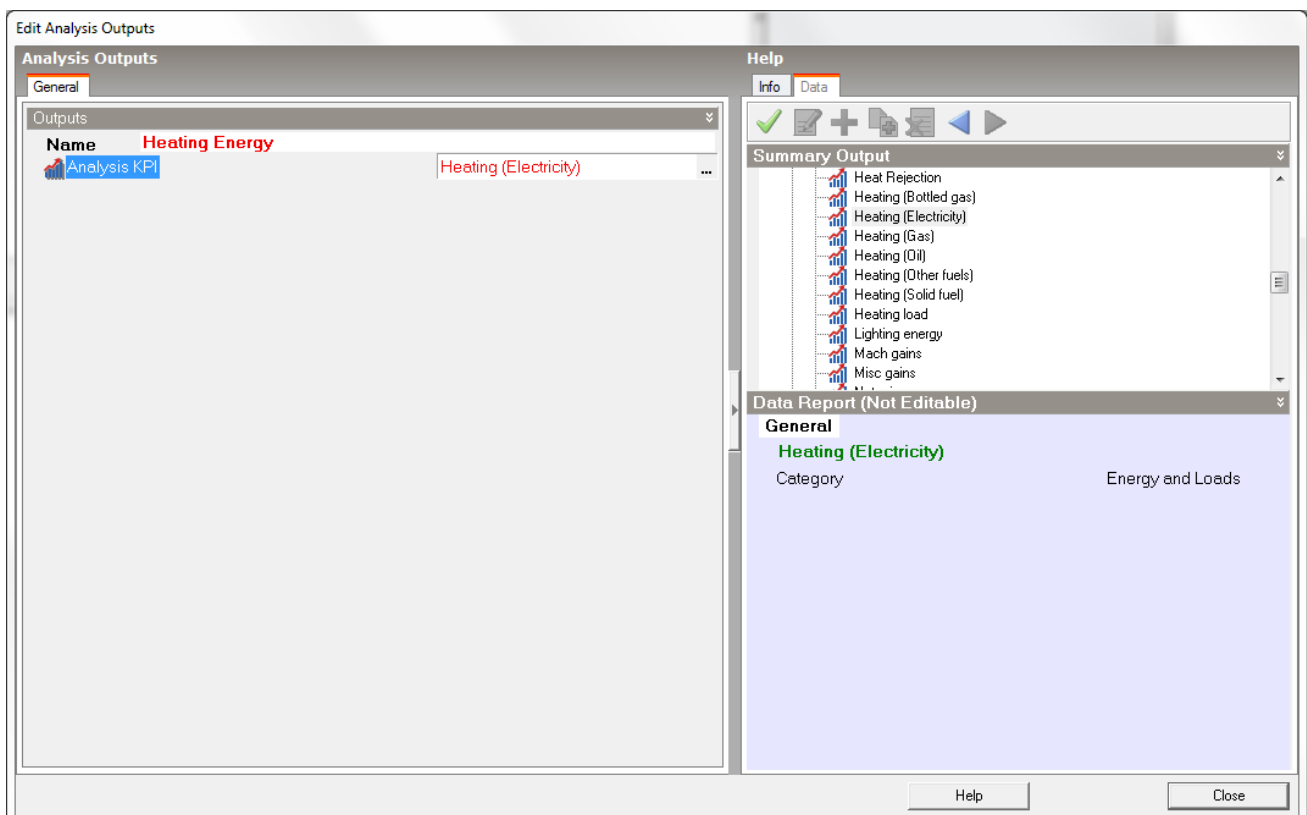
1. The uncertainty in heating energy use of the building due to the uncertainty of various inputs (uncertainty analysis).
2. The inputs that have the most influence on heating energy consumption (sensitivity analysis).

Output

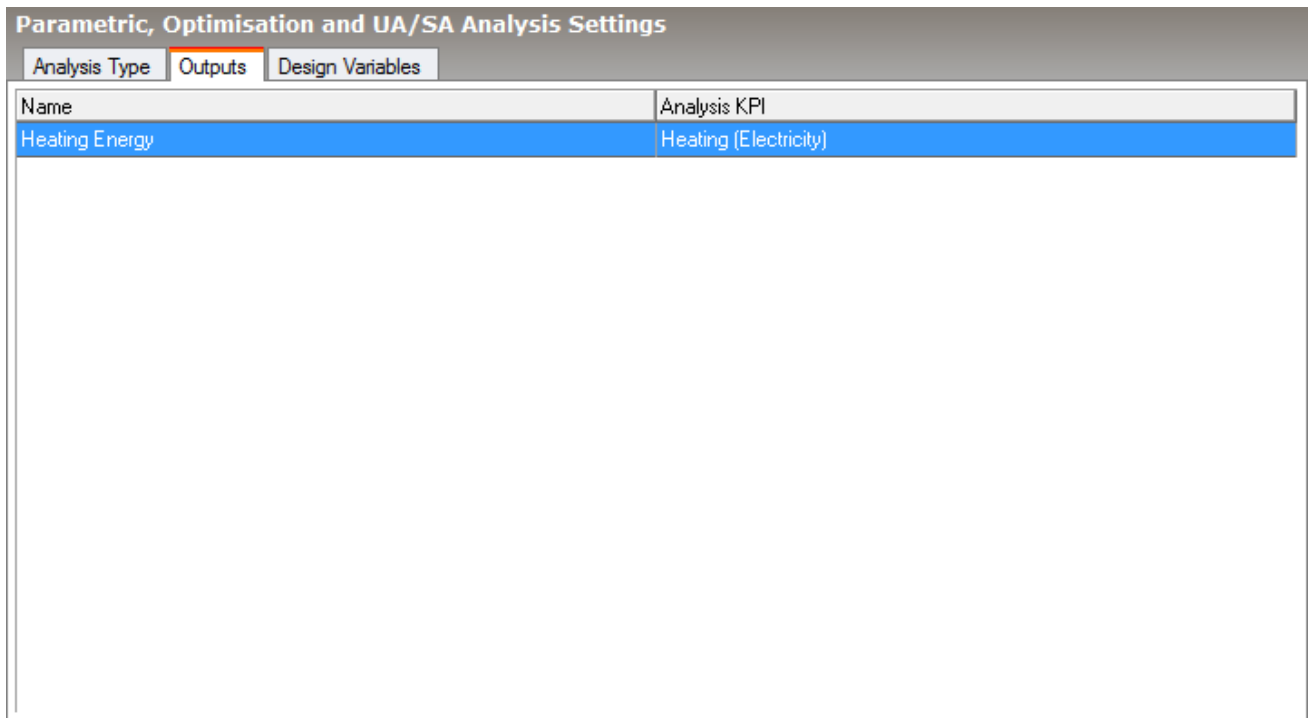
Add a new output by using the **Add Output** tool to access the [Edit Analysis Outputs](#) dialog. The procedure to select the **Heating (Electricity)** output is shown below.



When finished, the Heating (Electricity) Analysis Output Dialog should look like this:



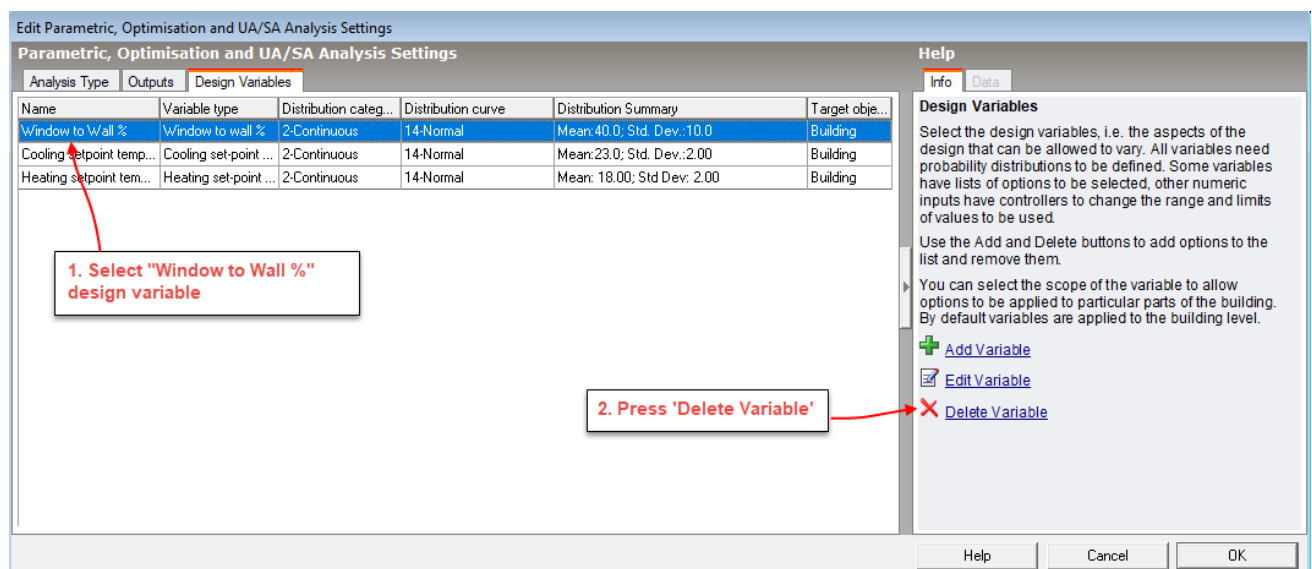
And when the Output dialog is closed the Outputs table should look like this:



Design Variables

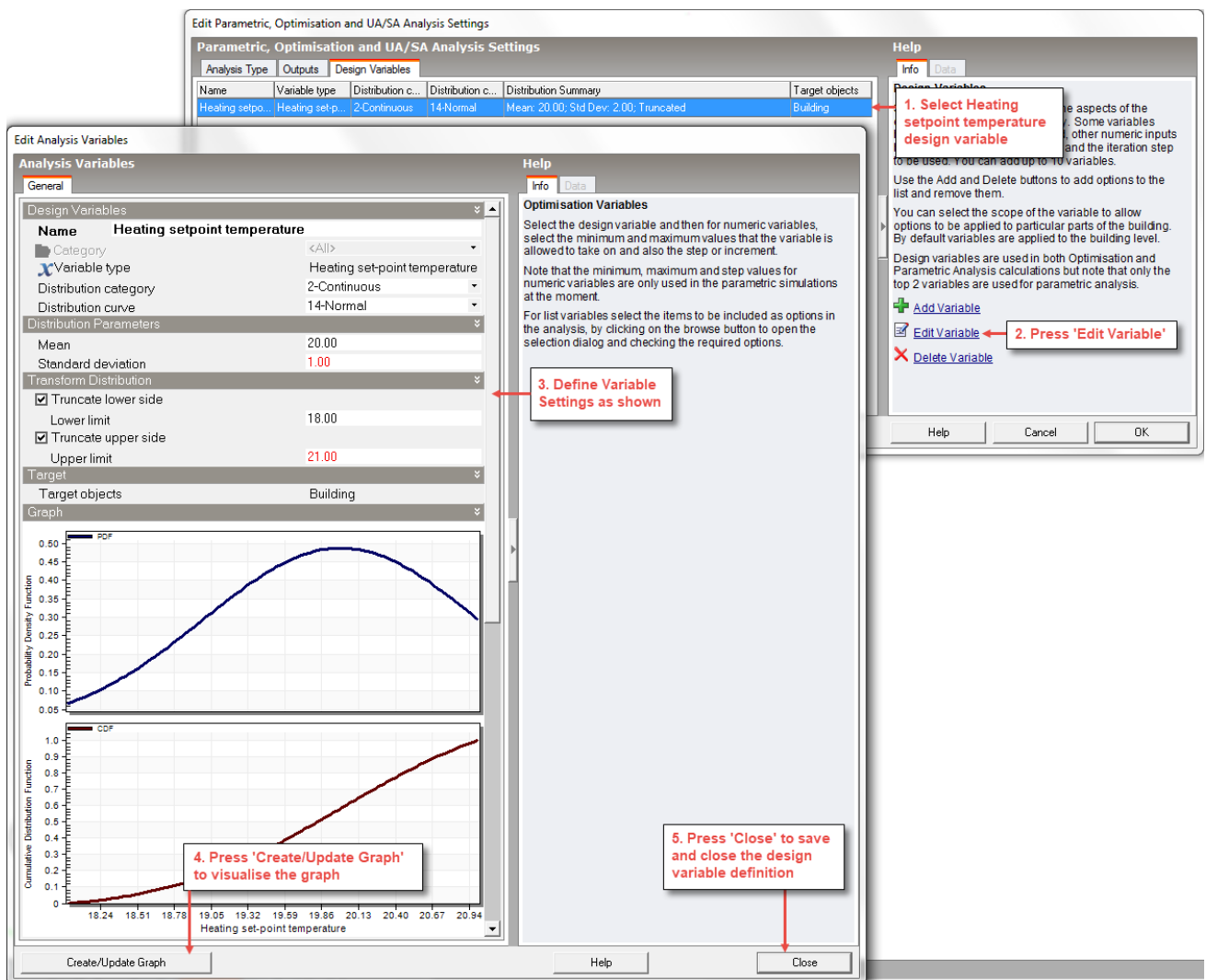
Go to the **Design Variables** tab. There are three pre-defined design variables. For this tutorial we need to remove the **Window to Wall %** design variable, modify the Heating and Cooling Setpoint variables and add 5 new variables.

1. To remove **Window to Wall %** and **Cooling setpoint** variables, follow the steps below:



2. Edit the existing **Heating setpoint temperature** variable by following the steps described below:

- Click on the **Edit variable** info panel link.
- Change the **Standard deviation** to 1.00.
- Change the **Truncation upper side, Upper Limit** to 21.00.
- Press **Create/Update Graph** to visualise the graph.
- Press the **Close** button to save the data and close the dialog.



4. Add a new **Equipment power density** variable by following the steps described below:

- Select **Add variable** on the **Design Variables** tab to open the **Edit Design variables** dialog.
- Select **Variable type** and on the right-hand list find the **Equipment power density** variable under the **Internal Gains/Lighting** folder. Select it.
- Set the **Distribution curve** to **18-Triangular**.
- Change the **Peak value** to 13.00, **Min value** to 10.00 and **Max value** to 20.00.
- Press the **Create/Update Graph** button to visualise the distribution.
- Press the **Close** button to save the data and close the dialog.

Edit Parametric, Optimisation and UA/SA Analysis Settings

Parametric, Optimisation and UA/SA Analysis Settings

Name	Variable type	Distribution c...	Distribution c...	Distribution Summary	Target objects
Heating setp...	Heating set-p...	2-Continuous	14-Normal	Mean: 20.00; Std Dev: 1.00; Truncated	Building

Edit Analysis Variables

General

Design Variables

Name	Variable type	Distribution category	Distribution curve	Peak value	Min Value	Max Value	Target	Target objects
Equipment power density	Equipment power density	2-Continuous	18-Triangular	13.00	10.00	20.00		Building

Graph

Probability Density Function (PDF) and Cumulative Distribution Function (CDF) plots for Equipment power density. The PDF plot shows a triangular distribution with a peak at 13.00. The CDF plot shows a cumulative distribution curve.

Help

Design Variables

Select the design variables, i.e. the aspects of the design that can be allowed to vary. Some variables have lists of options to be defined, other numeric inputs have minimum, maximum values and the iteration step to be used. You can add up to 10 variables.

Use the Add and Delete buttons to add options to the list and remove them.

You can select the scope of the variable to allow options to be applied to particular parts of the building. By default variables are applied to the building level.

Design variables are used in both Optimisation and Parametric Analysis calculations but note that only the top 2 variables are used for parametric analysis.

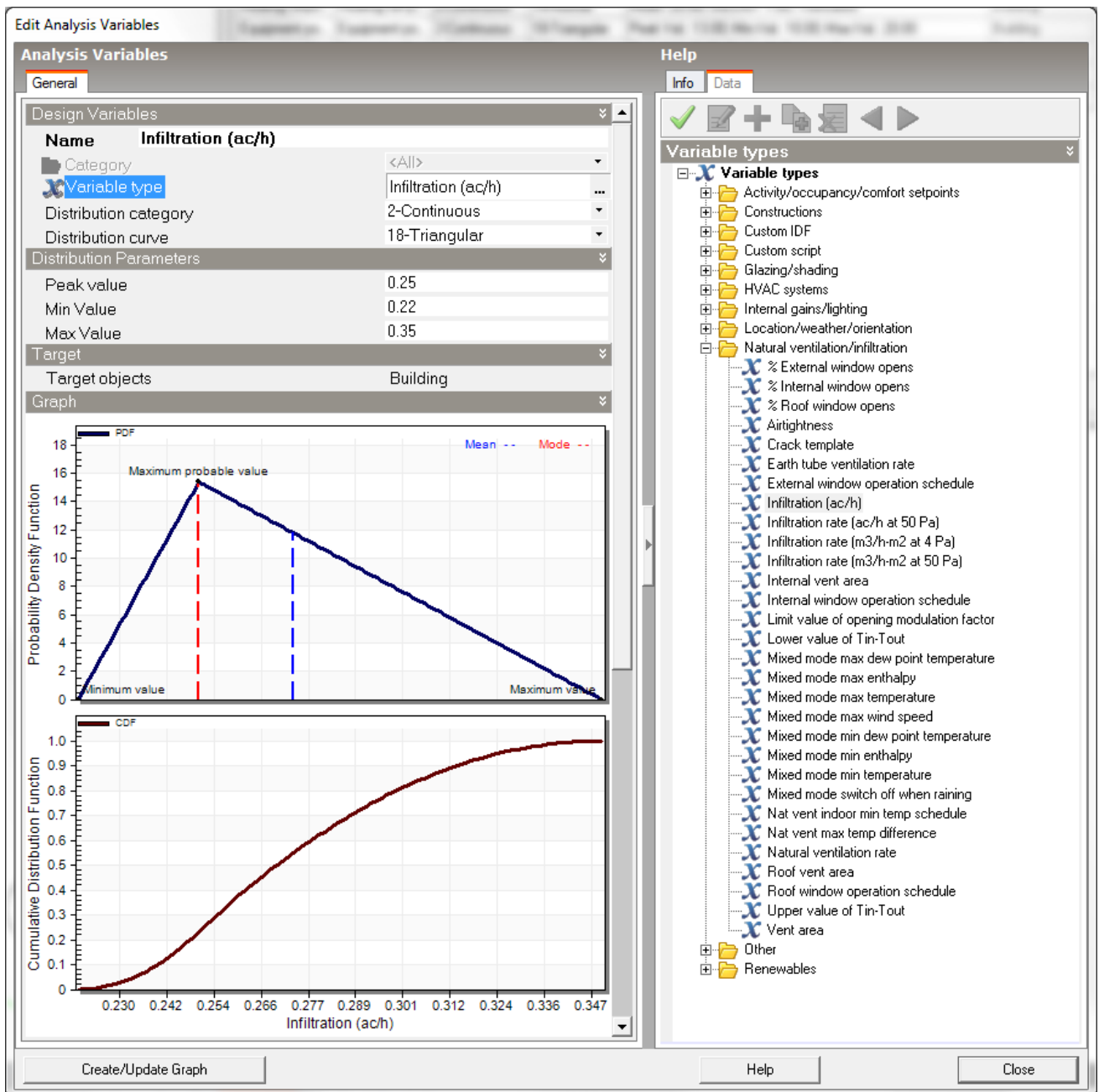
Buttons: Add Variable, Edit Variable, Delete Variable

Annotations:

- 2. Select 'Variable type'
- 4. Select 'Equipment power density' and press the green tick
- 5. Define Variable Settings as shown
- 3. Select Variable type as 'Equipment power density'
- 6. Press 'Create/Update Graph' to visualise the graph
- 7. Press 'Close' to save and close the design variable definition
- 1. Press 'Add Variable'

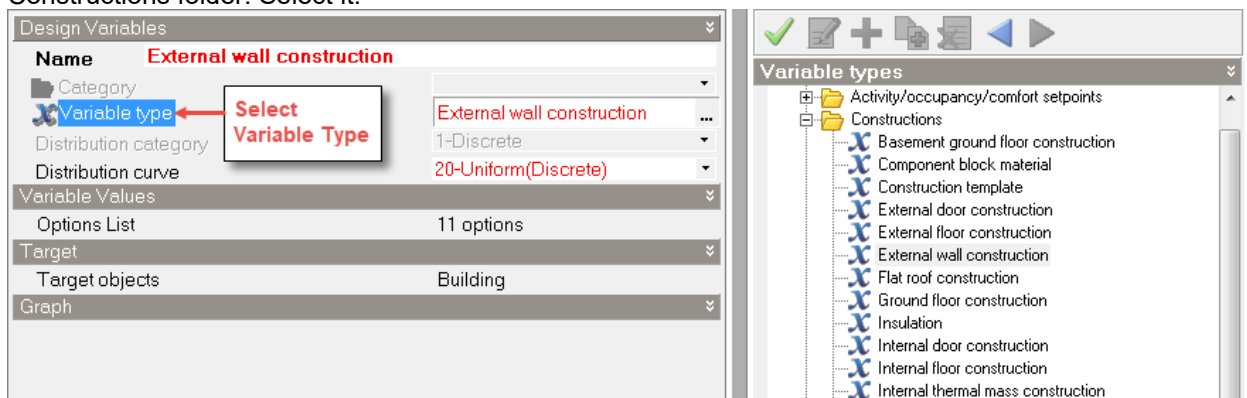
5. Add a new Infiltration (ac/h) variable by following the steps described below:

- Select **Add variable** on the **Design Variables** tab to open the **Edit Design variables** dialog.
- Select **Variable type** and on the right-hand list find the **Infiltration (ac/h)** variable under the **Natural ventilation/Infiltration** folder. Select it.
- Change the **Distribution curve** to **18-Triangular**.
- Change the **Peak value** to 0.25, **Min value** to 0.22 and **Max value** to 0.35.
- Press the **Create/Update Graph** button to visualise the distribution.
- Press the **Close** button to save the data and close the dialog.



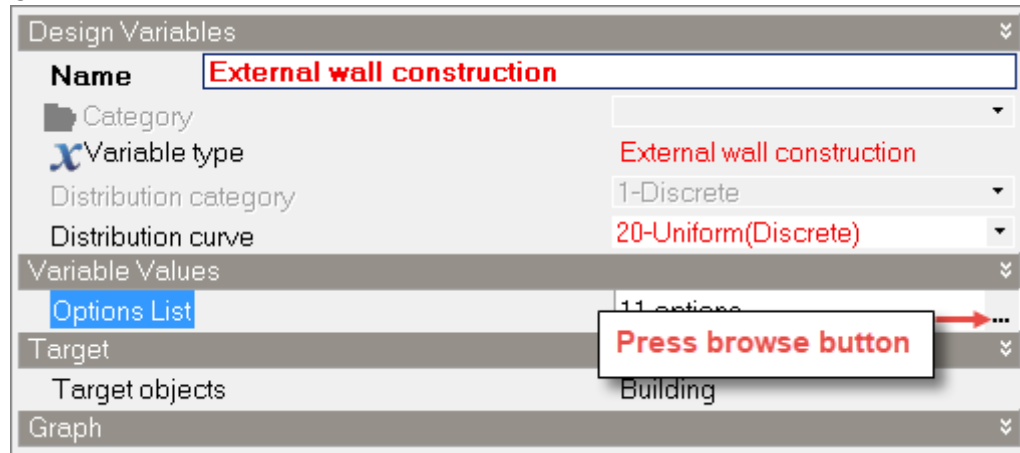
6. Add a new **External Wall Construction** variable by following the steps below:

- Select **Add variable** on the **Design Variables** tab to open the **Edit Design variables** dialog.
- Select **Variable type** and from the right-hand list find the **External wall construction** variable under **Constructions** folder. Select it.

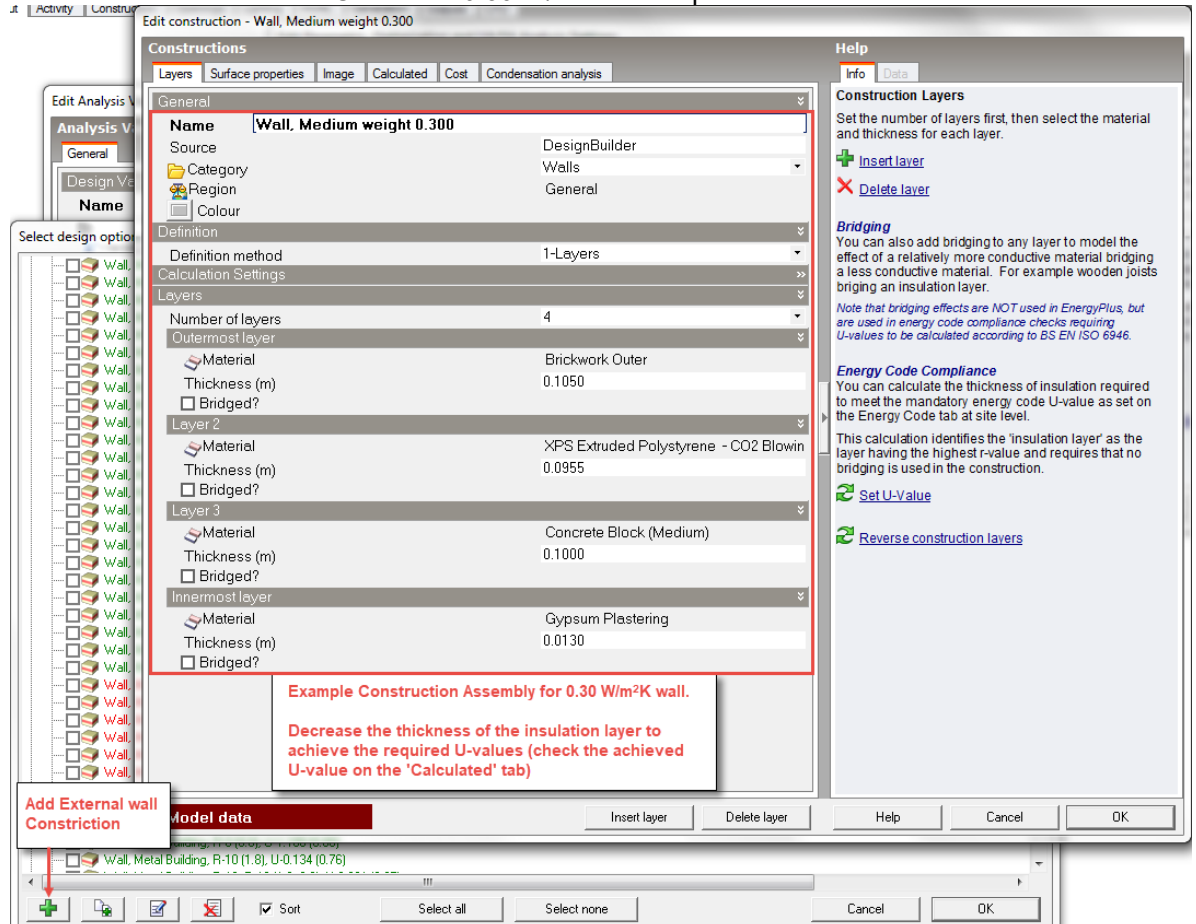


- Click on the **Option list** and follow these steps:

- o Click on the Browse Button:

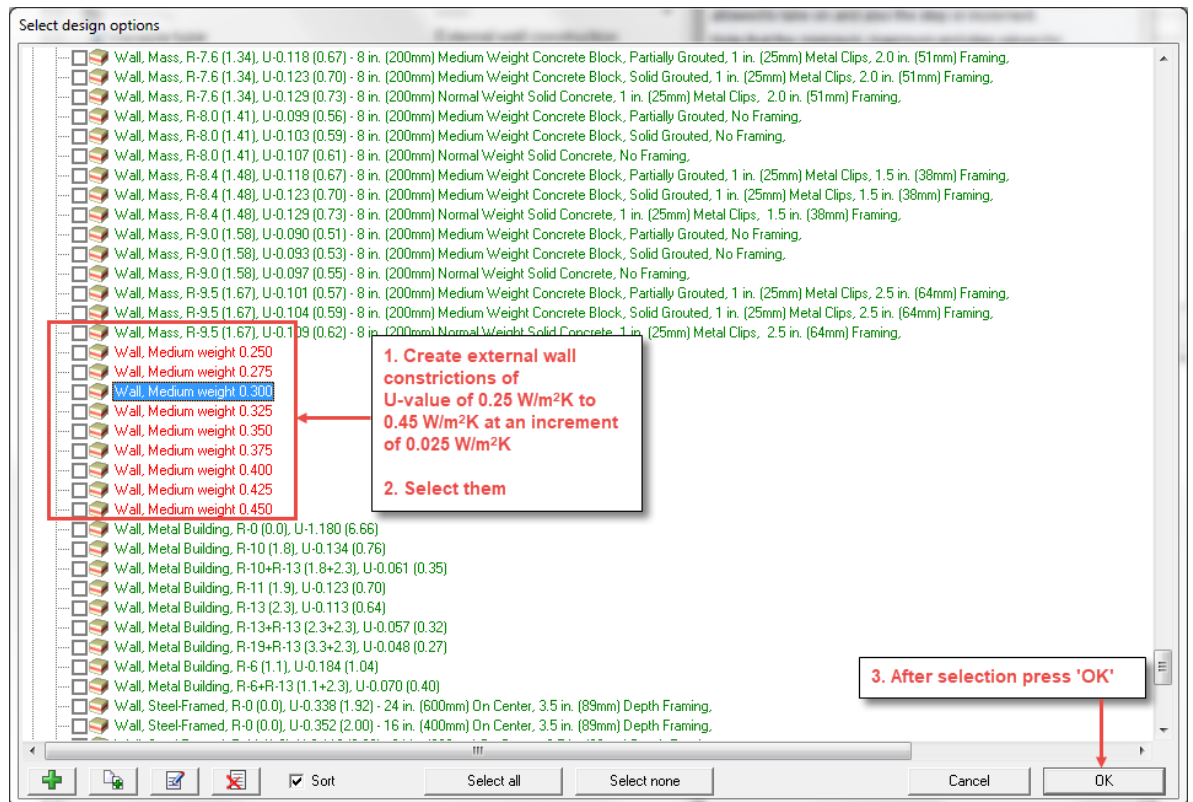


- o Add a new construction for U-value of 0.30 W/m²K. Example shown below:

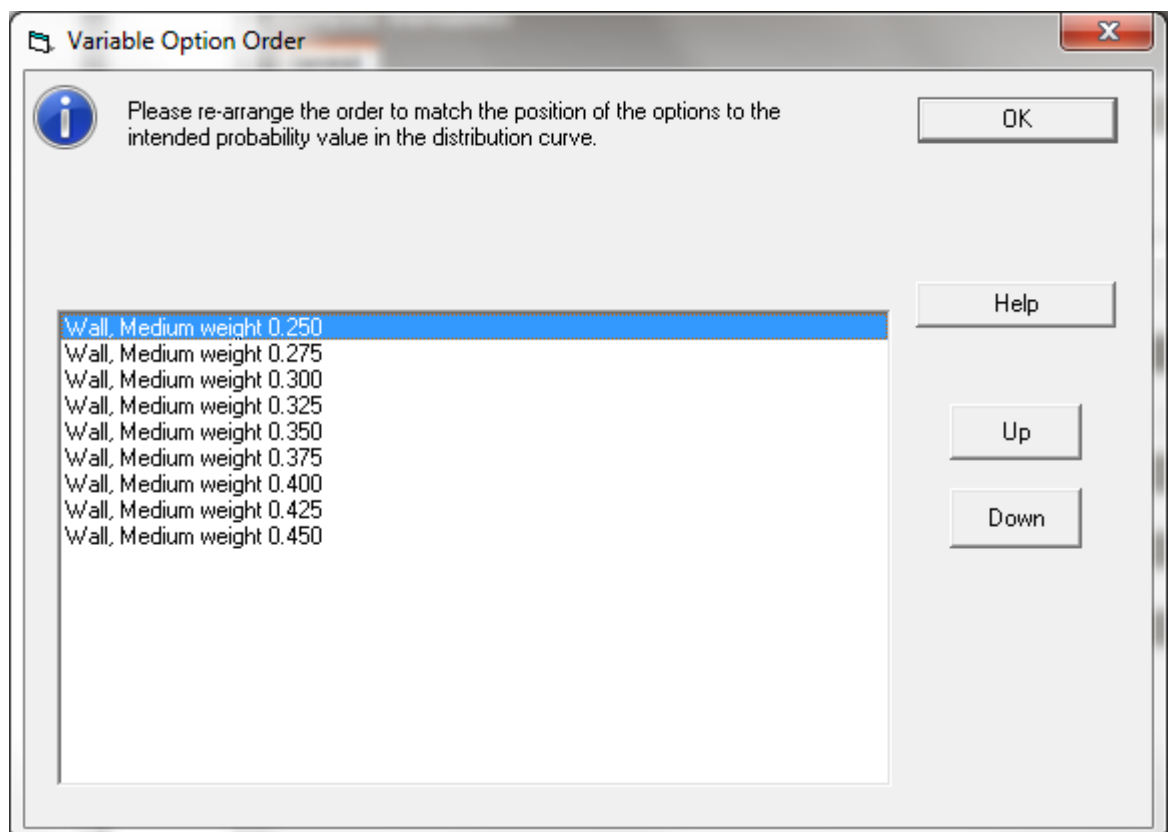


Note: You can use the **Set U-value** tool in the Info panel to achieve a specific U-value.

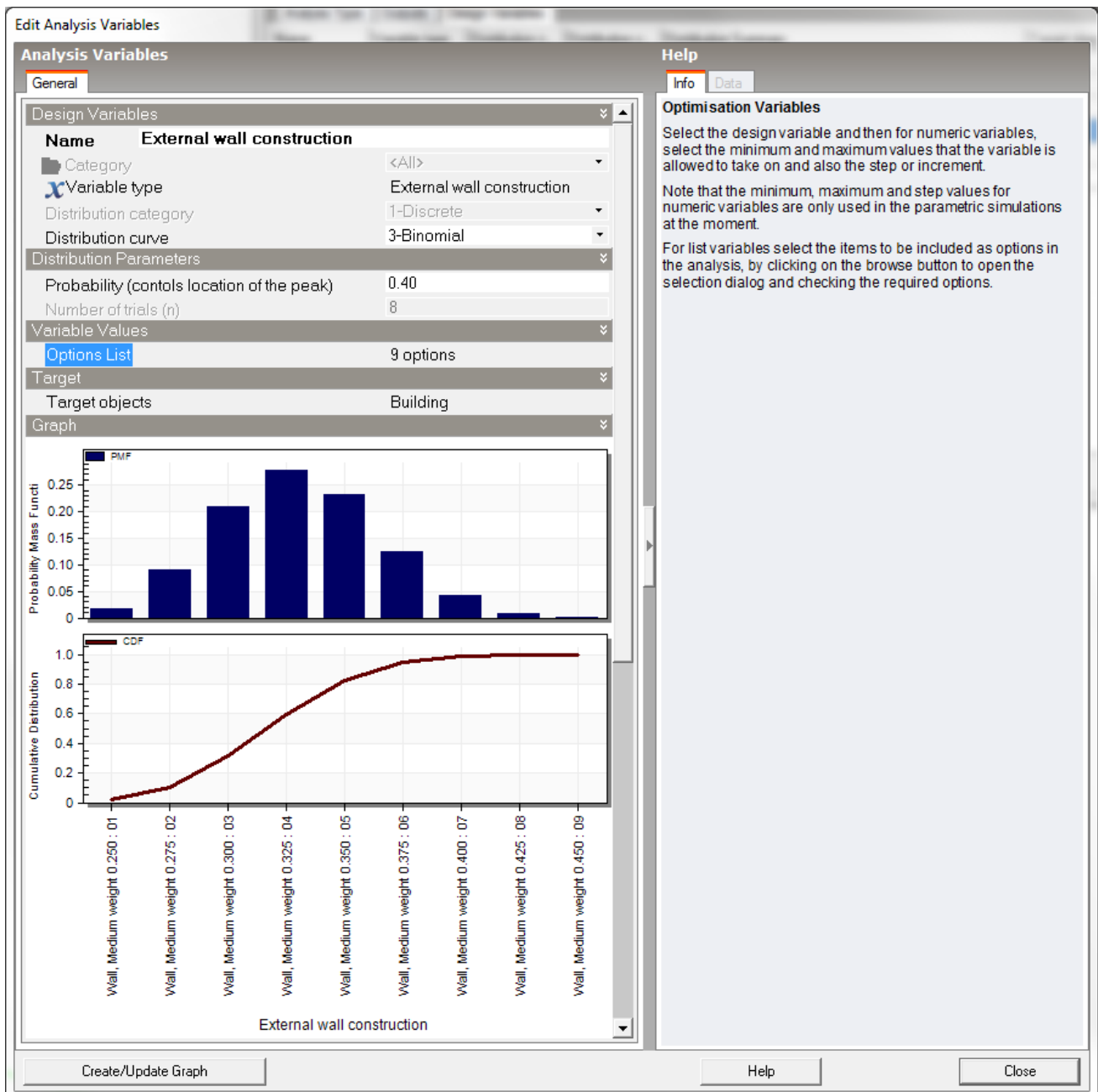
- o Similarly, create 9 external wall constructions of U-value of 0.25 W/m²K to 0.45 W/m²K with an increment of 0.025 W/m²K.



- o Configure the options list to ensure that selection list is in order of increasing U-value order and press OK to close the dialog:



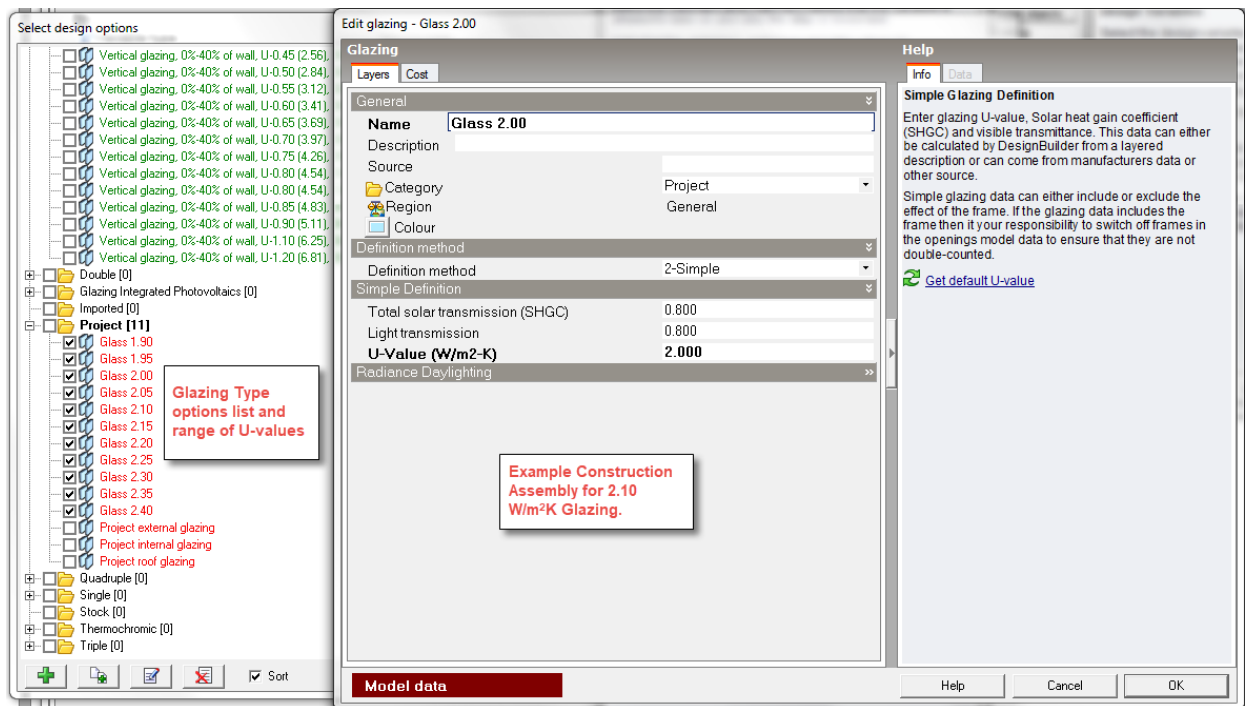
- Change the **Distribution curve** to **3-Binomial**.
- Change the **Probability** to 0.40.
- Press the **Create/Update Graph** button to visualise the distribution.
- Press the **Close** button to save the data and close the dialog.



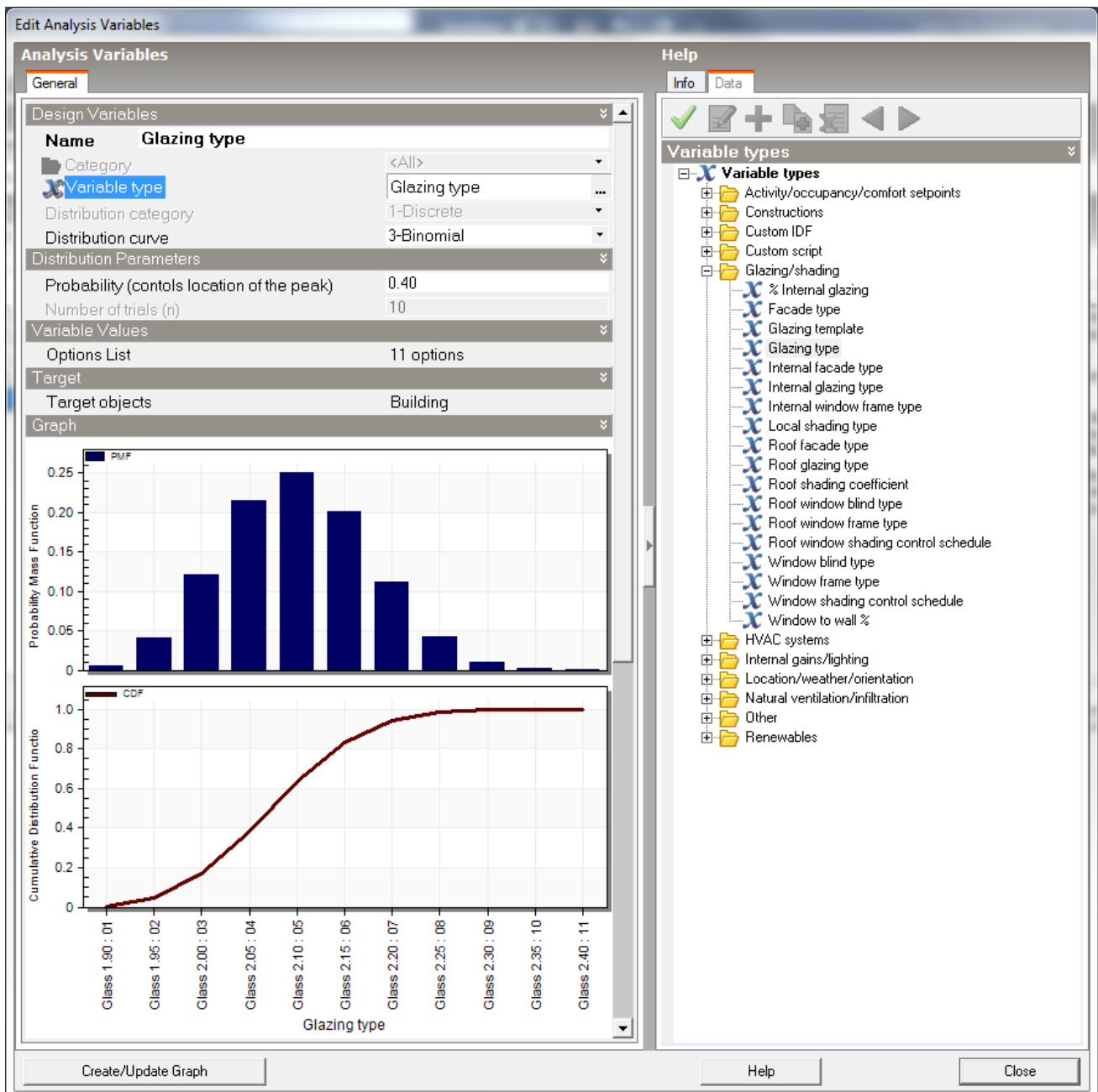
7. Similarly add a new **Glazing type** variable by following the steps below:

- Select **Add variable** on the **Design Variables** tab to open the **Edit Design variables** dialog.
- Select **Variable type** and, from the right-hand list, find the **Glazing type** variable under **Glazing/Shading** folder. Select it.
- Follow the steps described above for **External Wall Construction** to create a list of glazing components, ordering the list by ascending U-value. Create and select 11 Glazing type of U-value of 1.90 W/m²K to 2.40 W/m²K with an increment of 0.05 W/m²K.

Note: Use the **2-Simple** glazing Definition method where you can enter the U-value directly.

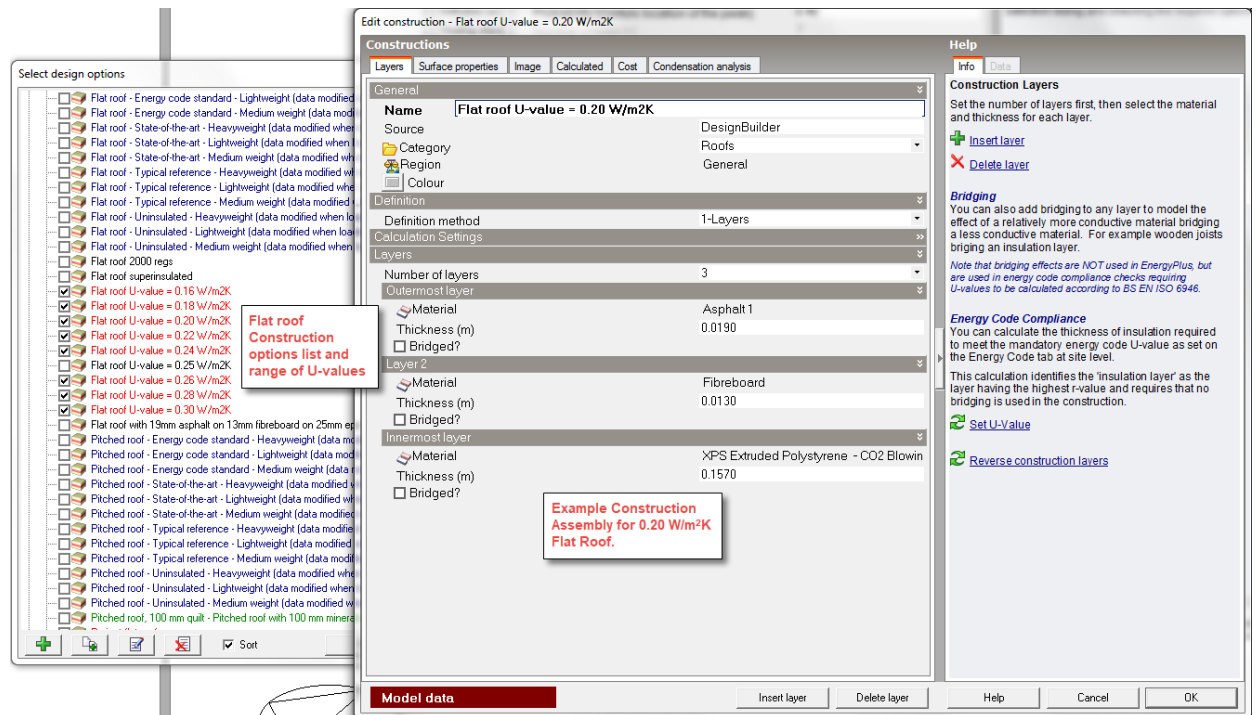


- Change the **Distribution curve** to **3-Binomial**.
- Change the **Probability** to 0.40.
- Press the **Create/Update Graph** button to visualise the distribution.
- Press the **Close** button to save the data and close the dialog.

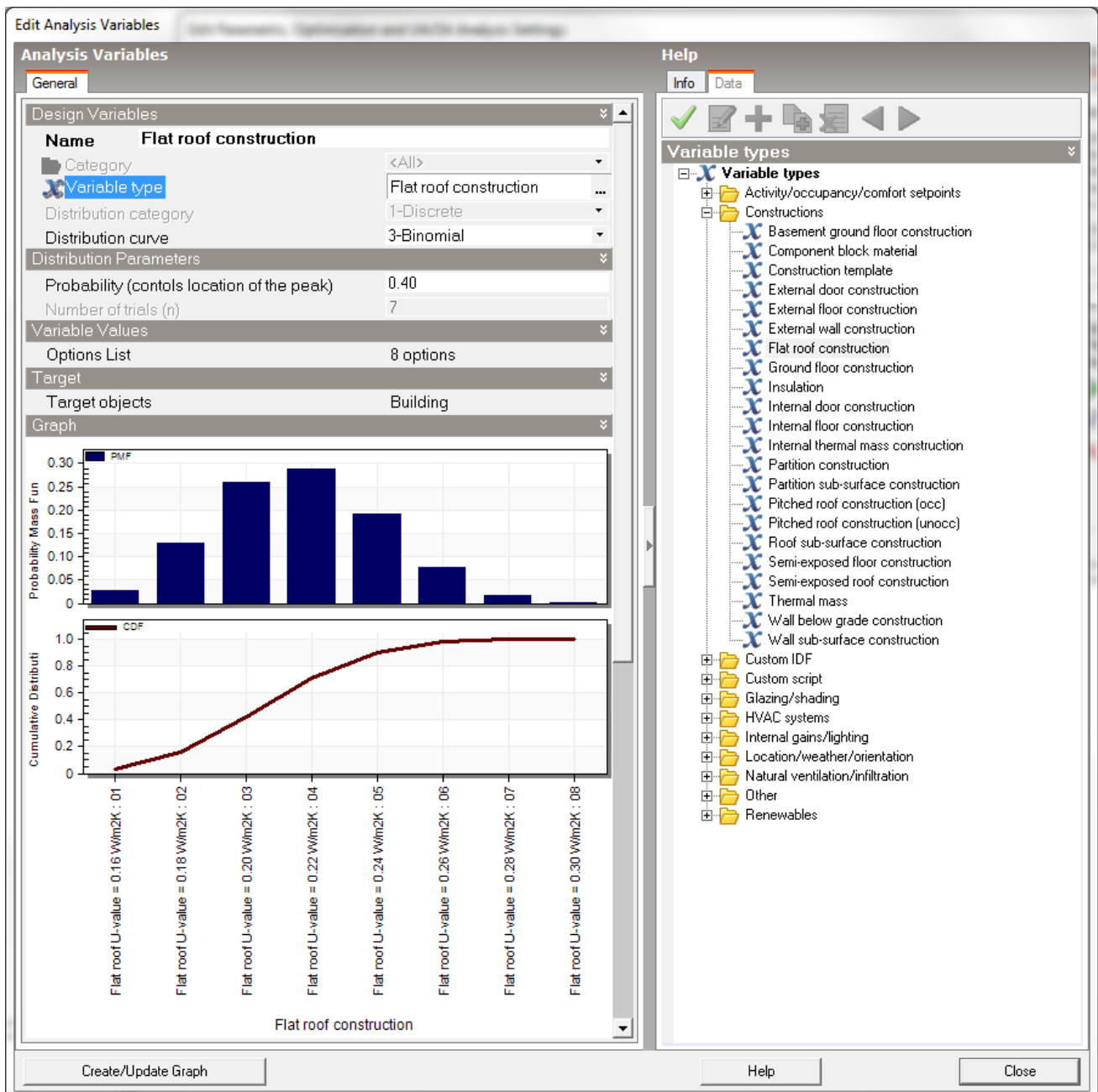


8. Similarly add a new **Flat roof construction** variable.

- Select **Add variable** on the **Design Variables** tab to open the **Edit Design variables** dialog
- Select **Variable type** and, from the right-hand list, find the **Flat roof construction** variable under the **Constructions** folder. Select it.
- Use the steps defined for **External Wall Construction** to create and configure an option list, ordering it in ascending U-value. Create and select 9 Flat roof constructions of U-value of 0.16 W/m²K to 0.30 W/m²K with an increment of 0.02 W/m²K.



- Change the **Distribution curve** to **3-Binomial**.
- Change the **Probability** to **0.40**.
- Press the **Create/Update Graph** button to visualise the distribution.
- Press the **Close** button to save the data and close the dialog.



9. The final list of design variables should look like this:

Parametric, Optimisation and UA/SA Analysis Settings

Analysis Type	Outputs	Design Variables			
Name	Variable type	Distribution c...	Distribution c...	Distribution Summary	Target objects
Infiltration (ac...	Infiltration (ac...	2-Continuous	18-Triangular	Peak Val.: 0.25; Min.Val.: 0.22; Max.Val.: 0.35	Building
Heating setpo...	Heating set-p...	2-Continuous	14-Normal	Mean: 20.00; Std Dev: 1.00; Truncated	Building
Equipment po...	Equipment po...	2-Continuous	18-Triangular	Peak Val.: 13.00; Min.Val.: 10.00; Max.Val.: 20.00	Building
External wall ...	External wall ...	1-Discrete	3-Binomial	Prob. Val.: 0.40; Trials: 8; Options: 9	Building
Flat roof const...	Flat roof cons...	1-Discrete	3-Binomial	Prob. Val.: 0.40; Trials: 7; Options: 8	Building
Glazing type	Glazing type	1-Discrete	3-Binomial	Prob. Val.: 0.40; Trials: 10; Options: 11	Building

Step 4 - Open Calculation Options Dialog

Having confirmed the uncertainty and sensitivity analysis options, the next dialog to open will be the Calculation options. Change the number of runs to 500 for this detailed analysis and, otherwise, use default settings.

Calculation Description

From

Start date1 Jan

To

Start date31 Dec

☐ Pause on errors
 ☐ Quietly retry if server becomes unavailable

Uncertainty/Sensitivity Analysis Options

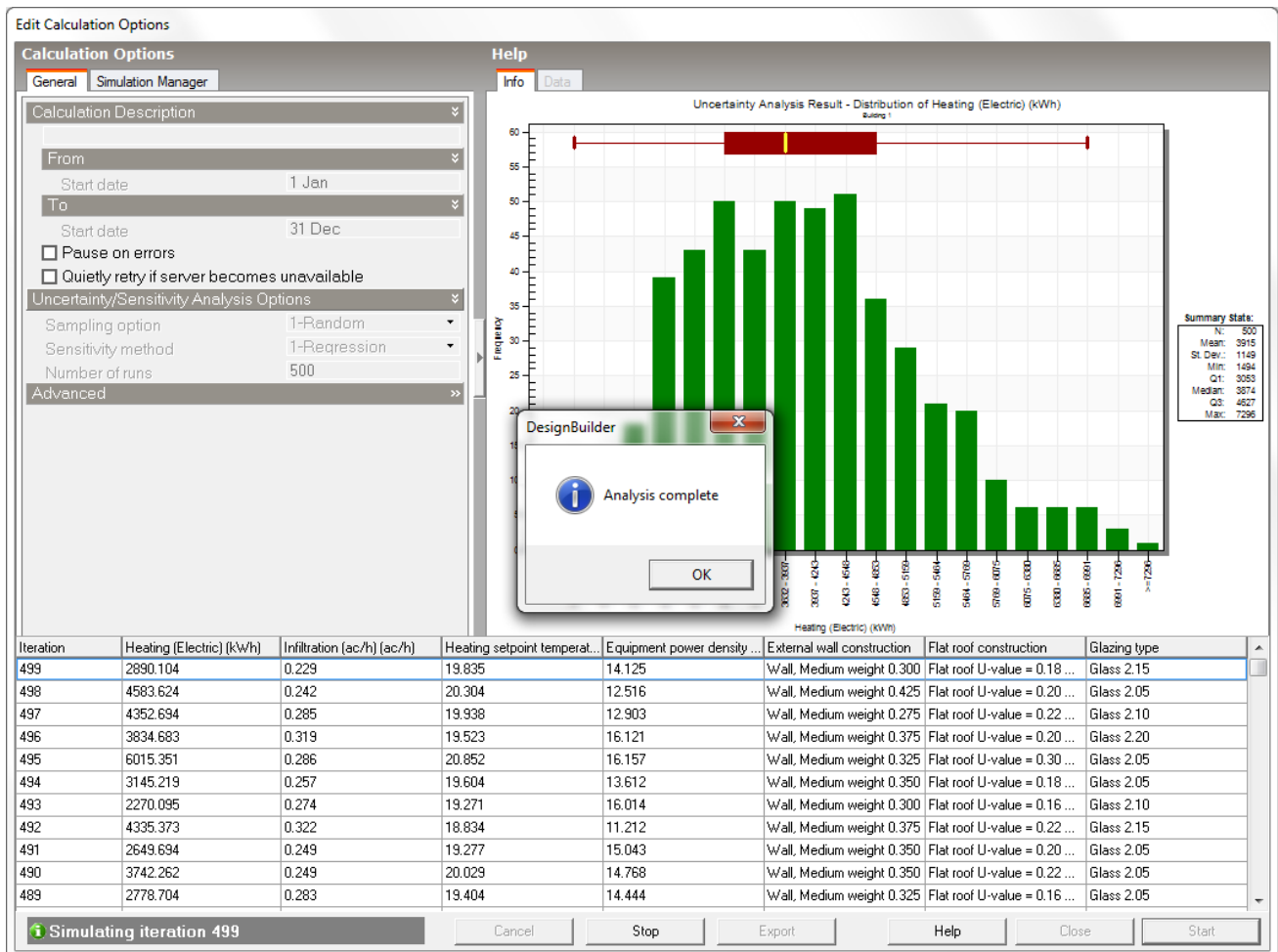
Sampling option1-Random
 Sensitivity method1-Regression
 Number of runs500

Step 5 - Start Uncertainty And Sensitivity Analysis

Once you have completed your review of the options press the **Start** button in the bottom right of the dialog. The uncertainty and sensitivity analysis process will involve running a lot of simulations. With the recommended settings there will be 500 runs with 20 runs being plotted in each batch, updating the histogram as each batch is finished. That means that 25 batches will be run! For our simple office building this shouldn't take too long with the Simulation manager running the simulations in parallel. However, if time is an issue it is usually worth starting with a smaller number of runs. During the runs uncertainty outputs are plotted at the end of each batch.

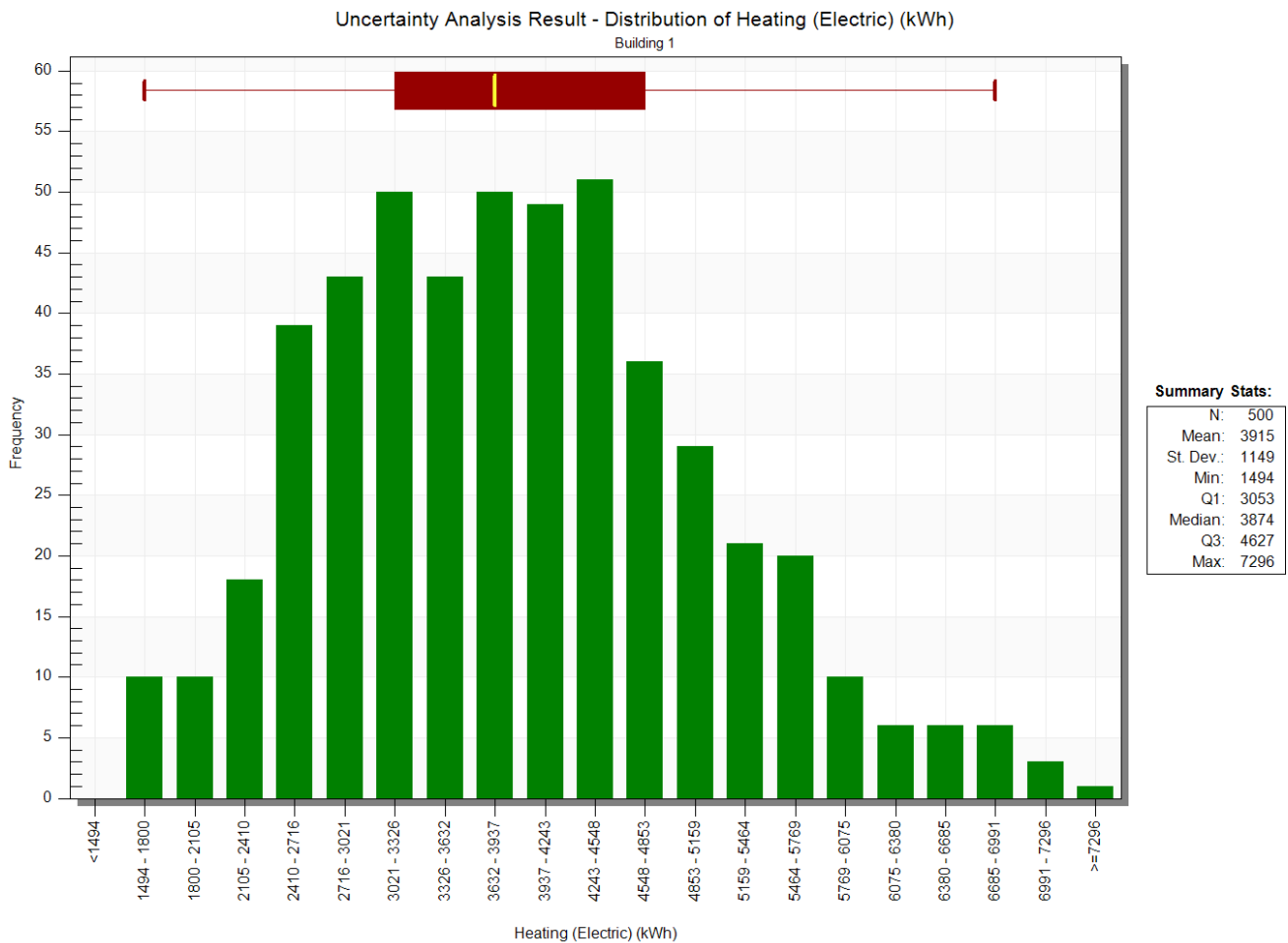
Note: While it is possible to stop the simulations before the UA/SA process is incomplete, this is not recommended as in this case uncertainty analysis results could be incorrect and sensitivity analysis results will not be created.

After all the runs have been finished an alert is displayed as shown in the screenshot below. Press OK to dismiss the alert and then Press **Close** to return to the **Optimisation +UA/SA** tab of the simulation screen to analyse the results.



Step 6 - Analyse Results

Uncertainty analysis results



Keep the default **1-Uncertainty analysis** Analysis type in the Display options panel result to view UA results first. You should be able to see from the histogram and summary statistics in the right hand panel on the graph that:

- The total heating energy has a high degree of variability, from a minimum value of 1494 kWh to a maximum value of 7296 kWh.
- While some of the data points are outliers, the mean value of Heating Energy use is 3915 kWh with standard deviation of ± 1149 kWh.

Sensitivity analysis results

To visualise the sensitivity analysis results, change the **Analysis type** to **2-Sensitivity Analysis**.

Display Options

General Detailed

Uncertainty/Sensitivity Analysis v

☒ Show analysis

Analysis type 2-Sensitivity an

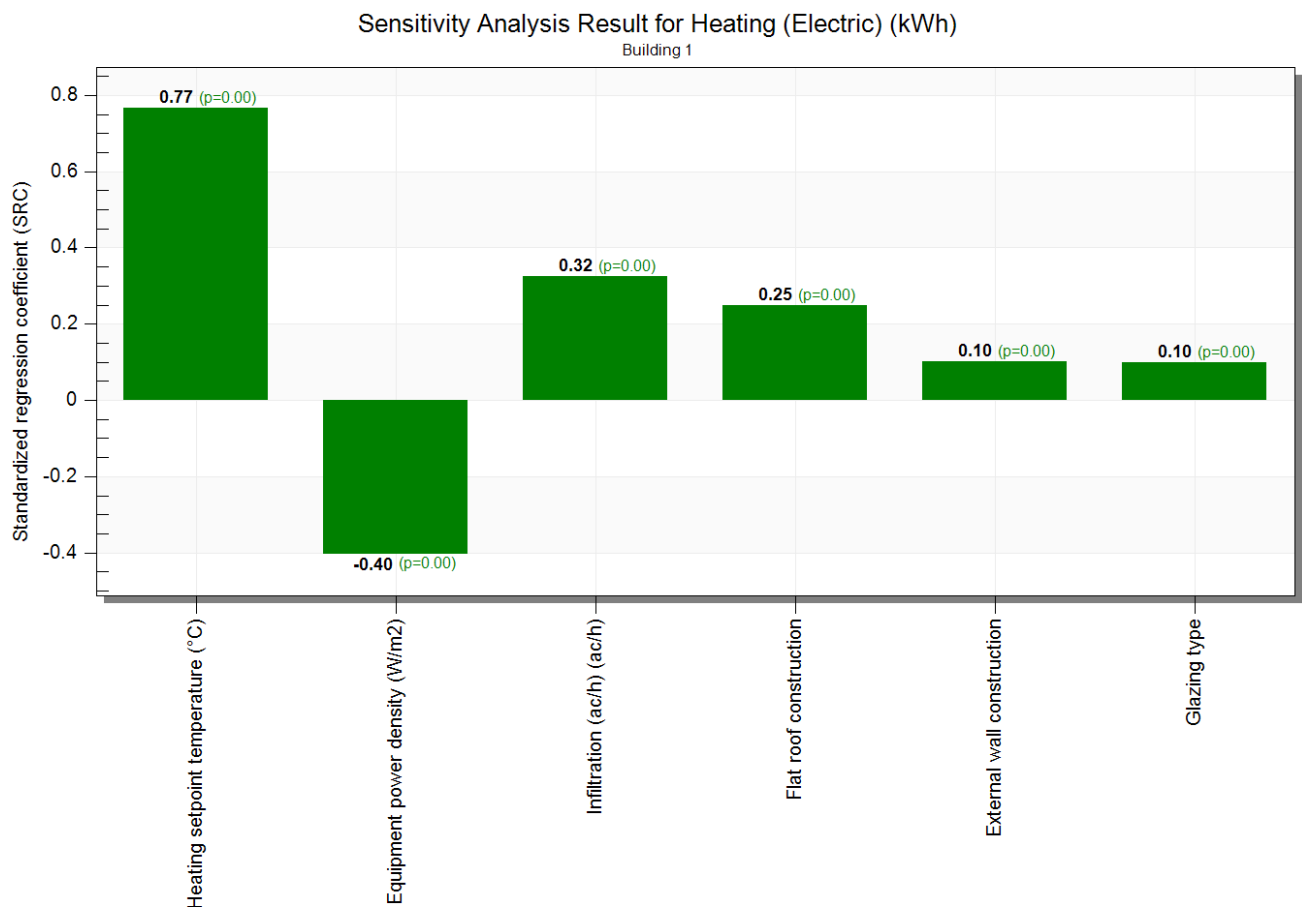
Display type 1-Graph (SRC)

☐ Show data grid

Graph Controls v

Output selected 1-Heating (Elect

Input order 2-Decreasing I



The Sensitivity analysis results are shown by default in decreasing order of importance with the most important variable on the left and the least important on the right. The value written above each bar in black bold is the standardised regression coefficient (SRC), telling us the relative importance of each input. In green to the right of each SRC value is the **p-value**. You should see in our example that the p-value for all inputs are 0. Note that a low p-value (<0.05) indicates a high level of confidence in the result for that variable.

- The sensitivity analysis reveals that for this building, based on the ranges of uncertainty in the input variables defined above, occupant behaviour contributes more to uncertainty in heating energy consumption than does fabric performance. We can draw this conclusion because the **Heating setpoint** and **Equipment power density** have higher SRC than any of the fabric performance parameters.
- Infiltration is the third most influential parameter and therefore, among the fabric performance variables factors, envelope sealing is the most critical factor in determining the heating energy use. The positive value of the SRC, as expected, tells that there is a direct relationship between Heating energy use and infiltration rate (an increase in infiltration leads to an increase in heating energy use).
- Glazing has the widest U-value range (1.9 W/m²K to 2.4 W/m²K) among the envelope elements (Wall and Roof range from 0.25 W/m²K to 0.45 W/m²K and 0.16 W/m²K to 0.30 W/m²K). However, among the three, the roof has the most influence over the heating energy because of the larger exposed area it has relative to the others.
- Equipment power density has an inverse relationship with heating energy use. This confirms that as the equipment load increases, the internal gains increase which reduces the heating energy use.
- The analysis suggests that managing occupant behaviour should be a priority at the building operation stage to minimise the performance variability.
- The two most important construction stage measures to improve the Heating energy use of this building are a) adequate envelope sealing and b) proper roof insulation. Another interpretation could be that if these two variables are measured more accurately then the uncertainty in model predictions can be reduced.

ASHRAE 90.1 Reporting

See: "ASHRAE 90.1 Appendix G PRM Reference" on page 2135

Data Visualisation

The Data Visualisation tools allow you to view a wide range of simulation outputs rendered using false colours on the model surfaces and openings. This way of presenting outputs can be useful when a graphic of summary of data over the whole building (or part of a building) is required. Example applications include:

- Solar incident radiation over the year on each surface which can be useful to help with locating solar panels or to identify windows or roof surfaces in particular need of shading.
- Checking that internal gains for all zones over the year correspond with what is expected.
- Viewing total annual solar gains through all building windows to identify windows that may need upgraded shading.

The data visualisation tools are available from the Simulation screen on the Data visualisation tab. You can run a simulation from this tab by pressing the **Update** toolbar icon.

See also the [Analysing results in DesignBuilder's interface](#) tutorial

Make Settings

When running simulations from the Data Visualisation tab you will see a **Make settings** link in the info panel of the Simulation calculation options dialog.

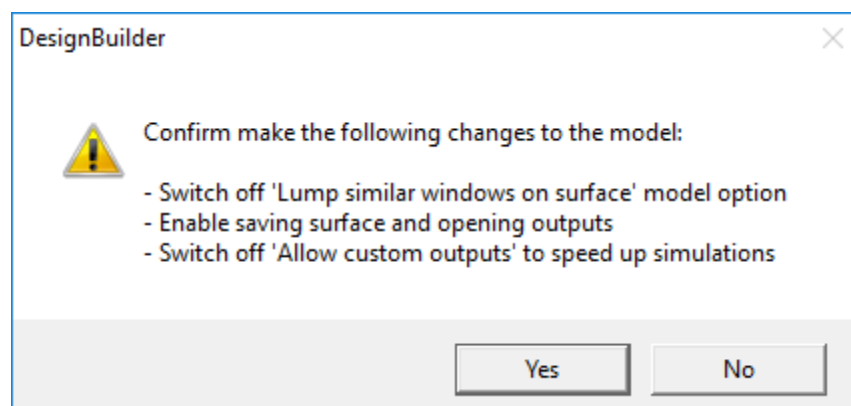
Data Visualisation

Before generating simulation results suitable for data visualisation you are advised to make some settings. You can ensure sensible default settings by clicking on the 'Make settings' link below.

Note this will ensure that the 'Lump similar openings on surface' option is switched off, which it currently is not.

[Make settings](#)

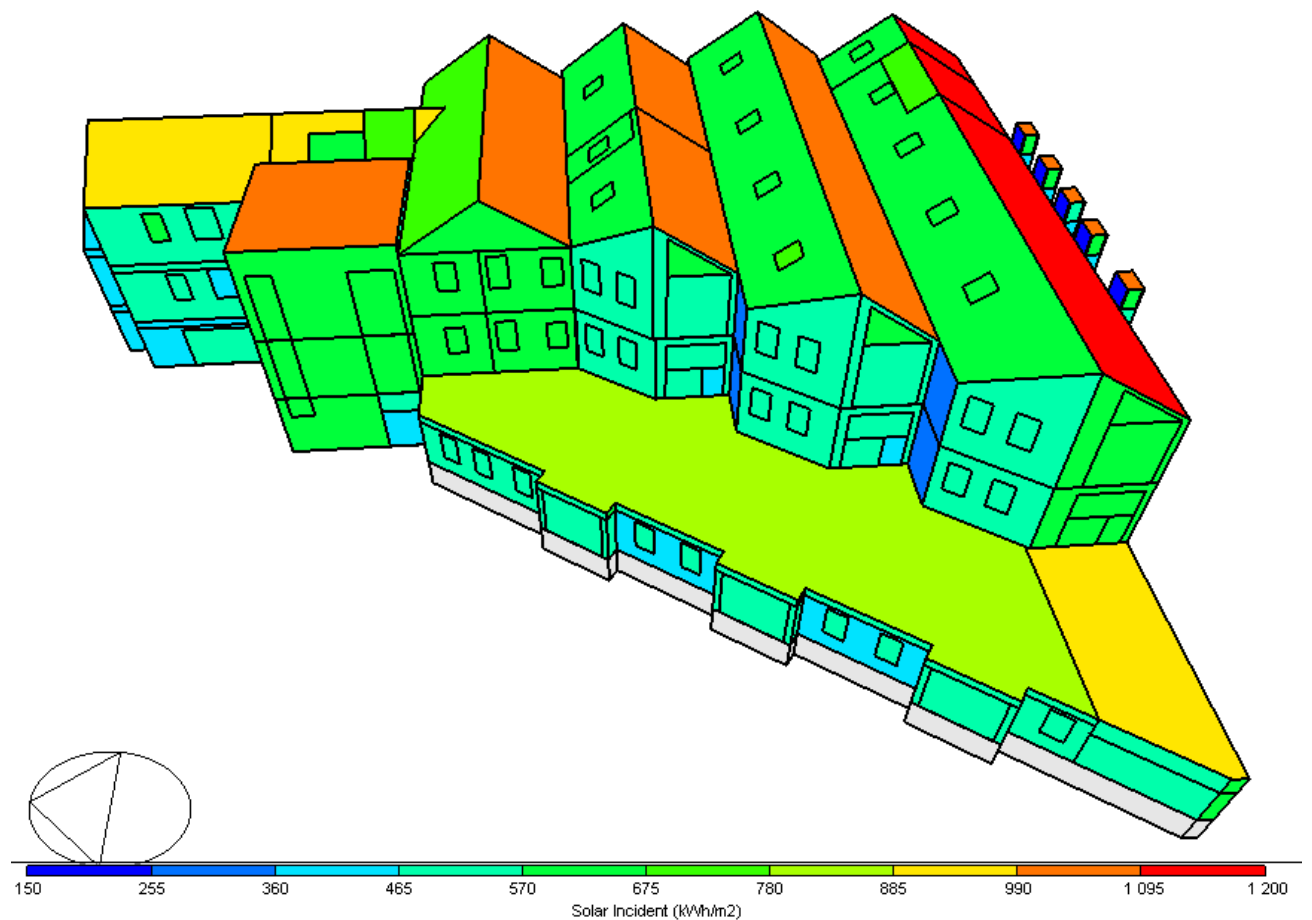
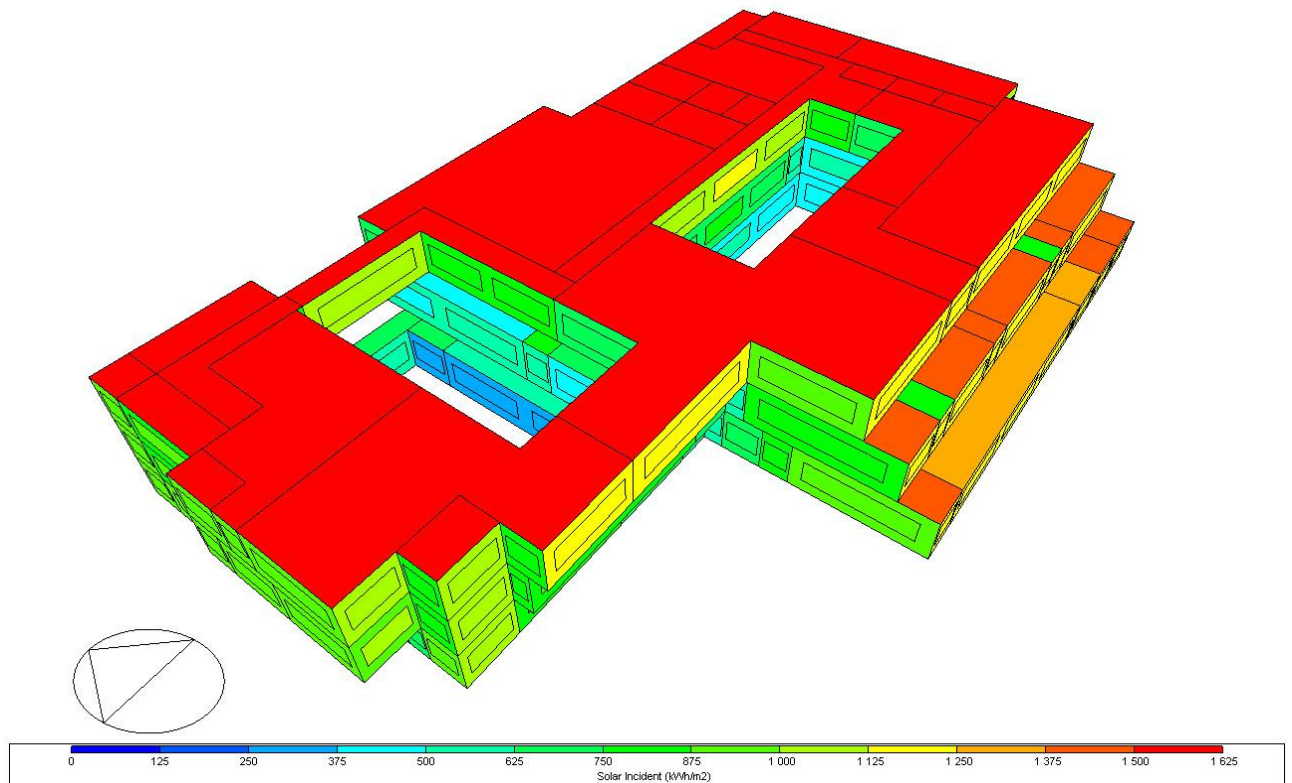
If the note is shown in red it means that DesignBuilder recommends checking that your output options are set up correctly for data visualisation. It is usually easiest simply to click on the **Make settings** info panel link to automatically make the recommended settings, in which case you will be asked to confirm the changes to the model with the message below:



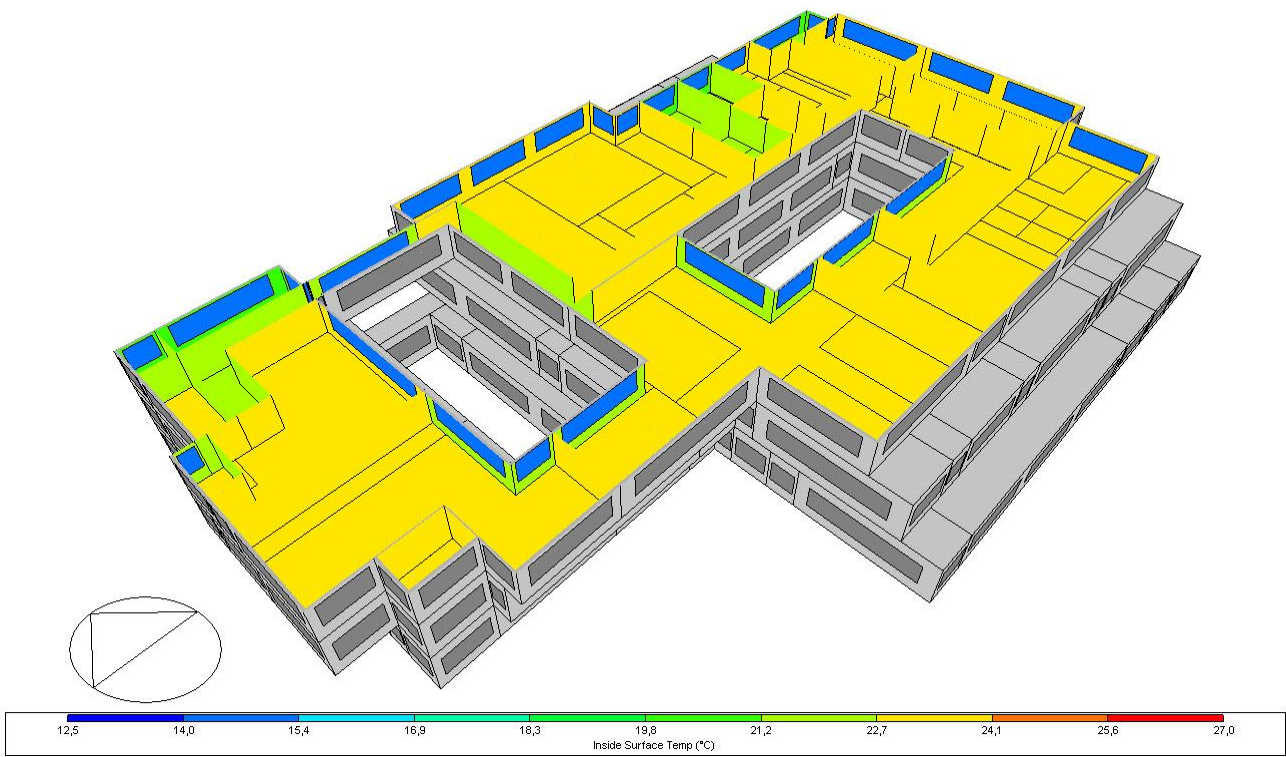
Confirm by selecting the Yes button, or No to cancel the changes.

Sample Outputs

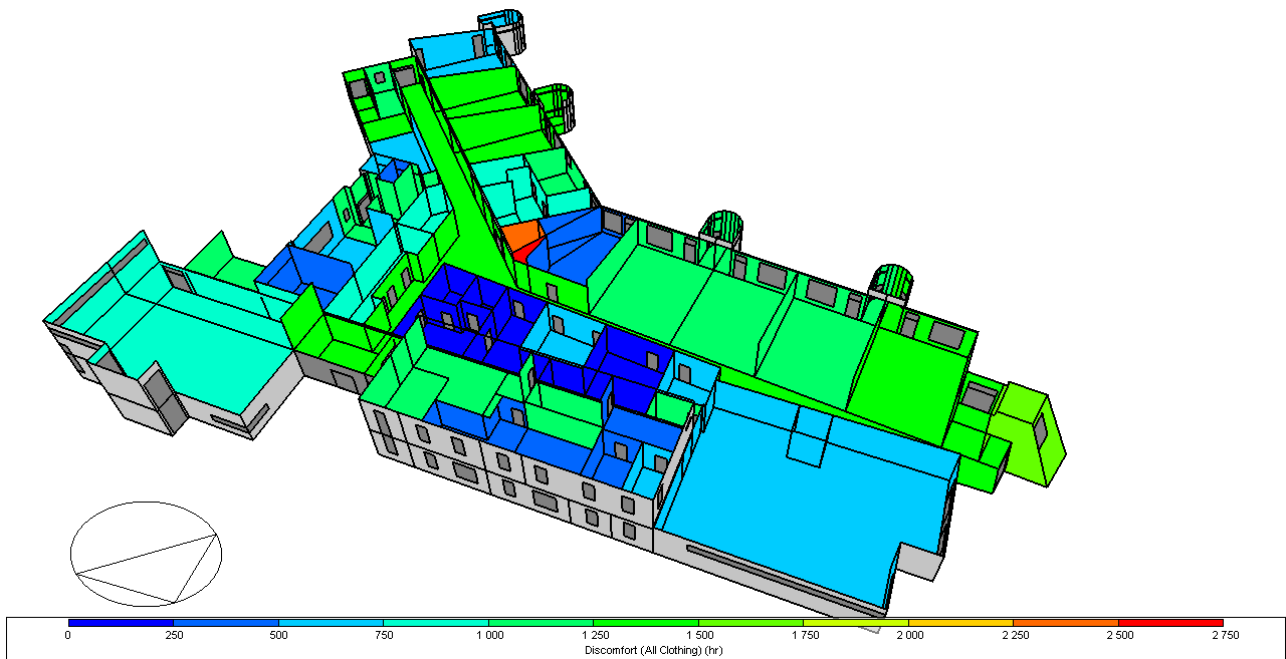
Some examples of data visualisation outputs that can be generated are shown below.



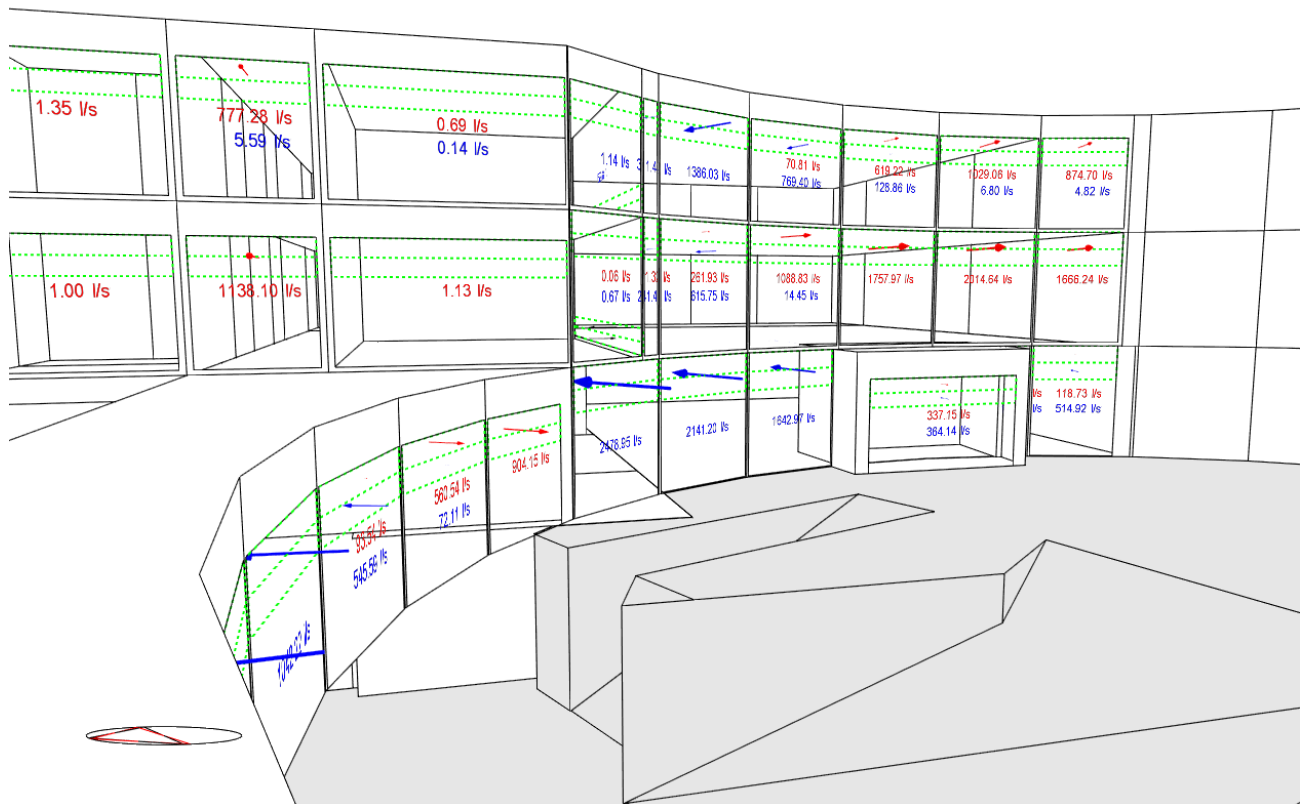
Annual Solar Incident Radiation Totals



Annual Average Inside Surface Temperatures



Annual Discomfort Hours for each Zone



Airflow Vectors showing air entering the building (blue arrows) air leaving the building (red arrows).

Plot type

The plot types available are:

- **1-External surfaces** where selected data is plotted on each external surface, e.g. Solar incident radiation.
- **2-Internal surfaces** where selected data for internal surfaces is plotted on each surface, e.g. internal surface temperatures.
- **3-Zone data** which allows calculated summary zone data to be plotted, e.g. total occupancy gains for each zone.
- **4-Airflow data** to view airflow vectors at windows, vents, doors and holes when using Calculated natural ventilation. This option allows airflow vectors to be viewed for hourly and sub-hourly intervals only. See also "Airflow Visualisation Controls" on page 1905

The Plot type selected affects the data that can be selected for display in the Output type option below.

Output type

Depending on which **Plot type** is selected (above) you can select the output type to be viewed from the options listed below.

Output list

Zone data

- Glazing gain
- Walls gain
- Ceilings gain
- Floors gain
- Solid floors gain
- Partitions gain
- Roofs gain
- Rooflights gain
- Ext floors gain
- Sens cooling energy

- Ext airflow gain
- Ext infiltration gain
- Ext nat vent gain
- Ext mech vent gain
- AHU heating energy
- Cooling energy
- Fans
- Pumps
- Preheat energy
- Reheat energy
- Radiant heater energy
- DHW
- Lights energy
- Process energy
- Catering energy
- Display lighting gains
- General lighting gains
- Miscellaneous gains
- Process gains
- Catering gains
- Computer + Equipment gains
- Occupancy gains
- Solar gains internal windows
- Solar gains external windows
- Zone sensible heating
- Zone sensible cooling
- Air Temp
- Radiant Temp
- Comfort Temp
- Occupancy Latent Gain
- Discomfort ASHRAE 55 (sum clo)
- Discomfort ASHRAE 55 (win clo)
- Discomfort ASHRAE 55 (all clo)

External surface data

- Outside Surface Temperature
- Solar Incident
- Ext Sunlit %
- External Convective Heat Transfer Coefficient

Internal surface data

- Surface gain
- Inside Surface Temperature
- Internal Convective Heat Transfer Coefficient
- Solar Transmitted
- Airflow in
- Airflow out

Airflow data

- Airflow in
- Airflow out

Interval

Select the interval of the data that you would like to view from the list:

- **1-Annual** to view annual data.

- **2-Monthly** to view monthly total and average data. In this case you must also select the month.
- **3-Daily** to view daily total and average data. In this case you must also select the month and day of the month.
- **4-Hourly** to view hourly total and average data. In this case you must also select the month, day of the month and hour.
- **5-Sub-hourly** sub-hourly total and average data. In this case you must also select the month, day of the month and time.

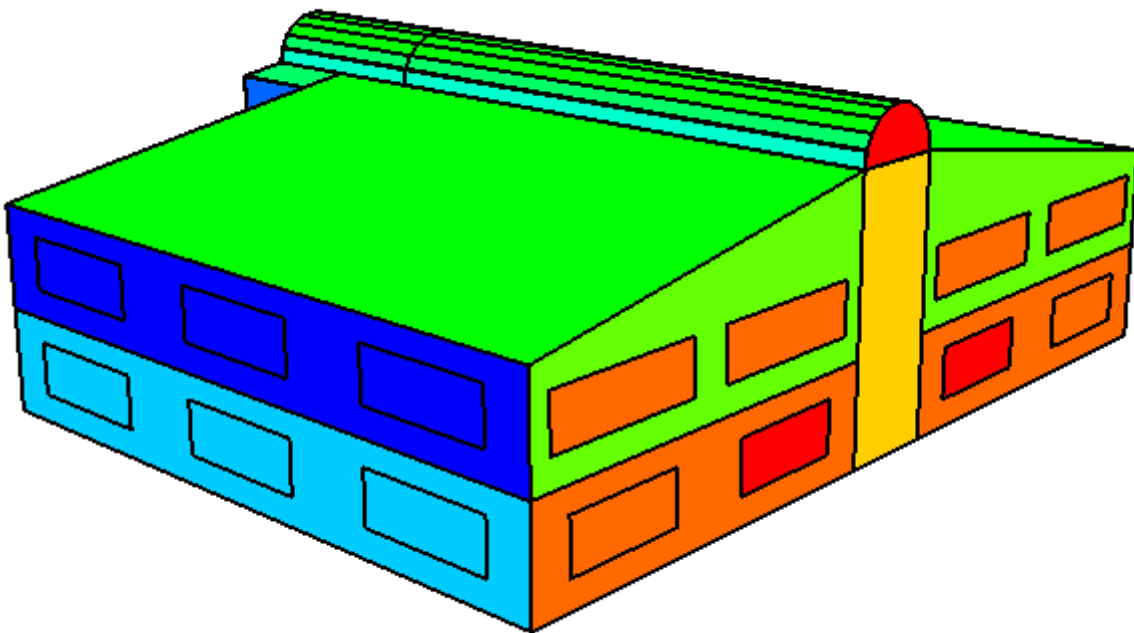
For example to view annual solar incident radiation set the Plot type to **1-External surfaces**, the Output type to **Solar incident** and the interval to **1-Annual**.

Normalise by area

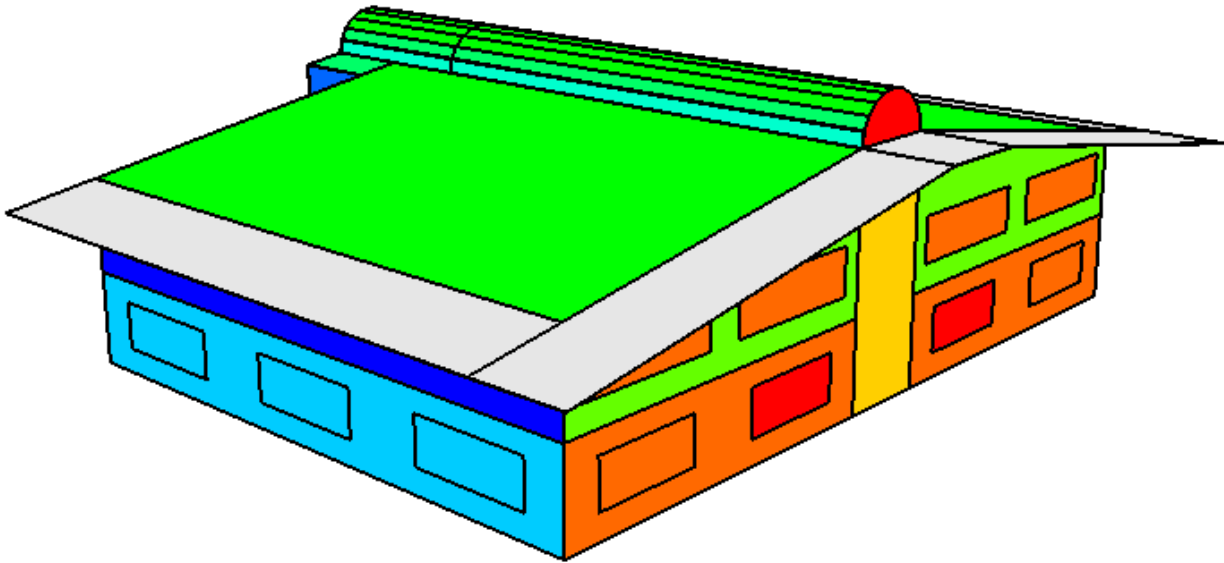
Check this checkbox if you would like results to be displayed normalised by area, i.e. divided by the area of the surface or opening.

Show component blocks and assemblies

To see component blocks and assemblies on the data visualisation check this checkbox.



Option switched off



Option switched on

Legend

DesignBuilder provides you with control over how the legend is generated and displayed.

Range method

Select from the options:

- **1-Default** where the most appropriate lower and upper values are calculated in such a way that the legend includes a manageable set of value intervals using "round" numbers avoiding unnecessary decimal places etc.
- **2-Min/max** where the actual calculated minimum and maximum values plotted are used as the minimum and maximum range without any rounding applied. This can be useful to find the maximum and minimum values.

The range method cannot be changed while the **Override range** option below is selected.

Override range

Use this option to override the default calculated range. DesignBuilder calculates an appropriate range for the scale based on the data, but sometimes it can be useful to override that. For example if one zone has very high occupancy gains compared with all the others, it may be best to reduce the upper range Max value (below) to allow the variation across the other zones to be seen, avoiding the distortion of effect of the "outlier" zone on the overall scale.

Min value

When selecting the **2-Min/max** Range method, the actual minimum and maximum values are used to define the scale. Checking the Override range option allows you to see the calculated min and max values. This can be useful data in itself, but you can also adjust the minimum value as required by typing in the required lowest value to be used in the scale here.

Max value

See above under Min value.

Min band colour

Select the colour to be used for the minimum band. Choose from:

- **1-Blue,**
- **2-Cyan,**
- **3-Green,**
- **4-Yellow**

By default the colours will range from blue (min value band) to red (max value band), through green, yellow and orange.

Note: the colour for the Min band selection must have a lower number than for the Max band selection (below).

Max band colour

Select the colour to be used for the minimum band. Choose from:

- **2-Cyan,**
- **3-Green,**
- **4-Yellow,**
- **5-Red**

Airflow Visualisation Controls

When the Plot type is set to **4-Airflow data**, natural ventilation airflows are displayed for each opening using coloured vectors. The length of the arrows indicates the airflow rate. Arrows are coloured as explained below.

- **Blue** - air entering the building from outside (fresh air)
- **Red** - air leaving the building to the outside (exhaust air)
- **Black** - air transfer between zones (interzone air)

For an example airflow visualisation, see under [Sample Outputs](#) above.

Vector scale factor

You can use the vector scale factor to adjust the size of all of the airflow vectors. Enter a higher number to see larger vectors and vice-versa.

Show opening airflow labels

If you would like to see the airflow rate printed on the rendered view then check this checkbox.

Label size

When Show opening airflow labels is checked you can select the size of the labels from the list:

- **1-Small**
- **2-Medium**
- **3-Large**

Show North arrow

Check this checkbox if you would like the North arrow to be printed on the rendered view.

Note: The direction of the wind is shown by the direction of the red arrow which is printed towards the bottom left of the screen regardless of whether the North arrow is displayed. For example, when the red arrow points upwards, the wind is moving in the direction of the arrow from the South towards the North, i.e. a southerly wind.

Using The Data Visualisation Tools

As well as all the usual view control tools (orbit, zoom etc), some extra tools are provided to help you get the most out of the data visualisation data plots and these are described below.

Note: You will need to use either or both of the Surface removal or Section cut tools to access data when the **Plot type** is **2-Internal surfaces** or **3-Zone data**.

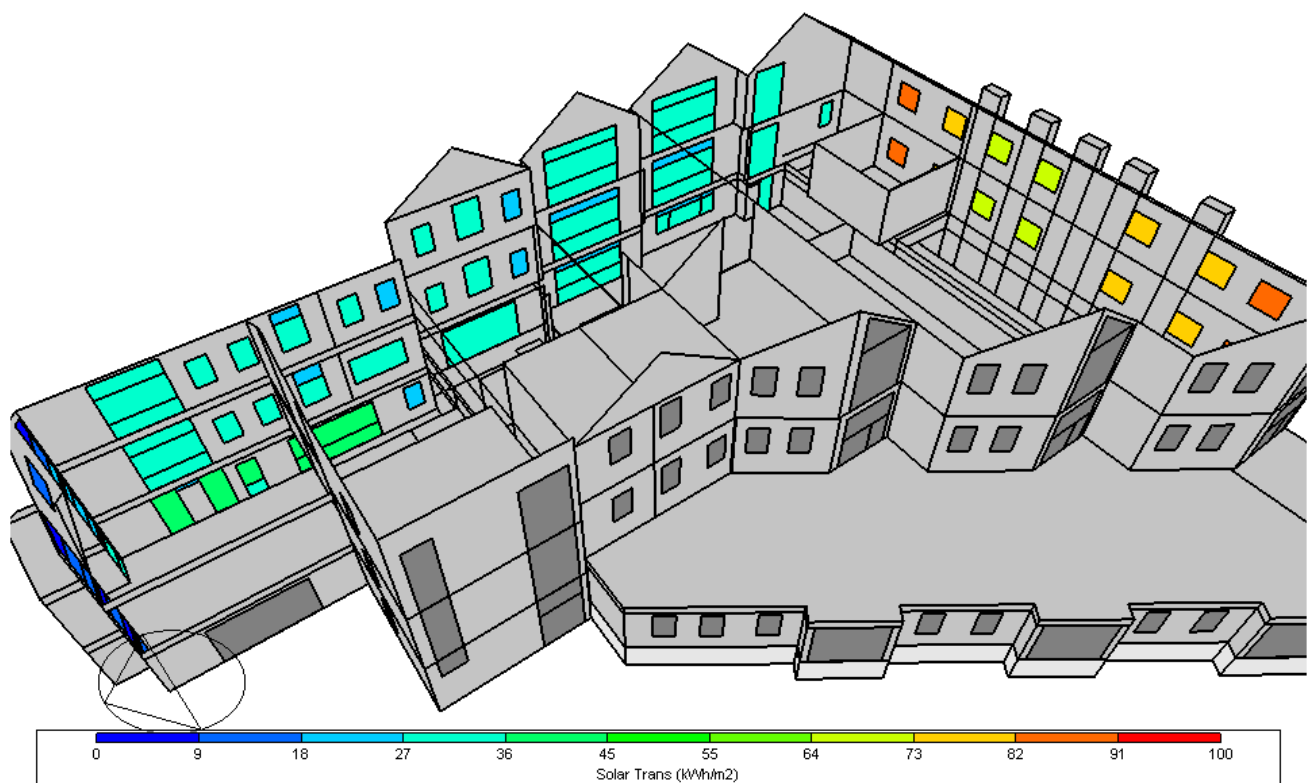
Surface removal

Specific to the Data Visualisation function is the option to remove surfaces to allow internal surface data to be viewed. This option is available when the Plot type is **2-Internal surfaces** or **3-Zone data**. To use the surface removal tool, simply click with the mouse on any external or internal surface that you would like to be removed. You can use this tool to selectively expose data deep within a building without losing other data that you might also wish to view.

To restore a surface that was removed hold the <Ctrl> key down and click with the mouse again on the space where the removed surface should be.

To restore all removed surfaces press the <Esc> key.

The example screenshot below shows how the distribution of building window solar gains can be viewed by removing surfaces.



Note: Removing surfaces in this way only affects the data visualisation view and does not affect the model itself.

Section cut

The [Section cut](#) tool can be used to cut a slice through the model to allow internal data to be viewed when the Plot type is **2-Internal surfaces** or **3-Zone data**. This tool can be especially useful for viewing data across a whole floor, or across a whole facade.

Movie generation

When the Interval is **2-Monthly** or **4-Hourly** you can generate a movie showing how the selected data changes with time.

Numeric outputs

To obtain a direct numeric output for a particular zone, surface or opening hold the <Shift> key down and click with the mouse on the surface of interest. The data for the current surface will be displayed in a tooltip until you release the mouse button.

Sub-Dividing Surfaces

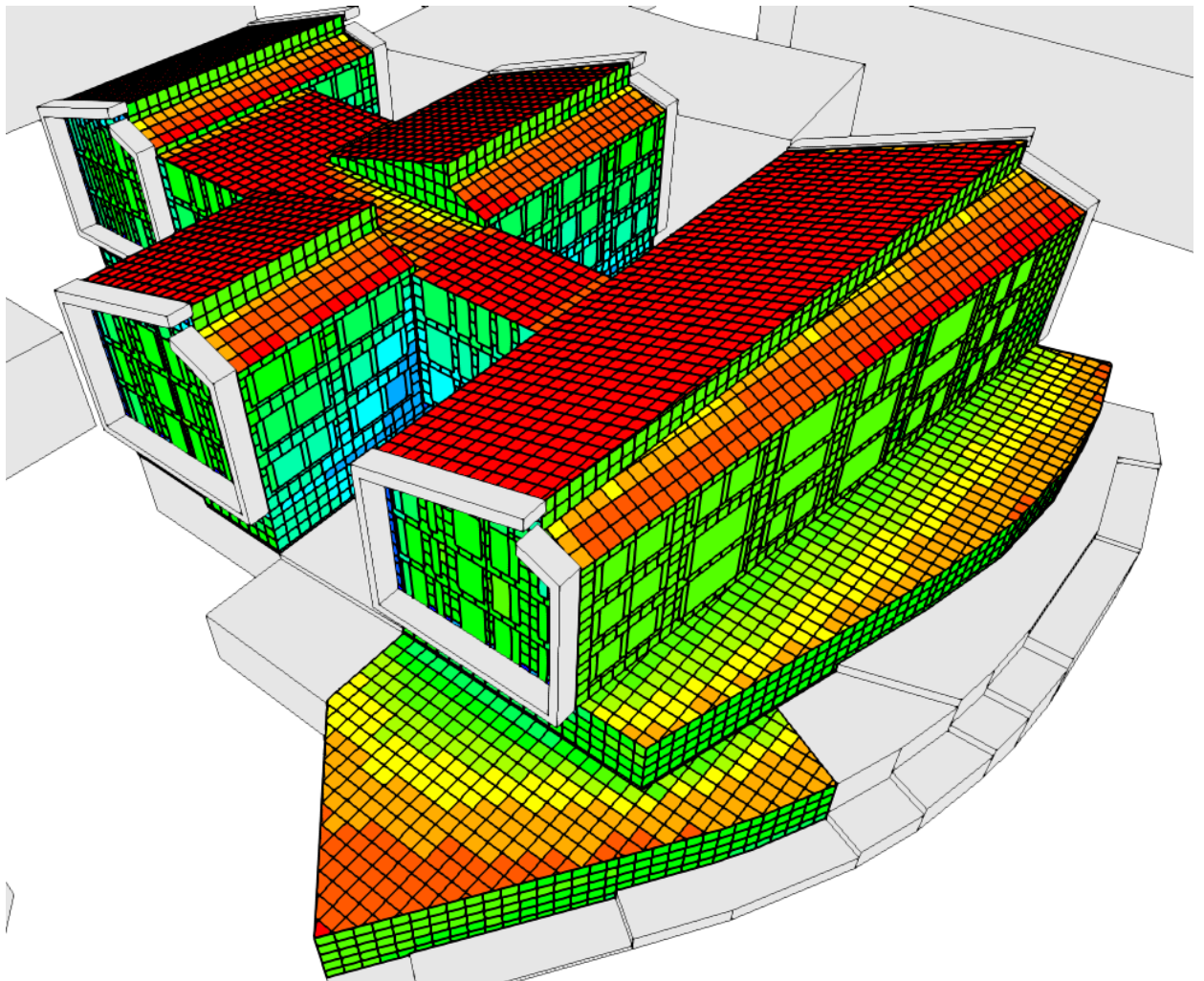
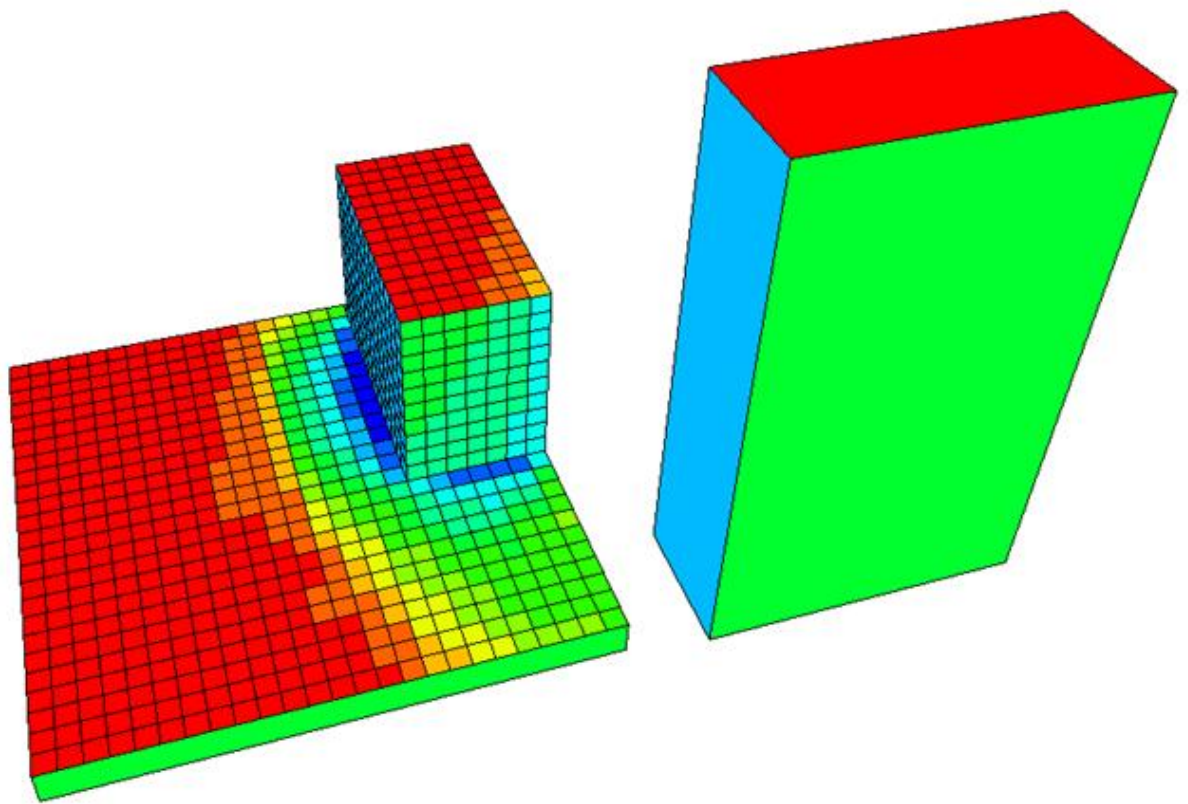
Sometimes it is useful to be able to sub-divide surfaces and display gridded results for [sub-surfaces](#) within each surface. This can be done by manually adding sub-surfaces to each surface that is to be sub-divided and generate and display results for sub-surfaces. The process is as follows:

1. Navigate to surface level for each surface to be sub-divided manually.
2. Navigate to building level and use the [clone](#) tool to copy any previously drawn grid of sub-surfaces to other surfaces in the model as required. This speeds up the process significantly
3. Ensure that you will generate the required results for sub-surfaces by checking the settings on the Miscellaneous tab.
4. Use the Simulation manager when running simulations to give the software maximum access to the machine's RAM.
5. Select appropriate simulation periods and data generation intervals bearing in mind that generating hourly or sub-hourly results for long simulation periods will generate a lot of data and could cause problems when attempting to load data back to the model for display. You should only generate the data you need. For example if you need to view hourly results over some design days in a large model, it may be best to run separate simulations for each design day only and load results back separately for each day of interest.

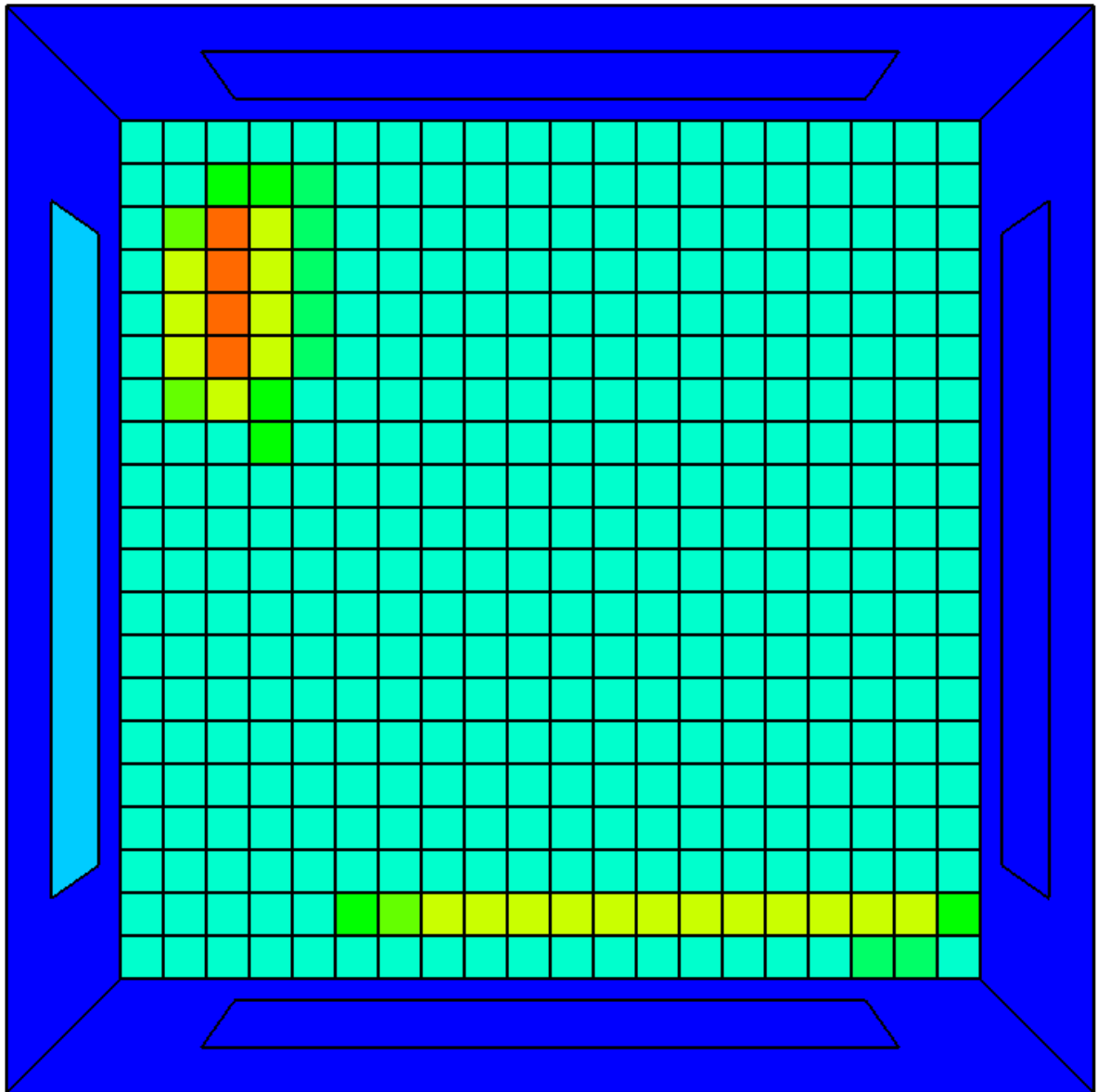
Tip: To avoid software memory issues, you should not run long simulations with hourly or sub-hourly surface or sub-surface results requested in large models!

Sub-dividing surfaces in this way can useful for 2 reasons:

1. In DesignBuilder complex and non-convex surfaces are sub-divided automatically behind the scenes into smaller polygons to meet EnergyPlus requirements, in which case multiple EnergyPlus polygons will be used to represent a single DesignBuilder surface. When reading simulation results, DesignBuilder uses the value for the last polygon it reads in to represent the data for the whole surface rather than calculating an average for all polygons. If you see surprising results for large complex surfaces, this is probably the cause and, if you need a more accurate representation, you may prefer to manually sub-divide the surface in question using sub-surfaces as described above.
2. You may need to view the distribution of conditions across larger surfaces for example to see how the solar incident radiation varies over a large flat roof to help decide on positioning PV panels. You can do this by placing a grid of sub-surfaces on any surfaces that need to be broken down and running the simulation as normal, making sure that surface data is stored for openings. In this case you can generate outputs such as that shown for the examples below.



You can also report the distribution of conditions on internal surfaces. For example the screenshot below shows the distribution of temperatures on the surface of a floor of an example model.



Note: The technique of using sub-surfaces to display the distribution of conditions across larger surfaces does not work for adiabatic surfaces because it is not possible to simulate openings on adiabatic surfaces.

Computational Fluid Dynamics (CFD)

Click on the **CFD** screen tab to start CFD calculations.

The rest of this section covers the processes involved in running CFD calculations in DesignBuilder.

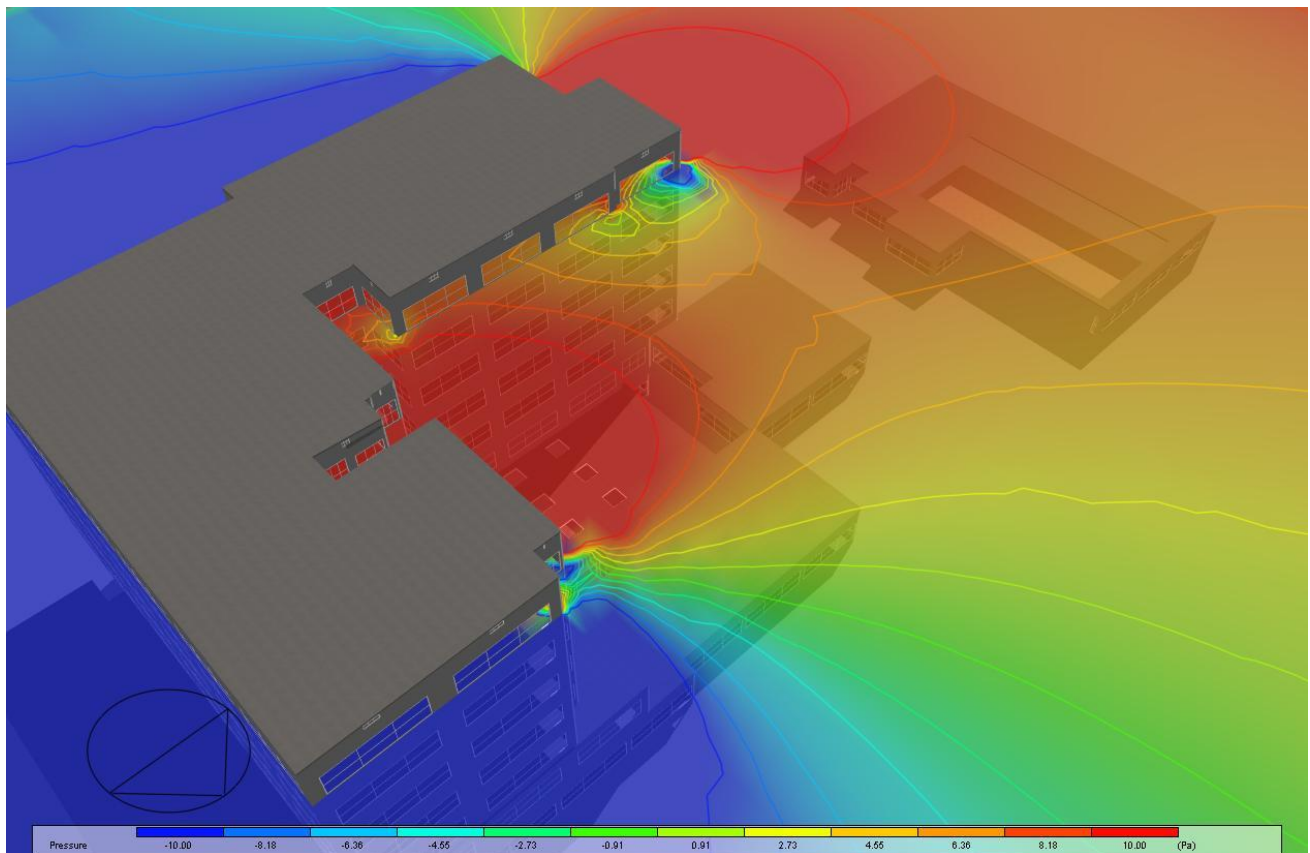
What is CFD?

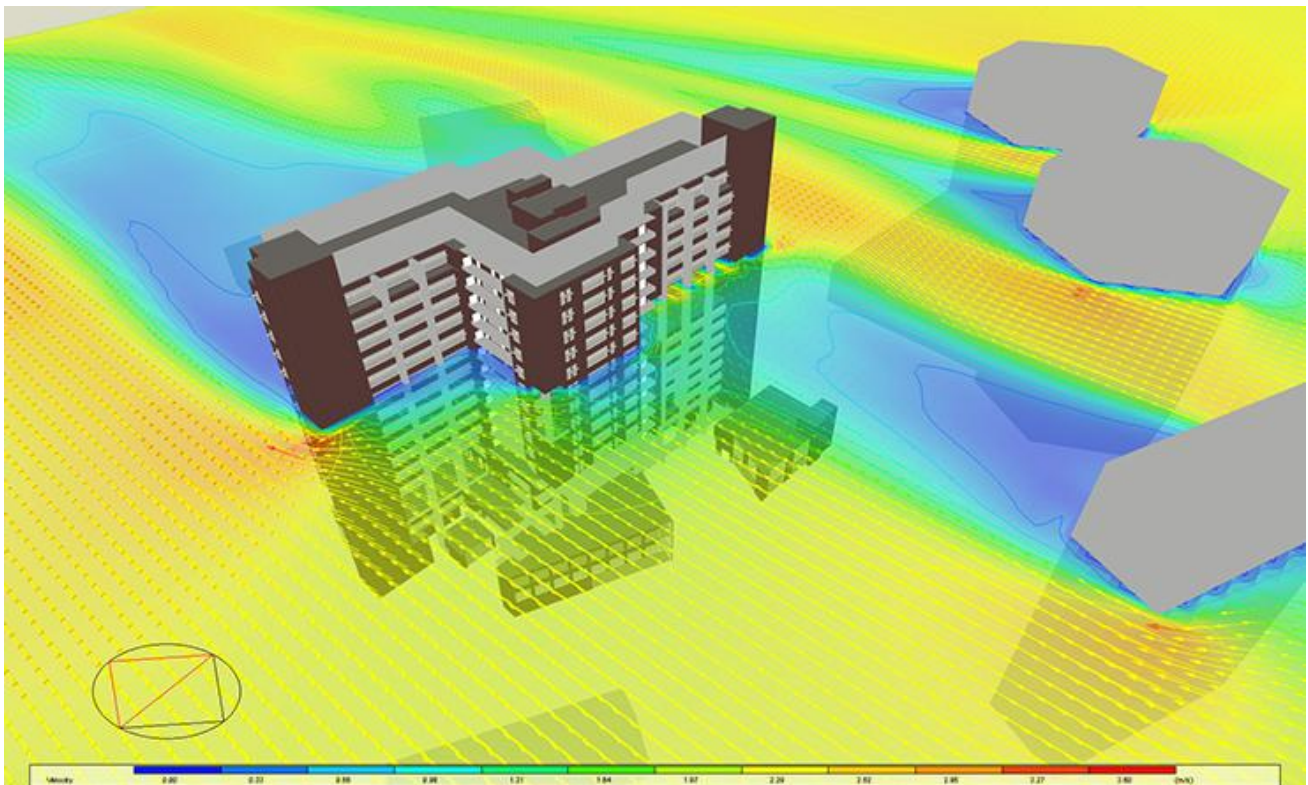
Computational Fluid Dynamics (CFD) is the term used to describe a family of numerical methods used to calculate the temperature, velocity and various other fluid properties throughout a region of space.

CFD when applied to buildings can provide the designer with information on probable air velocities, pressures and temperatures that will occur at any point through a predefined air volume in and around building spaces. Boundary conditions are specified which may include the effects of climate, internal heat gains and HVAC systems. DesignBuilder CFD can be used for both external and internal analyses.

External CFD analysis

External CFD analysis provides the distribution of air velocity and pressure around building structures due to the wind effect and this information can be used to assess pedestrian comfort, determine local pressures for positioning HVAC supply and extract grilles and to calculate more accurate pressure coefficients for EnergyPlus calculated natural ventilation simulations.

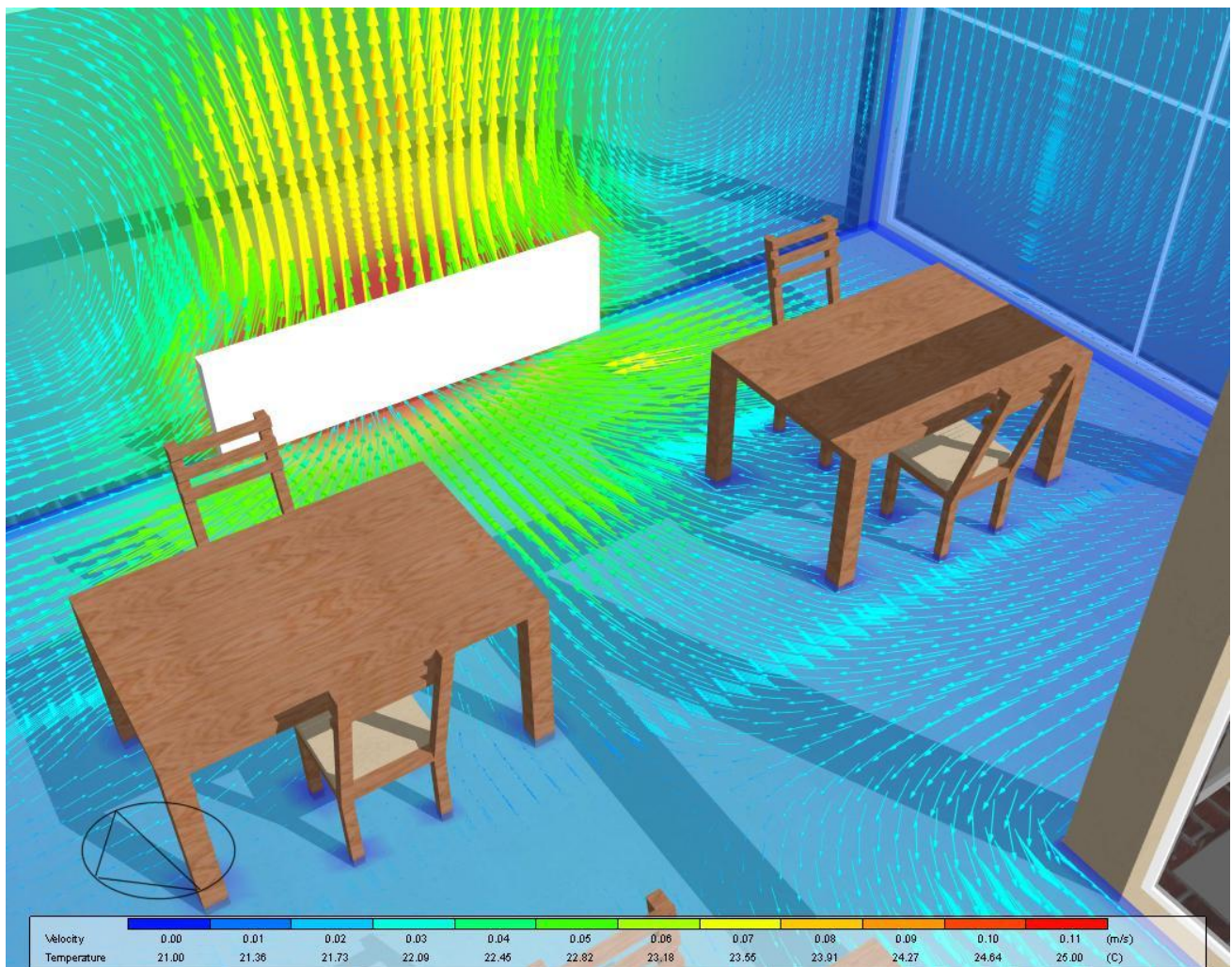




Example External CFD Analysis Output - Wind Pressure Study

Internal CFD analysis

Internal CFD analysis provides information on the distribution of air velocity, pressure and temperature (and several other calculated parameters) throughout the inside of building spaces. Also calculated is 'age of air' to indicate the relative 'freshness' of the air through the domain and also a comfort index. This information can be used to assess the effectiveness of various HVAC and natural ventilation system designs and to evaluate consequent interior comfort conditions.



Example Internal CFD Analysis Output - Occupant Comfort

CFD Calculations Topics Contents

- [Creating Model Geometry for CFD](#)
- [Finite Volume Grid and Geometric Modelling Considerations](#)
- [Building Levels and CFD Analyses](#)
- [CFD Boundary Conditions](#)
- [Boundary Conditions for Internal CFD Analyses](#)
- [Component Blocks and Assemblies as CFD Boundaries](#)
- [Setting Up a New External CFD Analysis](#)
- [Setting Up a New Internal CFD Analysis](#)
- [Editing the CFD Grid](#)
- [CFD Grid Information](#)
- [Setting Up CFD Cell Monitor Points](#)
- [CFD Calculation Options](#)
- [Residuals and Cell Monitor Graphs](#)
- [Convergence Difficulties and Troubleshooting](#)
- [Displaying Results](#)
- [CFD Comfort Calculations](#)
- "e Age of Air and Air Change Effectiveness" on page 1945
- [CFD Analysis Manager](#)
- "Example - CFD External Analysis" on page 1949
- "Example - CFD Internal Analysis Basics" on page 1955
- "Example - CFD Internal Analysis Including Assemblies" on page 1959
- [Appendix - CFD Calculations and Convergence](#)

Tip: You can find details on the calculation method used in the [CFD Technical](#) online document.

Creating Model Geometry For CFD

If you have already created a model for the purpose of an additional analysis (e.g. thermal simulation or SBEM calculation), you can use exactly the same model for CFD, although you should first read the [Finite volume grid and geometric considerations](#) section.

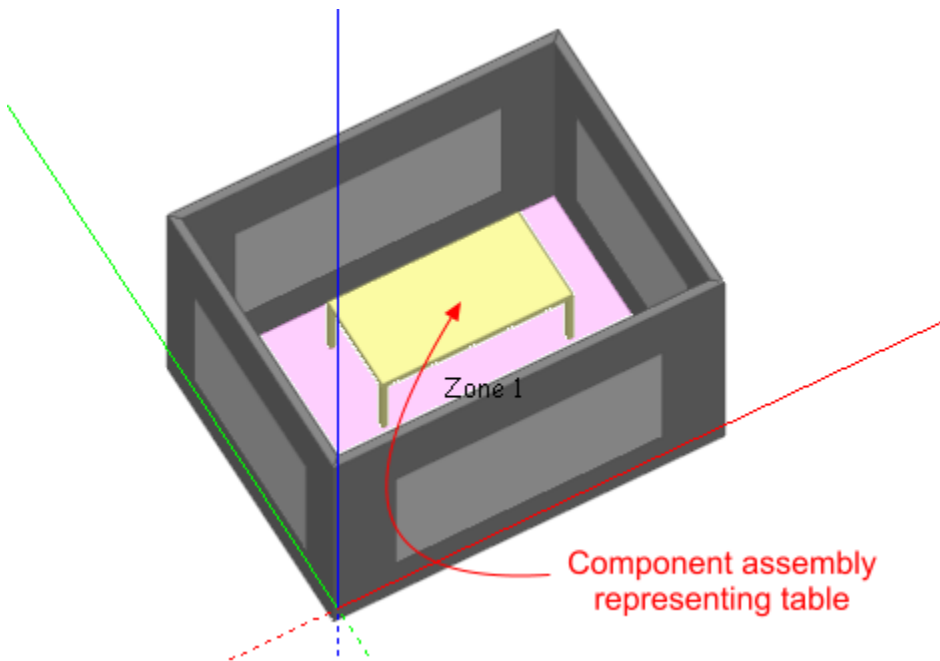
On the other hand, if you intend to conduct a CFD analysis from scratch, you should refer to the [Building Models](#) section for information on creating models.

Finite Volume Grid And Geometric Modelling Considerations

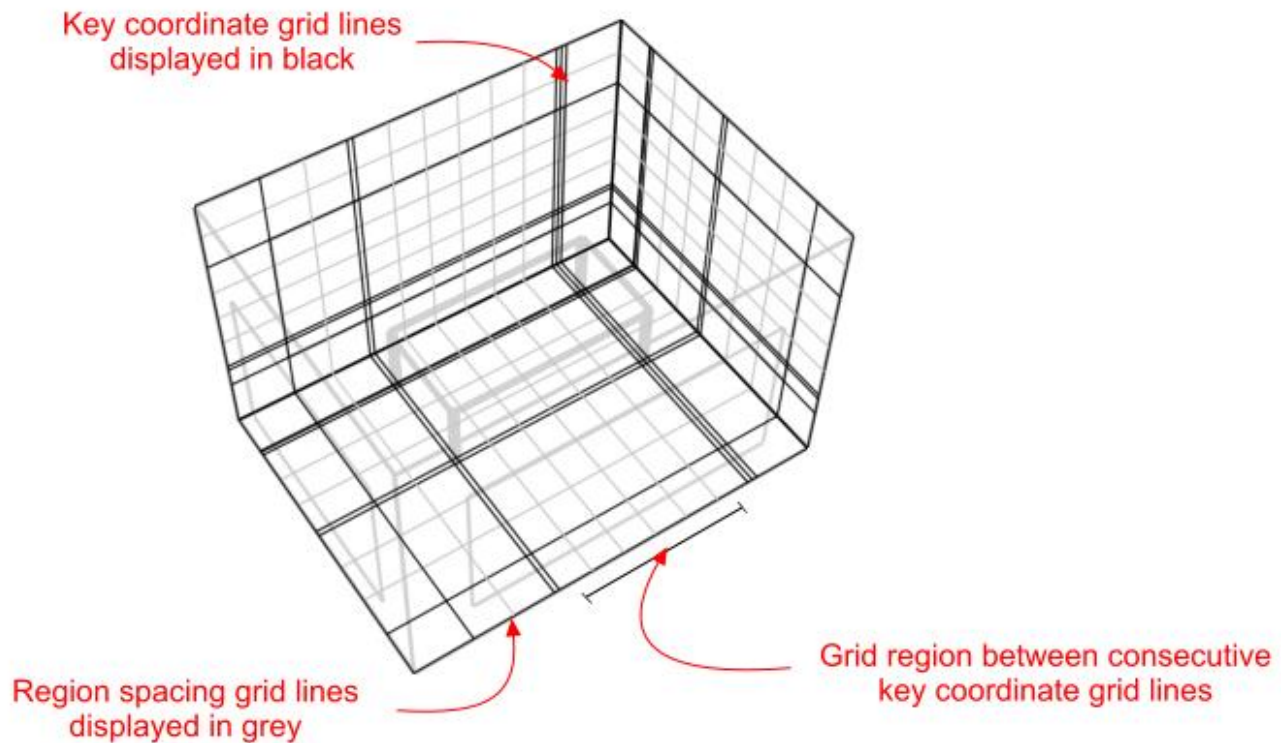
The calculation method requires that the geometric space across which the calculations are to be conducted is first divided into a number of non-overlapping adjoining cells which are collectively known as the finite volume grid.

When a CFD project is created, a grid is automatically generated for the required model domain by identifying all contained model object vertices and then generating key coordinates from these vertices along the major grid axes. These key coordinates, extended from the X, Y and Z-axes across the width, depth and height of the domain respectively are known as 'grid lines'. The distance between grid lines along each axis is known as a grid 'region' and these regions are initially spaced employing user-defined default grid spacing in order to complete the grid generation. The grid used by DesignBuilder CFD is a non-uniform rectilinear Cartesian grid, which means that the grid lines are parallel with the major axes and the spacing between the grid lines enables non-uniformity.

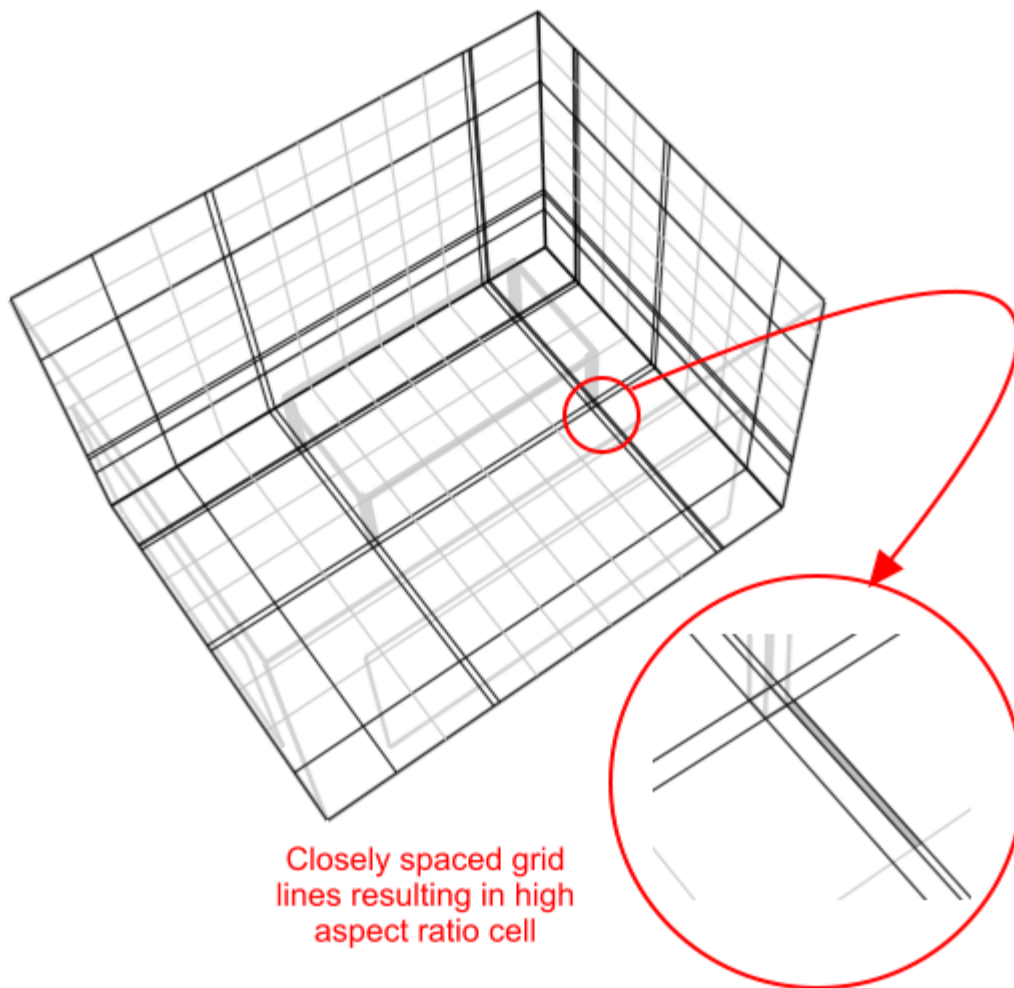
For example, looking at a simple building block with a single component assembly representing a table:



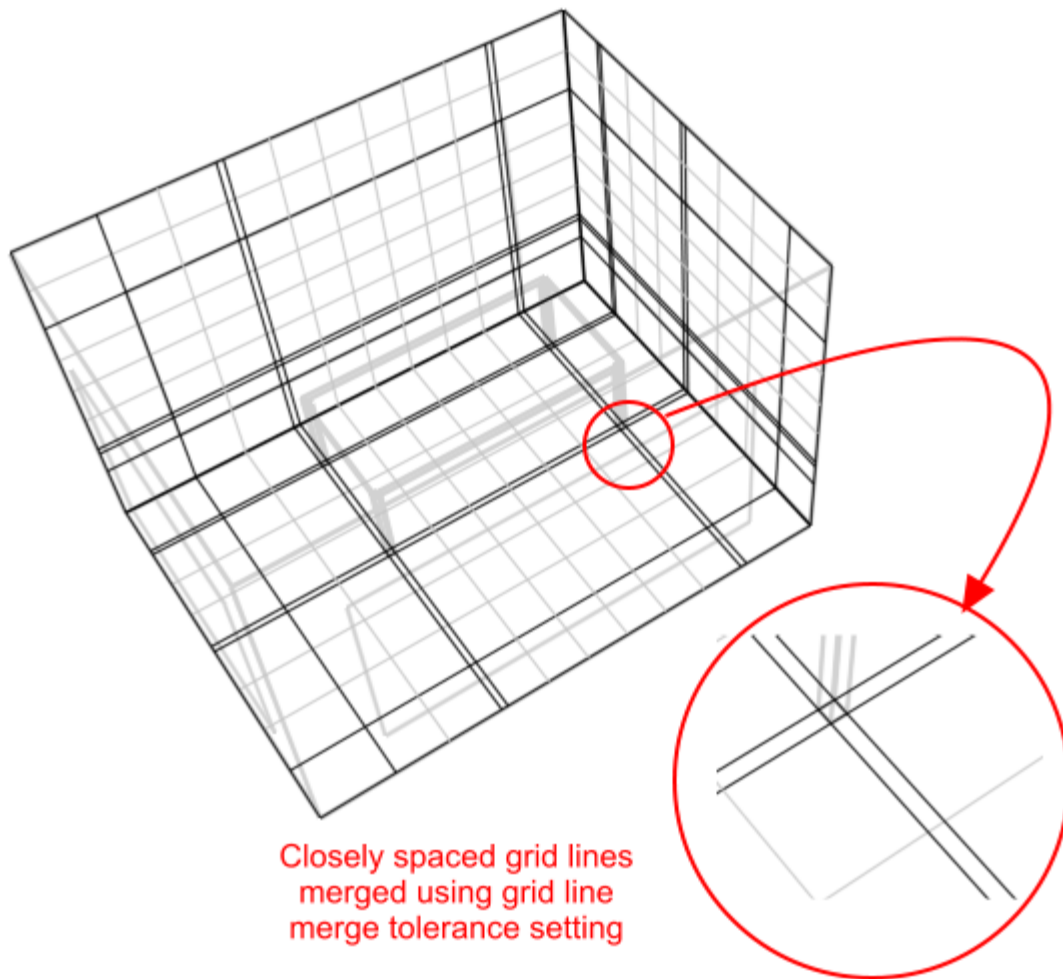
The resulting grid, generated with 0.3m default grid spacing would be as follows:



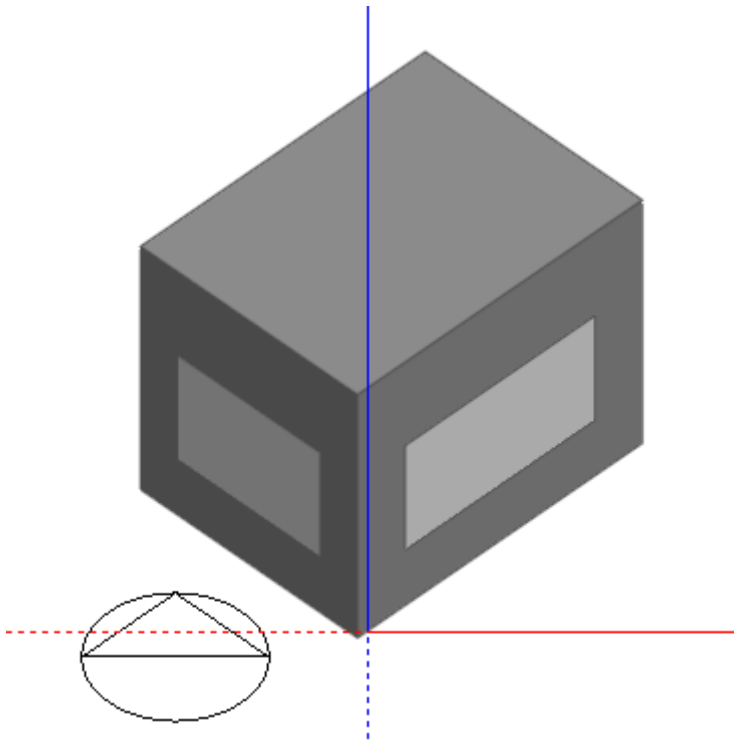
By default, grid regions are spaced uniformly using a spacing that is calculated to be as close to the user-defined default grid spacing as possible. Notice the narrow regions created between key coordinates associated with the tabletop and table legs. In this case, the distance between these key coordinates is of an acceptable value. However, very narrow regions resulting in long narrow grid cells or cells having a high aspect ratio should be avoided, as they tend to result in unstable solutions that can fail to converge. Highly detailed component assemblies can result in very large numbers of closely spaced key coordinates resulting in cells having high aspect ratios. Large numbers of key coordinates can also lead to overly complex grids and correspondingly high calculation run times and excessive memory usage which can be avoided by replacing very detailed assemblies with cruder representations for the purpose of the CFD calculation. However, where very narrow grid regions are unavoidable, adjacent grid lines formed from key coordinates can be merged together using the merge tolerance setting which is accessed through the new CFD analysis dialogs (see the [Setting Up a New External CFD Analysis](#) and 'Setting Up a New Internal CFD Analysis' sections). For instance, in the above example, if the table assembly had been located closer to the edge of the adjacent window, this could result in unacceptably close grid lines:



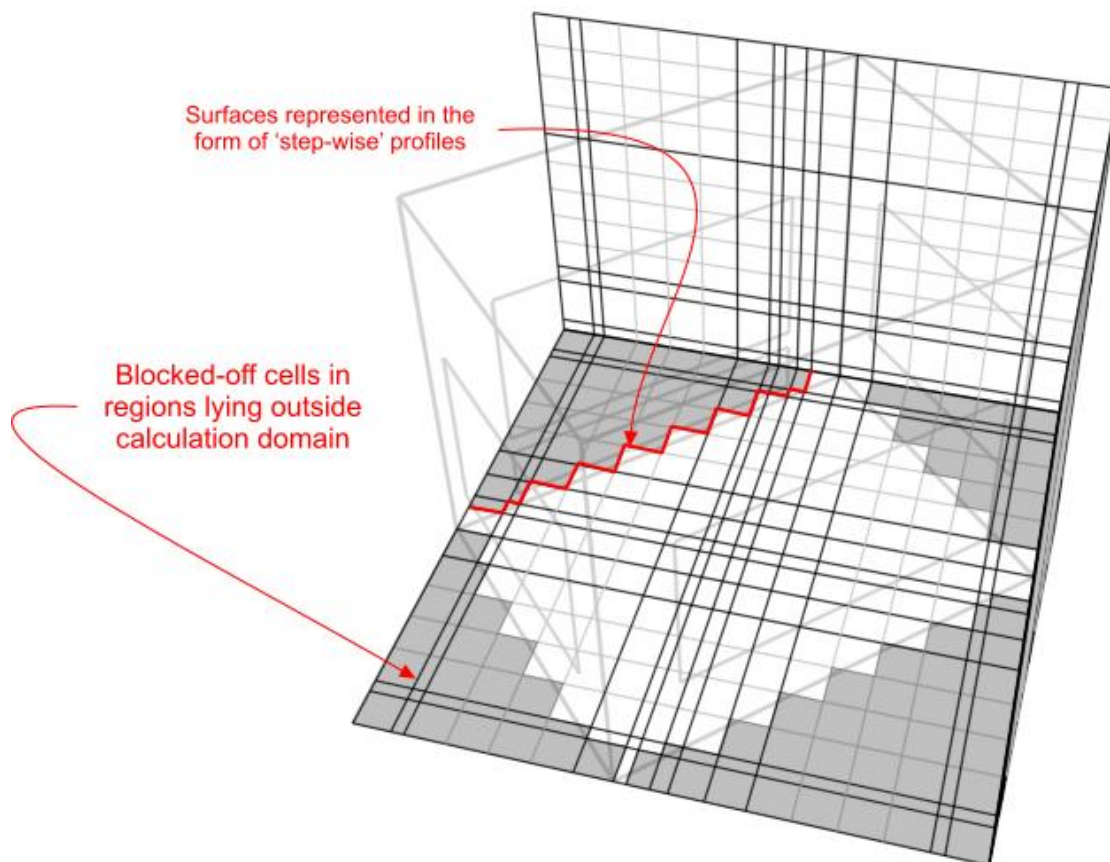
These closely spaced grid lines can be merged by creating a new analysis and increasing the grid line merge tolerance setting to say 0.025m:



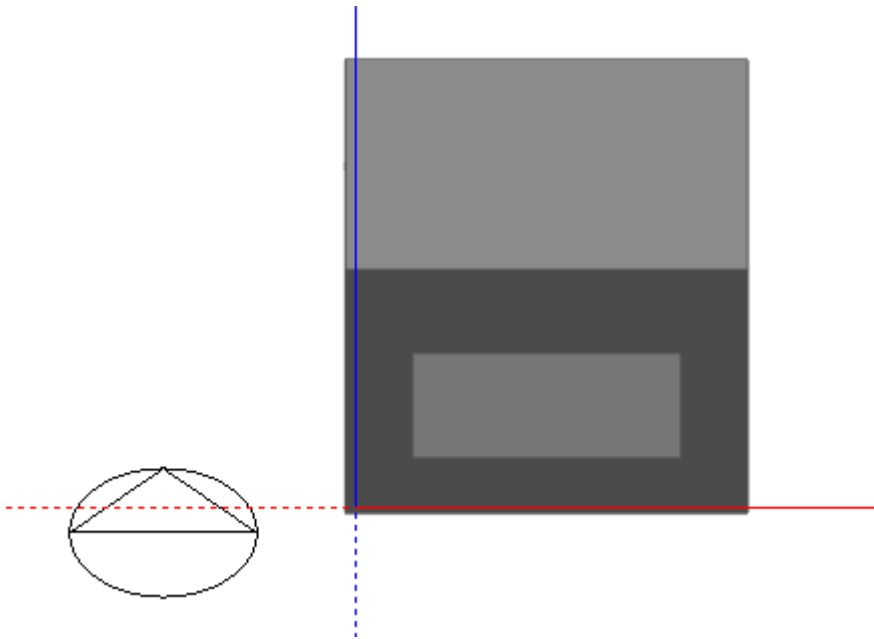
Due to the strict rectilinear nature of the grid, grid cells that lie in regions outside of the domain required for calculation are 'blocked-off' in order to cater for irregular geometries. It is important to take this into account when creating a model in order to maximise the efficiency of grid generation and/or to ensure that surface CFD boundaries will lie in the plane of a major grid axis to achieve accuracy of boundary representation. In some cases, the model may be rotated in order to ensure that most of the wall surfaces are orthogonal with respect to the grid axes. To take an extreme example, if a simple rectangular space has been drawn at a 45° angle to the Z-axis:



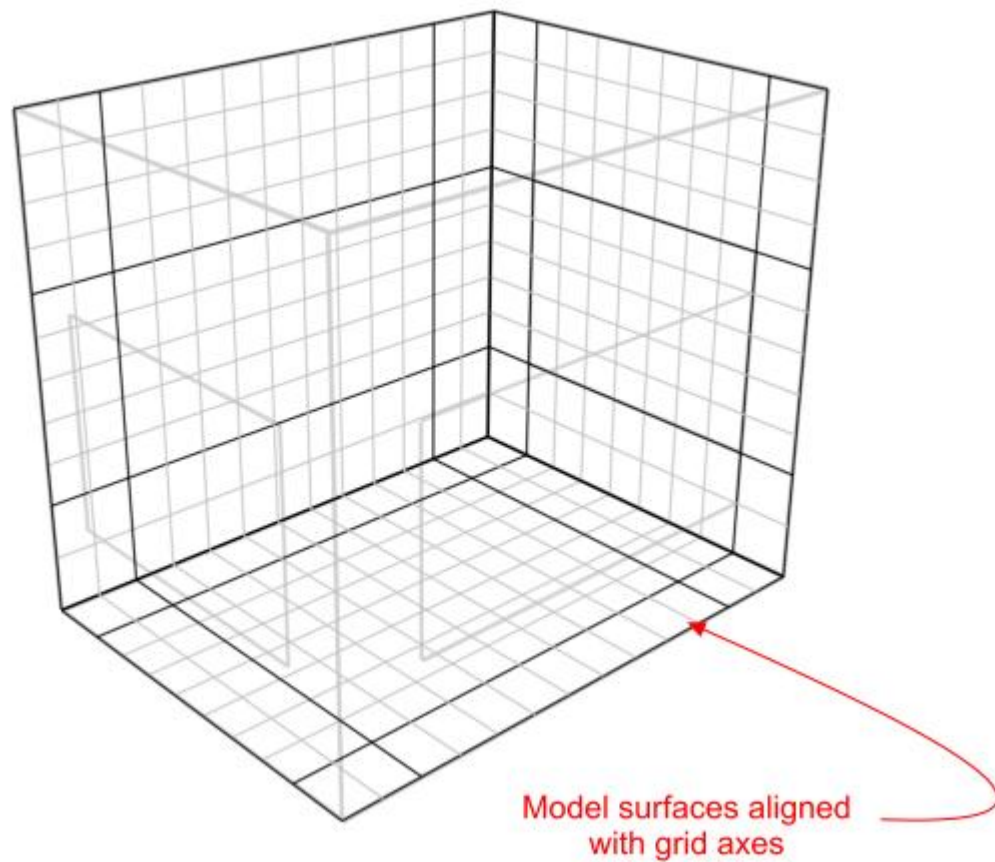
The resulting grid, generated using 0.3m default grid spacing, contains a great deal of redundancy in the form of blocked-off regions and inaccuracy of surface representation:



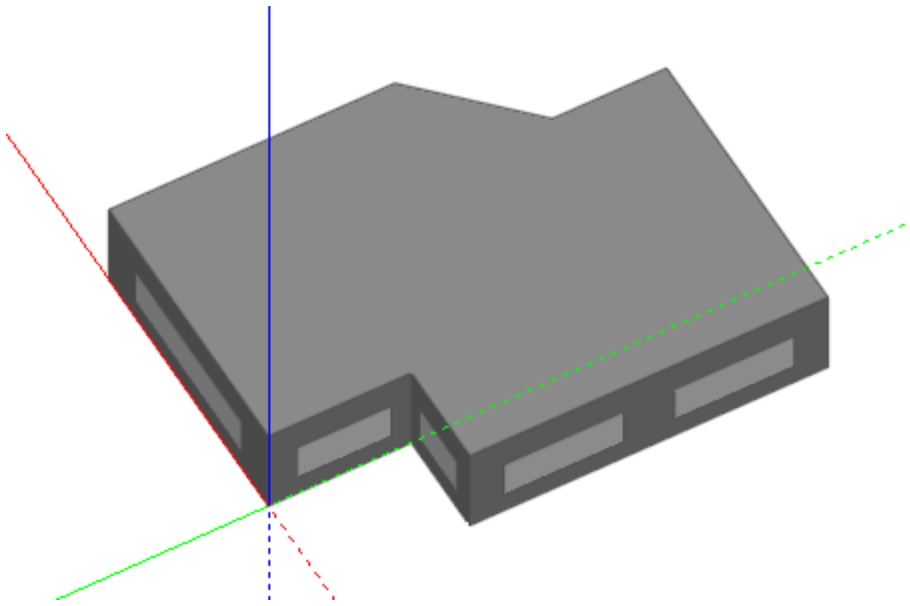
If the block is first rotated, to be aligned with the X-axis:



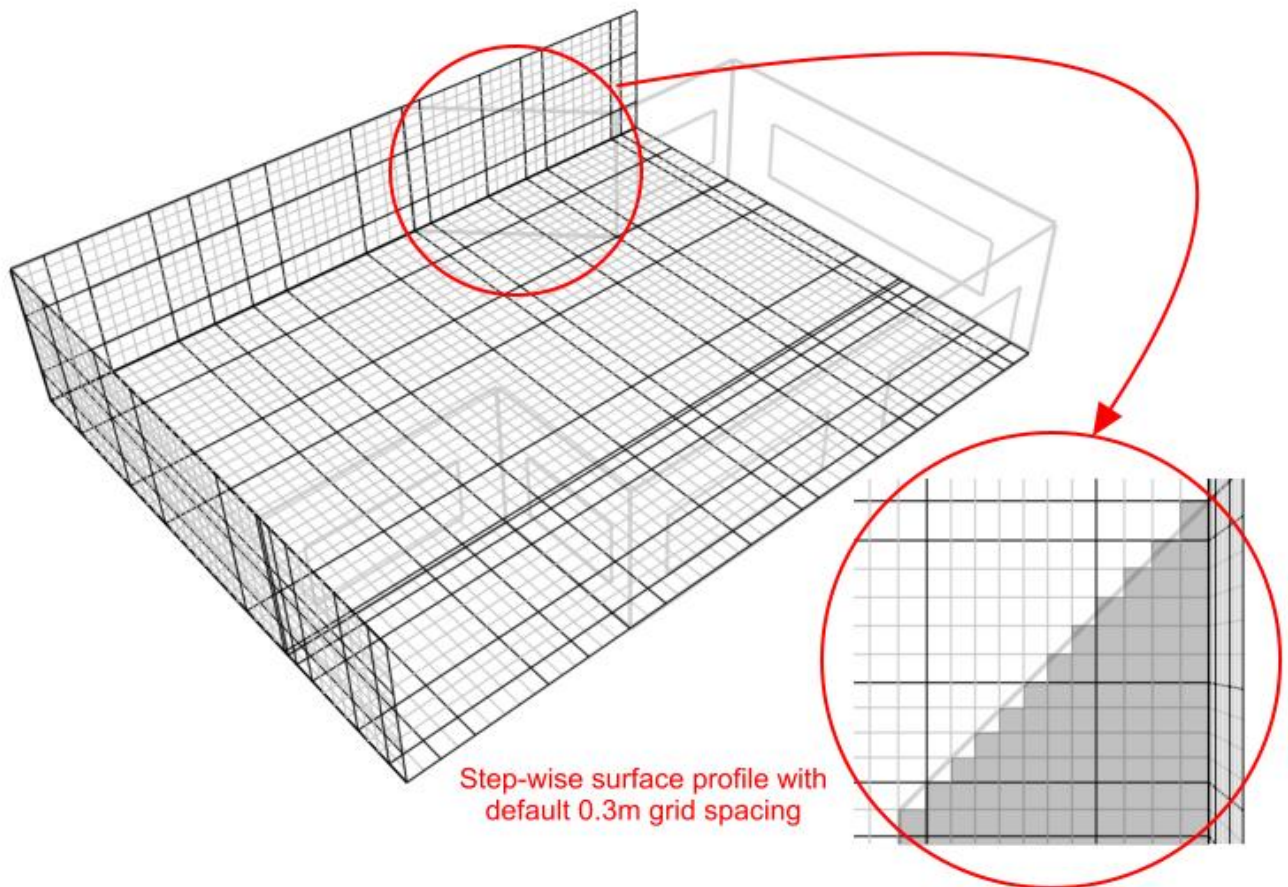
The resulting grid exhibits no redundancy and there is no inaccuracy in surface representation:



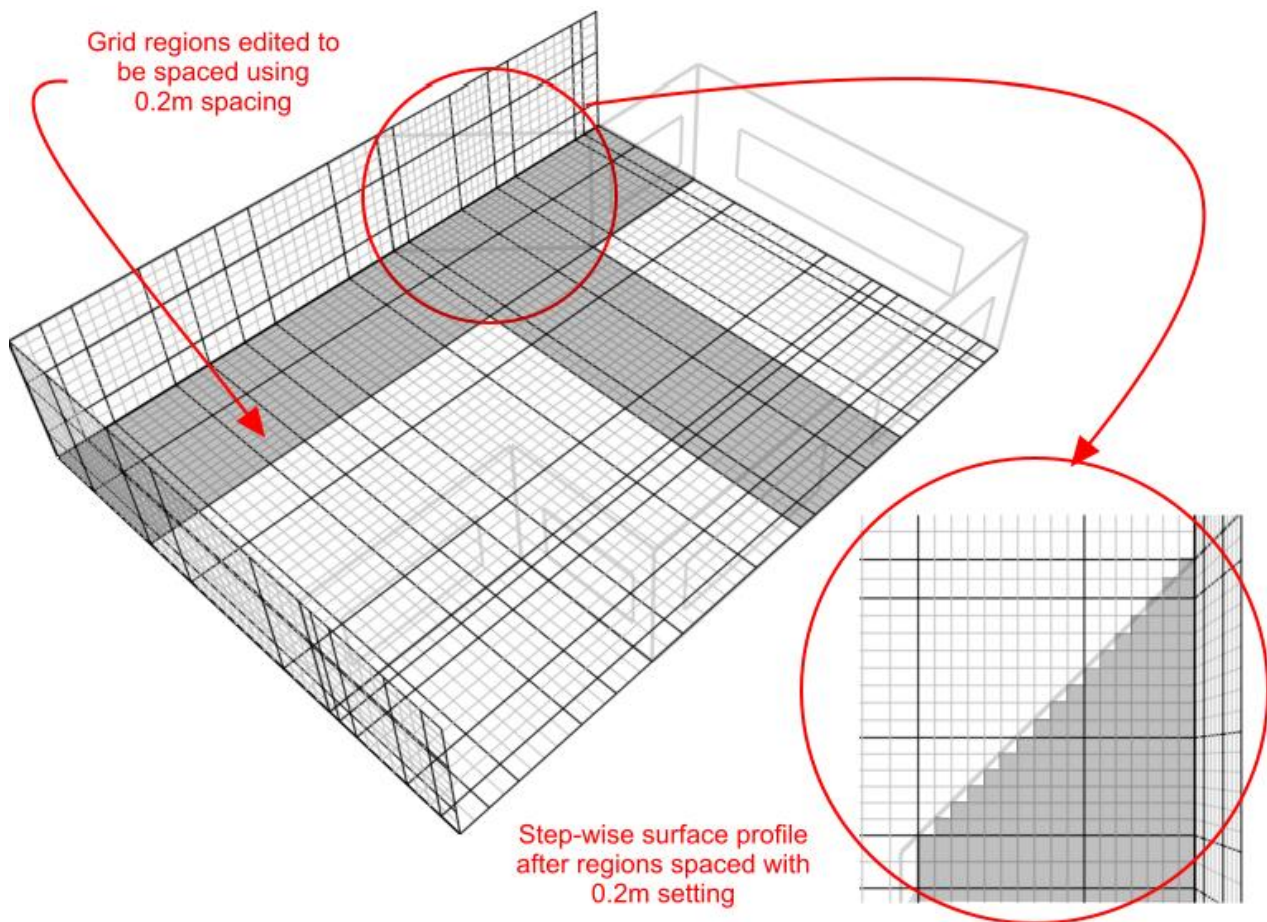
The accuracy of the representation of non-orthogonal surfaces can be improved by using smaller default grid spacing and in some cases specific grid regions can be modified to increase accuracy in a more localised fashion as illustrated by the following example:



Grid generated using default 0.3m grid spacing:



Using the 'Edit CFD Grid' tool, the grid spacing within just the regions spanning the angled section is reduced to 0.2m to improve the representation:



Grid regions between key coordinates can also be spaced using non-uniform spacing options and additional key coordinate regions can be added. Further information is provided on grid modification in the 'Editing the CFD Grid' section.

Building Levels And CFD Analyses

Internal CFD analyses can be conducted at zone, building block and building levels. Calculations can also be conducted for single zones that span several blocks by connecting them with holes and using the [Merge zones connected by holes](#) model option setting.

External analyses can only be conducted at the site level.

The level you are at when you create the CFD Analysis defines both the type of the CFD analysis (internal or external) and also the extent of the CFD domain. If you are at building level then the domain is the whole building, if you are at block level the domain will be the block and if you are at zone level the domain is the current zone.

CFD Boundary Conditions

An important initial concept for CFD analyses is that of boundary conditions. Each of the dependent variable equations requires meaningful values at the boundary of the calculation domain in order for the calculations to generate meaningful values throughout the domain. These values are known as boundary conditions and can be specified in a number of ways.

The specification of boundary conditions for external analyses is relatively straightforward because it is an isothermal calculation and so it only requires the building exposure, wind velocity and wind direction to be set and this is done in the **New CFD Analysis** dialog box.

Internal analysis boundary conditions are more involved and can require the addition of zone surface boundaries such as supply diffusers, extract grilles, temperature and heat flux patches and also the incorporation of model assemblies representing occupants, radiators, fan-coil units, etc. DesignBuilder provides default wall and window boundary temperatures automatically but it is important that you check that these defaults are appropriate and that you correctly specify any additional boundary conditions required for your project.

Tip: The internal surface temperature for each surface in the domain can be calculated using EnergyPlus simulation by switching on the appropriate options. Airflow rates through open windows, vents, doors and holes can be calculated using the [Calculated natural ventilation](#) option. You can find a webinar recording demonstrating the process involved at: <https://www.youtube.com/user/DesignBuilderUK>.

See also:

- [Boundary Conditions for Internal CFD Analyses](#)
- [CFD Model Data](#)
- [Component Blocks Assemblies as CFD Boundaries](#)

Boundary Conditions For Internal CFD Analyses

Internal boundary conditions can be defined in three ways:

1. Surface temperatures can be defined for all zone surfaces including windows and other openings.
2. Zone surface boundary conditions including supply diffusers, extract grilles, temperatures and heat fluxes can be added in the form of surface patches using a similar method to that used for adding windows and doors.
3. Component blocks and component assemblies can be defined as temperature or heat flux boundaries or they can also have boundary condition patches added as described above for Zone surfaces.

In all cases the CFD boundary conditions can be edited on the [CFD](#) Model data tab.

Note: The easiest way to accurately include the effect of **radiation heat transfer** in CFD calculations is to run an EnergyPlus simulation first and then import surface temperatures from the EnergyPlus simulation results as CFD boundary conditions using the [CFD boundary conditions dialog](#). The EnergyPlus surface temperatures include the effect of radiation heat transfer, however in order to arrive at the same heat balance in the CFD calculation, it is necessary to include the convective portion of any specified heat gain used in the EnergyPlus simulation as specified boundary conditions in the CFD model (using assembly flux, heat flux surface patches, etc).

Default Wall And Window Temperatures

All zone surfaces within a model have default wall and window boundary temperatures. These boundary temperatures are inherited from the building level down to the zone surface level and may be set at any intermediate level. This mechanism provides a convenient method for defining wall and window temperature boundary conditions throughout a building model in the absence of [imported EnergyPlus boundary data](#).

Default inside and outside wall and window boundary temperatures can be accessed under the **CFD Boundary** header on the [CFD](#) tab.

Note: You can include multiple zones in a single domain, but in this case it is important to ensure that all internal partitions and floors/ceilings separating zones in the domain have finite (non-zero) thickness. Any internal surfaces in the domain with zero thickness will be ignored and their boundary conditions will not take effect.

Zone And Component Block Surface Boundaries

Supply diffusers, extract grilles, temperature and heat flux patches can be added to zone and component block surfaces using the [Add CFD boundary](#) tool.

To add a surface boundary, move down to the surface on which you want to locate the boundary, either by selecting it from the model navigator or by navigating to it in the Edit screen. At the surface level, you can then select the [Add CFD boundary](#) tool from the toolbar.

Note: If you do not set a boundary condition for an internal block-level component block, when it comes to calculating the MRT distributions DesignBuilder assumes that the component block surfaces are at the mean room temperature.

Component Blocks & Assemblies As CFD Boundaries

Component blocks and assemblies can be used for CFD analyses to simply allow for the effect of obstructions on the airflow. However, they also allow the definition of CFD boundary attributes to enable them to act as constant temperature surfaces or heat flux sources and can also be modified to behave as non-solids to allow air to pass through them.

Note: Component blocks and assemblies defined at block level do not affect EnergyPlus simulations.

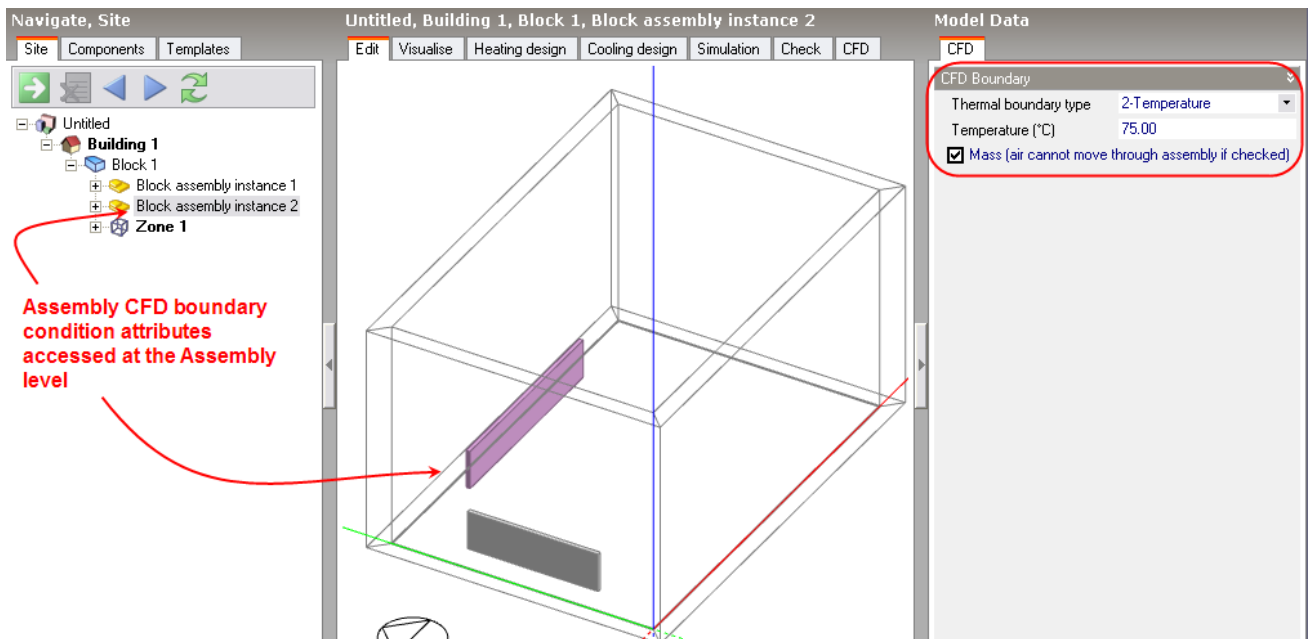
Component Blocks

CFD attributes for component blocks are inherited down from the building level to component blocks located at both building and building block levels. CFD attributes for component blocks are accessed on the [CFD](#) model data tab.

Assemblies

The CFD assembly library that is provided with DesignBuilder contains a number of pre-defined assemblies that can be used to add items such as occupants, radiators and furniture. Some of these pre-defined assemblies already have CFD boundary attributes associated with them, e.g. standing occupant assemblies have a defined convective heat flux of 56W. You can also define your own assemblies for use in CFD analyses and you will find details of how to do this in the [Assembly library](#) section.

CFD attributes for component assemblies are inherited from the parent assembly to instances of the assembly at the building and building block levels. CFD attributes for assemblies are accessed from the [CFD](#) tab after moving down to an assembly instance level by clicking on it in the navigator or double-clicking on it in the Edit screen. It is important to understand that if you change an attribute for a specific instance of an assembly, the attributes will be changed for all other instances of the same assembly.



The following CFD boundary settings can be set on the [CFD model data tab](#) for both component blocks and component assemblies:

Thermal boundary type: The thermal boundary type can be set to one of the following:

- **1-None** - component block does not act as a CFD boundary condition.
- **2-Temperature** - component block acts as a fixed temperature CFD boundary condition (enter temperature below).
- **3-Flux** - component block supplies a fixed heat flux to the surroundings (enter the convective flux below).

Temperature

If the thermal boundary type has been set to **2-Temperature**, enter the temperature of the block.

Heat flux (convective fraction only)

If the thermal boundary type has been set to **3-Flux**, enter the **convective only** portion of the heat emission of the block. If you know the total heat flux then you can calculate the convective part by multiplying by (1 - radiant fraction). For modelling building occupants you can assume a radiant fraction of 0.5. Sources of radiant fraction data for other equipment can be found at:

- "Office equipment gains" on page 348
- "Miscellaneous gains" on page 349
- "Catering gains" on page 350
- "Process gains" on page 352
- [General lighting](#)

Note: In cases where one or more faces of a component block are not in contact with the zone air (e.g. when the component block surface is adjacent to the floor or another component block) the total convective gain specified for the block is applied only to the surfaces in contact with the air.

Important Note: In the case of assemblies used to apply a heat flux boundary condition, the heat flux data on the CFD tab of the component blocks can be treated in one of 2 ways depending on whether or not it inherits this data from the assembly above:

a) When inheriting from the assembly above, the same heat flux data as for the parent assembly appears in blue on the CFD tab, but DesignBuilder doesn't use this directly. Instead the data is applied as if it is the total for the whole assembly and the actual value for this component block is reduced by a proportion

equal to the ratio of the component volume to the volume of the whole assembly.

b) When hard set heat flux data is entered (data shown in red on the CFD tab), this is used directly in CFD calculations and spread evenly around the component block surfaces.

Mass (air cannot move through block if checked)

Determines whether or not the block will act as a physical obstruction to the surrounding flow. In some cases, it may be desirable to fix the temperature or flux of the component but allow air to pass through the block, e.g. to represent an occupancy flux throughout a large volume without needing to locate individual occupants.

Advanced

Solid cell overlap tolerance

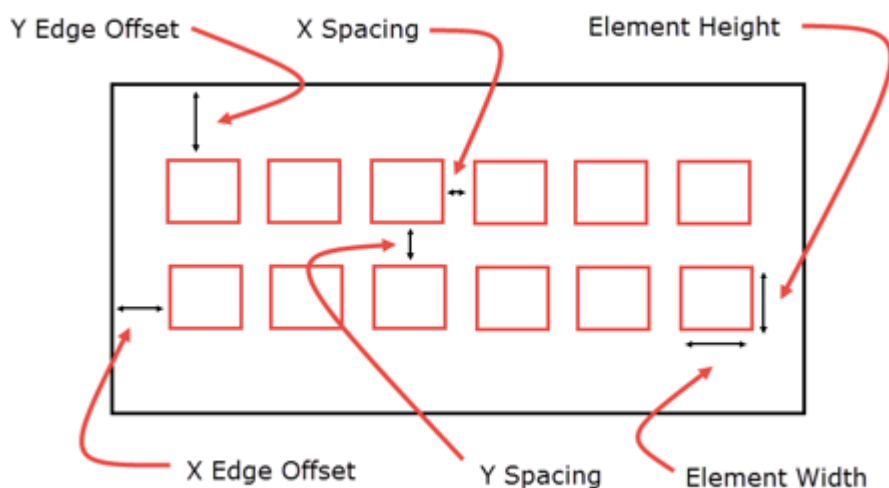
The **Solid cell overlap tolerance** is an advanced setting that allows some control over the identification of CFD grid cells that are to be interpreted as lying within a component and to be flagged as solid in the CFD simulation (assuming the mass flag is set for the component/assembly).

The overlap tolerance is a number between 0.0-1.0 representing a fractional overlap. The algorithm used to identify solid component cells marches up through the grid in the Z-dimension, creating a slice through each component at the height of each cell centre. An intersection check is then made for the resulting polygon list with each cell rectangle in plan. If the ratio of the intersection area to the cell area is greater than the tolerance, the cell is flagged as solid.

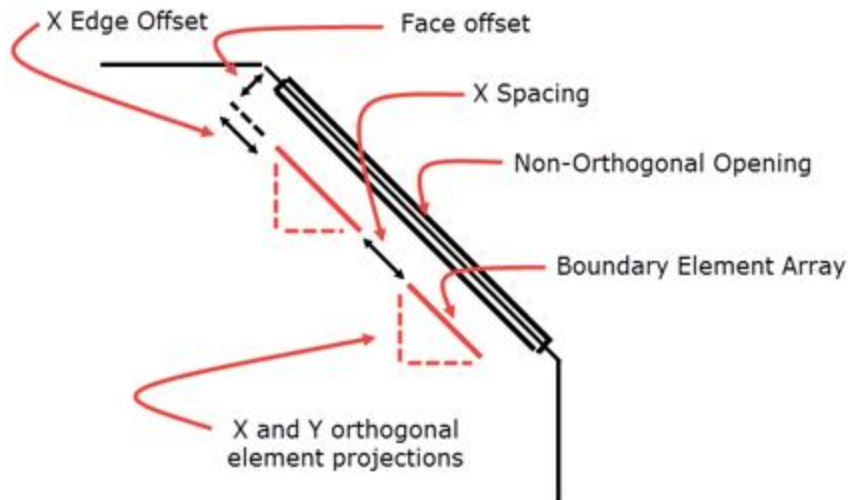
Non-Orthogonal Boundary Settings

CFD ventilation boundary conditions that are not orthogonal to the grid (i.e. not aligned with the grid) require special treatment to minimise problems caused by insufficient cell numbers in the vicinity of default boundary elements which can result in the flow stalling (mass flat-lining). The ventilation boundary mechanism for non-orthogonal ventilation boundaries involves automatically creating an array of rectangular elements across the face of the boundary. The resulting array of elements are then offset from the boundary face and projected onto the orthogonal X, Y, Z planes to form the orthogonal elements.

The diagrams below summarise the meaning of the various settings.



Normal View



Plan View

The following settings can be used at Building, Block and Zone levels as well as for non-orthogonal surfaces and their ventilation boundary conditions to allow hierarchical default settings to be defined.

Element width

The default value of 1.0m means that any opening having a dimension of 1.0m or less will have one projected element (max 0.9m distance from the opening with a 0.2m offset). This usually provides reasonable results without instability.

Element height

See above for Element width.

Face offset

The face offset is used to shift the ventilation boundary elements away from the parent surface into the body of the space by the defined offset distance. The reason for this setting is that in some cases, depending on the grid resolution, the algorithm that identifies solid cells can lead to a conflict between some solid cells and the boundary elements which can result in flow stalling (this was particularly a problem at the edges of full width windows which may not actually be such a problem now that we have an edge offset setting).

X-Spacing

X-spacing is the gap between the cells in the x-direction.

Y-Spacing

Y-spacing is the gap between the cells in the y-direction.

Z-Spacing

Z-spacing is the gap between the cells in the z-direction.

X-Edge offset

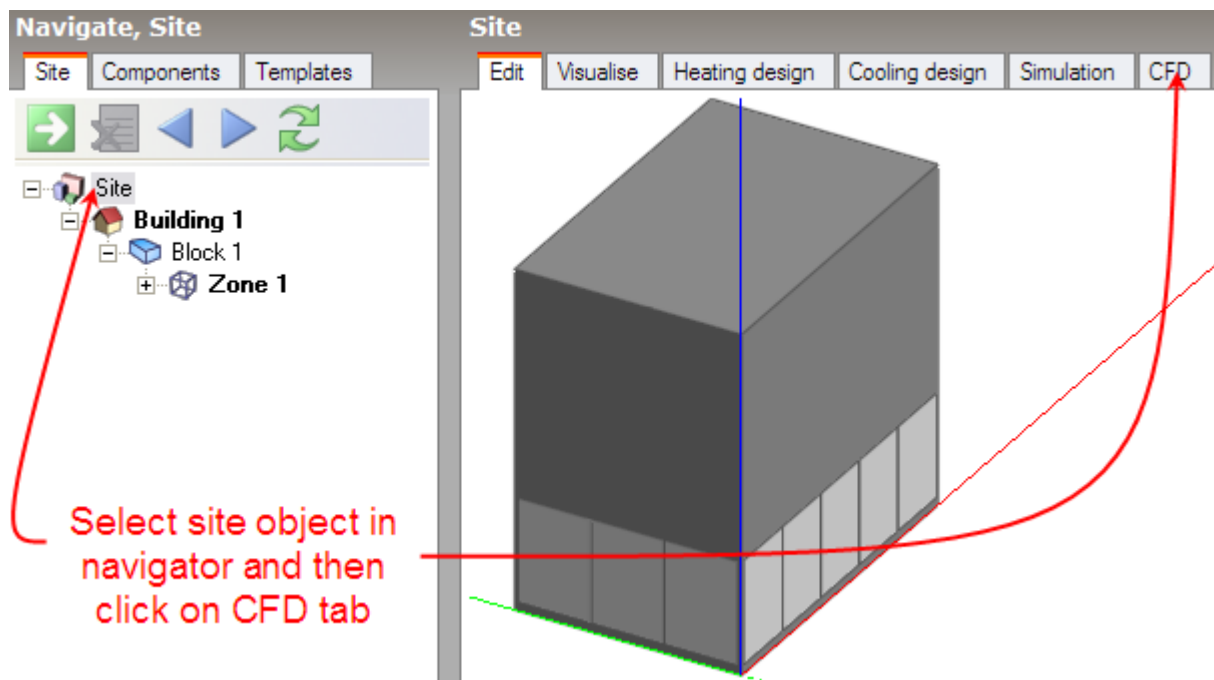
The offset in the x-direction between the edge of the boundary condition and the first cell.

Y-Edge offset

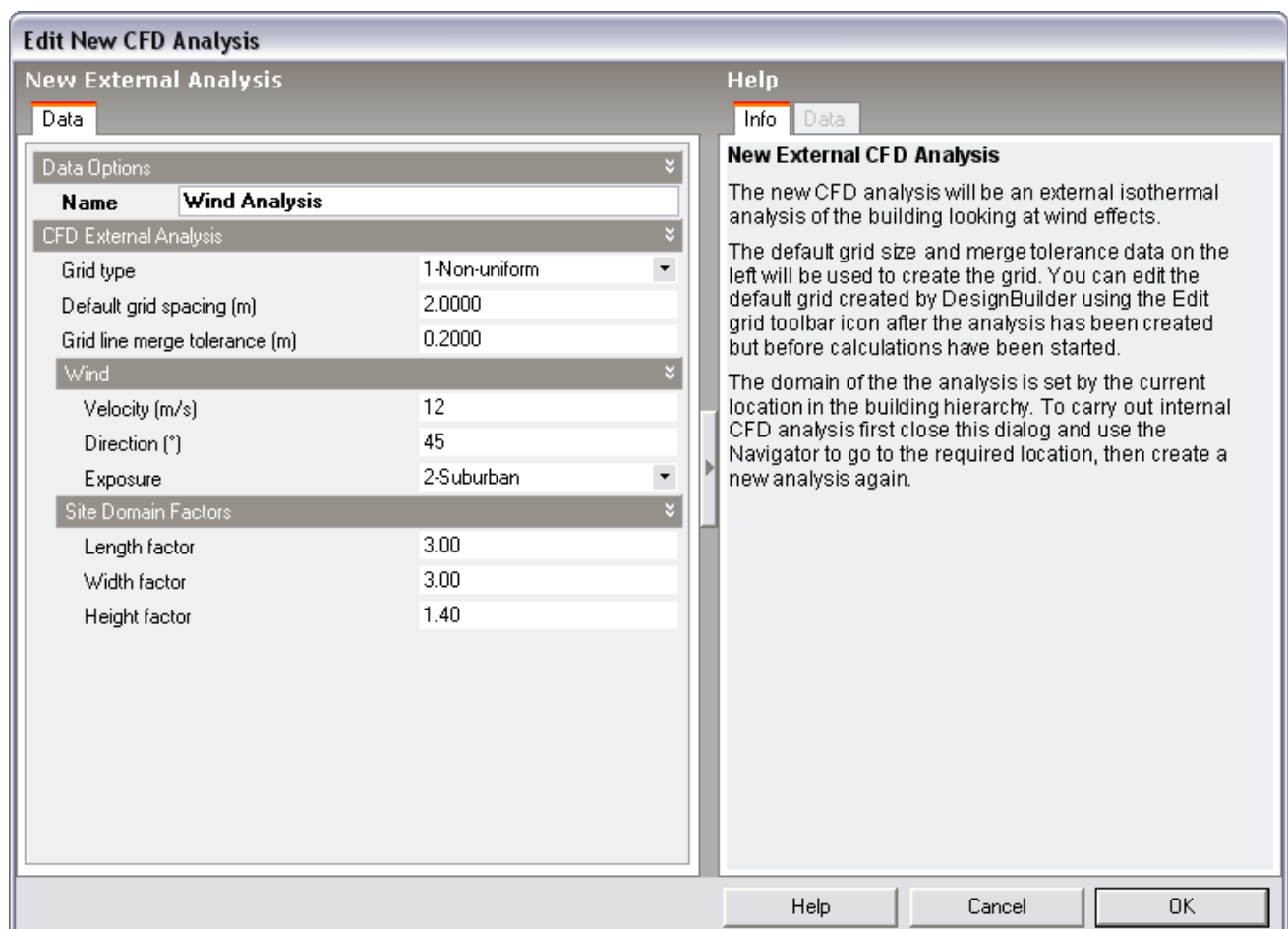
The offset in the y-direction between the edge of the boundary condition and the first cell.

New External CFD Analysis

To create a new external CFD analysis, first make sure that you have selected the site object in the model navigator and then click on the CFD screen tab:



The **New CFD Analysis** dialog is displayed which allows you to name the analysis, define grid generation variables, wind data and the extents of the domain to be included in the analysis:



Grid type

With External CFD analyses you can select from 2 grid types:

- **1-Non uniform** - the non-uniform grid used for external analyses is identical to the grid used for internal analyses whereby a list of key coordinates are identified from the geometry and these key coordinates are used to set up grid regions which are then spaced uniformly using the default grid spacing. The **Grid line merge tolerance** setting only applies to a non-uniform grid and may be used to merge grid lines that are separated by a distance less than the tolerance in order to minimise cell aspect ratio.
- **2-Uniform** - the uniform grid option generates cells of equal spacing along each of the major axes and makes no attempt to fit the grid to the model geometry, the software then identifies all cells that lie within the geometry as solid. The cells are generated using a spacing as close to the **Default grid spacing** setting as possible given the domain extents. The uniform grid calculation method is much more approximate than with the non-uniform grid but can result in faster calculations.

Tip: Uniform grids are preferred for large and or very complex external CFD analyses to encourage convergence.

Default grid spacing

On entry to the CFD screen after completing and closing the new CFD analysis dialog, the CFD grid is automatically generated throughout the overall external domain extents. The grid is created by first determining key points obtained from constituent model blocks along the major axes, the distance between each of these key points being known as regions. Each CFD grid region along each major axis is automatically spaced using the 'default grid spacing' dimension.

Grid line merge tolerance

A potential problem when generating the grid is the creation of cells with a high aspect ratio that can lead to instability in the equation solver. In order to avoid such cells, grid lines that are very close together can be merged. The **grid line merge tolerance** is the maximum dimension that will be used in determining whether or not to automatically merge grid lines.

Wind

Wind velocity

Enter the required free stream wind velocity in m/s (measured at 10m above ground).

Wind direction

The wind direction is defined clockwise from North. The default direction is 270°, i.e. Westerly.

Wind exposure

The free stream wind velocity is corrected for height above ground and surrounding terrain using an empirical relationship. The following exposures can be selected:

- **1-Open country**,
- **2-Suburban**
- **3- Urban**.

Wind speed profile used by DesignBuilder (obtained from ASHRAE Fundamentals "Airflow around buildings")

$$U_{(h)} = U_{met} \left(\frac{\delta_{met}}{h_{met}} \right)^{a_{met}} \left(\frac{h}{\delta} \right)^a$$

Where:

$U_{(h)}$ = wind speed (m/s) at height h (m)

U_{met} = wind speed (m/s) measured at 10m above ground level

δ_{met} = meteorological site air layer thickness (270m– ‘Country’ exposure)

a_{met} = exponent for meteorological site (0.14 – ‘Country’ exposure)

h_{met} = measurement height for meteorological (10m)

h = height above ground level (m)

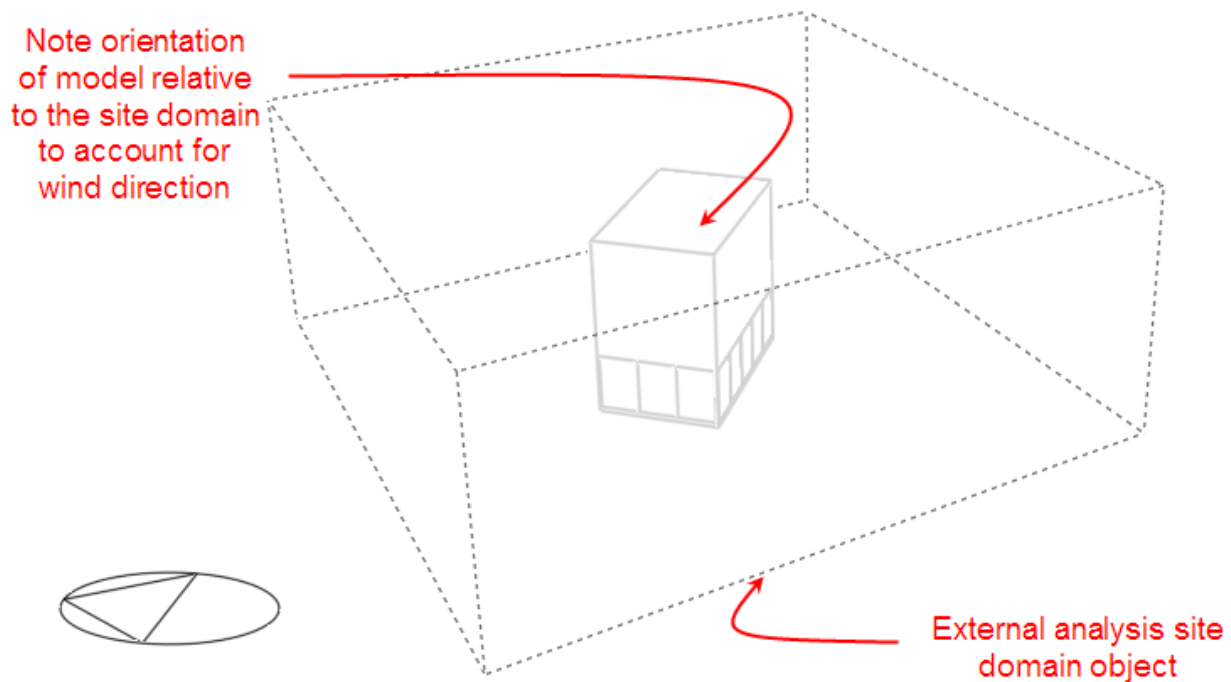
ASHRAE Exposure	Corresponding DesignBuilder Wind Exposure	a	δ (m)
Country	1-Open country	0.14	270
Urban	2-Suburban	0.22	370
City	3- Urban	0.33	460

Air layer thickness and exponent values for various exposures

Site Domain Factors

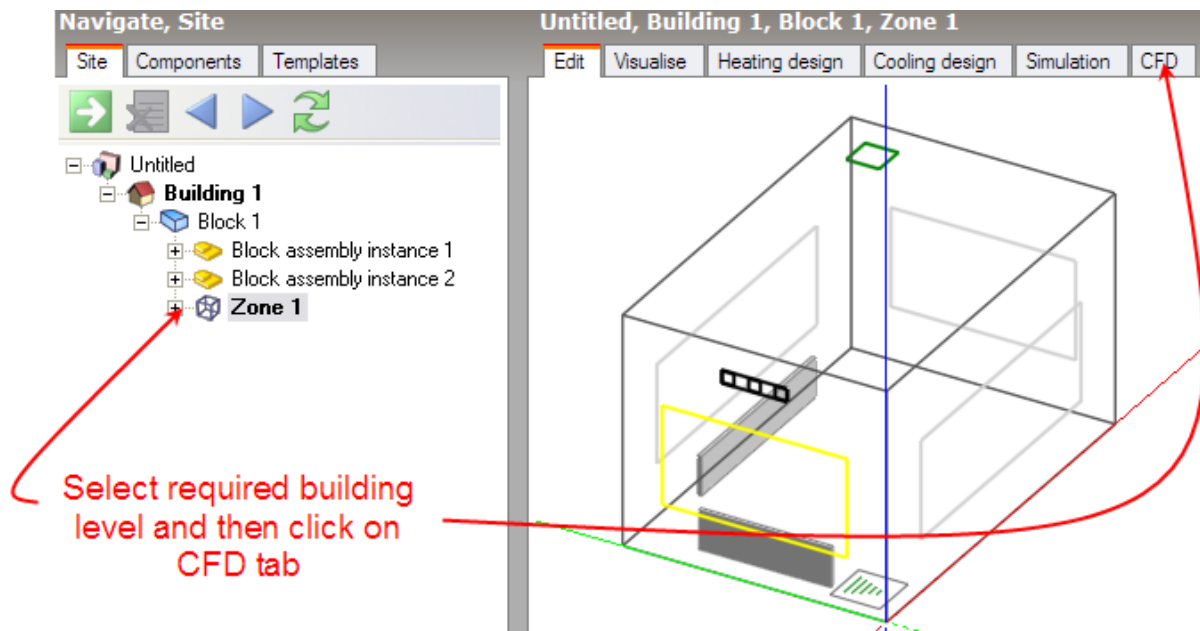
The length, width and height site domain factors are multipliers that are applied to the overall dimensions of the building model in order to arrive at an external volume or domain across which the analysis is carried out. The default factors are 3.0 applied to the length and width and a factor of 2.0 that is applied to the model height.

After clicking on the OK button, the CFD screen is displayed showing the model in conjunction with the site domain object:



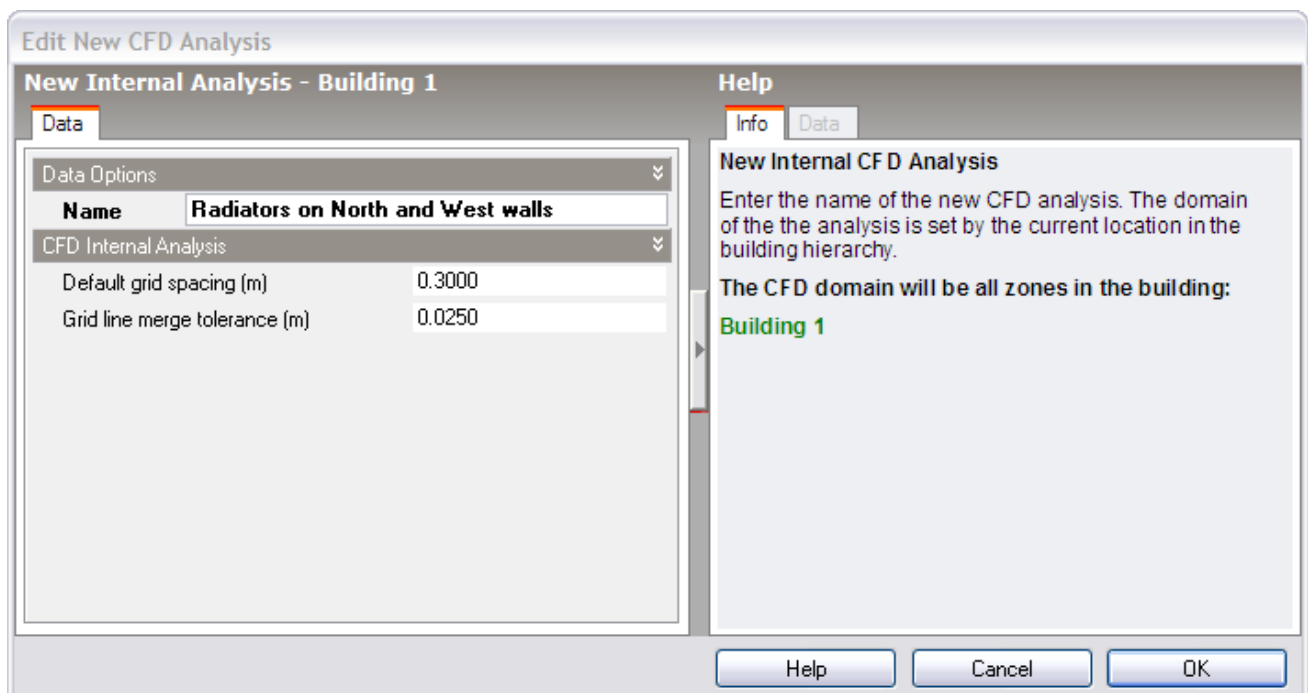
New Internal CFD Analysis

To create a new internal CFD analysis, first make sure that you have selected the appropriate level object (building, block or zone) in the model navigator and then click on the CFD screen tab:



New CFD analysis dialog

The **New CFD Analysis** dialog is displayed which allows you to name the analysis and define grid generation variables:



Default grid spacing

On entry to the CFD screen after completing and closing the new CFD analysis dialog, the CFD grid is automatically generated throughout the overall external domain extents. The grid is created by first determining key points obtained from constituent model blocks along the major axes, the distance between each of these key points being known as grid regions. Each CFD grid region along each major axis is automatically spaced using the 'default grid spacing' dimension.

Grid line merge tolerance

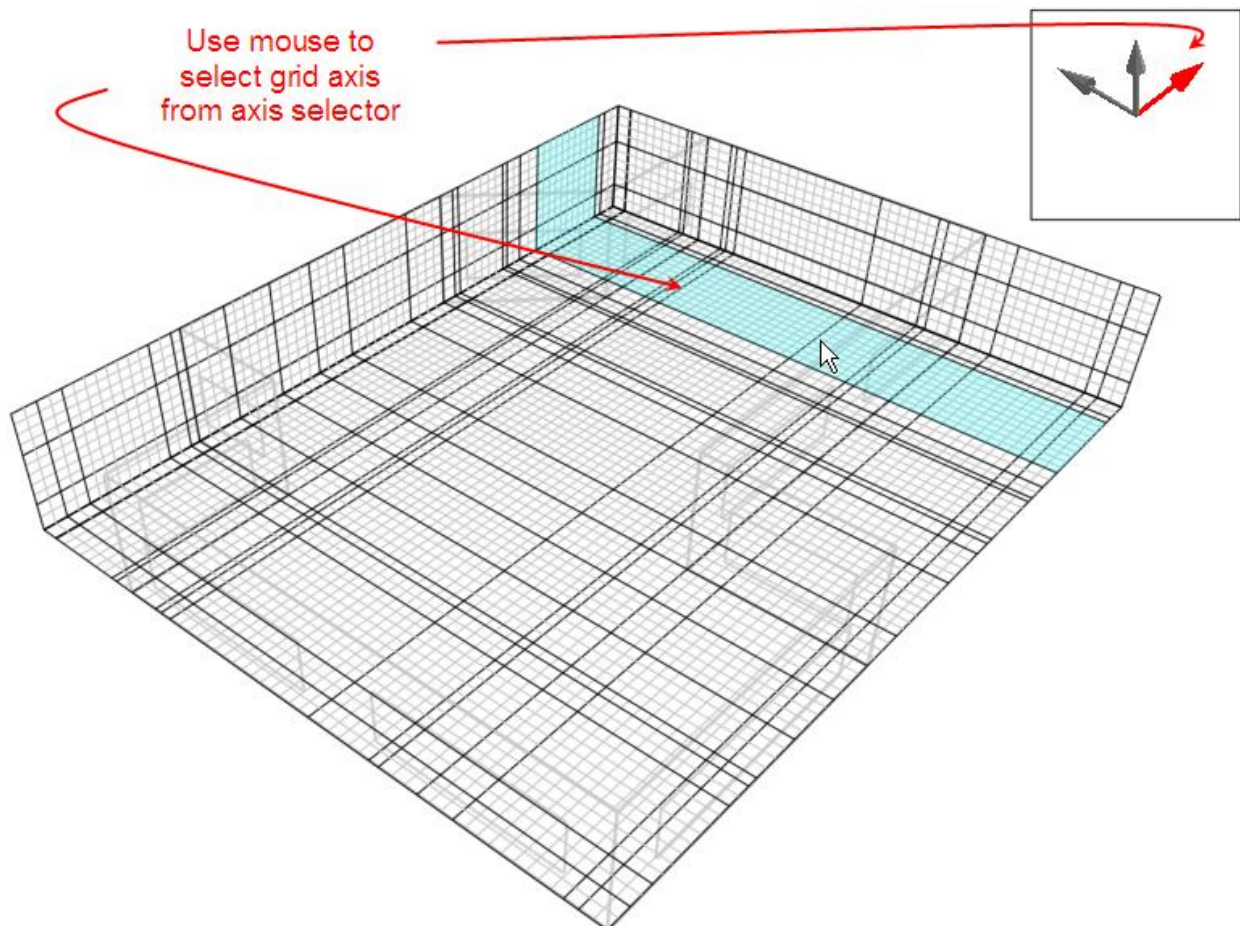
A potential problem when generating the grid is the creation of cells with a high aspect ratio that can lead to instability in the equation solver. In order to avoid such cells, grid lines that are very close together can be merged. The **grid line merge tolerance** is the maximum dimension that will be used in determining whether or not to automatically merge grid lines.

Editing The CFD Grid

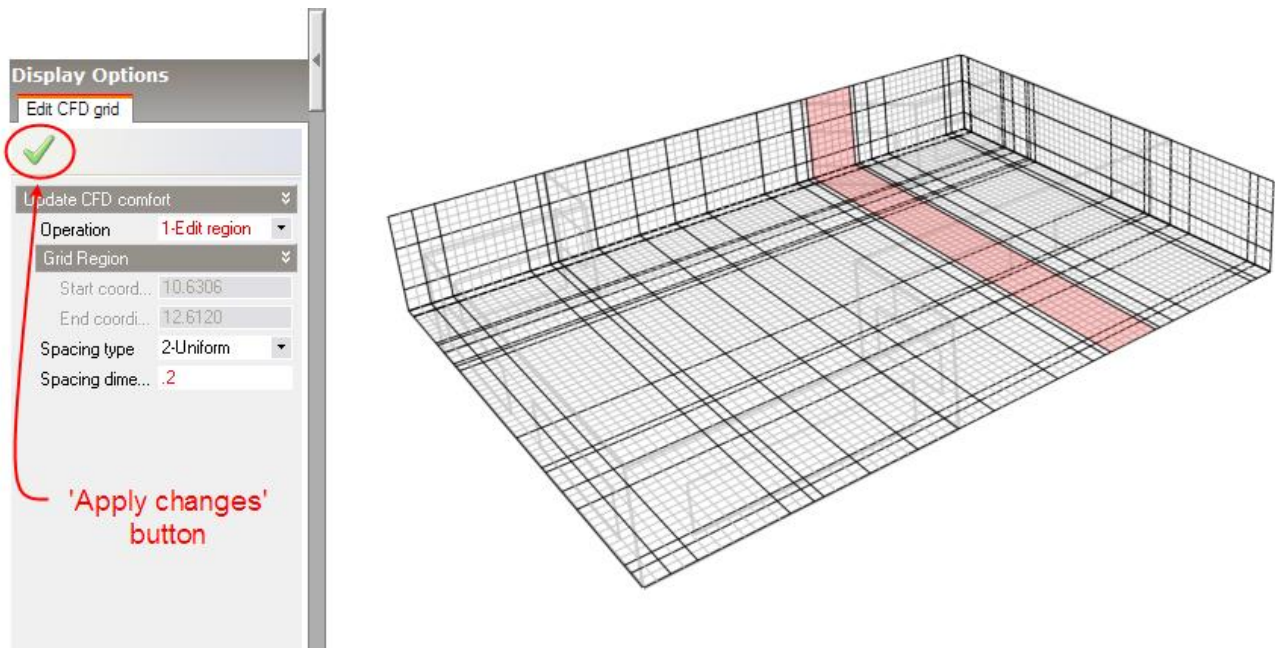


As soon as a new analysis data set has been created, a default CFD grid is generated using the default grid spacing defined as part of the data set. The default grid may be edited using the 'Edit CFD Grid' tool which allows you to change the spacing used for default regions, insert additional regions or remove previously inserted regions.

When you select the 'Edit CFD Grid' tool, the grid is displayed with the building outline overlaid as a ghosted wire frame. Grid regions are displayed along the major axes bounded by dark grey lines, the auto-generated grid region spacing lines being displayed in a lighter grey. The currently selected grid region is selected in cyan or yellow if the region is a non-default inserted region. To select regions on one of the other major axes, you can change the axis by clicking on the required axis of the axis selector displayed at the top right of the screen:



To edit a grid region, move the mouse cursor across the grid to the required region that will highlight to indicate that it has been selected and then click the mouse button. The selected region will then highlight in red and the 'Edit CFD Grid' data panel will be displayed at the bottom left of the screen:



The following settings are available on the **Edit CFD Grid** data panel:

Operation

- **1-Edit Region** - change the spacing of the currently selected region.
- **2-Insert Region** - insert a new custom key coordinate at a defined point in the current region to form an additional region.
- **3-Remove Region** - this option is only available if the current region has been previously inserted and allows you to remove it.

New coordinate (m)

If the 'Insert Region' operation has been selected, this setting is displayed allowing you to define a custom coordinate within the current region. Note that the coordinate must lie in the range defined by the displayed start and end coordinates.

Spacing Type

The spacing type can be set to one of the following:

- **1-None** - the region is not sub-divided.
- **2-Uniform** - the region is sub-divided into a number of equal sub-divisions, the dimension of which is calculated to be as close to the specified 'Spacing dimension' as possible.
- **3-Increasing power-law** - the location of each sub-division grid line within the region increases as the power of the spacing number, which starts at the beginning of the region. So that if i represents the index number of the grid line counted from the start of the region, the coordinate of the i^{th} sub-division grid line is calculated using the following relationship: $x_i = (\text{region dimension})(i/n)^{\text{power}} + x_s$
- **4-Decreasing power-law** - the location of each sub-division grid line within the region decreases as the power of the spacing number, which starts at the end of the region. So that if i represents the index number of the grid line counted from the start of the region, the coordinate of the i^{th} sub-division grid line is calculated using the following relationship: $x_i = (\text{region dimension})[1 - (i/n)^{\text{power}}] + x_s$
- **5-Symmetric power-law** - the coordinate of the i^{th} sub-division grid line is calculated using both increasing and decreasing power-law relationships that meet at the middle of the region:
For $i \leq n/2$: $x_i = [(\text{region dimension})/2](2i/n)^{\text{power}} + x_s$

For $i \geq n/2$: $x_i = [(\text{region dimension})/2][2 - (2i/n)^{\text{power}}] + x_s$

Spacing dimension (m)

If the uniform spacing type has been selected, this setting is displayed and allows you to enter the dimension used for sub-dividing the region.

Spacing power

If the Increasing/Decreasing/Symmetric power law spacing type has been selected, this setting is displayed and allows you to enter the power used in the associated power-law spacing relationship.

Number of divisions

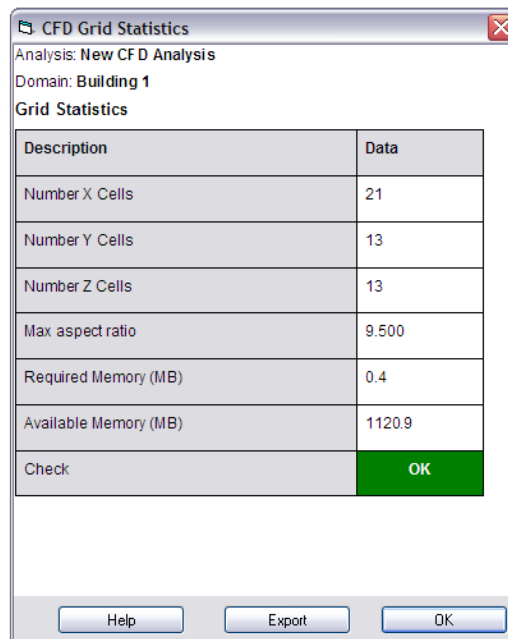
If the Increasing/Decreasing/Symmetric power law spacing type has been selected, this setting is displayed and allows you to enter the number of divisions used in the associated power-law spacing relationship.

After selecting an operation and adjusting the required settings, you then need to click on the 'Apply changes' button to update the selected grid region with the current settings.

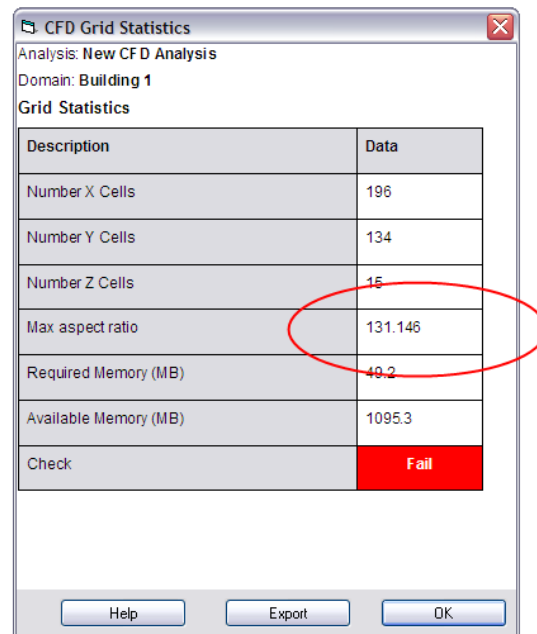
CFD Grid Information



Information about the CFD grid can be obtained using the **Show CFD grid statistics** tool, which displays the CFD grid statistics dialog:



If the maximum cell aspect ratio exceeds the allowable limit of 25, or the required memory exceeds the physical memory available, the 'Check' will indicate a failure:



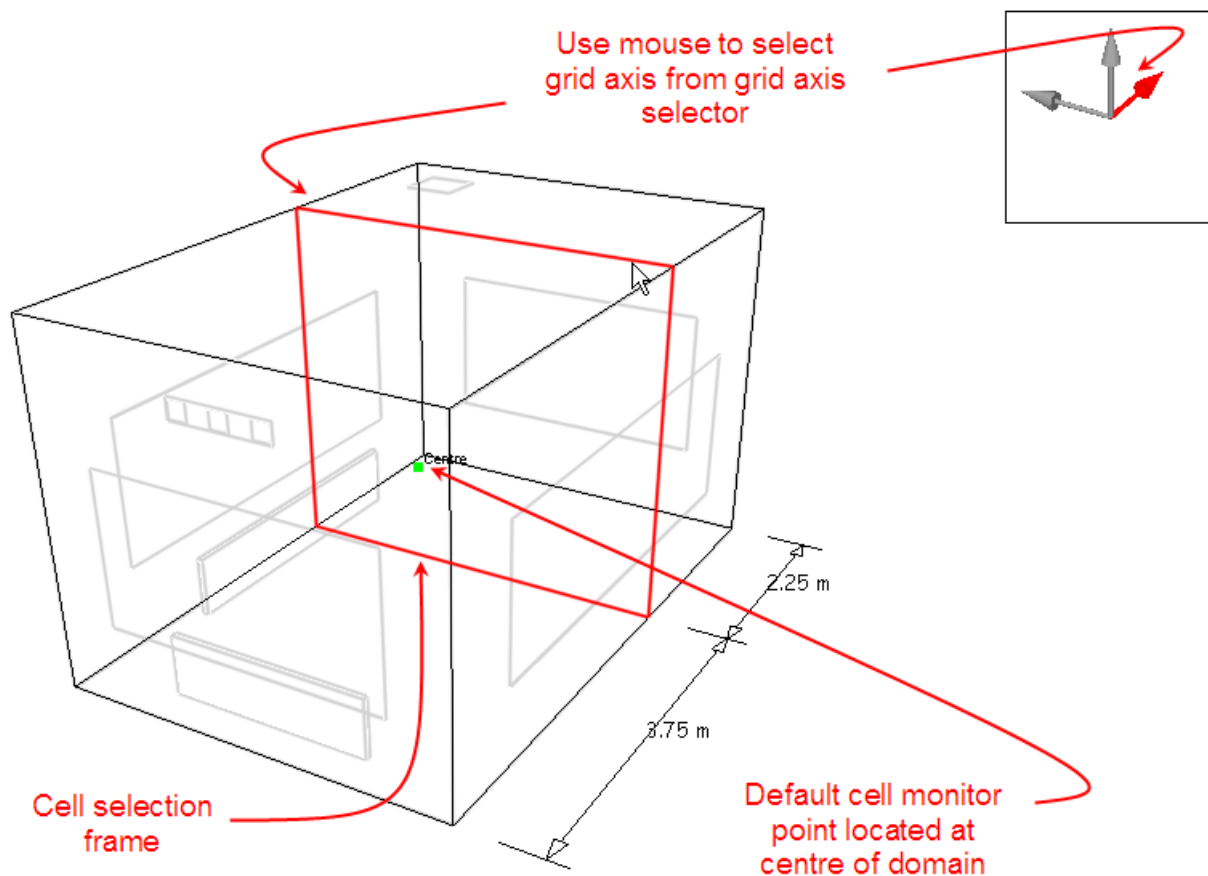
Setting Up CFD Cell Monitor Points



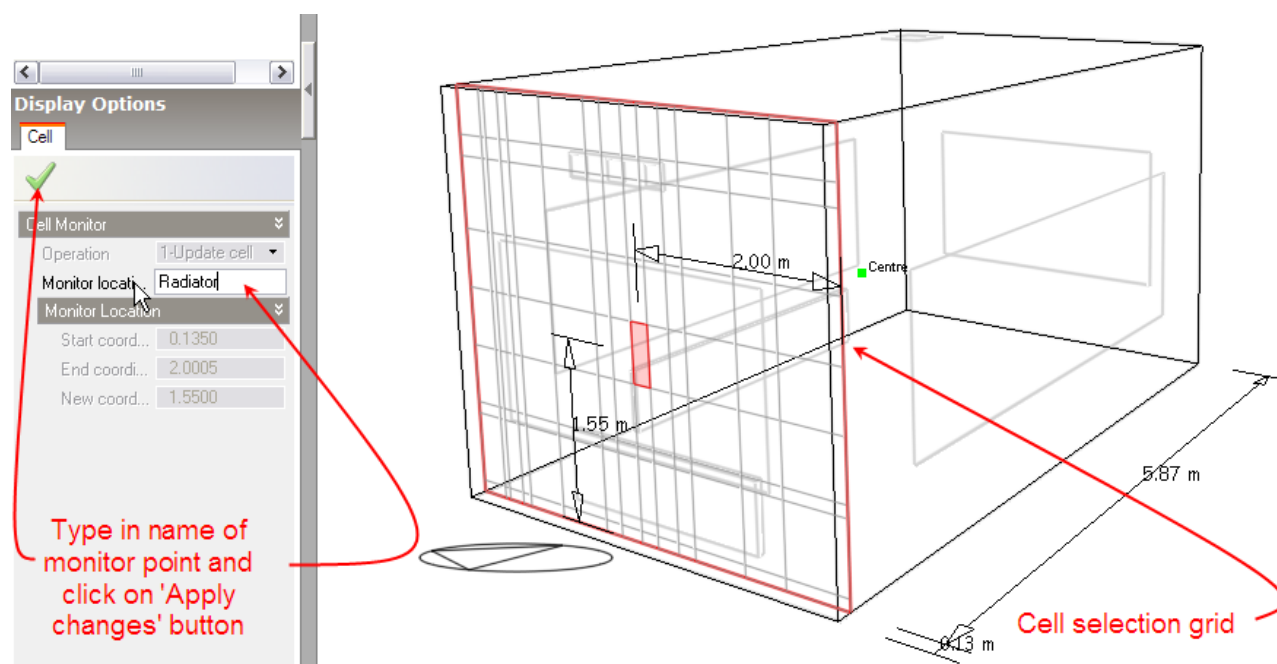
The CFD calculation process is iterative and is considered to have completed once 'convergence' has been achieved. The main indicator for convergence is the point at which the finite difference equations for all cells are satisfied by the current values of the appropriate dependent variables. However, it can be very useful in many cases to monitor the variation of the dependent variables at specific locations throughout the calculation domain in order to observe the point at which they stabilise. It may be acceptable, in many cases, to terminate a calculation as soon as the dependent variables have achieved adequate stability but before the residuals have arrived at their defined termination values

When a project is first created, a default central cell monitor point is automatically added to the domain. The Define CFD monitor points tool enables you to identify up to ten cells throughout the calculation domain at which you want to monitor the variation of the calculated variables.

After selecting the **Define CFD monitor points** tool, the cell monitor screen is displayed which includes the model view together with a cell selection frame, perpendicular to the currently selected axis. You can select a plane along the current axis by moving the mouse cursor across the model in the direction of the axis. To change the axis, click on the required axis on the axis selector tool, which is located at the top right of the screen:



After selecting the required axis, move the cell selection frame along the axis, and then click the mouse button to select a plane at the required distance along the axis. A cell selection grid is then displayed in the selected plane allowing you to move the mouse cursor to the required cell in that plane:



After moving the mouse cursor to the required cell, click the mouse button to display the cell monitor data panel at the bottom left of the screen.

The following settings are available on the cell monitor data panel:

Operation

This control is only active if you have selected a cell for which a monitor point has already been defined. The following options are available:

- **1-Update cell** - add a monitor point or edit the name of a previously defined point.
- **2-Remove cell** - remove a previously defined monitor point.

Monitor location name

Enter a name for monitor point - this is used later when selecting data for the monitor point to be displayed on the [CFD Calculation Options](#) dialog.

After changing the required settings, click on the 'Apply changes' button to update the selected cell. You can cancel the Define CFD monitor points command by pressing the <Esc> key or by selecting another tool.

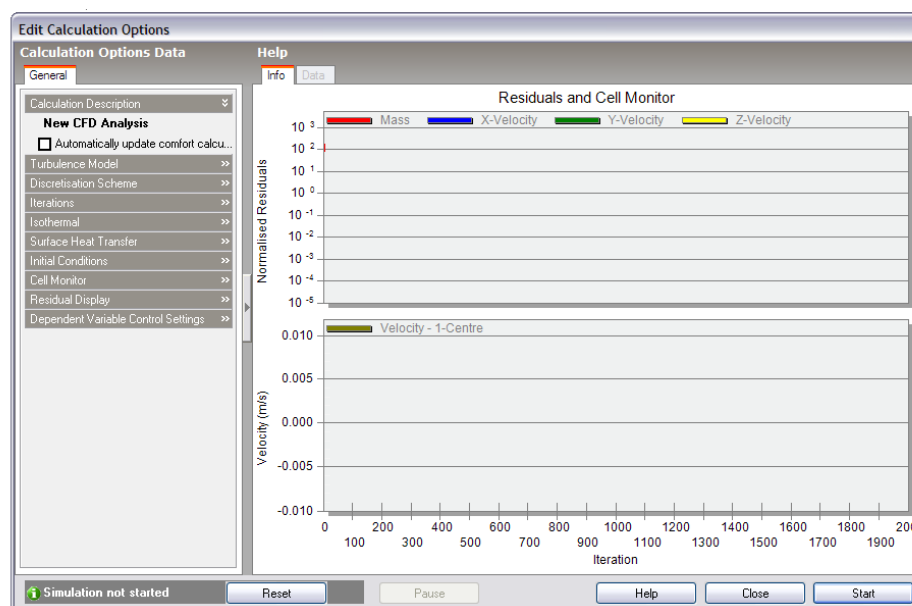
CFD Calculation Options



To run the calculations, click on the Update calculated data tool. If a problem is found with the grid generation, the [CFD Grid Statistics](#) dialog is displayed showing the error. There are three possible error conditions:

1. **Maximum cell aspect ratio exceeds the limit** - you will need to delete the existing project and create a new one either reducing the default grid spacing dimension or increasing the grid line merge tolerance by a suitable amount (see the [CFD Analysis Manager](#) section).
2. **Required memory exceeds the available memory** - you will need to delete the existing project and create a new project with larger default grid spacing or consider simplifying the problem definition in terms of complexity of assemblies, etc.
3. **Flow imbalance** – the total flow rate specified for supply diffusers does not balance with the total flow rate specified for extract grilles. You will need to add supply diffusers or extract grilles or edit the existing flow rates to establish the correct balance

If no problem is encountered in creating the grid, the 'Edit Calculation Options' dialog will be displayed:



The [Edit Calculation Options](#) dialog is divided into two main sections, the Residuals and Cell Monitor graphs and the Calculation Options Data panel.

Buttons

There are also a group of buttons at the bottom of the dialog, which allow you to control the calculations:

Start

Start, or if the calculations have been paused, re-start the calculations.

Pause

Interrupt or pause the calculations. Once the calculations have been paused, the calculation dialog can be temporarily closed in order to review the results.

Reset

After pausing the calculations, the calculations may then be re-initialised to start from scratch. If the solution is found to diverge and you pause the calculations to decrease the velocity false time steps, you will then need to reset the calculations before re-starting.

The calculation settings panel incorporates the following settings:

Turbulence Model

The following turbulence model options are available:

- **1-Constant effective viscosity:** The constant effective viscosity model is a very simple approach involving the replacement of the molecular viscosity in the Navier-Stokes equations with a constant effective viscosity (typically in the order of 100-1000). Although this model is incapable of modelling local turbulence or the transport of turbulence, it is computationally much less expensive than the k-e model and can be numerically much more stable.
- **2-k-e:** This model is one of the most widely used and tested of all turbulence models, belonging to the so-called RANS (Reynolds Averaged Navier-Stokes) family of models. These models involve replacing the instantaneous velocity in the Navier-Stokes and energy equations with a mean and fluctuating component. The resulting equations give rise to additional terms known as Reynolds stresses and turbulent heat flux components. Reynolds stresses are replaced with terms involving instantaneous velocities where molecular viscosities are substituted for effective viscosities and a similar substitution is conducted for the energy equation. The effective viscosity is the sum of the molecular viscosity and a turbulent viscosity, which is derived from the turbulence kinetic energy and the dissipation rate of turbulence kinetic energy:
where k = turbulence kinetic energy and ϵ = dissipation rate of turbulence kinetic energy
 k and ϵ are both derived from partial differential equations which are in turn derived from a manipulation of the Navier-Stokes equations.

Discretisation Scheme

The following discretisation schemes are available:

- **1-Upwind** - The calculation process involves replacing the defining set of partial differential equations with a set of finite difference equations. The conventional approach to this is to use a Taylor's series formulation, which leads to a set of central difference equations. However, although this approach is physically realistic for diffusion, it is not found to be realistic for convection because of its one-way nature, i.e. upwind conditions affect downwind conditions but not the reverse. The upwind scheme allows the convective term to be calculated assuming that the value of the dependent variable at a cell interface is equal to the value at the cell on the upwind side of the interface.
- **2-Hybrid** - a more computationally expensive approach than the upwind scheme but it reduces numerical diffusion at high values of Pe number.
- **3-Power-Law** - The power-law scheme is arguably more accurate than the hybrid scheme but is more computationally expensive.

Iterations

This is the maximum number of iterations conducted by the outer iterative calculation loop (see 'CFD Calculations and Convergence' section). The calculations will terminate when the number of iterations reaches this value regardless of whether or not the solution has converged.

Isothermal

If the Isothermal checkbox is checked the temperature is assumed to be constant throughout the calculation domain and the energy equation is removed from the calculations.

Surface Heat Transfer

Surface heat transfer option

When the **Isothermal** option is not selected inside surface heat transfer options can be selected from the following 2 options:

- **1-Calculated** - Surface heat transfer coefficients are calculated using a wall function if the k-e turbulence model has been selected.
- **2-User-defined** - Surface heat transfer coefficients can be specified separately for ceilings, wall and floors (below). The heat transfer is calculated by combining the specified surface transfer coefficient with the cell surface area and the delta T between the surface and the adjacent cell.

Surface Heat Transfer Coefficients

When the **2-User defined** surface heat transfer option has been selected, enter the inside convective surface heat transfer coefficients. Coefficients can be entered separately for ceilings, floors and walls.

Ceiling

The ceiling inside surface convective heat transfer coefficient is used in the heat flux calculations between all downward-facing surfaces, including zone ceilings, roofs, and downward facing component block surfaces for component blocks that have the **2-Temperature** boundary condition option set. The default user-defined ceiling heat transfer coefficient is 5 W/m²-K.

Wall

The wall inside surface convective heat transfer coefficient is used in the heat flux calculations between all wall and partition surfaces, including zone walls and partitions and non-horizontal component block surfaces for component blocks that have the **2-Temperature** boundary condition option set. The default user-defined wall heat transfer coefficient is 2.5 W/m²-K.

Floor

The floor inside surface convective heat transfer coefficient is used in the heat flux calculations between all upward-facing surfaces, including zone floors and upward facing component block surfaces for component blocks that have the **2-Temperature** boundary condition option set. The default user-defined floor heat transfer coefficient is 0.7 W/m²-K.

Initial Conditions

In some cases, a faster solution can be achieved by setting the initial conditions closer to the final expected conditions. Enter initial velocities in each of the x, y and z directions as well as the initial temperature of the whole domain.

Cell Monitor

Select any defined cell monitor point and associated dependent variable to be displayed on the cell monitor (see [Setting Up CFD Cell Monitor Points](#) section).

Residual Display

Residuals give an indication of the extent to which the calculations have converged to the final solution. They are a measure of the total 'error' still left on the solution. You can select the dependent variables and/or mass for which residuals are to be displayed in the [residuals monitor](#). The mass residual is similar to the dependent variable residuals but is extracted from a continuity equation mass balance for each cell and is the most significant residual in terms of indicating a successfully converged solution.

The residuals have no units and are normalised by scaling to enable similar termination residual magnitudes for each variable.

See also the [Termination residual](#) section below.

It is also important to use monitor points (above) in conjunction with termination residuals because they can provide a more meaningful indication of convergence (i.e. when the values of the dependent variables at the monitor points level out).

Dependent Variable Control Settings

Some CFD models may not converge with default control options and may require some adjustment of settings under the **Dependent Variable Control Settings** header.

The calculations involve a nested iterative scheme whereby dependent variable equations are solved iteratively within an overall outer iterative loop (see [CFD Calculations and Convergence](#) section). The dependent variable control settings enable control inner iterative dependent variable calculations.

Inner iteration

The number of iterations used for the calculation of the dependent variable.

False time step

The finite difference equation set is formulated in the form of a transient equation set although the calculations are essentially a 'snap-shot' in time. The reason for this formulation is that the transient term behaves as a very effective relaxation method, which can slow the change in dependent variables in order to arrive at a more stable solution. The false time step is the time step used in the pseudo-transient term of the dependent variable equation. For forced convection flows, a 'best-guess' optimal false time step is automatically calculated for velocities, however for buoyancy driven flows, a default value of 0.2 is used. Reducing the false time step has the effect of slowing down the change in the dependent variable and can be a helpful remedy for unstable solutions.

Relaxation factor

This is used in the 'text book' relaxation method, which can be used to allow only a proportion of the calculated value of the current iteration dependent variable to be assigned to the variable. However, the false time step is normally the preferred method of achieving under-relaxation.

Termination residual

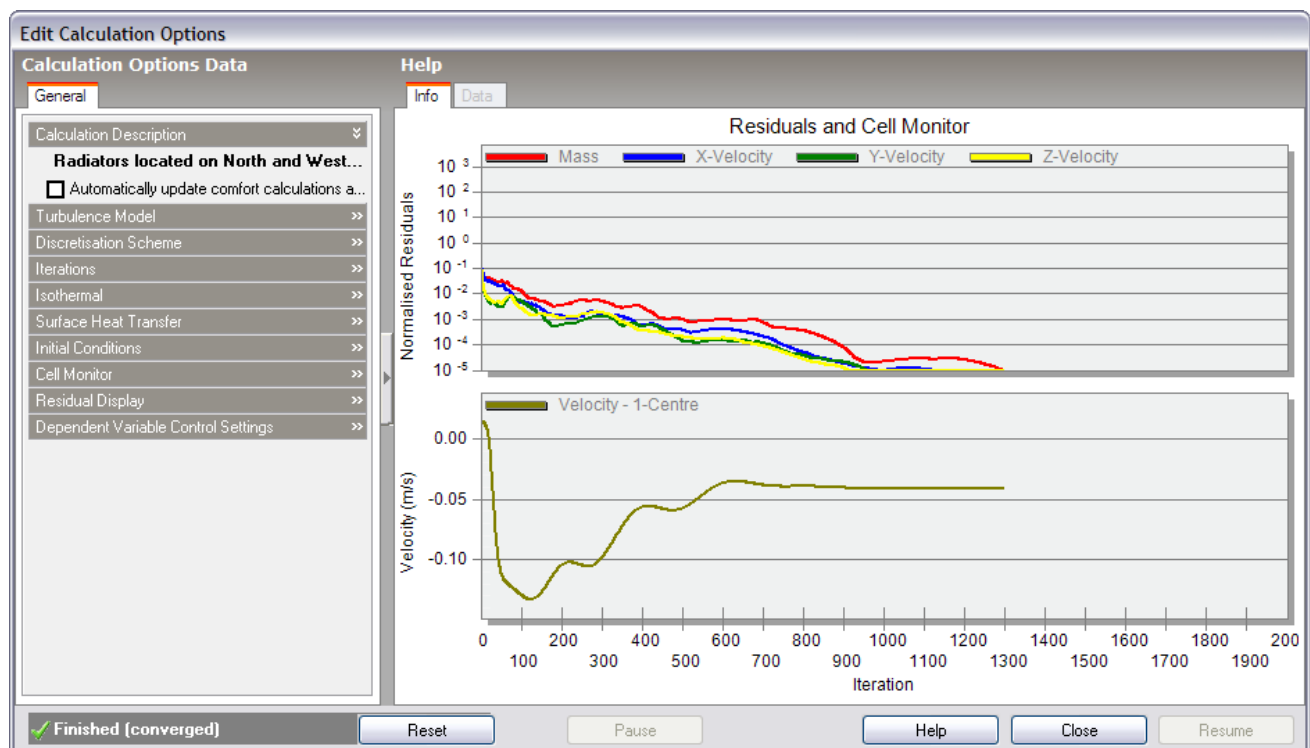
The outer iterative calculation loop is repeated until the finite difference equations for all cells are satisfied by the current values of the appropriate dependent variables, at which point the scheme is said to have 'converged'. The dependent variable residual is the maximum residual quantity for the equation balance across all cells in the domain. The solution is deemed to have converged for each dependent variable when the residual is less than the termination residual.

The [CFD Calculations and Convergence](#) section gives an overview of the calculation methodology that will help in understanding the concepts used to initialise, control and monitor the calculations.

Residuals And Cell Monitor Graphs

During the CFD simulation, you can monitor the progress of calculations by keeping an eye on residual and cell monitor graphs. The value of each selected dependent variable [residual](#) together with the mass residual is plotted on the residuals graph, at the end of each outer iteration of the calculation. You can use the residuals plot to monitor the overall convergence of the solution and to determine whether or not any remedial action is required.

When considering the residual plots, you should bear in mind that the residuals can fluctuate quite markedly throughout the period of the calculations and in some cases rise appreciably before falling and so you need to get an idea of the overall trend after several hundred iterations. The following screenshot shows a plot of residuals and a default centrally located monitor cell plot for the internal analysis illustrated in the [Displaying Results](#) section. Notice that the monitored velocity has reached a steady final value.



If you notice that the residuals are fluctuating wildly or steadily increasing over several hundred iterations, you should pause the calculations and take remedial action (see [Convergence Difficulties and Troubleshooting](#) section).

The cell monitor graph displays the variation in the selected dependent variable for the currently selected monitor cell (see [Setting Up CFD Cell Monitor Points](#) section and **Cell Monitor** setting in the [CFD Calculation Options](#) section). The variation of monitored cell point variables provides a good indication of solution convergence, i.e. when the variation of the variable stabilises.

Convergence Difficulties And Troubleshooting

Use of residuals and monitored variables to check for convergence

Reasonable convergence of the CFD solution can be assumed to have occurred once the [plotted normalised residuals](#) have all stabilised (i.e. they are not oscillating) and the values have dropped to around 0.001 or 10⁻⁴. If the residuals have dropped to very low levels and the [monitored variables](#) are fairly constant (they don't have to be absolutely constant), then you can be confident that you have a good solution (even though a non-

convergence message is displayed). The message is always displayed even if the residuals are very slightly above the termination values. These are conservative values and don't need to be strictly adhered to.

Common convergence problems and solutions

The default CFD settings and grid provided in DesignBuilder should enable a converged solution to be found in most cases, but occasionally, adjustments to the settings or to the problem specification may be needed.

One common cause of CFD simulations not converging is when there are insufficient grid cells between an extract and a supply diffuser. When these components are positioned too close together, the flow can become trapped and to avoid this situation, *we recommend that there are at least 3 grid cells between the extract and supply objects*. You can [edit the grid region](#) between these components and manually add the extra cells. An alternative solution is to use a finer grid.

Also, very small dimensions are likely to lead to flow instabilities (e.g. mass residual showing a "flat" line) and preventing convergence.

An initial large deviation in the monitored variable is expected which normally reduces fairly quickly tending towards stability and terminates at a value of 0.0001.

Generally, if you experience a solution that doesn't converge easily where the residuals appear to remain constant, it can help to re-structure the model definition by removing unnecessary obstructions and aligning any repeating component/assemblies with each other in order to reduce the number of grid lines.

Turbulence models

In some cases a solution can be very difficult, if not impossible, to achieve using the [standard k-ε turbulence model](#). Particularly difficult problems involve very low velocity buoyancy-driven flows where diffusion is dominated by convection and this can cause velocities to fluctuate around a mean value. In these cases, [reducing false time steps](#) may not improve the situation. In order to force a solution, you may consider introducing additional boundary conditions such as occupants, which can change the flow sufficiently to obtain a solution without the problem necessarily becoming unrealistic.

Alternatively, you may consider using the [constant effective viscosity turbulence model](#) with a relatively high turbulent viscosity or turbulent viscosity multiplier but bear in mind that this will introduce an artificially high level of diffusion into the flow and also, the constant effective viscosity model is incapable of modelling the transport of turbulence.

Velocity false time steps

If the [residuals](#) are found to diverge or fluctuate significantly, in many cases the situation can be improved by reducing the [false time steps](#) for the velocity components. The recommended procedure is to continuously half the time steps until a more stable solution is found.

You should only need to reduce the velocity false time steps if you experience a very unstable solution which exhibits diverging residuals or high amplitude oscillations. The time steps have a 'damping' effect on the solution which can increase stability but increases the overall simulation time.

Speeding up CFD simulations

Some tips that can help reduce CFD calculation times are:

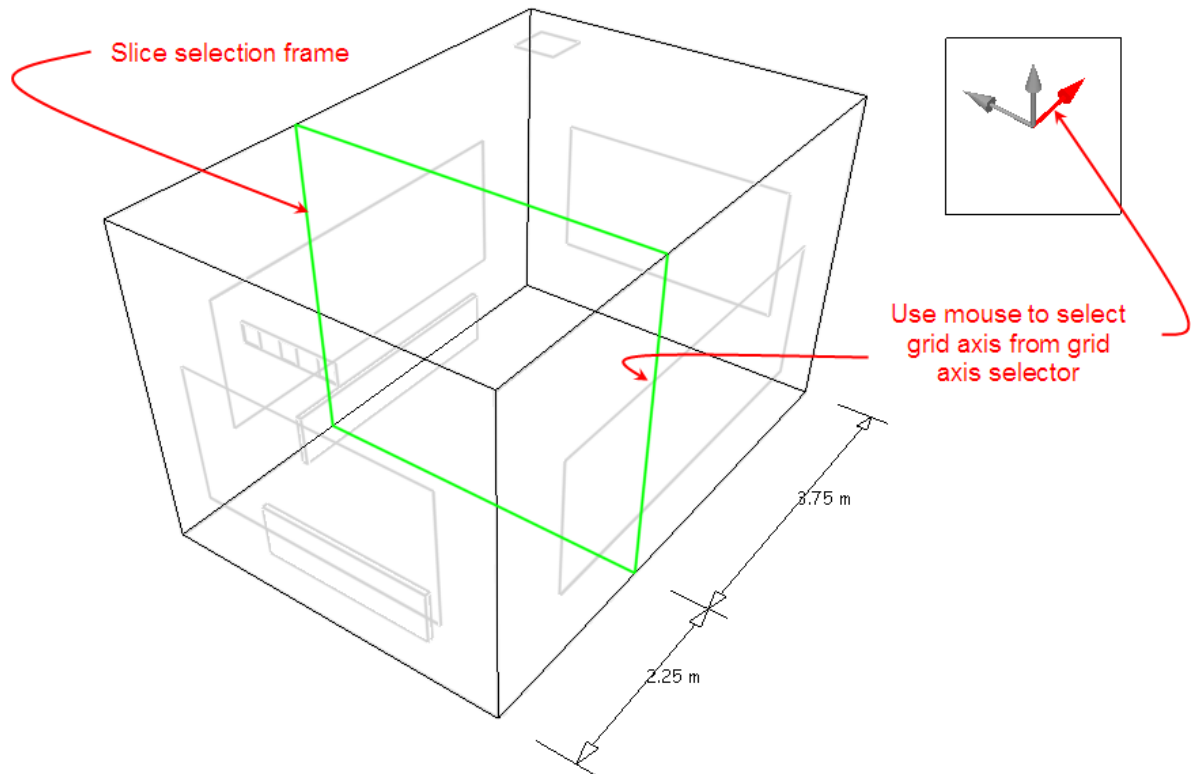
- Keep the model geometry as orthogonal as possible, i.e. ensuring that wherever possible surfaces are aligned with the axes.
- **Grid merge tolerance** should generally not be higher than 10% of grid size.

Displaying Results

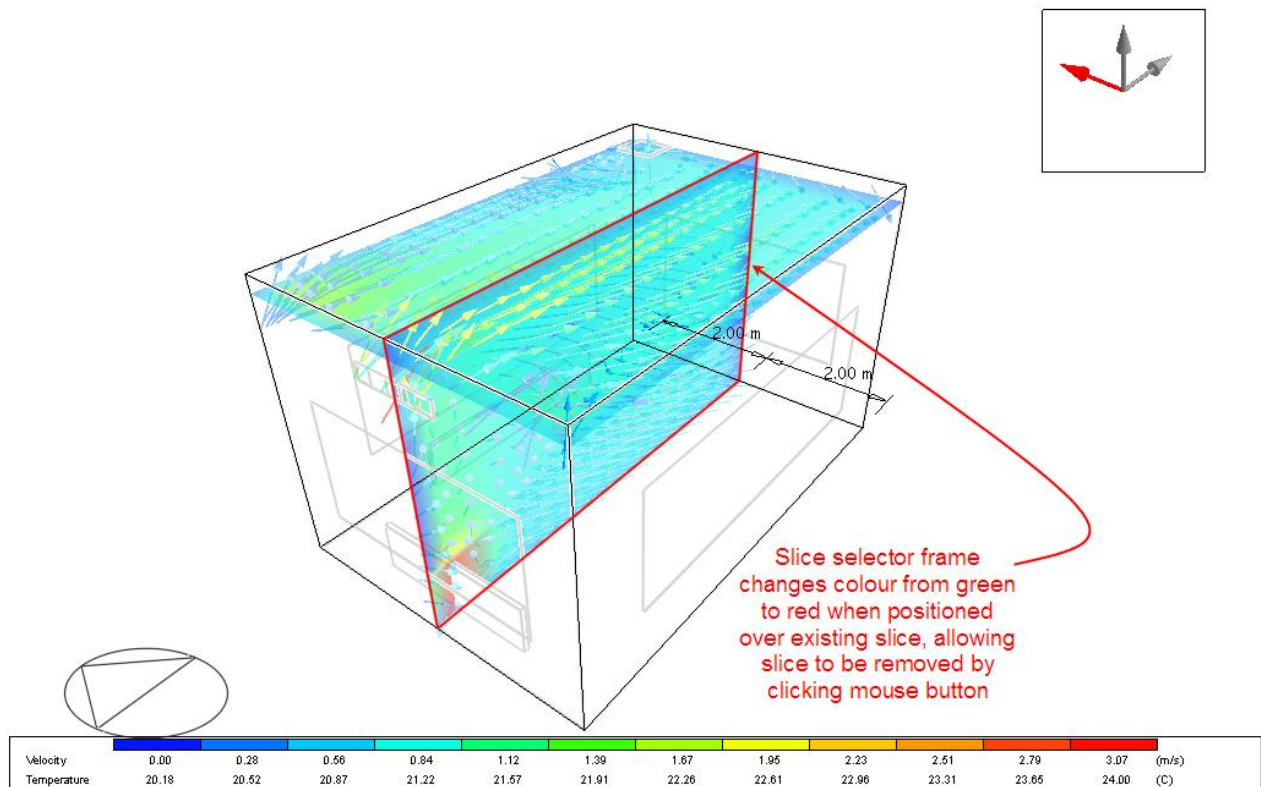


After completing or pausing the calculations, the CFD Display Options data panel is shown in the bottom left of the screen and the **Select CFD slice** tool becomes enabled (above). This is the main mechanism for displaying results. It allows you to select a slice along one of the main grid axes and perpendicular to it, within which any of the selected results can be displayed.

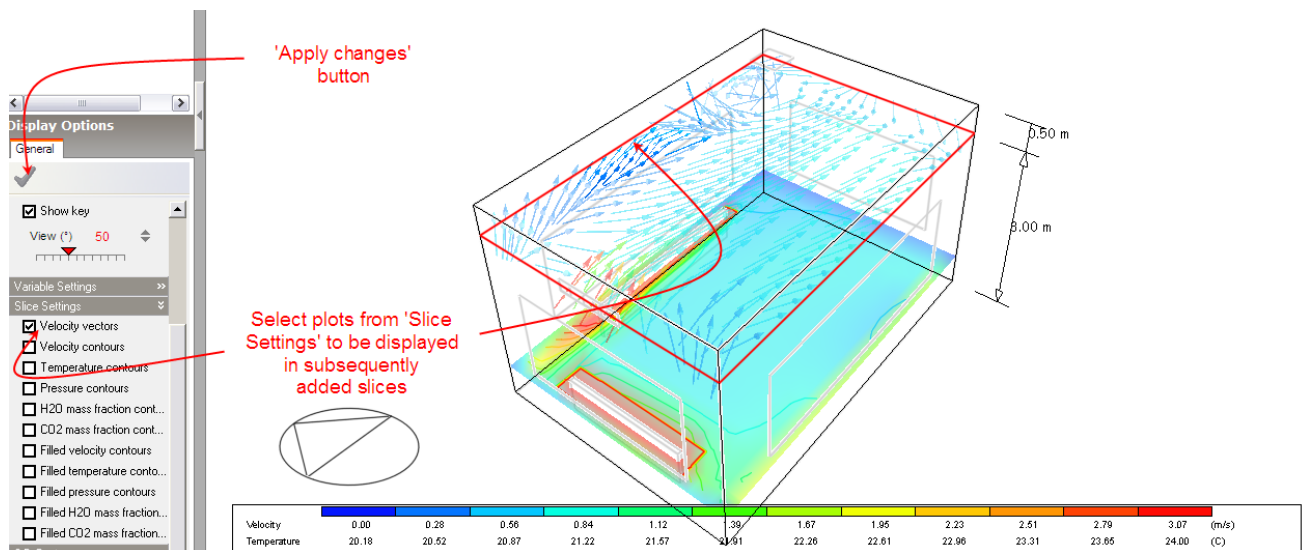
After selecting the **Select CFD slice** tool, the CFD results screen is displayed which includes the model view together with a slice selection frame, perpendicular to the currently selected axis. You can select a slice along the current axis by moving the mouse cursor across the model in the direction of the axis. To change the axis, click on the required axis on the axis selector tool, which is located at the top right of the screen:



After selecting the required axis and moving the slice selector frame to the required position along the axis, click the mouse button to add the slice to the display. Notice that as you move the slice selection frame to a slice that was previously added to the display, the frame colour changes from green to red and if you click the mouse button, the slice is removed from the display:



Slice variable plots are selected under the 'Slice Settings' header on the Display Options data panel and controlled under the 'Variable Settings' header:



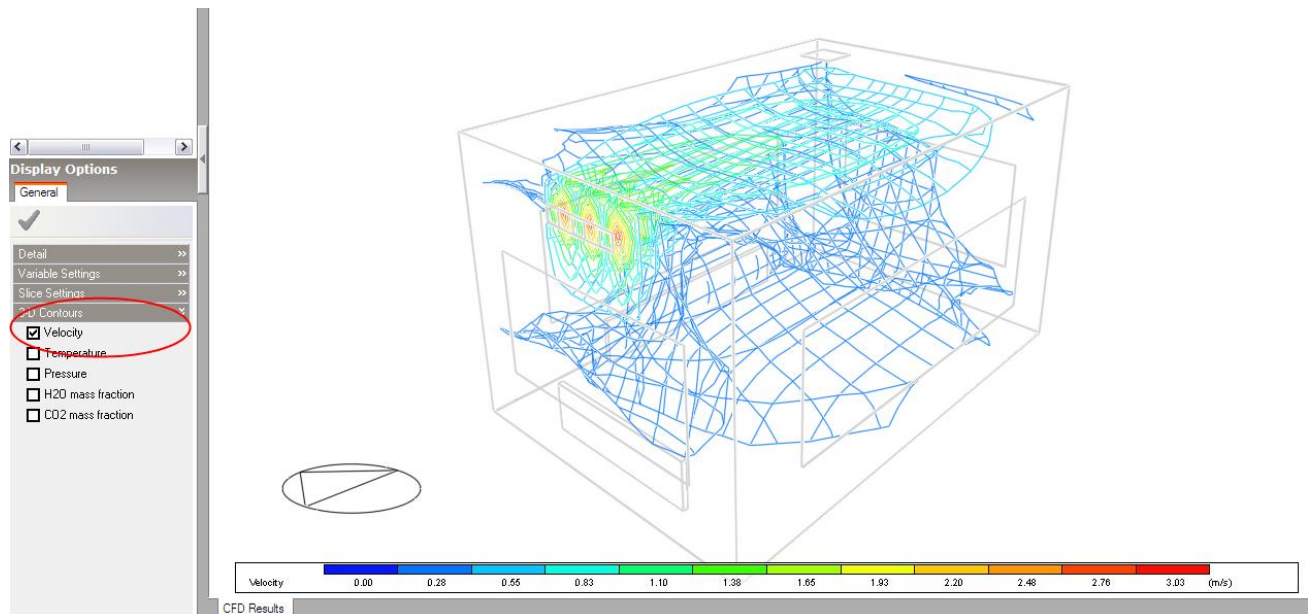
Note that any modifications made to items on the Display Options data panel will only come into effect after clicking on the 'Apply changes' button.

The 'Variable Settings' group contains groups of variable banding data for velocity, temperature and pressure variables. Also, under the variable settings 'Velocity' header, there are two velocity vector settings, 'Maximum vector length' and 'Velocity scale factor'. Velocity vectors are displayed as arrows, the length of which corresponds to the magnitude of the velocity and with the default vector scale factor of 1.0, a length of 1.0m corresponds to a magnitude of 1.0m/s. The maximum vector length is the maximum length of a vector that will be displayed to prevent the display from becoming cluttered with excessively large vectors. Any vector, which has a magnitude greater than this maximum, will be displayed translucently to distinguish it from vectors with lengths that do represent magnitude.

The variable banding data comprises the defined band range, followed by twelve contour band values within the defined range. The band range defines the minimum and maximum variable values between which data will be displayed and the band values are the actual values within that range that are displayed in the form of contours or in the case of velocity vectors, vector colours. The default minimum and maximum variable display values are extracted from the calculated values and this range is then divided into twelve equal increments in order to arrive at the dependent variable contour bands. Each contour band can be edited or switched off altogether. When the band range values are modified, the individual band values are automatically re-calculated.

The 'Slice Settings' group is used to select the plots required to be included in subsequently added slices.

The last item on the Display Options data panel is the 3-D contours group that allows you to select any of the available variables for which a 3-D contour plot is required:



Tip: You can export the calculated values for all displayed CFD slices to a csv file by clicking on the **File > Export > Export CFD results slices** item.

CFD Comfort Calculations



DesignBuilder CFD comfort calculations allow you to carry out a distributed analysis of the comfort conditions in the domain. The comfort calculations are carried out for each cell in the grid and include consideration of the local air temperature and velocity. Radiant temperatures of the surrounding surfaces are also considered. Note that solar radiation through windows or from zone equipment is not considered in DesignBuilder CFD.

After completing or pausing the CFD calculations the **Update CFD comfort** tool will become enabled. Clicking on this tool opens the comfort Calculation options dialog.

Comfort Calculation Options

Before the comfort calculations are run the comfort Calculation options dialog is displayed prompting for information on the values for metabolic rate, clothing and Relative humidity to be assumed in the comfort calculations (which use the [Fanger comfort model](#)).

Metabolic rate

Enter the metabolic rate to be used in the CFD comfort analysis. You should choose a value depending on the activity, age and sex of the occupants. If you are aiming to reproduce conditions in a previous EnergyPlus simulation then you should enter the same value used in that simulation. The default value for simulation is 0.90 met (the average of the recommended values for men and women). The default metabolic rate on this dialog is also 0.9.

Clothing level

Enter the clothing level for the CFD comfort calculations. The default is 1.0 which represents typical winter clothing.

If you are aiming to match comfort calculations with those in EnergyPlus you should ensure that you use the appropriate value. Note that EnergyPlus clothing levels are provided for summer and winter separately so you should make sure to choose the right value depending on the season for this CFD simulation.

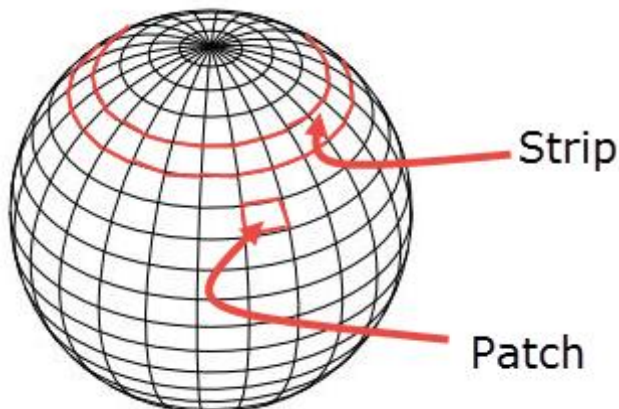
Relative humidity

DesignBuilder CFD does not calculate relative humidity (RH) so this value must be defined manually by the user when calculating comfort. Again if you are aiming to reproduce the same values from a simulation you should copy the appropriate RH from the EnergyPlus results to this dialog. The default value is 50%.

Number of sphere segments in MRT calculations

Select the number sphere segments used in CFD comfort MRT (Mean Radiant Temperature) calculations. Select from 8, 24 and 32. More segments should be selected for more accurate the MRT calculations, especially in geometrically complex spaces. Higher values of this option will lead to longer calculations times.

The MRT calculation works by constructing a sphere at the centre of each grid using a number of 'strips' and 'patches':



Number of strips = Number of segments / 2
Number of patches = Number of segments * Number of strips

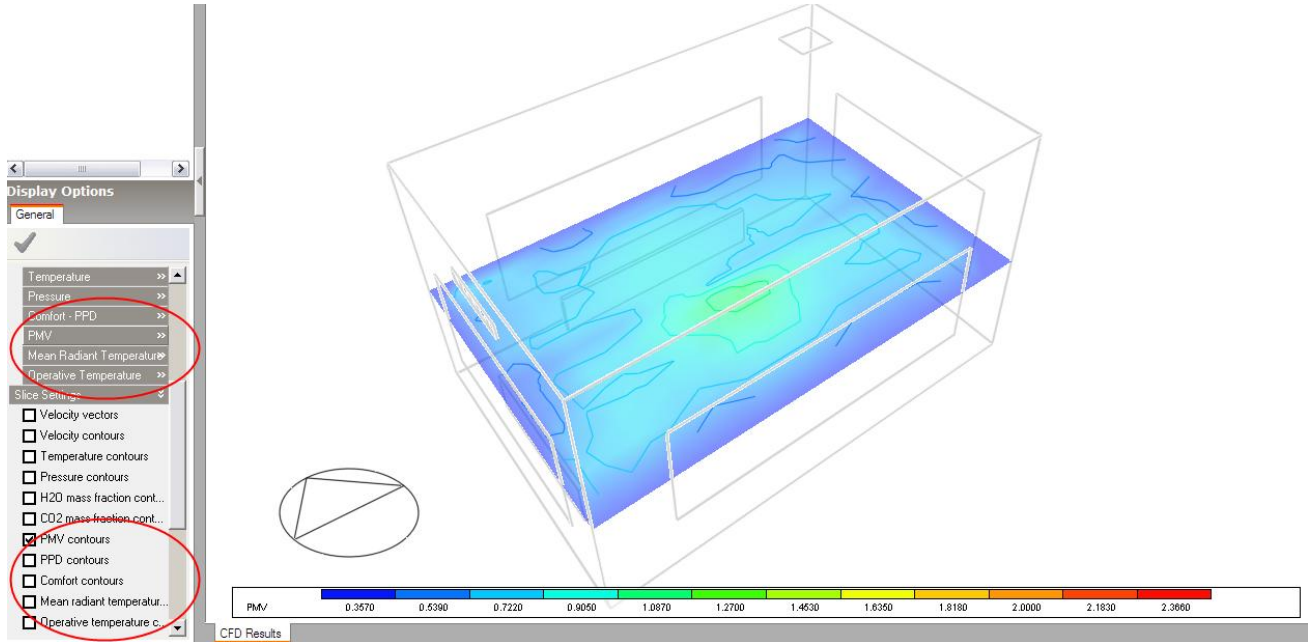
During the MRT calculations the procedure projects a 'ray' from the centre of each sphere through the centre of each 'patch' looking for the nearest surface that intersects with this ray (enclosure, component or assembly surface). The temperature of this intersected surface is then assigned to the patch. The MRT is finally calculated by summing the area/temperature contributions for the whole sphere.

Comfort Results

After pressing OK on the Calculation options dialog, a progress bar will be displayed indicating that mean radiant temperature (MRT) calculations are in progress, followed by another progress bar for the comfort calculations themselves.

After the comfort calculations have completed additional comfort data will become available and corresponding comfort slice display variables are available on the Display options data panel:

- [PMV](#) - Predicted Mean Vote calculated using Fanger equations according to ISO 7730
- PPD - Predicted Percentage Dissatisfied calculated using Fanger equations according to ISO 7730
- Mean Radiant Temperature
- Operative Temperature



You can find out more about Fanger comfort calculations in the [EnergyPlus Thermal Comfort](#) section.

E Age Of Air And Air Change Effectiveness

Age of Air (LMA)

The Local Mean Age of Air (LMA) is the mean time taken for (assumed fresh) air molecules arriving within the domain to travel to the point of interest. The calculation of the local mean age-of-air in DesignBuilder CFD is made by solving an additional partial differential equation. The equation is derived from the contaminant concentration equation (a 'passive scalar' equation because it does not react with the velocity field). The assumption for the calculation of age-of-air is that the production of contaminants throughout the room is uniform:

$$\frac{\partial}{\partial x} \left[\rho u \theta - \left(\frac{\mu}{\sigma_l} + \frac{\mu_t}{\sigma_t} \right) \frac{\partial \theta}{\partial x} \right] + \frac{\partial}{\partial y} \left[\rho v \theta - \left(\frac{\mu}{\sigma_l} + \frac{\mu_t}{\sigma_t} \right) \frac{\partial \theta}{\partial y} \right] + \frac{\partial}{\partial z} \left[\rho w \theta - \left(\frac{\mu}{\sigma_l} + \frac{\mu_t}{\sigma_t} \right) \frac{\partial \theta}{\partial z} \right] = 1$$

Where:

θ = mean age of air

u , v and w = velocity components

ρ = density of air

μ and μ_t = physical and turbulent viscosities respectively

σ_l and σ_t = laminar and turbulent Schmidt numbers respectively

This equation uses the results from the velocity distribution as well as the distribution of turbulent viscosities throughout the domain. Because the distribution of theta does not affect the velocity field, the age-of-air calculations are conducted after the main simulation.

You can request age of air to be calculated on the CFD tab after the main CFD simulation has completed as a separate operation. To do this click on the **Calculate LMA** tool.

Local Air Change Effectiveness (ACE)

After completing the Local Mean Age of Air (LMA) calculation the Air Change Effectiveness (ACE) tool is available.

The ACE value is an indicator of how well a mechanical ventilation system performs in terms of the degree to which air entering a zone is mixed with air that is already contained within the occupied regions of the zone.

The air change effectiveness (ACE), is defined as the age of air that would occur throughout the room if the air was perfectly mixed, divided by the average age of air where occupants breathe. Because the average age of air exiting the room is identical to the age of air that would occur throughout the room if the indoor air were perfectly mixed, the ACE is also the exhaust-air age divided by the average age of air where occupants breathe. In practical terms, the ACE equals the effective ventilation rate at the breathing zone divided by the ventilation rate that would occur throughout the building with perfect mixing and at the same outdoor air supply. A short-circuiting flow pattern between the fresh air supply and return outlets, increases the room-air age and causes ACE to be less than unity. Perfect mixing results in an ACE of unity. Preferentially ventilating the breathing zone with outside air will cause the ACE to be greater than unity.

ASHRAE states that “One common definition of air change effectiveness is the ratio of a time constant to an age of air”. The equation used by DesignBuilder CFD for ACE is:

$$\varepsilon_{I,N} = \frac{\tau_N}{Q_{age}}$$

ACE compliance criteria refers to ASHRAE Standard 129. This standard describes a method for measuring ACE of mechanically ventilated spaces and buildings. The standard gives the following equation for the time constant

$$\tau_N = \frac{\sum_m (Q_{ex,m} A_{ex,m})}{\sum_m Q_{ex,m}}$$

Where:

τ_N = nominal time constant

m = exhaust air stream index

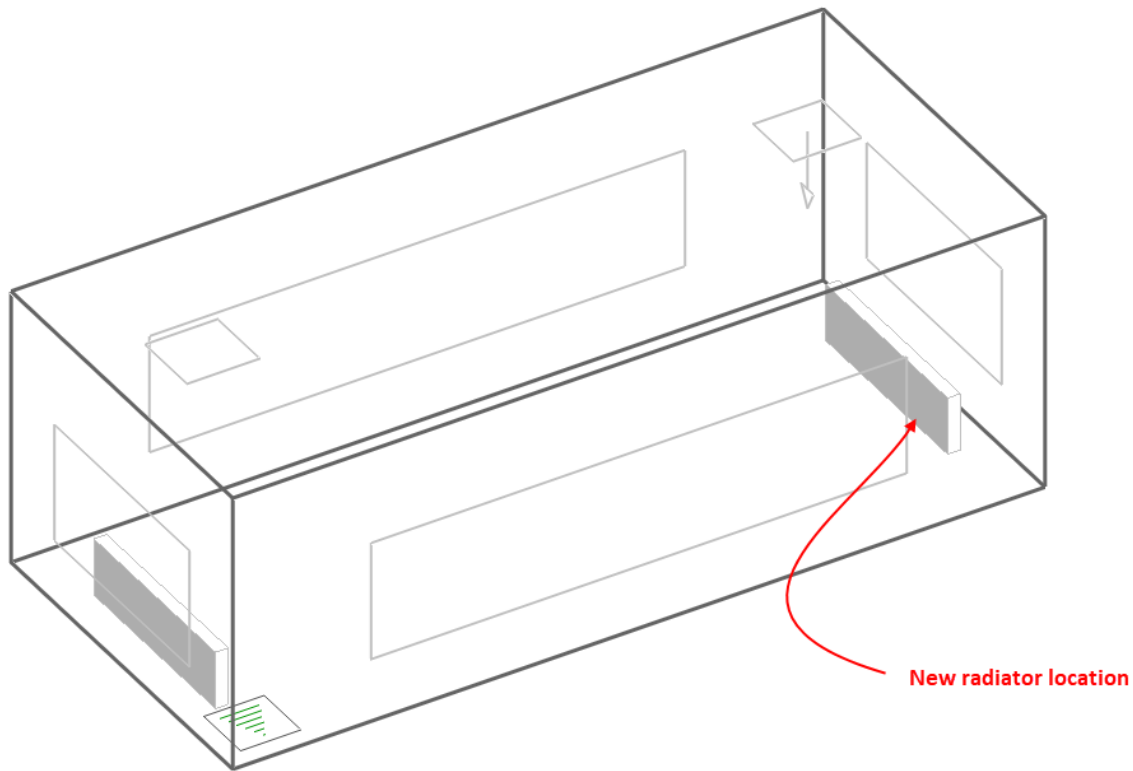
Q_{ex} = exhaust air stream flow rate

A_{ex} = age of air at exhaust

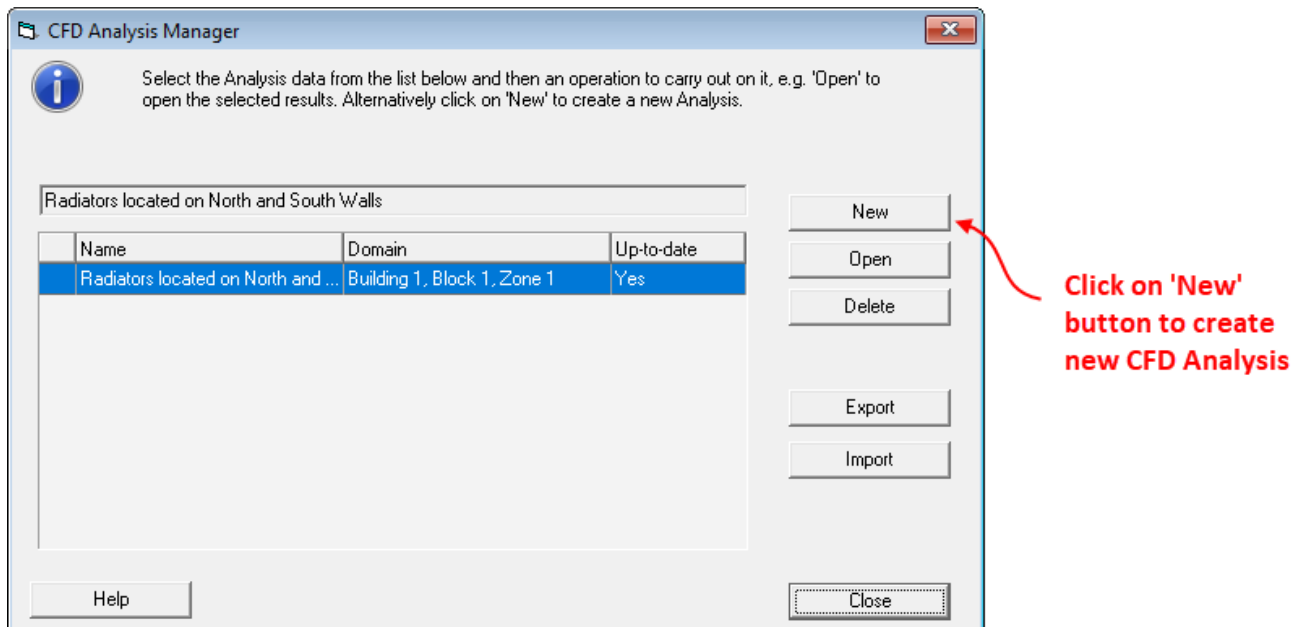
CFD Analysis Manager



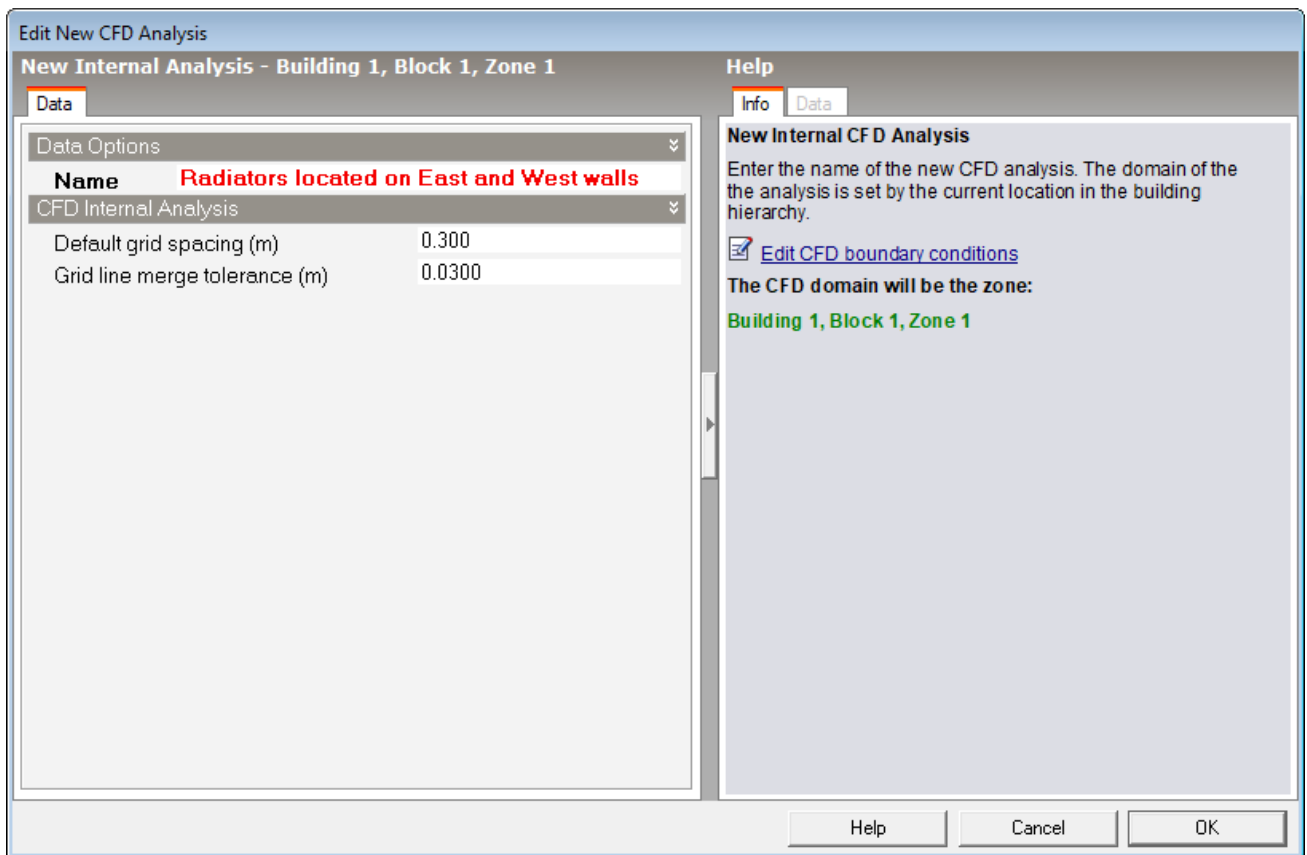
The CFD Analysis Manager dialog allows you to create new CFD projects, open existing projects or remove existing projects. For example, considering the internal analysis example illustrated in the [Displaying Results](#) section, you may want to conduct another analysis after moving the position of a radiator in order to look at the effect on air distribution. You would first go to the Edit screen and modify the radiator layout:



After returning to the CFD screen, you can then click on the CFD results manager tool to open the CFD Results Manager dialog:

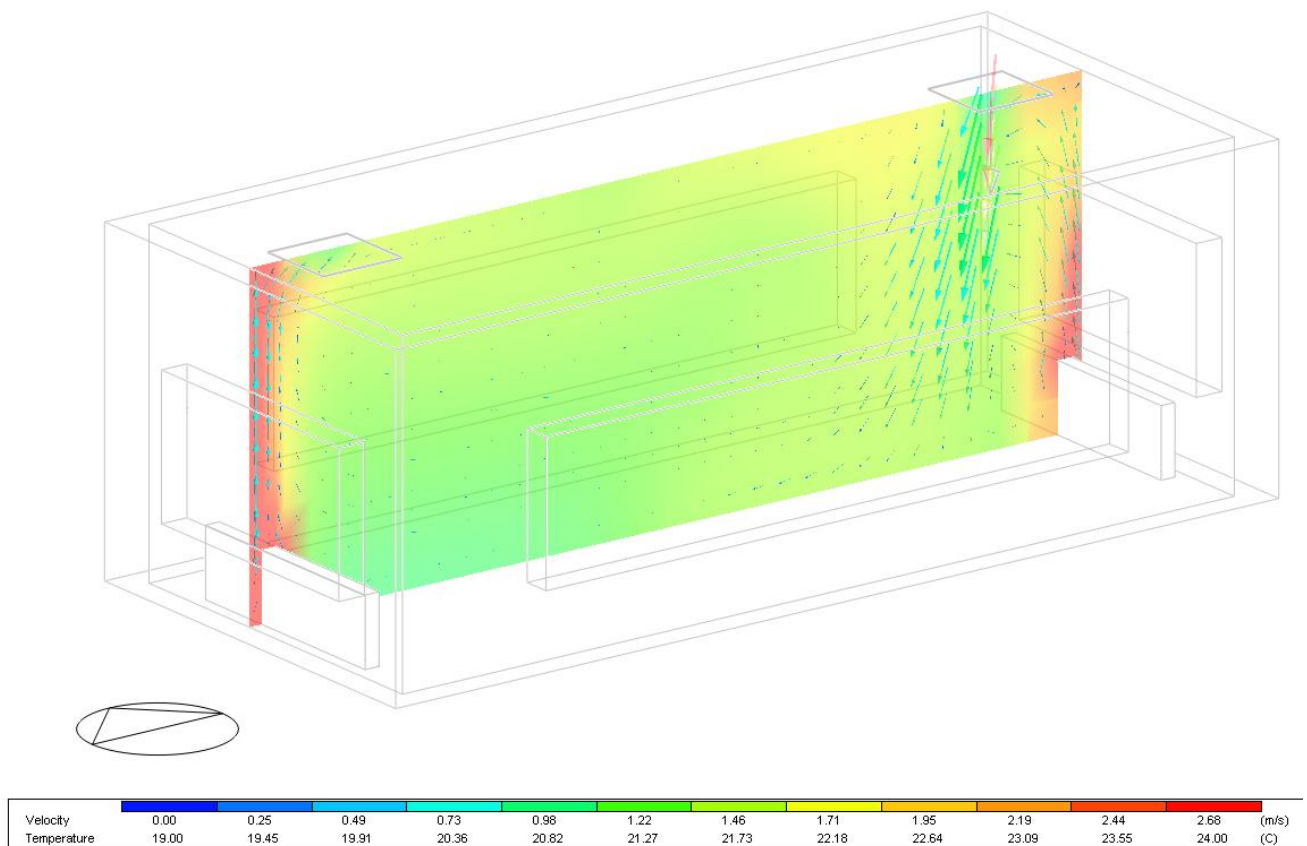


The CFD Analysis Manager dialog incorporates a list of all available projects together with buttons that allow you to create a new project, open a project or delete a project. For each project entry in the list, the first column indicates the project name, the second column the model domain object used for the calculations (site, building, building block, zone) and the third column indicates whether or not the model geometry contained within the results is up-to-date. In this case, you would click on the **New** button to create a new project:

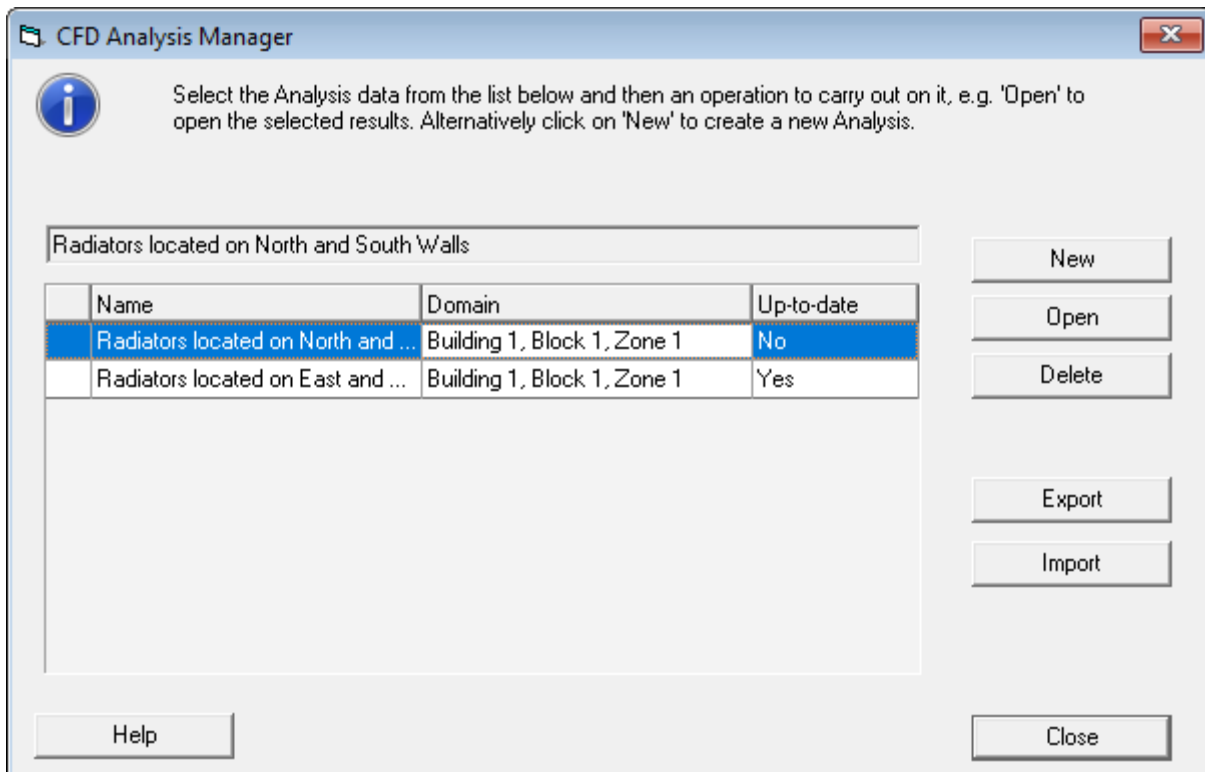


See the [Setting Up a New Internal CFD Analysis](#) section for details of creating a new internal analysis.

After creating the new analysis, you can then conduct the calculations:



You can then use the CFD Analysis Manager to open the previous project in order to compare results:



Importing And Exporting Results

Export

You can export a CFD analysis using the **Export** button on this dialog. The exported file might be imported later to a different version of the same model, perhaps on different computer on in a different file. CFD analysis files have extension .cfd.

Import

You can import a CFD analysis using the **Import** button on this dialog. The file for import will typically have previously been exported from a different version of the same model, perhaps on different computer on in a different file. CFD analysis files have extension .cfd.

Example - CFD External Analysis

This CFD example will go through the steps required for a typical external CFD analysis:

- Create model for external analysis.
- Switch to the CFD screen and create a new CFD analysis.
- Perform the CFD calculation.
- Review the results.

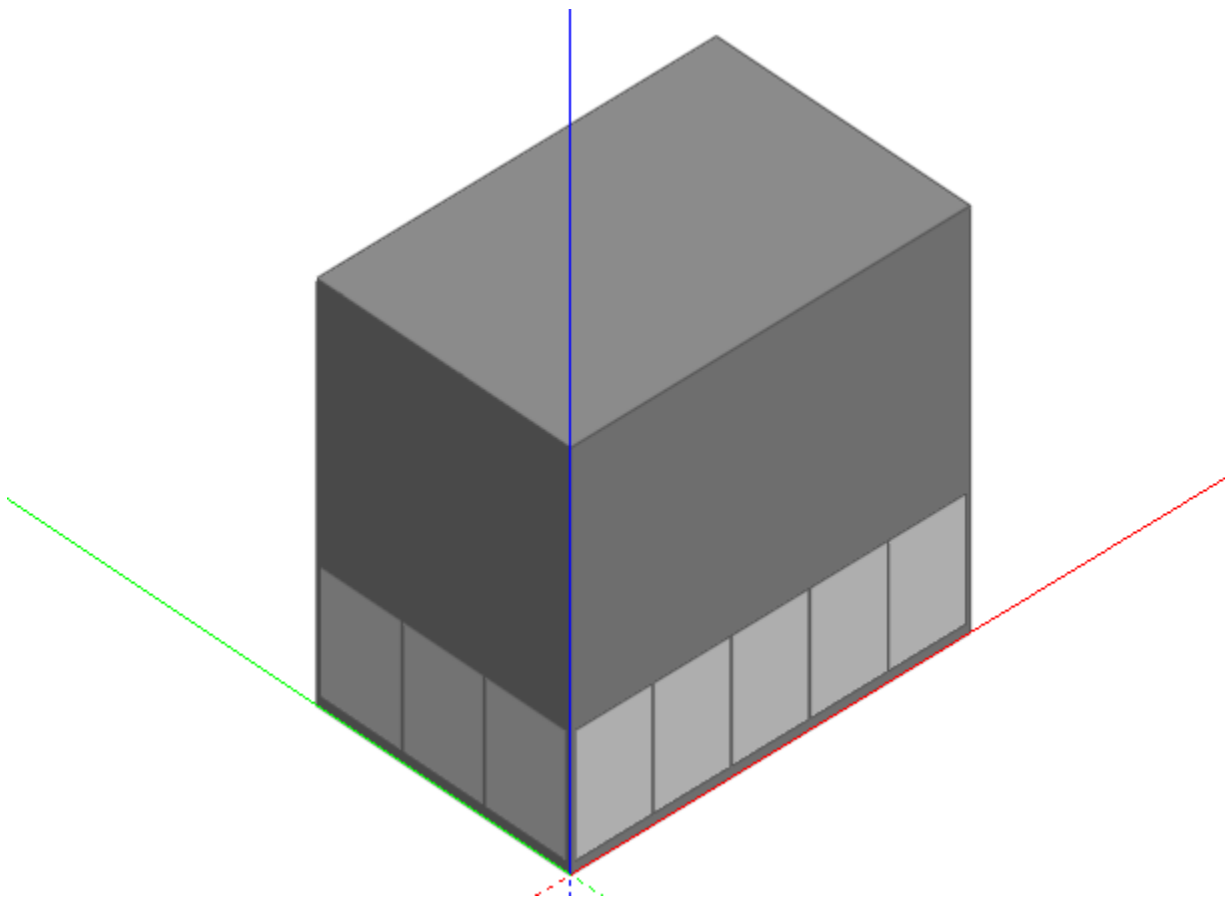
External analyses are simpler than internal analyses in that you do not need to define any internal boundary conditions. Also they are isothermal and take account principally of only velocity and pressure.

It is possible to [edit the grid](#) and [select monitor cells](#) with external CFD as well as with internal CFD.

Creating The Model

The model comprises a simple rectangular block with dimensions 30.0m x 20.0m x 30.0m.

Click on the Add new block tool and change the Block type to Building block and the Height to 30.0m. To place the origin of the building at the site origin, type in 0 0 and press the <Return> key. To then create the block with a 30.0m x 20.0m perimeter, type in 30 20 and press the <Return> key. A block will be created with the required dimensions. Alternatively create the block by drawing the base perimeter in the normal way.



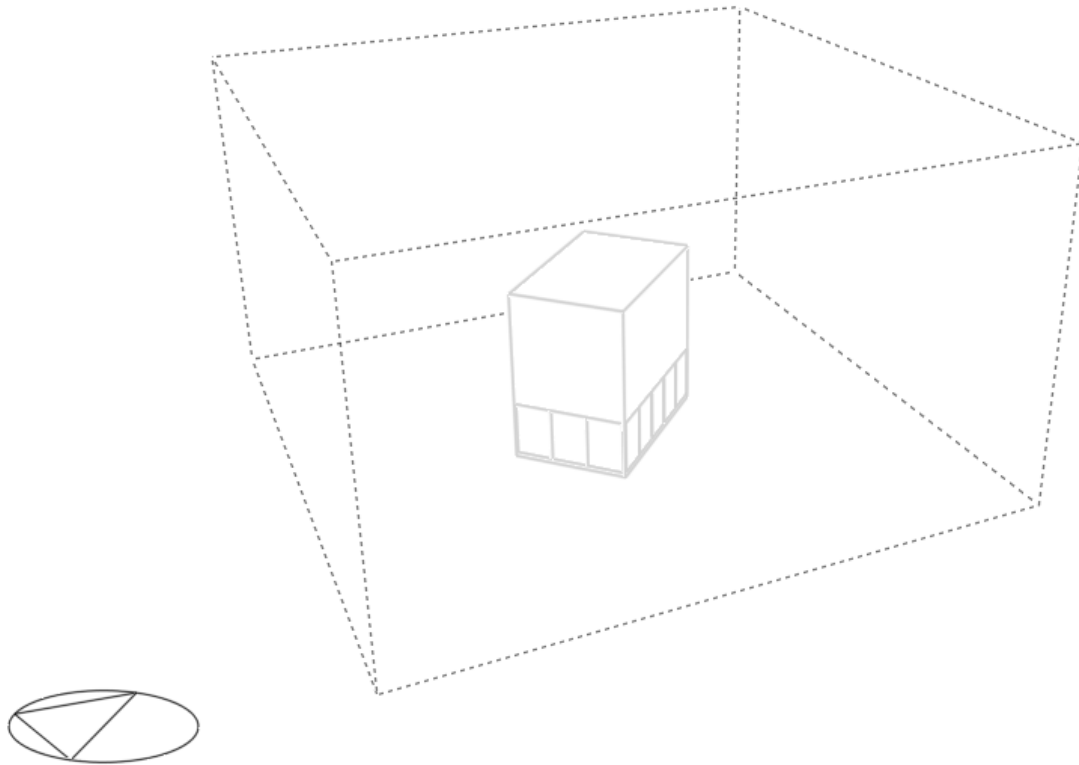
Creating A New CFD External Analysis

Each DesignBuilder project can have one or more CFD analyses. Each CFD Analysis can be a different set of results for a different variation on the model. Unlike simulation data, CFD Analyses are not deleted automatically when the model is modified on the Edit screen.

1. Click on the site in the navigator to move to the site level.

External CFD analyses can only be conducted at the site level.

2. Click on the CFD tab to set up an external CFD analysis.
3. After clicking on the CFD tab, the New CFD analysis dialog is displayed. Enter Wind analysis in the Name field and leave the grid spacing and grid line merge tolerance in their default settings.
4. Default grid spacing and grid line merge tolerance settings are covered later.
5. Under the Wind header, change the free-stream velocity to 12.0m/s, the Direction to 45° and the Exposure to Urban. The wind direction is defined clockwise from North, so the default of 270° represents a westerly and a setting of 45° a North-easterly.
6. The Site domain factors are multipliers that are applied to the overall model extent in the X (length), Y (width) and Z (height) dimensions to derive a site domain across which the calculations will be carried out. Keep the defaults of 3, 3 and 2.
7. After clicking on OK to create the analysis data set, the CFD screen displays the model in wire-frame together with the site domain, which is outlined in dotted lines.

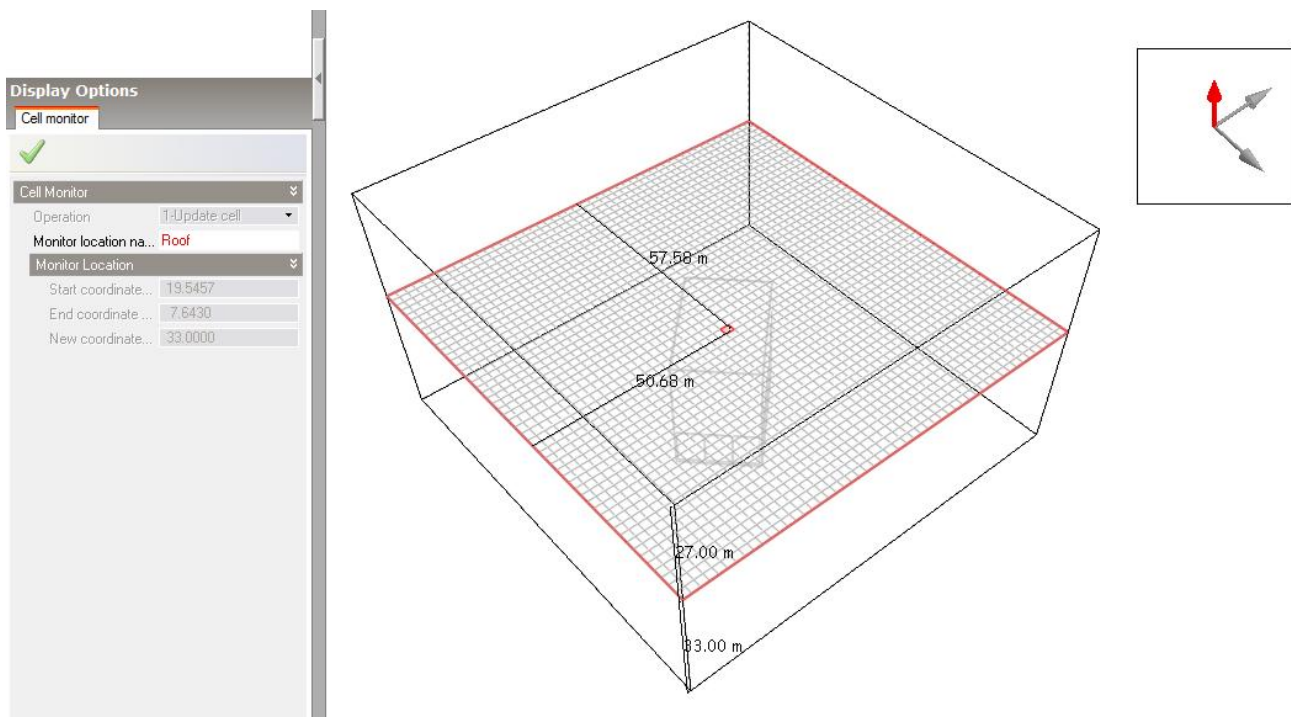


Notice the rotation of the domain with respect to the building to account for the wind direction.

Setting Up Monitor Cells

It is usually a good idea to display the state of one or more monitor cells in the domain so that you can 'monitor' the extent to which the calculations have settled down to a final converged solution. DesignBuilder automatically sets up a single monitor cell at the centre of the domain. You can add new monitor points at any time as follows.

Click on the Define CFD monitor points tool and select the Z grid axis using the grid axis selector tool. Move the cursor along the axis to select a plane a couple of grid cells above the building and then move the cursor across the selected plane and select a cell roughly above the centre of the building. The Cell monitor data panel is displayed at the bottom left of the screen. Enter the name Roof into the Monitor location name field and click on the green tick Apply changes button.



Press <Esc> to cancel the Define CFD monitor points tool.

Running The CFD Calculations

To set up calculation data and run the CFD calculations, click on the Update calculated data button, which will bring up the CFD Calculation options dialog. The left hand side of the calculation dialog contains various calculation options.

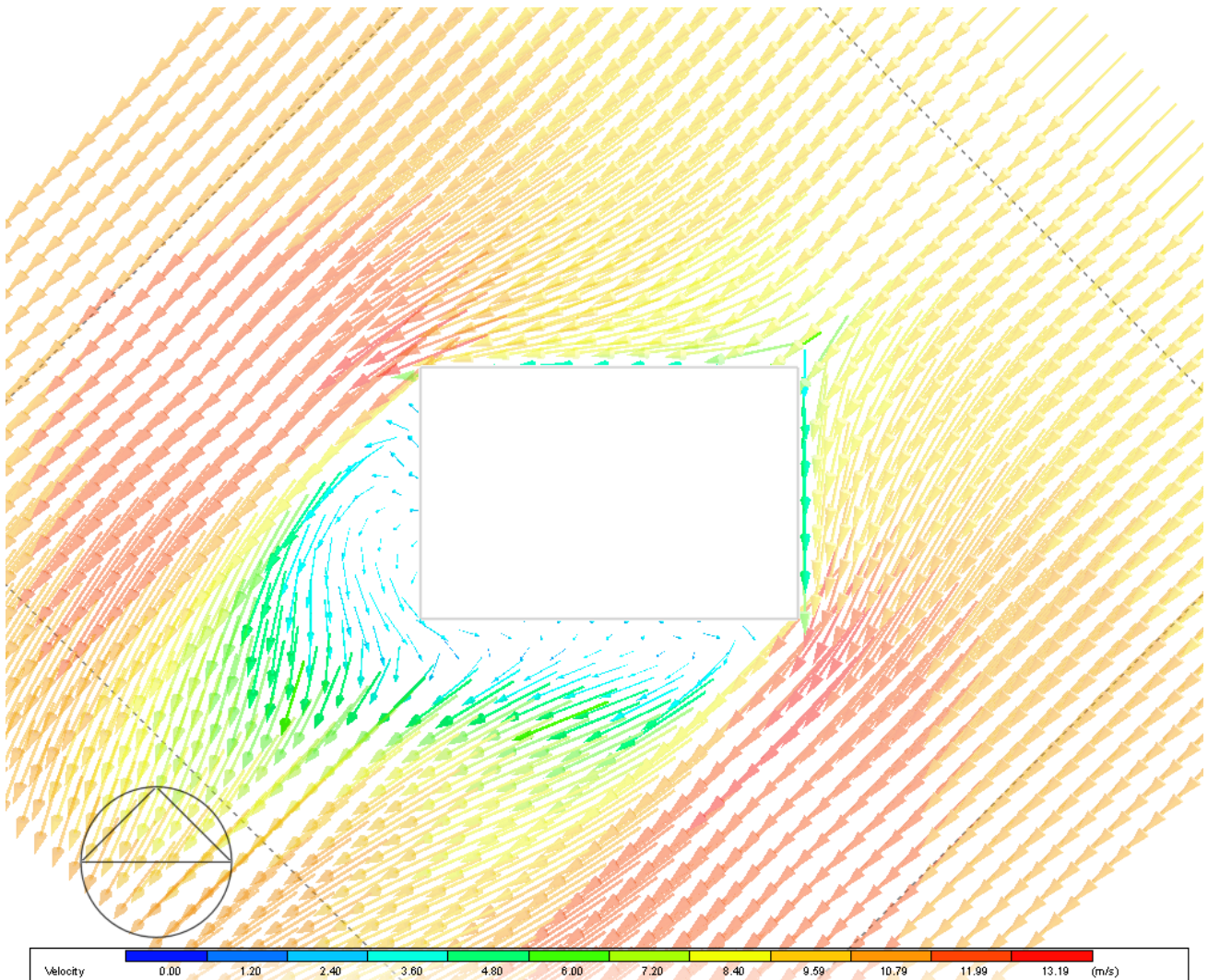
Keep the default options except for Iterations which should be set to 5000 and under the Residual Display header, switch on the X, Y and Z-axis velocity residuals.

Click on the Start button to begin the calculations. Wait until the calculation is fully converged when the residuals have reached the specified minimum and the “Finished (Converged)” indicator is displayed at the bottom left of the dialog. Notice that the displayed velocity for the supply monitor point becomes constant.

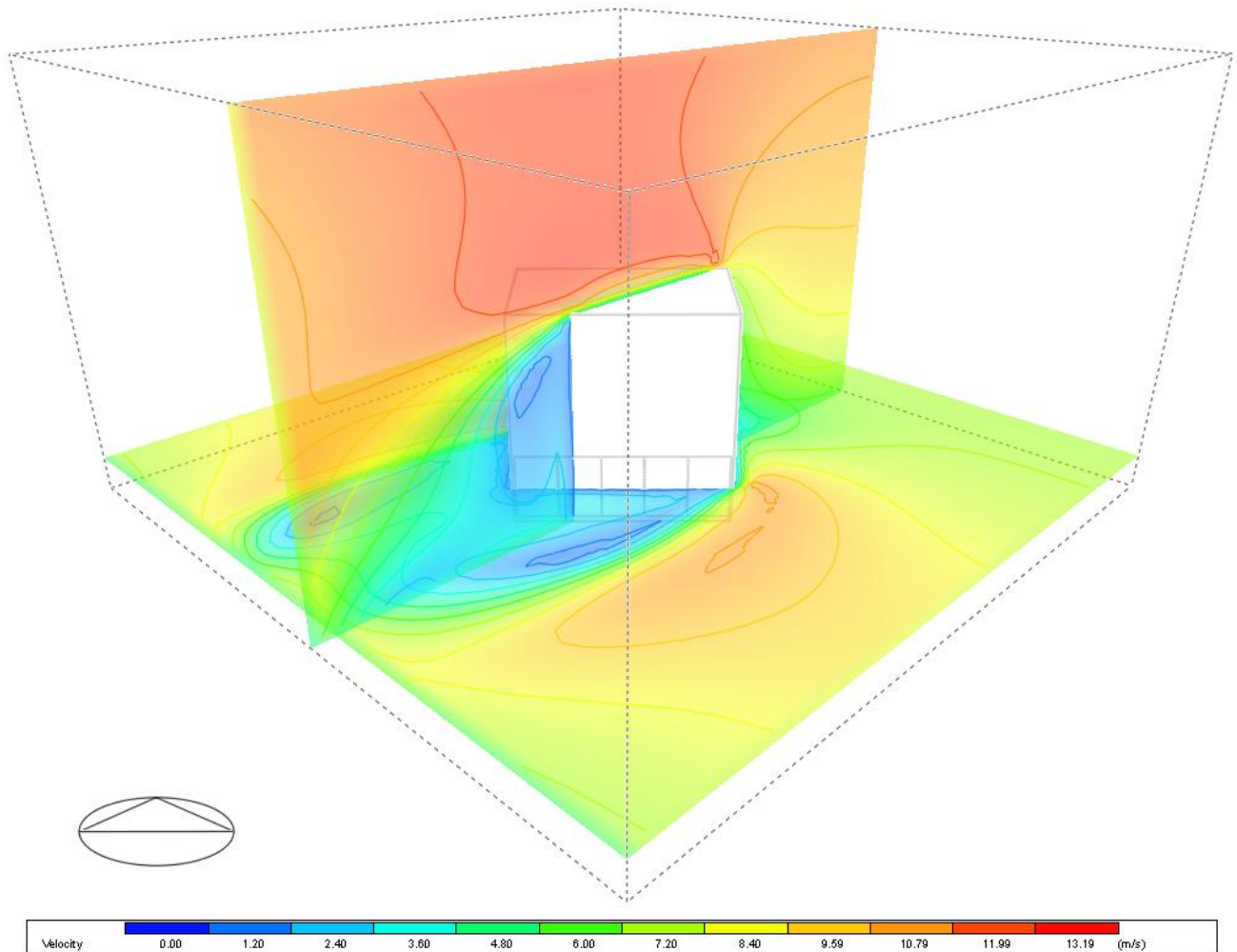
Press the Cancel button to close the calculation dialog.

Displaying CFD Results

1. Go to the Display Options panel and open the Velocity header under the Variable Settings header and set the Vector scale factor to 2.0 and the Maximum vector length to 5m.
2. Under the Slice Settings header, make sure that only Velocity vectors is switched on.
3. Click on the green tick Apply changes button.
4. Click on the Select CFD slice tool. Use the grid axis selector to change the current axis to the Z-axis and then move the mouse across one of the side surfaces of the site domain along the Z-axis until the slice selection frame is located 25m above the ground plane. Click on the mouse to add the currently selected slice to the display.



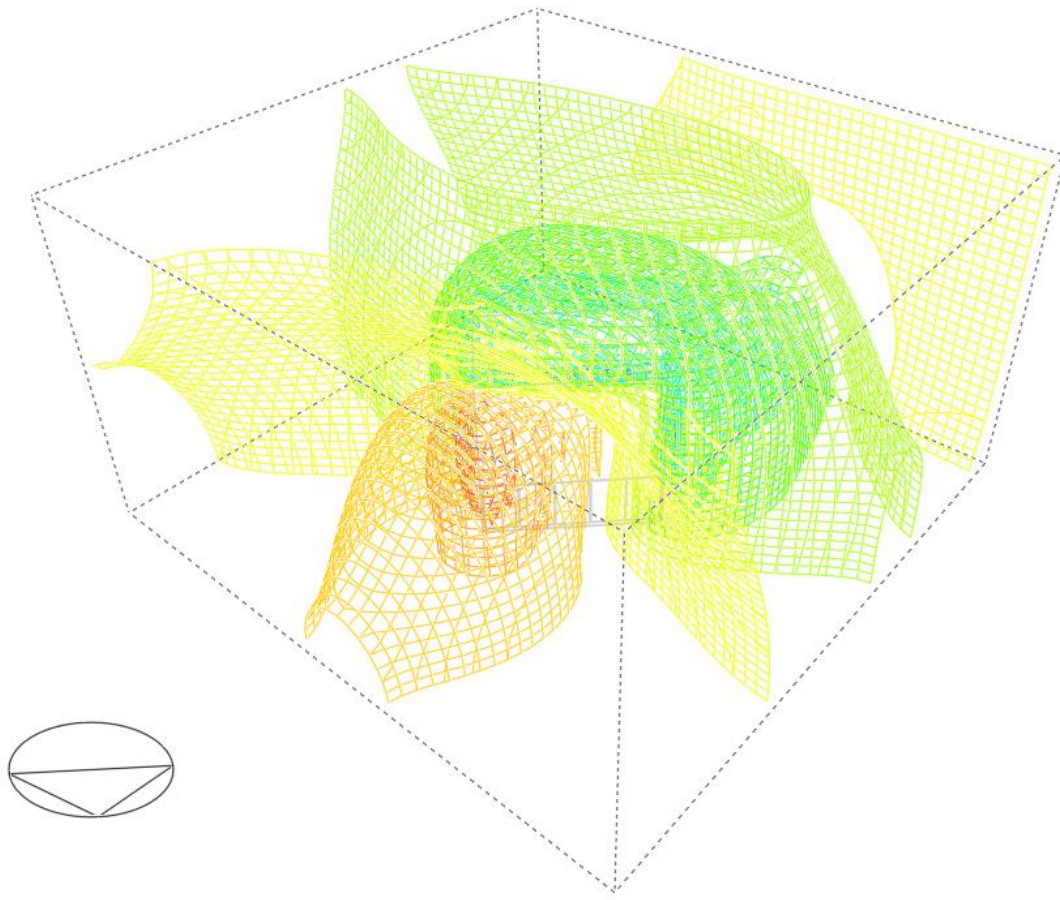
1. Clear the current slice from the display using the Clear all slices tool.
2. Switch off Velocity vectors under the Slice Settings header on the Display Options panel and switch on the Velocity contours and Filled Velocity contours.
3. Click on the green tick Apply changes button.
4. Select the Y-axis from the grid axis selector and move the cursor over the top surface of the site domain along the Y-axis until the slice selection frame is 50.68m from the origin and click the mouse to add the slice to the display.
5. To add a horizontal slice 5m from the ground, select the Z-axis from the grid axis selector and move the cursor over one of the side surfaces of the site domain along the Z-axis until the slice selection frame is 5.0m above the ground plane and click the mouse to add the slice to the display.



Display 3-D Contours

3-D contours can be displayed to show the values of a variable throughout the domain. 3-D contours can display a lot of data:

1. Clear the current slice from the display using the Tools > Clear all slices menu option.
2. Under the 3-D Contours header on the Display Options data panel, switch on Pressure. Click on the green tick Apply changes button.



Example - CFD Internal Analysis Basics

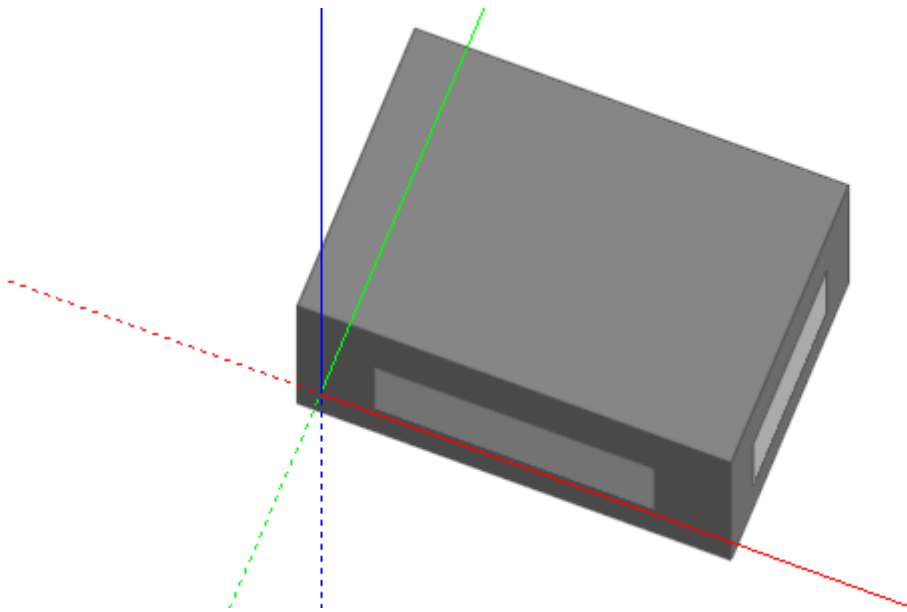
This internal CFD example will guide you through the basic minimum steps required for an internal CFD analysis:

- Create a simple single zone model.
- Add CFD surface boundaries to the model including a simple sidewall supply diffuser and ceiling mounted extract grille.
- Switch to the CFD screen and create a new CFD analysis.
- Perform a CFD calculation using default calculation options and grid.
- Review the results.

Creating The Model

The model comprises a simple rectangular block housing a zone with dimensions 6.0m x 4.0m x 3.5m.

Create a new site and add a new building using default settings. From the building level Click on the 'Add new block' tool and change the "Block type" to "Building block", the "Height" to 3.5m and set the wall thickness to 0.25m.



Default Wall And Window Boundary Temperatures

Default wall and window boundary temperatures can be defined from the building level down to the surface level. As with other DesignBuilder attribute data, these attributes are inherited from the level above unless deliberately overwritten.

The default wall and window boundary temperatures can be accessed under the **CFD Boundary** header on the CFD tab of the model data.

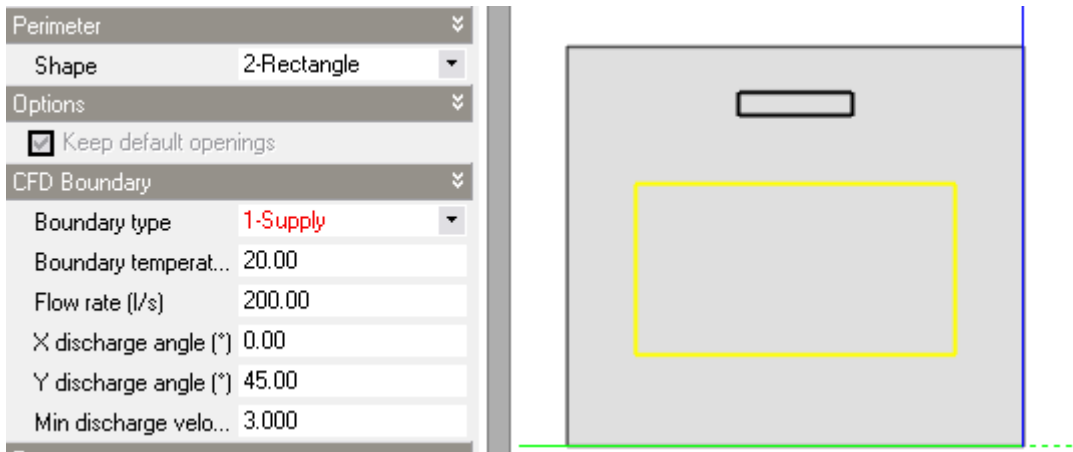
At the building level, change the default window temperature to 20°C.

Go down to the zone surface facing North and change the default wall temperature to 18°C and the default window temperature to 10°C. Go through the same procedure for the west facing zone surface.

Adding Surface Supply Diffuser And Extract Grille

Supply grille

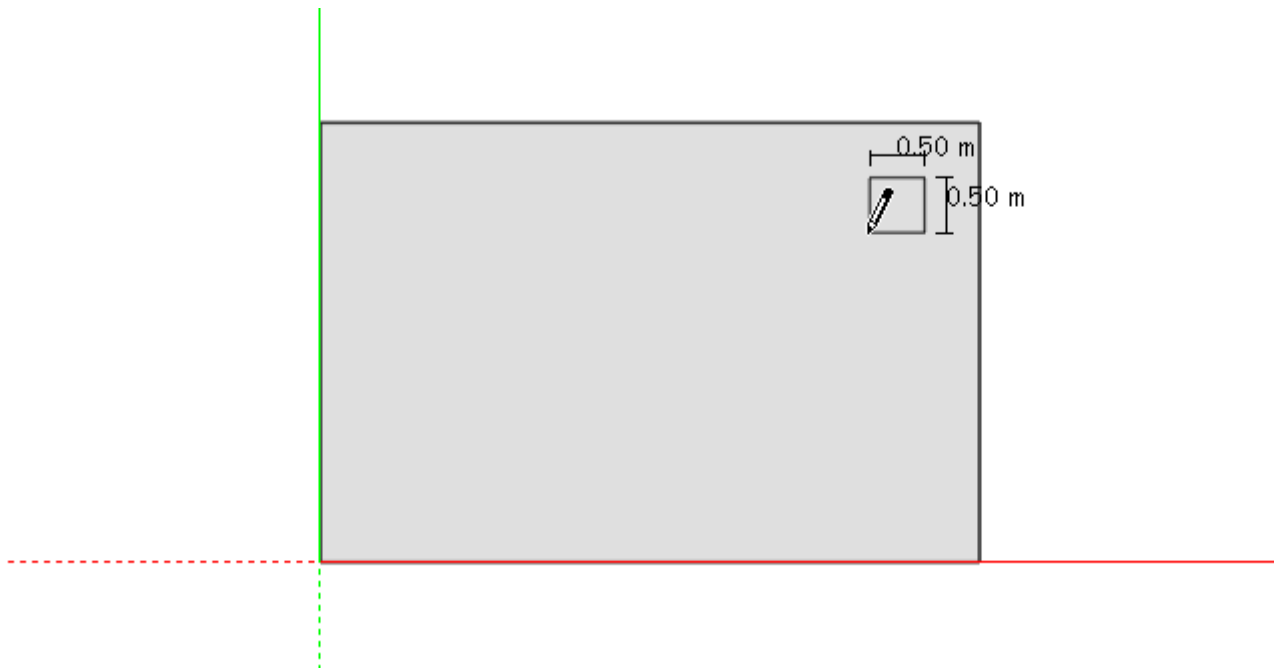
1. Click on the zone in the navigator and select the North-facing surface in the Edit screen. Double-click to drop down to the surface level.
2. Click on the Normal view button to obtain a normal view of the surface.
3. Click on the Add CFD boundary tool. The CFD boundary data panel is displayed at the bottom left of the screen.
4. Select Supply from the Boundary type drop list. Keep all of the remaining boundary defaults but change the Y discharge angle to 45° in order that the supply jet will be projected upwards at an angle of 45° from the horizontal.
5. The supply diffuser is to be 1.0m x 0.2m and is to be located midway between the North and South surface edges, 0.4m from the top edge of the surface. With the Increment snap switched on, move the cursor to the top left corner of the surface and press <SHIFT> to lock the origin to the surface corner, move the cursor away from the corner until the cursor is snapped 0.4m from the top edge and 1.5m from the left edge and click the mouse button to place the top left corner of the boundary. Move the cursor to place the bottom right corner of the boundary 1.0m to the right and 0.2m below the top left corner and click the mouse to place the supply.



Extract grille

A 0.5m x 0.5m extract grille is to be located in the corner of the ceiling 0.5 m from the North and east surface edges.

1. Click on the zone roof surface in the navigator and again click on the Normal view button to obtain a normal view.
2. Click on the Add CFD boundary tool and select Extract from the Boundary type drop list. Keep the default flow rate of 200 l/s to match the supply flow rate.
3. Move the cursor until it is snapped 0.5m from the top edge of the surface and 0.5m from the right edge and click the mouse to place the top right corner of the extract, move the mouse 0.5m to the left and 0.5m below the top right corner of the extract and click the mouse to place it.



Creating A New CFD Internal Analysis

Click on the building in the navigator to move to the building level and click on the CFD tab to set up an internal CFD analysis. An internal analysis can be set up for any level from a single zone up to the whole building. In this case, because we are dealing with a single zone building, the analyses for each level would be identical. After clicking on the CFD tab, the New CFD analysis dialog is displayed. Enter Basic Vent in the Name field and leave the grid spacing and grid line merge tolerance at their default settings. More on grids in the next exercise.

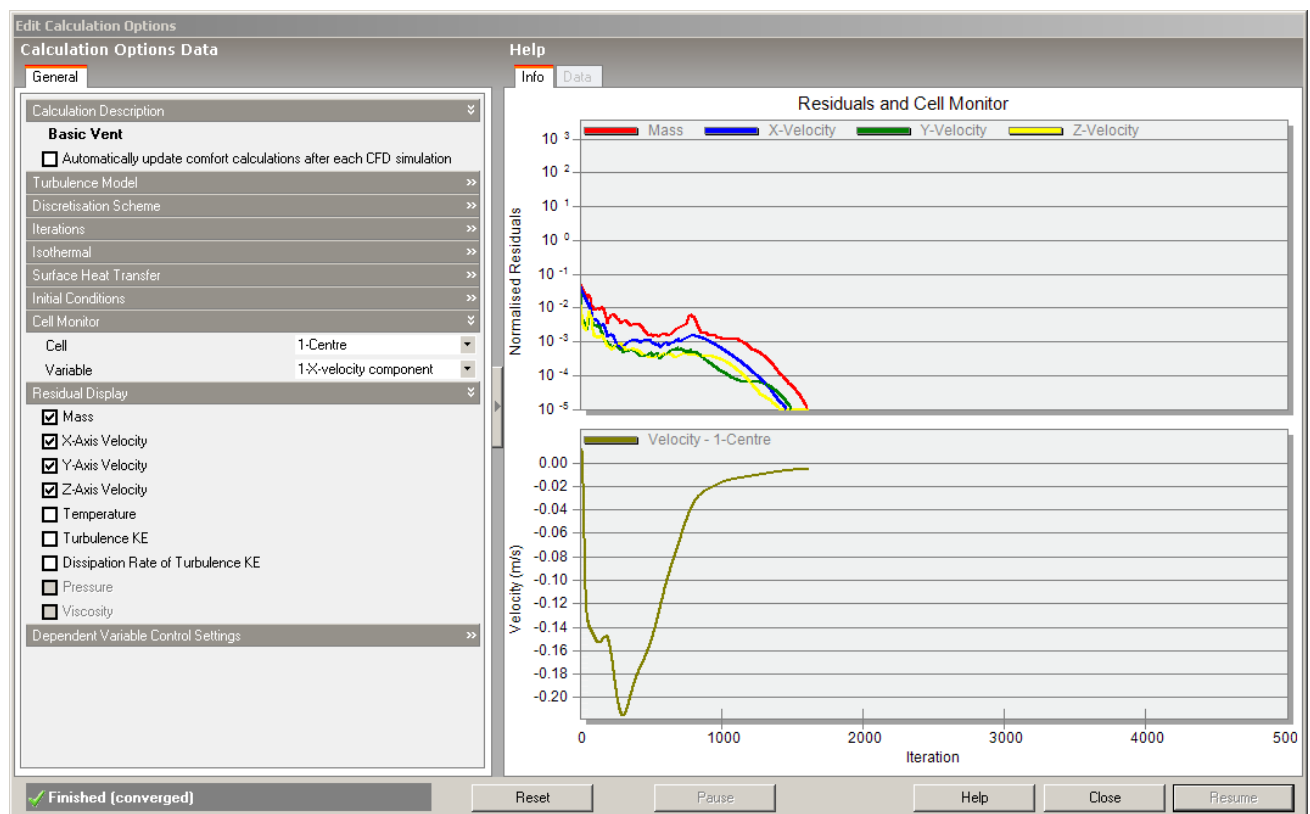
Click OK to create the analysis data set.

You will see the model in wire-frame mode by default and at this stage the CFD analysis has been created including the default grid and the model is ready for calculations. If you were going to edit the grid now is the time – before starting the calculations, but for now we will use the default grid.

Running The CFD Calculations

To set up calculation data and run the CFD calculations, click on the “Update calculated data” button, which will bring up the CFD Calculation options dialog. The left hand side of the calculation dialog contains various calculation options. Keep the default options except under the Residual Display header, switch on the X, Y and Z-axis velocity residuals.

Click on the Start button to begin the calculations. Wait until the calculation is fully converged when the residuals have reached the specified minimum (1×10^{-5} is the default for all variables) and the Finished (Converged) indicator is displayed at the bottom left of the dialog. Notice that the displayed velocity for the default Centre monitor point becomes constant (bottom graph).

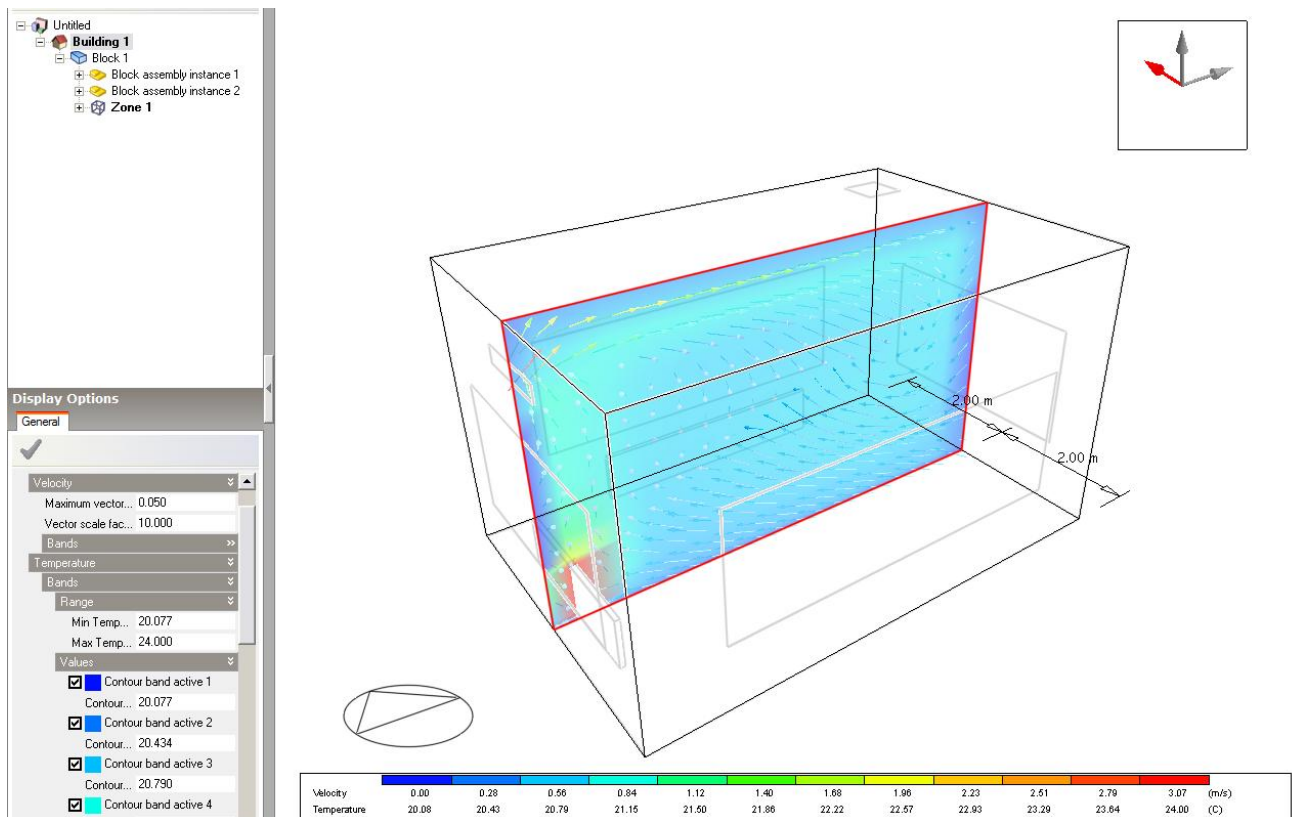


Press the Cancel button to close the calculation dialog.

Displaying CFD Results

After the calculations have been completed, the CFD results display options data panel is enabled allowing you to set various results display options. Use default Display options.

Click on the Select CFD slice tool. Click on the Y-axis of the grid axis selector (top right of screen) and then move the cursor over the ceiling surface along the Y-axis to move the slice selection frame midway between the North and South-facing walls, 2.0m from each wall and click the mouse button to add the currently selected slice to the display.



Notice that as you move the slice selection frame to a slice that was previously added to the display, the frame colour changes from green to red and if you click the mouse button, the slice is removed from the display.

Try adding more slices in other planes to see the key characteristics of temperature and velocity distribution in the zone.

Press the ESC key to cancel Select CFD slice tool.

Example - CFD Internal Analysis Including Assemblies

In this second internal CFD exercise, the model is modified to include radiators as well as the supply and extract vents entered in the first tutorial. Also some more advanced features of the CFD calculations and display are explored. You will learn how to:

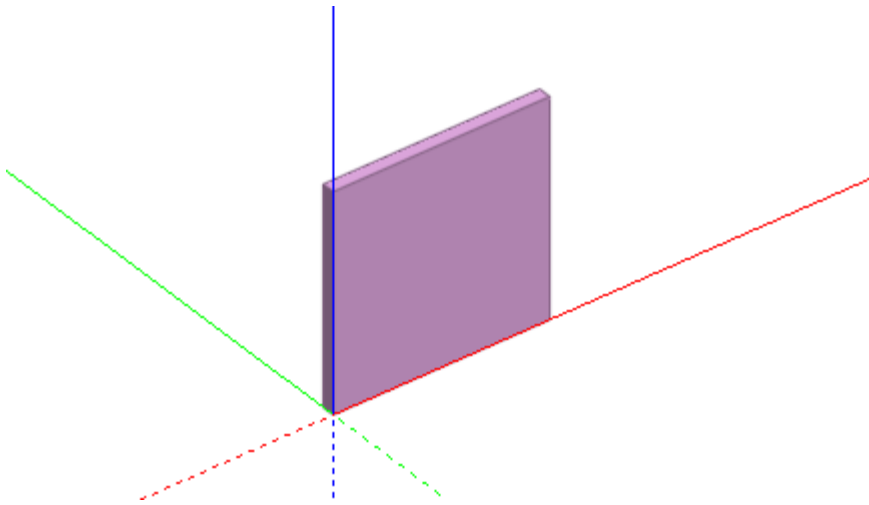
- Create a component representing a constant temperature radiator, convert the component to an assembly, assign CFD boundary conditions to the assembly and add the assembly to the model assembly library.
- Place the previously created radiator assembly within the zone.
- Edit the CFD grid.
- Add cell monitor points for calculation monitoring purposes.
- Edit the CFD Calculation options.
- Deal with common errors and non-convergence.
- Perform comfort calculations.

Creating The Radiator Assembly

Assemblies consist of 1 or more component blocks combined and are used to create boundary conditions inside or outside blocks. They can be used to create furniture, heat generating equipment, airflow supply and extract etc.

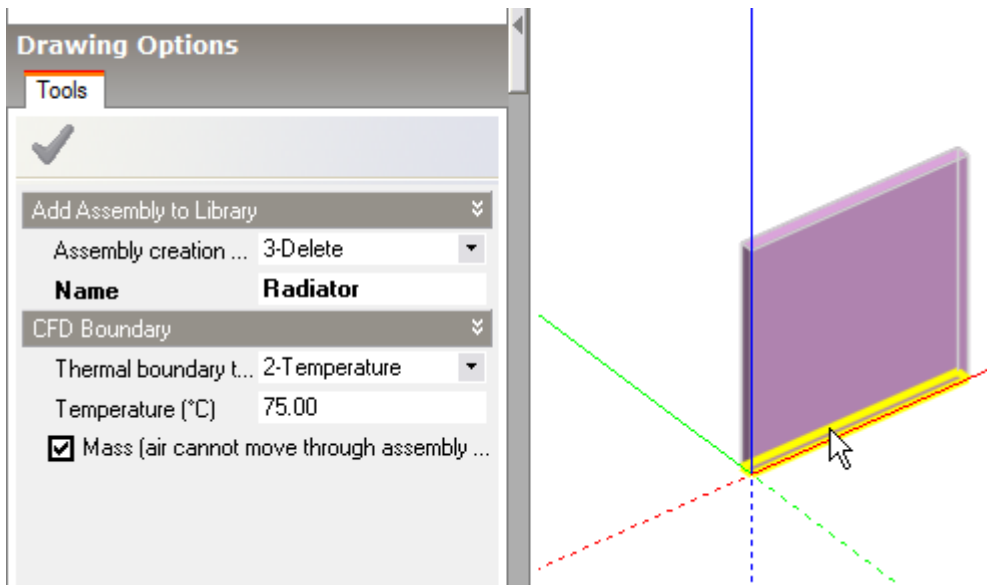
In this exercise a radiator assembly is to be created from a single component block, which is to be drawn using a rectangular extrusion.

1. Click on the Add new block tool and change the Block type on the Drawing Options panel to Component block and the Height to 1.0m.
2. To place the origin of the radiator component block at the site origin, type in 0 0 and press the return key.
3. Now to create the radiator component with a length of 1.0m and say a thickness of 60mm to represent a single panel convector type radiator, type in 1.0 0.06. The component block will then appear with the specified dimensions. Alternatively use the mouse to define the component block dimensions in the usual way.



To convert the component block to an assembly and add the resulting assembly to the model assembly library:

1. First select the component block and then click on the Add assembly to library tool. The selected component block will be displayed with a wire frame bounding-box and the Add assembly to library data panel is displayed at the bottom left of the screen.
2. Set the Assembly creation mode to Delete because the original component block will not be needed after the conversion and type the name Radiator into the Name field.
3. Set the Thermal boundary type to Temperature in order to create a constant temperature radiator and set the Temperature to 75.0°C.
4. Make sure that the Mass check box is checked to ensure that the assembly will represent a solid boundary and that air will not be able to pass through it.
5. The next stage is to select the attachment plane for the assembly, which is the surface plane of the assembly that will be used when attaching the assembly to a model surface during the assembly placement process. In this case, the base (bottom) surface of the radiator should be selected as the attachment plane. To select the attachment plane, move the cursor over the radiator assembly and as you do so, each surface of the assembly bounding-box will highlight in yellow, move the cursor to the base surface of the bounding-box and click the mouse. The assembly will then be automatically added to the model assembly library and because the Delete creation mode has been selected, the original component will be deleted.



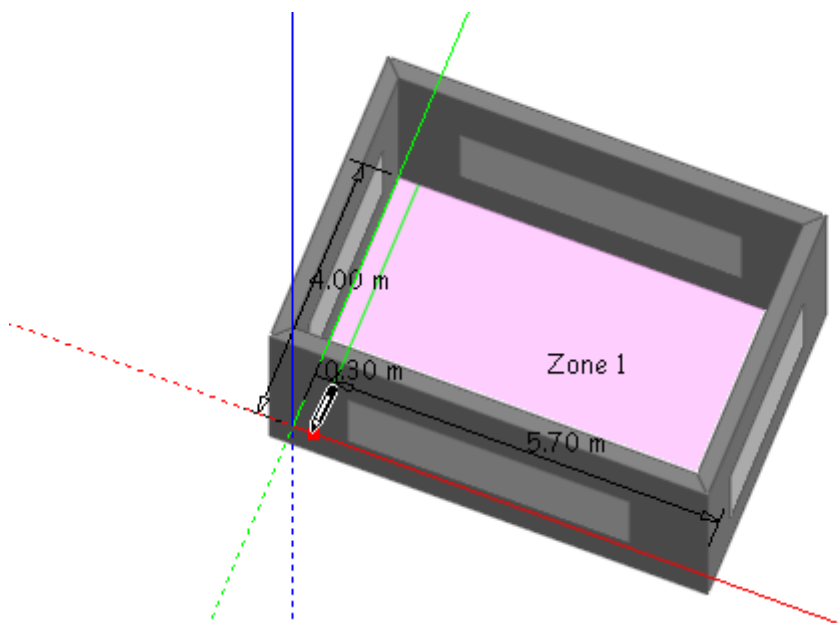
Positioning Radiators Within The Block

Assemblies inside the building must be placed at Block level (not zone level) so to place a radiator assembly first click on the block in the navigator to move down to the block level.

To position assemblies accurately you must use [construction lines](#).

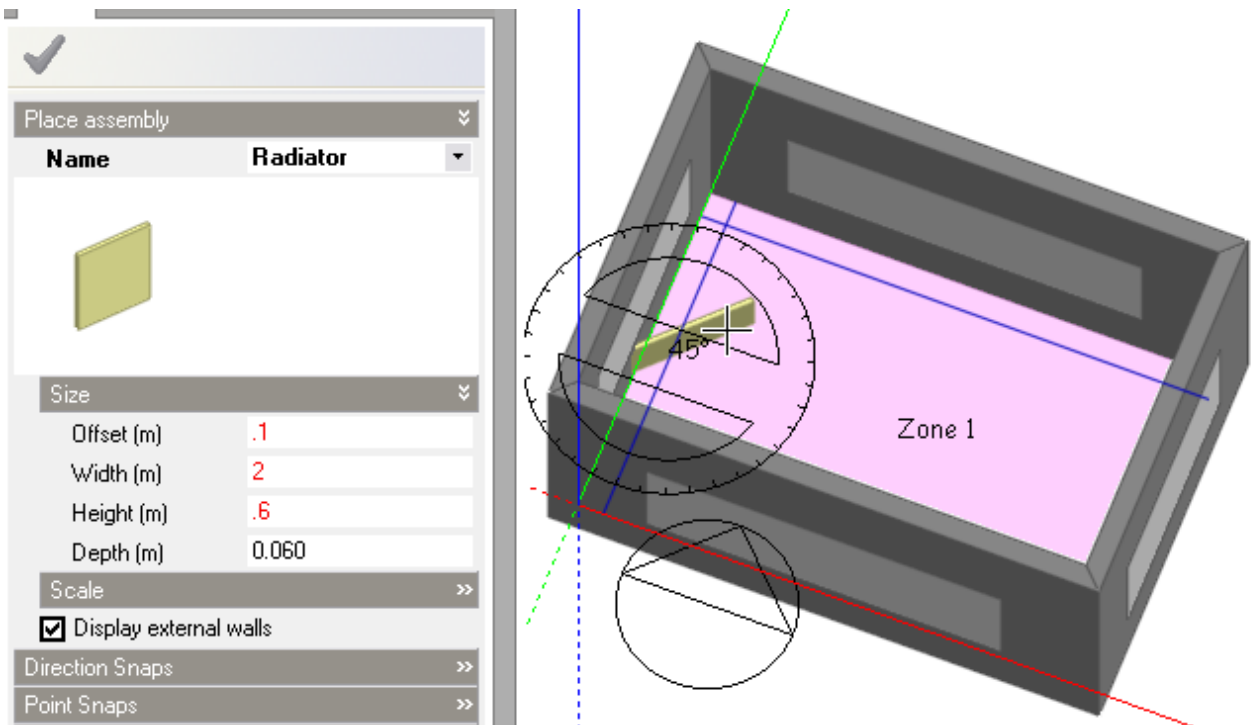
To do this, orbit the model to get a clear view of the west-facing end wall from the inside of the block and then click on the Place construction line tool.

To position the construction line accurately, switch on the Increment snap under Point Snaps on the Drawing Options data panel. Move the cursor to snap to the lower surface edge at the bottom of the North-facing wall and move the cursor to obtain an edge snap 0.3m from the end nearest the west-facing wall. Click the mouse to start the construction line and move the mouse to the opposite edge at the bottom of the South-facing wall, snapped to the Y-axis and click the mouse once the edge snap is established on the opposite edge to place the construction line.

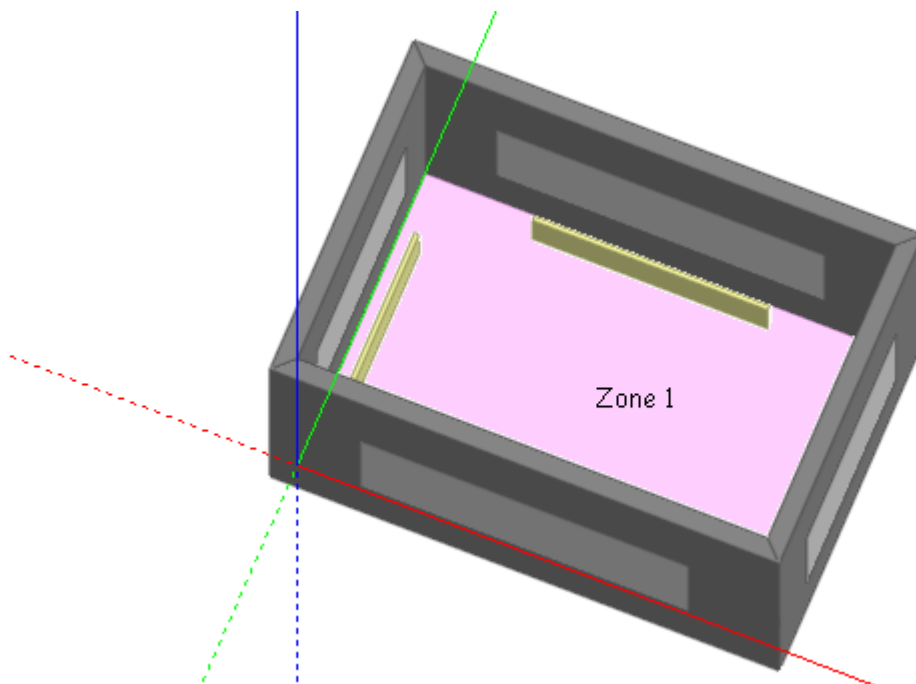


Click on the Place assembly tool and as you move the cursor back across the Edit screen, you will see that a copy of the previously created radiator assembly is attached to the cursor and the Place assembly data panel is displayed at the bottom left of the screen. Set the Offset to 0.1m to offset the base of the radiator 0.1m above the floor level, set the Width to 2.0m, the height to 0.6m and leave the Depth at 0.06m. Move the

cursor to establish a snap on the construction line and click the mouse to place the radiator origin. Once the radiator has been placed, a protractor is displayed at the assembly origin enabling you to define the rotation. Rotate the radiator assembly until it is parallel with the nearest wall and then click the mouse to set the rotation.

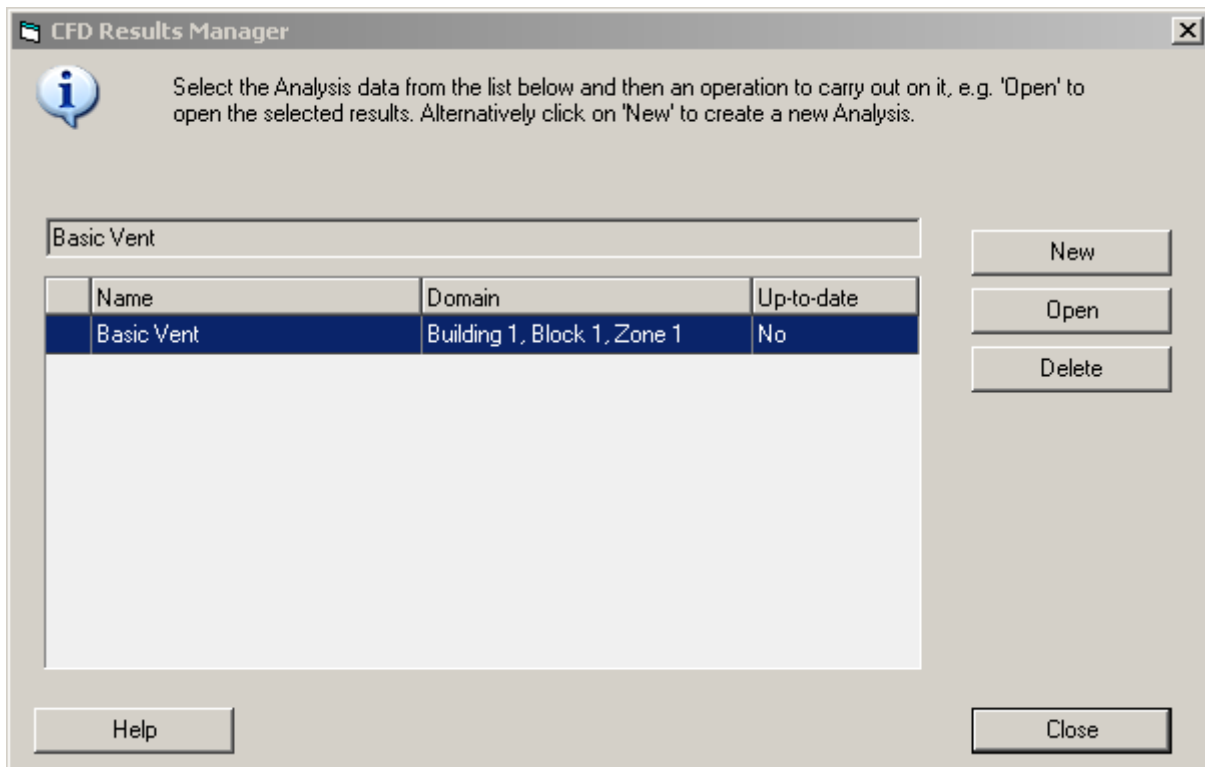


Repeat this procedure to place a 3.0m radiator adjacent to the North-facing wall.



Creating A New CFD Internal Analysis

We are now ready to start a new CFD calculation so again click on the building in the navigator to move to the building level and click on the CFD tab. If you have just finished the first CFD exercise, the analysis results for these calculations will be displayed. You should now create a new analysis because the model has changed with the addition of the radiators. To do this click on the CFD Results Manager toolbar icon. This displays the dialog below.



Note that the first Basic Vent analysis is flagged as not up-to-date. This is because the model has been edited since the last calculation was made.

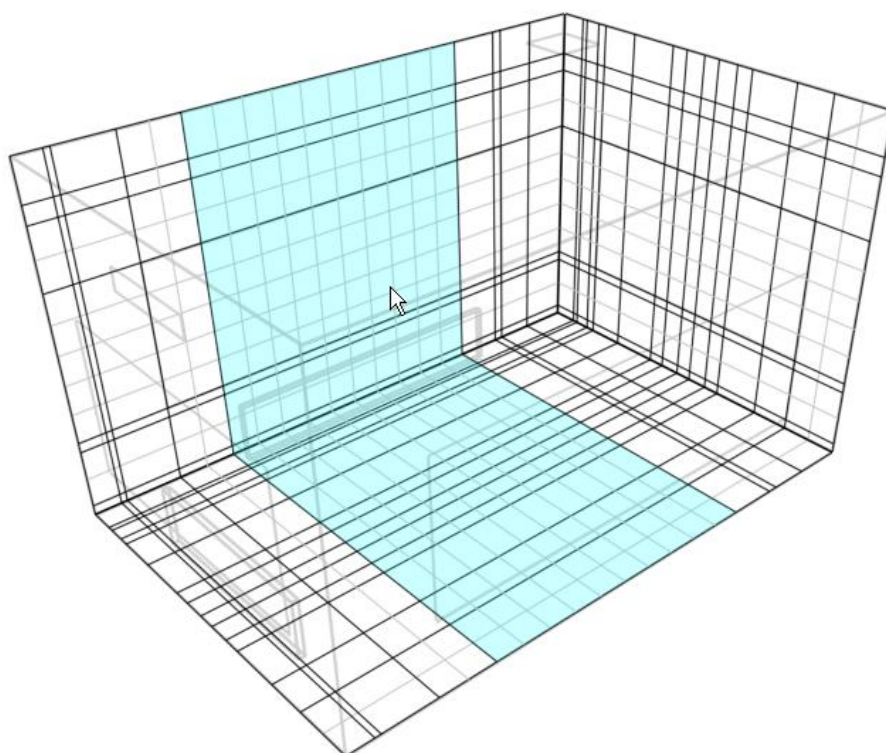
Click on the New button to create a new analysis using the model in its current state. The New CFD analysis dialog is displayed. Enter Radiators and Vent in the Name field and leave the default grid spacing and grid line merge tolerance at their default settings. The default grid spacing and grid line merge tolerance are used in generating the CFD grid. During the process of creating a new CFD analysis data set, a 3-D CFD grid is automatically generated by extracting key points from the model geometry along each of the major axes. The spaces between the key points are known as grid regions and are automatically divided into sub-regions using the default grid spacing entered on the Calculation options dialog. The grid line merge tolerance enables key point grid lines that are very close together to be merged in order to prevent very high aspect ratio cells from being created which can lead to calculation instability.

Click OK to create the analysis data set.

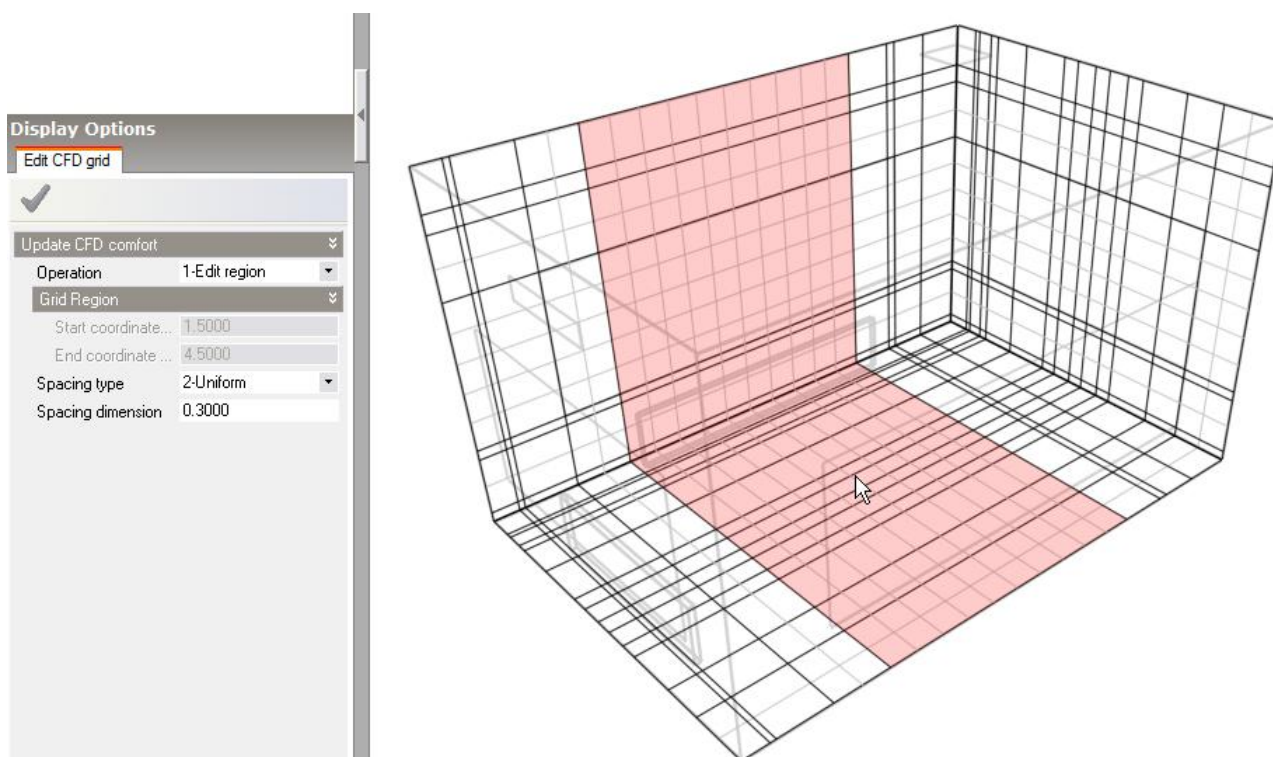
You will see the model in wire-frame mode by default and at this stage the CFD analysis has been created including the default grid and the model is ready for calculations.

Editing The CFD Grid

But before starting the calculations we will explore the edit grid capability. First click on the Edit CFD grid tool. Move the cursor across the displayed CFD grid to select grid regions along the current grid axis. Selected grid regions are highlighted in cyan.



To change the current grid axis, click on one of the major axes in the grid axis selector tool that is displayed at the top right of the CFD screen. To edit a selected region, click the mouse on the currently selected region. The selected region will turn red and the Edit CFD grid region data panel will be displayed at the bottom left of the screen. You can select Edit region from the Operation drop list to change the default spacing of the region or Insert region to create a new region and in the case of a previously inserted region, you can select Remove region to remove it.



To cancel the Edit CFD grid tool press the <Esc> key.

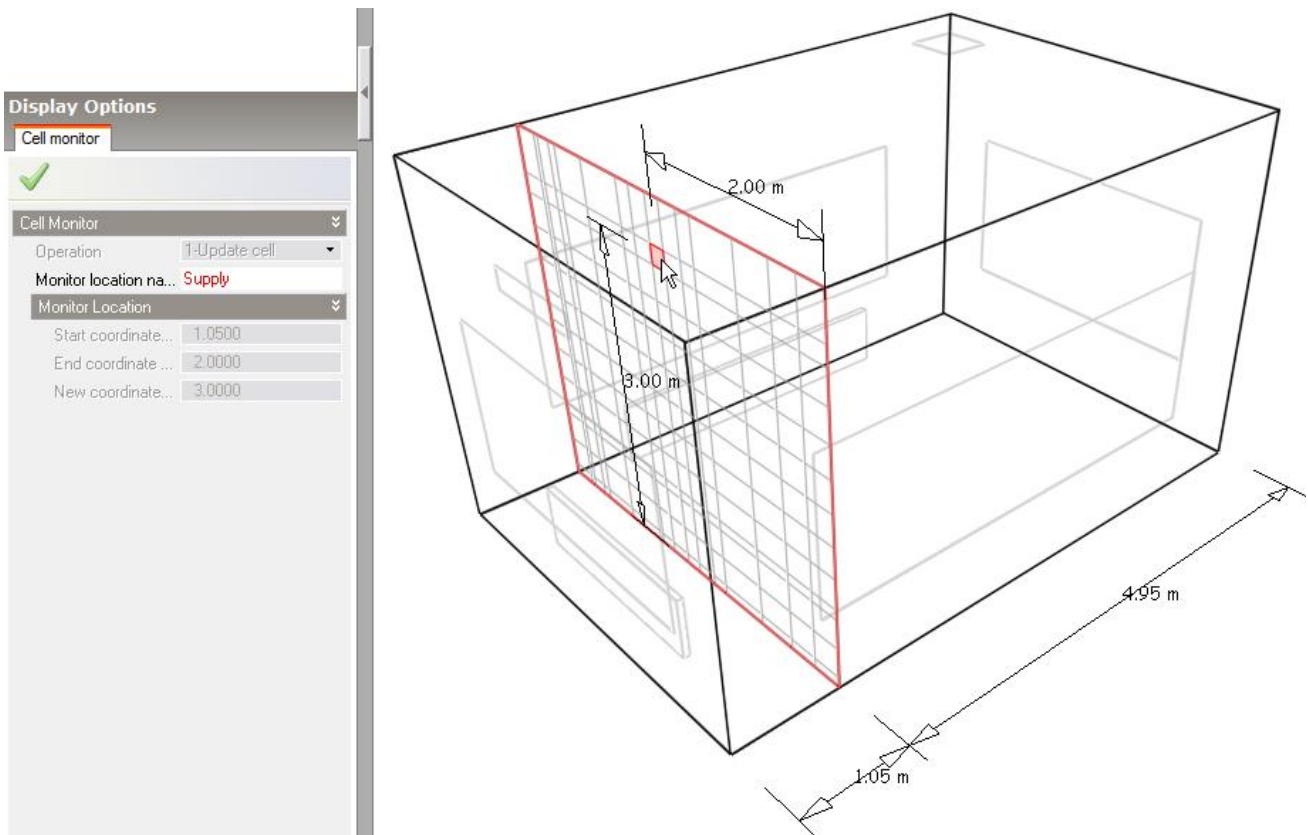
After editing the default grid, you can reset it to the default by clicking on the Reset CFD grid tool. Also, to obtain information about the grid, you can click on the Show CFD grid statistics tool which brings up a dialog indicating the number of cells, required memory, available memory and maximum cell aspect ratio.

For the purpose of this tutorial, the default grid will be used.

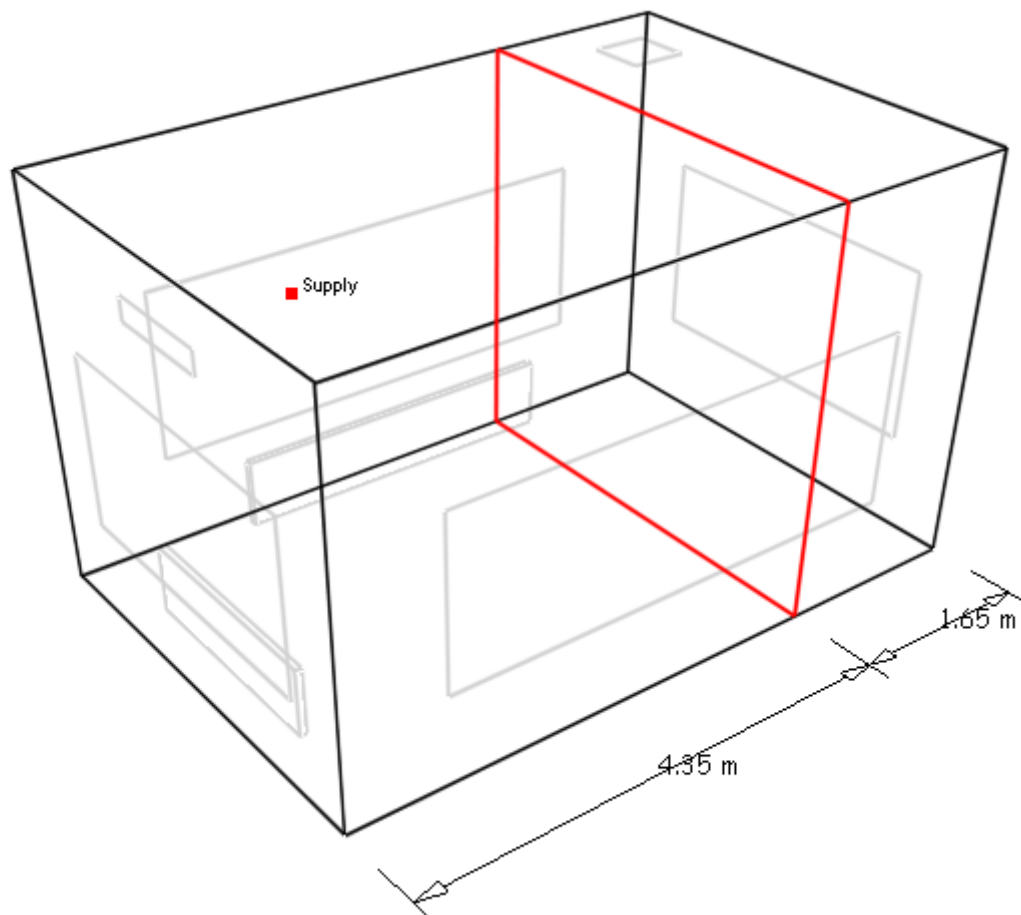
Setting Up Monitor Cells

In order to monitor the convergence of the solution, it can be useful to set up a number of monitor points throughout the calculation domain. During the calculation it is then possible to enable a plot of the variation of the various dependent variables for each of the defined monitor points in order to see if the variables have reached a constant value as the CFD calculation proceed.

1. To set up a monitor point, click on the Define CFD monitor points tool.
2. Move the cursor along the current grid axis to select the plane in which the monitor cell is to be selected. You can change the current grid axis using the grid axis selector tool as before.
3. Once the plane has been selected, you can move the cursor to select the required cell in that plane. With the current grid axis set to the default X axis, move the cursor along the axis to select a plane about 1.0m from the supply diffuser and then move the cursor across the selected plane and select a cell directly in front of the diffuser.
4. The Cell monitor data panel is displayed at the bottom left of the screen. Enter the name Supply in the Monitor location name field and click on the green tick Apply changes button



When a monitor cell has been added, a cell marker is displayed together with the cell name.



You can press <Esc> to cancel the Define CFD monitor points tool.

Running The CFD Calculations

To set up calculation data and run the CFD calculations, click on the Update calculated data button, which will bring up the CFD Calculation options dialog. The left hand side of the calculation dialog contains various calculation options. Keep the default options except under the Residual Display header, switch on the X, Y and Z-axis velocity residuals.

Click on the Start button to begin the calculations. Wait until the calculation is fully converged when the residuals have reached the specified minimum and the “Finished (Converged)” indicator is displayed at the bottom left of the dialog. Notice that the displayed velocity for the default Centre monitor point becomes constant (bottom graph).

CFD Display Options

In the CFD results slices, velocity vectors are displayed as arrows, the length of which corresponds to the magnitude of the velocity and with the default vector scale factor of 1.0, a length of 1.0m corresponds to a magnitude of 1.0m/s. The maximum vector length is the maximum length of a vector that will be displayed to prevent the display from becoming cluttered with excessively large vectors. Any vector, which has a magnitude greater than this maximum, will be displayed translucently to distinguish it from vectors with lengths that do represent magnitude.

The default minimum and maximum dependent variable values are extracted from the minimum and maximum values found in the domain and this range is then divided into twelve equal increments in order to arrive at the dependent variable contour bands. Each contour band can be edited or switched off altogether.

In this tutorial the maximum temperature will be 75.0°C (the temperature of the radiators), which is well above the mean temperature of the space, and consequently the default contour bands would be far from the mean and the resulting display would not be very informative. So in order to compress the contour bands to be

within the likely temperature range throughout the domain DesignBuilder automatically sets the Maximum Temperature under temperature bands range to 24.0°C and the minimum value to 18°C.

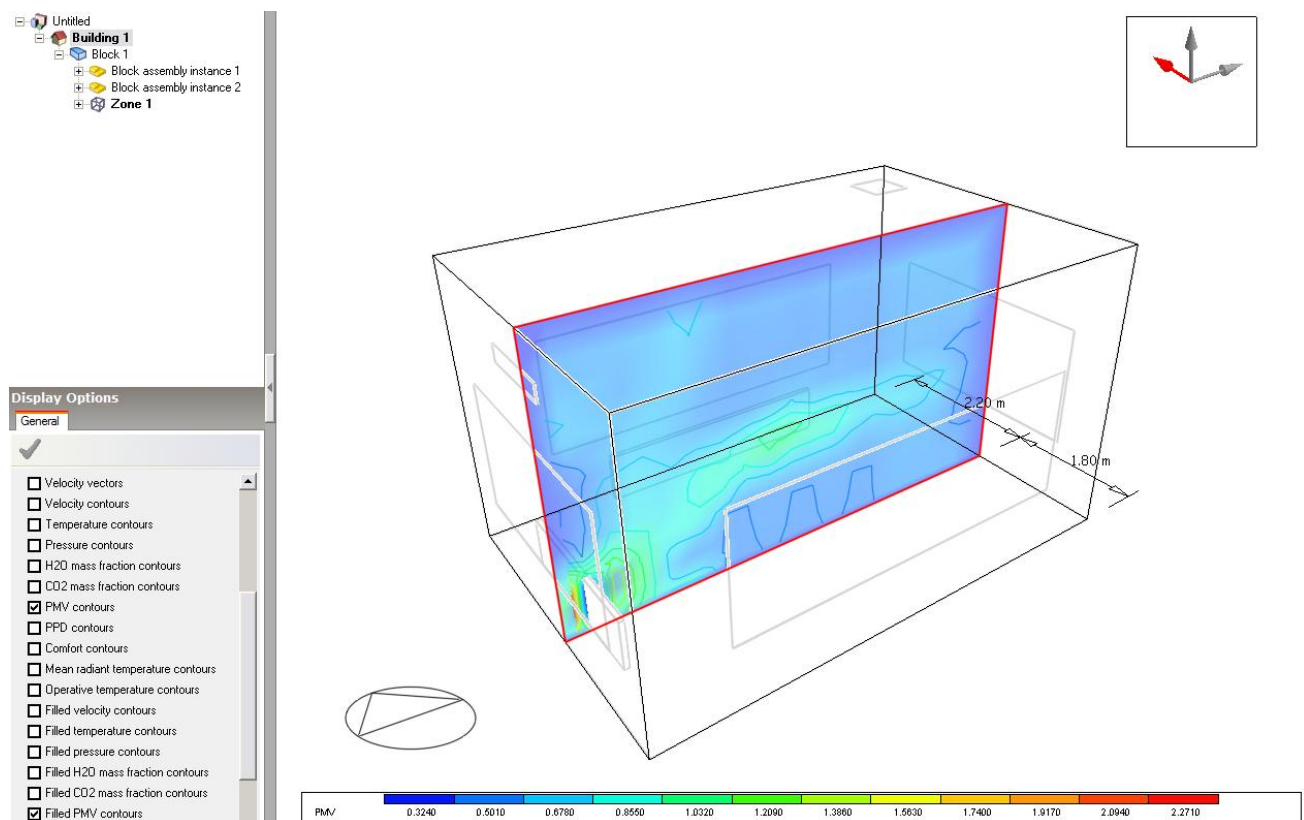
You can manually change these maximum and minimum values in the Display options panel. Note that you need to click on the green tick Apply changes button in order for the display attributes to take effect. Notice also that the temperature contour bands are updated using the new temperature range.

You can revert to the absolute minimum and maximum values in the model by using the Tools > Reset CFD display variables menu command.

Comfort Calculations

Click on the **Update CFD comfort** tool to carry out comfort calculations. After the calculations have been completed, various comfort variables are available for display, PMV, PPD, operative temperature and mean radiant temperature. Clear all slices from the display using the Clear all slices tool. Switch off Velocity vectors and Filled temperature contours under the Slice Settings header on the Display options panel and switch on the PMV contours and Filled PMV contours. Click on the green tick Apply changes button.

Click on the **Select CFD slice** tool and click on the Y-axis of the grid axis selector. Move the cursor over the ceiling surface along the Y-axis to move the slice selection frame again to be midway between the North and South-facing walls and click the mouse button to add the slice to the display.



Change CFD Analysis

You can now switch between the 2 CFD analyses you have carried out using the **CFD Analysis Manager** dialog. Select the analysis you wish to switch to and use the **Open** button to load results. Any slices associated with the analysis will be restored. These 2 analyses will be stored in the dsb file by default and are not deleted when changes are made to the model.

Appendix - CFD Calculations And Convergence

The numerical method used by DesignBuilder CFD is known as a primitive variable method, which involves the solution of a set of equations that describe the conservation of heat, mass and momentum. The equation set includes the three velocity component momentum equations (known as the Navier-Stokes equations), the temperature equation and where the k-ε turbulence model is used, equations for turbulence kinetic energy and the dissipation rate of turbulence kinetic energy. The equations comprise a set of coupled non-linear second-order partial differential equations having the following general form, in which f represents the dependent variables:

$$\underbrace{\frac{\partial}{\partial t}(\rho\phi)}_1 + \underbrace{\text{div}(\rho u\phi)}_2 = \underbrace{\text{div}(\Gamma \text{ grad } \phi)}_3 + \underbrace{S}_4$$

Term 1 represents the rate of change, term 2 represents convection, term 3 represents diffusion and S is a source term.

Due to its non-linearity, the equation set cannot be solved using analytical techniques, which necessitates the requirement for a numerical method. The numerical method employed by DesignBuilder involves re-casting the differential equations into the form of a set of finite difference equations by sub-dividing the required building space (or calculation domain) into a set of non-overlapping adjoining rectilinear volumes or cells, which is collectively known as a finite volume grid. The equation set is then expressed in the form of a set of linear algebraic equations for each cell within the grid and the overall set of equations is solved using an iterative scheme. The non-linearity of the equation set is accounted for by the use of a nested iterative scheme whereby each dependent variable equation set (velocity components, temperature, etc.) are themselves solved iteratively within an overall outer iterative loop and at the termination of each outer iteration, the most recent values of the dependent variables are fed back into the dependent variable coefficients. The outer iterative loop is repeated until the finite difference equations for all cells are satisfied by the current values of the appropriate dependent variables, at which point the scheme is said to have 'converged'. An appreciation of the requirement for convergence and the nested iterative procedure used to achieve it will help in understanding the meaning of the various calculation options that are described in the 'Conducting CFD Calculations' section.

The Iterative Nature of the Procedure

The calculation procedure has been developed to ensure that the iterative solution of the equation set would be guaranteed to converge if the equation coefficients were constant. However, the equation set is non-linear and the coefficients actually contain the dependent variables themselves, and consequently convergence cannot be guaranteed in all cases. Although in the majority of cases, as long as the dependent variables and particularly the velocities only change slowly, a converged solution is normally achieved.

The main mechanism to ensure that the variables change slowly is that of false time steps. The finite difference equation set is formulated in the form of a transient equation set although the calculations are steady state, i.e. essentially a 'snap-shot' in time. The reason for this formulation is that the transient term behaves as a very effective relaxation method, which can slow the change in dependent variables in order to arrive at a more stable solution. The false time step is the time step used in the pseudo-transient term of the dependent variable equation.

Air Compressibility

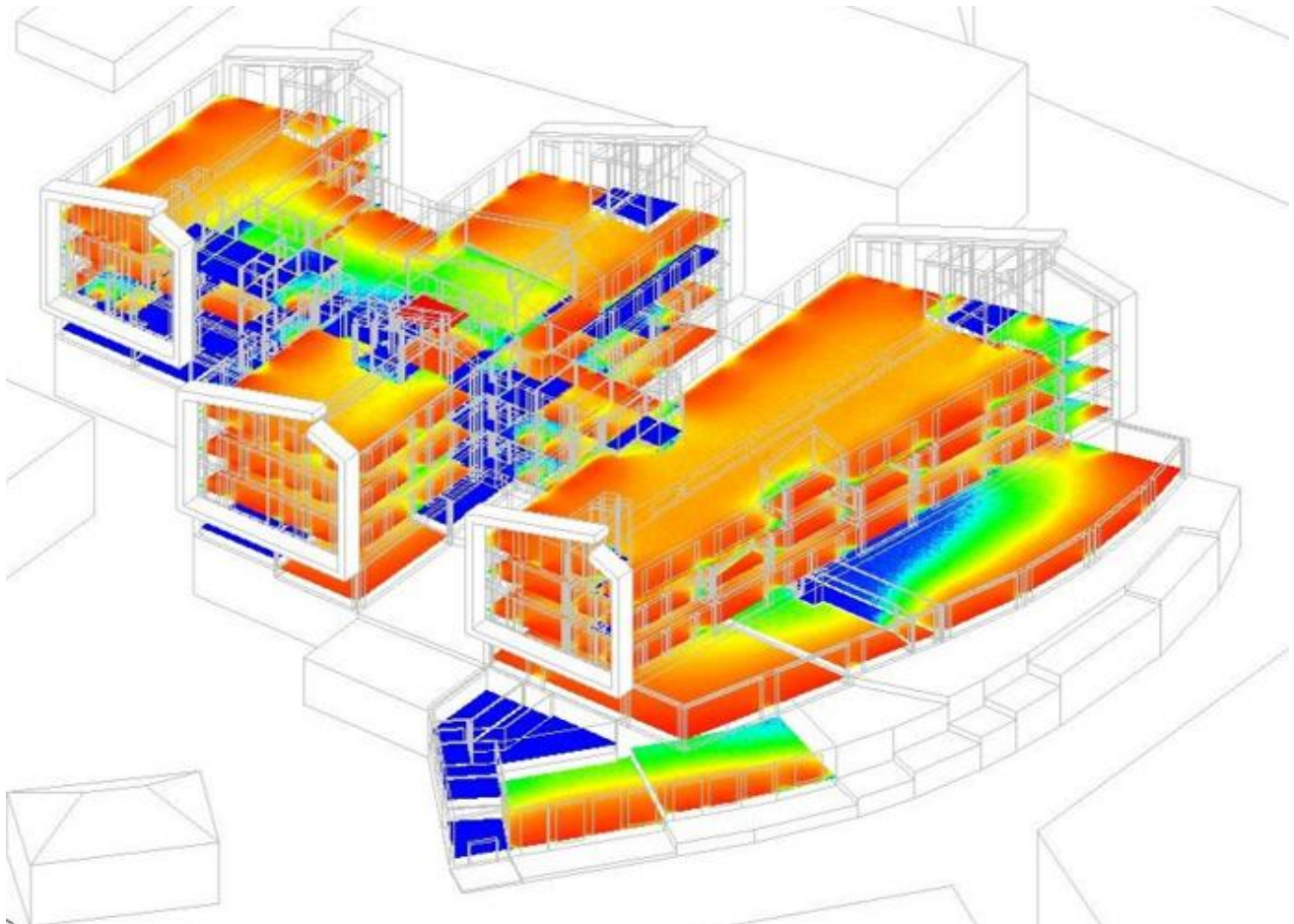
CFD calculation uses the so-called Boussinesq approximation that assumes the flow is incompressible except in the case of the buoyancy source term.

Tip: You can find more details on the calculation method used in the [CFD Technical](#) online document.

CFD Analysis Update

When you create a new CFD analysis, all of the model geometry and boundary conditions at that point are stored for that analysis and will always be used for simulations regardless of any subsequent changes to the model. So if you do make any further changes to the model they will not be taken into account in the simulations until you create a new analysis. So after you have made a change to the model that you wish to be included in subsequent simulations you must create a new CFD analysis.

Daylighting Calculations



DesignBuilder daylighting tools enable designers and assessors to check for good use of natural daylight and visual comfort in their building models by providing a wide range of daylight performance data. The results can be used to help guide the building design process and assess eligibility for credits with various rating schemes.

The [Daylighting module](#) is fully integrated with the core DesignBuilder thermal simulation feature, so, in general, no extra data needs to be added following previous energy simulation or certification project work. In particular building geometry, zone layouts, surface reflection, and glazing visible transmission properties from the thermal model are used in the daylighting calculations.

There are 4 main types of daylighting calculations in DesignBuilder:

1. [Point-in-time](#) illuminance simulations involve multi-zone physics-based calculation of illumination levels for static sky conditions using the Radiance simulation engine. Radiance applies a backward ray-tracing method to generate a map of illuminance on the working plane(s) in each zone. Point-in-time calculations allow you to generate Daylight Illuminance, Daylight Factor, and Uniformity results and other related reports.

Reports generated include: Daylight factor (DF and ADF), illuminance, uniformity etc. Standard reports for commonly used rating systems are also created, detailing eligibility for LEED v4, v4.1 (and older versions), BREEAM (HEA 01 4a) and Green Star (Credit IEQ4).

2. [Annual Climate Based Daylight Modelling](#) (CBDM, aka "dynamic daylighting") is the calculation of illuminance on the working plane using realistic sun and sky conditions and real climate data. Annual daylight simulations are run for a full year at a time-step of an hour to capture the daily and seasonal dynamics of natural daylight. The Radiance daylighting engine is also used in these simulations.

Results generated include: sDA, ASE and UDI distribution plots and summary grid reports. Standard formatted reports are also generated for: LEED (v4, v4.1); BREEAM HEA option b.

3. [Right to Light calculations](#) help check that new building developments meet planning requirements with regard to their impact on daylight availability in neighbouring buildings.

Results generated: Annual probable sunlight hours (APSH), Vertical sky component (VSC), No sky line (NSL) and Average daylight factor (ADF). VSC, APSH, ADF are calculated using Radiance while NSL calculations use an internal DesignBuilder ray tracing method.

4. [Daylighting integrated within EnergyPlus](#) simulations allow savings in electric lighting to be calculated based hour-by-hour on daylight availability throughout the year.

More information on Radiance daylighting calculations can be found in the [Introduction to Daylighting Tutorial](#)

See also:

- "Daylighting Model Preparation" on page 1970
- "Daylighting Calculation Options" on page 1974
- "Daylighting Outputs and Display Options" on page 2002
- "Daylighting Metrics" on page 2048
- "Guidance for Calculating Daylighting Credits for LEED, BREEAM, Green Star etc" on page 2055
- "Radiance Reference" on page 2057

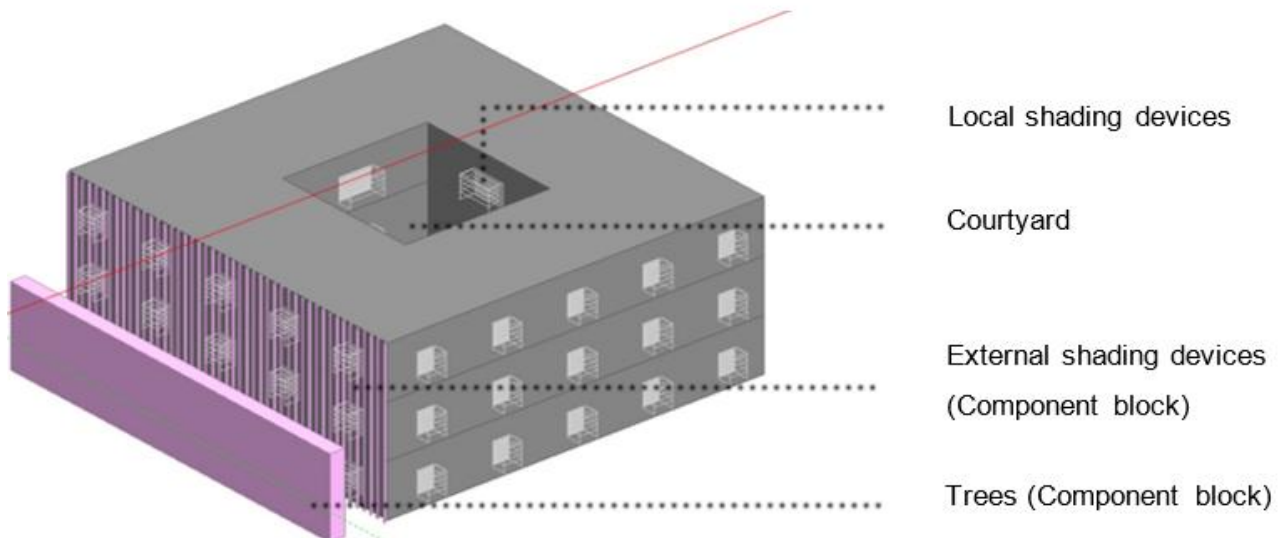
Daylighting Model Preparation

Before starting a Daylight simulation, the building model needs to be correctly defined. In this section the key factors that affect the daylighting results and their relevant settings, are discussed. These factors are:

1. Model geometry
2. Zones included in daylight simulation
3. Surface properties of outermost and innermost construction material
4. Site level ground reflectance
5. Glazing diffusing properties
6. Glazing visible properties
7. Window frames and dividers
8. Shading and components block

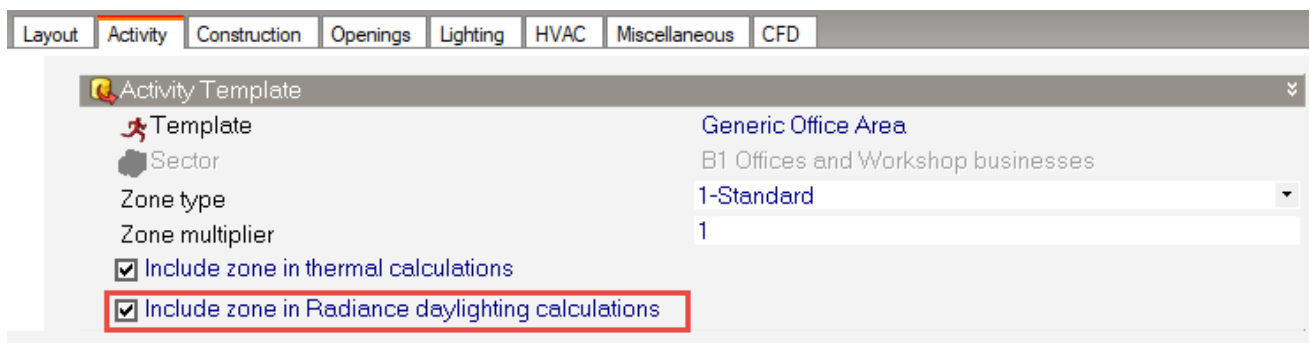
Model Geometry

To ensure the accuracy of the daylighting results, the building geometry needs to be modelled precisely with all the architectural elements such as courtyards, light wells, atriums, external shading devices etc included. Any obstructions near the building such as buildings, trees etc. should be modelled as component blocks or assemblies.



Zones Included in Daylight Simulation

To ensure that the zones for which the daylight analysis is to be conducted are included for the daylight calculation, navigate to that zone then go to the Activity model data tab. Under the Activity Template header check the [Include zone in Radiance daylighting calculations](#) checkbox.



Tip: in larger models, a convenient way to change the "Include zone in Radiance daylighting calculations" status for many zones at once is to use the "Model Data Grid View (MDGV)" on page 656 with the "Include Zones" Layout.

Surface Properties of Outermost and Innermost Construction Materials

The surface properties of materials used to define innermost and outermost layers of constructions for wall, roofs, partitions, floors etc play a significant role in light reflection and distribution inside spaces. It is therefore important to check the daylight-related properties of constructions, that is the surface properties of the outermost and innermost material applied to the construction.

Material surface properties can be checked and if necessary edited, from the Construction model data tab and editing the construction. On the Construction dialog the materials used in the innermost and outermost layers can be edited. The surface properties of the material can be accessed from [Surface properties](#) tab of the Materials dialog.

Edit material - Gypsum Plastering

Materials

General Surface properties Green roof Embodied carbon Phase change Cost

Surface Properties

Thermal absorptance (emissivity)	0.9000000
Solar absorptance	0.500
Visible absorptance	0.500
Roughness	3-Rough
☐ Colour	
☒ Texture	SandstoneBrown7402M

Radiance Daylighting

Material class	1-Plastic
Reflectance	0.500
Specularity	0.000
Roughness	0.000

From here the main settings of interest for daylighting are found under the [Radiance Daylighting](#) header:

- Material class
- Reflectance
- Specularity
- Roughness
- Transmissivity
- Fraction diffused

Important Note: It is important to consider that the visible properties of a material may be modified by applying paint or other surface finishes in the actual building. This can result in a very different properties compared to the value used for the "natural" material. For example, the default Visible absorptance of Gypsum plastering in the DesignBuilder database is 0.5, but when painted with bright white paint this could be as low as 0.1.

You can, if necessary, add a further innermost material to account for surface finishes such as paint.

Site level ground reflectance

As for the surface properties of the building outermost and innermost construction materials, the reflectance of the ground plane plays a significant role in reflecting light into the building. So it is important to enter a reasonable average value to represent the various surfaces that will reflect light from the external ground of the real site.

Enter the reflectance of the ground surface in the [Surface solar and visible reflectance](#) textbox. This setting is used to create a ground plane that reflects light in a diffuse manner.

Glazing diffusing properties

Glazing diffusing properties are set up on the Glazing dialog. They control the extent to which transmitted light is scattered in Radiance daylighting simulations.

Glazing

Layers | Calculated | Cost

General

Name: Project external glazing

Description:

Source:

Category: Project

Region: England and Wales

Colour:

Definition method

Definition method: 1-Material layers

Layers

Number layers: 2

Outermost pane

Pane type: Generic PYR B CLEAR 3MM

Flip layer:

Window gas 1

Window gas type: AIR 13MM

Innermost pane

Pane type: Generic CLEAR 3MM

Flip layer:

Radiance Daylighting

☒ Diffusing

Fraction diffused: 0.5

Specular (non-diffusing) glazing should have the [Diffusing](#) checkbox under the [Radiance Daylighting](#) header unticked whereas a fully diffused glazing system should have it ticked and the [Fraction diffused](#) set to 1.

Glazing visible properties

When the **1-Material layers** Glazing definition method is used, the visible properties of the selected panes will affect the transmission and reflection of light in Radiance daylight simulations. Likewise when the **2-Simple** definition method is set, the data on the Visible tab for the overall Glazing component is used in the daylighting simulations.

To change the Glazing and Pane's visible properties go to the Openings tab of Model data. You can access the Glazing types under the **External Windows** and **Internal Windows** headers. From the Glazing dialog, the glazing panes can be viewed, and if necessary changed, using the Pane dialog.

Window frames, dividers and reveals

The effect of [Window frames, dividers and reveals](#) can be defined on the Opening tab and are included in the Daylighting calculations.

Reveals account for the way that glazing is set back from the outside and inside surfaces of the wall they are located in.

Note: Frames and dividers are only applied to rectangular windows (default and custom) but not to other shaped windows. They are also not applied when using the Fill surface (100%) facade type.

Tip: To gain a clearer understanding of the window geometry being used in Daylighting simulations it can help to view the model on the Visualise screen.

Shading and components blocks

There are two ways to apply window shading:

1. Window shading (internal, external and mid-pane blinds) and Local shading (overhangs, sidefins and louvres).
2. Component or assembly blocks.

On the Openings tab under the **Shading** header make appropriate settings for [Window shading](#) and/or [Local shading](#). These need to be appropriately set and their material properties should be correctly defined as well, so that they absorb and reflect light appropriately.

[Component blocks](#) can also be used to define shading and surrounding blocks which may affect the total daylight. The material and material properties of component block should be set appropriately to accurately absorb and reflect light. Additionally, transmittance of component blocks can be changed to make them transparent or translucent to model changeable objects such as Trees.

When using component blocks, you can modify their properties from the Construction tab. From here the Maximum transmittance and transmittance schedule can be changed.

[Assemblies](#) can be used to define objects such as trees. They cannot be edited after they have been created and loaded into a model.

Note: The [assembly library](#) provided with DesignBuilder includes 4 types of trees for loading directly into models. These can be scaled or stretched to meet the requirements of your model.

Hourly weather file

Select the [Hourly weather data](#) from the list of [Hourly weather data components](#). This data is used to provide Radiance with hourly direct and diffuse illuminance data for annual daylighting simulations.

The hourly weather data is also needed to prepare point-in-time [Sky Settings for LEED v4 and v4.1 Option 2](#) Daylighting calculations. Note that in this case the weather file is not actually used in the Radiance calculations.

Daylighting Calculation Options

Daylighting calculations are controlled using a set of options on the Daylighting calculation options dialog which is displayed before the calculations are started. To access this dialog (and the Daylighting calculations)

press the Update toolbar icon  from the [Daylighting](#) screen.

Calculation Description

Calculation description

Naming the calculation will provide a useful reference for your simulation results. You may wish to include information on any variation made for each calculation.

General Options

Simulation type

Select the type/purpose of the simulations that you require. The available options are:

- **1-General** - the default option, for general purpose daylighting calculations when none of the other options 2 - 6 from the list below apply and/or full flexibility is required.
- **2-LEED** - provides access to various types of LEED daylighting calculations from LEED v2 through to v4.1 including option 1 (annual) and option 2 (point in time) simulations when LEED v4 or v4.1 is selected.
- **3-BREEAM** - provides access to BREEAM HEA Options 4a (point in time) and 4b (annual) calculations.
- **4-LM-83-12** - the option to choose for sDA, ASE and UDI simulations when they are not for LEED or BREEAM.
- **5-Right to light (BS8206)** - provides access to right to light calculations for assessing the impact of new developments on daylight availability in a building. [Annual probably sunlight hours \(APSH\)](#), [Vertical sky component \(VSC\)](#), No sky line (NSL) and Average daylight factor (ADF) reports can be generated.
- **6-Green star** - provides access to the point in time Green star daylight factor calculations and report.

If you do not require any particular report or you require full flexibility in the calculation settings then select **1-General**.

Note: Daylight results are stored separately for each Simulation type so you can generate and store multiple results sets, one for each of the above, each using different calculation option settings.

Detail template

This selection allows you to set the output quality of the simulation. The quality and accuracy increase down the list. For most typical simulations, the **4-Good** setting is sufficient.

Technical Details

Selecting a detail template from the drop list loads a set of Radiance settings and ambient parameters to the dialog as shown in the table below.

You do not necessarily need to have an in-depth understanding of the meaning of each of these parameters to carry out an accurate assessment in DesignBuilder. However knowing a little bit about the way the calculations work and how the options affect the calculations will help you to understand how to get the best trade-off between accuracy and simulation times.

Detail template	Ambient bounces	Ambient accuracy	Ambient resolution	Ambient divisions	Ambient super-samples
	<i>Radiance code</i> -ab	-aa	-ar	-ad	-as
1-Fast Not recommended for project work	1	0.3	128	256	128
2-Standard Lowest usable quality - but will underestimate illuminance	3	0.25	256	512	256
3-Good (no interpolation)	4	0.00	512	1024	512
4-Good Default setting for balanced accuracy and simulation speed	4	0.22	512	1024	512
5-Accurate Best setting for highest practical accuracy	5	0.20	512	2048	1024
6-High quality Use when highest quality is required and a long wait for results is acceptable	7	0.18	1024	4096	2048

Tip: For a discussion on the relative merits of using some of the above settings for daylight credit calculations see "Guidance for Calculating Daylighting Credits for LEED, BREEAM, Green Star etc" on page 2055.

Working plane height

Enter the height of the working plane above floor level for each zone in the daylight simulation (in m or ft). The default value will depend on the [Simulation type](#) selected. The working plane height entered on this dialog corresponds with the building level setting on the Output model data tab.

The working plane height is normally set at the average height of the top surface of the desks above the floor. A typical value will be in the range 0.7-0.8m.

Note: The working plane height used in Radiance calculations is measured from the top of the floor surface level and not the base of block.

Tip: To define different working plane heights for different zones, you can override the building level setting on this dialog by entering values under the [Daylighting outputs](#) header on the Outputs model data tab at block or zone level.

Minimum grid size

This is the smallest grid size that can be used to fill in gaps in the working plane in cases where the maximum grid size is too large, especially at the edges of the plane. Use smaller values (e.g. 0.01m) to fill in most gaps between the main grid cells and the borders of the working plane for a more continuous output. It is common to use a Minimum grid size that is a factor of between 0.5 and 0.1 times the Maximum grid size.

To maintain a *uniform grid* across the working plane set the Minimum grid size the same value as the Maximum grid size (below).

Maximum grid size

The main grid size that can be used to divide up the working plane when calculating illuminance. Larger values will speed up calculations but give lower resolution in daylighting outputs.

You should consider the size and number of zones in the building when selecting the grid size. A very fine grid will of course cause Radiance calculations to take a lot longer to complete. On the other hand a very coarse grid in a small zone may not give a good enough distribution.

You should use a maximum grid size no greater than 5ft or 1.5m for LEED daylighting calculations.

Ground plane extension

If you enter a value greater than zero in this cell, DesignBuilder automatically generates a ground plane below all site objects to model ground-reflected light. The ground plane is generated behind the scenes and its dimensions are calculated by setting up a rectangular box in the plan view around the building extents and adding the [Ground plane extension](#) at each edge.

The auto-generated ground plane uses the [Ground reflectance](#) defined at site level.

An alternative approach to model ground reflectance is to set the [Ground plane extension](#) to zero and to use [ground component blocks](#) instead. Or a combination of ground component blocks and the ground plane extension can be used.

Tip: While it may be tempting to enter a very large number here to cover a large area, that is not usually recommended as it would significantly change the scene dimensions used within Radiance. Larger scene dimensions require high detail settings which may require very long simulation times.

Margin

Enter the margin (in m or ft) around the zone boundaries where illuminance data is not to be included in summary results. This option can be used to help avoid inclusion of potentially misleading illuminance data close to walls and windows. DesignBuilder sets the default margin depending on the simulation type selected.

Include all buildings

If you would like other buildings to be included in the daylighting study as shading/reflecting surfaces then check the **Include all buildings** checkbox. Note that daylighting results will only ever be generated for the current building.

Very high solar altitudes

The whole sky distribution is normalized to the zenith value, so having the sun too near to the zenith could be a problem and therefore Radiance limits the solar position a maximum altitude of 87 degrees. If the calculated solar altitude is higher than this limit then Radiance uses the maximum allowed value of 87 degrees in the calculations. A warning message will be displayed informing this change. Generally this restriction should not impact the daylighting results.

Technical discussion on hemispherical sampling

Hemispherical sampling at the first level will always be initiated from the first point supplied to rtrace. From these hemispherical sampling rays, the ambient calculation will predict the way the indirect irradiance is changing about that point (the indirect irradiance gradient). The calculation also evaluates an estimation of error associated with the prediction for the irradiance gradient. These quantities, together with the Ambient accuracy parameter, are used to determine a “radius of validity” for the gradient estimate. If the next point supplied to rtrace is within this radius, the indirect irradiance is evaluated from the gradient estimate and not from further hemispherical sampling. In other words, the value is obtained by a form of interpolation rather than by actual sampling.

Irradiance interpolation can occur across the points supplied to rtrace, so hemispherical sampling at the first level will not necessarily be initiated from every point in the working plane supplied to rtrace.

It is important to appreciate the element of chance at work whenever hemispherical sampling is used. If the number of initial sampling rays were set too small, the calculation might, for example, miss a bright patch even though it was visible from the point at which the rays were spawned. Likewise, an unrepresentative chance “hit” of a small bright patch by one of the sampling rays can produce a gross overestimate for indirect irradiance. In a rendering, the artefacts associated with ambient undersampling can cause bright and dark blotches.

To minimise blotches we need to set a sufficiently high value for the number of initial sampling rays, Ambient divisions. Hemispherical sampling is generally too expensive to initiate at every surface visible from the eye point. The calculation needs good indirect irradiance estimates from sampling at a limited number of locations. We then rely on the irradiance interpolation algorithm to estimate missing values. To generate a fairly smooth rendering for a daylight calculation accounting for the first level of inter-reflection, we would need to set moderately high resolution values for the ambient parameters.

Simulation Options

Each [Simulation type](#) has a different set of calculation options and you can find help for each in topics below.

- [General](#)
- [LEED](#)
- [BREEAM](#)

- [LM-83-12](#)
- [Right to Light](#)
- [Green Star](#)

Advanced Options

The following options are available on the Daylight Calculation Options Dialog for static illuminance and annual CBDM simulations. They govern the quality of the simulation and are best set through the [Detail template](#) by non-expert users.

Ambient bounces (-ab)

The number of ambient bounces is the maximum number of diffuse bounces computed by the indirect calculation. The number of ambient bounces that Radiance should apply varies depending on the type of building and daylighting system you are analysing. It can be set based on the number of reflections typically required by the light to reach the task plus one or two extra for inter reflection within the space.

The number of bounces should normally be set to 3 for most accurate calculations or 2 if some trade-off is acceptable. 4 does not usually add much accuracy over and above 3 and 1 and 0 are inadequate for most daylighting calculations.

Note: When the number of ambient bounces is set to 0 the ambient lighting calculations are switched off, so only direct sun/sky light patches are considered.

Tip: It is possible to calculate the fraction of the working plane which receives direct sunlight by setting the **Ambient bounces** calculation option to 0 and the Illuminance threshold display option to 0, then viewing the **Floor Area above Threshold** data in the Grid output.

If the number of ambient bounces is set to 1, light is considered to reach the interior surfaces from the sun's direct rays, from the diffuse sky, and from first bounce reflections of direct sunlight rays from both interior and exterior surfaces. Reflections of sky light off interior or exterior surfaces to other interior surfaces will not be considered. Additional bounces can be added to consider additional flux paths.

Note that a value of 3 for the number of ambient bounces is often enough for daylight factor calculations. However, if the daylight factor plot is "lumpy", setting a higher value alone will not fix it.

Doubling ambient bounces can double rendering time.

Ambient accuracy (-aa)

Ambient accuracy (-aa) is the maximum error (expressed as a fraction) permitted in the indirect irradiance interpolation.

You should normally use a value between 1 and 0.1, with lower values giving the best accuracy. A value of zero gives no interpolation.

Halving Ambient accuracy approximately quadruples rendering time.

Ambient resolution (-ar)

The Ambient resolution sets the distance between ambient calculations by determining the maximum density of ambient values used in interpolation. Factors that influence the scale over which interpolation may occur are:

- The ambient accuracy.
- The ambient resolution.
- The maximum scene dimension.

The minimum possible spacing between hemispherical indirect irradiance sampling points is the maximum scene dimension multiplied by the ambient accuracy divided by the ambient resolution. In other words, for distances less than this minimum, the calculation will always resort to interpolation, rather than initiate more sampling, regardless of the error estimate associated with that interpolation. This prevents the calculation from spending unnecessary time resolving irradiance gradients over negligible scales. This distance gives the scale at which the interpolation accuracy begins to deteriorate from the ambient accuracy setting. The minimum separation for calculated irradiances, S_{min} , is:

$$S_{min} = D_{max} \times \text{Ambient accuracy} / \text{Ambient resolution}$$

Where the scene dimension, D_{max} , is a measure of the maximum dimension of the zone being treated.

The effect of increasing Ambient resolution depends on the scene but it can quadruple calculation times for double the value.

Ambient divisions (-ad)

Ambient divisions sets the number of initial sampling rays sent from each ambient point into the hemisphere to determine the indirect incident light. The error in the Monte Carlo calculation of indirect illuminance will be inversely proportional to the square root of this number. A value of zero implies no indirect calculation.

The Ambient divisions and super-samples parameters can be used to help reduce "noise" in a calculation. By setting these options higher more rays will be tested when calculating an ambient value for a point.

High values of this parameter (4096 is perhaps the highest value that would be used under normal circumstances) will minimise "patchiness" of daylighting outputs but slow calculations. Doubling Ambient divisions will approximately quadruple calculation time.

Ambient super-samples (-as)

The number of extra rays that will be used to sample areas in the divided hemisphere that appear to have high variance. Ambient super sampling should usually be set to about one half or one quarter of the Ambient divisions parameter. Super-samples are applied only to the ambient divisions which show a significant change.

The effect of increasing Ambient super-samples is to reduce "patchiness" in regions where indirect illuminance is changing rapidly, but adding to the Ambient divisions and calculation times.

Setting Ambient Parameters

Some general advice on the approach is:

1. Set [Ambient divisions](#) high enough to capture the visible luminous features at the first bounce.
2. Give sufficient [Ambient bounces](#) to redistribute the light.
3. Set the remaining ambient parameters to sufficiently high resolution to deliver acceptably smooth results.

Monte Carlo random sampling

When this option is selected Radiance uses Monte Carlo random sampling during the simulations which causes results for each simulation to have small variations. In other words you will see small differences in results each time you run the same simulation. To ensure that results are consistent between simulations you should leave this option unchecked.

Sky patches

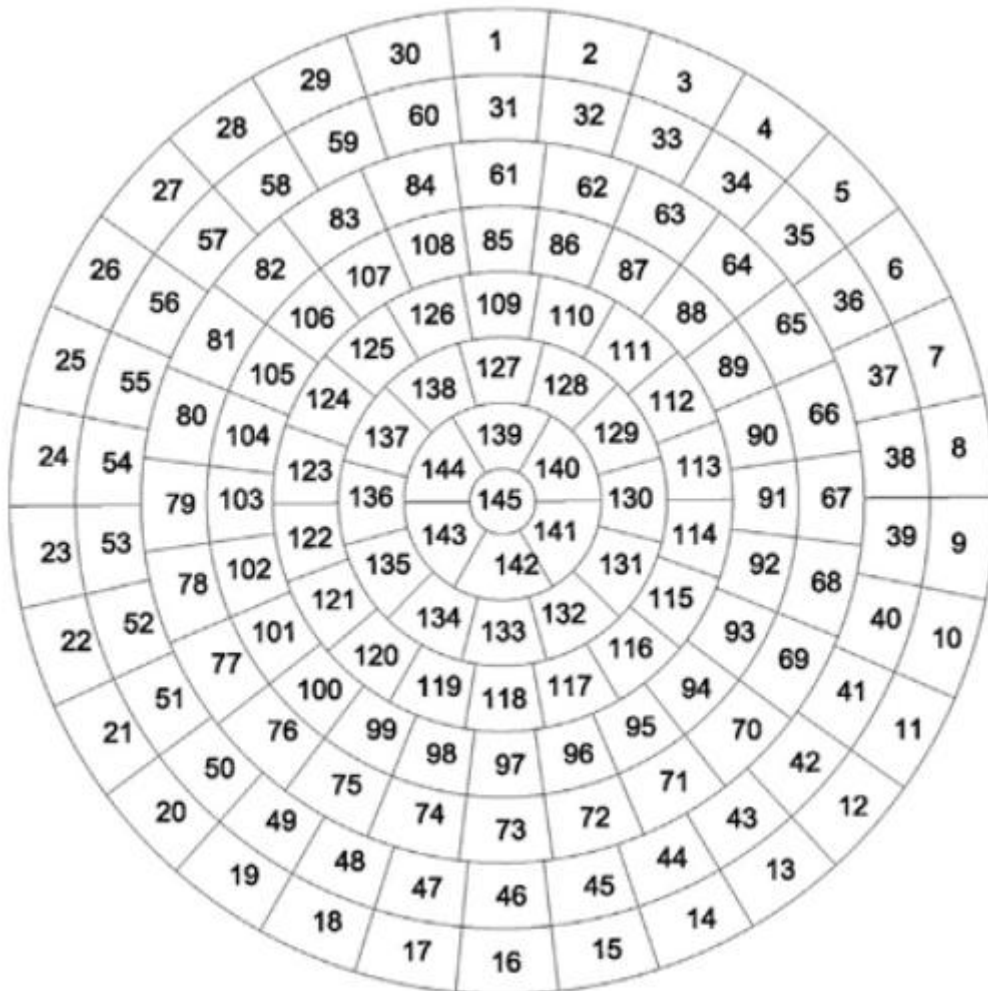
In daylighting simulations, the sky is broken up (discretised) into a number of "patches". The more finely the sky is discretised, the more accurate the daylighting results and longer the simulations will take.

This option is used in all Annual daylighting simulations and in No Sky Line Right to Light calculations.

Important Note: For **No Sky Line** Right to Light calculations you should select the greatest available detail, option **6-5185**.

You can select the number of sky patches that will be used from the options listed below:

- **1-145** - the original Tregenza sky discretised into 145 patches.



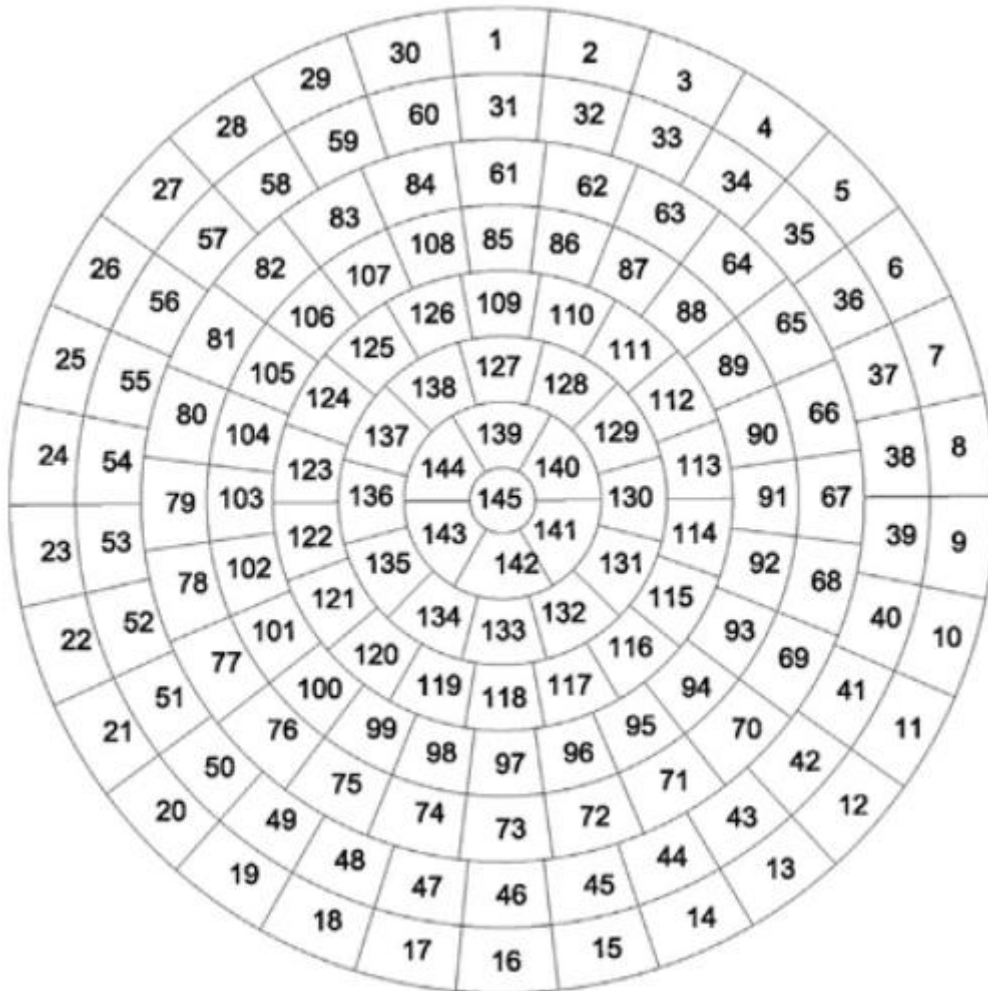
- **2-577** - sky is discretised into 577 patches. This is the default option corresponding to Reinhart's extension of the Tregenza sky, where the original 145 patches are subdivided into 4 subpatches, except at the zenith. MF:2 Reinhart subdivision scheme.
- **3-1297** - sky is discretised into 1297 patches. MF:3 Reinhart subdivision scheme.
- **4-2305** - sky is discretised into 2305 patches. This corresponding to Reinhart's extension of the Tregenza sky, where the original 145 patches are subdivided into 16 subpatches, except at the zenith. MF:4 Reinhart subdivision scheme.
- **5-3601** - sky is discretised into 3601 patches. MF:5 Reinhart subdivision scheme.
- **6-5185** - sky is discretised into 5185 patches. MF:6 Reinhart subdivision scheme. This is the most detailed option and is recommended for use with the No sky line Right to Light calculation which requires a fine sub-division of the sky for accurate results.

Sun patches

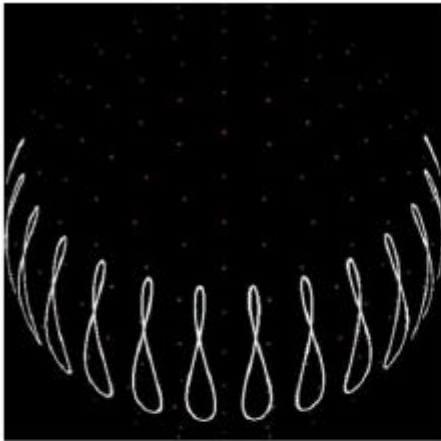
In annual daylighting simulations, the position of the sun at any particular timestep is approximated to lie on a grid of Sun patches. This option allows you to define the granularity of the grid.

This setting is used in all Annual daylighting simulations.

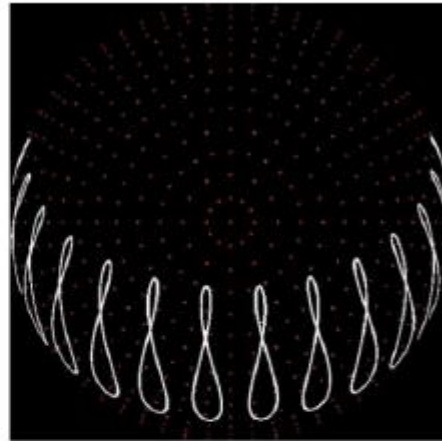
- **1-145** - the original Tregenza sky discretised into 145 patches.



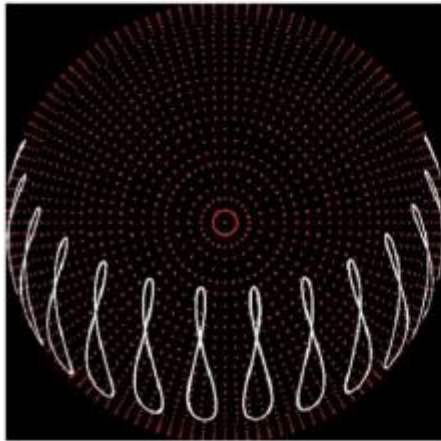
- **2-577** - sky is discretised into 577 patches. This is the default option corresponding to Reinhart's extension of the Tregenza sky, where the original 145 patches are subdivided into 4 subpatches, except at the zenith. MF:2 Reinhart subdivision scheme.
- **3-1297** - sky is discretised into 1297 patches. MF:3 Reinhart subdivision scheme.
- **4-2305** - sky is discretised into 2305 patches. This corresponding to Reinhart's extension of the Tregenza sky, where the original 145 patches are subdivided into 16 subpatches, except at the zenith. MF:4 Reinhart subdivision scheme.
- **5-3601** - sky is discretised into 3601 patches. MF:5 Reinhart subdivision scheme.
- **6-5185** - sky is discretised into 5185 patches. MF:6 Reinhart subdivision scheme. This is the most detailed option.



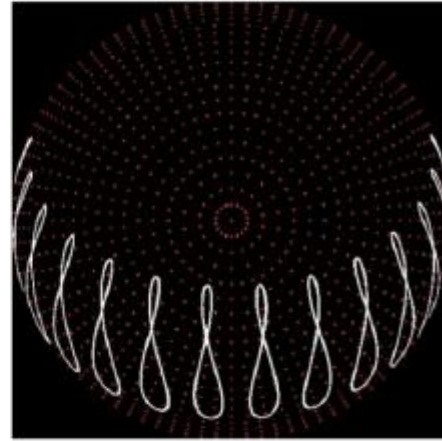
MF:1 (145)



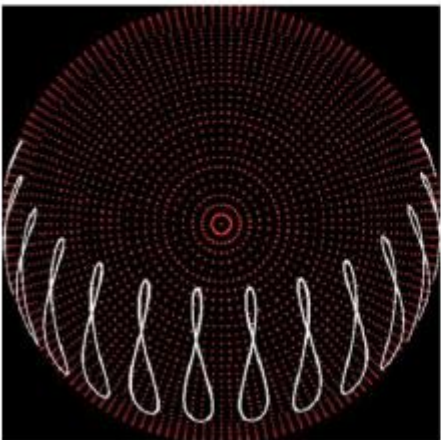
MF:2 (577)



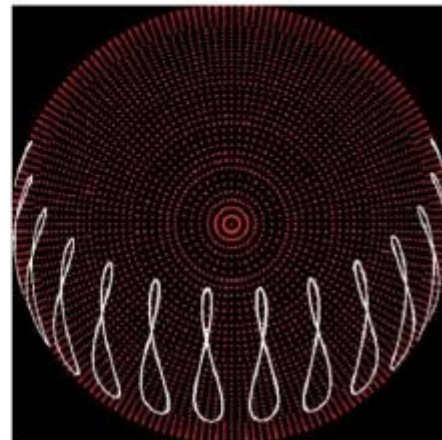
MF:4 (2305)



MF:3 (1297)

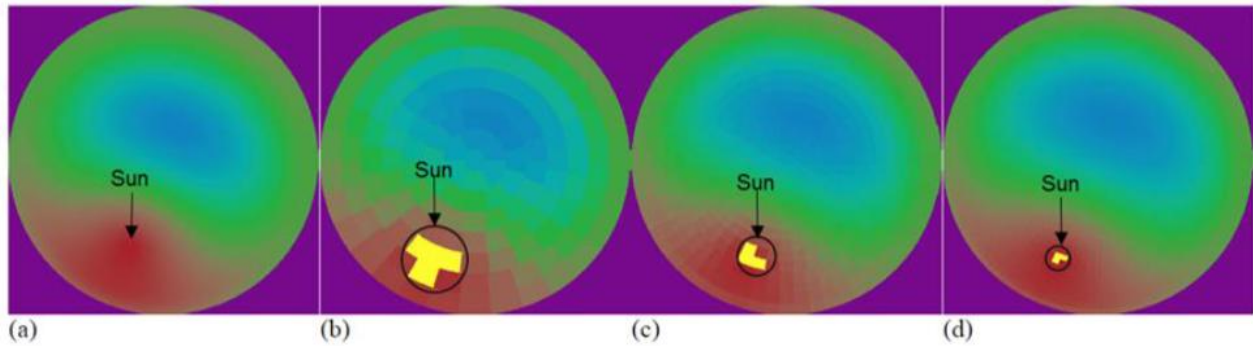


MF:5 (3601)



MF:6 (5185)

Illustration of the Sun patch concept. Each of the above images is a fisheye rendering for the solar positions corresponding to a particular value of MF (Radiance parameter defining the number of subdivisions for each Tregenza patch). The sky discretisation is shown with red dots and the actual positions of the sun for every hour of the year are shown as white lines. Clearly the greater the number of Sun patches, the more accurate the solar position in the simulations.



The above image shows fish-eye projections of a continuous sky model (a) along with corresponding discretised versions. Images (b), (c) and (d) use 145, 580 and 2305 sky patches respectively. The images show that even with a high degree of discretisation, the size of the sun is overestimated in discrete sky models as the actual position of the sun in the sky at a given time is approximated to 3-4 sky patches. Clearly, this approximation in position is accompanied by an overestimation of the size of the sun with respect to the sky.

Max usable cores

Enter the maximum number of cores that DesignBuilder should use for the Radiance simulations. Enter 0 to use all cores. Note that if you allocate all cores to Radiance, your machine may not be sufficiently responsive to allow you to work on other tasks while the simulations run.

Daylighting Calculation Options - General

General Simulation Options

Perform annual simulation

If you would like one or more of the annual Climate Based Daylighting Method annual simulations to be run tick this checkbox.

Perform point in time simulation

If you would like one or more of the point in time illuminance simulations to be run tick this checkbox.

Annual Options

Window shading

This setting allows any window blinds selected on the Openings tab to be applied for daylighting calculations as well, or LM-83-12 rules for dynamic blinds can be applied.

- **1-No shading** - don't apply any [Window shading](#) for the daylighting simulation, even if it is selected on the Openings model data tab.
- **2-Fixed blinds** - apply any [Window shading](#) model data settings from the Openings tab. Any blinds selected will be applied in their activated (closed) state.
- **3-Dynamic blinds (LM-83-12)** - use the rules defined in Section 2.2.7 of LM-83-12 to dynamically control blinds to maximise daylight availability while avoiding glare. Note that when this option is selected, windows have dynamically controlled blinds applied regardless of whether the [Window shading](#) model data is selected. When dynamic blinds are selected and internally positioned slatted or diffusing blinds are specified on the Openings tab then the [blind properties](#) set in model data are used in the dynamic blinds. Otherwise, if Window shading is not set, or if blinds are not internal (they are external or mid-pane), or they are neither slatted nor diffusing blinds then a diffusing blind with reflectance of 0.2, transmittance of 0.05 and specularity of 0 are used for the dynamic blinds. This option is automatically selected behind the scenes when the Simulation type is **2-LEED** or **4-LM-83-12**.

Note: [Local shading](#) model data settings are *always* applied to Radiance simulations.

Dynamic shading method

This setting allows you to control the algorithm used to simulate the dynamic blinds. Two methods are provided, **1-Standard** and **2-Detailed**. The **2-Detailed** option allows dynamic shading to be fully optimised for buildings where light is transferred through internal windows that connect zones with different occupancy schedules. However, these optimised calculations can take a long time to run in large buildings with many internal windows due to the ray tracing method used to identify the dependency of each room on every other. So a second **1-Standard** option is also provided which introduces some simplifying assumptions which help it to run much more quickly in such buildings without significantly impacting the accuracy of the sDA results. If all rooms in the building have the same occupancy or if there are no internal windows then the issue doesn't arise and either method can be used.

- **1-Standard** –The standard method assumes that rooms with external windows always define a host window's shading schedule for all possible hours, regardless of whether the room is occupied or not. This simplification avoids the need for the "dependency calculations" referred to below for the **2-Detailed** method and significantly reduces overall simulation time in large models with many interconnected rooms. There is some trade-off in accuracy but in most cases, this will not have a significant impact on sDA results.
- **2-Detailed** - The Detailed method identifies all external windows that have an impact on every other room, accounting for light transmission through internal windows. When calculating the dynamic shading for external windows, the occupancy schedule of the host room is applied while it is occupied. However, during hours when the host room is unoccupied, some sophistication is required to determine the dependency of rooms on each other so as to ensure that host rooms are calculated before dependent rooms. In this case, dynamic shading schedules are set up for external windows according to the requirements of any dependent rooms which are occupied. In cases where there are multiple dependent rooms, the occupancy schedule of the room with the greatest floor area is used. The dependency calculations are carried out using a ray tracing technique which can take a long time to run in large buildings with many interconnected rooms.

Note 1: Adding partitions to subdivide large zones reduces simulation time for both methods. However, the effect is particularly pronounced for the **2-Detailed** method, where the reduction in time is significant.

Note 2: Although the **1-Standard** method is generally faster, its simulation time increases substantially with the addition of more floors. By comparison, the **2-Detailed** method remains relatively stable, with only a modest increase in simulation time.

Technical Notes

According to LM-82-12, when calculating sDA, the use of shading is permitted and when calculating ASE, the use of shading is not permitted. The two metrics are used together, with ASE informing us when blinds should be used when simulating sDA. sDA provides the lower bound of what is an acceptable amount of light and ASE provides an upper bound. When the 2 methods are combined we are aiming to get as much light into a zone as possible whilst making sure we only allow direct sunlight into a maximum of 2% of the working plane. Dynamic shading solves this by calculating which combinations of external window blinds should be open or closed for any given hour so that as much light as possible is allowed into a space without exceeding the limits imposed by ASE and so ensuring that glare is avoided.

When performing an annual daylight simulation with dynamic shading, the process can be summarised as follows:

- 1 For each zone:
 - 2 For each occupied hour of the zone's schedule (hours outside of a zone's schedule are not used)
 - 3 Calculate ASE. If the zone exceeded the ASE thresholds, this zone will need blinds closed during this hour. If not, the blinds may remain open during this hour.
 - 4 For each occupied hour that failed ASE: (i.e. all the hours that will need closed blinds)

5 Calculate how much direct sunlight each window group contributes to the ASE result and use that to determine the best combination of blinds to close, and which ones can remain open so that we allow the most amount of light through without failing ASE

6 For each occupied hour of the zone's schedule:

Calculate sDA using the blind configuration determined in step 5 if this hour failed ASE, or using open blinds if this hour passed ASE

Note that for each zone that has access to blinds, the configuration of the blinds may change every hour. However, when determining the occupancy hours where dynamic shading should be applied (step 5), things can get complicated when considering the effect of light through internal windows.

Before discussing the challenges in detail, some terminology is needed:

- A "host room" is a room in which one or more external windows allow light to enter that is also transmitted to other "dependent rooms" or "adjacent rooms" through internal windows.
- A "shading schedule" is a schedule of values which defines the hours in the year when dynamic shading is to be applied for each external window

The problem in DesignBuilder v7.0

In DesignBuilder v7.0, the shading schedule for external windows is defined only during the hours the host room is occupied and only used when calculating the host room's sDA values. However, this simple approach can create issues where a dependent room could get higher sDA values than a host room when the dependent room's only source of light came from the host room's external windows. There were 2 problems to solve:

1. How to ensure that an adjacent room uses the shading schedule of all external windows that affect the adjacent room (there may be more than one host room).
2. What to do when an adjacent room is occupied outside of the hours defined by the host room's shading schedule:
 - a. What if there are multiple host rooms?
 - b. What if there are multiple adjacent rooms that are affected by the same host rooms?

The solution in v7.3 and later

The first problem was solved by first creating a dependency graph of rooms affected by one another and calculating the shading schedule for each set of rooms in topological order (a linear ordering such that for every directed edge uv from vertex u to vertex v , u comes before v), to ensure the host rooms are calculated before the adjacent rooms. The room dependency ordering was created by tracing a ray from every sample point of each room to every external window in the building. If the ray hit an obstruction on the way (a wall, for example) it was discarded. If a ray managed to make it to an external window then the source room must be dependent on that window.

The second problem was solved by first calculating ASE for the entire range of occupied hours in the building, not just those used by each room. This allowed us to calculate shading schedules for any hours that may be required by dependent rooms later on. For example, if a room is occupied between the hours of 09:00 and 14:00, but the range of occupied hours in the building is 06:00 – 18:00, ASE is now calculated for 06:00 – 18:00 for every room, regardless of how many hours it is actually occupied. Then, for every host room, the shading schedule is calculated according to the room's occupancy schedule. For any occupancy hours shared by a host room and a dependent room, the shading schedule of the host room is used in the sDA calculation of the dependent room. For occupied hours of a dependent room that lay outside of the occupancy schedule of a host room, a new shading schedule is derived for those hours.

In the case where a host room had many dependants with overlapping occupancy schedules, rooms that would have a bigger effect on the final result were given priority. For example, a host room is occupied between 11:00 and 13:00. Dependant room A is occupied between 09:00 and 12:00. Dependant room B is occupied between 09:00 and 15:00. Between the hours of 11:00 and 13:00 all three rooms use the shading schedule of the host room since it is the owner of the windows. It is determined that dependant room A has a bigger floor area than room B, so takes priority. Therefore, between the hours of 09:00 and 11:00 room A and room B use the shading schedule of the host's windows as defined by room A. Finally, room B uses its own shading schedule for the remaining hours of 13:00 to 15:00. This is what has become known as the "Detailed" Dynamic shading method.

Although this Detailed method provides accurate results it can take a very long time to calculate in large buildings with many connected rooms. The bottleneck is mainly caused by the computation of the room dependencies. The standard method avoids the need for the dependency calculations by defining the shading schedule for all possible hours regardless of occupancy in the host zone. A shading schedule is

calculated for every external window for every hour occupied by some room in the building as though the host room itself was occupied during that time. The new method then partitions all occupancy hours of every room into overlapping schedules. For the previous example of three rooms with overlapping schedules, the new method would calculate all three rooms at the same time for the hours 11:00 to 13:00 using the shading schedule of the host room. Then it would calculate rooms A and B together between the hours of 09:00 and 11:00 again using the shading schedule of the host room even though the host room isn't occupied during this time. Finally, it would calculate room B between 13:00 and 15:00 with a shading schedule again defined by the host.

The key insight is that all rooms that share an occupancy time are calculated together even if there were no dependencies between them. Because the shading schedules of all external windows had already been defined, it didn't matter whether the rooms interacted or not because they had no influence on the shading.

By dropping the need to calculate dependencies, the performance of the Standard method was vastly improved over that of the Detailed method. However, because the Standard method does not take into account the individual needs of dependent rooms when calculating hours outside of a host room's occupied hours, results are not quite as accurate as with the Detailed method. Nevertheless, in most cases, the difference between the two is negligible. Any differences are primarily in cases where a dependent room requires different shading to the host.

Override zone occupancy schedule

By default the occupancy schedule used for each zone in the Annual simulations are based on the zone [Occupancy schedule](#) defined on the Activity tab, however you can override this to use the same occupancy schedule for each zone by selecting this option and selecting the schedule in the browse control below.

Occupancy schedule

When the above **Override zone occupancy schedule** option is selected, you can select the occupancy schedule to be used to define when zones are occupied for all zones in the simulation. Values in the schedule greater than 0 indicate occupancy.

The default occupancy schedule provided by DesignBuilder does not include daylight saving time (DST). DST has to be added manually to the schedule. For example for the Northern Hemisphere you can use a schedule like the following:

```
Schedule:Compact,  
8:00 - 18:00 Every day (DST Northern Hemisphere),  
Fraction,  
Through: 03/30,  
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0,  
Through: 10/26,  
For: AllDays,  
Until: 07:00, 0,  
Until: 17:00, 1,  
Until: 24:00, 0,  
Through: 12/31,  
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0;
```

Exclude unlit hours

You can exclude hours where no light is received in a zone from results by checking this checkbox. This option is only available when the **1-Custom** Simulation type is selected. For all other Simulation type settings the calculations are carried out as if this option is unchecked.

sDA Thresholds

sDA lower illuminance threshold

When annual results are being generated you can enter the lower illuminance threshold, the minimum level of illuminance (in lux or fc) below which lighting levels are considered to be inadequate. A typical lower threshold value for sDA calculations is 300 lux.

ASE Thresholds

ASE illuminance threshold

When annual results are being generated you can enter the maximum level of illuminance (in lux or fc) above which lighting levels are considered to be likely to lead to visual discomfort, i.e. glare. A typical upper threshold for ASE calculations is 1,000 lux.

UDI Thresholds

UDI lower illuminance threshold

When annual results are being generated you can enter the minimum level of illuminance (in lux or fc) below which lighting levels are considered to be inadequate. A typical lower threshold value for UDI calculations is 300 lux.

UDI upper illuminance threshold

When annual results are being generated you can enter the maximum level of illuminance (in lux or fc) above which lighting levels are considered to be excessive and lead to visual discomfort, i.e. glare. A typical upper value for UDI calculations is 3,000 lux.

Point-in-time Options

Window shading

This setting allows any window blinds selected on the Openings tab to be applied for daylighting calculations as well.

- **1-No shading** - don't apply any [Window shading](#) for the daylighting simulation, even if it is selected on the Openings model data tab.
- **2-Fixed blinds** - apply any [Window shading](#) model data settings from the Openings tab. Any blinds selected will be applied in their activated (closed) state.

Note: [Local shading](#) model data settings are *always* applied to Radiance simulations.

Sky method

There are 2 methods for generating the sky illuminance distribution for the Radiance calculation:

- **1-Standard sky** - the default method where one of the predefined standard sky distributions are applied using the Radiance [gensky](#) tool.
- **2-Perez all weather** - an alternative method suitable for all weather types based on the Perez sky clearness (ϵ) and sky brightness (Δ) parameters and using the Radiance [gendaylit](#) tool. This option

must be selected for LEED v4 Option 2 calculations. There are 3 Perez methods to choose from as described below.

Perez method

When the Sky method is set to **2-Perez all weather** there are 4 methods to choose from depending on the data you have available. In all cases the time, day and month of the analysis must be set to define the location of the sun in the sky.

- **1-Brightness and clearness** - if you know the Perez sky clearness (ϵ) and sky brightness (Δ) parameters of the sky to be analysed choose this option.
- **2-Direct normal irradiance** for when measured direct normal and diffuse horizontal irradiance data is available. This option can be useful since it allows readily available solar data in W/m² from the EnergyPlus weather file to be used to define sky illuminance. Go to the Simulation tab, navigate to site level and select hourly data to see solar data suitable for the location of your building for any hour of the year. Note that using full spectrum solar data to define illuminance is seen by daylighting purists as less accurate than using measured values of actual illuminance (see option 4).
- **3-Direct horizontal irradiance** for when measured (or calculated) direct horizontal and diffuse horizontal irradiance data is available. Such data can usually be found in EnergyPlus epw files in W/m².
- **4-Direct normal illuminance** for when measured (or calculated) direct normal and diffuse horizontal illuminance data is available. Such data can often be found in EnergyPlus epw files but DesignBuilder does not provide access to it directly. This is considered to provide the most accurate method if the required illuminance data is available.

Sky clearness (Epsilon)

Perez sky clearness (ϵ). Enter a value from 1.0 to 12.0.

Available only when using the **2-Perez all weather** [Sky method](#) and the **1-Brightness and clearness** [Perez method](#)

See: [Perez, R., R. Seals, and J. Michalsky, 1993. All-weather model for sky luminance distribution—Preliminary configuration and validation](#)

Sky brightness (Delta)

Perez sky brightness (Δ). Enter a value from 0.01 to 0.6.

Available only when using the **2-Perez all weather** [Sky method](#) and the **1-Brightness and clearness** [Perez method](#)

See: [Perez, R., R. Seals, and J. Michalsky, 1993. All-weather model for sky luminance distribution—Preliminary configuration and validation](#)

Direct normal irradiance

The direct normal solar broad spectrum irradiance (in W/m² or W/ft²). This value can be found from the site level hourly weather data.

Available only when using the **2-Perez all weather** [Sky method](#) and the **2-Direct normal irradiance** [Perez method](#).

Diffuse horizontal irradiance

The diffuse horizontal solar broad spectrum irradiance (in W/m² or W/ft²). This value can be found from the site level hourly weather data.

Available only when using the **2-Perez all weather** [Sky method](#) and the **2-Direct normal irradiance** or **3-Direct horizontal irradiance** [Perez method](#) have been selected.

Direct normal illuminance

The direct normal illuminance (visible spectrum) (in lux or fc).

Available only when using the **2-Perez all weather** [Sky method](#) and the **4-Direct normal illuminance** [Perez method](#)

Diffuse horizontal illuminance

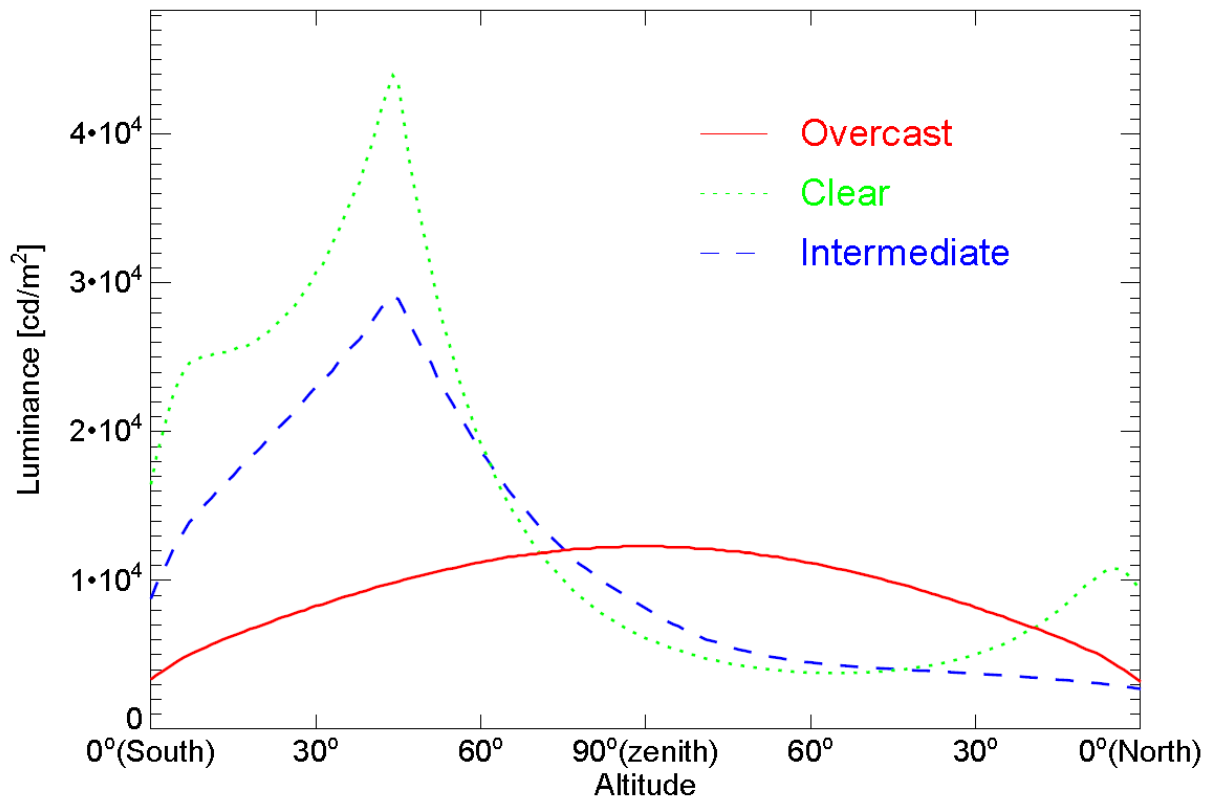
The diffuse horizontal illuminance (visible spectrum) (in lux or fc).

Available only when using the **2-Perez all weather** [Sky method](#) and the **4-Direct normal illuminance** [Perez method](#)

Sky model

When using the **1-Standard sky** [Sky method](#), select the sky model to be used for the daylighting calculations. Select from:

- **1-CIE sunny clear day**, the sky distribution corresponds to the standard CIE clear sky condition with additional direct illumination from the sun included. Expect very bright patches due to direct illumination from the sun with relatively dark areas where direct sunlight does not fall.
- **2-CIE clear day**, the standard CIE clear sky distribution *without* direct illumination from the sun.
- **3-CIE sunny intermediate day**, sky with intermediate conditions between the overcast and clear skies (see graph below) with *direct illuminance from the sun included*.
- **4-CIE intermediate day**, the CIE standard intermediate sky with illuminance distribution conditions between the overcast and clear skies (see graph below). *Direct illuminance from the sun is not included*.
- **5-CIE overcast day**, The CIE Standard Overcast Sky, originally known as the Moon and Spencer Sky, was devised to represent the luminance distribution observed for overcast skies. Adopted as a standard by the CIE in 1955, this description is the one most frequently used for illuminance modelling. In this model, the sky brightness increases gradually with altitude from the horizon to the zenith, but it does not vary with azimuth.
- **6-CIE overcast day (specify illuminance)** This option is similar to **5-CIE overcast day** described above but is scaled using a standard sky illuminance at the zenith. By default the zenith illuminance is set to 10,000 lux so that daylight factors can be calculated simply as working plane illuminance values divided by 100. This option is frequently used for daylight factor calculations. Other zenith illuminance values can be entered though.
- **7-Uniform cloudy sky** where the sky illumination is completely uniform.



Sky luminance profiles for 3 sky models (Source [Rendering with Radiance: A Practical Tool for Global Illumination](#))

Solar position

When using the **1-Standard sky** [Sky method](#) and one of the sunny or intermediate [Sky models](#) (options 1-4), you must define the position of the sun by entering the Month, Day of the month and Hour.

Point-in-time Options	
Window shading	1-No shading
Sky method	1-Standard sky
Sky model	1-CIE sunny clear day
Solar position	
Month	Sep
Day	21
Hour	9

Zenith illuminance

When the [Sky model](#) (above) is set to **6-CIE overcast day (scaled)** you can define the scaling illuminance at the Zenith (in lux or fc). The default is 10,000 lux which allows daylight factors to be calculated simply as working plane illuminance values divided by 100.

Daylighting Calculation Options - LEED

This topic covers the Daylighting calculation options specific to the **2-LEED** [Simulation type](#).

LEED Simulation Options

LEED version

Select from the supported versions of LEED:

- **1-LEED v2** - for NC 2.2 Credit EQ 8.1 (LEED v2)
- **2-LEED v3** - for Daylighting BD+C: New Construction v3
- **3-LEED v4** - for Daylighting BD+C: New Construction, Options 1 and 2
- **4-LEED v4.1** - for EQ Credit, Options 1 and 2

Perform LEED daylight option 1 (annual) simulation

When the LEED version is set to one of **1-LEED v4** or **2-LEED v4.1**, check this checkbox if you would like to run an annual sDA + ASE daylighting simulation suitable for assessing credits for Daylighting BD+C: New Construction, Option 1.

Perform LEED daylight option 2 (point in time) simulation

When the LEED version is set to one of **1-LEED v4** or **2-LEED v4.1**, check this checkbox if you would like to run a point in time illuminance daylighting simulation suitable for assessing credits for Daylighting BD+C: New Construction, Option 2.

Option 1 (Annual) Options

Dynamic shading method

When running LEED annual daylight simulations, Dynamic blinds are simulated according to the requirements set out in LM-83-12.

For an explanation of the Dynamic shading method options, see the "Dynamic shading method" on page 1984 article under the General Simulation type.

Override zone occupancy schedule

By default the occupancy schedule used for each zone in the Annual simulations are based on the zone [Occupancy schedule](#) defined on the Activity tab, however you can override this to use the same occupancy schedule for each zone by selecting this option and selecting the schedule in the browse control below.

Occupancy schedule

When the above **Override zone occupancy schedule** option is selected, you can select the occupancy schedule to be used to define when zones are occupied for all zones in the simulation. Values in the schedule greater than 0 indicate occupancy.

The default occupancy schedule provided by DesignBuilder does not include daylight saving time (DST). DST has to be added manually to the schedule. For example for the Northern Hemisphere you can use a schedule like the following:

```
Schedule:Compact,
8:00 - 18:00 Every day (DST Northern Hemisphere),
Fraction,
Through: 03/30,
For: AllDays,
Until: 08:00, 0,
Until: 18:00, 1,
Until: 24:00, 0,
Through: 10/26,
For: AllDays,
Until: 07:00, 0,
Until: 17:00, 1,
Until: 24:00, 0,
Through: 12/31,
```

```
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0;
```

Option 2 (Point-in-time) Options

Sky method

There are 2 methods for generating the sky illuminance distribution when running the LEED v2 and v4 the Radiance calculations:

- **1-Standard sky** - the default method where one of the predefined standard sky distributions are applied using the Radiance **gensky** tool.
- **2-Perez all weather** - an alternative method suitable for all weather types based on the Perez sky clearness (ϵ) and sky brightness (Δ) parameters and using the Radiance **gendaylit** tool. There are 3 Perez methods to choose from as described below.

For LEED v4 and v4.1 point in time calculations, the **2-Perez all weather** option is always used and this setting is not displayed.

Perez method

There are 4 methods to choose from.

- **1-Brightness and clearness** - if you know the Perez sky clearness (ϵ) and sky brightness (Δ) parameters of the sky to be analysed choose this option.
- **2-Direct normal irradiance** for when measured direct normal and diffuse horizontal irradiance data is available. This option can be useful since it allows readily available solar data in W/m² from the EnergyPlus weather file to be used to define sky illuminance. Go to the Simulation tab, navigate to site level and select hourly data to see solar data suitable for the location of your building for any hour of the year. Note that using full spectrum solar data to define illuminance is seen by daylighting purists as less accurate than using measured values of actual illuminance (see option 4).
- **3-Direct horizontal irradiance** for when measured (or calculated) direct horizontal and diffuse horizontal irradiance data is available. Such data can usually be found in EnergyPlus epw files in W/m².
- **4-Direct normal illuminance** for when measured (or calculated) direct normal and diffuse horizontal illuminance data is available. Such data can often be found in EnergyPlus epw files but DesignBuilder does not provide access to it directly. This is considered to provide the most accurate method if the required illuminance data is available.

The sky properties must be set separately for morning and afternoon using data that depends on the **Perez method** selection .

Sky clearness (Epsilon)

Perez sky clearness (ϵ). Enter a value from 1.0 to 12.0.

Available only when using the **2-Perez all weather** [Sky method](#) and the **1-Brightness and clearness** [Perez method](#)

See: [Perez, R., R. Seals, and J. Michalsky, 1993. All-weather model for sky luminance distribution—Preliminary configuration and validation](#)

Sky brightness (Delta)

Perez sky brightness (Δ). Enter a value from 0.01 to 0.6.

Available only when using the **2-Perez all weather** [Sky method](#) and the **1-Brightness and clearness** [Perez method](#)

See: [Perez, R., R. Seals, and J. Michalsky, 1993. All-weather model for sky luminance distribution—Preliminary configuration and validation](#)

Direct normal irradiance

The direct normal solar broad spectrum irradiance (in W/m² or W/ft²). This value can be found from the site level hourly weather data.

Available only when using the **2-Perez all weather** [Sky method](#) and the **2-Direct normal irradiance** [Perez method](#).

Diffuse horizontal irradiance

The diffuse horizontal solar broad spectrum irradiance (in W/m² or W/ft²). This value can be found from the site level hourly weather data.

Available only when using the **2-Perez all weather** [Sky method](#) and the **2-Direct normal irradiance** or **3-Direct horizontal irradiance** [Perez method](#) have been selected.

Direct normal illuminance

The direct normal illuminance (visible spectrum) (in lux or fc).

Available only when using the **2-Perez all weather** [Sky method](#) and the **4-Direct normal illuminance** [Perez method](#)

Diffuse horizontal illuminance

The diffuse horizontal illuminance (visible spectrum) (in lux or fc).

Available only when using the **2-Perez all weather** [Sky method](#) and the **4-Direct normal illuminance** [Perez method](#)

LEED v4 and v4.1 Option 2 Sky Settings

An important part of the LEED v4 Option 2 daylight calculation is to set the sky conditions to be used in the simulations. This must be done using the **2-Perez all weather** [Sky method](#). The procedure required for this is described by USGBC:

Calculate illuminance intensity for sun (direct component) and sky (diffuse component) for clear-sky conditions as follows:

- Use typical meteorological year data, or an equivalent, for the nearest available weather station.
- Select one day within 15 days of September 21 and one day within 15 days of March 21 that represent the clearest sky condition.
- Use the average of the hourly value for the two selected days.
- Exclude blinds or shades from the model. Include any permanent interior obstructions. Movable furniture and partitions may be excluded.

Method 1 using Direct horizontal irradiance

Based on LEED documentation:

1. Obtain typical meteorological year data. Use local weather data or TMY weather data files for the nearest city.
2. To calculate the illuminance intensity for sun (direct component) and sky (diffuse component) in a TMY2 or TMY 3 file, you can export the data into spreadsheet format.
3. From the TMY, select the day within 15 days of September 21 that has the clearest sky condition (total sky cover at its lowest value) at 9 A.M.

4. From the TMY, select the day within 15 days of March 21 that has the clearest sky condition at 9 A.M.
5. Determine the direct horizontal irradiance (W/m²) values at 9 A.M. for the day selected in September and at 9 A.M. for the day selected in March. Average the two values and use the result in the 9 A.M. simulation as the direct horizontal irradiance input. If the file does not explicitly state direct horizontal irradiance, calculate it as follows:

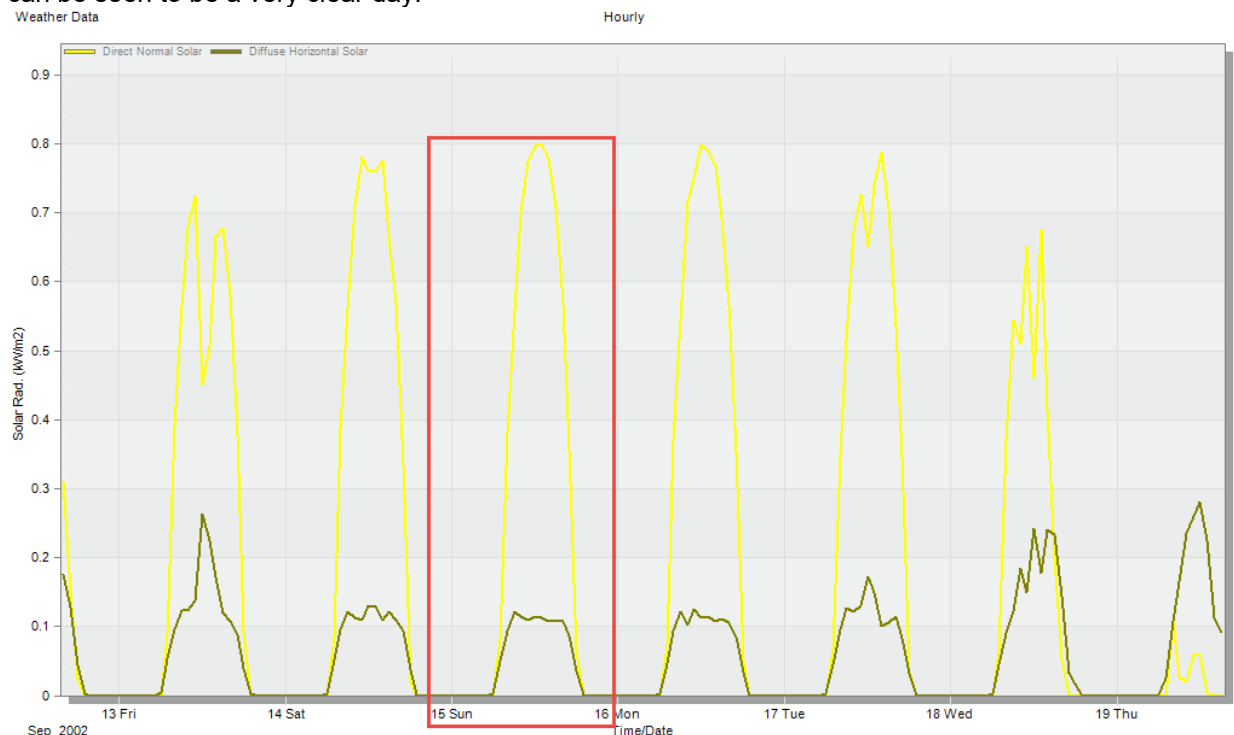
$$\text{Direct horizontal irradiance} = \text{Global horizontal irradiance} - \text{Diffuse horizontal irradiance}$$

6. Determine the diffuse horizontal irradiance (W/m²) values at 9 A.M. for the day selected in September and at 9 A.M. for the day selected in March. Average the two values and use the result in the 9 A.M. simulation as the diffuse horizontal irradiance input.
7. Repeat procedures 3–6 for 3 P.M.
8. Choose the [2-Perez all weather Sky method](#) (this is done behind the scenes for LEED v4 and v4.1).
9. Choose the [3-Direct horizontal irradiance Perez method](#).
10. Input the custom values for [Direct horizontal irradiance](#) and [Diffuse horizontal irradiance](#) in the Calculation options dialog and run the Radiance simulation.

Method 2 using Direct normal irradiance

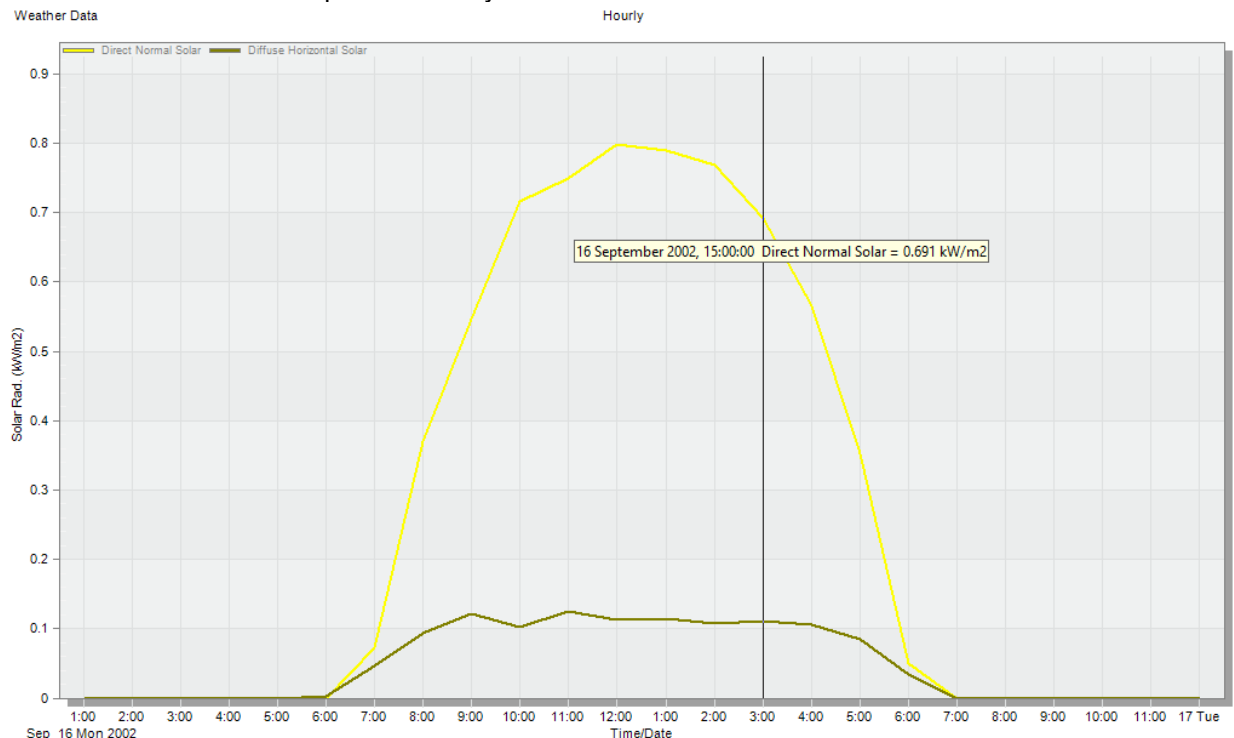
Another (simpler) procedure to provide the sky condition data within DesignBuilder is as follows:

1. Click on the [Simulation](#) tab and go to Site level to view the site solar radiation data. To focus on this click on the Detailed tab of the Display options panel, scroll down to the **Site Weather** settings and check only the **Direct normal solar** and the **Diffuse horizontal solar** checkboxes. Click on the Save options for future use display options toolbar button and save the settings.
2. Scroll to view September data and identify the clearest day within 15 days of 21 September. This might be by checking to see the highest value of beam solar radiation. For example 16 September can be seen to be a very clear day:



3. Change to view 1 day at a time by entering 1 in the [Days per page](#) text box and pressing the [Update](#) display options toolbar button.

4. Scroll to 16 September and write down the **Direct normal solar** and the **Diffuse horizontal solar** W/m² values at 9am and 3pm on this day.



5. Repeat the above process for March to find the **Direct normal solar** and the **Diffuse horizontal solar** W/m² values at 9am and 3pm for the clearest day within 15 days of 21 March.
6. Calculate average **Direct normal solar** and the **Diffuse horizontal solar** values for 9am and 3pm for both the March and September days.
7. Choose the **2-Perez all weather Sky method** (this is done behind the scenes for LEED v4 and v4.1).
8. Choose the **2-Direct normal irradiance Perez method**.
9. Enter these calculated **Direct normal irradiance** and **Diffuse horizontal irradiance** values in the Calculation options dialog and run the Radiance simulation.

Daylighting Calculation Options - BREEAM

This topic describes the BREEAM-specific Daylighting Calculation options which can be accessed when the [Simulation type](#) is set to **3-BREEAM**.

BREEAM Simulation Options

Perform BREEAM HEA option 4b (annual) simulation

Check this checkbox if you would like to run an annual daylighting simulation suitable for assessing credits for BREEAM Health and Wellbeing Credit HEA 01 option 4b .

Perform BREEAM HEA option 4a (point in time) simulation

Check this checkbox if you would like to run a point in time illuminance daylighting simulation suitable for assessing credits for BREEAM Health and Wellbeing Credit HEA 01 option 4a .

Option 4b (Annual) Options

Override zone occupancy schedule

By default the occupancy schedule used for each zone in the Annual simulations are based on the zone [Occupancy schedule](#) defined on the Activity tab, however you can override this to use the same occupancy schedule for each zone by selecting this option and selecting the schedule in the browse control below.

Occupancy schedule

When the above **Override zone occupancy schedule** option is selected, you can select the occupancy schedule to be used to define when zones are occupied for all zones in the simulation. Values in the schedule greater than 0 indicate occupancy.

The default occupancy schedule provided by DesignBuilder does not include daylight saving time (DST). DST has to be added manually to the schedule. For example for the Northern Hemisphere you can use a schedule like the following:

```
Schedule:Compact,  
8:00 - 18:00 Every day (DST Northern Hemisphere),  
Fraction,  
Through: 03/30,  
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0,  
Through: 10/26,  
For: AllDays,  
Until: 07:00, 0,  
Until: 17:00, 1,  
Until: 24:00, 0,  
Through: 12/31,  
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0;
```

Min illuminance threshold

For **BREEAM HEA 01 Option 4b**, you can enter a minimum illuminance threshold value from Table 12 based on the area type of the zones being analysed.

Note: You can view BREEAM HEA01 2016 Option b Table 12 [here](#). If your project requires a different version of BREEAM, please make sure to use the table values from the appropriate documentation.

Avg illuminance threshold

For **BREEAM HEA 01 Option 4b**, you can enter an average illuminance threshold value from [Table 12](#) based on the area type of the zones being analysed. See note above regarding BREEAM versions.

Shading

BREEAM annual daylighting calculations do not apply LM-83-12 dynamic shading or any other window shading algorithms. Any [Window shading](#) settings on the Openings tab are ignored.

Option 4a (Point-in-time) Options

In general you should make the following settings for BREEAM point in time daylighting calculations.

- Sky method: **1-Standard sky**
- Sky model: **6-CIE overcast day (specify illuminance)**
- Zenith illuminance: 10000

Sky method

There are 2 general methods for generating the sky illuminance distribution for the Radiance calculation:

- **1-Standard sky** - the default method where one of the predefined standard sky distributions are applied using the Radiance **gensky** tool.
- **2-Perez all weather** - an alternative method suitable for all weather types based on the Perez sky clearness (ϵ) and sky brightness (Δ) parameters and using the Radiance **gendaylit** tool. There are 3 Perez methods to choose from as described below.

Perez method

When the Sky method is set to **2-Perez all weather** there are 4 methods to choose from depending on the data you have available. In all cases the time, day and month of the analysis must be set to define the location of the sun in the sky.

- **1-Brightness and clearness** - if you know the Perez sky clearness (ϵ) and sky brightness (Δ) parameters of the sky to be analysed choose this option.
- **2-Direct normal irradiance** for when measured direct normal and diffuse horizontal irradiance data is available. This option can be useful since it allows readily available solar data in W/m² from the EnergyPlus weather file to be used to define sky illuminance. Go to the Simulation tab, navigate to site level and select hourly data to see solar data suitable for the location of your building for any hour of the year. Note that using full spectrum solar data to define illuminance is seen by daylighting purists as less accurate than using measured values of actual illuminance (see option 4).
- **3-Direct horizontal irradiance** for when measured (or calculated) direct horizontal and diffuse horizontal irradiance data is available. Such data can usually be found in EnergyPlus epw files in W/m².
- **4-Direct normal illuminance** for when measured (or calculated) direct normal and diffuse horizontal illuminance data is available. Such data can often be found in EnergyPlus epw files but DesignBuilder does not provide access to it directly. This is considered to provide the most accurate method if the required illuminance data is available.

Sky clearness (Epsilon)

Perez sky clearness (ϵ). Enter a value from 1.0 to 12.0.

Available only when using the **2-Perez all weather** [Sky method](#) and the **1-Brightness and clearness** [Perez method](#)

See: [Perez, R., R. Seals, and J. Michalsky, 1993. All-weather model for sky luminance distribution—Preliminary configuration and validation](#)

Sky brightness (Delta)

Perez sky brightness (Δ). Enter a value from 0.01 to 0.6.

Available only when using the **2-Perez all weather** [Sky method](#) and the **1-Brightness and clearness** [Perez method](#)

See: [Perez, R., R. Seals, and J. Michalsky, 1993. All-weather model for sky luminance distribution—Preliminary configuration and validation](#)

Direct normal irradiance

The direct normal solar broad spectrum irradiance (in W/m² or W/ft²). This value can be found from the site level hourly weather data.

Available only when using the **2-Perez all weather** [Sky method](#) and the **2-Direct normal irradiance** [Perez method](#).

Diffuse horizontal irradiance

The diffuse horizontal solar broad spectrum irradiance (in W/m² or W/ft²). This value can be found from the site level hourly weather data.

Available only when using the **2-Perez all weather Sky method** and the **2-Direct normal irradiance** or **3-Direct horizontal irradiance Perez method** have been selected.

Direct normal illuminance

The direct normal illuminance (visible spectrum) (in lux or fc).

Available only when using the **2-Perez all weather Sky method** and the **4-Direct normal illuminance Perez method**

Diffuse horizontal illuminance

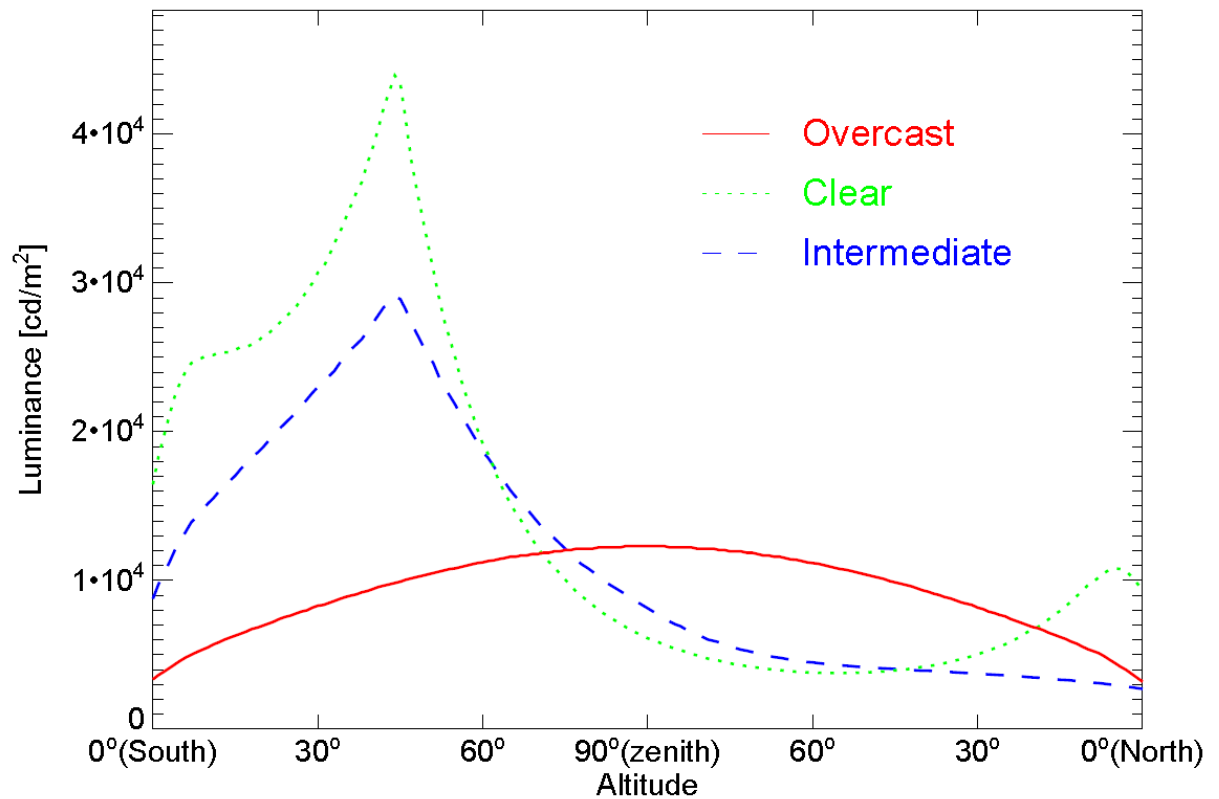
The diffuse horizontal illuminance (visible spectrum) (in lux or fc).

Available only when using the **2-Perez all weather Sky method** and the **4-Direct normal illuminance Perez method**

Sky model

When using the **1-Standard sky Sky method**, select the sky model to be used for the daylighting calculations. Select from:

- **1-CIE sunny clear day**, the sky distribution corresponds to the standard CIE clear sky condition with additional direct illumination from the sun included. Expect very bright patches due to direct illumination from the sun with relatively dark areas where direct sunlight does not fall.
- **2-CIE clear day**, the standard CIE clear sky distribution *without* direct illumination from the sun.
- **3-CIE sunny intermediate day**, sky with intermediate conditions between the overcast and clear skies (see graph below) with *direct illuminance from the sun included*.
- **4-CIE intermediate day**, the CIE standard intermediate sky with illuminance distribution conditions between the overcast and clear skies (see graph below). *Direct illuminance from the sun is not included*.
- **5-CIE overcast day**, The CIE Standard Overcast Sky, originally known as the Moon and Spencer Sky, was devised to represent the luminance distribution observed for overcast skies. Adopted as a standard by the CIE in 1955, this description is the one most frequently used for illuminance modelling. In this model, the sky brightness increases gradually with altitude from the horizon to the zenith, but it does not vary with azimuth.
- **6-CIE overcast day (specify illuminance)** This default option is similar to **5-CIE overcast day** described above but is scaled using a standard sky illuminance at the zenith. By default the zenith illuminance is set to 10,000 lux so that daylight factors can be calculated simply as working plane illuminance values divided by 100. This option is frequently used for daylight factor calculations. Other zenith illuminance values can be entered though.
- **7-Uniform cloudy sky** where the sky illumination is completely uniform.



Sky luminance profiles for 3 sky models (Source [Rendering with Radiance: A Practical Tool for Global Illumination](#))

Solar position

When using the **1-Standard sky** [Sky method](#) and one of the sunny or intermediate [Sky models](#) (options 1-4), you must define the position of the sun by entering the Month, Day of the month and Hour.

Point-in-time Options	
Window shading	1-No shading
Sky method	1-Standard sky
Sky model	1-CIE sunny clear day
Solar position	
Month	Sep
Day	21
Hour	9

Zenith illuminance

When the [Sky model](#) (above) is set to **6-CIE overcast day (scaled)** you can define the scaling illuminance at the Zenith (in lux or fc). The default is 10,000 lux which allows daylight factors to be calculated simply as working plane illuminance values divided by 100.

Daylighting Calculation Options - LM-83-12

This topic covers the Daylighting calculation options specific to the **4-LM-83-12** [Simulation type](#).

LM-83-12 Simulation Options

Dynamic shading method

For an explanation of the Dynamic shading method options, see the "Dynamic shading method" on page 1984 article.

Override zone occupancy schedule

By default the occupancy schedule used for each zone in the Annual simulations are based on the zone [Occupancy schedule](#) defined on the Activity tab, however you can override this to use the same occupancy schedule for each zone by selecting this option and selecting the schedule in the browse control below.

Occupancy schedule

When the above **Override zone occupancy schedule** option is selected, you can select the occupancy schedule to be used to define when zones are occupied for all zones in the simulation. Values in the schedule greater than 0 indicate occupancy.

The default occupancy schedule provided by DesignBuilder does not include daylight saving time (DST). DST has to be added manually to the schedule. For example for the Northern Hemisphere you can use a schedule like the following:

```
Schedule:Compact,  
8:00 - 18:00 Every day (DST Northern Hemisphere),  
Fraction,  
Through: 03/30,  
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0,  
Through: 10/26,  
For: AllDays,  
Until: 07:00, 0,  
Until: 17:00, 1,  
Until: 24:00, 0,  
Through: 12/31,  
For: AllDays,  
Until: 08:00, 0,  
Until: 18:00, 1,  
Until: 24:00, 0;
```

sDA Thresholds

sDA lower illuminance threshold

Enter the lower illuminance threshold, the minimum level of illuminance (in lux or fc) below which lighting levels are considered to be inadequate. A typical lower threshold value for sDA calculations is 300 lux.

ASE Thresholds

ASE illuminance threshold

Enter the maximum level of illuminance (in lux or fc) above which lighting levels are considered to be likely to lead to visual discomfort, i.e. glare. A typical upper threshold for ASE calculations is 1000 lux.

Daylighting Calculation Options - Right To Light

This topic covers the Daylighting calculation options specific to the **5-Right to light (BS8206)** [Simulation type](#).

Right to Light was formerly defined in the BS 8206 standard, now updated to the BS EN 17037:2018. DesignBuilder currently supports the BS 8206 standard, some aspects of which are transferable (but not equivalent) to the updated standard. Four types of Right to Light daylight calculations are available:

1. Vertical Sky Component (VSC)
2. Annual probable sun hour (APSH)
3. No-skyline
4. Average daylight factor

When simulation type is set as: **5-Right to light**, the relevant simulation options are shown and the required working plane height is automatically set to 0.85m in accordance to the standard.

You can generate reports for 4 types of Right to Light assessments by checking the appropriate checkboxes as described below.

Perform vertical sky component simulation

Check this checkbox to carry out a [vertical sky component](#) simulation.

Perform annual probable sunlight hours simulation

Check this checkbox to carry out an [annual probable sunlight hours](#) simulation.

Perform no sky line simulation

Check this checkbox to carry out a [no sky line](#) simulation.

Note: For **No Sky Line** simulations you should select the greatest available detail for the [Sky patches](#) setting, option **6-5185**.

Perform average daylight factor simulation

Check this checkbox to carry out an [average daylight factor](#) simulation.

Daylighting Calculation Options - Green Star

This topic covers the Daylighting calculation options specific to the **6-Green star** [Simulation type](#).

Green Star Simulation Options

Please refer to the [Point in time](#) settings for the **1-General** [Simulation type](#) for information on the following Green Star settings:

- Sky method
- Perez method
- Sky clearness (Epsilon)
- Sky brightness (Delta)
- Direct normal irradiance
- Diffuse horizontal irradiance
- Direct normal illuminance
- Diffuse horizontal illuminance
- Sky model
- Solar position
- Zenith illuminance

Daylighting Outputs And Display Options

This section describes the available daylighting outputs and the corresponding Display options. The Display options panel at the bottom left of the screen is used to select which results are to be displayed.

View

Simulation type

Select the type of simulation results to be viewed. Generally, you should select the same [Simulation type](#) as selected on the Calculation options dialog.

The following options are available:

- **1-General** - access results generated by any previous **1-General** daylighting simulations that have been run.
- **2-LEED** - provides access to LEED daylighting reports for any LEED v2 through to v4.1 including option 1 (annual) and option 2 (point in time) simulations that have been carried out.
- **3-BREEAM** - provides access to BREEAM HEA Options 4a (point in time) and 4b (annual) reports.
- **4-LM-83-12** - the option to choose for sDA, ASE and UDI simulations when they are not for LEED or BREEAM.
- **5-Right to light (BS8206)** - Annual probable sunlight hours (APSH), Vertical sky component (VSC), No sky line (NSL) and Average daylight factor (ADF) reports can be viewed.
- **6-Green Star** - Green Star daylight factor reports.

These options together with their associated Display options are described in more detail below.

Note: The corresponding daylight simulation must have been carried out to view results for any of the above. For example, you will only see results for the **1-General** simulation type if you have previously run a calculation using the equivalent **1-General** [Simulation type](#) set on the Calculation Options dialog.

Several Daylighting Display options depend on the Simulation type selected as detailed in the topics listed below.

- [General](#)
- [LEED](#)
- [BREEAM](#)
- [LM-83-12](#)
- [Right to Light](#)
- [Green Star](#)

The options described below apply to all daylight map outputs.

Show Object Options

This set of "Show.." object options are available when one of the map options has been selected.

Show component blocks

Tick this checkbox if you would like building or block level [component blocks](#) to be displayed with the model.

Show assemblies

Tick this checkbox if you would like [assemblies](#) to be displayed with the model.

Show shading planes

Tick this checkbox if you would like [shading planes](#) to be displayed with the model.

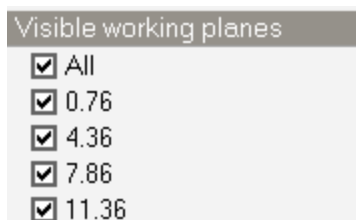
Show North arrow

Tick this checkbox if you would like the North arrow to be displayed on the ground plane.

Visible Working Planes

The **Visible Working Planes** settings are available when the **Report type** is set to one of the map options.

When viewing map data for a building, block or zone, there can be multiple working planes at various heights. DesignBuilder therefore provides a list of all relevant working plane heights in the current building, block or zone to allow you to select which are to be displayed. The working plane heights listed are relative to z=0 site ground level.



Selecting the **All** option toggles all working planes to be checked or unchecked depending on current state.

Zone Labels

The Zone Labels settings are available when the **Report type** is set to one of the map options.

Show zone labels

Check this option if you wish zones to be labelled with their name in the Map view.

Result Values

The Cell Values settings are available when the [Report type](#) is set to one of the Map options.

Show values

Choose whether or not to show numeric values printed within each grid cell. Tick the checkbox to display numeric values.

Note: for small grid cells (or large spaces with lots of grid cells) there may not be enough space to comfortably fit the numeric values, in which case it is usually best to leave this option unchecked.

Font colour mode

There are options to allow you to configure the font colour used to display cell value numeric text. Select from:

- **1-Automatic** where the font colour is chosen for each cell individually to ensure strong contrast with the background colour.
- **2-User specified** in which case you can select a single fixed colour for the numeric text.

Font colour

When the Font colour mode is set to **2-User specified** you can select the font colour manually using this control.

Colour

The Colour settings are available when the **Report type** is set to one of the map options.

Palette

Select the palette to be used from the options:

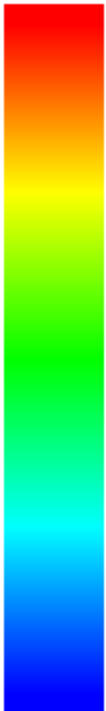
- **1-Red to Black** - where Red is used for brightest parts of the working plane and Black is used for the darkest. This is the default option.



- **2-Red to Yellow** - where Red is used for brightest parts of the working plane and Yellow is used for the darkest:



- **3-Red to Blue** - where Red is used for brightest parts of the working plane and Blue is used for the darkest. This is the palette used in DesignBuilder v5.0 and earlier:



- **4-White to Black** - where White is used for brightest parts of the working plane and Black is used for the darkest.



Daylighting Display Options - General

This topic covers the Daylighting display options specific to the **1-General** [Simulation type](#).

Option

Select the reports you would like to display. Options for the **1-General** Simulation type are:

- **1-Point in time** - show illuminance and daylight factor results.
- **2-Annual** - show annual sDA, ASE, UDI results.

Report type

When the **1-Point in time** [Option](#) setting is made, the following report types are available:

- **1-Map** - a daylight distribution contour map on the working plane of the current object is displayed.
- **2-Grid** - a table of summary daylight factor and illuminance daylight statistics for each zone is displayed.

When the **2-Annual** [Option](#) setting is made, the following report types are available:

- **1-DA map** - shows the distribution of annual hours on the working plane for which the DA ([Daylight Autonomy](#)) criteria are met.
- **2-ASE map** - shows the distribution of annual hours on the working plane for which the ASE ([Annual Sunlight Exposure](#)) criteria are met.
- **3-UDI map** - shows the distribution of annual hours on the working plane for which the UDI ([Useful Daylight Illuminance](#)) criteria are met.
- **4-Grid** - a table of summary annual daylight statistics is displayed for each zone.

Margin

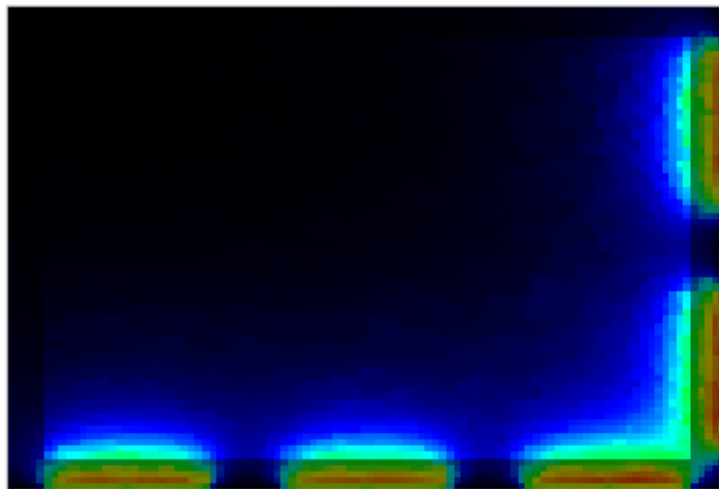
If a non-zero [margin](#) was specified in the calculation options, you can choose whether or not to include data points within the margin in the selected reports. If the margin calculation option was set to zero you can ignore these options.

The margin display options depends on the [Report type](#) setting.

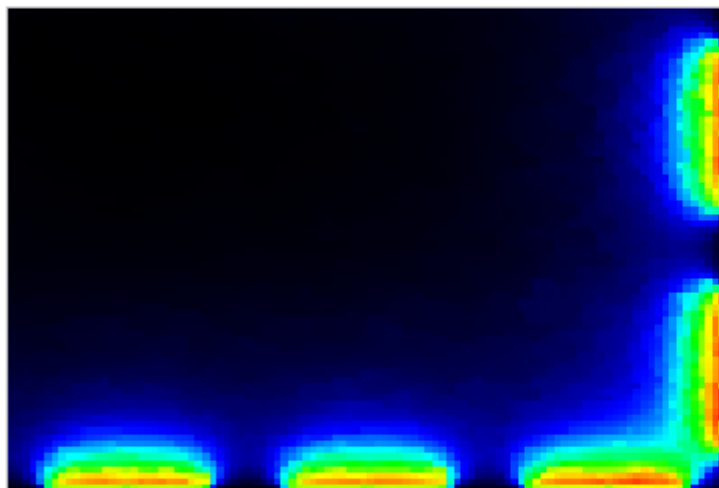
Map Margin option

When the [Report type](#) is set to one of the map options, you can choose how the margin region at the edges of the working plane is displayed on the map. Choose from the options below.

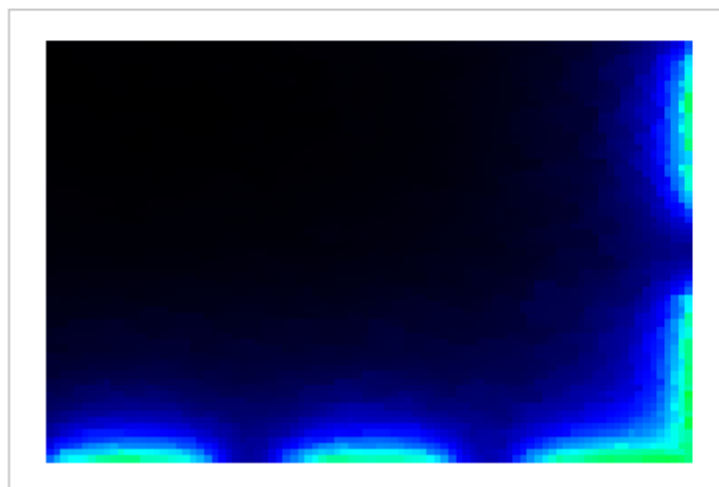
- **1-Shaded** - results in the margin are plotted, but the colours are darkened.
- **2-Not shaded** - results in the margin are plotted the same as for the main area of the working plane.
- **3-Hidden** - no results are plotted in the margin region which is displayed in white.



1-Shaded - Colours in margin area darkened



2-Shaded - Colours in margin area plotted as for rest of working plane



3-Hidden - Data in margin area not plotted

Grid/Report Margin options

When the **1-Point in time Option** setting is made and the **Report type** is set to **2-Grid**, you can choose whether or not to include data points within that margin for the average and or min/max daylight factor calculations in summary reports and grids.

Include margin area in average daylight factor calculation

Check this option to include all areas in the average daylight factor summary results. Uncheck it to exclude data points in the margin.

Include margin area in min/max daylight factor calculation

Check this option to include all areas in the min/max daylight factor summary results. Uncheck it to exclude data points in the margin.

Metric Pass Thresholds (Annual)

When viewing annual results and the **2-Grid** Report type is set, the annual daylighting metric pass thresholds should be checked and, if necessary, changed.

Note: The annual daylighting thresholds are used to define acceptability and are split into two parts:

- a. the [illuminance thresholds](#) which define the acceptable level of lighting at any one point in time over the year, and are defined on the [Calculation options dialog](#), and,
- b. metric pass thresholds, which are defined here the Display options panel, as described below.

Changing any of the Metric pass threshold settings in the Display options will typically change the **Area in Range** values shown on the grid.

sDA percentage annual hours

Enter the % of the occupied year pass threshold used for each cell in the [sDA calculations](#). sDA is usually defined as sDA_{300,50%}, which means that a given grid cell must achieve at least 300 lux ([sDA lower illuminance threshold](#)) for at least 50% ([sDA Percentage annual hours](#)) of the occupied year for its area to be considered to "pass" the criteria. The overall sDA metric for a zone is the % of the floor area that passes in this way.

ASE annual hours

Enter the number of occupied hours in the year pass threshold used for each cell in the [ASE calculations](#). ASE_{1000,250} means for a given cell the illuminance must not be greater than the 1000 lux threshold ([ASE illuminance threshold](#)) for more than 250 hours ([ASE annual hours](#)) in the (occupied) year to pass.

UDI percentage annual hours

Enter the % of the occupied year pass threshold used for each cell in the [UDI calculations](#). A pass threshold value of 50% means that a given cell must achieve an illuminance of between 100 and 3000 lux (or whatever values were specified as the lux bounds in Calculation options) for at least 50% of the occupied hours in the year.

Thresholds (Point In Time)

When viewing point in time results and the **2-Grid** Report type is set, the illuminance daylighting pass thresholds should be checked and, if necessary, changed.

Threshold type

Select from:

- **1-Daylight factor**, set the min average daylight factor threshold.
- **2-Illuminance**, set the min and max illuminance threshold values.

Min average DF threshold

When the Threshold type is set to **1-Daylight factor**, set the minimum average daylight factor to be used in calculating the **Floor Area Within Limits** grid outputs.

Min illuminance

When the Threshold type is set to **2-Illuminance**, set the minimum illuminance to be used in calculating the **Floor Area Within Limits** grid outputs.

Max illuminance

When the Threshold type is set to **2-Illuminance**, set the maximum illuminance to be used in calculating the **Floor Area Within Limits** grid outputs.

Scale

The Scale settings are available when the [Report type](#) is set to one of the map options:

Source

The scale of the daylight maps can be defined using data from the building, block or zone level or it can be entered manually. It is a way to define the minimum and maximum values on the scale. Select from one of the following options.

- **1-Building**, where all of the values calculated for the whole building are scanned and the lowest and highest values are used. This allows plots to be generated using a consistent scale even when displaying data at block or zone level.
- **2-Block**, where all of the values calculated for the current block are scanned and the lowest and highest values for are used. This allows plots to be generated using a consistent scale within a block even when displaying data at zone level.
- **3-Zone**, where the scale is tailored to the current zone allowing in many cases a more even distribution of colours through the map. When using this option, images generated for each zone will use a different scale and so will not be so easily comparable as they would be if one of the **1-Building** or **2-Block** options had been used.
- **4-User defined**, where you can enter the minimum and maximum values on the scale manually. This has the advantage of allowing a specific lower and upper scale value to be used. It also allows you to define colours to represent areas that exceed the maximum or are lower than the minimum. For example, for many point-in-time daylighting calculations, daylight factors lower than 2% are considered to be inadequately daylit. These areas can be highlighted in a special colour to help provide a clear visual indication of the compliant and non-compliant areas.

Value source

When the **1-Point in time** [Option](#) setting is made, **Value source** defines whether to view Daylight factor or Illuminance results. Select from:

- **1-Daylight factor** - daylight distribution maps are plotted as daylight factors.
- **2-Illuminance** - daylight distribution maps are displayed based on illuminance data.

When the **2-Annual** [Option](#) setting is made, **Value source** defines whether to view results as Annual hours or Percentage annual hours. Select from:

- **1-Annual hours** - daylight distribution maps are plotted as the number of annual hours for which the selected criteria are met.
- **2-Percentage annual hours** - daylight distribution maps are plotted as the % of annual hours for which the selected criteria are met. The denominator used in the % calculation is the total number of occupied hours.

Note: when [Source](#) is set to **4-User defined**, **Value source** is hard set to **1-Annual hours**.

User-Defined Scale Settings

Low value

When using the **4-User defined** Scale source you can enter the minimum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

High value

When using the **4-User defined** Scale source you can enter the maximum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

Apply scale "too dark" colour

If you wish to set a specific colour to represent the areas of the working plane that have values below the entered **Low value** criterion then check this box and define the colour in the colour selector below.

Scale "too dark" colour

Select the colour to be used to represent areas of the working plane that are *below* the entered **Low value**. Typically a dark grey or black colour might be used to show areas of the working plane that are inadequately daylit.

Apply scale "too bright" colour

If you wish to set a specific colour to represent areas of the working plane that are *above* the entered **High value** then check this box and define the colour in the colour selector below.

Scale "too bright" colour

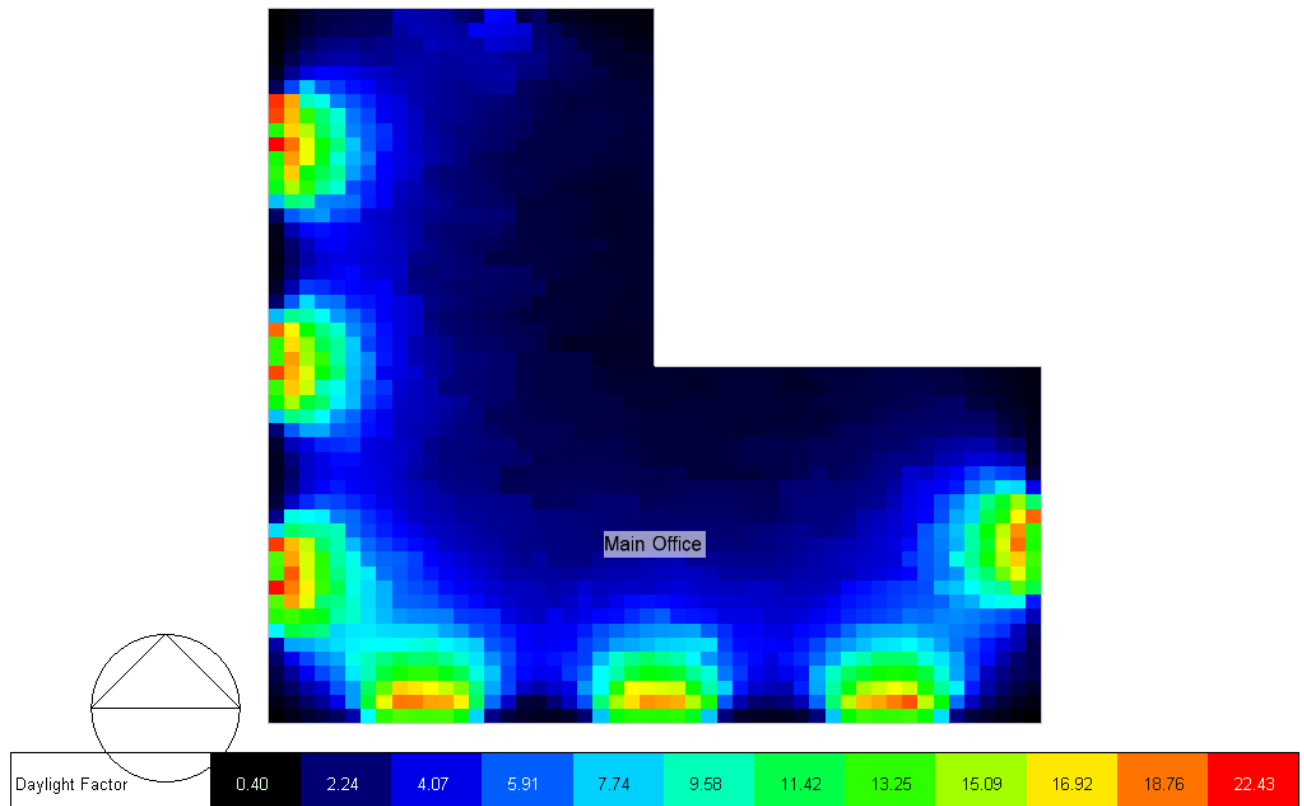
Select the colour to be used to represent areas of the working plane that are *above* the entered **High value**.

Point In Time Illuminance Results

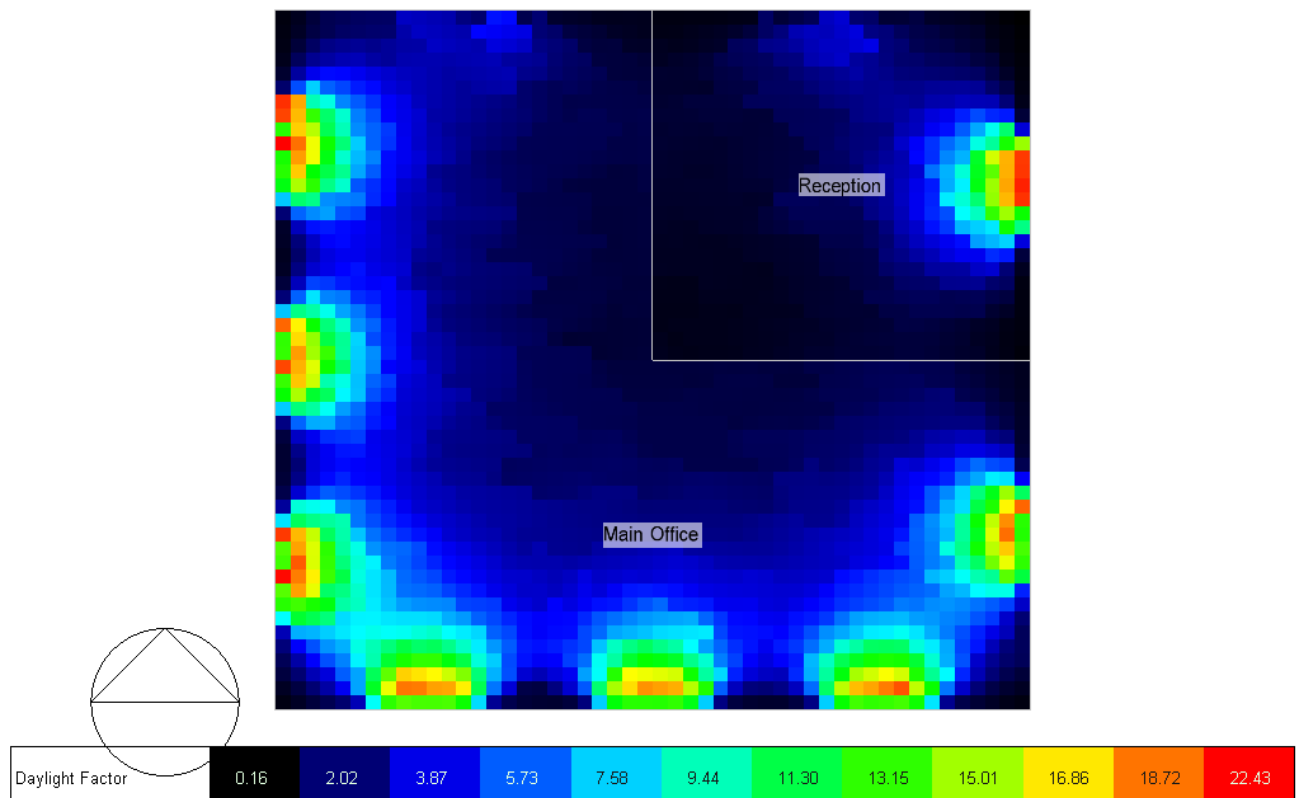
This section explains the results generated when the **1-Point in time Option** setting is made.

Map - Daylight Illuminance Distribution

When the **Report type** Display option is set to **1-Map** a contour map showing the distribution of the selected output across the working plane of the object is displayed. An example daylight map for a single zone is shown below.



When at building and block levels maps of daylight distribution across the working plane of the current object can be displayed. An example daylight distribution map displayed across multiple zones is shown below.



Grid

When the **Report type** Display option is set to **2-Grid**, a grid displays information on average daylight factors, daylight uniformity and illuminance values for each zone together with a total for the building. The following data is calculated for each zone:

- **Floor Area (m2 or ft2)**
- **Floor Area within Limits (m2 or ft2)** - the floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.
- **Floor Area within Limits (%)** - the % floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.
- **Average Daylight Factor (%)**
- **Minimum Daylight Factor (%)**
- **Maximum Daylight Factor (%)**
- **Uniformity ratio (Min / Avg)**
- **Uniformity ratio (Min / Max)**
- **Min Illuminance (lux or fc)**
- **Max Illuminance (lux or fc)**

This option can be especially useful if the built-in daylight reports don't cover the particular calculation you need to make. In this case you can export the summary data for each zone for further analysis in a spreadsheet.

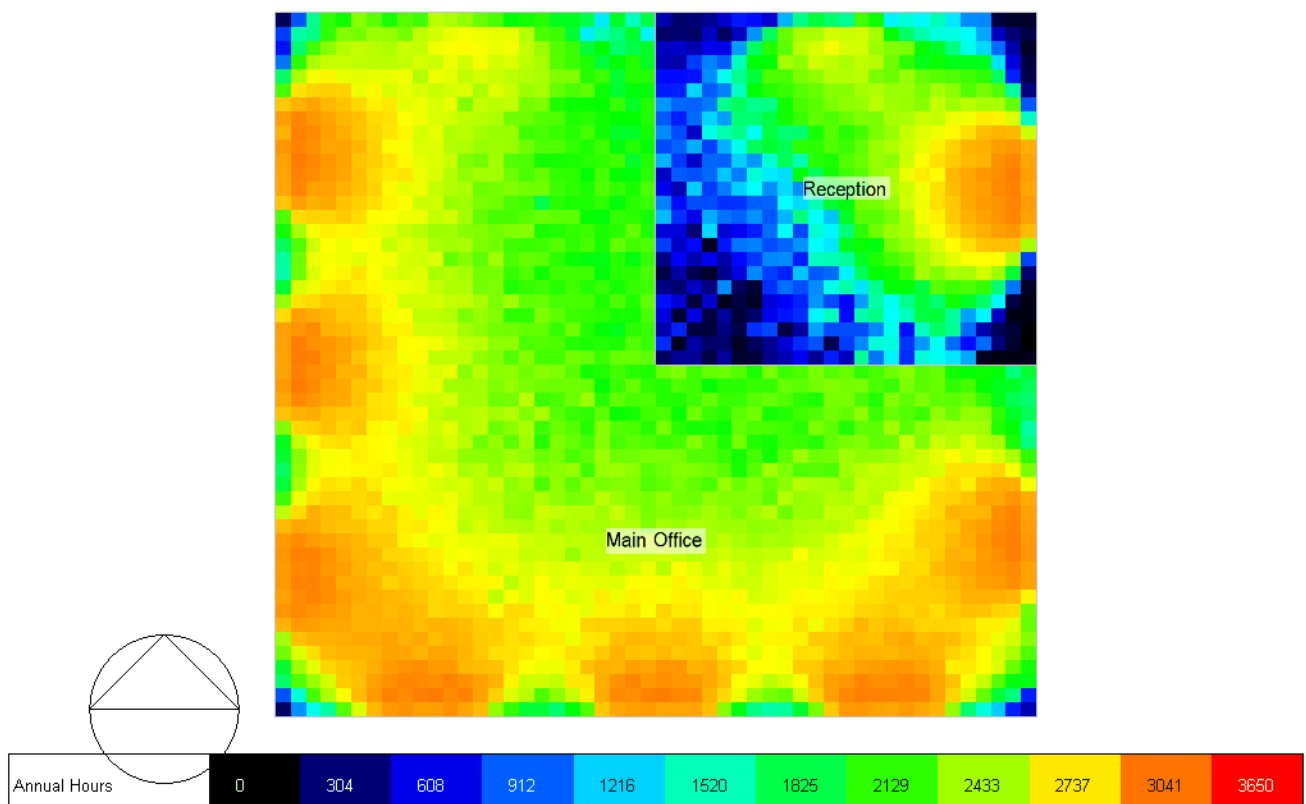
An example grid output is shown below.

Block	Zone	Floor Area (m2)	Floor Area within Li...	Floor Area within Li...	Average Daylight F...	Minimum Daylight F...	Maximum Daylight F...	Uniformity ratio (Min...	Uniformity ratio (Min...	Min Illuminance (lux)	Max Illuminance (lux)
First Floor	Room	100.000	85.032	85.032	7.668	0.781	24.763	0.102	0.032	78.26	2481.10
Extension	Room	50.000	49.905	99.811	9.264	1.850	25.508	0.200	0.073	185.69	2560.26
Ground Floor	South	50.000	49.811	99.621	9.997	1.192	25.098	0.119	0.047	119.61	2518.30
Ground Floor	North	50.000	48.864	97.727	6.092	1.572	23.896	0.258	0.066	157.43	2392.76
Total		250.000	233.612	93.445	8.138	0.781	25.508	0.000	0.000	78.26	2560.26

Annual Results

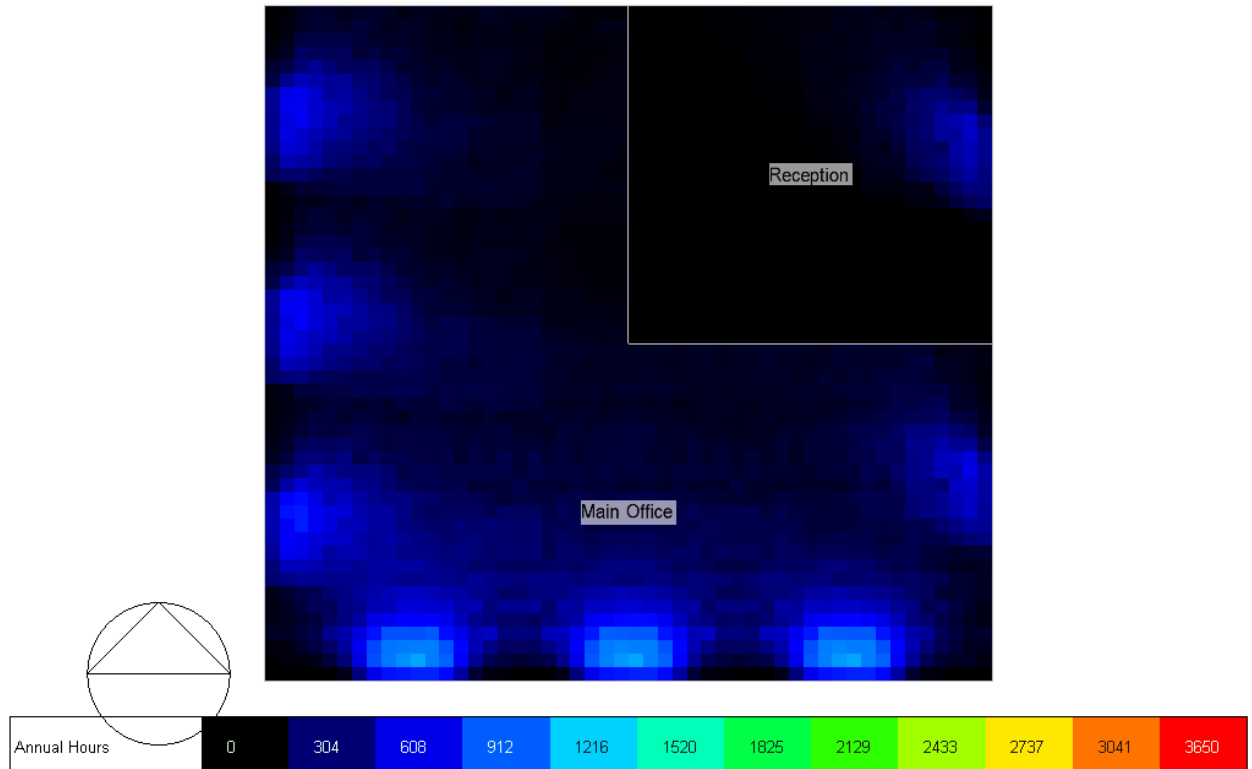
This section explains the results generated when the **2-Annual Option** setting is made.

- When the **1-sDA Map** Option is selected, the distribution of **sDA** across the working plane for the current object is displayed. The main metric, **sDA_{300/50%}**, used in LEED v4 Option 1 daylighting calculations represents the percentage of the floor area that exceeds 300 lux for at least 50% of the occupied time. Unoccupied periods are ignored. The minimum working plane illuminance threshold is set under the **Metric pass thresholds** header in the Display options panel.



- When the **2-ASE Map** Option is selected, the distribution of [ASE](#) across the working plane for the current object is displayed.

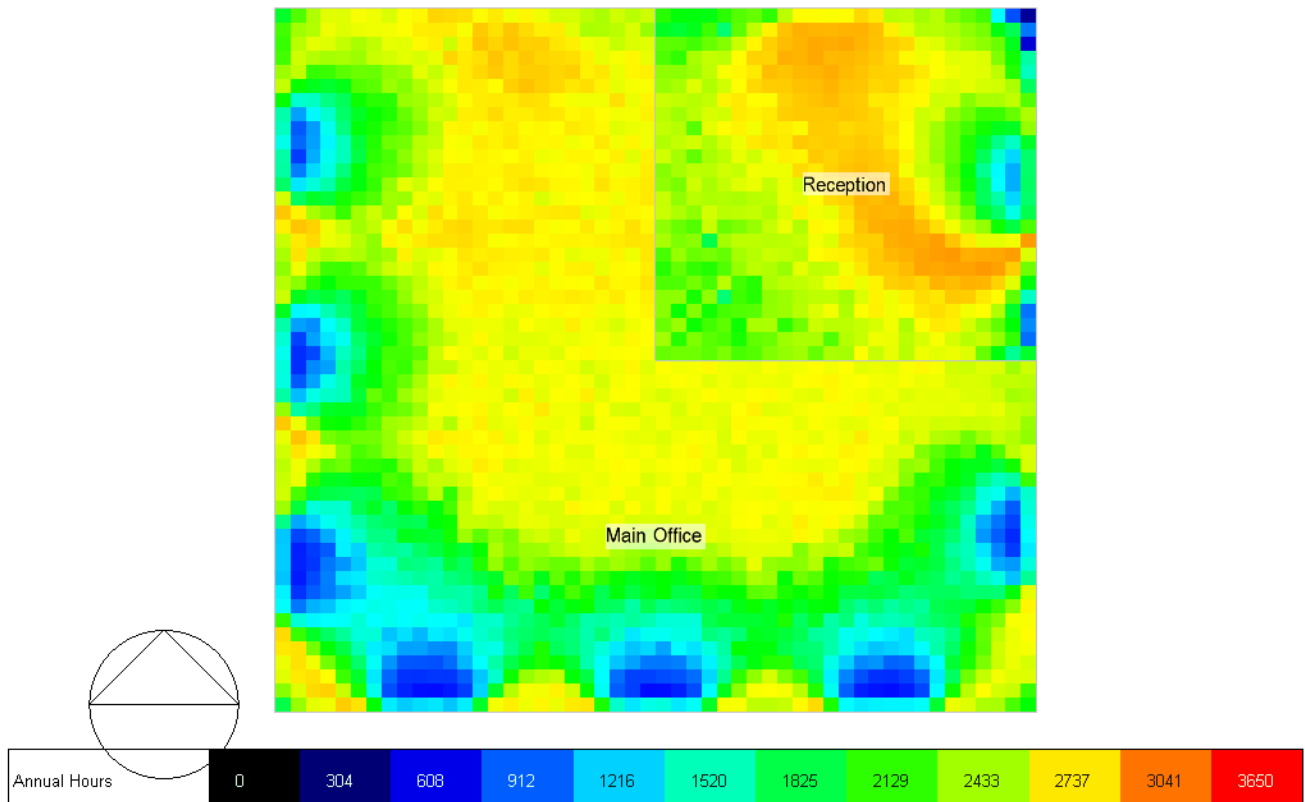
The minimum working plane illuminance threshold is set under the [Metric pass thresholds](#) header in the Display options panel.



Useful Daylight Illuminance (UDI)

Useful Daylight Illuminance (UDI) is the annual occurrence of illuminance distribution across the working plane that are within a range considered “useful” by occupants. It excludes any times when there is inadequate natural daylight or excessive direct sunlight which would give rise to a risk of glare. UDI therefore not only provides information about useful daylight illuminance, but also on the propensity for excessive levels of daylight that are associated with glare, occupant discomfort and unwanted solar gains. In this way the single UDI metric can be used to summarise the overall daylight performance of a space.

More specifically, the UDI bins hourly time values based upon three illumination ranges which are typically, 0-100 lux, 100-2000 lux, and over 2000 lux. It provides full credit only to values between 100 lux and 2,000 lux, based on the assumption that horizontal illumination values outside of this range are not useful. The graphical percent values represent the percentage of the floor area that meets the UDI criteria at least 50% of the time.



Grid

A table of summary annual daylight statistics for each zone is displayed, including areas and % for each zone that satisfy the requirements of the sDA, ASE and UDI metrics.

Block	Zone	Floor Area (m2)	sDA Area in Rang...	sDA Area in Rang...	ASE Area in Rang...	ASE Area in Rang...	UDI Area in Rang...	UDI Area in Rang...
1F first floor East	Open office	239,578	220,777	92,152	66,084	27,584	0,000	0,000
1F first floor West	Open office	239,578	212,772	88,811	80,139	33,450	0,000	0,000
0F ground floor W...	Open office	239,575	153,853	64,219	119,229	49,767	0,000	0,000
0F ground floor East	Open office 1	239,575	150,130	62,665	121,835	50,855	0,000	0,000
Core circulation	Circulation	101,566	101,566	100,000	0,000	0,000	0,000	0,000
Atrium Reception	Reception	44,759	44,759	100,000	0,000	0,000	0,000	0,000
Total		1104,632	883,857	80,014	387,287	35,060	0,000	0,000

- **Floor Area** - the floor area of the zone.
- **sDA Area in Range** (m2 or ft2) - the area of the zone where the sDA meets the criteria set (typically 300 lux), including the [sDA lower illuminance threshold](#).
- **sDA Area in Range** (%) - calculated as **sDA Area in Range** divided by the **Floor Area**.
- **ASE Area in Range** (m2 or ft2) - the area of the zone where the ASE meets the criteria set (typically 1000 lux), including the [ASE illuminance threshold](#).
- **ASE Area in Range** (%) - calculated as **ASE Area in Range** divided by the **Floor Area**.
- **UDI Area in Range** (m2 or ft2) - the area of the zone where the UDI meets the criteria set, including the [UDI lower illuminance threshold](#) and [UDI upper illuminance threshold](#).
- **UDI Area in Range** (%) - calculated as **UDI Area in Range** divided by the **Floor Area**.

Daylighting Display Options - LEED

This topic covers the Daylighting display options specific to the [2-LEED Simulation type](#).

LEED version

Having selected the [2-LEED Simulation type](#), the version of LEED should be selected. Choose from:

- [1-LEED v2](#) - for NC 2.2 Credit EQ 8.1 (LEED v2).
- [2-LEED v3](#) - for Daylighting BD+C: New Construction v3.
- [3-LEED v4](#) - for Daylighting BD+C: New Construction, Options 1 and 2.
- [4-LEED v4.1](#) - for EQ Credit, Options 1 and 2.

Option

For LEED v4 and v4.1 you can choose to display results for Option 1 or Option 2. Select from:

- **1-Option 1 (annual)** - view LEED Daylighting BD+C: New Construction, Option 1 results.
- **2-Option 2 (point in time)** - view LEED Daylighting BD+C: New Construction, Option 2 results.

Report type

The report types available depend on the [LEED version](#) and [Option](#) settings.

When the **1-Option 1 (annual)** [Option](#) setting is made (for LEED versions **3-LEED v4** or **4-LEED v4.1**), the following report types are available:

- **1-DA map** - shows the distribution of annual hours on the working plane for which the [DA \(Daylight Autonomy\)](#) criteria are met.
- **2-ASE map** - shows the distribution of annual hours on the working plane for which the [ASE \(Annual Sunlight Exposure\)](#) criteria are met.
- **3-Grid** - a table of summary annual daylight statistics is displayed for each zone.
- **4-Report** - displays a report with a summary of the results and the number of credits that can be awarded for the selected calculation.

When the **2-Option 1 (point in time)** [Option](#) setting is made and [LEED version](#) is set to **3-LEED v4** or **4-LEED v4.1**, or when the [LEED version](#) is set to **1-LEED v2** or **2-LEED v3**, the following report types are available:

- **1-Map** - a daylight factor or illuminance distribution contour map on the working plane of the current object is displayed.
- **2-Grid** - a table of summary daylight statistics for each zone is displayed.
- **3-Report** - displays a report with a summary of the results and the number of credits that can be awarded for the selected calculation.

Clear sky time (LEED v3 only)

When the [LEED version](#) is set to **2-LEED v3** and a LEED v3 calculation has been carried out, 2 sets of illuminance maps will be stored, one for 9 am and the other for 3 pm. You can select which of these to display using the **Clear sky time** display option. Choose from the following options:

- **1-AM 9:00 - 21 Sep**
- **2-PM 15:00 21 Sep**

Construction type

When displaying a report output for LEED v4 or v4.1 Option 1, you must first select the appropriate Construction type to ensure that the right floor area thresholds are used in the calculation of credits. Choose from:

- **1-New Construction**
- **2-Core and Shell**
- **3-School**
- **4-Retail**
- **5-Data Centre**
- **6-Warehouse**
- **7-Hospitality**

Credits are calculated based on the table below.

Points for daylit floor area: Spatial daylight autonomy

New Construction, Core and Shell, Schools, Retail, Data Centers, Warehouses & Distribution Centers, CI, Hospitality		Healthcare	
sDA (for regularly occupied floor area)	Points	sDA (for perimeter floor area)	Points
55%	2	75%	1
75%	3	90%	2

Margin

Refer to the Margin help for the [General](#) Simulation type.

Metric Pass Thresholds (Annual)

When one of the grid or report types is set, the annual daylighting metric pass thresholds should be reviewed and, if necessary, changed.

Note: The annual daylighting thresholds are used to define acceptability and are split into two parts:

- the [illuminance thresholds](#) which define the acceptable level of lighting at any one point in time over the year, and are defined on the [Calculation options dialog](#), and,
- metric pass thresholds, which are defined here the Display options panel, as described below.

Changing any of the Metric pass threshold settings in the Display options will typically change the **Area in Range** values shown on the grid or report.

sDA percentage annual hours

Enter the % of the occupied year pass threshold used for each cell in the [sDA calculations](#). sDA is usually defined as sDA_{300,50%}, which means that a given grid cell must achieve at least 300 lux ([sDA lower illuminance threshold](#)) for at least 50% ([sDA Percentage annual hours](#)) of the occupied year for its area to be considered to "pass" the criteria. The overall sDA metric for a zone is the % of its floor area that passes in this way.

ASE annual hours

Enter the number of occupied hours in the year pass threshold used for each cell in the [ASE calculations](#). The default is 250 hours. ASE_{1000,250} means, for a given cell, the illuminance must not be greater than the 1000 lux threshold ([ASE illuminance threshold](#)) for more than 250 hours ([ASE annual hours](#)) in the (occupied) year to pass.

Metric Pass Thresholds (Point In Time)

When viewing point in time results and one of the grid or report types is set, the illuminance daylighting pass thresholds should be checked and, if necessary, changed.

Threshold type

Select from:

- **1-Daylight factor**, set the min average daylight factor threshold.
- **2-Illuminance**, set the min and max illuminance threshold values.

Min average DF threshold

When the Threshold type is set to **1-Daylight factor**, set the minimum average daylight factor to be used in calculating the **Floor Area Within Limits** grid outputs.

Min illuminance

When the Threshold type is set to **2-Illuminance**, set the minimum illuminance to be used in calculating the **Floor Area Within Limits** grid outputs.

Max illuminance

When the Threshold type is set to **2-Illuminance**, set the maximum illuminance to be used in calculating the **Floor Area Within Limits** grid outputs.

Scale

The Scale settings are available when the [Report type](#) is set to one of the map options. They provide control over how the map legend scale is defined.

Source

The scale of the daylight maps can be defined using data from the building, block or zone level or it can be entered manually. It is a way to define the minimum and maximum values on the scale. Select from one of the following options:

- **1-Building**, where all of the values calculated for the whole building are scanned and the lowest and highest values are used. This allows plots to be generated using a consistent scale even when displaying data at block or zone level.
- **2-Block**, where all of the values calculated for the current block are scanned and the lowest and highest values for are used. This allows plots to be generated using a consistent scale within a block even when displaying data at zone level.
- **3-Zone**, where the scale is tailored to the current zone allowing in many cases a more even distribution of colours through the map. When using this option, images generated for each zone will use a different scale and so will not be so easily comparable as they would be if one of the **1-Building** or **2-Block** options had been used.
- **4-User defined**, where you can enter the minimum and maximum values on the scale manually. This has the advantage of allowing a specific lower and upper scale value to be used. It also allows you to define colours to represent areas that exceed the maximum or are lower than the minimum. For example, LEED Option 2 states that areas of the working plane where illuminance is lower than 300 lux are inadequately daylighted. Likewise, areas of the working plane where illuminance is higher than 3,000 lux are considered to be subject to glare. These areas can be highlighted in special colours to help provide a clear visual indication of the compliant and non-compliant areas.

Value source

When the **2-Option 2 (point-in-time) Option** setting is made, **Value source** defines whether to view Daylight factor or Illuminance results. Select from:

- **1-Daylight factor** - daylight distribution maps are plotted as daylight factors.
- **2-Illuminance** - daylight distribution maps are plotted as illuminance values.

When the **1-Option 1 (annual) Option** setting is made, **Value source** defines whether to view results as Annual hours or Percentage annual hours. Select from:

- **1-Annual hours** - daylight distribution maps are plotted as the number of annual hours for which the selected criteria are met.
- **2-Percentage annual hours** - daylight distribution maps are plotted as the % of annual hours for which the selected criteria are met. The denominator used in the % calculation is the total number of occupied hours.

Note: when [Source](#) is set to **4-User defined**, **Value source** is hard set to **1-Annual hours**.

User-Defined Scale Settings

Low value

When using the **4-User defined** Scale source you can enter the minimum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

High value

When using the **4-User defined** Scale source you can enter the maximum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

Apply scale "too dark" colour

If you wish to set a specific colour to represent the areas of the working plane that have values below the entered **Low value** criterion then check this box and define the colour in the colour selector below.

Scale "too dark" colour

Select the colour to be used to represent areas of the working plane that are *below* the entered **Low value**. Typically a dark grey or black colour might be used to show areas of the working plane that are inadequately daylight.

Apply scale "too bright" colour

If you wish to set a specific colour to represent areas of the working plane that are *above* the entered **High value** then check this box and define the colour in the colour selector below.

Scale "too bright" colour

Select the colour to be used to represent areas of the working plane that are *above* the entered **High value**.

LEED Version-Dependent Settings

LEED v4.1

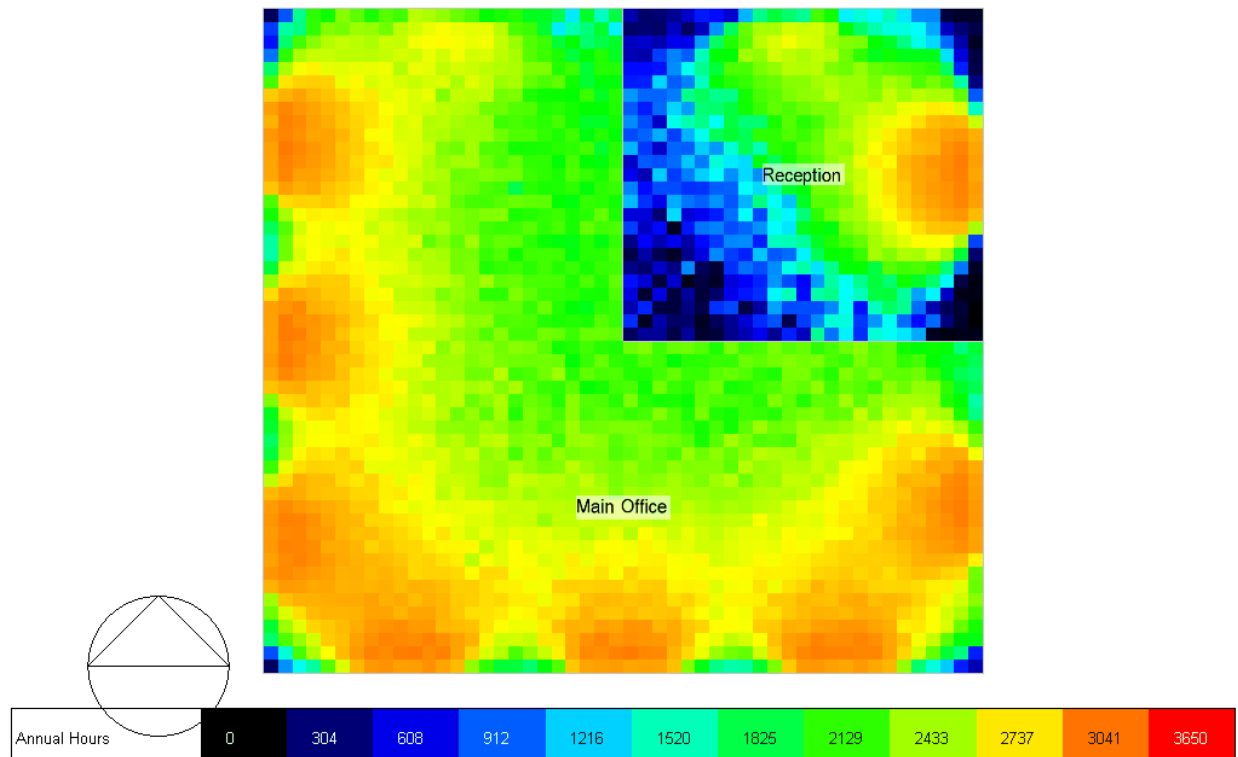
LEED V4.1 Option 1 (Annual) Results

This section summarises the results generated when the **1-Option 1 (annual)** [Option](#) setting is made.

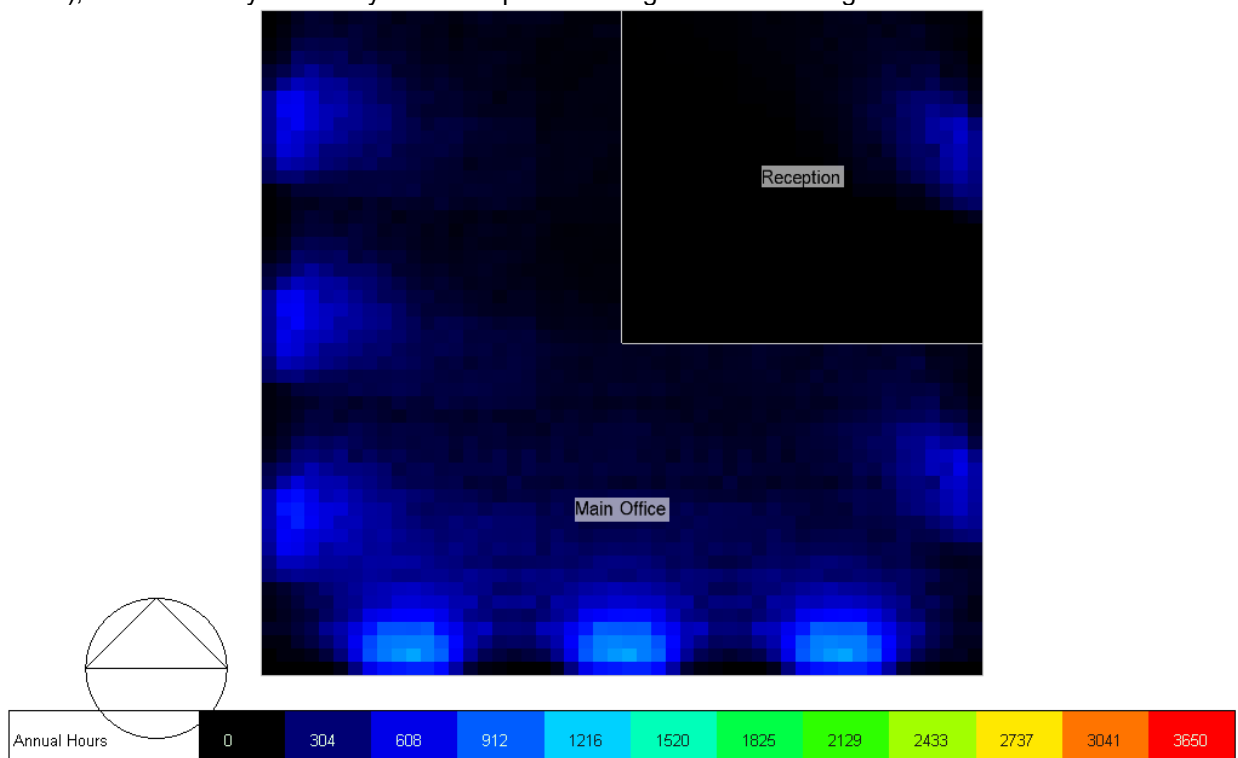
LEED v4.1 Option 1 Map Outputs

When the LEED v4.1 version is set, the distribution of sDA and ASE results on the working plane can be displayed in map form by setting the Report type to **1-DA Map** or **2-ASE Map**.

- When the **1-DA Map** Option is selected, the distribution of [sDA](#) across the working plane for the current object is displayed. The main metric, sDA_{300/50%}, used in LEED v4.1 Option 1 daylighting calculations represents the percentage of the floor area that exceeds 300 lux for at least 50% of the occupied time. Unoccupied periods are ignored.
The minimum working plane illuminance threshold is set to 300 lux behind the scenes. If you need to use a different threshold then you should select the **1-General** Simulation type instead of **2-LEED**.



- When the **2-ASE Map** Option is selected, the distribution of **ASE** across the working plane for the current object is displayed. The main glare metric used in LEED v4.1, $ASE_{1000/250}$, represents the percentage of the floor area that exceeds 1000 lux for 250 or more occupied hours over the year. For any regularly occupied spaces with $ASE_{1000/250}$ greater than 10% (**ASE % Area in Range** of 90% or more), it is necessary to identify how the space is designed to address glare.



LEED v4.1 Option 1 Grid Outputs

To view a summary of the areas and % areas that meet the sDA and ASE requirements set the Report type to **3-Grid**.

The grid output for LEED v4.1 Option 1 summarises the daylighting performance of each zone in terms of areas and % areas that meet the sDA and ASE requirements.

Before viewing results, you should check the **Metric pass thresholds** set in Display options. **sDA percentage annual hours** should be set to 50 and **ASE annual hours** should be set to 250.

Block	Zone	Floor Area (m2)	sDA Area in Range (m2)	sDA Area in Range (%)	ASE Area in Range (m2)	ASE Area in Range (%)
First Floor	Open plan	100.000	98.599	98.599	69.708	69.708
Extension	Room	50.000	47.118	94.235	45.997	91.994
Ground Floor	Main Office	75.000	73.799	98.399	48.506	64.674
Ground Floor	Reception	25.000	11.378	45.513	23.397	93.590
Total		250.000	230.895	92.358	187.608	75.043

Report for LEED v4.1 Option 1 Daylighting BD+C

A report that can be used towards obtaining LEED v4.1 Option 1 daylighting BD+C points through a combination of sDA and ASE metrics. The report includes background information about the requirements as can be seen in the example report below.

When generating a report for LEED v4.1 Option 1, the first step is to select the appropriate [Construction type](#) to ensure that the right thresholds are used in the calculation of credit points.

Before viewing results, you should check the **Metric pass thresholds** set in Display options. **sDA percentage annual hours** should be set to 50 and **ASE annual hours** should be set to 250.

LEED v4.1 EQ Credit: Daylight Option 1

Intent

To connect building occupants with the outdoors, reinforce circadian rhythms, and reduce the use of electrical lighting by introducing daylight into the space.

Requirements

Provide manual or automatic (with manual override) glare-control devices for all regularly occupied spaces.

Option 1. Simulation: Spatial Daylight Autonomy and Annual Sunlight Exposure (1–3 points, 1-2 points Healthcare)

Perform annual computer simulations for spatial daylight autonomy 300/50% (sDA300/50%), and annual sunlight exposure 1000,250 (ASE1000,250) as defined in IES LM-83-12 for each regularly occupied space. Healthcare projects must use each regularly occupied space located in the perimeter area determined under EQ Credit Quality Views. Additionally, calculate the average sDA300/50% value for the total regularly occupied floor area.

For any regularly occupied spaces with ASE1000,250 greater than 10%, identify how the space is designed to address glare.

Points are awarded as follows:

Healthcare	
The average sDA value for the zone is at least 40%	1 point(s)
The average sDA value for the zone is at least 55%	2 point(s)
The average sDA value for the zone is at least 75%	Exemplary performance
Each regularly occupied space achieves an sDA value of at least 55%	Exemplary performance or 1 additional point if only 1 point achieved above

The results below were calculated using the Radiance simulation engine, which provides a detailed multi-zone physics-based calculation of illumination levels on the working planes of the building over the whole year using real climate data.

Daylighting data	
Project file	Daylighting Modelling Guide Rev for Help.dsb
Report generation time	26/07/2024 19:03:50
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.760
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200
sDA lower illuminance threshold (lux)	300.0
ASE upper illuminance threshold (lux)	1000.0

Summary Results	
Total area (m2)	250.000
sDA Area in Range (m2)	230.895
sDA Area in Range (%)	92.36
ASE Area in Range (m2)	187.608
ASE Area in Range (%)	75.04
Points credited	Exemplary performance

* This construction must identify how it will address glare

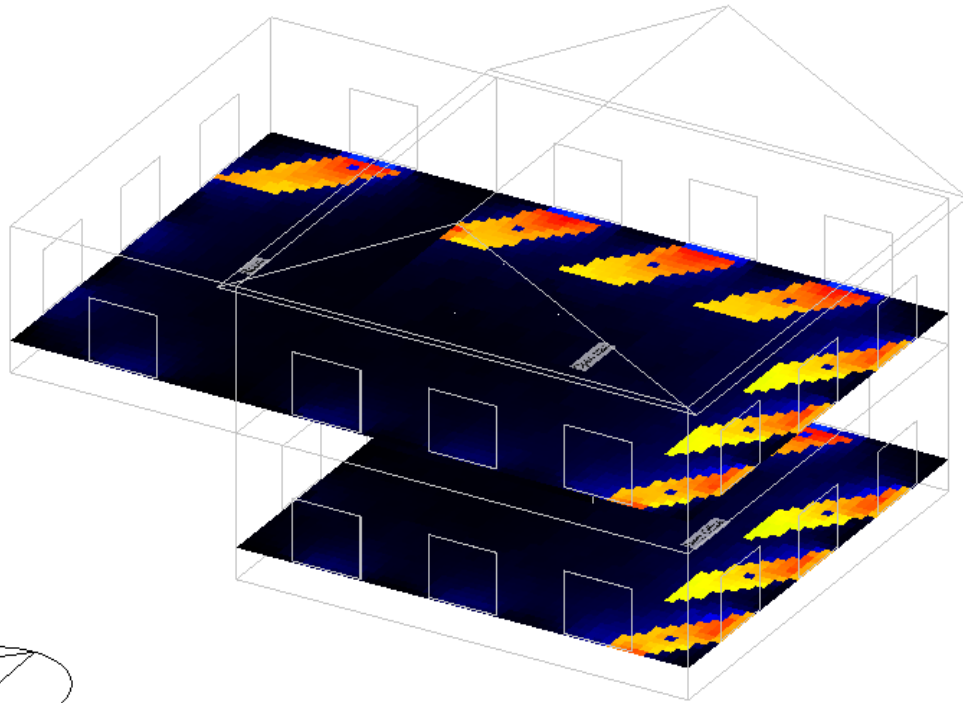
LEED v4.1 Option 2 (Point In Time) Results

This section explains the results generated when the **2-Option 2 (point in time)** [Option](#) setting is made.

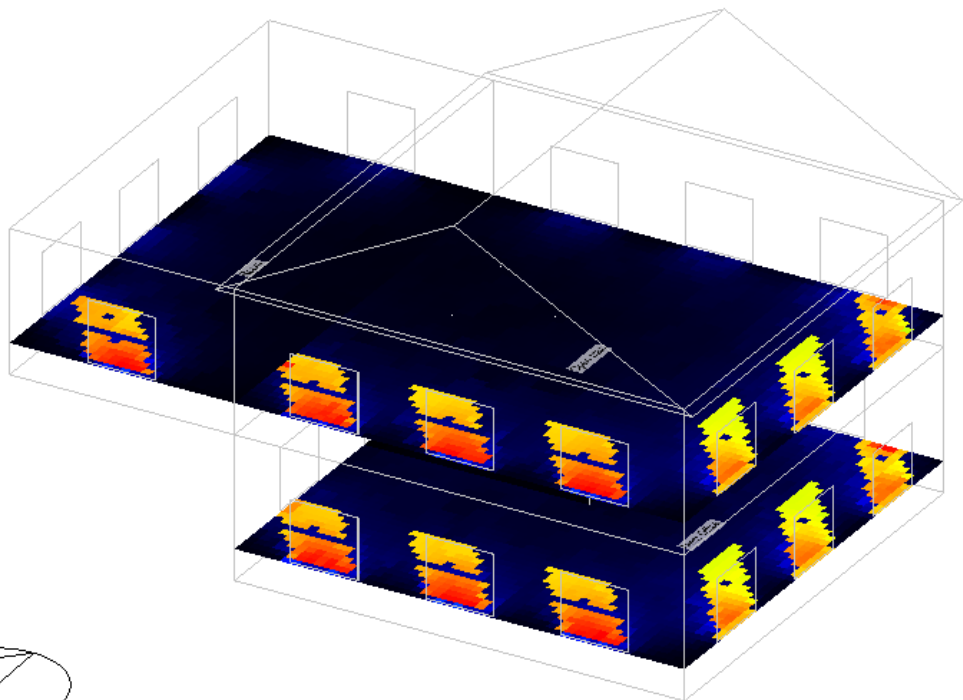
LEED v4.1 Option 2 Map Outputs

Use the **Clear sky time** setting to request results for morning or afternoon. The options are:

- 1-AM 9:00 - 21 Sep
- 2-PM 15:00 - 21 Sep



Daylight Factor	0.34	5.28	10.22	15.16	20.10	25.04	29.99	34.93	39.87	44.81	49.75	59.63
-----------------	------	------	-------	-------	-------	-------	-------	-------	-------	-------	-------	-------



Daylight Factor	0.08	5.04	10.00	14.95	19.91	24.87	29.83	34.79	39.74	44.70	49.66	59.58
-----------------	------	------	-------	-------	-------	-------	-------	-------	-------	-------	-------	-------

LEED v4.1 Option 2 Grid Output

The grid output for LEED v4.1 Option 2 summarises the daylighting performance of each zone in terms of minimum and maximum daylight factor and illuminance values. Uniformity ratios are provided as minimum/average and minimum/maximum.

The following data is calculated for each zone:

- **Floor Area (m2 or ft2)**
- **Floor Area within Limits (m2 or ft2)** - the floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.
- **Floor Area within Limits (%)** - the % floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.
- **Average Daylight Factor (%)**
- **Minimum Daylight Factor (%)**
- **Maximum Daylight Factor (%)**
- **Uniformity ratio (Min / Avg)**
- **Uniformity ratio (Min / Max)**
- **Min Illuminance (lux or fc)**
- **Max Illuminance (lux or fc)**

This option can be especially useful if the built-in daylight reports don't cover the particular calculation you need to make. In this case you can export the summary data for each zone for further analysis in a spreadsheet.

An example grid output is shown below.

Block	Zone	Floor Area (m2)	Floor Area within ...	Floor Area within ...	Average Daylight ...	Minimum Daylight ...	Maximum Daylight ...	Uniformity ratio (M...	Uniformity ratio (M...	Min Illuminance (L...	Max Illuminance (L...
First Floor	Open plan	100.000	64.200	64.200	16.572	0.827	59.665	0.050	0.014	475.89	34343.58
Extension	Room	50.000	41.520	83.040	7.645	0.340	58.928	0.045	0.006	196.01	33939.01
Ground Floor	Main Office	75.000	45.400	60.533	17.686	0.454	59.194	0.026	0.008	261.21	34049.08
Ground Floor	Reception	25.000	10.960	43.840	6.832	0.078	58.263	0.011	0.001	44.90	33531.59
Total		250.000	162.080	64.832	14.147	0.078	59.665	0.000	0.000	44.90	34343.58

Before viewing grid results, you should ensure that the Thresholds set in Display Options define minimum and maximum illuminance values of 300 lux and 3,000 lux respectively:

Threshold	
Threshold type	2-Illuminance
Min illuminance (lux)	300.000
Max illuminance (lux)	3000.000

LEED v4.1 Option 2 Report

A report is generated detailing eligibility for the LEED v4.1 daylighting points for Option 2 based on % area of the occupied part of the building that has daylight factor between 300 and 3,000 lux at 9 am and 3 pm. LEED v4.1 Option 2 requires two calculations using clear day conditions recorded for the nearest weather station to the site, the first within 15 days of September 21 and the second within 15 days of March 21.

The easiest way to achieve this is described under [LEED v4 Option 2 Sky Settings](#) in the Calculation options help.

When reporting cell values in grid and report views, the minimum, maximum, or average cell value is calculated and applied as summarised below:

The minimum value for each cell of the two simulations is recorded if it is lower than the lower threshold.

The maximum value for each cell of the two simulations is recorded if it is higher than the upper threshold.

Otherwise, the average value of the two simulations is recorded if both values for the cell are within the threshold limits.

LEED v4.1 EQ Credit: Daylight Option 2**Intent**

To connect building occupants with the outdoors, reinforce circadian rhythms, and reduce the use of electrical lighting by introducing daylight into the space.

Requirements

Provide manual or automatic (with manual override) glare-control devices for all regularly occupied spaces.

Option 2. Simulation: Illuminance Calculations (1–3 points, 1-2 points Healthcare)

Perform computer simulations for illuminance at 9 a.m. and 3 p.m. on a clear-sky day at the equinox for each regularly occupied space. Healthcare projects should use the regularly occupied spaces located in the perimeter area determined under EQ Credit Quality Views.

Demonstrate illuminance levels are between 300 lux and 3000 lux at both 9 a.m. and 3 p.m. Spaces with view-preserving automatic (with manual override) glare-control devices may demonstrate compliance for only the minimum 300 lux illuminance level.

Points are awarded as follows:

New Construction, Core and Shell, Schools, Retail, Data Centers, Warehouses, Hospitality	
Percentage of regularly occupied floor area	Points
55%	1
75%	2
90%	3

The results below were calculated using the Radiance simulation engine which provides a detailed multi-zone physics-based calculation of illumination levels on the working planes of the building.

Daylighting data	
Project file	Daylighting Modelling Guide Rev for Help.dsb
Report generation time	19/02/2024 17:28:36
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.760
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200
Summary Results	
Total area (m2)	250.0
Total area meeting requirements (m2)	162.1
Percentage of regularly occupied floor area	64.8
Points credited	1

Eligible zones for daylighting, worst case values for 9:00 and 15:00

Block	Zone	Floor area (m2)	Min Illumination (lux)	Working plane area within Limits (%)
First Floor	Open plan	100.0	475.89	64.2
Extension	Room	50.0	196.01	83.0
Ground Floor	Main Office	75.0	261.21	60.5
Ground Floor	Reception	25.0	44.00	43.8

Before viewing the report, you should ensure that the Thresholds set in Display options define minimum and maximum illuminance values of 300 lux and 3,000 lux respectively:

Threshold	
Threshold type	2-Illuminance
Min illuminance (lux)	300.000
Max illuminance (lux)	3000.000

LEED v4**LEED v4 Option 1 (Annual) Results**

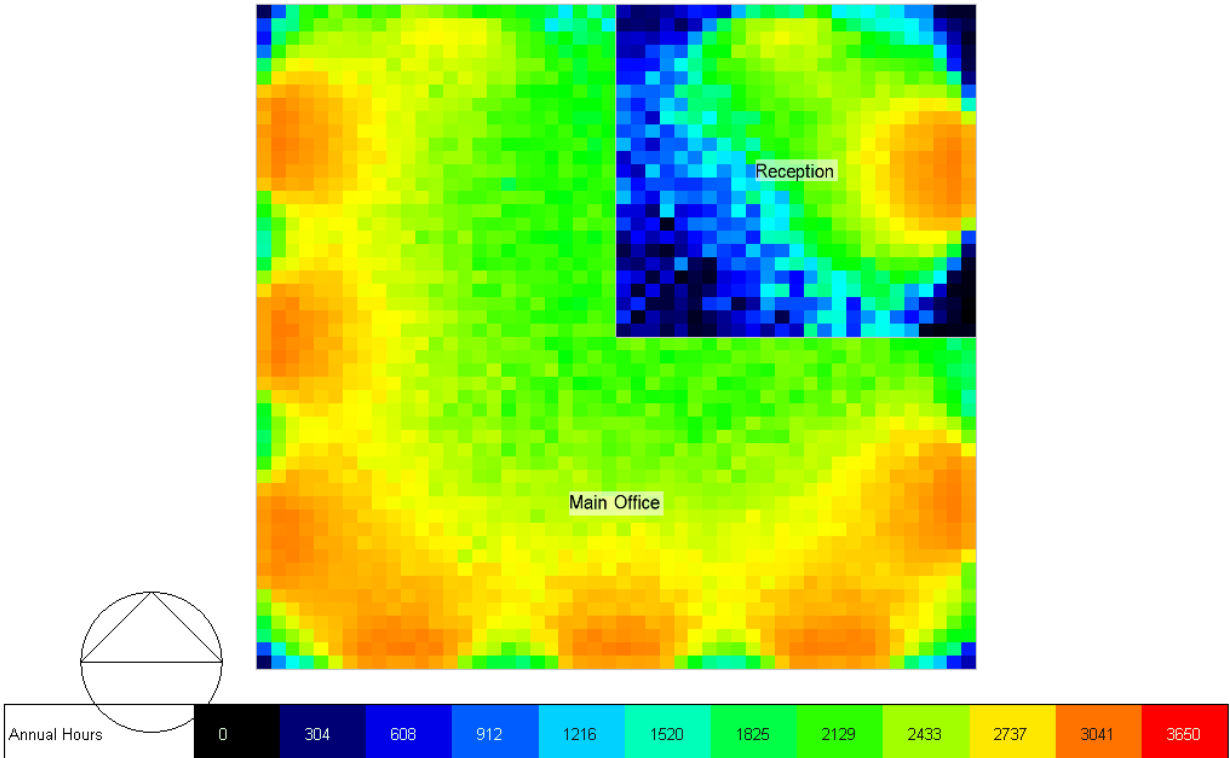
This section summarises the results generated when the **1-Option 1 (annual) Option** setting is made.

LEED v4 Option 1 Map Outputs

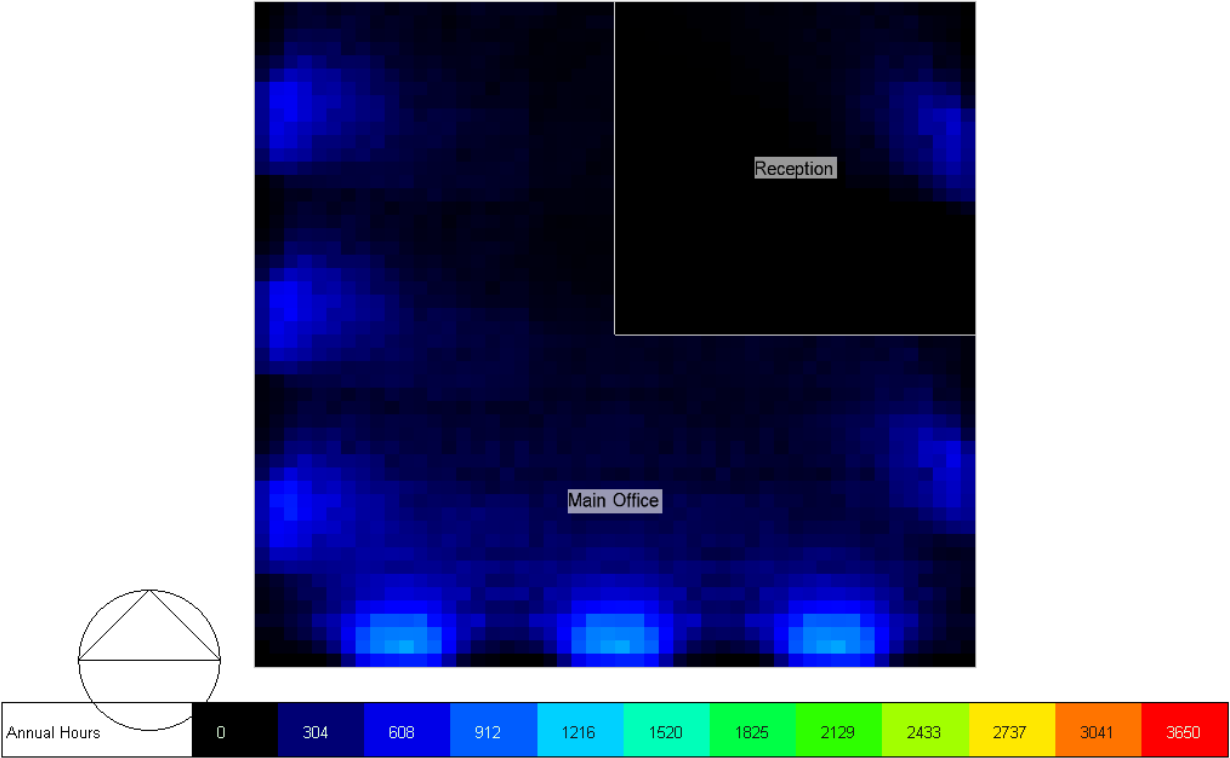
When the LEED v4 version is set, the distribution of sDA and ASE results on the working plane can be displayed in map form by setting the Report type to **1-DA Map** or **2-ASE Map**.

When the **1-DA Map** Option is selected, the distribution of **sDA** across the working plane for the current object is displayed. The main metric, sDA_{300/50%}, used in LEED v4.1 Option 1 daylighting calculations represents the percentage of the floor area that exceeds 300 lux for at least 50% of the occupied time. Unoccupied periods are ignored.

The minimum working plane illuminance threshold is set to 300 lux behind the scenes. If you need to use a different threshold then you should select the **1-General** Simulation type instead of **2-LEED**.



When the **2-ASE Map** Option is selected, the distribution of ASE across the working plane for the current object is displayed. The main glare metric used in LEED v4, ASE_{1000/250}, represents the percentage of the floor area that exceeds 1000 lux for 250 or more occupied hours over the year. For any regularly occupied spaces with ASE_{1000,250} greater than 10% (**ASE % Area in Range** of 90% or more), it is necessary to identify how the space is designed to address glare.



LEED v4 Option 1 Grid Outputs

To view a summary of the areas and % areas that meet the sDA and ASE requirements set the Report type to **3-Grid**.

The grid output for LEED v4 Option 1 summarises the daylighting performance of each zone in terms of areas and % areas that meet the sDA and ASE requirements.

Before viewing results, you should check the **Metric pass thresholds** set in Display options. **sDA percentage annual hours** should be set to 50 and **ASE annual hours** should be set to 250.

You can also use the **ASE Min area** Display option to allow very small zones to pass ASE criteria when required. To avoid small zones failing the ASE criteria, enter the minimum floor area for zones to be checked for ASE. Enter zero if you don't want this facility to be used.

Block	Zone	Floor Area (m2)	sDA Area in Range (m2)	sDA Area in Range (%)	ASE Area in Range (m2)	ASE Area in Range (%)
First Floor	Open plan	100.000	98.599	98.599	69.708	69.708
Extension	Room	50.000	47.118	94.235	45.997	91.994
Ground Floor	Main Office	75.000	73.799	98.399	48.506	64.674
Ground Floor	Reception	25.000	11.378	45.513	23.397	93.590
Total		250.000	230.895	92.358	187.608	75.043

Report for LEED v4 Option 1 Daylighting BD+C

A report that can be used towards obtaining LEED v4 Option 1 daylighting BD+C points through a combination of sDA and ASE metrics. The report includes background information on the requirements as can be seen in the example report below.

When generating a report for LEED v4 Option 1, the first step is to select the appropriate [Construction type](#) to ensure that the right thresholds are used in the calculation of credit points.

Before viewing results, you should check the **Metric pass thresholds** set in Display options. **sDA percentage annual hours** should be set to 50 and **ASE annual hours** should be set to 250.

You can also use the **ASE Min area** Display option to allow very small zones to pass ASE criteria when required. To avoid small zones failing the ASE criteria, enter the minimum floor area for zones to be checked for ASE. Enter zero if you don't want this facility to be used.

LEED v4 Daylighting BD+C: New Construction (sDA and ASE)

Intent

To connect building occupants with the outdoors, reinforce circadian rhythms, and reduce the use of electrical lighting by introducing daylight into the space.

Requirements

Provide manual or automatic (with manual override) glare-control devices for all regularly occupied spaces.

Option 1. Simulation: Spatial Daylight Autonomy (2-3 points, 1-2 points Healthcare)

Demonstrate through annual computer simulations that spatial daylight autonomy 300/50% (sDA 300/50%) of at least 55%, 75%, or 90% is achieved. Use regularly occupied floor area. Healthcare projects should use the perimeter area determined under EQ Credit Quality Views. Points are awarded according to Table 1 towards the bottom of the report.

AND

Demonstrate through annual computer simulations that annual sunlight exposure 1000/250 (ASE 1000/250) of no more than 10% is achieved. Use the regularly occupied floor area that is daylight per the sDA 300/50% simulations.

The results below were calculated using the Radiance simulation engine, which provides a detailed multi-zone physics-based calculation of illumination levels on the working planes of the building over the whole year using real climate data.

Daylighting data	
Project file	Daylighting Modelling Guide Rev for Help.dsb
Report generation time	19/02/2024 16:55:53
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.760
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200
sDA lower illuminance threshold (lux)	300.0
ASE upper illuminance threshold (lux)	1000.0

Summary Results	
Total area (m2)	250.000
sDA area in range (m2)	230.895
ASE area in range (m2)	187.6
sDA Points	3
ASE Status	FAIL
Overall points	0

Eligible zones for daylighting

Block	Zone	Floor area (m2)	sDA area in range (m2)	sDA % in Range%	ASE area in range (m2)	ASE % in range
First Floor	Open plan	100.0	98.60	98.60	69.71	69.7
Extension	Room	50.0	47.12	94.24	46.00	92.0
Ground Floor	Main Office	75.0	73.80	98.40	48.51	64.7
Ground Floor	Reception	25.0	11.38	45.51	23.40	93.6

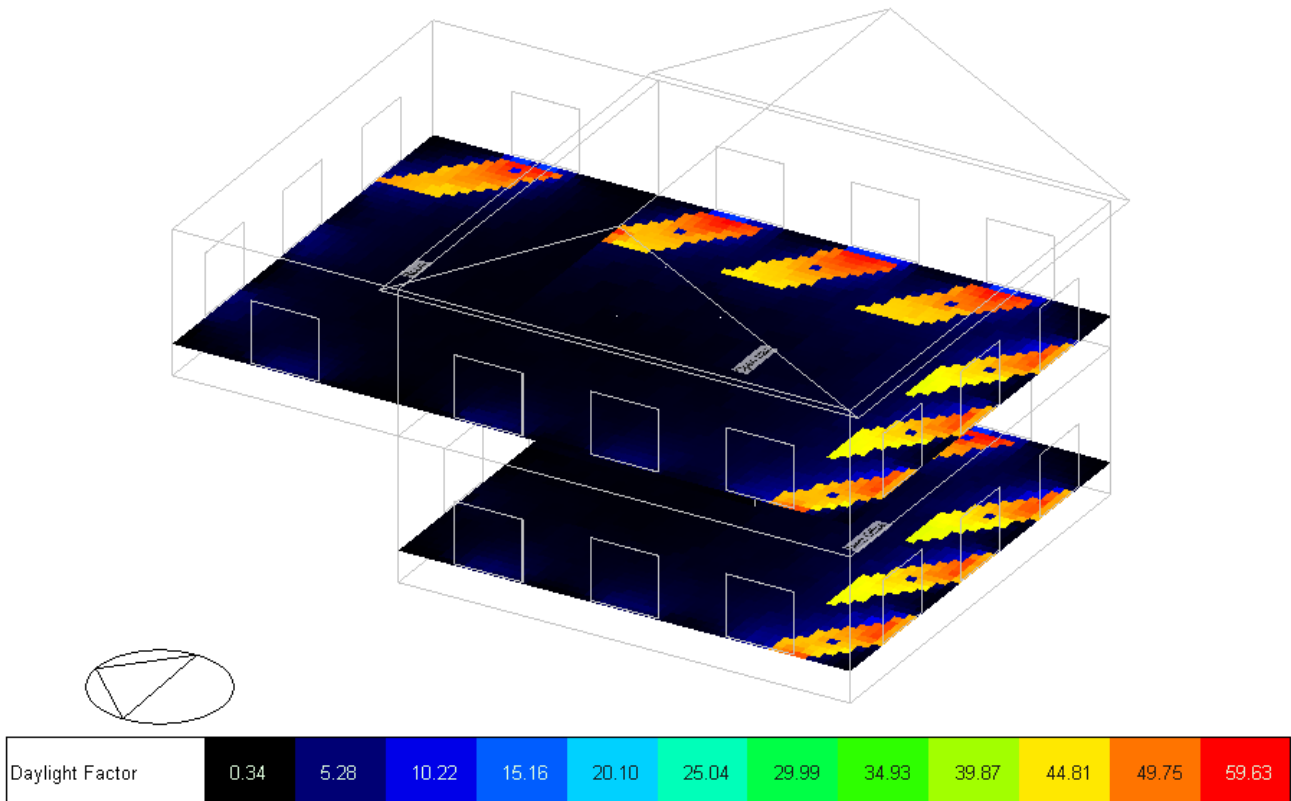
LEED v4 Option 2 (Point In Time) Results

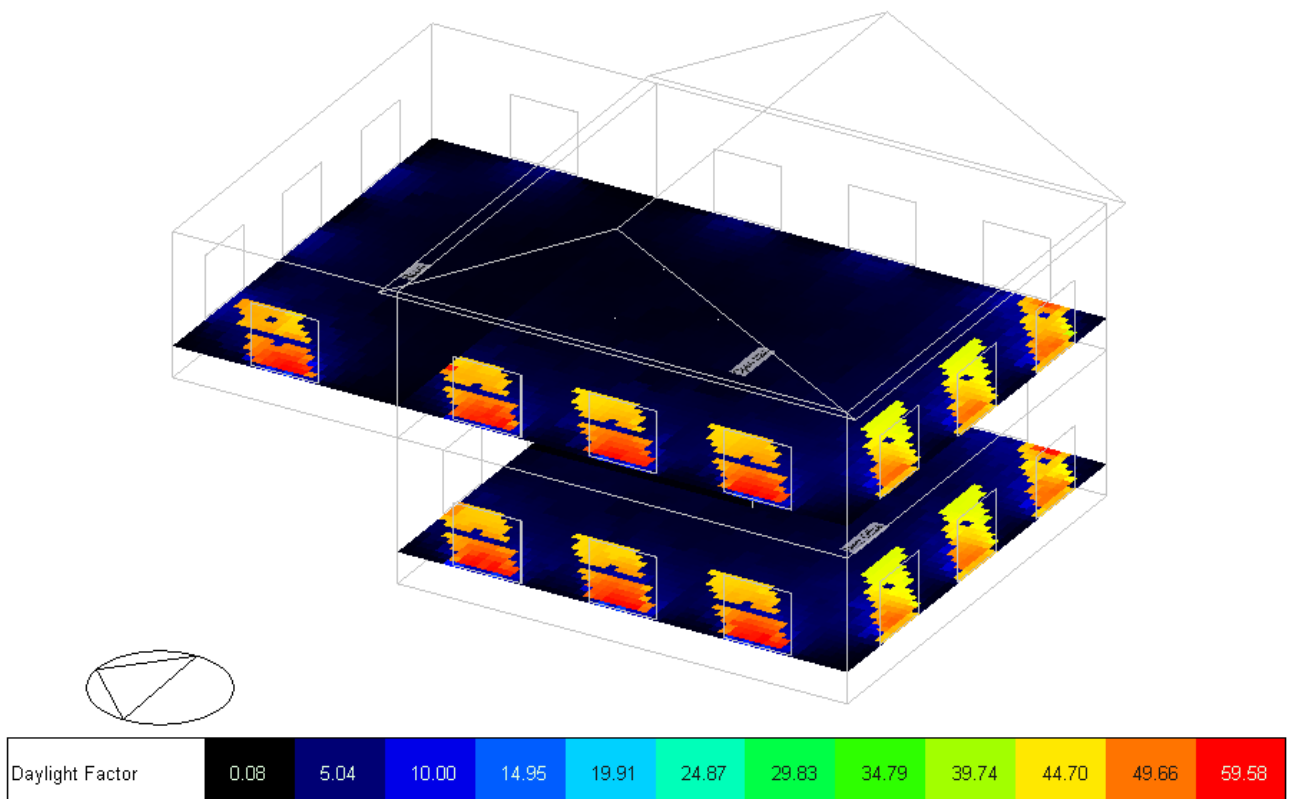
This section explains the LEED v4 results generated when the **2-Option 2 (point in time)** [Option](#) setting is made.

LEED v4 Option 2 Map Outputs

Use the **Clear sky time** setting to request results for morning or afternoon. The options are:

- 1-AM 9:00 - 21 Sep
- 2-PM 15:00 - 21 Sep





LEED v4 Option 2 Grid Output

The grid output for LEED v4 Option 2 summarises the daylighting performance of each zone in terms of minimum and maximum daylight factor and illuminance values. Uniformity ratios are provided as minimum/average and minimum/maximum.

The following data is calculated for each zone:

Floor Area (m2 or ft2)

Floor Area within Limits (m2 or ft2) - the floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.

Floor Area within Limits (%) - the % floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.

Average Daylight Factor (%)

Minimum Daylight Factor (%)

Maximum Daylight Factor (%)

Uniformity ratio (Min / Avg)

Uniformity ratio (Min / Max)

Min Illuminance (lux or fc)

Max Illuminance (lux or fc)

This option can be especially useful if the built-in daylight reports don't cover the particular calculation you need to make. In this case you can export the summary data for each zone for further analysis in a spreadsheet.

An example grid output is shown below.

Block	Zone	Floor Area (m2)	Floor Area within ...	Floor Area within ...	Average Daylight ...	Minimum Daylight ...	Maximum Daylight ...	Uniformity ratio (M...	Uniformity ratio (M...	Min Illuminance (L...	Max Illuminance (L...
First Floor	Open plan	100.000	64.200	64.200	16.572	0.827	59.665	0.050	0.014	475.89	34343.58
Extension	Room	50.000	41.520	83.040	7.645	0.340	58.928	0.045	0.006	196.01	33939.01
Ground Floor	Main Office	75.000	45.400	60.533	17.686	0.454	59.194	0.026	0.008	261.21	34049.08
Ground Floor	Reception	25.000	10.960	43.840	6.832	0.078	58.263	0.011	0.001	44.90	33531.59
Total		250.000	162.080	64.832	14.147	0.078	59.665	0.000	0.000	44.90	34343.58

Before viewing grid results, you should ensure that the thresholds set in Display Options define minimum and maximum illuminance values of 300 lux and 3,000 lux respectively:

Threshold	
Threshold type	2-Illuminance
Min illuminance (lux)	300.000
Max illuminance (lux)	3000.000

LEED v4 Option 2 Report

A report is generated detailing eligibility for the LEED v4 daylighting points for Option 2 based on % area of the occupied part of the building that has daylight factor between 300 and 3,000 lux at 9 am and 3 pm. LEED v4 Option 2 requires two calculations using clear day conditions recorded for the nearest weather station to the site, the first within 15 days of September 21 and the second within 15 days of March 21. The average of the two hourly values are used in the calculations. The easiest way to achieve this is described under [LEED v4 Option 2 Sky Settings](#) in the Calculation options help.

Daylighting Credit - LEED v4 Option 2

The credit is based on demonstrating through computer modeling that illuminance levels will be between 300 lux and 3,000 lux for 9 a.m. and 3 p.m., both on a clear-sky day at the equinox. If 90% of floor area meets the requirement then 2 credits are awarded, otherwise if 75% of floor area meets the requirements then 1 credit is awarded. Calculations are based on regularly occupied floor area.

The results below were calculated using the Radiance simulation engine which provides a detailed multi-zone physics-based calculation of illumination levels on the working planes of the building.

Daylighting data	
Project file	C:\Users\AndyT1\OneDrive - Designbuilder Software Ltd\Documents\DesignBuilder Data\Daylighting Modelling Guide Rev for Help.dsb
Report generation time	19/02/2024 17:11:58
Sky model	2-Perez all weather, 3-Direct horizontal irradiance
Time 1	9:00, 21 Mar/Sep
Time 2	15:00, 21 Mar/Sep
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.760
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200
Illuminance lower threshold (lux)	300.000
Illuminance upper threshold (lux)	3000.000
Summary Results	
Total area (m2)	250.0
Total area meeting requirements (m2)	162.1
% Area within illuminance threshold limits	64.8
LEED v4 Option 2 Credits	FAIL

Eligible zones for daylighting, worst case values for 9:00 and 15:00

Block	Zone	Floor area (m2)	Min Illumination (lux)	Working plane area within Limits (%)
First Floor	Open plan	100.0	475.89	64.2
Extension	Room	50.0	196.01	83.0
Ground Floor	Main Office	75.0	261.21	60.5
Ground Floor	Reception	25.0	44.90	43.8
Total		250.0	44.90	64.8

Before viewing the report, you should ensure that the Thresholds set in Display options define minimum and maximum illuminance values of 300 lux and 3,000 lux respectively:

Threshold	
Threshold type	2-Illuminance
Min illuminance (lux)	300.000
Max illuminance (lux)	3000.000

LEED v3

LEED v3 NC 2009 IEQ 8.1 Results

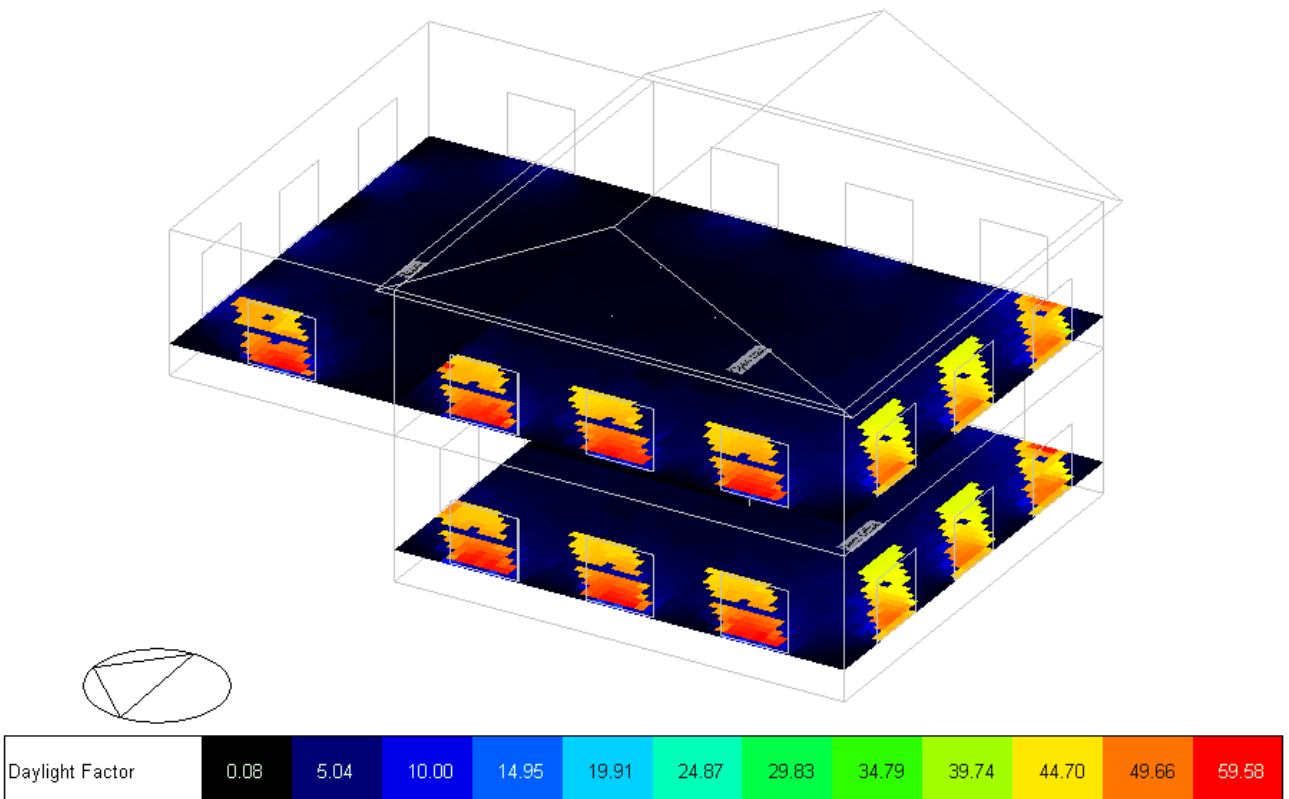
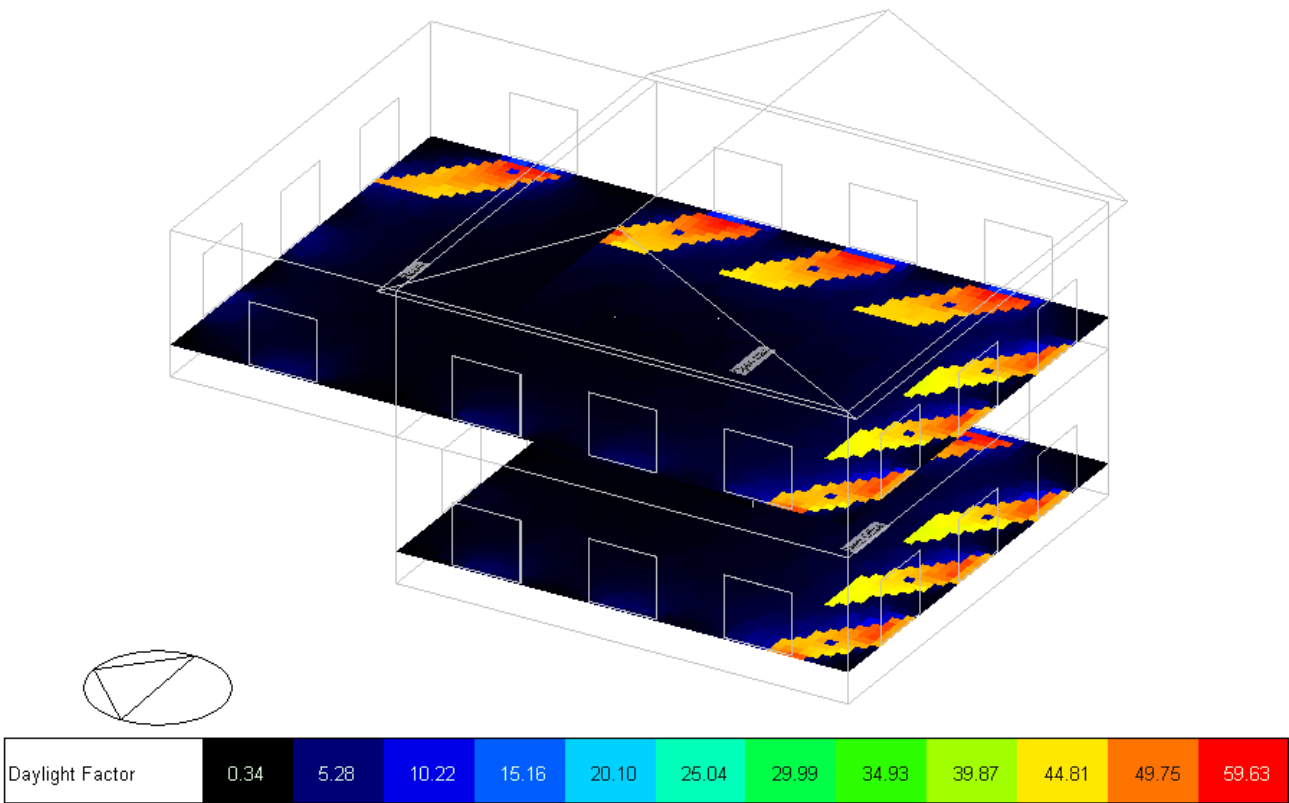
LEED v3 requires computer simulations to demonstrate that the applicable spaces achieve daylight illuminance levels of a minimum of 10 footcandles (fc) (108 lux) and a maximum of 500 fc (5,400 lux) in a clear sky condition on September 21 at 9 a.m. and 3 p.m. DesignBuilder provides a report to summarise the building's compliance with these requirements, but you can also drill down to see detailed daylight performance for each zones.

This section summarises the results generated when the **2-LEED v3** Version setting is made.

LEED v3 Map Outputs

Use the **Clear sky time** setting to request results for morning or afternoon. The options are:

- 1-AM 9:00 - 21 Sep
- 2-PM 15:00 - 21 Sep



LEED v3 Grid Outputs

The grid output for LEED v3 summarises the daylighting performance of each zone in terms of minimum and maximum daylight factor and illuminance values. Uniformity ratios are provided as minimum/average and minimum/maximum.

The following data is calculated for each zone:

Floor Area (m2 or ft2)

Floor Area within Limits (m2 or ft2) - the floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.

Floor Area within Limits (%) - the % floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.

Average Daylight Factor (%)

Minimum Daylight Factor (%)

Maximum Daylight Factor (%)

Uniformity ratio (Min / Avg)

Uniformity ratio (Min / Max)

Min Illuminance (lux or fc)

Max Illuminance (lux or fc)

This option can be especially useful if the built-in daylight reports don't cover the particular calculation you need to make. In this case you can export the summary data for each zone for further analysis in a spreadsheet.

An example grid output is shown below.

Block	Zone	Floor Area (m2)	Floor Area within ...	Floor Area within ...	Average Daylight ...	Minimum Daylight ...	Maximum Daylight ...	Uniformity ratio (M...	Uniformity ratio (M...	Min Illuminance (L...	Max Illuminance (L...
First Floor	Open plan	100.000	64.200	64.200	16.572	0.827	59.665	0.050	0.014	475.89	34343.58
Extension	Room	50.000	41.520	83.040	7.645	0.340	58.928	0.045	0.006	196.01	33939.01
Ground Floor	Main Office	75.000	45.400	60.533	17.686	0.454	59.194	0.026	0.008	261.21	34049.08
Ground Floor	Reception	25.000	10.960	43.840	6.832	0.078	58.263	0.011	0.001	44.90	33531.59
Total		250.000	162.080	64.832	14.147	0.078	59.665	0.000	0.000	44.90	34343.58

Before viewing grid results, you should ensure that the Thresholds set in Display Options define minimum and maximum illuminance values of 270 lux and 5,400 lux respectively:

Threshold

Threshold type
2-Illuminance

Min illuminance (lux)
270

Max illuminance (lux)
5400

LEED v3 Credit EQ8.1 Report

A report is generated detailing eligibility for the LEED v3 daylighting credit EQ8.1 based on % area of the occupied part of the building that has better than the minimum required daylight factor. LEED v3 EQ8.1 requires 2 calculations using [CIE clear day](#) conditions at 9am and 3pm on 21 September. When the LEED v3 calculation option is selected these calculations are carried out automatically.

An example LEED v3 daylight report is shown below.

Daylighting Credit - LEED v3 NC 2009 IEQ 8.1

The aim of the daylighting credit is to encourage and recognize designs that provide appropriate levels of daylight for building users.
A daylighting credit is available if at least 75% of net lettable area in occupied spaces is adequately daylit, having illuminance within the range 10-500 fc.

The results below were calculated using the Radiance simulation engine which provides a detailed multi-zone physics-based calculation of illumination levels on the working planes of the building.

Daylighting data	
Project file	C:\Users\AndyT1\OneDrive - Designbuilder Software Ltd\Documents\DesignBuilder Data\Daylighting Modelling Guide Rev for Help.dsb
Report generation time	19/02/2024 19:01:00
Sky model	2-Perez all weather, 3-Direct horizontal irradiance
Time 1	9:00, 21 Sep
Time 2	15:00, 21 Sep
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.750
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200
Illuminance lower threshold (lux)	270.000
Illuminance upper threshold (lux)	5400.000
Summary Results	
Total area (m2)	250.0
Total area meeting requirements (m2)	179.2
% Area within illuminance threshold limits	71.7
LEED v3 NC 2009 IEQ 8.1 Status	FAIL

Eligible zones for daylighting, worst case values for 9:00 and 15:00

Block	Zone	Floor area (m2)	Min Illumination (lux)	Working plane area within Limits (%)		
First Floor	Open plan	100.0	514.96	70.4		
Extension	Room	50.0	174.12	86.9		
Ground Floor	Main Office	75.0	220.86	67.7		
Ground Floor	Reception	25.0	67.88	58.4		
Total		250.0	67.88	71.7		

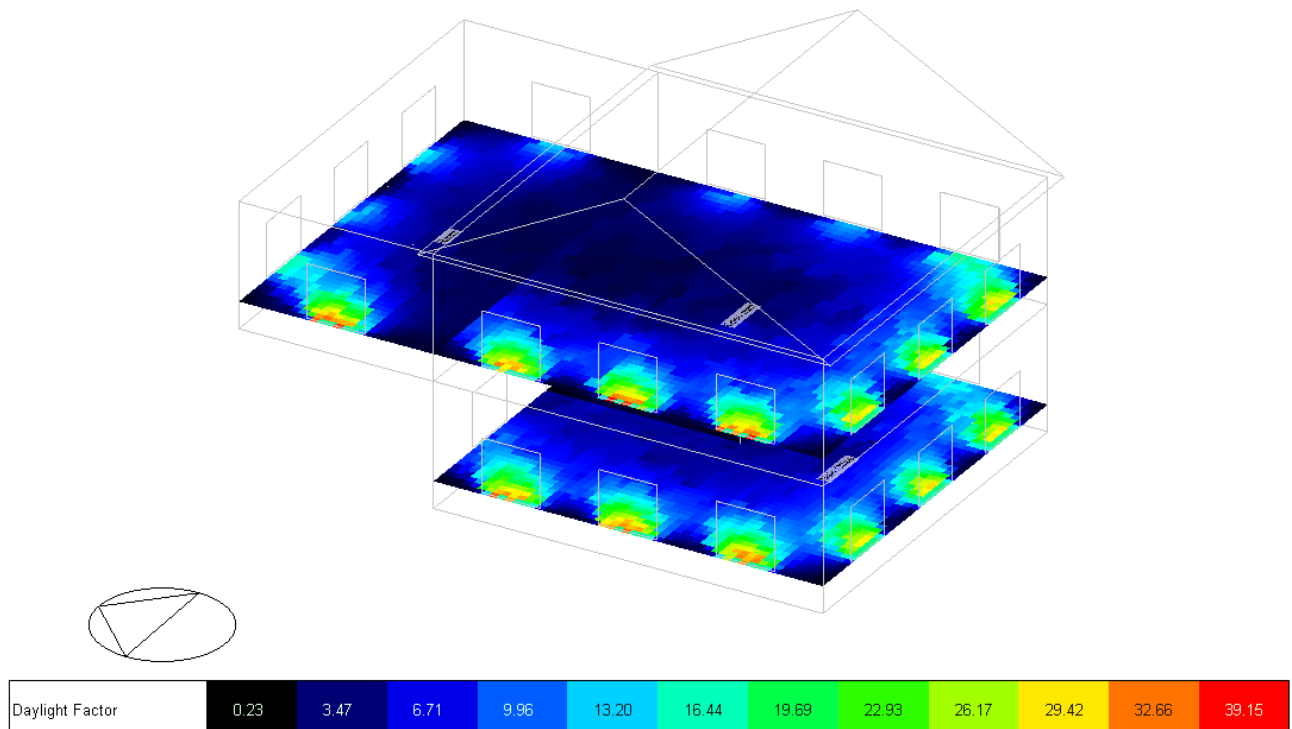
LEED v2

LEED v2 NC 2. EQ 8.1 Results

LEED v2 requires computer simulations to demonstrate that the applicable spaces achieve daylight illuminance levels of a minimum of 10 footcandles (fc) (108 lux) in a clear sky condition on September 21 at 9 a.m. and 3 p.m. DesignBuilder provides a report to summarise the building's compliance with these requirements, but you can also drill down to see detailed daylight performance for each zones.

LEED v2 Map Results

With LEED v2, the calculations for morning and afternoon need to be run separately. Likewise, results must be reviewed separately.



LEED v2 Grid Results

The grid output for LEED v2 summarises the daylighting performance of each zone in terms of minimum and maximum daylight factor and illuminance values. Uniformity ratios are provided as minimum/average and minimum/maximum.

Block	Zone	Floor Area (m2)	Floor Area within...	Floor Area within...	Average Dayligh...	Minimum Dayligh...	Maximum Daylig...	Uniformity ratio (...)	Uniformity ratio (...)	Min Illuminance (...)	Max Illuminance ...
First Floor	Open plan	100.000	71.720	71.720	8.724	1.249	39.365	0.143	0.032	83.18	2622.60
Extension	Room	50.000	27.760	55.520	6.337	0.861	37.385	0.136	0.023	57.66	2504.50
Ground Floor	Main Office	75.000	69.560	92.747	10.276	1.026	36.512	0.100	0.028	68.57	2438.82
Ground Floor	Reception	25.000	3.240	12.960	2.604	0.228	13.547	0.087	0.017	15.11	898.64
Total		250.000	172.280	68.912	8.100	0.228	39.365	0.000	0.000	15.11	2622.60

LEED v2 Credit EQ8.1 Report

A report is generated detailing eligibility for the LEED v2 daylighting credit EQ8.1 based on % area of the occupied part of the building that has better than the minimum daylight factor. The [CIE overcast sky \(10000 lux\)](#) sky model is used. An example LEED daylight report is shown below.

LEED NC 2.2 Credit EQ 8.1

The aim of the daylighting credit is to encourage and recognize designs that provide appropriate levels of daylight for building users. A pass requires at least 75% of net lettable area in occupied spaces to be adequately daylight, having illuminance over the minimum threshold value.

The results below were calculated using the Radiance simulation engine which provides a detailed multi-zone physics-based calculation of illumination levels on the working planes of the building.

Daylighting data	
Project file	C:\Users\AndyT1\OneDrive - Designbuilder Software Ltd\Documents\DesignBuilder Data\Daylighting Modelling Guide Rev for Help.dsb
Report generation time	20/02/2024 17:33:10
Sky model	1-Standard sky, 2-CIE clear day
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.750
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200
Illuminance lower threshold	270.000

Summary Results	
Total area (m2)	250.0
Total area meeting requirements (m2)	172.3
% area meeting requirements	68.9
LEED NC 2.2 Credit EQ 8.1 Status	FAIL

Eligible zones for daylighting

Block	Zone	Floor area (m2)	Min Illumination (lux)	Working plane area within Limits (%)		
First Floor	Open plan	100.0	83.18	71.7		
Extension	Room	50.0	57.66	55.5		
Ground Floor	Main Office	75.0	68.57	92.7		
Ground Floor	Reception	25.0	15.11	13.0		
Total		250.0	15.11	68.9		

Daylighting Display Options - BREEAM

This topic covers the Daylighting display options specific to the **3-BREEAM** [Simulation type](#).

Option

Select the reports you would like to display. Options for the **3-BREEAM** Simulation type are:

1-Option 4a (point in time) - view BREEAM HEA 01 Option 4a results based on point in time daylight simulation.

2-Option 4b (annual) - view BREEAM HEA 01 Option 4b results based on an annual daylight simulation.

Report type

The report types available depend on the **Option** setting, above.

When the **1-Option 4a (point in time)** Option setting is made, the following report types are available:

- 1-Map** - a daylight distribution contour map on the working plane of the current object is displayed.
- 2-Grid** - a table of summary daylight statistics for each zone is displayed.
- 3-Report** - displays a report with a summary of the results and the number of credits that can be awarded for the selected calculation.

When the **2-Option 4b (annual)** Option setting is made, the map results are not available and you can choose from the following report types:

- 1-Grid** - a table of summary daylight statistics for each zone is displayed.
- 2-Report** - displays a report with a summary of the results and the number of credits that can be awarded for the selected calculation.

Margin

Refer to the Margin help for the [General](#) Simulation type.

Metric Pass Thresholds (Annual)

When one of the grid or report types is set, the annual daylighting metric pass thresholds should be reviewed and, if necessary, changed.

The annual daylighting thresholds are used to define acceptability and are split into two parts:

the [illuminance thresholds](#) which define the acceptable level of lighting at any one point in time over the year, and are defined on the [Calculation options dialog](#), and, metric pass thresholds, which are defined here the Display options panel, as described below.

Changing any of the Metric pass threshold settings in the Display options will typically change the **Area in Range** values shown on the grid or report.

Avg illuminance annual hours

Enter the threshold to be compared with the count of hours failing the average illuminance criterion in each zone, in the [HEA 01 Option 4b annual daylighting calculations](#). If less hours fail than the value entered here then the zone passes.

Min illuminance annual hours

Enter the threshold to be compared with the count of hours failing the minimum illuminance criterion in each zone, in the [HEA 01 Option 4b annual daylighting calculations](#). If less hours fail than the value entered here then the zone passes.

Min area to comply (%)

Enter the minimum threshold for the occupied floor area that needs to comply for the overall building to comply.

Metric Pass Thresholds (Point In Time)

When viewing point in time results and one of the grid or report types is set, the illuminance daylighting pass thresholds should be checked and, if necessary, changed.

Threshold type

Select from:

- 1-Daylight factor**, set the min average daylight factor threshold.
- 2-Illuminance**, set the min and max illuminance threshold values.

Min average DF threshold

When the Threshold type is set to **1-Daylight factor**, set the minimum average daylight factor to be used in calculating the **Floor Area Within Limits** grid outputs.

Min uniformity ratio

When the Report type is set to **3-Report** set the minimum uniformity ratio to be used as a threshold for checking each zone.

Min point daylight factor

When the Threshold type is set to **1-Daylight factor**, set the minimum average daylight factor to be used in calculating the **Floor Area Within Limits** grid outputs.

Min illuminance

When the Threshold type is set to **2-Illuminance**, set the minimum illuminance to be used in calculating the **Floor Area Within Limits** grid outputs.

Max illuminance

When the Threshold type is set to **2-Illuminance**, set the maximum illuminance to be used in calculating the **Floor Area Within Limits** grid outputs.

Min % area adequately daylit

Enter the minimum floor area than must be adequately daylit for the building to pass overall.

Scale

The Scale settings are available when the [Report type](#) is set to the map option. They provide control over how the map legend scale is defined.

Source

The scale of the daylight maps can be defined using data from the building, block or zone level or it can be entered manually. It is a way to define the minimum and maximum values on the scale. Select from one of the following options:

1-Building, where all of the values calculated for the whole building are scanned and the lowest and highest values are used. This allows plots to be generated using a consistent scale even when displaying data at block or zone level.

2-Block, where all of the values calculated for the current block are scanned and the lowest and highest values for are used. This allows plots to be generated using a consistent scale within a block even when displaying data at zone level.

3-Zone, where the scale is tailored to the current zone allowing in many cases a more even distribution of colours through the map. When using this option, images generated for each zone will use a different scale and so will not be so easily comparable as they would be if one of the **1-Building** or **2-Block** options had been used.

4-User defined, where you can enter the minimum and maximum values on the scale manually. This has the advantage of allowing a specific lower and upper scale value to be used. It also allows you to define colours to represent areas that exceed the maximum or are lower than the minimum. For example, in BREEAM option 4a point-in-time daylighting calculations, daylight factors lower than 2% are considered to be inadequately daylit. These areas can be highlighted in a special colour to help provide a clear visual indication of the compliant and non-compliant areas.

Value source

When the **2-Option 2 (point-in-time) Option** setting is made, **Value source** defines whether to view Daylight factor or Illuminance results. Select from:

1-Daylight factor - daylight distribution maps are plotted as daylight factors.

2-Illuminance - daylight distribution maps are plotted as illuminance values.

When the **1-Option 1 (annual) Option** setting is made, **Value source** defines whether to view results as Annual hours or Percentage annual hours. Select from:

1-Annual hours - daylight distribution maps are plotted as the number of annual hours for which the selected criteria are met.

2-Percentage annual hours - daylight distribution maps are plotted as the % of annual hours for which the selected criteria are met. The denominator used in the % calculation is the total number of occupied hours.

Note: when [Source](#) is set to **4-User defined**, **Value source** is hard set to **1-Annual hours**.

User-Defined Scale Settings

Low value

When using the **4-User defined** Scale source you can enter the minimum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

High value

When using the **4-User defined** Scale source you can enter the maximum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

Apply scale "too dark" colour

If you wish to set a specific colour to represent the areas of the working plane that have values below the entered **Low value** criterion then check this box and define the colour in the colour selector below.

Scale "too dark" colour

Select the colour to be used to represent areas of the working plane that are *below* the entered **Low value**. Typically a dark grey or black colour might be used to show areas of the working plane that are inadequately daylight.

Apply scale "too bright" colour

If you wish to set a specific colour to represent areas of the working plane that are *above* the entered **High value** then check this box and define the colour in the colour selector below.

Scale "too bright" colour

Select the colour to be used to represent areas of the working plane that are *above* the entered **High value**.

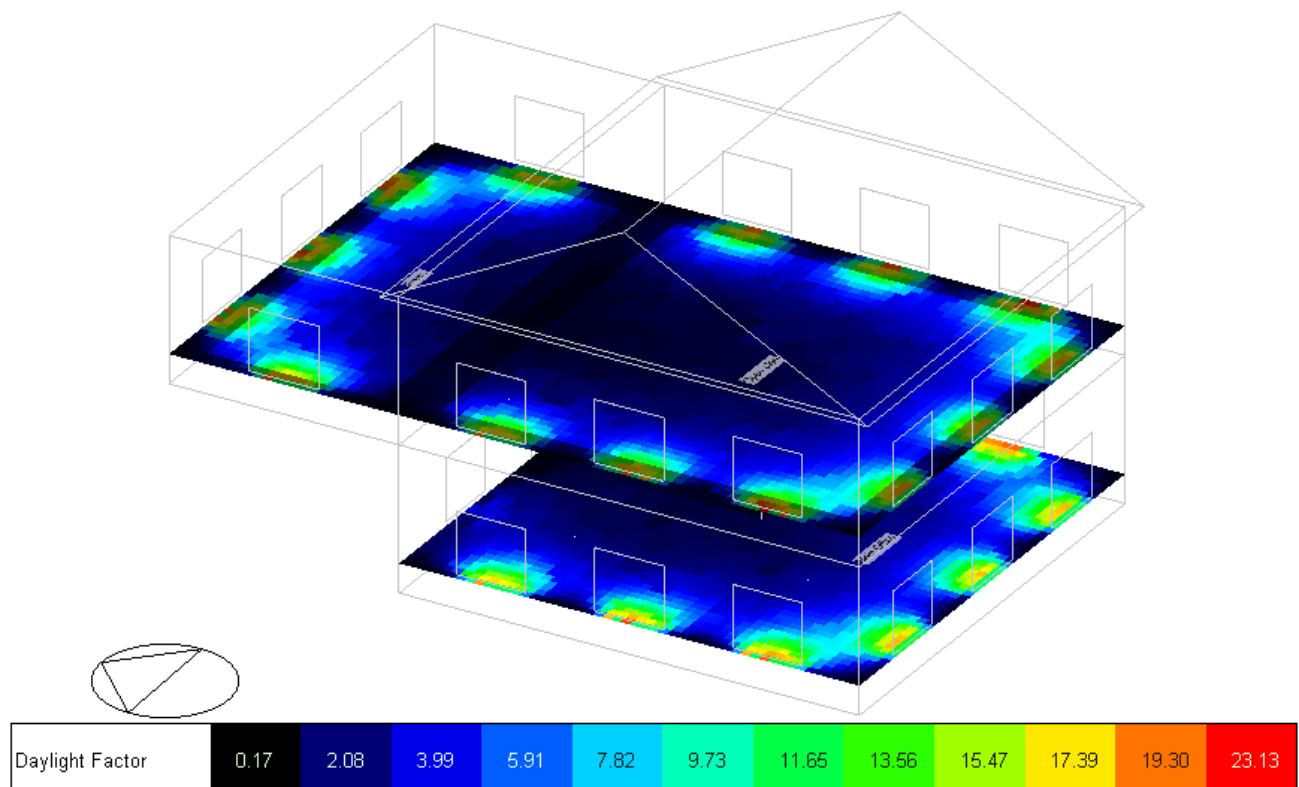
BREEAM Option 4a Results

This section summarises the results generated when the **1-Option 4a (point in time) Option** setting is made.

BREEAM HEA 01 4a Map Output

The distribution of [daylight factors](#) across the working plane can be displayed in map form by setting the Report type to **1-Map**. In this case, the minimum working plane daylight factor threshold is set to 2% behind the scenes.

The screenshot below shows an example of an Option 4a map output.



BREEAM HEA 01 4a Grid Output

The grid output for BREEAM HEA 01 Option 4a can be viewed by setting the Report type to **2-Grid**. It summarises the daylighting performance of each zone in terms of minimum and maximum daylight factor and illuminance values. Uniformity ratios are provided as minimum/average and minimum/maximum.

The following data is listed for each zone:

Floor Area (m2 or ft2)

Floor Area within Limits (m2 or ft2) - the floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.

Floor Area within Limits (%) - the % floor area that is within the min/max illuminance or daylight factor limits set on the Display Options panel.

Average Daylight Factor (%)

Minimum Daylight Factor (%)

Maximum Daylight Factor (%)

Uniformity ratio (Min / Avg)

Uniformity ratio (Min / Max)

Min Illuminance (lux or fc)

Max Illuminance (lux or fc)

This option can be especially useful if the built-in daylight reports don't cover the particular calculation you need to make. In this case you can export the summary data for each zone for further analysis in a spreadsheet.

An example grid output is shown below.

Block	Zone	Floor Area (m2)	Floor Area within ...	Floor Area within ...	Average Daylight...	Minimum Daylight...	Maximum Daylight...	Uniformity ratio (...)	Uniformity ratio (...)	Min Illuminance (L...	Max Illuminance (...)
First Floor	Open plan	100.000	61.467	61.467	3.739	0.788	15.142	0.211	0.052	78.91	1516.32
Extension	Room	50.000	38.655	77.310	4.429	1.214	17.854	0.274	0.068	121.66	1788.83
Ground Floor	Main Office	75.000	54.048	72.064	4.859	0.451	21.914	0.093	0.021	45.25	2196.84
Ground Floor	Reception	25.000	8.754	35.016	2.462	0.334	20.812	0.136	0.016	33.48	2085.60
Total		250.000	162.924	65.170	4.085	0.334	21.914	0.000	0.000	33.48	2196.84

Before viewing grid results, you should ensure that the Threshold set in Display Options defines a minimum daylight factor of 2%:

Threshold	
Threshold type	1-Daylight factor
Min average DF threshold	2.000

BREEAM HEA 01 4a Report

A report detailing eligibility for the BREEAM HEA 01 Option 4a can be viewed by setting the Report type to **3-Report**. These daylighting credits are available based on % area of the occupied part of the building that has better than minimum daylight factors plus better than minimum uniformity ratios.

Specifically, for each zone, a pass requires that at least 80% of net lettable floor area in occupied spaces is adequately daylighted on the working plane, 0.7m above the floor under a uniform CIE overcast design sky. A zone is adequately daylighted if both the following conditions are met:

Average daylight factor is at least 2.0%.

A uniformity ratio of at least 0.3 or a minimum point daylight factor of 0.8%.

You should define the required thresholds before viewing the report by ensuring that the settings under the [Threshold](#) header in the Display Options panel are as follows:

[Min average DF threshold](#) = 2%

[Min uniformity ratio](#) = 0.3

[Min point daylight factor](#) = 0.8

[Min % area adequately daylight](#) = 80%

See screenshot below.

Threshold	
Threshold type	1-Daylight factor
Min average DF threshold	2.000
Min uniformity ratio	0.30
Min point daylight factor	0.80
Min % area adequately daylight	80

These thresholds can be adjusted for cases that are different from the default.

BREEAM Option 4b Results

This section summarises the results generated when the **1-Option 4b (annual) Option** setting is made.

BREEAM HEA 01 4b Grid Output

The grid output for [BREEAM HEA 01 Option 4b](#) can be viewed by setting the Report type to **1-Grid**. It summarises the daylighting performance of each zone in terms of number of hours over the year where the minimum illuminance is above the [Min illuminance threshold](#) and the average illuminance is above the [Avg illuminance threshold](#). These thresholds are set on the Calculations options dialog.

The following data is listed for each zone:

Floor Area (m2 or ft2)

Min Lux Above Threshold (hr) - the number of hours over the year where the minimum illuminance is above the [Min illuminance threshold](#).

Avg Lux Above Threshold (hr) - the number of hours over the year where the average illuminance is above the [Avg illuminance threshold](#).

Block	Zone	Floor Area (m2)	Min Lux Above Threshold...	Avg Lux Above Threshold...
First Floor	Open plan	100.000	2152	2968
Extension	Room	50.000	1540	2945
Ground Floor	Main Office	75.000	1602	2986
Ground Floor	Reception	25.000	193	2423
Total		250.000	0	0

BREEAM HEA 01 4b Report

A report detailing eligibility for the BREEAM HEA 01 Option 4b can be viewed by setting the Report type to **2-Report**.

Option 4b considers that a building is adequately daylit if at least 80% of its zones can achieve at least an average daylight illuminance of 300 lux for 2000 hours per year and the minimum daylight illuminance at the worst lit point is at least 90 lux for 2000 hours.

BREEAM Health and Wellbeing Credit HEA 01

The aim of the daylighting credit is to encourage and recognize designs that provide appropriate levels of daylight for building users.

A building is adequately daylit if at least 80% of its zones can achieve at least an average daylight illuminance of 300 lux for 2000 hours per year and the minimum daylight illuminance at the worst lit point is at least 90 lux for 2000 hours.

Daylighting data	
Project file	C:\Users\AndyT1\OneDrive - Designbuilder Software Ltd\Documents\DesignBuilder Data\Daylighting Modelling Guide Rev for Help.dsb
Report generation time	22/02/2024 16:23:44
Location	LONDON/GATWICK ARPT
Working plane height (m)	0.700
Maximum grid size (m)	0.200
Minimum grid size (m)	0.200

Summary Results	
Total area (m2)	250.0
Total area meeting requirements (m2)	100.0
% area meeting requirements	40.0
BREEAM Health and Wellbeing Credit HEA 01 Status	FAIL

Eligible zones for daylighting

Block	Zone	Floor area (m2)	Min illuminance annual hours (hr)	Avg illuminance annual hours (hr)
First Floor	Open plan	100.0	2152	2968
Extension	Room	50.0	1540	2945
Ground Floor	Main Office	75.0	1602	2986
Ground Floor	Reception	25.0	193	2423

You should define the required thresholds before viewing the report by ensuring that the settings under the [Metric pass thresholds](#) header in the Display Options panel are as follows:

[Avg illuminance annual hours](#) = 2,000

[Min illuminance annual hours](#) = 2,000

[Min area to comply \(%\)](#) = 80

See screenshot below.

Metric pass thresholds	
Avg illuminance annual hours	2000
Min illuminance annual hours	2000
Min area to comply (%)	80

These thresholds can be adjusted for cases that are different from the default.

Daylighting Display Options LM-83-12

This topic covers the Daylighting display options specific to the [4-LM-83-12 Simulation type](#).

The outputs for this Simulation type are the same as those for the **2-LEED** Simulation type when the [LEED version](#) is set to **3-LEED v4** or **4-LEED v4.1** and [Option](#) is set to **1-Option 1 (annual)**. The only difference is that with LM-83-12 there is no report option.

View the [LEED v4.1 Option 1 outputs](#) section.

Daylighting Display Options - Right To Light

This topic covers the Daylighting display options specific to the [5-Right to light \(BS8206\) Simulation type](#).

Option

The following right to light outputs can be displayed:

- [1-Vertical sky component](#)
- [2-Annual probable sunlight hours](#)
- [3-No sky line](#)
- [4-Average daylight factor](#)

Report type

The following report types are available for all Right to light outputs:

- [1-Map](#) - a graphical display of the results is displayed.
- [2-Grid](#) - a table of summary results for each zone or window is displayed (depending on the selected Option).

Scale

The Scale settings are available when the [Report type](#) is set to one of the map options:

Source

The scale of the daylight maps can be defined using data from the building, block or zone level or it can be entered manually. It is a way to define the minimum and maximum values on the scale. Select from one of the following options.

- 1-Building**, where all of the values calculated for the whole building are scanned and the lowest and highest values are used. This allows plots to be generated using a consistent scale even when displaying data at block or zone level.
- 2-Block**, where all of the values calculated for the current block are scanned and the lowest and highest values for are used. This allows plots to be generated using a consistent scale within a block even when displaying data at zone level.

3-Zone, where the scale is tailored to the current zone allowing in many cases a more even distribution of colours through the map. When using this option, images generated for each zone will use a different scale and so will not be so easily comparable as they would be if one of the **1-Building** or **2-Block** options had been used.

4-User defined, where you can enter the minimum and maximum values on the scale manually. This has the advantage of allowing a specific lower and upper scale value to be used. It also allows you to define colours to represent areas that exceed the maximum or are lower than the minimum. For example, for many point-in-time daylighting calculations, daylight factors lower than 2% are considered to be inadequately daylit. These areas can be highlighted in a special colour to help provide a clear visual indication of the compliant and non-compliant areas.

Value source

When [Option](#) is set to **1-Vertical sky component** or **4-Average daylight factor**, **Value source** defines whether to view Daylight factor or Illuminance results. Select from:

1-Daylight factor - daylight distribution maps are plotted as daylight factors. Note that in the case of VSC, the Daylight factor values are displayed as percentages with the illuminance on each window divided by the illuminance on an unshaded horizontal plane.

2-Illuminance - daylight distribution maps are displayed based on illuminance data.

When [Option](#) is set to **2-Annual probable sunlight hours (APSH)**, **Value source** defines whether to view results as Annual hours or Percentage annual hours. Select from:

1-Annual hours - windows are coloured and labelled based on the number of annual hours for which the APSH criteria are met.

2-Percentage annual hours - windows are coloured and labelled based on the percentage of occupied annual hours for which the APSH criteria are met. The denominator used in the % calculation is the total number of occupied hours.

Note: when [Source](#) is set to **4-User defined**, **Value source** is hard set to **1-Annual hours**.

User-Defined Scale Settings

Low value

When using the **4-User defined** Scale source you can enter the minimum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

High value

When using the **4-User defined** Scale source you can enter the maximum scale value to be displayed. The units and meaning of this setting depend on the **Value source** setting, i.e. the data that is being plotted.

Apply scale "too dark" colour

If you wish to set a specific colour to represent the areas of the working plane that have values below the entered **Low value** criterion then check this box and define the colour in the colour selector below.

Scale "too dark" colour

Select the colour to be used to represent areas of the working plane that are *below* the entered **Low value**. Typically a dark grey or black colour might be used to show areas of the working plane that are inadequately daylit.

Apply scale "too bright" colour

If you wish to set a specific colour to represent areas of the working plane that are *above* the entered **High value** then check this box and define the colour in the colour selector below.

Scale "too bright" colour

Select the colour to be used to represent areas of the working plane that are *above* the entered **High value**.

Right To Light Results

Vertical Sky Component (VSC)

[VSC](#) results can be displayed either graphically (as a map) or in grid form.

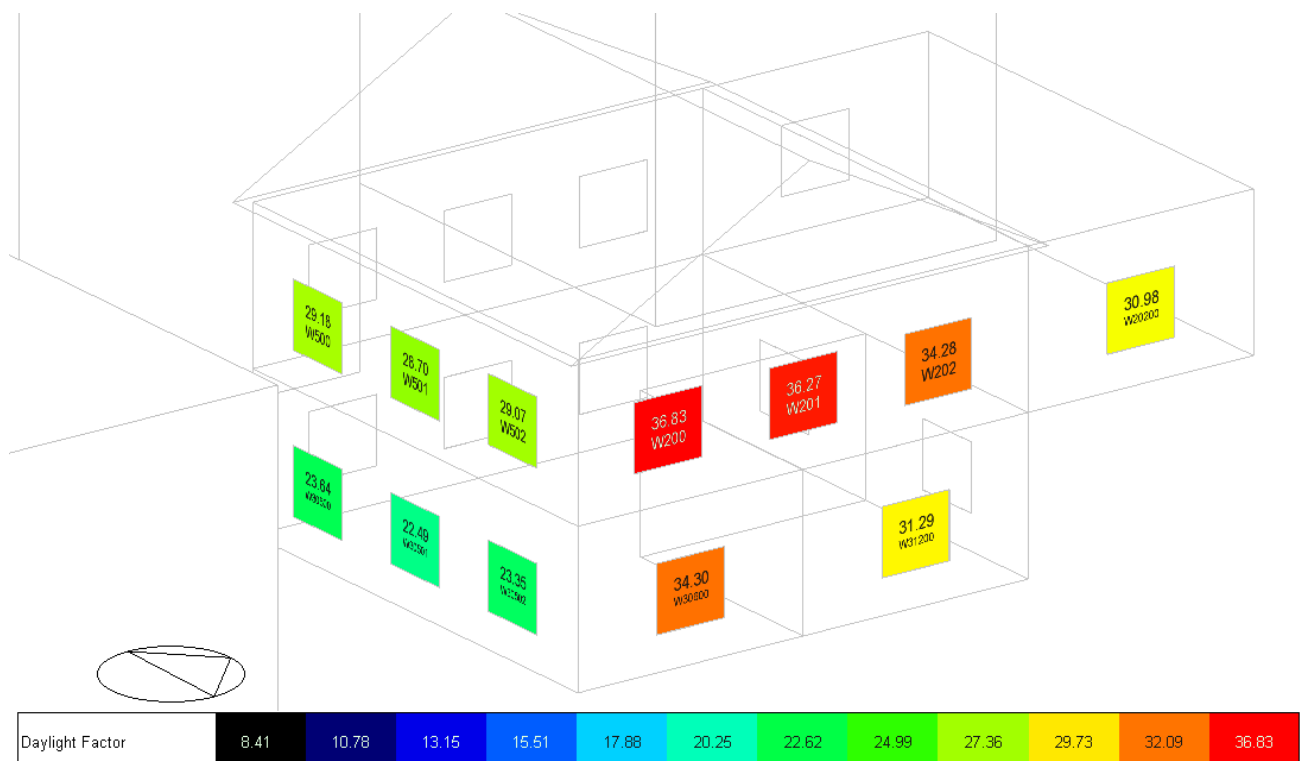
Map

When [Report type](#) is set to **1-Map**, the VSC results are displayed on the model by printing the value on each exterior window and colouring each window according to the scale.

When [Value source](#) is set to **1-Daylight factor**, the VSC is displayed as a percentage:
 $100 \times \text{illuminance on the centre point of the outside face of the window} / \text{illuminance on an unobstructed horizontal surface}$.

Alternatively, when [Value source](#) is set to **2-Illuminance**, the VSC is displayed as the illuminance on the centre point of the outside face of the window.

An example VSC map output is shown below.



Each window also has a unique Window ID label printed on it below the daylight factor or illuminance value. This can be used to cross reference the window when viewing the grid output, see below.

Grid

When [Report type](#) is set to **2-Grid**, the VSC results are displayed in a grid, with each window on a separate line with the following data:

Block
Zone

Window ID can be used to cross reference with Window ID values printed on each window in the map view.

Illuminance (lux or fc)

Daylight factor - the Vertical Sky Component output calculated as described above.

Window Azimuth - orientation of the window expressed as degrees from North. East-facing windows have a value of 90°.

Window Slope - vertical windows have a slope of 90° while horizontal windows have a slope of 0°.

Block	Zone	Window ID	Illuminance (lux)	Daylight Factor (%)	Window Azimuth	Slope
First Floor	Open plan	W200	3077.552	36.831	90.00	90.00
First Floor	Open plan	W201	3030.960	36.274	90.00	90.00
First Floor	Open plan	W202	2864.752	34.284	90.00	90.00
First Floor	Open plan	W400	1982.557	23.727	270.00	90.00
First Floor	Open plan	W401	1863.846	22.306	270.00	90.00
First Floor	Open plan	W402	1906.520	22.817	270.00	90.00
First Floor	Open plan	W500	2438.189	29.179	180.00	90.00
First Floor	Open plan	W501	2397.793	28.696	180.00	90.00
First Floor	Open plan	W502	2429.334	29.073	180.00	90.00
Extension	Room	W20200	2588.609	30.980	90.00	90.00
Extension	Room	W20400	1826.874	21.863	270.00	90.00
Ground Floor	Main Office	W30300	702.684	8.409	0.00	90.00
Ground Floor	Main Office	W30400	1448.816	17.339	270.00	90.00
Ground Floor	Main Office	W30401	1307.838	15.652	270.00	90.00
Ground Floor	Main Office	W30402	1412.037	16.899	270.00	90.00
Ground Floor	Main Office	W30500	1975.602	23.643	180.00	90.00
Ground Floor	Main Office	W30501	1878.843	22.485	180.00	90.00
Ground Floor	Main Office	W30502	1950.928	23.348	180.00	90.00
Ground Floor	Main Office	W30600	2866.173	34.301	90.00	90.00
Ground Floor	Reception	W31200	2614.868	31.294	90.00	90.00
Ground Floor	Reception	W31300	777.145	9.301	0.00	90.00

Annual Probable Sunlight Hours (APSH)

[APSH](#) results can be displayed either graphically (as a map) or in grid form.

Map

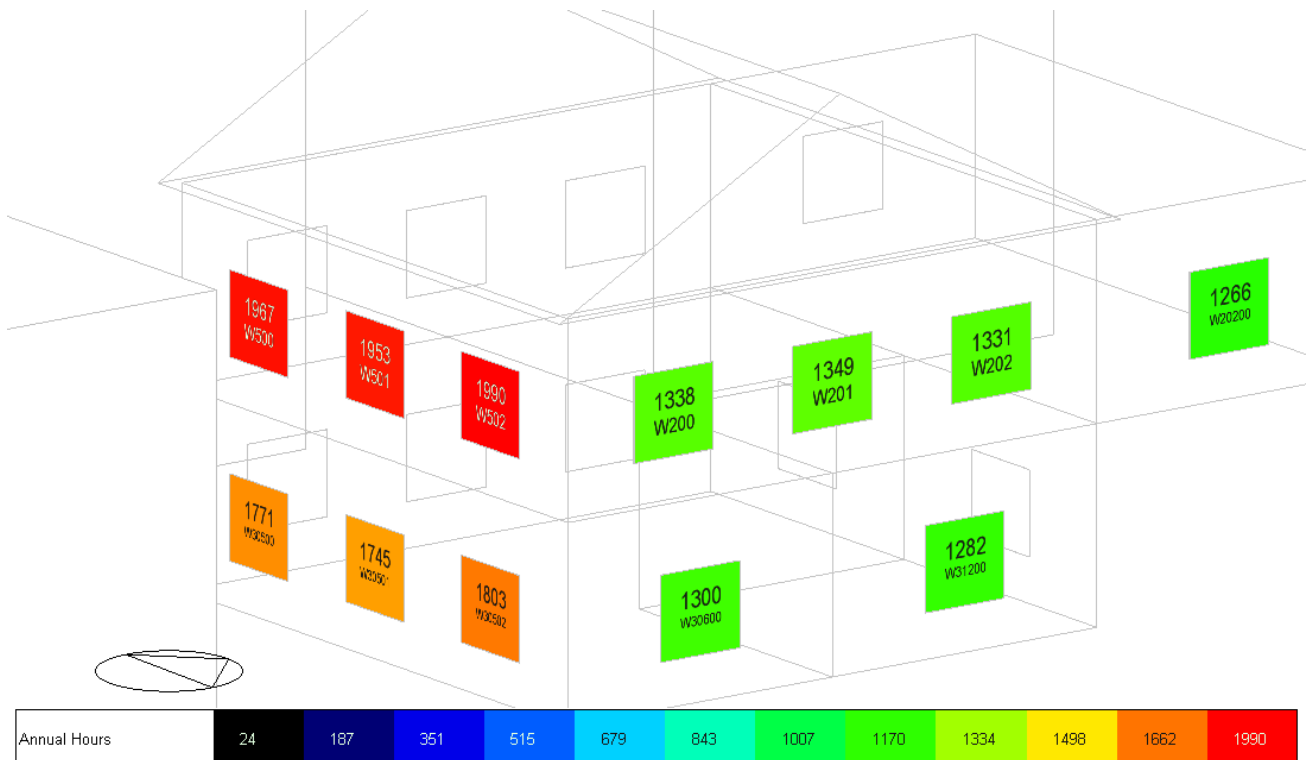
When [Report type](#) is set to **1-Map**, the APSH results are displayed on the model by printing values on each exterior window and colouring each window according to the scale.

When [Value source](#) is set to **1-Annual hours**, the APSH is displayed as the number of hours when direct sunlight fall on the centre point of the outside face of the window.

Alternatively, when [Value source](#) is set to **2-Percentage annual hours**, the APSH is displayed as:

$$\frac{100 \times \text{Number of hours when direct sunlight falls on the centre point of the outside face of the window}}{\text{Number of hours when the sky was clear with sun}}$$

An example APSH map output is shown below.



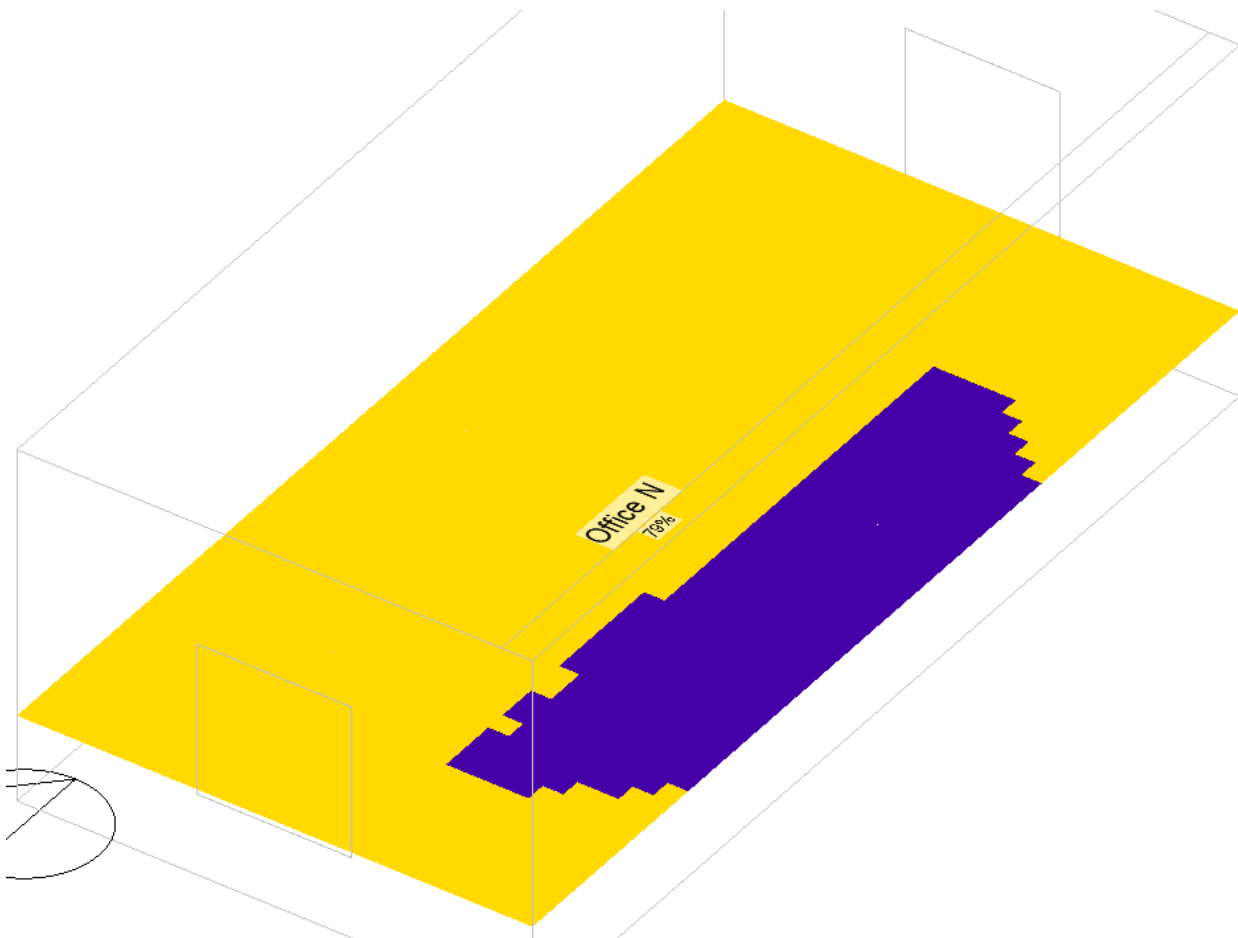
No Sky Line (NSL)

[NSL](#) results can be displayed either graphically (as a map) or in grid form.

Map

When [Report type](#) is set to **1-Map**, the NSL results are displayed graphically on the working plane in the model. Yellow areas indicate the parts of the working plane that have a view of the sky, while purple areas have no view of the sky.

Each zone has a label with the name of the zone and the % of the zone's working plane area where the sky is visible. See screenshot below.



Grid

When [Report type](#) is set to **2-Grid**, the NSL results for all zones are displayed in grid form with the following data:

Block

Zone

Sky Visible %, i.e. the % of the zone's working plane area where the sky can be seen.

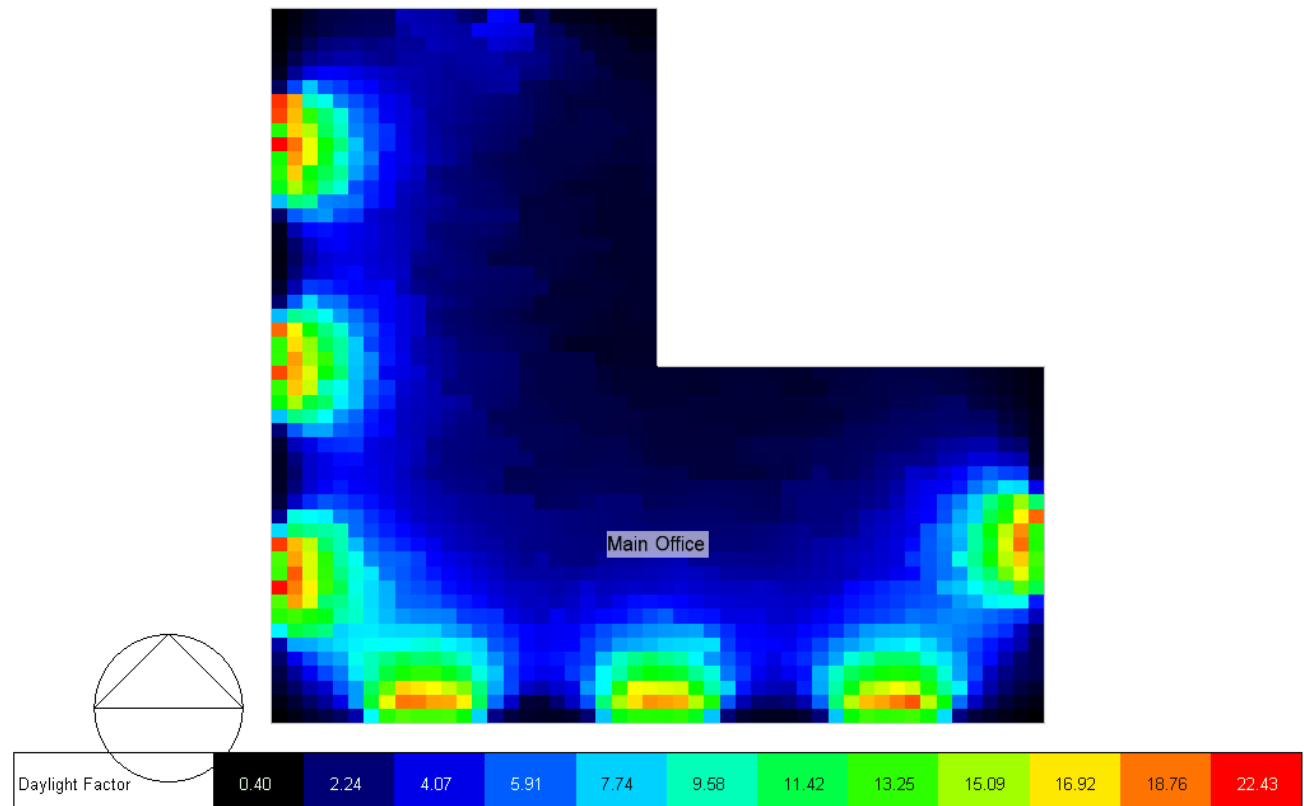
Block	Zone	Sky visible (%)
First Floor	Open plan	100.00
Extension	Office N	79.76
Ground Floor	Main Office	100.00
Ground Floor	Reception	100.00

Average Daylight Factor (ADF)

[ADF](#) results can be displayed either graphically (as a map) or in grid form.

Map

When [Report type](#) is set to **1-Map**, the distribution of the Daylight factors across the working plane is displayed graphically on the model. An example daylight map for a single zone is shown below.



Grid

When [Report type](#) is set to **2-Grid**, the ADF results for all zones are displayed in grid form with the following data:

Block
Zone
Floor Area (m2 or ft2)
Average Daylight Factor (%)
Minimum Daylight Factor (%)
Maximum Daylight Factor (%)

Block	Zone	Floor Area (m2)	Average Daylight Factor (%)	Minimum Daylight Factor (%)	Maximum Daylight Factor (%)
First Floor	Open plan	100.000	3.693	0.432	24.250
Extension	Office N	50.000	1.485	0.149	22.153
Ground Floor	Main Office	75.000	3.142	0.285	23.521
Ground Floor	Reception	25.000	2.058	0.169	21.557

Daylighting Display Options - Green Star

This topic covers the Daylighting display options specific to the **6-Green Star** [Simulation type](#).

Results

Green Star IEQ4

A report detailing the number of Green Star daylighting credit achieved (a maximum of 3 credits are possible). The calculations are based on % area of the occupied part of the building that has better than minimum daylight factors.

Daylighting Metrics

DesignBuilder provides calculations and outputs for the following daylighting metrics:

Point in time, illuminance

[Daylight factor](#)
[Illuminance](#)

Annual, climate based

[Daylight Autonomy \(sDA\)](#)
[Annual Sunlight Exposure \(ASE\)](#)
[Useful Daylight Illuminance \(UDI\)](#)

Right to Light

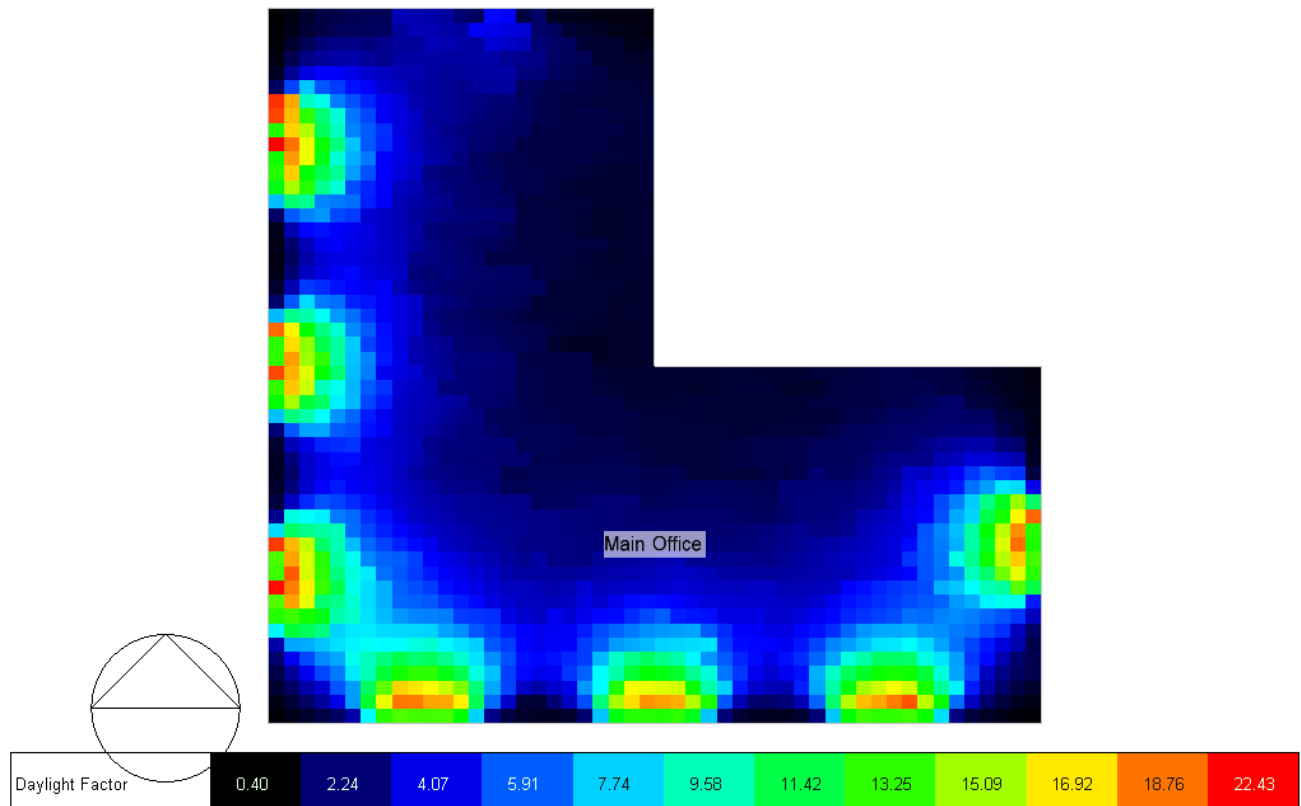
[Vertical Sky Component \(VSC\)](#)
[No Sky Line \(NSL\)](#)
[Annual Probable Sunlight Hours \(APSH\)](#)
[Average Daylight Factor](#)

Point In Time Illuminance Daylighting

The DesignBuilder Radiance illuminance simulations provide a detailed multi-zone physics-based calculation of illumination levels on the working plane of a building using fixed static external lighting conditions and sky distributions.

Daylight factor

Daylight factors are calculated as the ratio of daylight illuminance to that received by an unobstructed horizontal external plane. They are commonly calculated across a grid of points on the working plane using a CIE overcast sky. An average daylight factor can be calculated for a zone as the area-weighted average of the illuminance on each cell on the gridded working plane.



Illuminance

The illuminance on the working plane can be reported instead of daylight factors as described above but without the division by the unobstructed horizontal external plane. Illuminance is used as a key metric in the LEED Daylight Credits Option 2 calculations using a Clear sky.

BREEAM HEA 01 Option 4a

BREEAM HEA 01 Option 4a is a point in time daylight factor-based calculation method used to check that at least 80% of net lettable floor area in occupied spaces is adequately daylit on the working plane, 0.7m above the floor under a uniform CIE overcast design sky. A zone is adequately daylit if both the following conditions are met:

Average daylight factor is at least 2.0%.

A uniformity ratio of at least 0.3 or a minimum point daylight factor of 0.8%.

Annual Climate-Based Daylighting

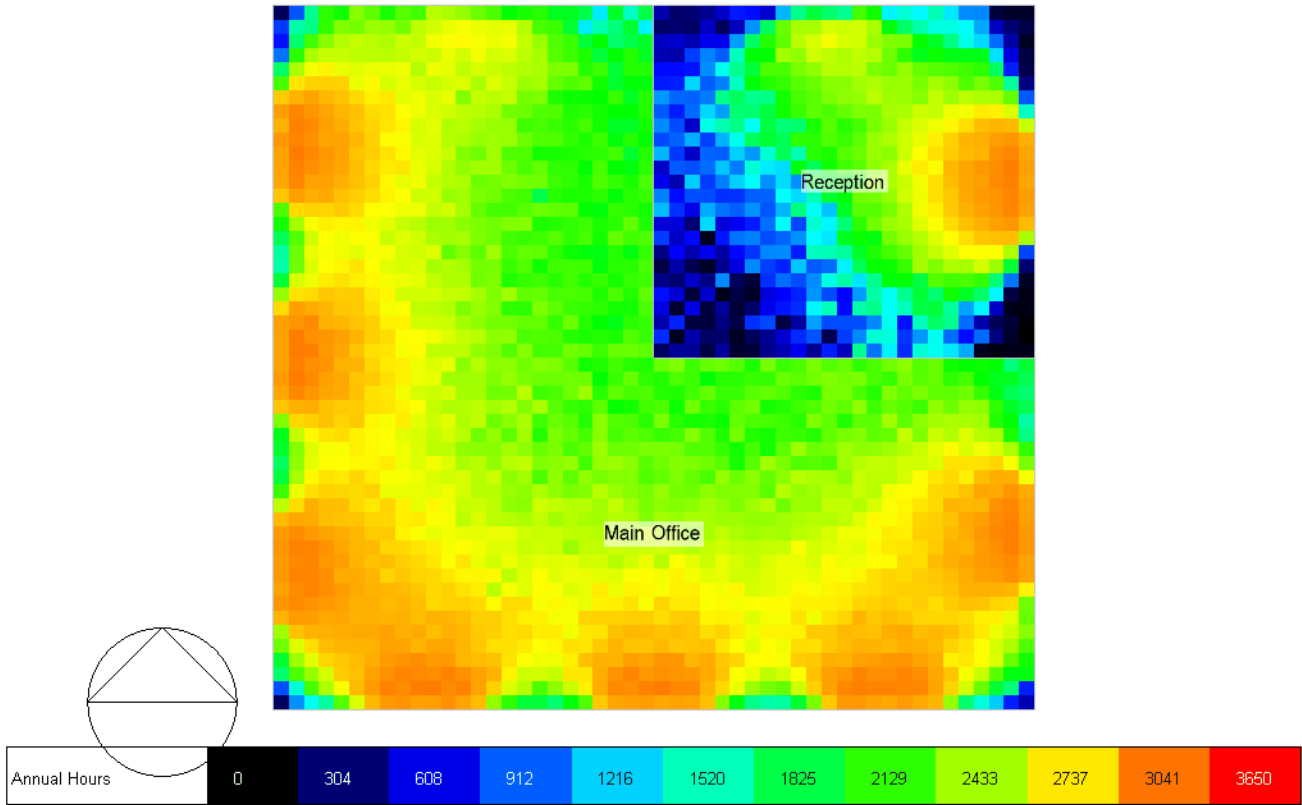
Annual climate-based daylight modelling (CBDM) is the prediction of illuminance on the working plane using realistic sun and sky conditions based on standardised hourly climate data. In DesignBuilder, CBDM evaluations are carried out for a full year at a time-step of an hour in order to capture the daily and seasonal dynamics of natural daylight. The Radiance daylighting engine is used in these simulations. You can read more about Radiance and the CBDM metrics output by DesignBuilder below. CBDM is also known as "dynamic daylighting". These metrics summarise the daylight availability over the whole year and throughout a space. Three main dynamic daylighting metrics calculated by DesignBuilder are described below.

Daylight Autonomy (sDA)

Spatial Daylight Autonomy (sDA) assesses whether a space receives sufficient daylight on the working plane. It is expressed as the percentage of occupied time during the year when the minimum work plane illuminance (typically 300 lux) can be maintained by daylight alone. It defines the extent to which a space has enough natural daylight to allow the prescribed activity to be carried out without the need for electric lighting. This is a recommended metric by the Illuminating Engineering Society of North America (IESNA).

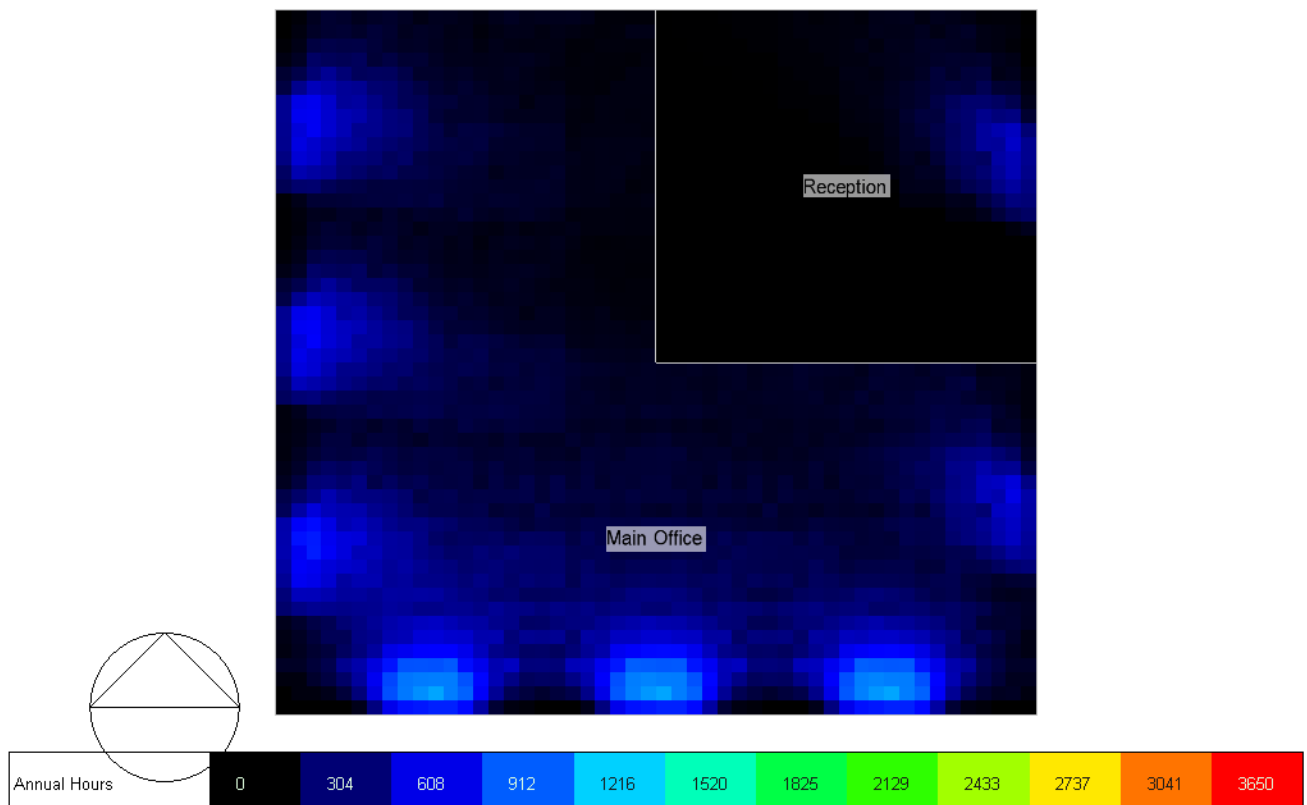
Dynamic window shading as specified in IES LM-83-12 is applied. For more specific information on this, see the [Dynamic shading method](#) section under Calculation options, **1-General** Simulation type.

The most commonly used metric, $sDA_{300/50\%}$, used in LEED v4 and v4.1 Option 1 daylighting calculations, represents the percentage of the floor area that exceeds 300 lux for at least 50% of the occupied time. Unoccupied periods are ignored.



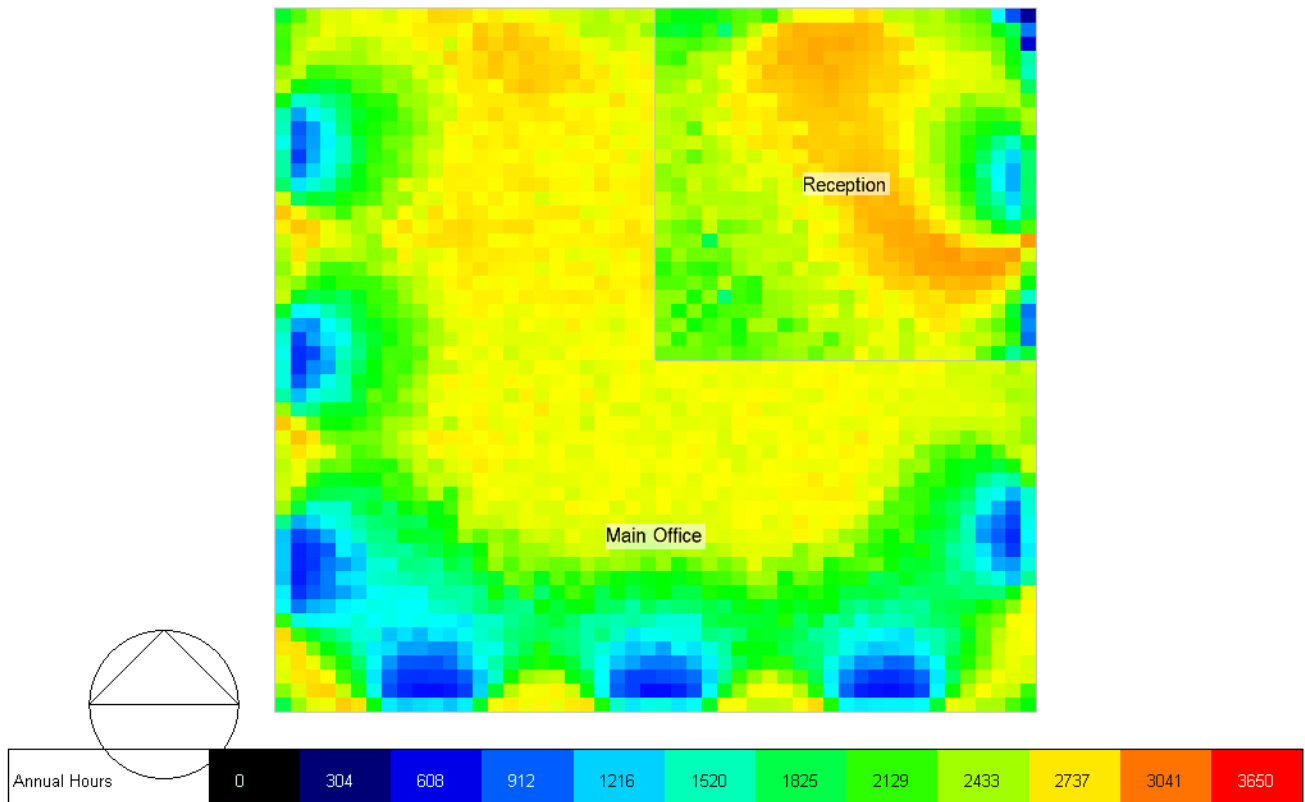
Annual Sunlight Exposure (ASE)

Annual Sunlight Exposure (ASE) is used alongside sDA to quantify the risk of visual discomfort due to glare. At the zone level, it measures the percentage of the working plane exceeding the threshold of 1000 lux for more than 250 occupied hours per year. For grid points on the working level it measures the number of occupied hours per year the point receives direct sunlight greater than the 1000 lux threshold value.



Useful Daylight Illuminance (UDI)

UDI is the percentage of the occupied time when a target range of illuminance values at a point on the working plane is met by daylight alone. At the zone level, it measures the percentage of the working plane that is within the target illuminance range for more than 50% of the occupied hours. It excludes any times when there is inadequate natural daylight or excessive direct sunlight which would give rise to a risk of glare. UDI therefore not only provides information about useful daylight illuminance, but also on the propensity for excessive levels of daylight that are associated with glare, occupant discomfort and unwanted solar gains. In this way the single UDI metric can be used to summarise the overall daylight performance of a space.



BREEAM HEA 01 Option 4b

BREEAM annual daylighting calculations are based on an hourly calculation of illuminance on the working plane of each zone of the model. For each hour and each zone, the average and minimum point illuminance are calculated. If the average illuminance for that hour is above the [Avg illuminance threshold](#), this hour passes for the average criteria. If for the hour the minimum point illuminance is above a given [Min illuminance threshold](#), this hour passes the minimum criteria. The number of hours, for which both the average and minimum criteria are met, is compared to the thresholds defined in Display options. If both criteria are satisfied, then the space is considered satisfactory.

The Option 4b simulations are carried out without window shading being applied.

Note: The worst lit point is defined as the same fixed point over the whole year. To identify this, the number of hours in a year each point on the grid receives the minimum illuminance required is calculated. The worst lit point is the one with the fewest hours. Compliance is then based on whether the number of hours at this point exceeds the threshold.

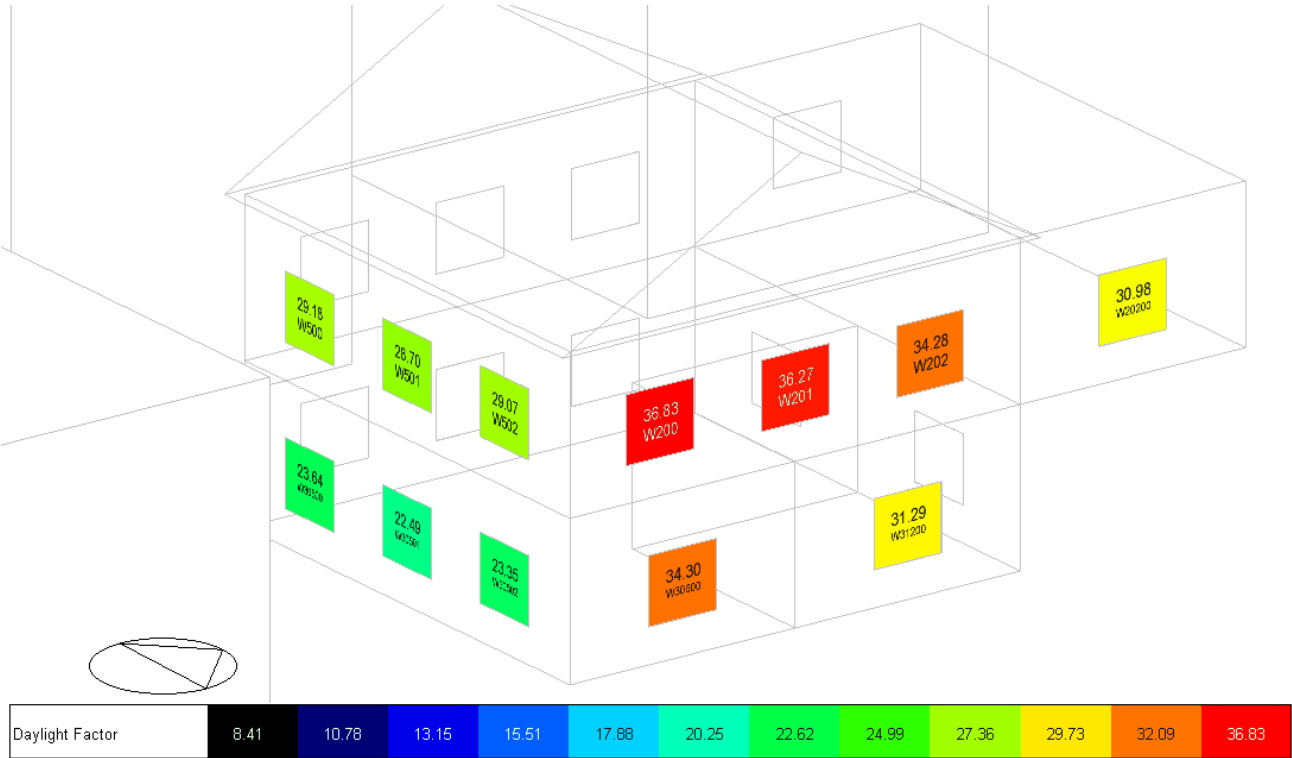
Right To Light

Generally, a right to light refers to the right to receive sufficient light through windows. Right to light generally become an issue when a new development, or proposed development affects the access to light of an adjoining property. Right to light also apply to obstructions caused by trees, hedges etc. DesignBuilder can be used to assess the impact of neighbouring buildings or other obstructions on the windows of an existing building.

Vertical Sky Component (VSC)

Vertical Sky Component is a measure of the amount of sky visible from a given point on the outside face of the window. It is the ratio of the illuminance at that point due to the light received directly from a standard CIE overcast sky to the illuminance on an unobstructed outside horizontal plane under the same sky.

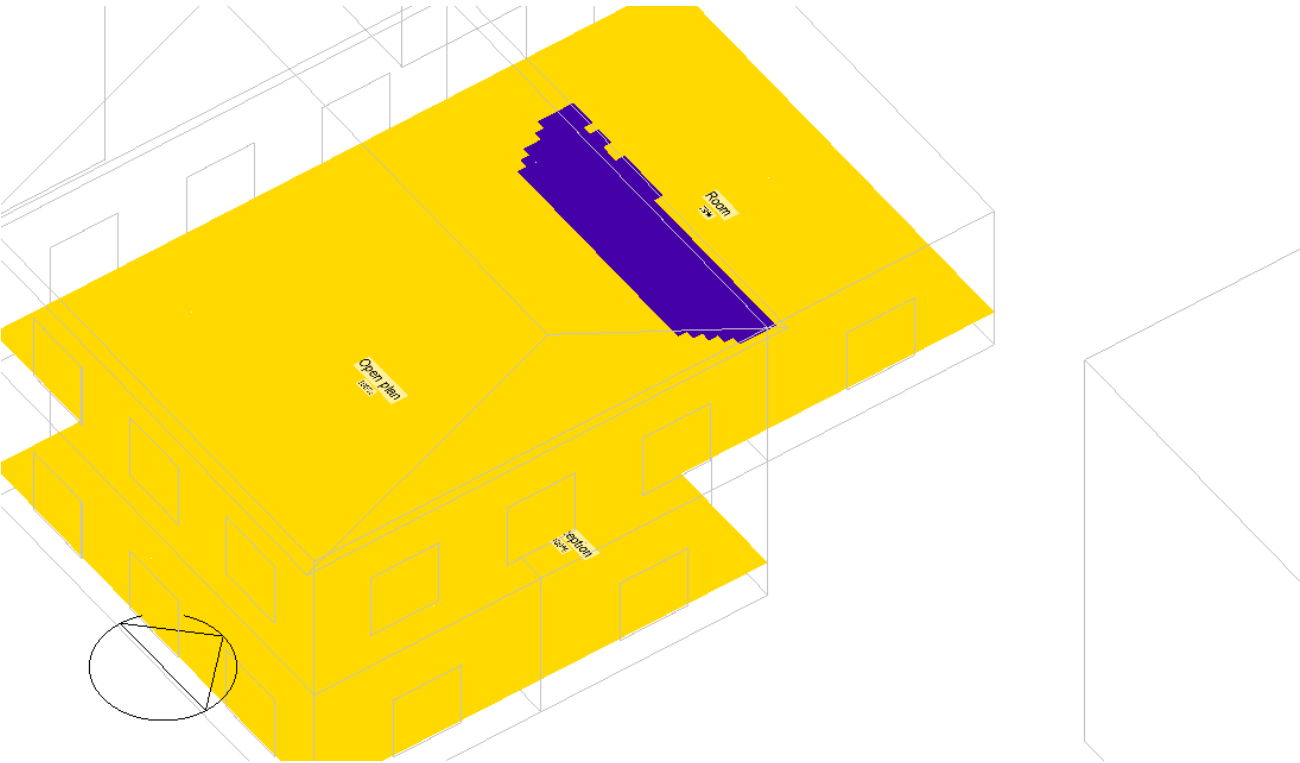
An unobstructed vertical surface will have a VSC of ~40% and 27% is generally considered adequate for new residential developments.



No Sky Line (NSL)

No Sky Line divides areas of the working plane which can receive light directly (i.e. with no bounces) from the sky, from those which cannot.

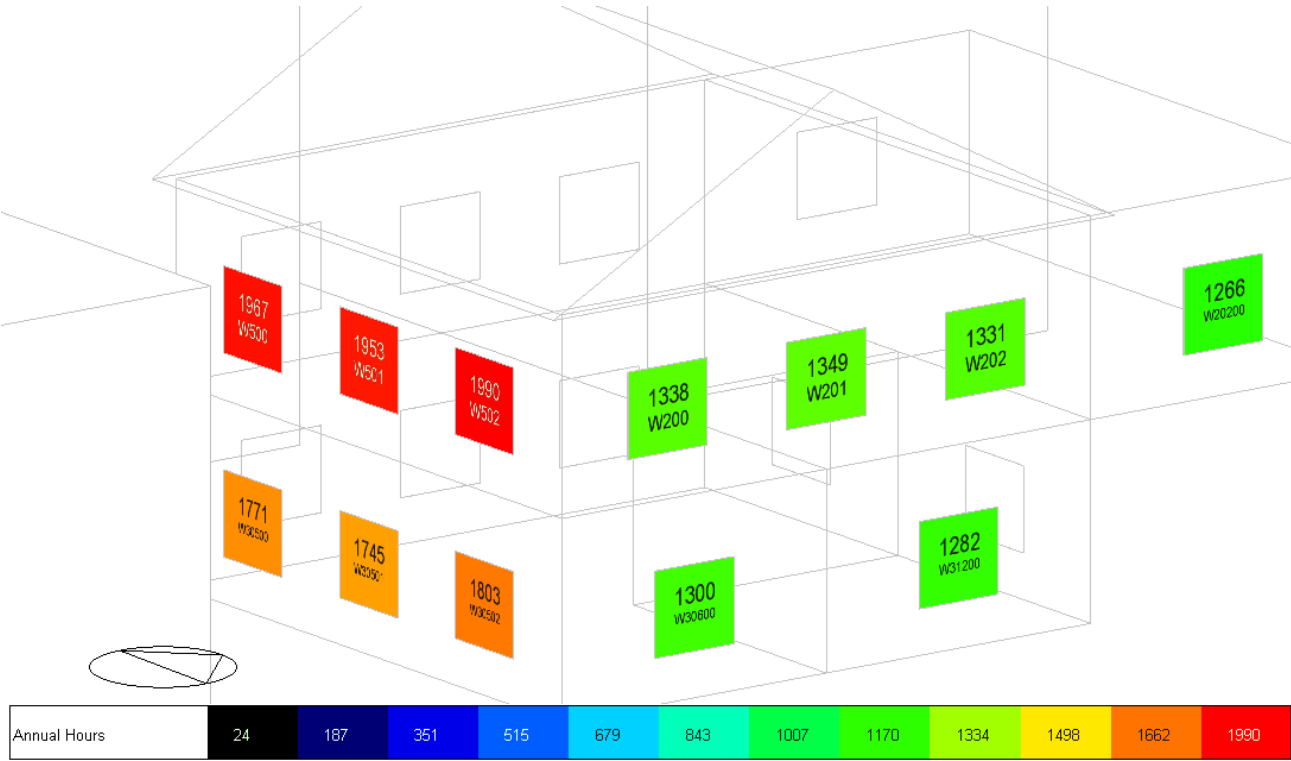
No more than 20% of floor area should be behind the No Sky Line, i.e. with no view of the sky.



Annual Probable Sunlight Hours (APSH)

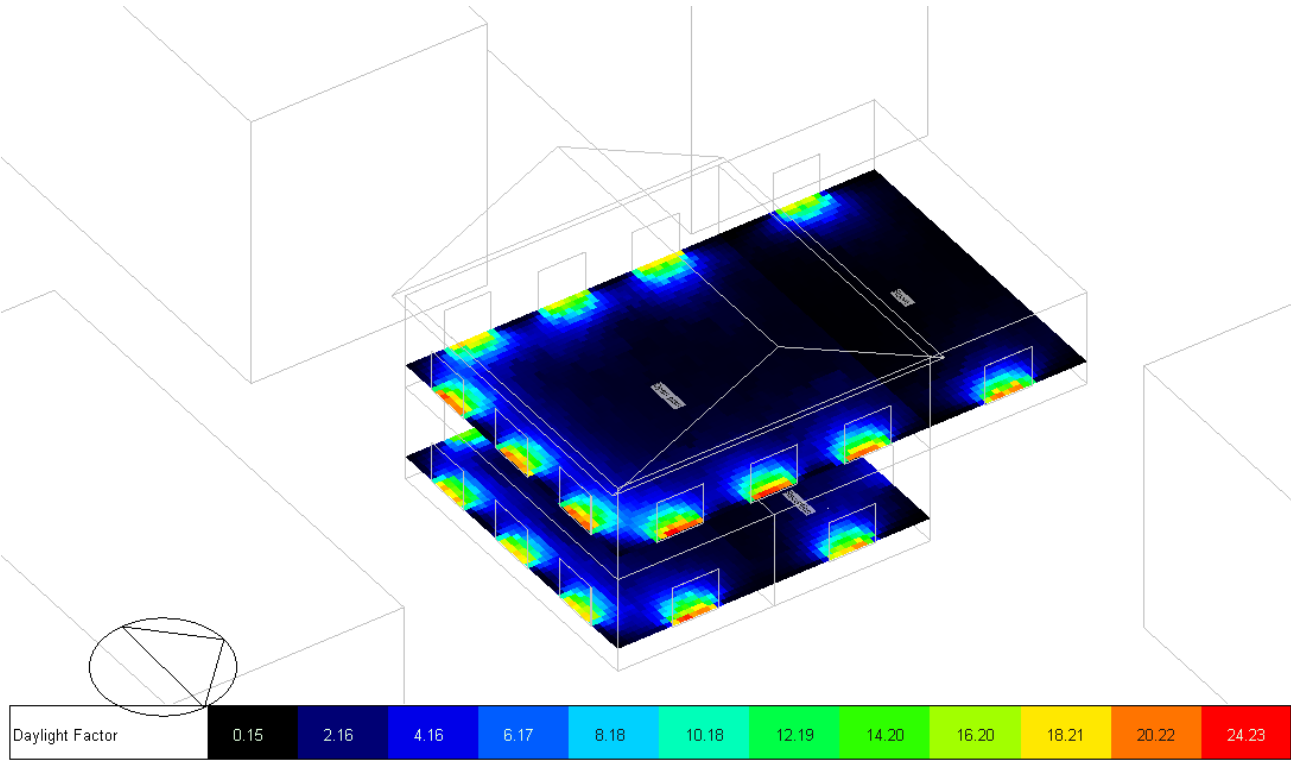
Annual Probable Sunlight Hours is the percentage of direct sunlight hours on the outside face of a window divided by number of hours when sky was clear with sun. Calculations involve counting hours over the year

where the sensor on the mid-point of the window receives any direct light and also the hours where the direct normal illuminance (from weather file) is greater than zero and dividing one by the other.



Average Daylight Factor

The Average Daylight Factor gives the average illuminance from daylight on the working plane relative to that received by an unobstructed horizontal external plane. It is calculated as the area-weighted average of the illuminance on each cell on the gridded working plane. A single value defines the performance of a zone.



Guidance For Calculating Daylighting Credits For LEED, BREEAM, Green Star Etc

DesignBuilder Radiance is ideal for calculating daylighting credits for certification schemes such as LEED, BREEAM and Green Star. In these cases there is often pressure to provide highest possible illuminance reports and, because some settings can cause illuminance values to be lower than expected, this section provides advice on the most appropriate settings to use.

Many of the LEED, BREEAM and Green Star daylighting credit calculations revolve around a "% of floor area having daylight illuminance over a threshold" report for each zone and for the whole building. The table below describes how to get the most accurate results from these calculations.

Setting	Recommended Setting	Advice
Detail template	5-Accurate	You should use the most detailed setting you can afford to wait for. The 1-Fast setting is not intended for project output and should be avoided for all credit calculations as it tends to lead to severe underestimates of illuminance levels. The accuracy increases as the level of detail in the calculation increases up to and including the 5-Accurate setting. The 6-High quality setting is also not recommended for these sort of calculations as it does not add much in terms of accuracy but takes a lot longer than the 5-Accurate setting.
Ambient Bounces	3-8	The 5 built in templates are ideal for "simple daylight scenes that have shallow spaces with relatively large window openings". However in deep plan models or models with small windows or greater external/internal complexity of the model you may want to increase the number of bounces from the default set by the Detail template. Models which have complex exterior geometry may require up to 8 bounces to ensure that the light reflection from exterior building surfaces is correctly treated. Likewise to obtain the correct impact of variations in the floor reflectance you will need to use 5 or more bounces. Tip: If you aren't sure, try experimenting with different values for this setting to test sensitivity and learn impact on results.
Grid size	Max: 0.2m Min: 0.05m	Of course, the finer the overall grid the more accurate the results. Perhaps less obvious however is that using lower values for Minimum grid size can lead to more accurate and higher zone average illuminance results. This is because the small grid cells generated with the low minimum grid size can fill in around the edges of the working plane where often the highest levels of daylight are to be found.
Margin	0	Use a zero margin if you need the entire working plane to be included in the results. This will often lead to higher % areas over daylight thresholds because the edges of the working plane are usually closest to where many of the windows are located.

BREEAM Daylighting Guidance

BREEAM Daylighting Calculations For Mixed-Use Buildings

BREEAM has different daylighting requirements for different space types (offices, educational spaces, retail areas, etc.), and it is currently not possible to run a single daylighting simulation that accounts for all these variations simultaneously. This section provides guidance on how to set up daylighting calculations and how to review and post-process results for buildings with more than one space type.

For Option 4a (Point in Time) Simulations

Only one daylighting simulation is required for all space types
Only requires changes to Display options

The four Display options settings that need to be controlled are:

Depending on the [Threshold Type](#), either [Min average DF threshold](#), or [Min](#) and [Max illuminance thresholds](#)

[Min uniformity ratio](#)

[Min point daylight factor](#)

[Min % area adequately daylit](#)

Threshold	
Threshold type	1-Daylight factor
Min average DF threshold	2
Min uniformity ratio	0.30
Min point daylight factor	0.80
Min % area adequately daylit	80

For Option 4b (Annual) Simulations

Requires changes to both Calculation options and Display options

Has two separate illuminance threshold fields in the Calculation options that need to be set before the simulation starts.

May require multiple simulations if the model has more than one space type

The five variables that need to be controlled are:

[Min illuminance threshold](#) (Calculation option)

[Avg illuminance threshold](#) (Calculation option)

Option 4b (annual) Options	
☐ Override zone occupancy schedule	
Min illuminance threshold (lux)	90
Avg illuminance threshold (lux)	300

[Avg illuminance annual hours](#) (Display option)

[Min illuminance annual hours](#) (Display option)

[Min area to comply \(%\)](#) (Display option)

Metric pass thresholds	
Avg illuminance annual hours	2000
Min illuminance annual hours	2000
Min area to comply (%)	80

The main recommendations for 4b (Annual) calculations and models with multiple space types are:

Use the [Include zone in Radiance daylighting calculations](#) checkbox to selectively include/exclude zones

Run multiple simulations in grouped batches (by space type)

Be careful with zones with internal transparent surfaces and openings between different space types, as they should be included to pass light to inner zones.

This approach allows for proper and efficient assessment of each space type according to its specific BREEAM requirements.

Radiance Reference

Radiance Implementation

Radiance was developed at LBNL California, for predicting the distribution of visible radiation in illuminated spaces. It is internationally recognised as one of the most accurate and extensively validated lighting simulation tools available. The implementation of Radiance within DesignBuilder uses the full 3-D geometric model of the physical environment as seen on the [Visualisation](#) screen as input, and applies a "backward ray-tracing" method in which light is followed backwards from the working plane to the source(s) to produce a map of illuminance on the working plane. The calculations allow light to be transmitted through exterior and interior windows, and the shading and reflective effect of local shading devices and component and assembly blocks is included.

Cost And Carbon

The **Cost and Carbon** screen allows you to review the projected construction cost of the building design and the embodied carbon in the building fabric.

[Construction Cost](#)
[LCA / Embodied Carbon](#)

Construction Cost

Depending on the [Cost model](#) selected on the Building Cost Summary dialog, the construction cost report will be generated and displayed using one of 3 formats:

1-Basic, a simple report including building only construction costs.

2-UNIFORMAT II, a summary report based on standard ASTM UNIFORMAT II conventions.

3-NRM1, a summary report based on standard RICS NRM1 conventions.

The outputs of the Cost calculation can be included automatically within the [Construction cost](#) data to be fed to EnergyPlus for LCC economic analysis and in optimisation simulations.

Note: Construction cost calculations for models where the [zone multiplier](#) is used are not always 100% accurate because the construction cost of internal surfaces between multiplied zones and adiabatic component blocks can be over-estimated.

Note: In order for the cost of renewable systems such as solar PV and hot water panels and wind turbines to be included in the Construction cost report (and any cost-related KPIs used in an optimisation) those generators must be specified in an "Electric Load Centre" on page 526.

LCA And Embodied Carbon

Embodied carbon in buildings refers to the greenhouse gas emissions associated with the entire life cycle of a building, from the extraction of raw materials to its eventual disposal. This includes emissions from manufacturing, transportation, construction, maintenance, refurbishment, and demolition. It's essentially the "upfront" carbon footprint of a building, before it even begins to be used. It is typically estimated from the energy used to extract and transport raw materials as well as emissions from manufacturing processes. The embodied carbon of a building can include all the emissions from the construction materials, the building process, all the fixtures and fittings inside as well as from deconstructing and disposing of it at the end of its lifetime.

It can be used to compare the environmental impacts of different building materials, designs and construction processes. It can help to identify elements which are carbon-intensive and promote alternative options which reduce the amount of CO₂ released. It can be used to design policies that reduce the CO₂ emissions from the construction sector.

Used together with the operational carbon (i.e. the carbon emitted due to use of the building for heating, cooling, equipment use etc), the embodied carbon can be included as part of a whole life cycle analysis (LCA) to understand the overall impact of the building on the environment over its whole life cycle.

DesignBuilder provides 2 methods to calculate environmental impact of the building construction (as opposed to its operation):

"One Click LCA Integration" on page 2058 where material quantities from the model are exported to the One Click LCA website for further analysis .

"DesignBuilder Embodied Carbon Calculations" on page 2066 where the embodied carbon calculations are carried out entirely within DesignBuilder.

Generally, we recommend the One Click LCA export for any LCA analysis for certification purposes as their material databases are certified and maintained up to date with current figures. Note that an additional licence must be purchased from One Click LCA to use their software. The DesignBuilder internal embodied carbon calculations, on the other hand, are intended for quick indicative calculations and for parametric studies such as optimisations where LCA or embodied carbon are KPIs in the analysis.

Technical Note: Geometry Conventions in LCA Carbon Assessments

In general, you should use the [2-Outer Zone geometry and surface areas](#) option when carrying out LCA and embodied carbon calculations. This is because the **2-Outer** setting typically provides a more realistic assessment of the material quantities than the **1-Inner** option. In particular, depending on how the blocks have been drawn and whether any [fixed thicknesses](#) surface have been applied, the **1-Inner** option may result in vertical gaps between floors and horizontal gaps between zones on the same level. This may in turn result in an underestimate of the surface areas in the LCA and embodied carbon results, regardless of which of the 2 methods are used.

One Click LCA Integration



The recommended way to carry out a complete Building Life Cycle Assessment, is through our native integration with [One Click LCA®](#), a powerful yet easy to use LCA and LCC platform. The integration enables DesignBuilder users to:

Transfer DesignBuilder energy models quickly and easily to One Click LCA for additional materials analyses.

Achieve BREEAM Mat 01 LCA and LCC credits

Achieve LEED v4 and v4.1 LCA materials credits.

Use One Click LCA's material database to get a true picture of the carbon & LCA performance.

Note: The One Click LCA integration uses the material names selected in the DesignBuilder model, mapping them to certified data contained in the extensive One Click LCA database. The [embodied carbon data](#) entered in the DesignBuilder Materials database is not used in these analyses.

User Guide

DesignBuilder's One Click LCA integration allows you to automatically submit the materials included in an EnergyPlus model to One Click LCA for analysis and reporting. The process is described below.

1. Set the model up with the required material definitions. This step requires little or no extra work for most simulation models.

The materials data is extracted from the model as follows:

Opaque materials - material quantities are calculated by summing the area and thickness of construction [material layers](#) for each surface in the model. The embodied carbon data for each material is defined later within One Click LCA by mapping project materials to data points in their database. DesignBuilder library materials should mostly be automatically mapped.

Glazing

For glazing defined using [Detailed layers](#), the quantity of glass in the model is calculated using the area of windows x number of panes of each.

For glazing defined using the [Simple method](#), the number of panes data is used to multiply a standard glass element of the defined window area.

Component blocks

Component blocks defined with their [Level](#) set to **1-Building** will be included in the LCA analysis. Component blocks defined with their [Level](#) set to **2-Site** are not included. The rationale is that such component blocks are typically used to represent other buildings, site objects such as trees and other site objects, and these should not be excluded in an LCA analysis. Additionally, component blocks can individually be switched out of the analysis by unchecking the [Shades and reflects](#) checkbox.

For [Non-flat component blocks](#) material quantities are included in the report on a *volumetric* basis using the [Component block material](#) setting.

For [Flat component blocks](#) with ~ zero thickness, material quantities are included in the report on an *area* basis. They use the same [Component block material](#) setting.

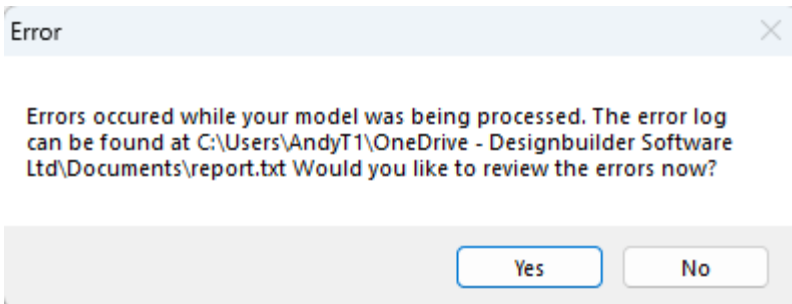
Local shading - Local shading surfaces are included on an *area* basis using the Local shading [Blade material](#) setting.

PV panel - a separate line item for PV panels is included in the data submitted.

Note: PV panels must be connected to a [Load centre](#) to be included in the analysis.

2. When the model has been set up as required, click on the **One Click LCA** toolbar icon  to start the submission process.

3. As DesignBuilder processes the model, if any errors are encountered, you will be asked if you would like to review them.



Click **Yes** to open the error log file in the application associated with the .txt file extension, typically a text editor such as Notepad.

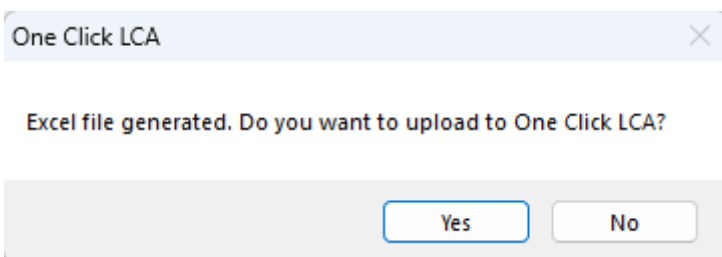
If errors are reported, the usual reason is that materials referenced by constructions or component blocks are missing from the model. See example errors below.

```
Getting Model Material Data
Building Name: Building
  Block Name: Block 5
    Zone Name: Zone 1
      Surface Name: Wall - 7.006 m2 - 180.0°
        Adjacency Number: 0
          Get Surface Construction Data
          Get Zone From Handle
          Surface Type is Wall
          Partition Type is not Internal
          Wall Construction
          Layer Number: 2
14:40:10 - _AirMed50mm: Unable to retrieve material record for layer 2.

Processing Component Blocks
Building Name: Building
  Add Component Model Element
  Is Not Solar Collector
14:40:10 - Building:Component block 3: Unable to retrieve component block material
```

4. You will be prompted to save the materials inventory in a spreadsheet report format. Enter the name of the report to be saved on your local machine and press the **Save** button.

5. Once the report has been created, you will be asked whether you would like to upload it to One Click LCA.



6. At this point, you can check the materials inventory spreadsheet you saved locally to make sure it reflects the data you would expect to see for your model, and if necessary, make any adjustments before uploading it. The screenshot below shows an example materials inventory spreadsheet for a simple model.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	CLASS	MATERIAL	QUANTITY	QTY_TYPE	THICKNESS_MM	THICKNESS_IN	LENGTH_M	LENGTH_FT	AREA_M2	AREA_SQ_FT	VOLUME_M3	VOLUME_CU_FT	VOLUME_CU_YD	UNIT_PCS
2	SLAB	Urea Formaldehyde Foam	575.7144348	M2	132.7	5.224409483	0	0	575.7144348	6196.938602	76.39730549	2698.34038	99.92390156	1
3	SLAB	Cast Concrete	575.7144348	M2	100	3.9370079	0	0	575.7144348	6196.938602	57.57144348	2033.414002	75.30060404	1
4	SLAB	Floor/Roof Screed	575.7144348	M2	70	2.75590553	0	0	575.7144348	6196.938602	40.30001043	1423.389801	52.71042283	1
5	SLAB	Timber Flooring	575.7144348	M2	30	1.18110237	0	0	575.7144348	6196.938602	17.27143304	610.0242005	22.59018121	1
6	ROOF	Asphalt 1	575.7144348	M2	10	0.39370079	0	0	575.7144348	6196.938602	5.757144348	203.3414002	7.530060404	1
7	ROOF	MW Glass Wool (rolls)	575.7144348	M2	144.5	5.688976416	0	0	575.7144348	6196.938602	83.19073582	2938.283233	108.8093728	1
8	ROOF	Air gap >=25mm	575.7144348	M2	200	7.8740158	0	0	575.7144348	6196.938602	115.142887	4066.828004	150.6012081	1
9	ROOF	Plasterboard	575.7144348	M2	13	0.511811027	0	0	575.7144348	6196.938602	7.484287652	264.3438202	9.789078525	1
10	WALL	Brickwork Outer	237.628238	M2	100	3.9370079	0	0	237.628238	2557.809067	23.7628238	839.2990644	31.08059965	4
11	WALL	XPS Extruded Polystyrene - CO2 Blowing	237.628238	M2	79.5	3.129921281	0	0	237.628238	2557.809067	18.89144492	667.2427562	24.70907672	4
12	WALL	Concrete Block (Medium)	237.628238	M2	100	3.9370079	0	0	237.628238	2557.809067	23.7628238	839.2990644	31.08059965	4
13	WALL	Gypsum Plastering	237.628238	M2	13	0.511811027	0	0	237.628238	2557.809067	3.089167094	109.1088784	4.040477955	4
14	WINDOW	Generic PYR B CLEAR 3MM	98.33067344	M2	3	0.118110237	65.55378229	215.0714642	98.33067344	1058.42256	0.29499202	10.41907009	0.38583499	16
15	WINDOW	Generic CLEAR 3MM	98.33067344	M2	3	0.118110237	65.55378229	215.0714642	98.33067344	1058.42256	0.29499202	10.41907009	0.38583499	16
16	WINDOW	Painted Oak	8.955378229	M2	20	0.78740158	131.1075646	430.1429284	8.955378229	96.39488901	0.179107565	6.326049994	0.234263847	32
17	OTHER	Project component block material	12.18872916	M2	0	0	24.37745818	79.97853734	12.18872916	131.1983888	0.304718229	10.76259819	0.398556391	1
18	BUILDINGTECH	photovoltaic panel	22.0443748	M2	0	0	8.446555316	27.71179566	22.0443748	237.2836756	0.551109409	19.46509451	0.720823882	1

7. When you are happy with the report, click on the Yes button to upload it to One Click LCA.

8. To access the data uploaded on the One Click LCA website, you must first sign in using the web form shown below.



English v

Sign in to your account

Email

Sign in

Or sign in with



Google



Microsoft

New user? [Register](#)

9. When signed in you will be presented with the One Click LCA "Settings" page where the process of refinement and reporting begins.

The screenshot shows the 'One Click LCA' software interface. At the top, there's a navigation bar with the logo, an '+ Add' button, and links for 'Licences', 'HELP', and a user profile 'Simon'. Below this, a breadcrumb trail reads 'Main > Import data'. A progress bar contains eight steps: DATA (checked), SETTINGS (active), CLASSIFY, FILTER, COMBINE, REVIEW, MAPPING, and UPDATING. Under the progress bar, checkboxes for 'Review manually' are shown for each step: DATA (unchecked), SETTINGS (checked), CLASSIFY (unchecked), FILTER (unchecked), COMBINE (checked), REVIEW (unchecked), MAPPING (checked), and UPDATING (unchecked). A blue banner states 'You are importing 18 rows'. The 'Settings' section has buttons for 'Cancel', 'Download Excel', 'Send data to another user', and 'Continue'. A note says: 'Choose the project, design and calculation tool that you want to use for import. If you are unsure on what to do next, please visit our [Customer Support Center](#).' Below this are five sections: 'Choose the project or entity' (DesignBuilder), 'Choose or create a design' (New design), 'Choose the tool' (LCA for BREEAM Int'l/ES/UK), 'Filtering settings' (Building without building technol), and 'Existing data preservation' (Replace existing data with import). At the bottom, two checkboxes are checked: 'Remove empty data' and 'Convert all to metric unit system'.

For more information on the integration from the OCL side, see:

[One Click LCA DesignBuilder Integration Help Article.](#)
[Updated Import Process in One Click LCA](#)

Note: You should [contact One Click LCA](#) to obtain a licence to access their online system from DesignBuilder.

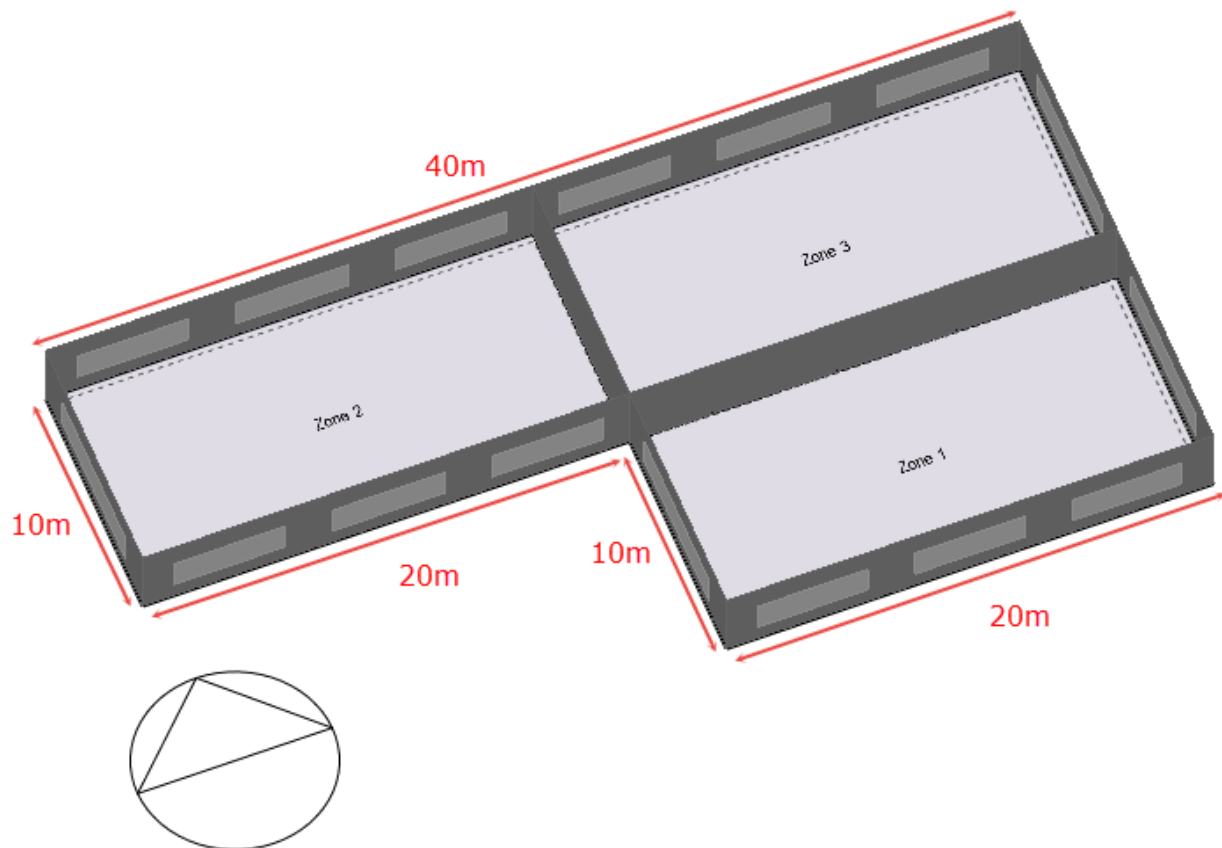
One Click LCA Example

The example below shows how to use DesignBuilder to set up and run a basic one-click LCA analysis. It includes an assessment of the embodied carbon in the opaque and glazed constructions, shading components, and a PV panel of a simple 2-storey model.

Model Setup

Geometry

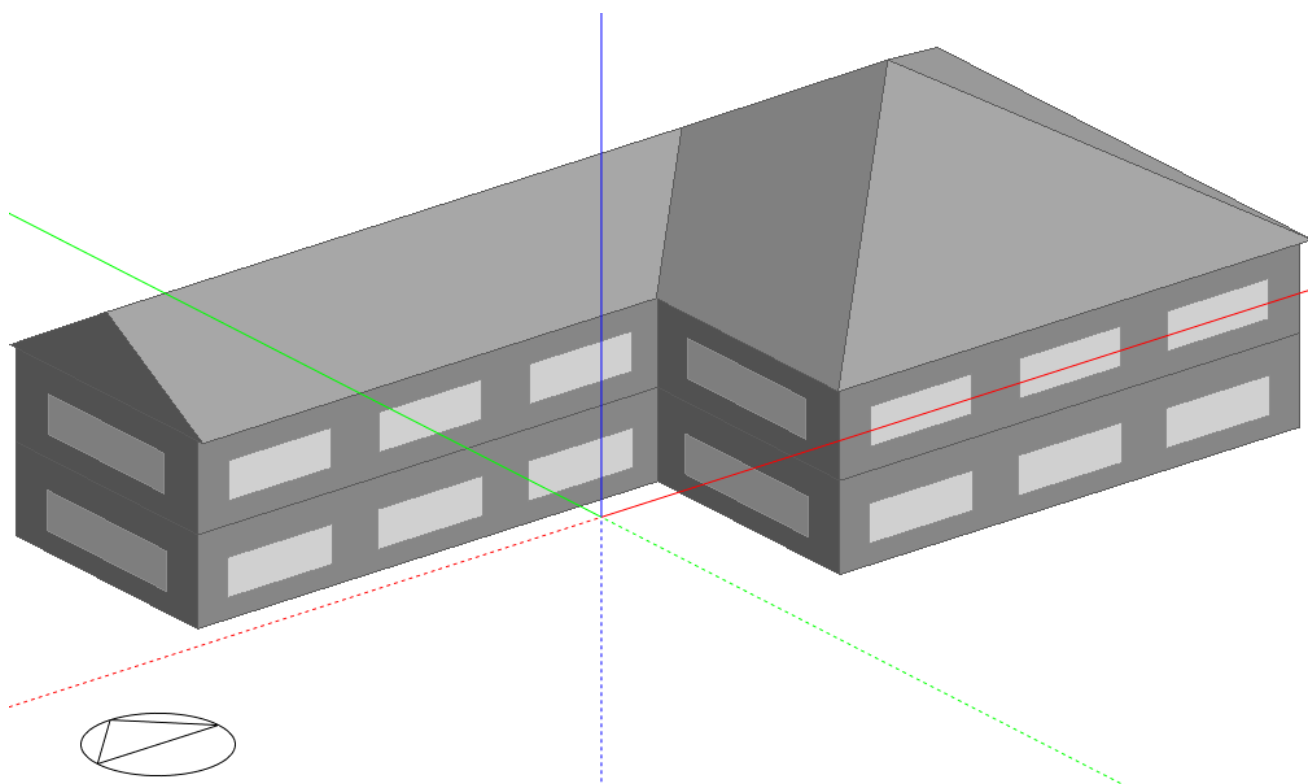
1. Start by opening DesignBuilder and creating a new project.
2. The location of the project is not important for our purposes. In this example you can choose the default location.
3. Add a building and create an L-shaped block with the dimensions below and partition it into three equal-sized zones.



Clone this block to create an identically dimensioned second block.

Draw an unoccupied gable roof block on top of that.

The model should now look similar to the screenshot below.



Constructions

4. Next, create a custom wall construction by editing the default "Project wall".
 - a. Change brick thickness to a typical thickness of 0.1025m.
 - b. Change insulation thickness to 0.1m.

Glazing

5. Similarly, for glazing, create a custom glazing component by editing the default "Project external glazing" component.
 - a. Choose "Generic Clear 4mm" for both inner and outer layers.
 - b. Choose "ARGON 13MM" for the cavity in between.

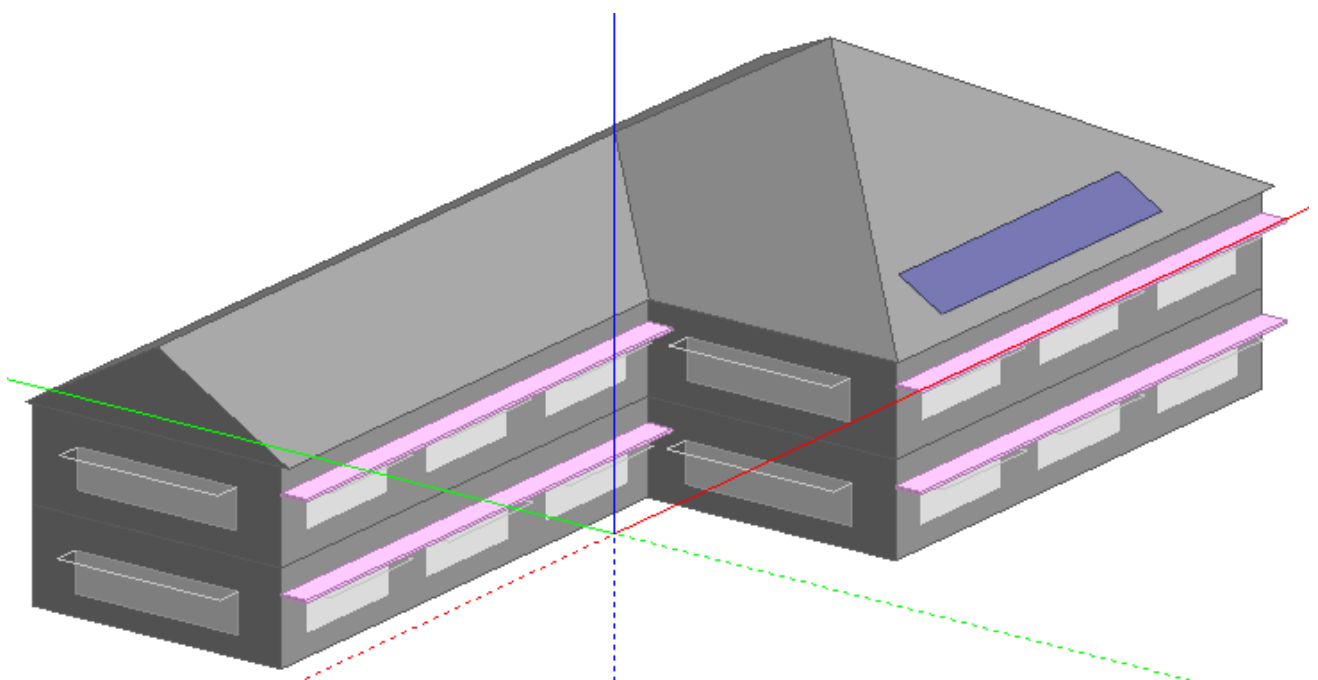
Shading

6. Shading will be represented using a set of 4 component blocks.
 - a. Draw a 1m overhang component blocks over one of the 20m the south-facing facades.
 - b. Clone it 3 times to give 4 component block overhangs.
 - c. Alternatively, you could use a Local shading component to achieve a similar effect, setting the shading material on the local shading dialog.

PV Panel

7. Now add a PV panel.
 - a. Draw a large 12 x 2m panel on the South-facing roof
 - b. Connect it to a Load centre:
 - Go to building level and open the [Generation](#) model data tab.
 - Check the [Include electric load centres](#) checkbox
 - Set the [Number of electric load centres](#) to 1.
 - Copy the default "DC with Inverter" library component.
 - Rename the copy to "PV with inverter".
 - Edit the new "PV with inverter" Electric load centre.
 - On the [Generator list](#) tab of the Electric load centre dialog, select the "Solar collector 1" PV panel, just added
 - Select the new Load centre.

The model should now look similar to the screenshot below.



We're now ready to export to the data to One Click LCA.

Export To One Click LCA

Before exporting the model to One Click LCA, you should consider checking the [Correlate materials in report by construction](#) setting on the Carbon tab of the Model options dialog. This allows you to track the source of each material instance in construction and glazing components and component blocks by creating a separate line item for each, instead of simply summing the quantity of each material.

Model Options - Building and Block

Model Options

Data Advanced Heating Design Cooling Design Simulation Display Drawing tools Block Project details **Carbon**

LCA

LCA time of study (years) 50

Glazing Carbon Model

Glazing CO2 embodied carbon emissions (kgCO2/kgGlass) 1.200

Glazing coating embodied CO2 (kg/m2) 0.700

Renewables

PV panels (kg/m2) 242.000

One Click LCA

☒ Correlate materials in report by construction

8. When the model has been set up as required, click on the One Click LCA toolbar icon (below) to start the submission process.



9. As DesignBuilder processes the model, if any errors are encountered, you are asked if you would like to review them.

When you are prompted to save the materials inventory in a report, enter the name of the report to be saved on your local machine and press the Save button.

10. Once the report has been created, you will be asked whether you would like to upload it to One Click LCA.

One Click LCA

Excel file generated. Do you want to upload to One Click LCA?

Yes No

11. Before you upload it, you should check the materials inventory in an external spreadsheet tool such as Excel, and if necessary, make any adjustments before uploading it.

If you applied the [Correlate materials in report by construction](#) Model option setting, the report will look similar to the screenshot below. Note the "Comments" column, which shows the source construction/glazing component for each material line item. Otherwise, the "Comments" column will be left blank.

#	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1	CLASS	MATERIAL	QUANTITY	QTY_TYPE	THICKNESS_MM	THICKNESS_IN	LENGTH_M	LENGTH_FT	AREA_M2	AREA_SQ_FT	VOLUME_M3	VOLUME_CU_FT	VOLUME_CU_YD	UNIT_PCS	MASS_KG	COMMENT
2	SLAB	Urea Formaldehyde Foam	600 M2		132.7	5.224409483	0	0	600	6458.34625	79.62	2812.165425	104.1390268	3		Project ground floor
3	SLAB	Cast Concrete	600 M2		100	3.9370079	0	0	600	6458.34625	60	2119.190222	78.477036	3		Project ground floor
4	SLAB	Floor/Roof Screed	600 M2		70	2.75590553	0	0	600	6458.34625	42	1483.433156	54.9339252	3		Project ground floor
5	SLAB	Timber Flooring	600 M2		30	1.18110237	0	0	600	6458.34625	18	635.7570667	23.5431108	3		Project ground floor
6	SLAB	Cast Concrete (Dense)	600 M2		100	3.9370079	0	0	600	6458.34625	60	2119.190222	78.477036	3		Project internal floor
7	WALL	Gypsum Plasterboard	420 M2		25	0.984251975	0	0	420	4520.842375	10.5	370.8582889	13.734813	8		Project partition
8	WALL	Air gap 10mm	210 M2		100	3.9370079	0	0	210	2260.421188	21	741.7165779	27.4669626	4		Project partition
9	WALL	Brickwork Outer	615.2801191 M2		102.5	4.054533098	0	0	615.2801191	6622.820083	63.06621221	2227.488338	82.48749009	17		Project wall
10	WALL	XPS Extruded Polystyrene - CO2 Blowing	615.2801191 M2		100	3.9370079	0	0	615.2801191	6622.820083	61.52801191	2173.159354	80.47560009	17		Project wall
11	WALL	Concrete Block (Medium)	615.2801191 M2		100	3.9370079	0	0	615.2801191	6622.820083	61.52801191	2173.159354	80.47560009	17		Project wall
12	WALL	Gypsum Plastering	615.2801191 M2		13	0.511811027	0	0	615.2801191	6622.820083	7.998641548	282.510716	10.46182801	17		Project wall
13	WINDOW	Generic LOW IRON 4MM	240.93765 M2		4	0.157480316	160.6251	526.9852362	240.93765	2593.431281	0.9637506	34.03951414	1.260538176	32		Project external glazing
14	WINDOW	Generic CLEAR 12MM	240.93765 M2		12	0.472440948	160.6251	526.9852362	240.93765	2593.431281	2.8912518	102.1185424	3.781614527	32		Project external glazing
15	WINDOW	Painted Oak	20.86251 M2		20	0.78740158	321.2502	1053.970472	20.86251	224.5621887	0.4172502	14.73720907	0.545742649	64		Painted Wooden window frame
16	SLAB	Plywood (Heavyweight)	600 M2		10	0.39370079	0	0	600	6458.34625	6	211.9190222	7.8477036	3		Project semi-exposed ceiling
17	SLAB	MW Glass Wool (rolls)	600 M2		131.9	5.19291342	0	0	600	6458.34625	79.14	2795.211903	103.5112105	3		Project semi-exposed ceiling
18	SLAB	Cast Concrete (Lightweight)	600 M2		100	3.9370079	0	0	600	6458.34625	60	2119.190222	78.477036	3		Project semi-exposed ceiling
19	SLAB	Air gap >=25mm	600 M2		200	7.8740158	0	0	600	6458.34625	120	4238.380445	156.954072	3		Project semi-exposed ceiling
20	SLAB	Plasterboard	600 M2		13	0.511811027	0	0	600	6458.34625	7.8	275.4947289	10.20201468	3		Project semi-exposed ceiling
21	ROOF	Clay Tile (roofing)	731.1332869 M2		25	0.984251975	0	0	731.1332869	7869.853203	18.27833217	645.5877137	23.90715553	5		Project unoccupied pitched roof
22	ROOF	Air gap 10mm	731.1332869 M2		20	0.78740158	0	0	731.1332869	7869.853203	14.62266574	516.470171	19.12572443	5		Project unoccupied pitched roof
23	ROOF	Roofing Felt	731.1332869 M2		5	0.196850395	0	0	731.1332869	7869.853203	3.65566434	129.1175427	4.781431106	5		Project unoccupied pitched roof
24	SLAB	External Rendering	33.18 M2		25	0.984251975	0	0	33.18	357.1465476	0.8295	29.29780483	1.084945023	1		Project external floor
25	SLAB	MW Stone Wool (rolls)	33.18 M2		148.2	5.834645708	0	0	33.18	357.1465476	4.917276	173.677387	6.431554095	1		Project external floor
26	SLAB	Timber Flooring	33.18 M2		5	0.196850395	0	0	33.18	357.1465476	0.1659	5.859560965	0.216898005	1		Project external floor

12. When you are happy with the report, close it and click Yes to upload it to the One Click LCA website.

13. From here, follow standard One Click LCA guidance for reviewing and modifying LCA reports.

DesignBuilder Embodied Carbon Calculations

Embodied Carbon Results

DesignBuilder provides a basic report of the embodied carbon and equivalent carbon data for the building on the **LCA and Embodied Carbon** tab of the **Cost and Carbon** screen. The carbon data covers only bulk construction, glazing and PV panel materials and excludes embodied carbon in lighting, HVAC and other building services equipment.

The calculation is made by summing the embodied carbon data for each surface, window and PV panel in the model as follows:

Opaque materials - embodied carbon is calculated using construction [material layers](#) and the [embodied carbon data](#) for each material.

Glazing - embodied carbon is calculated using the [Glazing CO2 carbon emissions](#) and [Glazing coating embodied CO2](#) settings on the Carbon tab of the Model options dialog and the area of windows x number of panes of each.

PV panel - embodied carbon is calculated by multiplying the area of PV panels by [PV panels](#) embodied carbon per area data on the Carbon tab of the Model options dialog.

The materials and constructions inventories are equivalent ways of showing the same data, but broken down in different ways. The total for the building is shown at the bottom.

Note: Embodied carbon calculations for models where the [zone multiplier](#) is used are not always 100% accurate because the embodied carbon for internal surfaces between multiplied zones and adiabatic component blocks can be over-estimated.

Note: The embodied carbon for glazing and PV panels can be defined on the [Carbon tab](#) of the Model options dialog.

An example Embodied carbon report is shown below.

Embodied and Equivalent Carbon for: Village Centre

The estimated embodied and equivalent carbon data shown below is based on bulk carbon data obtained from the Bath ICE and other data sources. The embodied carbon associated with building services such as lighting and HVAC equipment is not covered in these results.

Equivalent carbon is similar to embodied carbon but also includes the effects of other greenhouse gases so as to provide an equivalent amount of CO2 that would cause the same amount of global warming as the actual greenhouse gases (which may include sulphur dioxide, methane etc) emitted by the processes involved in production of the material.

You should make sure that all of the building materials and glazing systems used in the project have this data defined on the Embodied carbon tab of the materials and glazing dialogs.

The results are indicative only and it is the user's responsibility to check that input data is checked and the basis for the calculations is understood.

Materials Embodied Carbon and Inventory	Area (m2)	Embodied Carbon (kgCO2)	Equivalent CO2 (kgCO2)	Mass (kg)
JV3 Ref Metals - Iron (SA.0.7)	14.275	430.791	469.134	225.545
JV3 Ref Weatherboard	56.706	169.550	173.236	368.586
JV3 Ref Concrete cast - dense reinforced	1398.174	299069.300	318364.200	964739.762
JV3 Ref Sandstone Cladding	858.113	0.000	0.000	17604.583
Concrete cast - dense reinforced	843.369	90198.300	96017.550	290962.283
Timber Flooring	1852.937	16620.840	16982.160	36132.266
Plasterboard	597.265	6354.896	6689.365	16723.411
Gypsum Plastering	263.838	1002.584	1055.352	2638.380
Gypsum Plasterboard	4711.072	11224.570	12159.950	93538.049
MW Stone Wool (rolls)	263.838	182.840	195.029	174.133
MW Glass Wool (rolls)	739.369	844.447	927.236	551.926
Cast Concrete	1852.937	88940.970	88940.970	1111762.098
Cast Concrete (Dense)	2475.515	41588.660	41588.660	519858.200
Brickwork Inner Leaf	14.744	606.572	634.144	2757.147
Brickwork Outer Leaf	263.838	9867.541	10316.070	44852.461
Fibreboard	127.360	201.152	209.041	394.416
Sub Total		567303.100	594722.000	3103283.247
Constructions Embodied Carbon and Inventory	Area (m2)	Embodied Carbon (kgCO2)	Equivalent CO2 (kgCO2)	
JV3 Lightweight Reference (Total R2.8)	8.224	40.891	43.660	
JV3 Ref Lightweight roof w Ceil R3.2	14.275	612.555	661.821	
JV3 Reference Timber Clad (Total R2.8)	56.706	408.128	429.200	
JV3 Stone Clad Brick Reference (Total R2.3)	14.744	639.550	670.121	

JV3 Proposed Concrete Floor w Ceil R0.3	815.184	174367.800	185617.400
JV3 Ref timber ext concrete floor (R2)	62.431	3703.776	3724.753
Project internal floor	981.113	16482.700	16482.700
Project partition	1894.015	10227.680	11079.980
Project internal floor_Reversed	1494.402	25105.950	25105.950
JV3 ref block plasterboard wall (Total R1.0)	263.838	11052.970	11566.450
JV3 Stone Clad Concrete (Total R0.53)	843.369	91109.160	97004.300
JV3 Proposed Flat Roof Concrete w Ceil R2.0	582.990	131546.700	139981.400
JV3 Ref timber gnd concrete floor (R0.69)	1790.506	102005.100	102354.300
Sub Total	8821.796	1764359.134	

Glazing Embodied Carbon and Inventory	Area (m2)	Embodied Carbon (kgCO2)	Equivalent CO2 (kgCO2)
RS4 custom roof glazing	174.403	3706.060	3924.063
JV3 Simple external glazing_14	3.471	44.258	46.861
JV3 Simple external glazing_13	8.080	103.024	109.084
JV3 Simple external glazing_12	3.453	44.019	46.609
JV3 Simple external glazing_11	17.640	224.906	238.136
JV3 Simple external glazing_10	69.443	885.403	937.486
JV3 Simple external glazing_9	8.332	106.238	112.487
JV3 Simple external glazing_8	48.766	621.754	658.328
JV3 Simple external glazing_7	55.226	704.135	745.555
JV3 Simple external glazing_6	137.489	1752.985	1856.102
JV3 Simple external glazing_5	167.807	2139.534	2265.389
JV3 Simple external glazing_4	75.389	961.216	1017.758
JV3 Simple external glazing_3	10.288	131.173	138.889
JV3 Simple external glazing_2	58.849	750.326	794.463
JV3 Simple external glazing_1	108.658	1385.387	1466.880
Project external glazing	14.695	187.361	198.383
Sub Total	961.989	13747.780	14556.473
Building Total Embodied CO2	9783.784	581050.807	609278.500

Reporting

You can use DesignBuilder to generate reports using the [Compile Reports](#) and [Report topic](#) mechanism or you can [Export](#) individual report content to file, clipboard (for pasting into word processors) or directly to the printer.

[Export Data](#)
[Report Topics](#)
[Compile Reports](#)
[Export CSV](#)

Export Data



You can export data from DesignBuilder **screens**:

Edit Screen - exports a non-rendered 3-D image of the building. Alternatively a 3-D DXF file can be exported.
Visualisation Screen - exports a rendered 3-D image of the building optionally including shading and antialiasing..
Heating and Cooling Design Screens - exports design data (graphical, tabular, spreadsheet).
Simulation Screen - exports simulation data (graphical, tabular, spreadsheet).

You can export to a range of **destinations** (depending on the nature of the data):

File - store output in a file which can be imported into your documents/e-mails.
Printer - sends output directly to the printer.
Clipboard - copy the data to the clipboard for pasting into another application such as Word, Excel or email.
Report topic - store the data as a [report topic](#) which can later be included as a page in a report by using the [Compile report](#) option.
Insights - export parametric results to the Insights tool on the DesignBuilder Analytics website.

Tip: the default setting for the destination can be set on the [Dialogs](#) tab of the Program options dialog.

Export **format** options depend on the nature of the data. The full range of options is:

Bitmap - best bitmap based output quality but very large file sizes.
PNG - better quality compression than JPEG but sometimes have larger file sizes.
TIFF - best bitmap based output quality but very large file sizes.
GIF - good compression but black and white only.
JPEG - recommended for 3-D images of the model because of its good quality combined with small file sizes.
DXF - generate 3-D DXF data of the model - only available from Edit and Visualisation Screens. The 3-D DXF file generated by DesignBuilder is suitable for reading into other CAD software. Two options are available:
 1-Whole model with details which includes all of the details of the model as seen on the Visualisation tab, and,
 2-Zones which generates the internal surfaces as used in simulations and other energy calculations.
Windows metafile - results graphs only - scalable so graphs can be resized without losing quality.
CSV spreadsheet - grid results and library data only. Note that grid results saved in [csv format](#) can be loaded to Results Viewer for further analysis.
Text - reports only.

JSON - format used when exporting to Insights for further analysis.

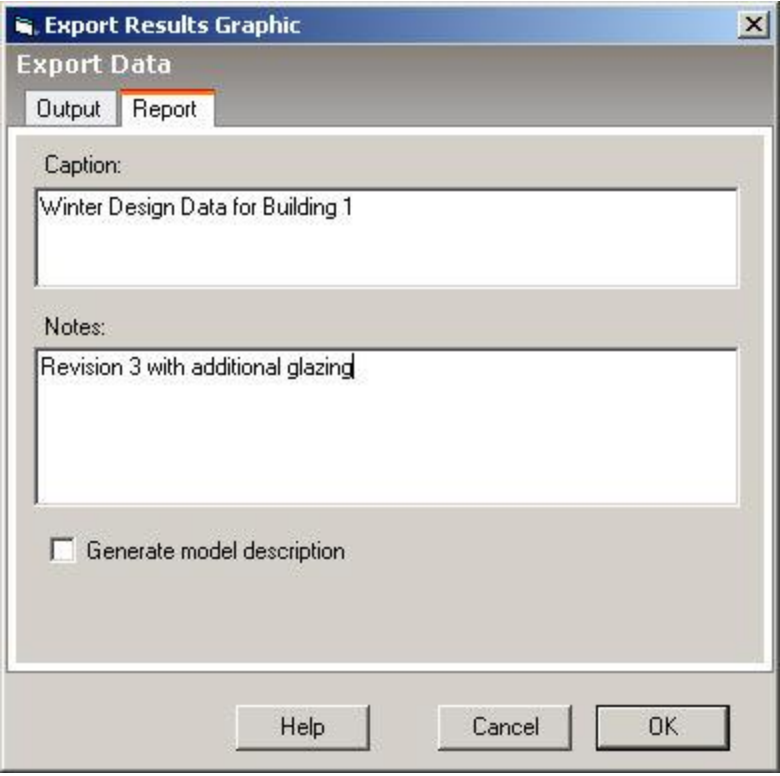
See also the [Exporting Data and Compiling Results Tutorial](#)

Tip: To improve the image resolution we recommend using a high resolution monitor.

Report Topics

Report topics are used to store snapshots of model data including model descriptions, graphic images of the model and results.

You can export to a report topic by selecting 'Report topic' from the 'Export to' option on the [Export Data](#) dialog. When Report topic is selected, a second tab becomes available called Report on which you can enter a title and caption for the topic. You can also choose to include a model description with the topic. This allows a building description to be included alongside the data when the report is compiled using the [Compile Report](#) dialog.



The screenshot shows a dialog box titled "Export Results Graphic" with a close button (X) in the top right corner. Inside the dialog, there are two tabs: "Output" and "Report". The "Report" tab is currently selected. Under the "Caption:" label, there is a text input field containing the text "Winter Design Data for Building 1". Below this, under the "Notes:" label, there is a larger text input field containing the text "Revision 3 with additional glazing". At the bottom of the dialog, there is a checkbox labeled "Generate model description" which is currently unchecked. At the very bottom of the dialog, there are three buttons: "Help", "Cancel", and "OK".

Report topics are retained within the dsb file and are available for compiling into a report if the file is saved and reloaded.

Compile Reports



The Compile Reports dialog allows you to control the generation of a report based on previously created [Report topics](#). Available report topics are listed and can be previewed by clicking on them.

The Title entered on this dialog is displayed in a table at the top of the report.

Check the checkbox to the left of the report topics to be included in the report and click on OK.

See also the [Exporting Data and Compiling Results Tutorial](#)

Insights

The DesignBuilder [Insights](#) platform enables fully interactive, in-depth analysis of parametric simulation results through [Parallel Coordinate diagrams](#), [Bubble Charts](#), and [Line Charts](#). These tools allow you to explore multi-dimensional datasets, identify trends, and understand relationships between variables more effectively. Insights generates high-quality graphical outputs suitable for reporting, and results can be easily shared across teams, supporting a more collaborative and transparent analysis process.



The following types of DesignBuilder parametric simulation results can be exported to Insights:

[Parametric analysis](#)

[Optimisation](#)

[Sensitivity analysis](#)

In each of these cases the process of exporting parametric data from DesignBuilder results screens is similar.

Detailed information on how to use Insights is provided in the topics linked below.

"Exporting Parametric Results to Insights" on page 2073

"Insights Preparation" on page 2071

"Insights Data Management" on page 2076

"Insights Data Analysis" on page 2079

"Insights Tutorials" on page 2090

Insights Preparation

The Insights website can be accessed through any web browser by pointing it to the DesignBuilder Analytics portal at:

analytics.designbuilder.co.uk

Registering And Signing In

You do not need to purchase a separate license to access the Insights website. However, you do need to register with an email address and password before being able to sign in.

Registration

If you have not done so already, the first step is to create an account on the Analytics platform by registering your user name, email address, password and your activation code. From the home page, analytics.designbuilder.co.uk, click on the large green button to register:

Email	Email
Password	Password
SIGN IN	
REGISTER	
Forgot Password?	

This takes you to the Registration page where you can enter your user name, email address and password. If you already have a [Climate Analytics licence](#), you can check the checkbox and enter it in the text box below:

User Name	Jim Smith
Email	jim@gmail.com
Password	*****
☐ I have a Climate Analytics activation code.	
☐ Agree to Terms of Use	
REGISTER USER	
CANCEL	

Signing In

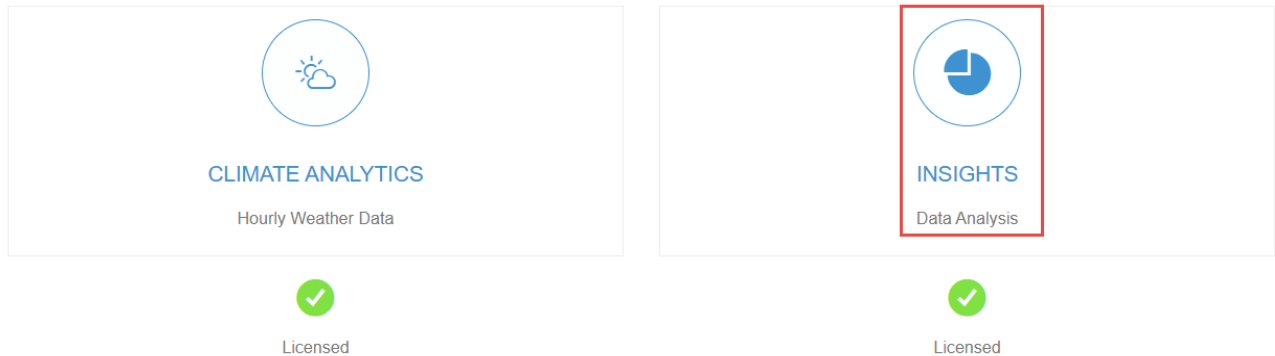
You must sign in before being able to access the Insights web page. Enter the same email address and password that you used when [registering](#).

Email	jim@gmail.com
Password	*****
SIGN IN	
REGISTER	
Forgot Password?	

If you have forgotten your password click on the **Forgot Password** link to receive a reset email with a link and instructions.

Accessing Insights

After successfully signing in you will see the Analytics opening page click. Now click on the "Insights" button to access the Insights web app.



Exporting Parametric Results To Insights

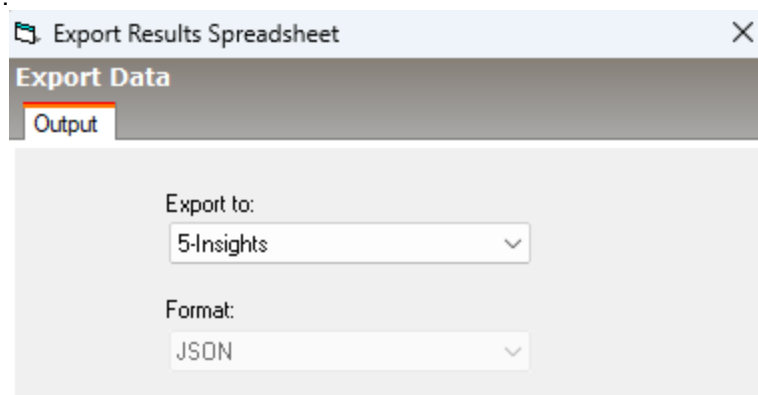
To export a set of existing parametric results to Insights, follow the steps below.

1. Open your model, go to the Simulation screen. To export optimisation or UA/SA results go to the **Optimisation + UA/SA** tab. Or, to export parametric analysis results go to the **Parametric** tab.

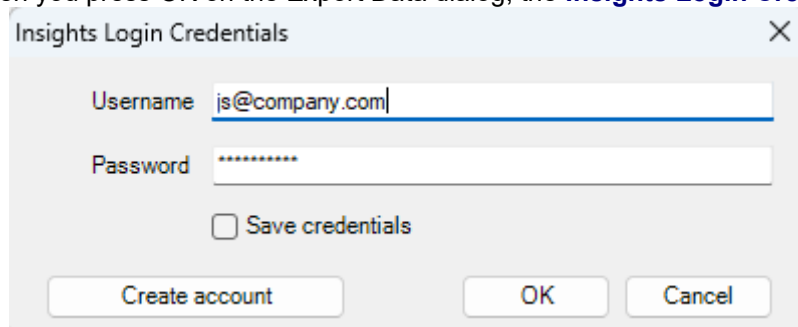
2. Check the graphical results are displayed.

3. Click on the **Export data** toolbar icon .

4. On the Export Data dialog, select the **5-Insights** "Export to" option. The Format will be automatically set to **JSON**:



5. When you press OK on the Export Data dialog, the **Insights Login Credentials** dialog is displayed:



6. If you have already registered on the [Analytics portal](#), enter your email address and password.

Check the **Save credentials** checkbox to avoid having to re-enter these settings for every export. Or, if you share your machine with others, you may prefer to leave it unchecked.

7. If you don't already have an account on the Analytics portal, click on the **Create account** button. This will take you to the main login page of the Analytics portal where you can click on the **Register** button. See below.

Email	Email
Password	Password
SIGN IN	
REGISTER	
Forgot Password?	

Once you have registered you can enter your email address and password on the **Insights Login Credentials** dialog.

For more details on registration see "Insights Preparation" on page 2071

8. Having entered your account credentials, press the OK button to close the credentials dialog and open the **Insights Upload Options** dialog. Here you can enter details on the analysis and add your parametric results to an existing Insights project or to a new project that you create from this dialog. Details on how to use this dialog are provided in the next section.

Insights Upload Options Dialog

Insights Upload Options

Projects

☒ New project

Name: 19-23 King Street

Owner: JS

Reference:

☐ Existing project

Properties

Name: Optimisation: CO2 vs Discomfort

Reference: Option 1

Chart defaults

X Axis: Discomfort (All Clothing) (hr)

Y Axis: CO2 (lb)

Size: Fixed size

Series:

☐ Open webpage after upload

OK Cancel

Projects

Insights allows you to store your parametric results within projects that you define. To create a new project, select the **New Project** radio button and enter the details of the new project, including the Project **Name**, the **Owner** of the project and a **Reference**. You can use these fields to provide any additional information associated with the data.

Alternatively, you can store your results with an existing project, defined when uploading a previous parametric data set. In this case, check the **Existing project** radio button and enter the **Owner** and **Reference** fields as described above.

Properties

Under the Properties header, you can enter a **Name** for this data set. This will be used as the main identifier for the data and as a header for graphs and diagrams when they are displayed. You can also enter **Reference** text.

Note: All **Reference** and **Owner** fields on this dialog are for your reference only and can be left blank if this data is not required.

Tip: All **Name**, **Reference** and **Owner** data you enter here can be edited later on the Insights website, if necessary.

Chart defaults

Under the **Chart defaults** header, you can select the default **X-Axis** and the default **Y-Axis** to be used when plotting results on bubble and line plots.

You can also select how the **Size** of bubbles are defined by default for [bubble charts](#) if displayed in Insights. If you wish X-Y data points to have a fixed size then keep the default **Fixed size** setting.

When exporting parametric analysis data, you can also select a variable as the default **Series**. This will be used to create one or more lines on the parametric line plot each with a constant variable value. The range of series variable values will be displayed in the legend of the line plot. For example, if you select the series to be "Glazing type", the parametric analysis results will be displayed with a set of lines of different colours, one line for each glazing type option. The line colours are mapped to glazing type options in the legend.

These settings can all be changed later within Insights.

Check the **Open webpage after upload** checkbox to gain quick access to the Insights website directly from DesignBuilder.

Press OK to upload the data to Insights.

Insights Data Management

Having logged in to the Insights website, you will see a list of projects that have previously been uploaded. If this is your first time using Insights, you may see a blank page, in which case you must [upload results](#) from your parametric simulations before being able to analyse them here.

List Of Projects

If you have previously uploaded analyses from DesignBuilder, you will see a list of projects here.

My Projects

Name	Owner	Reference	Created		
					
London Bridge 2025	JS		2025-06-02 - 17:09		
My new project	JS	Iteration 1	2025-06-02 - 15:50		
v8.0.0.077	AT		2025-05-30 - 16:04		
Opt Ex 2	AT		2025-05-30 - 12:28		

Each project includes the following meta data:

Name: The name of the project as entered when creating a new project from the [Insights Upload Options Dialog](#) when exporting the data from DesignBuilder. The name should provide sufficient information to differentiate it from any other projects.

Owner: The name or initials of the analyst who created the project. This data may be left blank, if necessary.

Reference: Any additional data provided for the project. This data may be left blank, if necessary.

Created: The date and time when the project was created.

From the project screen, you can:

View analyses: Click on a project row to see the list of analyses associated with it.

Edit properties: Edit the Name, Owner or Reference properties of any of the projects by clicking on the associated **Edit properties** button  towards the end of the project row.

Edit Project Details

Name:

My new project

Owner:

JS

Reference:

Iteration 1

Save

Delete project: Delete the project by clicking on the associated Delete  icon at the end of the row. Note that deleting the project will also delete all associated analyses.

Filter projects: To filter the list of projects by Project **Name**, **Owner**, or **Reference**, type the filter text in the filter box at the top of the relevant column and press <Enter>. For example, to see only the projects with "London" in their name, type "London" in the name filter field as shown below. Note that filtering is case-insensitive.

My Projects

Name	Owner	Reference	Created		
London					
London Bridge 2025	JS		2025-06-02 - 17:09		

Filter field, with "London" entered

List of projects that include "London" in their name.

Tip: You can easily return to the Projects list at any time by clicking on the **Projects** menu item at the top of the page.

List Of Analyses

Insights includes two types of analysis list:

List of analyses in a **selected project**, accessed by clicking on a project from the **Projects** list, as described above.

List of analyses in **all projects**, accessed by clicking on the **Analysis** menu item.

Both types of analysis list behave the same, so they are both covered here in this section.

Analysis Type	Analysis Name	Owner	Reference	Created		
	Measures 1, 2, 3 included	JS	Iteration 2	2025-06-02 - 17:15		
	All measures included	JS	Iteration 1	2025-06-02 - 17:09		

Each analysis includes the following meta data:

Analysis Type: The type of analysis is indicated by an icon on the left of each row. It can be one of:



Optimisation



Sensitivity Analysis



Parametric Analysis

Name: The name of the analysis as entered when exporting using the [Insights Upload Options Dialog](#) in DesignBuilder. The name should provide sufficient information to differentiate it from any other analyses associated with this project.

Owner: The name or initials of the analyst who exported the data. This data may be left blank, if necessary.

Reference: Any additional data provided for the analysis. This data may be left blank, if necessary.

Created: The date and time when the analysis was created.

From the **Analyses** listing page, you can:

View an analyse: Click on an analysis row to [view the data](#).

Edit properties: Edit the Name, Owner or Reference properties of any of the analyses by clicking on

the associated **Edit properties** button  towards the end of the analysis row.

Edit Analysis Properties 

Name:

Owner:

Reference:

Save

Delete analysis: Delete an analysis by clicking on the associated Delete  icon at the end of the row.

Filter analyses: To filter the list of analyses by Analysis **Name**, **Owner**, or **Reference**, type the filter text in the filter box at the top of the relevant column and press <Enter>. For example, to see only the analyses with "3" in their name, type "3" in the name filter field as shown below. Note that filtering is

case-insensitive.

Project Name: London Bridge 2025

Analysis Type	Analysis Name	Owner	Reference	Created		
	3					
	Measures 1, 2, 3 included	JS	Iteration 2	2025-06-02 - 17:15		

Tip: You can easily return to the Analyses list at any time by clicking on the **Analyses** menu item at the top of the page.

Insights Data Analysis

The usual way to view Insights data is from a Project's Analyses list. Having [signed in](#), click on a [Project](#) to list its [Analyses](#) and then click on the analysis row of interest. The parametric data for that analysis will be displayed.

Depending on whether the data selected is for an Optimisation, a Sensitivity Analysis, or a Parametric Analysis, you will be able to access different results:

Optimisation and **UA/SA** analyses provide [Bubble Charts](#) and [Parallel Coordinate Plots](#).
Parametric Analysis analyses provide [Line Charts](#) and [Parallel Coordinate Plots](#).

Click on one of the tabs at the top of the screen to select the required output. For example, for Optimisation analyses, the header and tabs are configured as shown below.

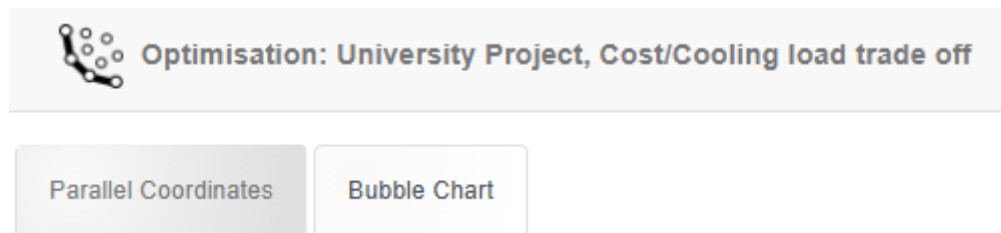


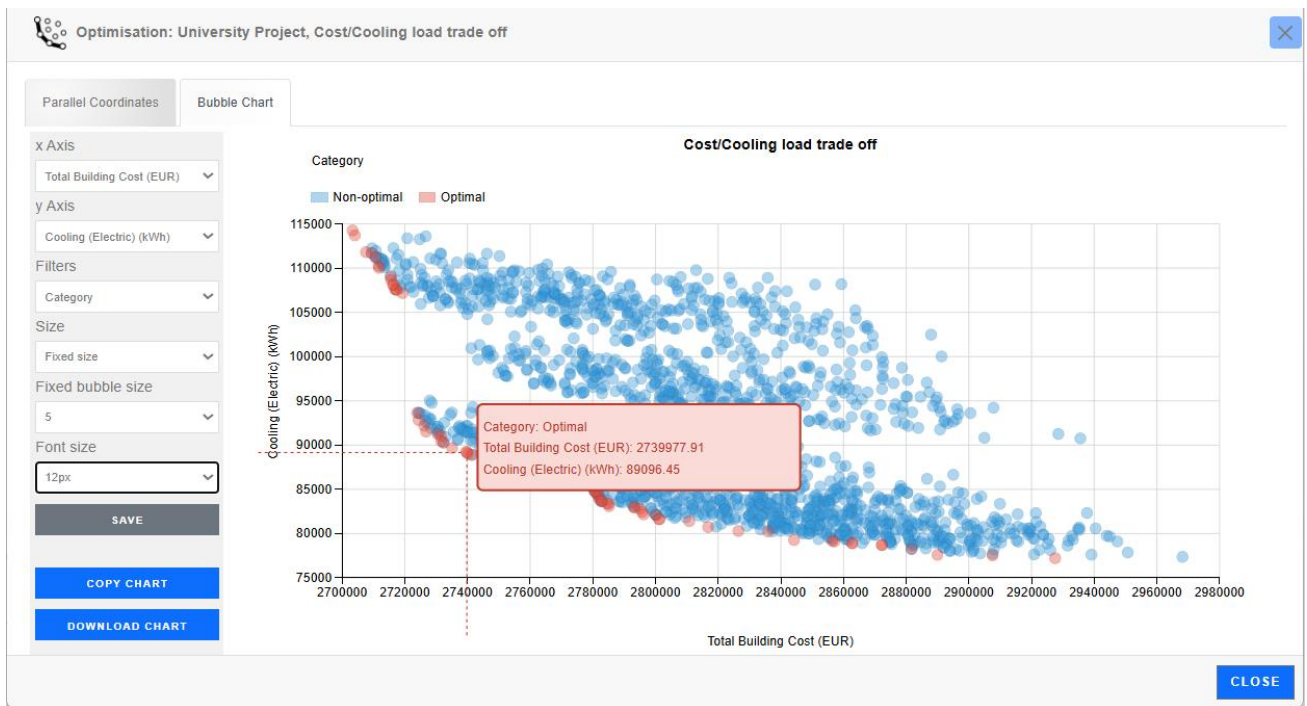
Chart Types

For more information on how to use the available Insights chart types, see the topics below:

[Bubble Charts](#)
[Parallel Coordinate Plots](#)
[Line Charts](#)

Insights Bubble Charts

When first opening an Optimisation analysis, you will see a bubble chart similar to the one below. By default, the settings should be similar to those used in DesignBuilder, so unless changes were made to defaults on the DesignBuilder [Insights Upload Options Dialog](#), the default Insights bubble chart should look similar to that displayed in DesignBuilder, with the same X and Y-axes, Filter and a fixed bubble size.



This shows the Pareto front of optimal solutions without any additional slicing and dicing of the data.

Moving the mouse over a data point provides a tooltip showing the X and Y-axis values and the Category which, in the above case, defines whether the point is Optimal or Non-optimal. The tooltip is displayed in the same colour as that used to define the category.

Panel settings

The panel on the left-hand side of the screen allows changes to be made to a range of settings:

X-axis

Select an [Objective](#), [Constraint](#), [Additional output](#), or a numeric [Design variable](#) from the drop list to be used as the X-axis variable. List type variables are not available for selection. For optimisation bubble charts, an objective should normally be selected here.

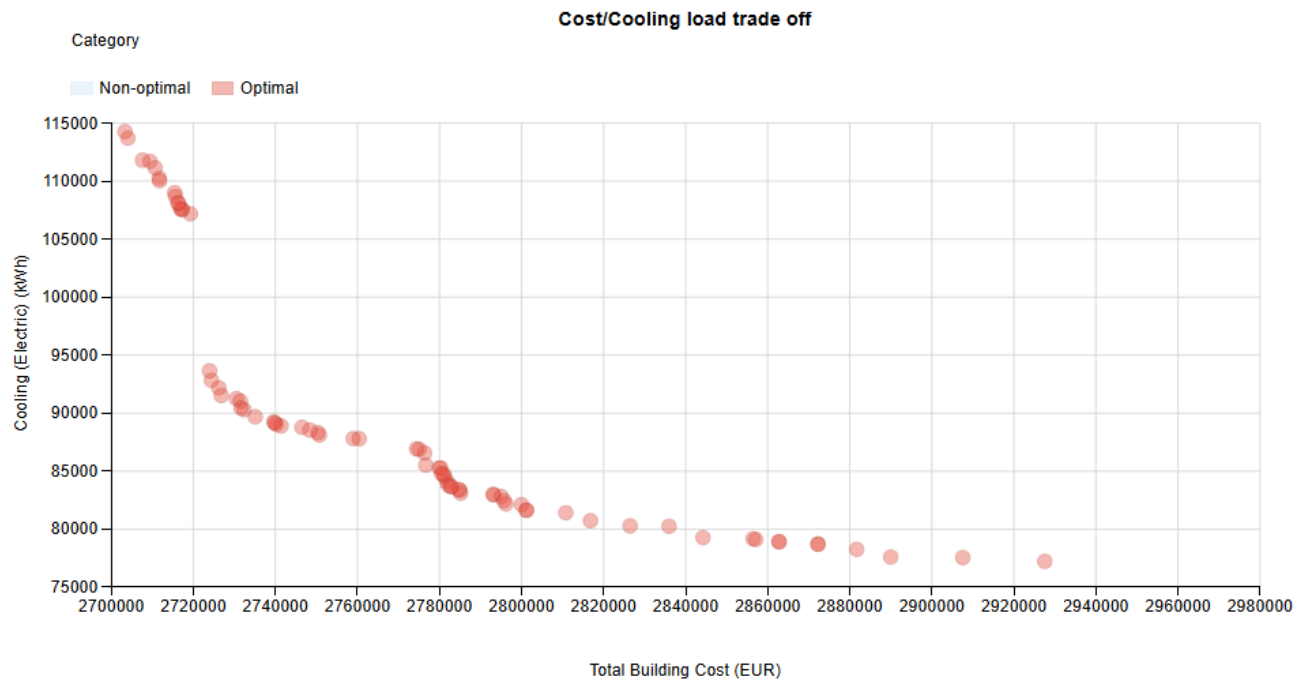
Y-axis

Select an [Objective](#), [Constraint](#), [Additional output](#), or a numeric [Design variable](#) from the drop list to be used as the Y-axis variable. List type variables are not available for selection. For optimisation bubble charts, an objective should normally be selected here.

Filters

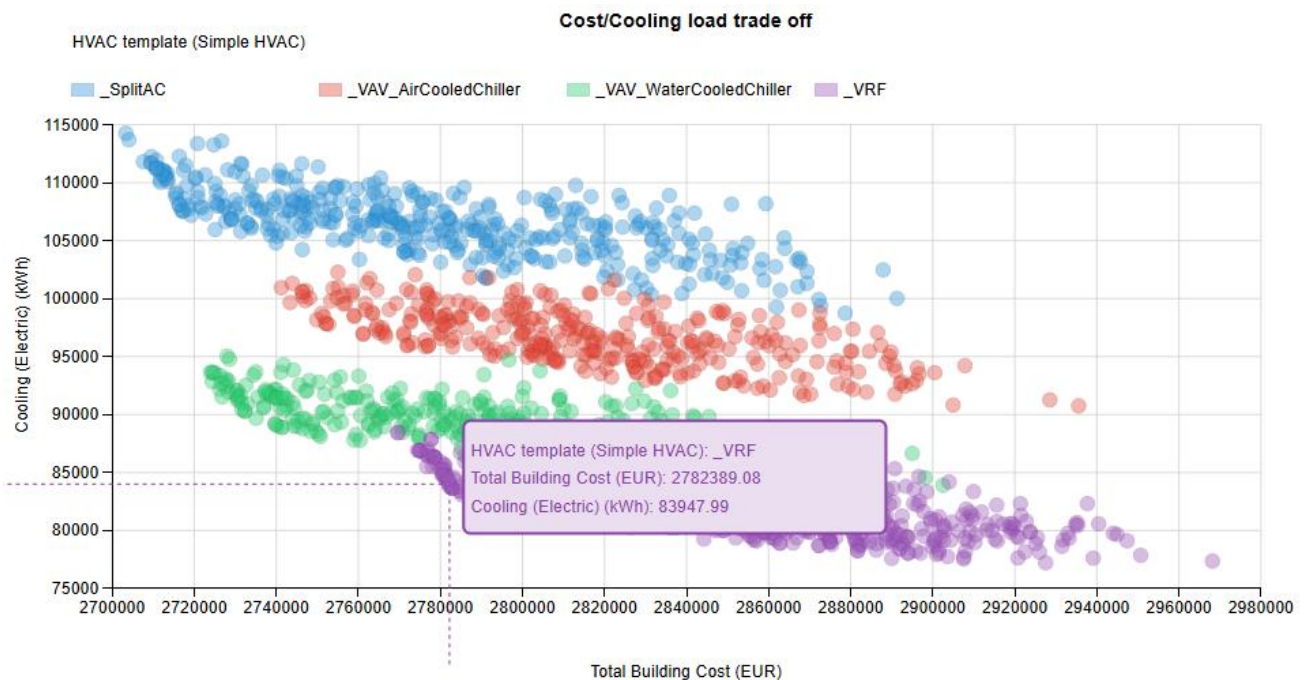
Select "Category" or a non-numeric List type variable from the list. The **Filter** setting is used to create a legend and to colour the bubbles. For example, the default "Category" filter colours each point either red or blue depending on whether it is Optimal or Non-optimal.

By clicking on the coloured box in the legend, it's possible to hide data points of one or categories. For example, to hide all non-optimal points, click on the blue "Non-optimal" legend box. The blue box is faded and the corresponding non-optimal points on the bubble plot are hidden. See below:



To display the non-optimal values again, click again on the "Non-optimal" points box in the legend.

Alternatively, select a list type variable from the filters list to colour each data point based on the setting for the selected variable. In the example below, the "HVAC template" was selected as the Filter. The result is that each data point is coloured based on its HVAC template.



Size

In addition to the **Fixed** Size option, you can also use the **Size** drop list to select an [Objective](#), [Constraint](#), [Additional output](#) or numeric [Design variable](#) to define the size of each bubble. In the example below, the roof solar absorptivity is used to define the bubble size.



Fixed bubble size

When using the **Fixed size** Size option, you can define the size of all bubbles by selecting a value from the **Fixed bubble size** drop list.

Min bubble size

When **Size** set to anything other than **Fixed size**, you can define the minimum size for bubbles by selecting a value from the **Min bubble size** drop list. This minimum size corresponds to the minimum value for the selected objective, constraint, extra output or numeric variable.

Max bubble size

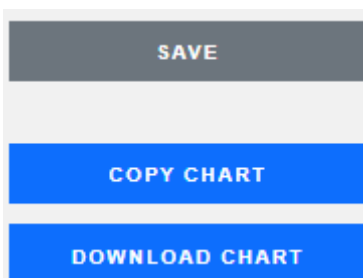
When **Size** set to anything other than **Fixed size**, you can define the maximum size for bubbles by selecting a value from the **Max bubble size** drop list. This maximum size corresponds to the maximum value for the selected objective, constraint, extra output or numeric variable.

Font size

Select the font size to be used for all legends, axis values, x and y-axis labels, and categories.

Panel command buttons

At the bottom of the left-hand panel, you can also find buttons to save settings and copy/download the chart.



These are described in more detail below.

Save

Pressing the **Save** button, the current set of options described above are stored, so when viewing the chart next time, it will look exactly as it does now.

Saving these settings only applies to the current chart and does not effect on any other analyses or charts.

Copy chart

Pressing the **Copy chart** button copies the currently displayed bubble chart onto the clipboard ready for pasting into an external document.

Download chart

Pressing the **Download chart** button causes the currently displayed bubble chart to be downloaded as a png file into the Downloads folder on your local machine. The file is named using the analysis Name.

Insights Parallel Coordinates Plots

Parallel coordinate plots (PCPs) are an interactive visualisation technique used to represent high-dimensional data in a two-dimensional space especially when checking for patterns and relationships. Each dimension of the data is represented by a parallel axis, and data points are plotted as lines connecting points on each axis. This allows for the visualisation and comparison of many outputs and variables simultaneously.

PCPs can be especially useful for:

Sensitivity Analysis and identifying key drivers: Understanding which input parameters have the most significant impact on a specific output. For example, when looking for "fan-out" patterns where a range of values on one input axis leads to a very wide range of values on an output axis (like EUI), that input is a highly sensitive parameter. Likewise "fan-in" patterns when a wide range of values on an input axis converges to a narrow range on an output axis, that parameter has less impact. This could be useful as part of an initial exploratory phase of design optimisation to help refine the range for variable values and which variables and outputs to include in any future analyses.

Identifying high-performance design solutions (optimisation): PCPs are excellent for sifting through very large design spaces to find combinations of parameters that meet multiple performance goals simultaneously. For example, when aiming for a near-net-zero building, the design team need to find design options that have a very low [EUI](#), minimal discomfort/Unmet Load Hours, and a reasonable construction cost. The PCP allows users to select a desired range on one or more axes by [brushing](#) the objective, constraint or output axes, typically selecting the top-performing range. For example, highlight the bottom 10% of EUI values, the bottom 20% of construction cost, and the bottom 5% of Discomfort or Unmet Load Hours. The resultant plot will show which input parameter ranges correspond to these high-performance outcomes.

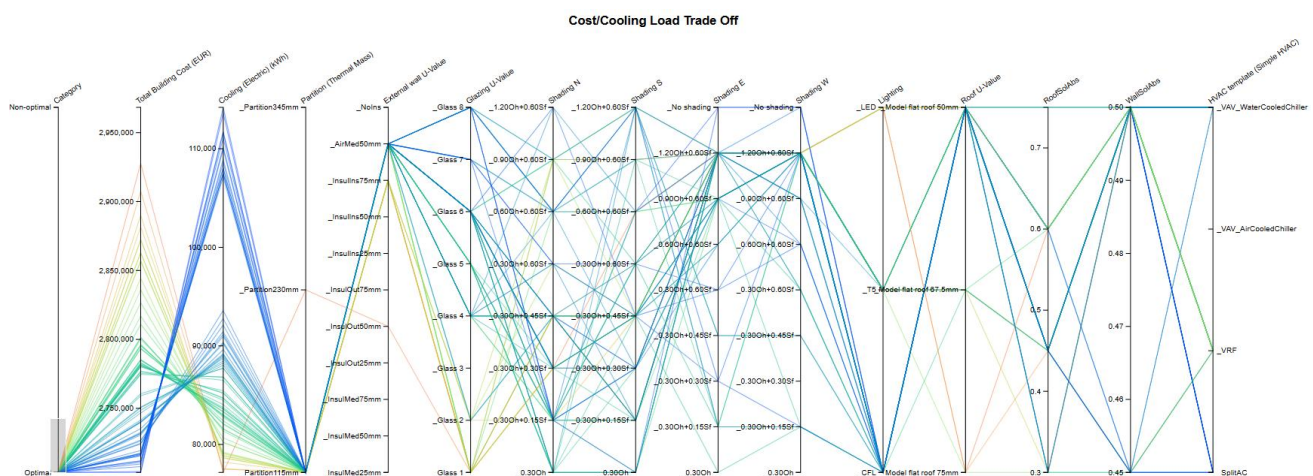
Understanding and visualising trade-offs: Building design inevitably involves making trade-offs. Improving one metric often worsens another. PCPs make these conflicting relationships obvious. See [below](#).

Their ability to visualise high-dimensional data, reveal complex trade-offs, and enable interactive analysis makes them a powerful complement to other the other Insights display methods.



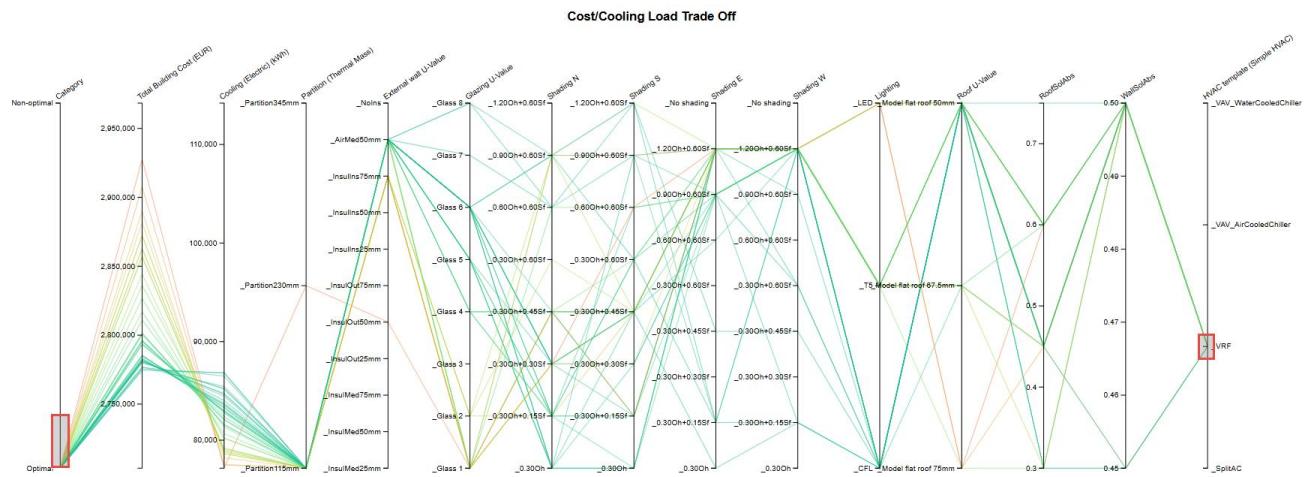
All data

As can be seen from the above diagram without any filtering, parallel coordinates plots can contain a lot of data and the clutter may not help with analysis. However, even in this unfiltered state they can provide visual clues to help [identify correlations](#).



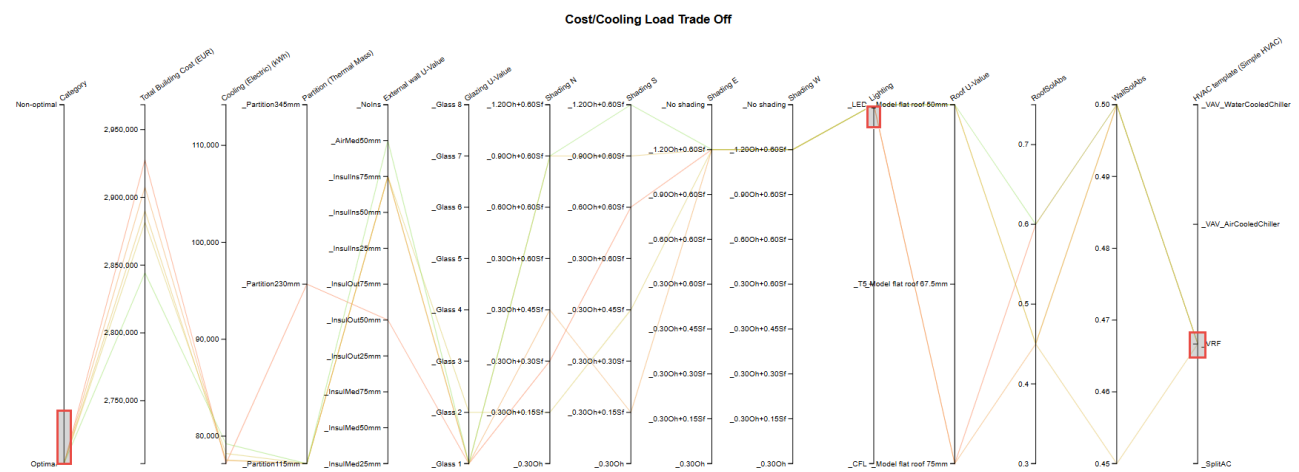
Optimal data only

The real value of these outputs becomes apparent when you start to explore the data by excluding potentially less interesting design options. For example, in the above output, it might help to limit the outputs to only the optimal ones. To do this, use the [brush tool](#) to select the "Optimal" Category only by clicking with the mouse on the "Category" axis just above the "Optimal" option, and dragging to include the "Optimal" option and releasing the mouse button. Limiting outputs to only the optimal ones reduces the data significantly, as can be seen in the above diagram.



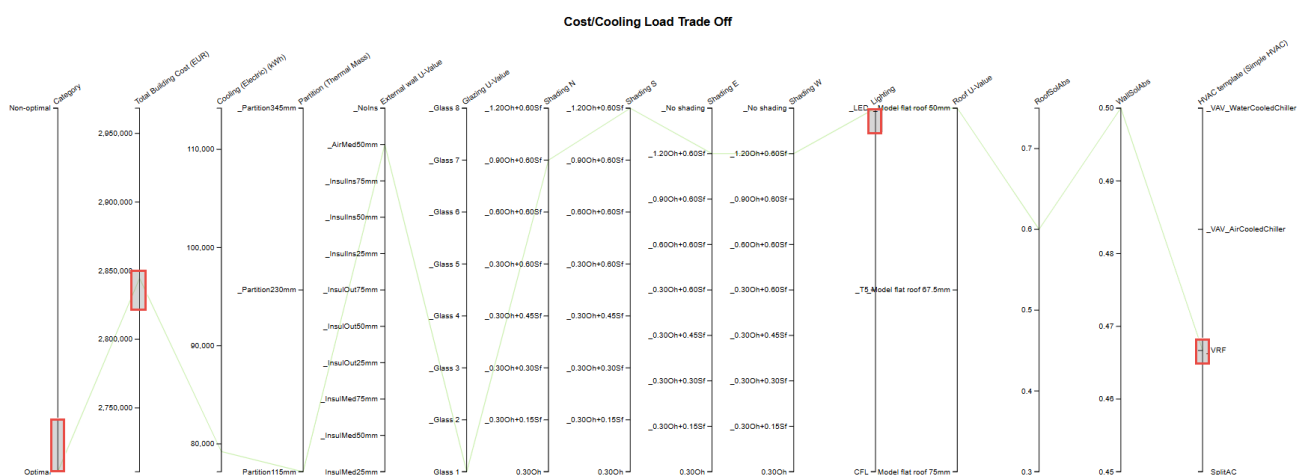
Optimal data with VRF HVAC system

Now limiting the outputs to include only design options with a VRF HVAC system further narrows down the results. See above.



Optimal data with VRF HVAC system and LED lighting

Further limiting the outputs to exclude design options without LED lighting results in the above plot.



Finally, selecting the option with the lowest "Total Building Cost" from the remaining options results in the above plot with a single output.

This process is further illustrated in a video below.

Brush tool

Parallel coordinates plots with a lot of results can be cluttered and difficult to read. The best way to remedy this is by selecting subsets of the lines of greatest interest while hiding the others thus isolating the data points you're most interested in. This is done by interactively "brushing" areas of interest on the plot using the mouse.

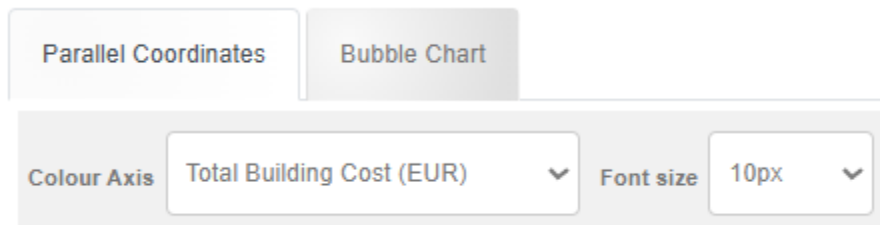
To do this, use the mouse to click and drag over one of the axes to define the intersecting design options of interest. You can do the same for any of the axes.

For brushes that have already been created, you can move these up and down by dragging with the mouse, or you can extend or reduce their range by using the mouse to drag the brush edges.

Tip: To clear brushing on any PCP Y-axis so that the full range of values are considered, double-click on the axis away from the existing brush. Alternatively, you can extend the brush size to cover the whole axis.

The process of creating and manipulating brushes is illustrated in the example video below which shows the steps described in the example above and additionally demonstrates how brushes can be resized and moved. Click on the maximise button for best clarity.

Panel settings



The horizontal panel at the top of the screen allows changes to be made to the following settings:

Color Axis

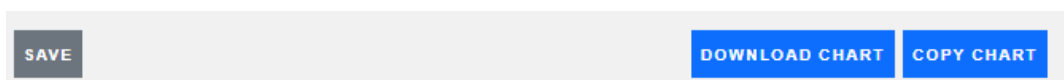
The **Colour Axis** control allows you to select which of the y-axes is used to colour the lines. In the [above example](#), "Total Building Cost" is selected as the axis to be used to colour the lines representing the design options. This results in lines with the highest building cost being displayed in red, lines with lowest building cost being displayed in blue, and intermediate options being displayed in colours between red and blue on the spectrum.

Colouring the lines in this way can help identify any trends or correlations in the data. In the [above example](#), most of the blue lines are positioned towards the top of the "Cooling (electric) (kWh)" axis, while most of the red and orange lines are towards the bottom. This indicates that the most expensive building options tend to give the lowest cooling and vice versa. In other words, there is a negative correlation between building cost and cooling electric energy consumption.

Font size

Select the font size to be used for all y-axis values and labels.

Panel command buttons



On the top panel, you can also find buttons to save settings and copy/download the chart.

These are described in more detail below.

Save

Pressing the **Save** button retains the current set of options described above, so that when viewing the same chart next time, it will look exactly as it does now. Saving these settings does not effect on any other analyses or charts.

Copy chart

Pressing the **Copy chart** button copies the Parallel coordinates diagram onto the clipboard, ready for pasting into any document.

Download chart

Pressing the **Download chart** button causes the currently displayed Parallel coordinates diagram to be downloaded as a png file into the Downloads folder on your local machine. The file is named using the analysis Name.

Insights Line Charts

When first opening a Parametric Analysis dataset, you will see a line chart similar to the one below. By default, the settings should be similar to those used in DesignBuilder, so unless changes were made to defaults on the DesignBuilder [Insights Upload Options Dialog](#), the default Insights bubble chart should look similar to that displayed in DesignBuilder, with the same X and Y-axes and Series.



Moving the mouse over a data point provides a tooltip showing the X and Y-axis values and the Series value which, in the above case, defines the Glazing type. The tooltip is displayed in the same colour as that used to define the Series.

Panel settings

The panel on the left-hand side of the screen allows changes to be made to the following settings:

X-Axis

Select an [Output](#), or a [Design variable](#) from the drop list to be used as the X-axis variable. Often, a Design variable will be used for the X-axis.

Y-Axis

Select an [Output](#), or a [Design variable](#) from the drop list to be used as the Y-axis variable. Often an Output will be used for the Y-axis.

Series

The Series drop list allows you to select a non-numeric variable to define. This will be used to create one or more lines on the line plot each with a constant variable value. The range of series variable values will be displayed in the legend of the line plot. For example, if you select the series to be "Glazing type", the parametric analysis results will be displayed with a set of lines of different colours, one line for each glazing type option. The line colours are mapped to glazing type options in the legend. See below.

x Axis

Rotation (°)

y Axis

Cooling (Electric) (kWh)

Series

Glazing type

Chart type

Line chart

Font size

14px

SAVE

COPY CHART

DOWNLOAD CHART

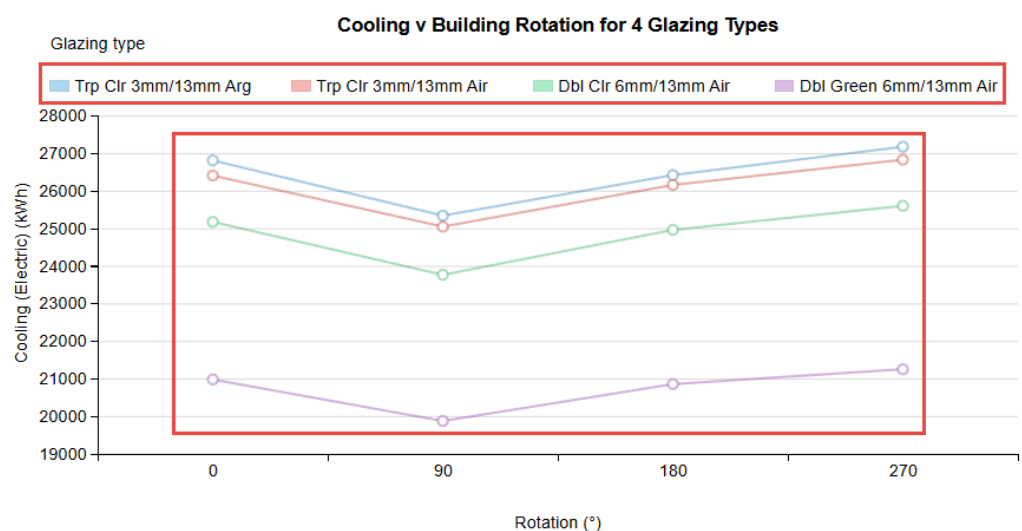
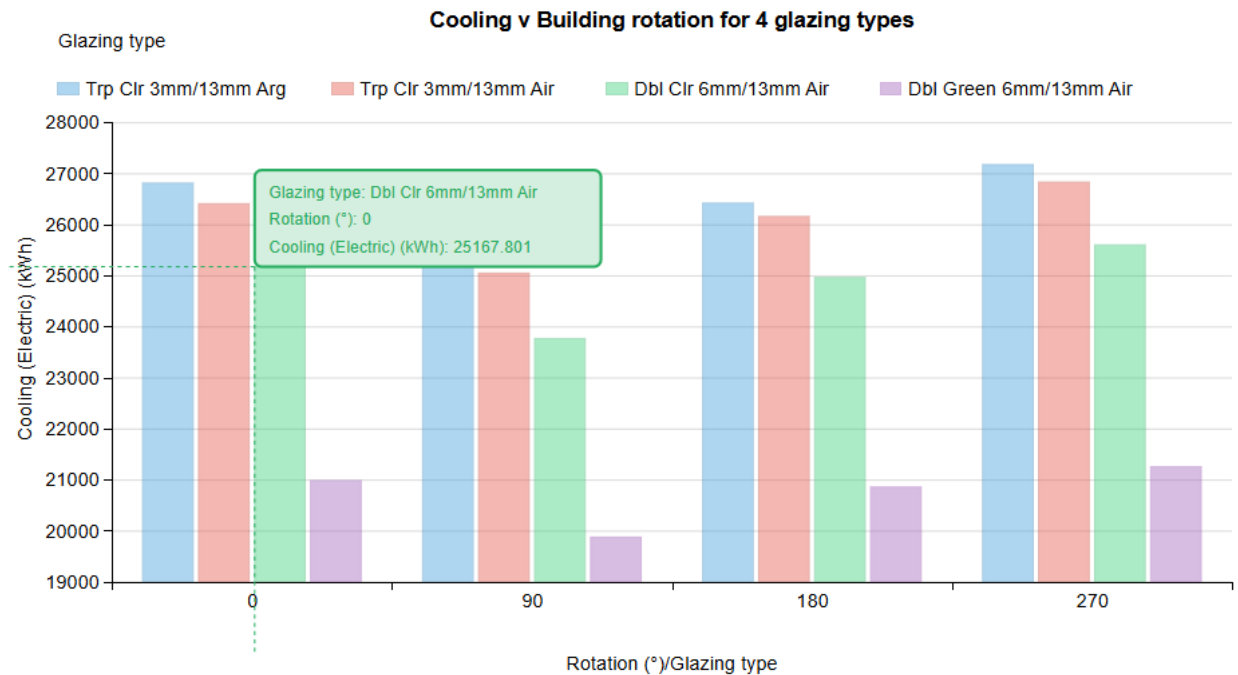


Chart type

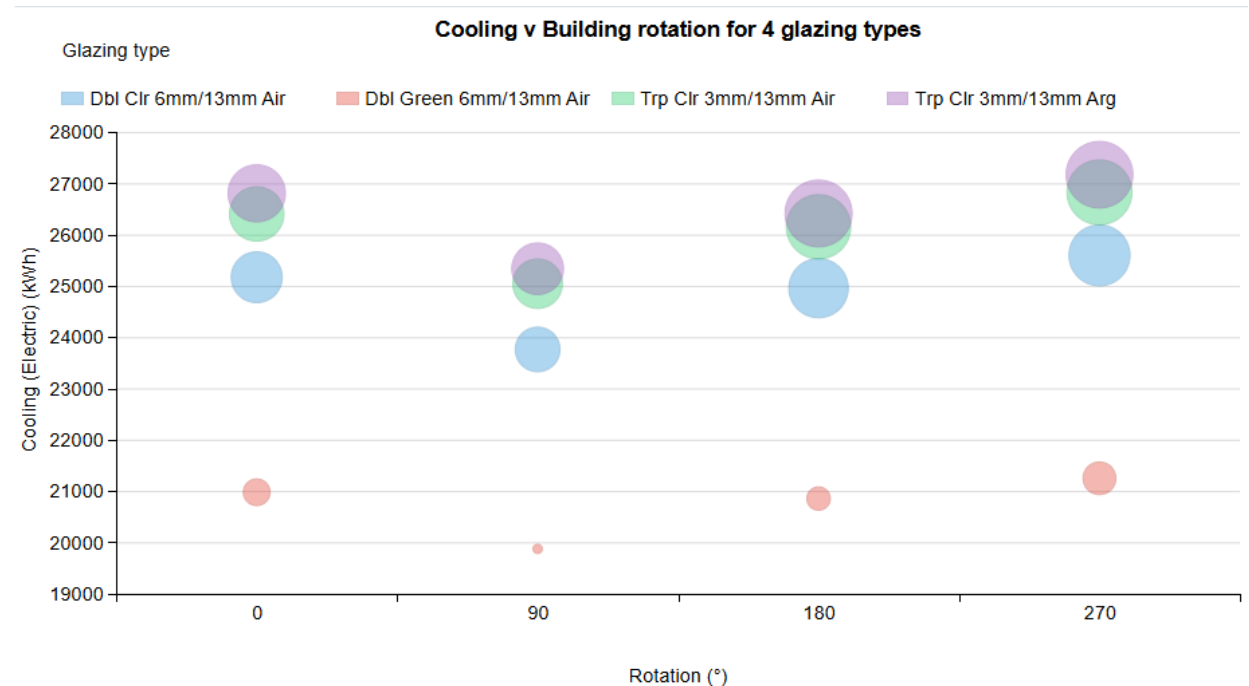
Choose from:

Line type, where a line is used to represent a set of values for each series.

Bar chart, where bars are used instead of points or bubbles to represent each value.



Bubble chart where bubbles are used instead of points or bars to provide an additional "dimension" through their size. The screenshot below shows an example where the bubble size is defined by the number of discomfort hours over the year.



Font size

See [above](#) under Bubble charts.

Panel command buttons

See [above](#) under Bubble charts.

Insights Tutorials

"Insights Tutorial - Basic Parametric Analysis " on page 2090

"Insights Tutorial - Basic Optimisation" on page 2095

Insights Tutorial - Basic Parametric Analysis

This tutorial provides a worked example to help you quickly get up to speed with using the Insights web app to analyse parametric analysis results. It uses the Parametric Analysis [Basic Tutorial](#) as a starting point.

Step 1 - Parametric Analysis Basic Tutorial

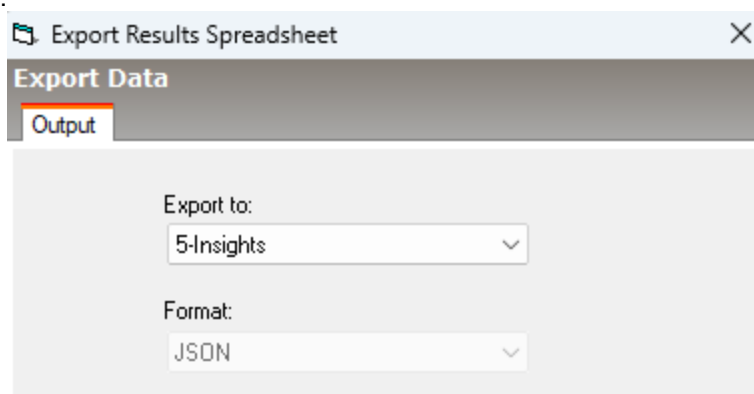
Start by working through the Parametric Analysis [Basic Tutorial](#), which walks you through the essentials of setting up and running a parametric analysis and analysing results in DesignBuilder.

Step 2 - Export results to Insights

Step 2.a - If you haven't already registered to create an account on the Insights website, go ahead and do that now. See "Insights Preparation" on page 2071 for details on this. You'll need to enter your account credentials to export results to Insights and subsequently access the website itself.

Step 2.b - Next, click on the **Export data** toolbar icon  to start the export process.

On the Export Data dialog, select the **5-Insights** "Export to" option. The Format will be automatically set to **JSON**:



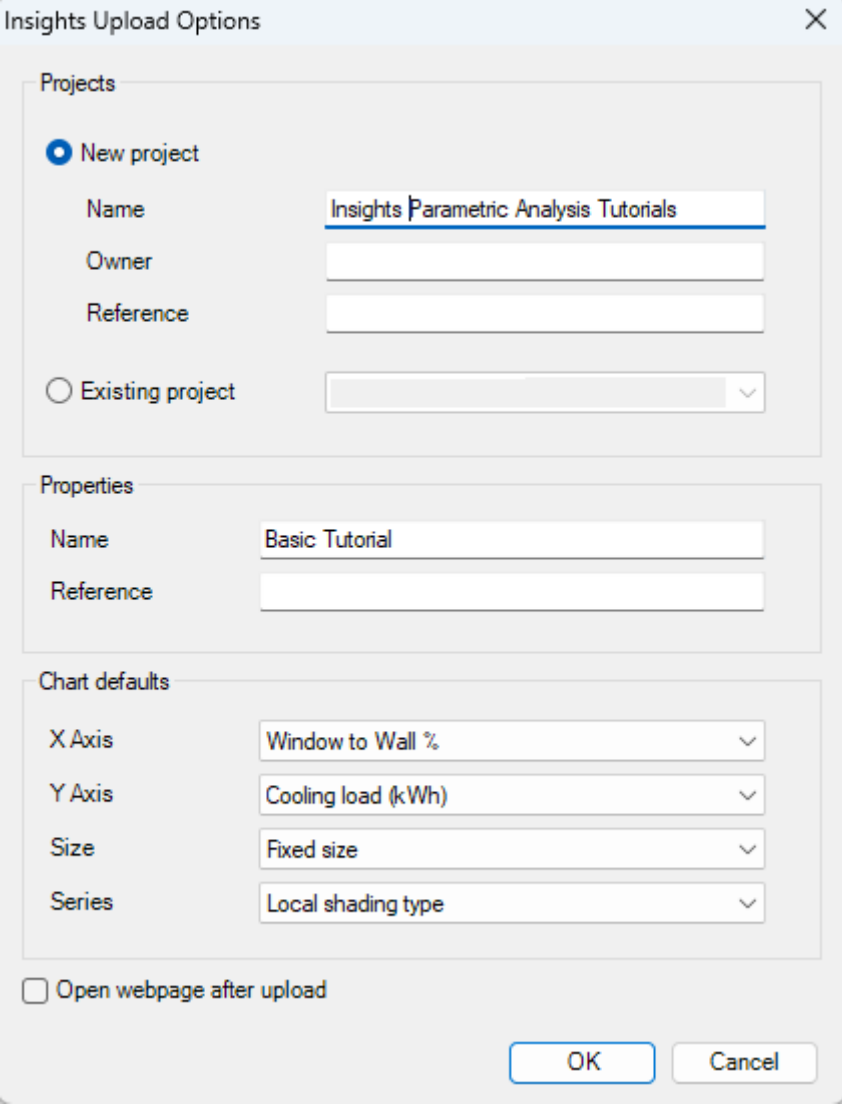
Step 2.c - When you press OK on the Export Data dialog, DesignBuilder's [Insights Login Credentials](#) dialog is displayed. Enter your email address and password and press OK to continue.

Step 2.d On the **Insights Upload Options** dialog you can upload your parametric results to Insights.

- Start by selecting the **New project** radio button to create a new project for this tutorial.
- Enter a name for the project: "Insights Parametric Analysis Tutorials".
- Enter a name for the analysis: "Basic Tutorial".

Leave the other settings in their default state.

The dialog should now look something like this:



The dialog box is titled "Insights Upload Options" and contains three main sections: Projects, Properties, and Chart defaults. At the bottom, there is a checkbox for "Open webpage after upload" and two buttons: "OK" and "Cancel".

Projects

- ☒ New project
 - Name: Insights Parametric Analysis Tutorials
 - Owner:
 - Reference:
- ☐ Existing project:

Properties

- Name: Basic Tutorial
- Reference:

Chart defaults

- X Axis: Window to Wall %
- Y Axis: Cooling load (kWh)
- Size: Fixed size
- Series: Local shading type

☐ Open webpage after upload

OK Cancel

Press OK to confirm.

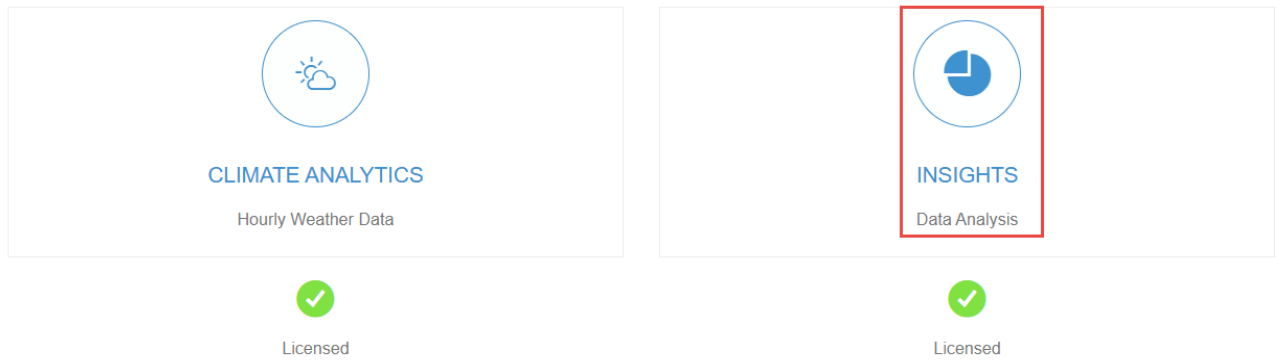
More details on the process involved with exporting results to Insights can be found at: "Exporting Parametric Results to Insights" on page 2073

Step 3 - Access the Analysis in Insights

3.a - In your web browser, navigate to: analytics.designbuilder.co.uk

3.b - Sign in using your login credentials.

3.c - Click on the large **Insights** button.



3.d - You should see the "Insights Parametric Analysis Tutorial" project listed on the projects page:

My Projects

Name	Owner	Reference	Created		
Insights Parametric Analysis Tutorials			2025-06-13 - 17:08		

Click on this project to open it to display the list of analyses associated with it.

3.e - You should see the "Basic Tutorial" Analysis listed:

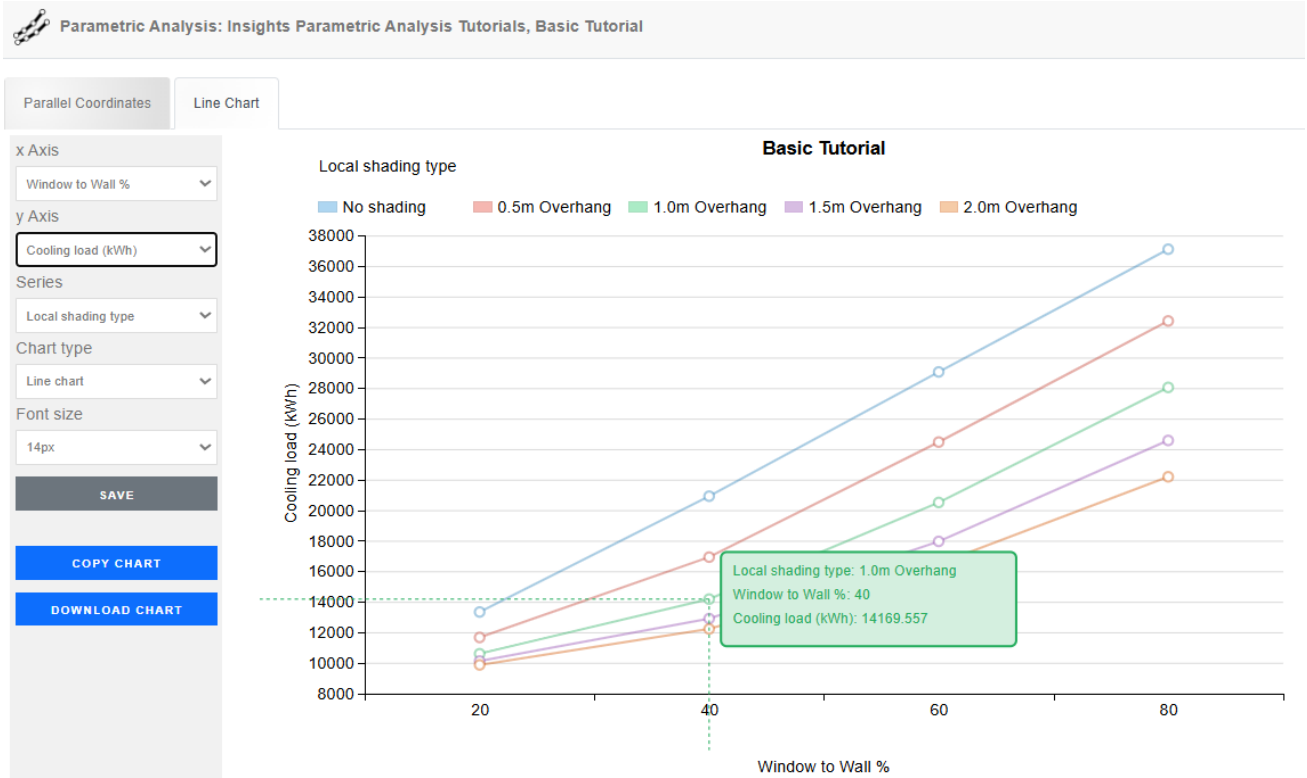
Project Name: Insights Parametric Analysis Tutorials

Analysis Type	Analysis Name	Owner	Reference	Created		
	Basic Tutorial			2025-06-13 - 17:08		

Click on the "Basic Tutorial" Analysis to display results.

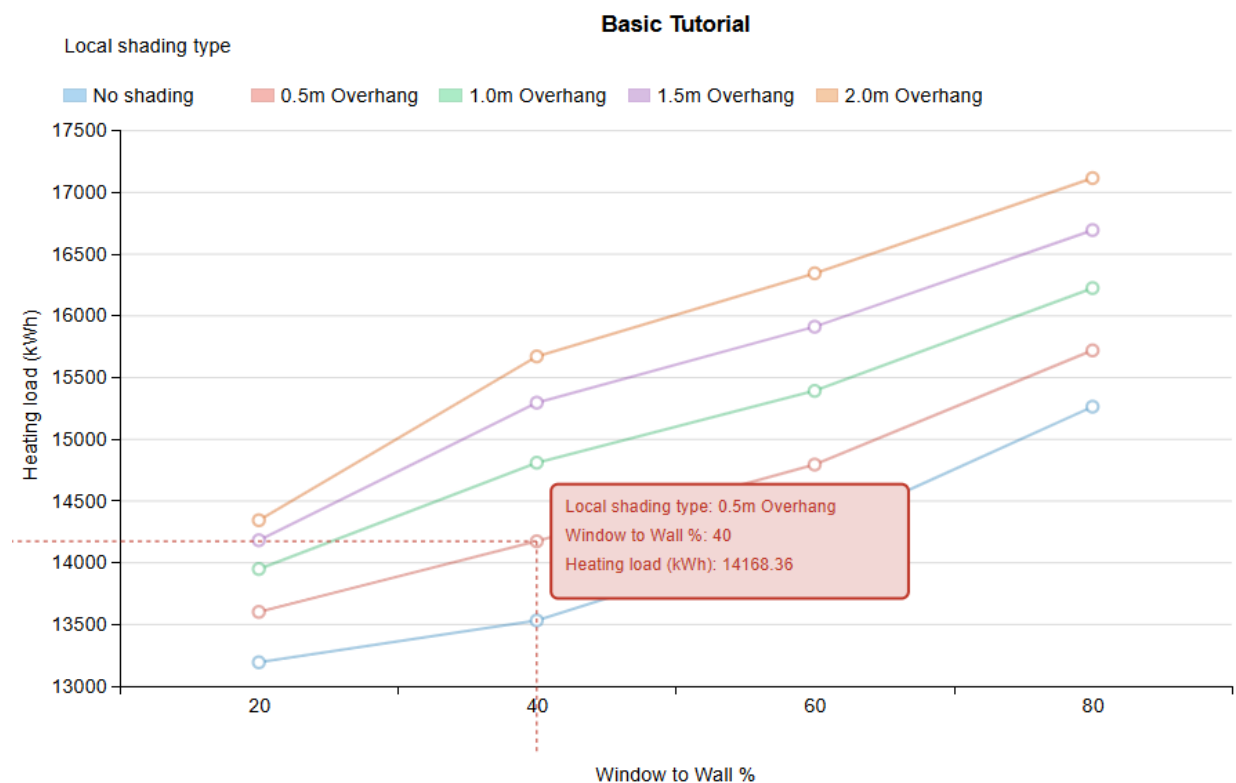
Step 4 - Analyse results in Insights

You should see a graph similar to the one below:



Now explore the results:

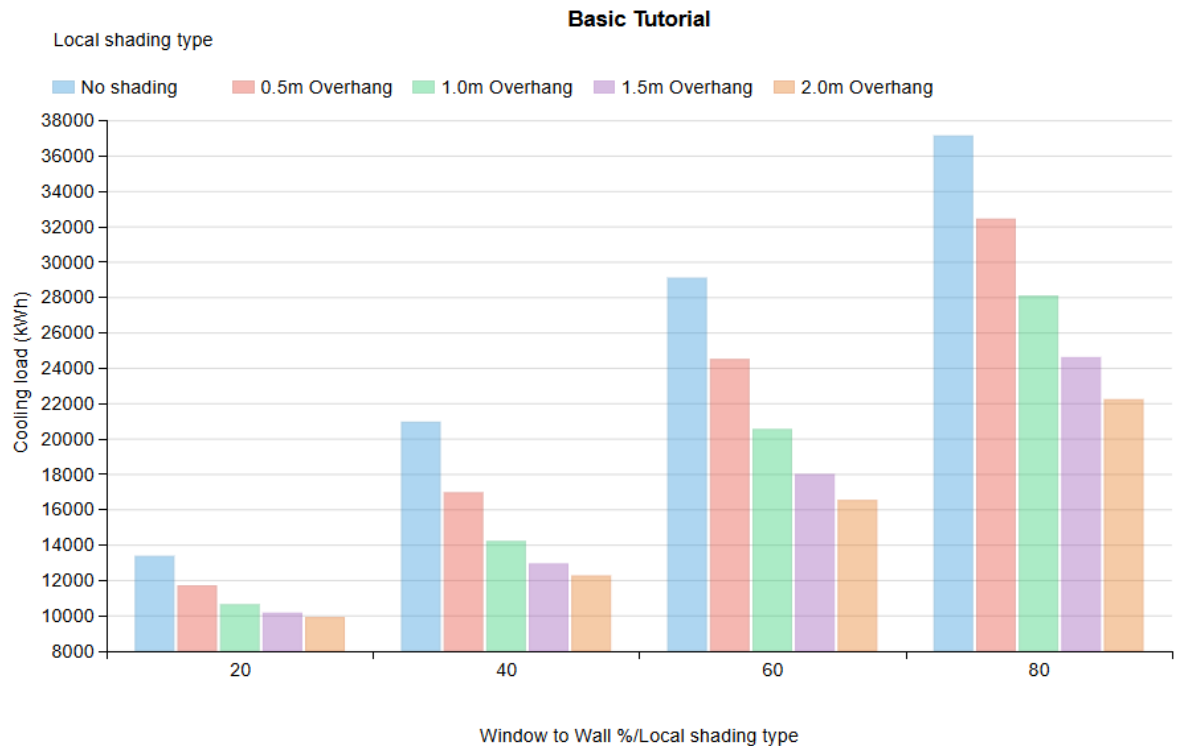
Select the "Heating load" as the Y-axis KPI.



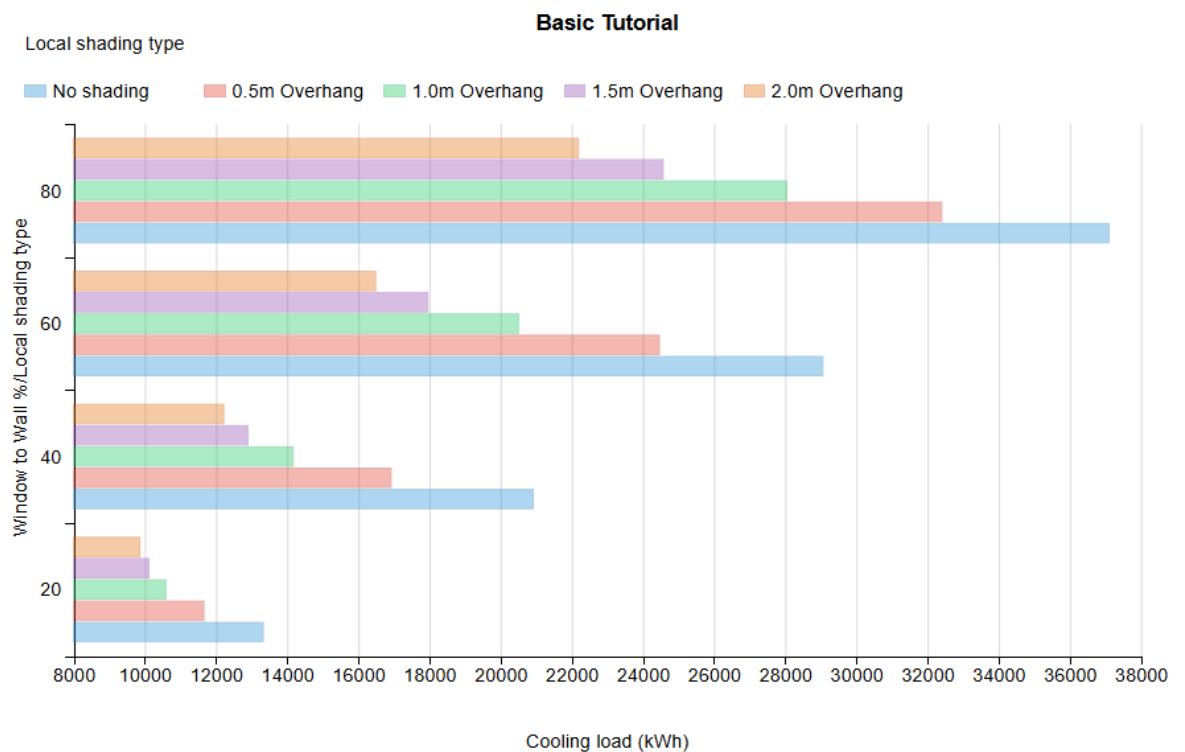
Try removing one or more of the series. For example, if you don't need to include the "2.0m Overhang" series option on the outputs, click on the corresponding legend box to remove its data.

Try different Chart types:

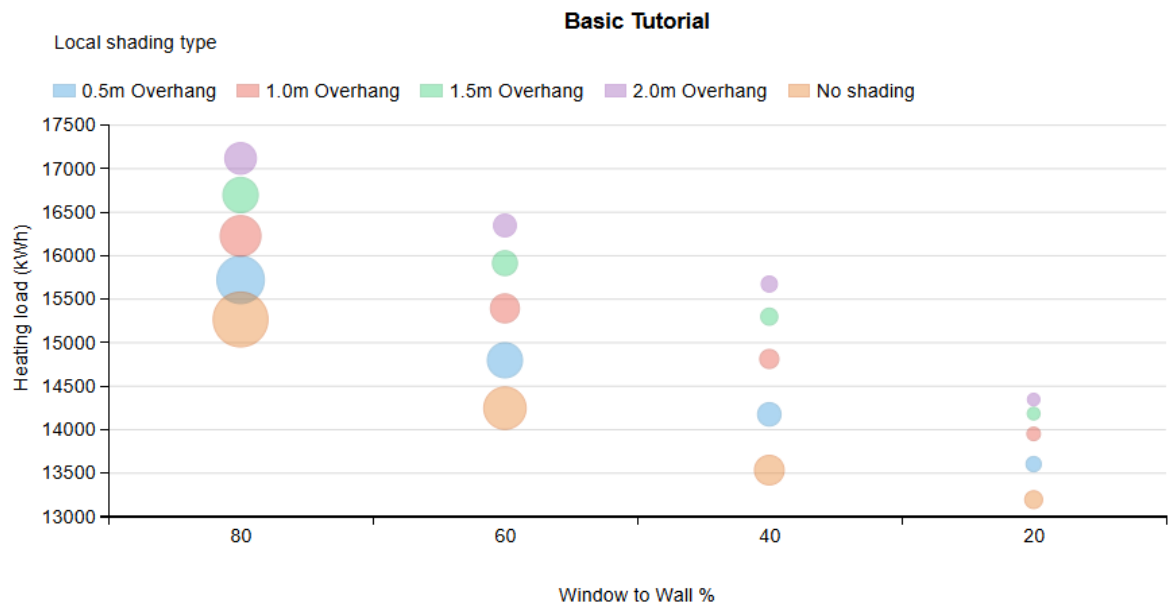
Set the **Chart type** to **Bar chart**:



Try switching the axes by setting the **x-axis** to be "Heating load" and the **y-axis** to be "Window to Wall %":



Bubble chart showing "Heating load" as the **y-axis** and "Cooling load" used to set the **Size** of the bubble.



Experiment with various **Font size** settings to see which works best for your outputs. Larger font sizes are often more appropriate when generating outputs for reports, for example.

Try using the **Copy Chart** and **Download Chart** buttons.

Use the **Save** button to save your preferred settings for your next Insights session.

Insights Tutorial - Basic Optimisation

This tutorial provides a worked example to help you quickly get up to speed with using the Insights web app to analyse optimisation results. It uses the Optimisation [Basic Tutorial](#) as a starting point.

Step 1 - Optimisation Basic Tutorial

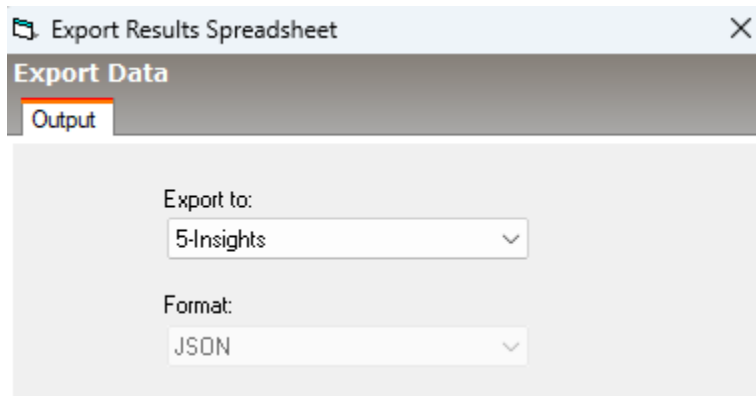
Start by working through the Optimisation [Basic Tutorial](#), which walks you through the essentials of setting up and running an optimisation and analysing results in DesignBuilder.

Step 2 - Export results to Insights

Step 2.a - If you haven't already registered to create an account on the Insights website, go ahead and do that now. See "Insights Preparation" on page 2071 for details on this. You'll need to enter your account credentials to export results to Insights and subsequently access the website itself.

Step 2.b - Next, click on the **Export data** toolbar icon  to start the export process.

On the Export Data dialog, select the **5-Insights** "Export to" option. The Format will be automatically set to **JSON**:



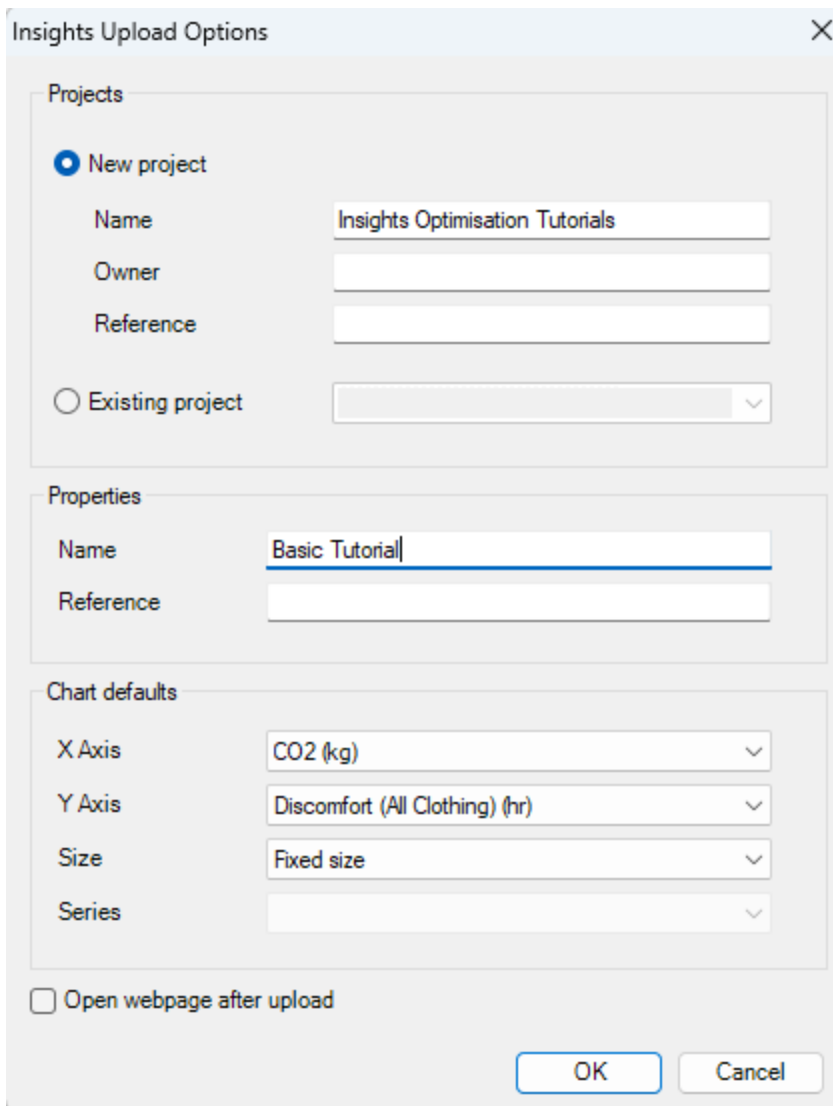
Step 2.c - When you press OK on the Export Data dialog, DesignBuilder's [Insights Login Credentials](#) dialog is displayed. Enter your email address and password and press OK to continue.

Step 2.d On the **Insights Upload Options** dialog you can upload your parametric results to Insights.

- a. Start by selecting the **New project** radio button to create a new project for this tutorial.
- b. Enter a name for the project: "Insights Optimisation Tutorials".
- c. Enter a name for the analysis: "Basic Tutorial".

Leave the other settings in their default state.

The dialog should now look something like this:



The dialog box is titled "Insights Upload Options" and contains three main sections: Projects, Properties, and Chart defaults.

Projects

- ☒ New project
 - Name: Insights Optimisation Tutorials
 - Owner:
 - Reference:
- ☐ Existing project
 - Dropdown menu:

Properties

- Name: Basic Tutorial
- Reference:

Chart defaults

- X Axis: CO2 (kg)
- Y Axis: Discomfort (All Clothing) (hr)
- Size: Fixed size
- Series:

☐ Open webpage after upload

Buttons: OK, Cancel

Press OK to confirm.

More details on the process involved with exporting results to Insights can be found at: "Exporting Parametric Results to Insights" on page 2073

Step 3 - Access the Analysis in Insights

3.a - In your web browser, navigate to: analytics.designbuilder.co.uk

3.b - Sign in using your login credentials.

3.c - Click on the large **Insights** button.



CLIMATE ANALYTICS

Hourly Weather Data

✓

Licensed



INSIGHTS

Data Analysis

✓

Licensed

3.d - You should see the "Insights Optimisation Tutorials" project listed on the projects page:

My Projects

Name	Owner	Reference	Created		
Insights Optimisation Tutorials			2025-06-16 - 15:18		

Click on this project to open it to display the list of analyses associated with it.

3.e - You should see the "Basic Tutorial" Analysis listed:

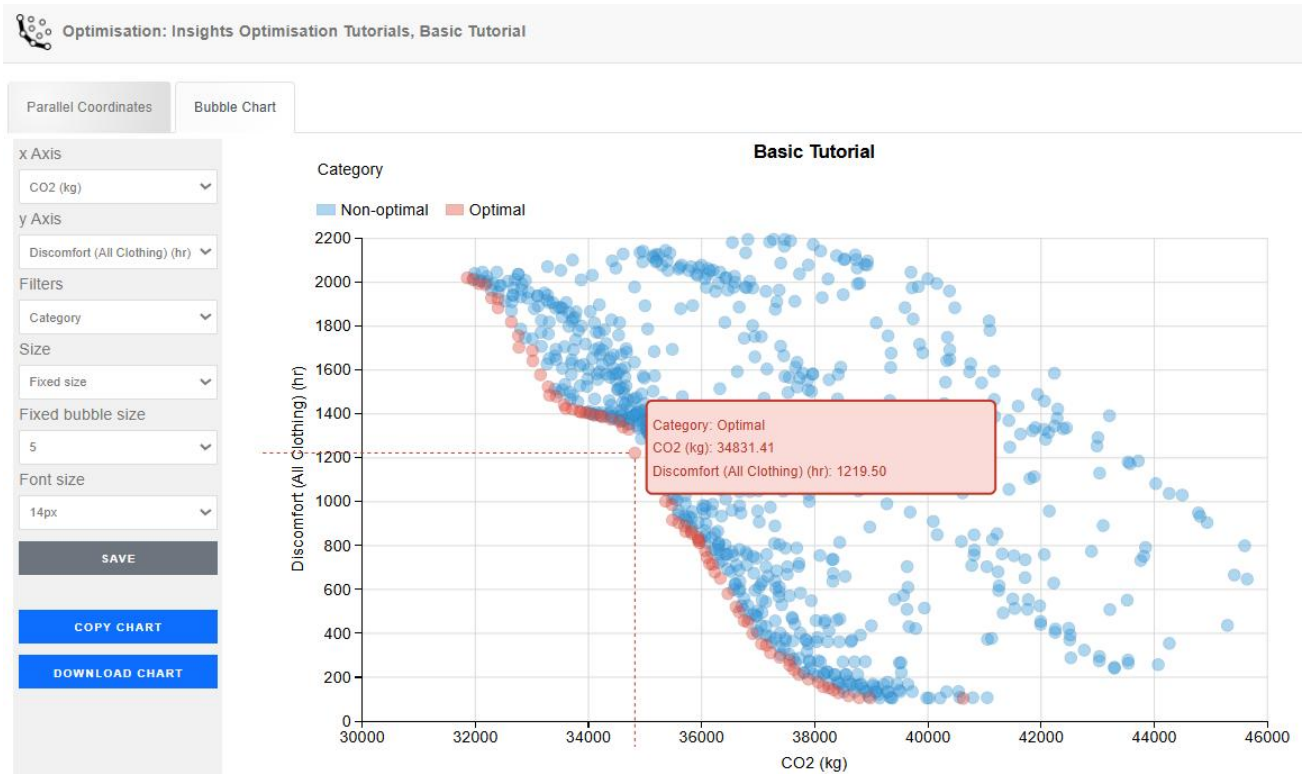
Project Name: Insights Optimisation Tutorials

Analysis Type	Analysis Name	Owner	Reference	Created		
	Basic Tutorial			2025-06-16 - 15:18		

Click on the "Basic Tutorial" Analysis to display results.

Step 4 - Analyse results in Insights

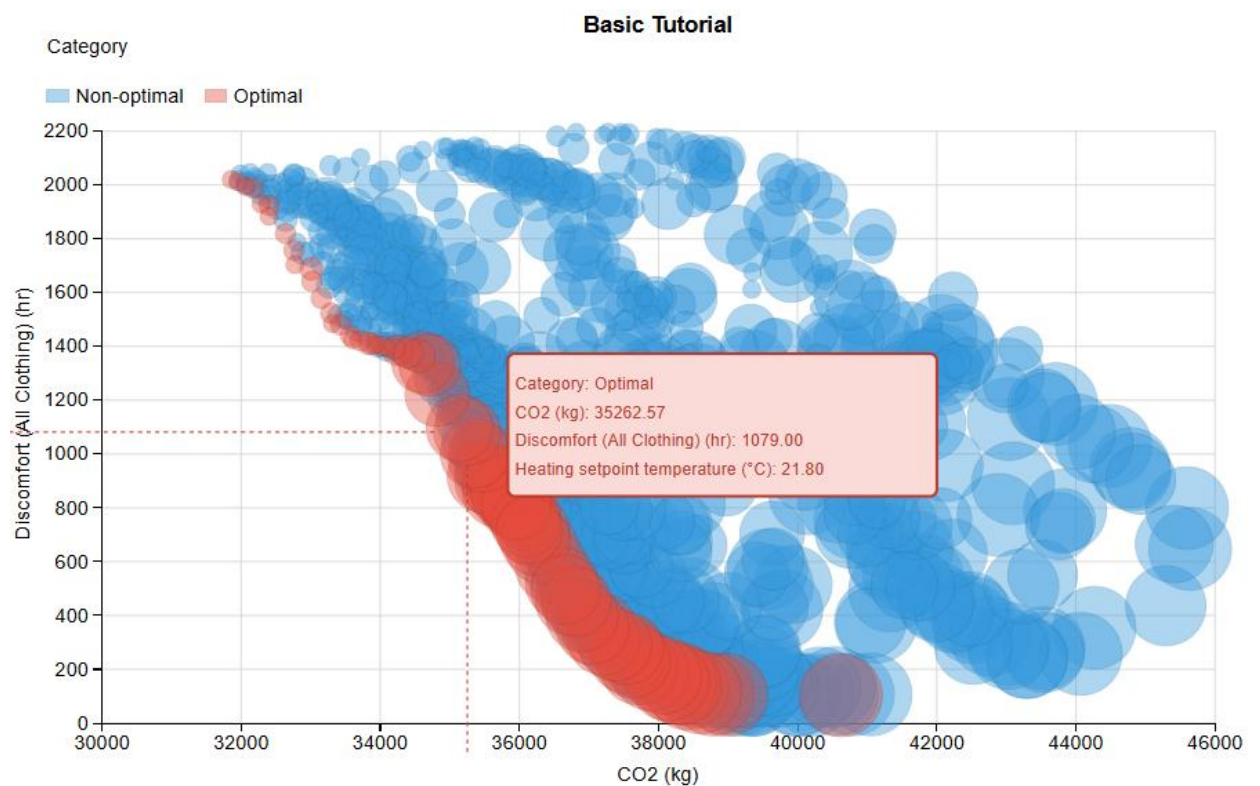
You should see a graph similar to the one below:



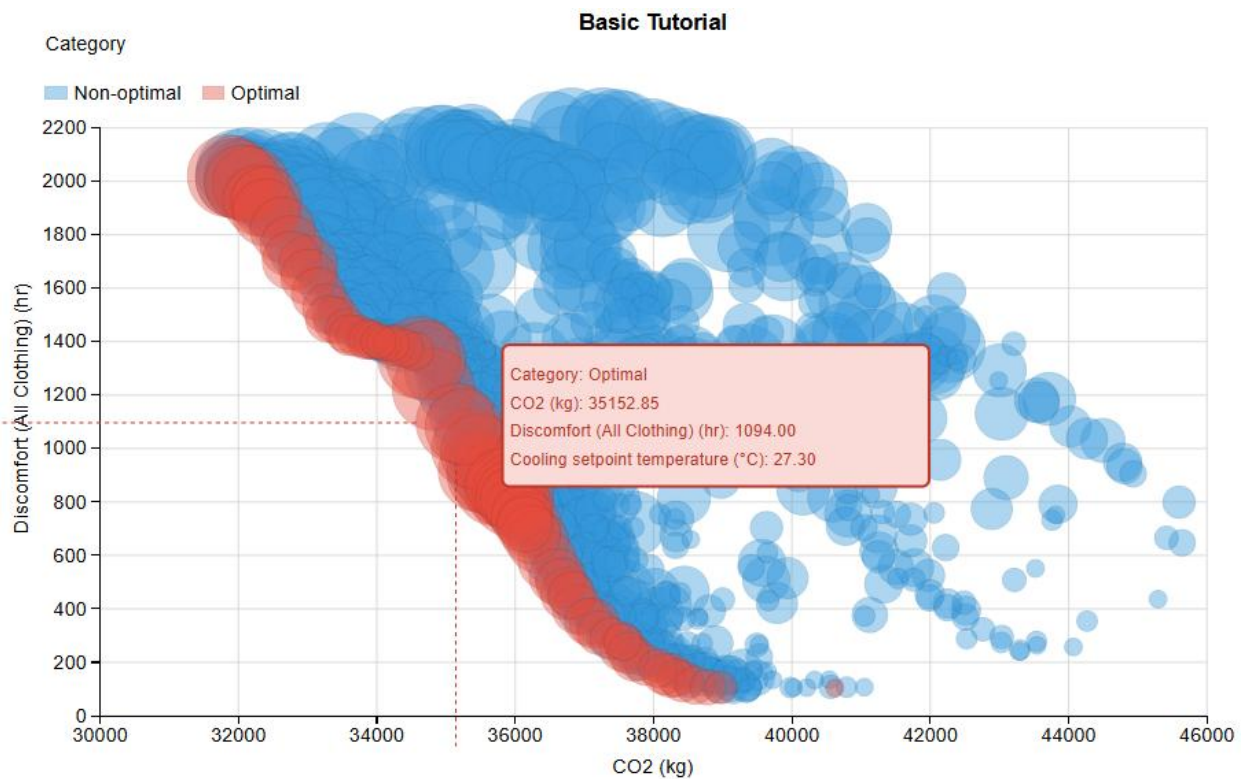
Now explore the results:

Move your mouse over the results points and note the dynamic tooltips and dotted lines that intersect with the x and y-axes.

Experiment with different Bubble **Size** settings. For example, selecting "Heating setpoint temperature" results in a graph like the one below which shows high values of "Heating setpoint temperature" giving rise to low discomfort hours over the year and high carbon emissions, and low values resulting in Low Carbon emissions/high discomfort. See below:



As expected, selecting "Cooling setpoint temperature" to define the Bubble Size gives the reverse trend:



Try various **Min/Max bubble size** settings to see which works best for your analyses.

Experiment with various **Font size** settings to see which works best for your outputs. Larger font sizes are often more appropriate when generating outputs for reports, for example.

Try using the **Copy Chart** and **Download Chart** buttons.

Use the **Save** button to save your preferred settings for your next Insights session.

HVAC Loads Report

See also: "Export HVAC Loads Report" on page 49

UK Certification

This Program help does not attempt to cover use of the software when the [Analysis type](#) is set to **2-SBEM** or **10-DSM**.

For details on how to work with DesignBuilder SBEM and DSM, please download the separate manuals from the [Downloads > Documents](#) section of our website.

Extensibility

DesignBuilder may be extended through various types of scripts or through plugins. Each system is targeted at a different use-case and different levels of complexity. Both systems may be used at the same time, and multiple active plugins and scripts are supported.

[Scripts](#) are intended to hook into certain stages of a DesignBuilder simulation in order to modify or report on the process.

[Plugins](#) are a superset of Scripts, providing extra functionality such as the ability to create a custom sub-menu in DesignBuilder's main-menu, which may allow for ad-hoc processing of DesignBuilder data.

Scripts can be written in the C# or Python programming languages. Plugins are written in the C# programming language.

In general scripts are used where a quick solution is required and it is acceptable for the program to control the calling at key hook points. Plugins on the other hand are more appropriate for larger customisation projects where the code is pre-compiled and can be executed under user control if necessary.

Note: The [Extensibility Tools and API User Guide](#) explains how users and developers can customise and extend DesignBuilder's functionality through scripts and plugins. It covers the use of the DesignBuilder API to access and manipulate building models, simulation processes, and data tables. The guide details how to write and enable scripts in C# or Python, implement hook points during simulations, and develop more advanced plugins that can add dynamic menus and integrate external libraries. It provides information on working with model attributes, database tables, and geometry extraction, offering a complete framework for creating tailored workflows and automating tasks within DesignBuilder.

Note: The [DesignBuilder API Reference Guide](#) is the main source of help on all of the API classes, collections, enumerations etc.

Scripting



DesignBuilder provides 3 different sets of scripting tools to help more expert users customise their simulations and generate special reports.

"Energy Management System (EMS)" on page 2102 - powerful set of EnergyPlus runtime scripting tools to help you write portable code to customise EnergyPlus' simulation behaviour.

[CS-Script](#) - where standard C# code can be executed, typically before and after simulations, to modify the default IDF generated by DesignBuilder, access the DesignBuilder model input and output data to produce custom reports.

[Python](#) - set of tools which allow Python code to be executed at key hook points, typically before and after simulations like CS-Script, to modify the default IDF generated by DesignBuilder through the [eppy](#) scripting tool, or for post-processing of results for custom reporting etc.

The Scripting tools can be accessed from the [Scripts](#)  toolbar icon and also from the **Tools > Scripts** menu option which opens the **Script Manager** dialog. More details on how to use the scripting tools can be found in the help topics on each type of script.

Note: The 3 types of script can also be used to develop [Custom EMS](#), [Custom IDF](#) and [Custom Script](#) design variables.

Plugins

Plugins are capable of all of the CS-Script functionality, but they also allow custom sub-menus to be added to the main DesignBuilder **Plugins** menu allowing general processing of DesignBuilder models and results. They are usually more suitable for developing larger projects where source code can be compiled into a DLL. For example if developing a new code compliance module that required an in-depth processing of the DesignBuilder model and results then this would normally be implemented as a plugin rather than a script.

You can access DesignBuilder plugins from our:

[Scripts and Plugins download page](#) - sample plugins developed by DesignBuilder

[Github repository](#) - various sample plugins provided with source code to help developers to develop their own plugins

Energy Management System (EMS)

EnergyPlus provides an encyclopaedic range of modelling capabilities, but every project is different and some may require special simulation functionality not provided in the standard engine. EMS provides custom simulation runtime control to override selected aspects of standard EnergyPlus behaviour. A simple programming language called EnergyPlus Runtime Language (Erl) is used to control custom behaviour. EnergyPlus interprets and executes the Erl program as the model is being simulated, thus the EMS provides a powerful way to extend the capability of EnergyPlus without the need to become an expert in programming its source code.

DesignBuilder also provides an additional set of [Erl language extensions](#) that allow scripts to be written to include logic and model data for selected zones, surfaces and windows. These extensions allow "portable" scripts to be written that apply to any model and so can be reused in future projects and shared with colleagues.

Tip: The help on DesignBuilder EMS is intended to be read together with the main EnergyPlus [EMS application guide](#) which provides most of the information required to understand how to use EMS with detailed descriptions and examples.

See the [EMS Demo Tutorial](#) for an introduction to using the DesignBuilder EMS runtime scripting system including DB Erl extensions.

Introduction

The Energy Management System (EMS) is one of the high-level control methods available in EnergyPlus. An EMS script is able to access a wide variety of "sensor" data and use this data to direct various types of control actions. The original concept was to emulate, inside EnergyPlus, the types of controls possible with the digital energy management systems used in real buildings. However there are many other application for the EMS including:

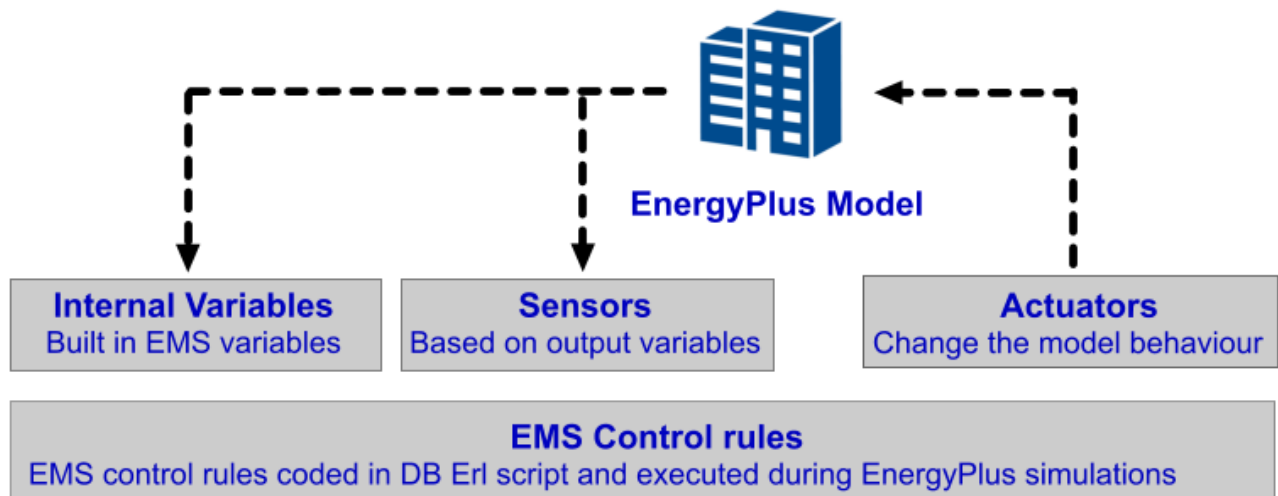
- Obtain more precise control over simulated behaviour for HVAC systems, glazing, lighting etc (e.g. override default EnergyPlus control linkage between economisers and heat recovery).
- Linking one system with another (e.g. link earth tube to HVAC inlet).
- Provide special outputs not provided natively by EnergyPlus (e.g. generate CIBSE TM52 or TM59 outputs).
- Researching new building systems.

The EMS allows the functionality of many of the existing high-level control objects to be overridden. For example the EMS can:

- Turn on and off pumps and fans and adjust their performance characteristics.

Turn on and off lights and electric equipment,
 Change the set points on zone thermostats,
 Dispatch on-site generation.
 Change the set points on system nodes.
 Turn on and off supply-side heating, cooling, and heat rejection equipment such as boilers, chillers,
 cooling towers.
 Turn on and off on-site generation equipment.

The EMS uses the concept of **sensors** to obtain information from elsewhere in the model for use in control calculations. **Actuators** are the mechanism for influencing changes on the model and code is written to tell the Erl processor how to apply the control laws.



Sensor objects are used to declare Erl variables which are linked to EnergyPlus output variables or meters. Actuators map Erl variables to control actuators elsewhere in EnergyPlus. The EMS then initiates control actions by changing the value of this variable inside an Erl program.

The way it works is probably best understood by studying working EMS scripts such as those provided by DesignBuilder alongside this documentation and that in the EnergyPlus [EMS application guide](#).

Note: EMS is an advanced feature of EnergyPlus and is not for beginners. To use it you will need to be comfortable writing your own custom code and have a thorough understanding of how you want your models to behave. If you are intimidated by the idea of writing programs to adjust the fine details of how an EnergyPlus model runs then EMS is probably not for you. However, if you relish the idea of being able to write code to override default behaviour, you will find that EMS can help you model many features to your exact specification, significantly extending EnergyPlus' normal range of capabilities.

DesignBuilder Extensions

DesignBuilder includes some important [extensions](#) to the standard EnergyPlus EMS and Erl system allowing DB EMS programs to be reused and shared. DesignBuilder extensions provide:

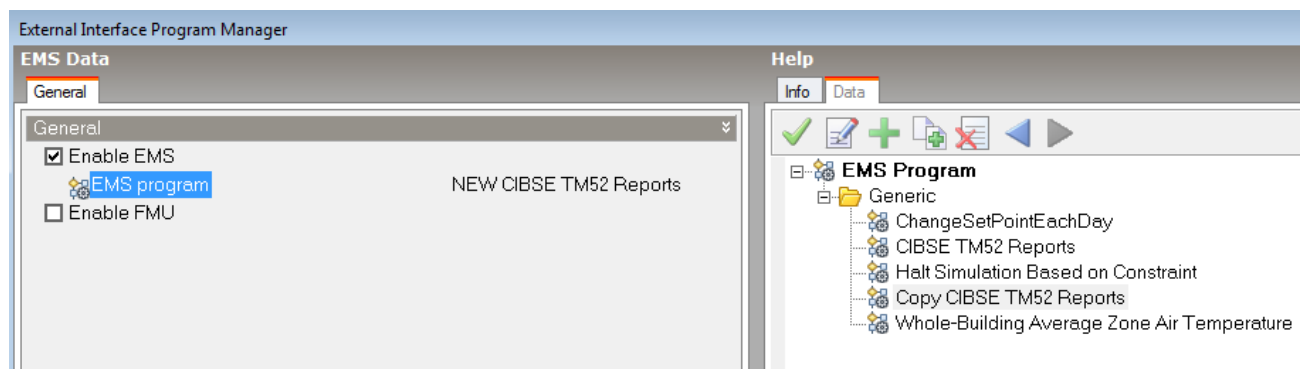
Portable Erl scripts

[Zone, surface and opening loops](#) allow code to be written that doesn't reference specific object names in Erl scripts.

All available [actuators, sensors, internal variables and program entry points](#) can be selected from lists. Connection between the script and the model, e.g. attribute settings from the model can be [inserted](#) in the script or used to control whether parts of the script are executed or not.

How to Use EMS

The DesignBuilder EMS functionality can be accessed from the **Scripts**  toolbar icon and also from the **Tools > Scripts** menu option which opens the **Script Manager** dialog.



Previously created EMS programs are listed on the data panel to the right. These can be selected, copied, deleted and edited in the usual way.

Two steps are required to enable an EMS program:

The **Enable scripts** checkbox on the **Script Manager** dialog must be checked.

The **Enable program** checkbox on the **Edit Script** dialog must be checked.

Note: Selecting an EMS program on the **Script Manager** dialog is not enough to ensure its script is included in the simulation - the **Enable program** checkbox is the important setting.

[DB Erl](#) scripts are defined on the **Edit Script** dialog which provides utilities for quickly adding template script for new sensors, actuators, internal variables and programs.

Note: EMS scripts are only run for Simulations and are not applied to Cooling and Heating design calculations.

Using EMS to Include Additional IDF

As well using EMS for advanced control and customisation you can also use the system to include simple additional IDF script with the model. This is an easy way to add more outputs or to add HVAC or other EnergyPlus IDF objects not yet supported by DesignBuilder. It offers a more flexible approach than the [Include IDF](#) option on the Model and Simulation options dialogs because the scripting system saves the IDF data within the dsb file making it completely self-contained and portable. Also you can store and manage a list of various IDF datasets with EMS which makes it easier to include/exclude various models and/or outputs in a controlled way. A further advantage when adding IDF for outputs is that the [Add Output](#) dialog lists all outputs available for the current model which provides a convenient way to select any extra outputs that are required.

Support

It is possible to write quite complex EMS programs and there is plenty of scope for making mistakes, which can take time to track down and fix. Please be aware that the DesignBuilder Support Desk is not generally able to offer a free service for debugging or advising on the possible cause of errors in EMS programs. See the [Support Desk Policy](#).

The EnergyPlus [EMS application guide](#) provides a section on debugging EMS programs which is a good starting point for tracking down problems.

EMS Edit Script Dialog

The EMS **Edit Script** dialog is used to create and edit individual EMS programs using [DesignBuilder Erl](#) code.

General

Name

Enter the name of the EMS script that will appear in the [Script Manager](#) list for selection.

Description

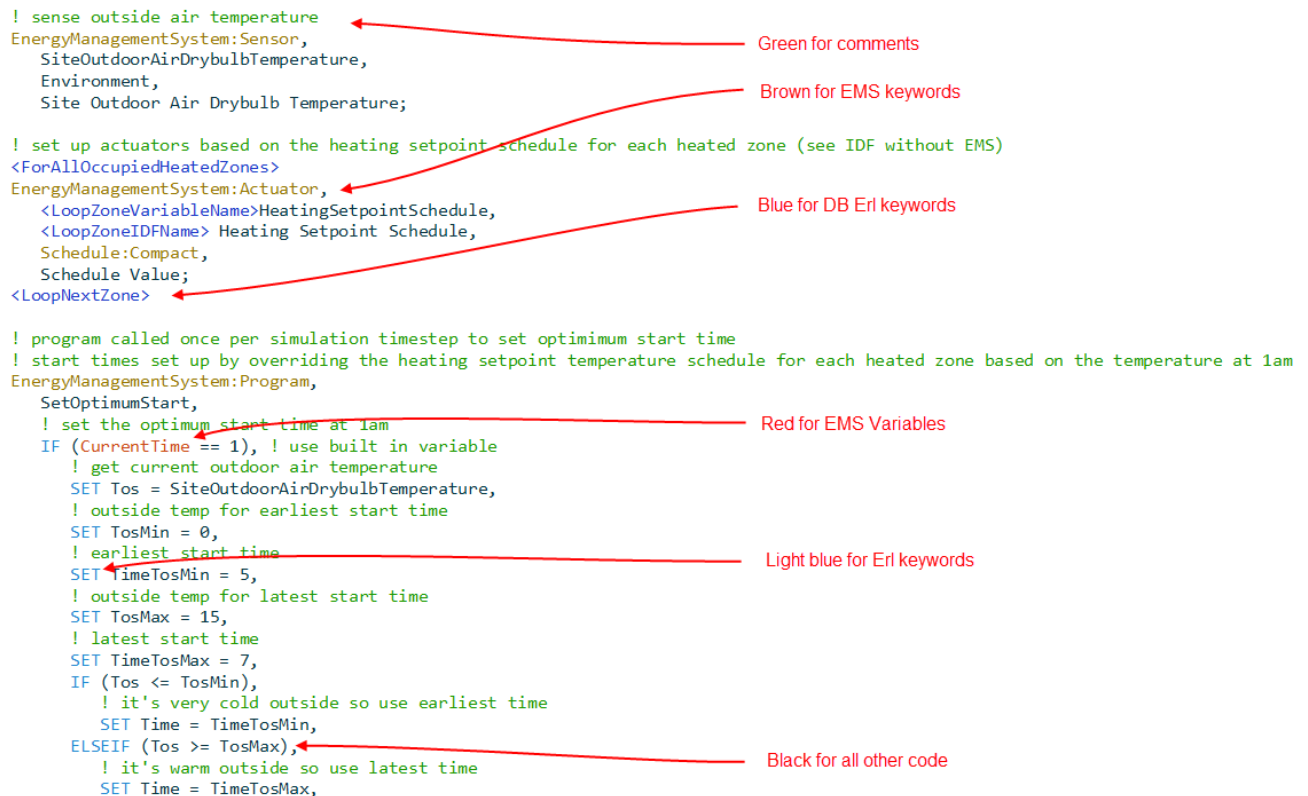
The description can be used to add any extra notes or description of the script.

Category

The **Category** for an EMS script is set to **EMS** and should not generally be changed once the script has been created.

Script

The EMS script editor provides colour syntax highlighting to make it easier to read the code.



```

! sense outside air temperature
EnergyManagementSystem:Sensor,
  SiteOutdoorAirDrybulbTemperature,
  Environment,
  Site Outdoor Air Drybulb Temperature;

! set up actuators based on the heating setpoint schedule for each heated zone (see IDF without EMS)
<ForAllOccupiedHeatedZones>
EnergyManagementSystem:Actuator,
  <LoopZoneVariableName>HeatingSetpointSchedule,
  <LoopZoneIDFName> Heating Setpoint Schedule,
  Schedule:Compact,
  Schedule Value;
<LoopNextZone>

! program called once per simulation timestep to set optimum start time
! start times set up by overriding the heating setpoint temperature schedule for each heated zone based on the temperature at 1am
EnergyManagementSystem:Program,
  SetOptimumStart,
  ! set the optimum start time at 1am
  IF (CurrentTime == 1), ! use built in variable
  ! get current outdoor air temperature
  SET Tos = SiteOutdoorAirDrybulbTemperature,
  ! outside temp for earliest start time
  SET TosMin = 0,
  ! earliest start time
  SET TimeTosMin = 5,
  ! outside temp for latest start time
  SET TosMax = 15,
  ! latest start time
  SET TimeTosMax = 7,
  IF (Tos <= TosMin),
  ! it's very cold outside so use earliest time
  SET Time = TimeTosMin,
  ELSEIF (Tos >= TosMax),
  ! it's warm outside so use latest time
  SET Time = TimeTosMax,

```

Green for comments

Brown for EMS keywords

Blue for DB Erl keywords

Red for EMS Variables

Light blue for Erl keywords

Black for all other code

Tip: Standard Erl syntax is described in the main [EMS application guide](#) and DesignBuilder extensions to the standard Erl syntax are described on the "DesignBuilder Erl" on page 2116 page.

Note: Sometimes it can be useful to create and/or edit scripts in an external text editor and paste the text into this dialog when finished. In this case you should bear in mind that the DesignBuilder Script Editor only supports ANSI characters and not the full UTF8 (Unicode) character set. If you paste Unicode text into the Script Editor there is a risk that it will fail to load correctly into EnergyPlus. You should therefore ensure that the script in the external text editor uses the ANSI character set before copying it.

If you are using Notepad++ as your external text editor, from the "Encoding" top menu item use "Convert to

ANSI". Alternatively, if using the standard Windows Notepad editor use the "File > Save As" menu option to specify encoding as ANSI.

It can also be opened into plain old Notepad and on "File" → "Save As" specify encoding as ANSI

Enable program

Check this checkbox to ensure that the EMS script is applied to any simulations carried out. EMS scripts are not applied to Heating and Cooling design calculations.

Utilities

Insert EMS/IDF code

DesignBuilder provides several utilities to make it easy to insert EMS and/or IDF code into the script for sensors, variables, internal variables, programs etc. These are accessed by clicking on the **Edit Script** dialog Info panel links:

-  "Add Output" on page 2106
-  "Add Program" on page 2108
-  "Add Sensor" on page 2109
-  "Add Actuator" on page 2112
-  "Add Internal Variable" on page 2113
-  "Add EMS Variable" on page 2115
-  "Add DB Variable" on page 2116

Some of the above tools add a simple keyword while others insert several lines of DB Erl code:

Other tools

Refresh EMS Sensor, Actuator and Variable lists - regenerates the .edd and .rdd files needed to create the lists for outputs, sensors, actuators and internal variables. There will be a delay (of several minutes for larger models) while EnergyPlus is run to generate the necessary files.

Align comments to cursor provides a quick way to ensure that all inline comments at the end of lines of code are left-aligned with the current cursor position. This can be a useful way to quickly tidy up comments in a program.

Maximise editor - click to maximise the **Edit Script** dialog and hide all controls apart from the actual editor. This can be useful to gain the maximum working area while working on a script. Click again to restore the controls to normal non-maximised configuration. A tick will appear in the icon to the left of the Maximise editor link in the info panel when it is maximised.

EMS code libraries

Publicly accessible libraries of EMS scripts can be accessed by clicking on the links at the bottom of the Info panel. The idea is for DesignBuilder users to share any scripts they have written and found useful.

[DesignBuilder UK EMS library](#)
[DesignBuilder Spain EMS library](#)

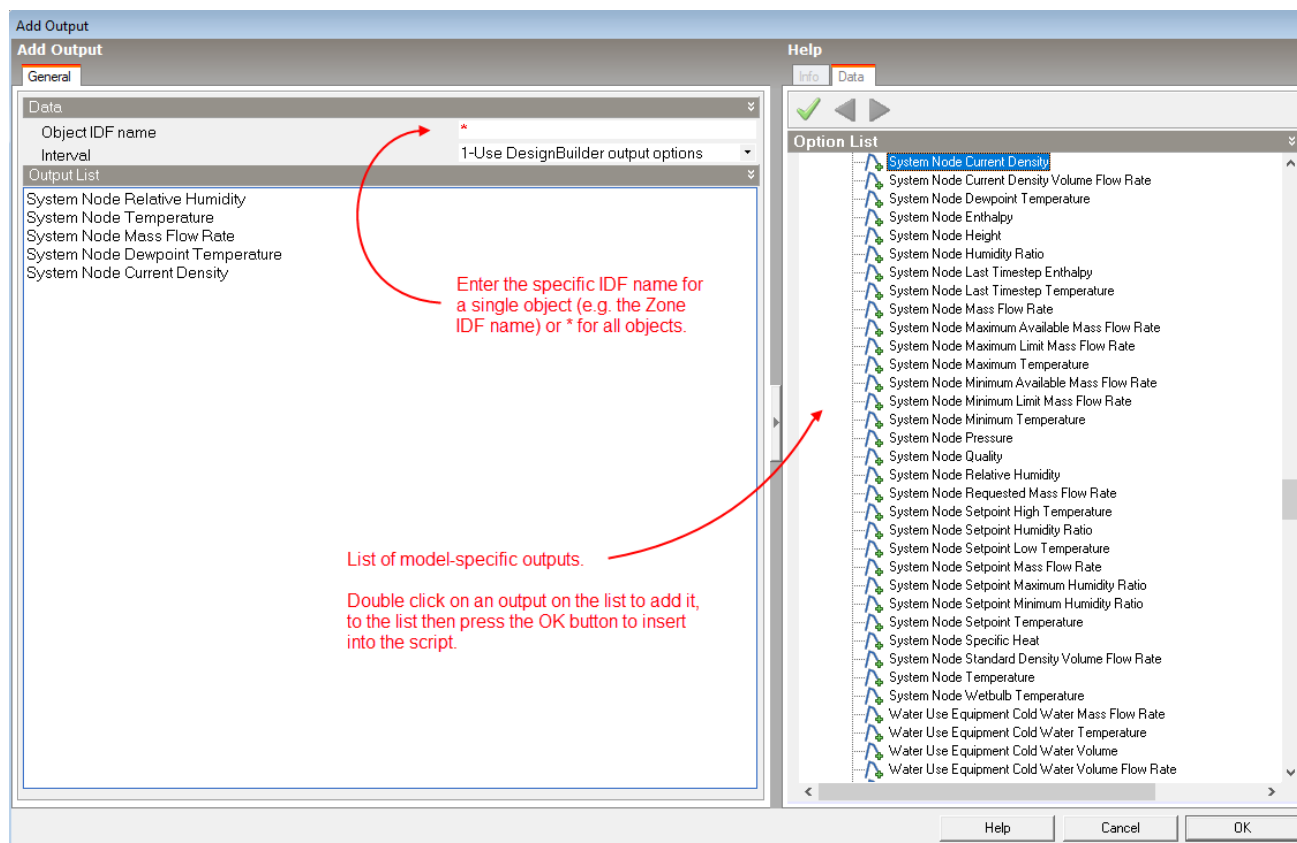
Add Output

It can sometimes be useful to add extra outputs either as part of an EMS program or to provide extra information on model performance.

The **Add Output** dialog is accessed by clicking on the **Add output** link in the Info panel of the [Edit Script](#) dialog when editing an EMS script.

Note: Extra outputs added in this way must be viewed in the [Results Viewer](#) and are not generally available for view in the main DesignBuilder results screen.

With the **Add Output** dialog open you can select from a list of outputs that is generated based on the actual model.



When you press the OK button the list of selected outputs is inserted into the script at the cursor position. Or press the Cancel button to not add any outputs to the script.

Add outputs to the list

To add an output listed in the right hand data panel to the list for insertion either a) double-click on the required output in the right-hand list, or b) single-click on the required output and then click on the Select



icon.

Tip: Any Outputs that have been added but are not actually required can be removed from the list by selecting the text that is not required and pressing the <Delete> key.

Object IDF name

Enter a specific object name such as a zone or surface IDF name, or enter "*" to indicate all relevant objects.

Tip: EnergyPlus runs faster if you use a single output report request using the "*" option instead of a long list of output report requests that use actual object names.

Interval

Select the reporting interval for the selects outputs from the list:

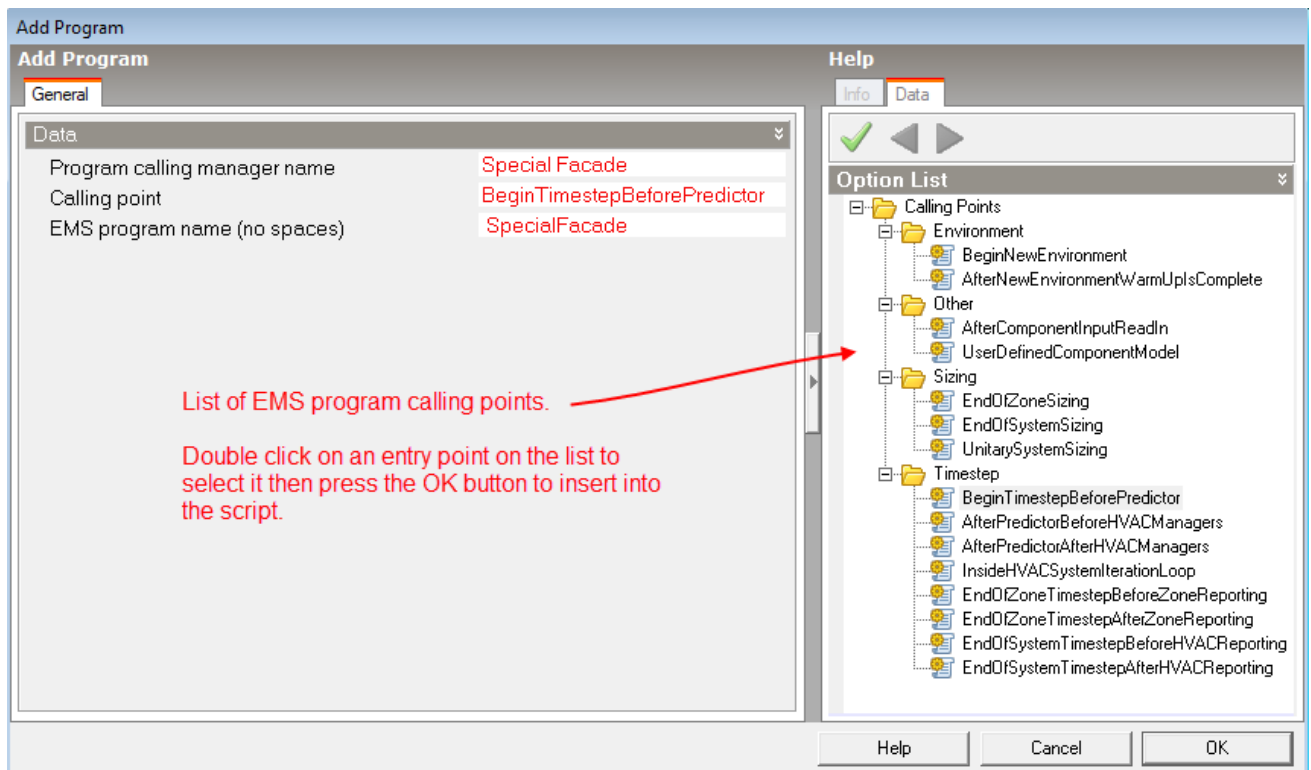
- 1-Use **DesignBuilder output options** which means that the selected output reports will be generated for the [intervals](#) selected on the Simulation calculation options dialog.
- 2-Timestep,
- 3-Hourly,
- 4-Daily,
- 5-Monthly,
- 6-Run period, totals and averages over the whole simulation period.

See also:

"How to Generate and View Extra EnergyPlus Outputs" on page 2205

Add Program

The **Add Program** dialog is accessed by clicking on the **Add program** link in the Info panel of the [Edit Script](#) dialog when editing an EMS program. With the **Add Program** dialog open you can select from a list of predefined program entry points.



For example, when the above settings are made the script below will be added:

```
EnergyManagementSystem:ProgramCallingManager,  
    Special Facade,  
    BeginTimestepBeforePredictor,  
    SpecialFacade;  
  
EnergyManagementSystem:Program,  
    SpecialFacade,  
    ! add program code  
    ;
```

You should add you own EMS logic at the point of the "Add program code" comment .

Program calling manager name

Enter the name of the program calling manager (see above).

Calling point

Select the one of the program entry points listed in the right hand data panel by either a) double-clicking on the required entry point , or b) by single-clicking on the required entry point and then clicking on the Select



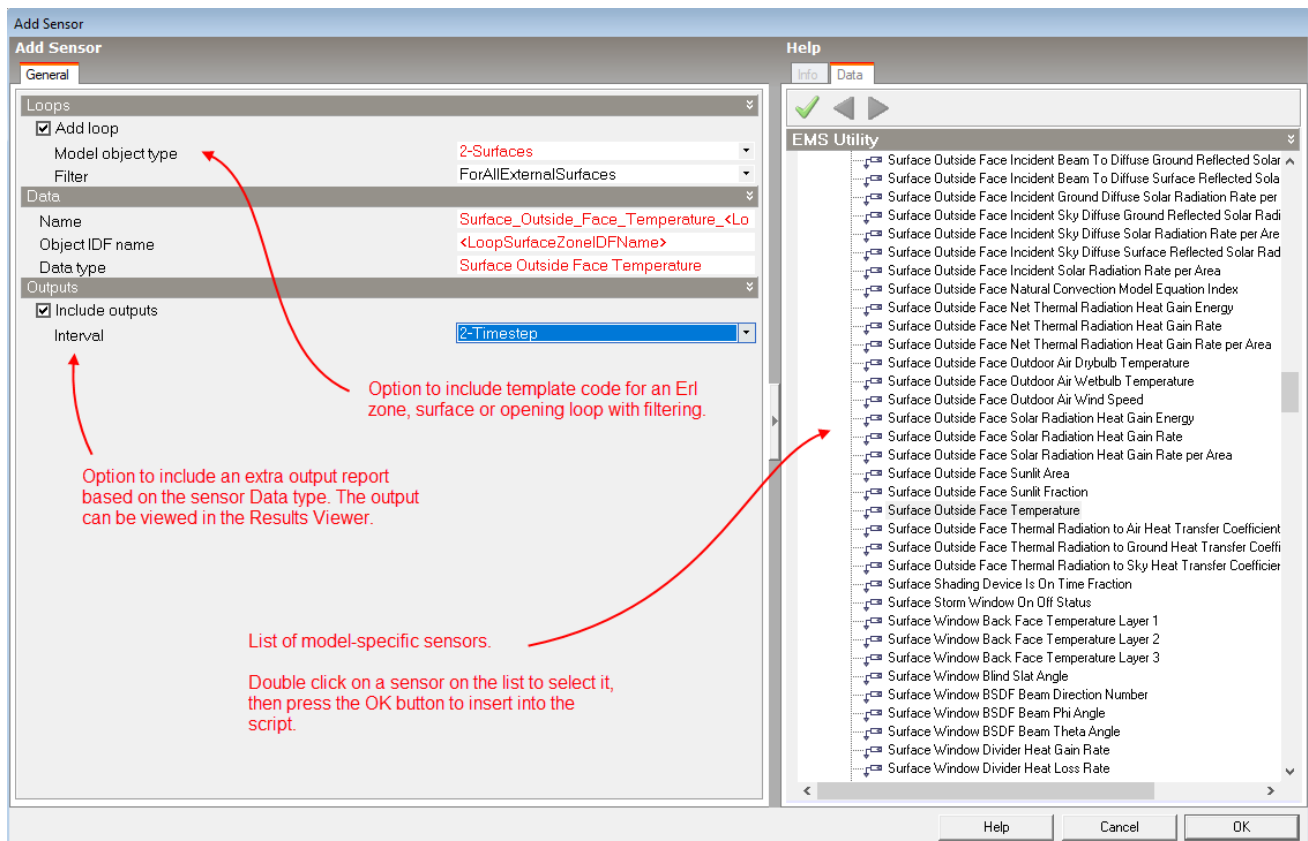
icon. When you press the OK button Program code is inserted into the script at the cursor position.

EMS program name

Enter the name of the EMS program (see above). The program name must follow Erl guidelines and in particular must not contain any spaces.

Add Sensor

The **Add Sensor** dialog is accessed by clicking on the **Add sensor** link in the Info panel of the [Edit Script](#) dialog when editing an EMS program. With the **Add Sensor** dialog open you can select from a list of sensors based on the actual model.



At the point the dialog is opened DesignBuilder asks you if you would like to update the list of available sensors by automatically running a preliminary simulation to generate an edd file with a list of all available sensors for the current model. If you agree to this there will be a delay (of several minutes for larger models) while EnergyPlus is run to generate the .edd and .rdd files.

Select the one of the sensors listed in the right hand data panel by either a) double-clicking on the required sensor, or b) by single-clicking on the required sensor and then clicking on the Select  icon.

When you press the OK button Erl code for the sensor is inserted into the script at the cursor position.

Loops

You can get DesignBuilder to set up a DB Erl loop to include script to generate multiple sensors by checking the **Add loop** checkbox. This allows you to make further 2 selections to define the type of loop.

Model object type

There are 3 different DB Erl Loop types. Select from:

1-Zones where the loop is for all zones in the building, with optional filters.

2-Surfaces where the loop is for all surfaces in the building, with optional filters.

3-Windows where the loop is for all windows in the building, with optional filters.

Note: The **Model object type** defines the type of loop and the type of sensor selected must be compatible. In particular for zone loops, only zone sensors should be selected. For surface loops only surface or zone sensors should be selected and for window loops only window or zone sensors should be selected. DesignBuilder checks for incompatible combinations of Model object type and sensor selected and generates an error where appropriate.

Loop keyword

The list of loop keywords presented will depend on the **Model object type** selected as shown below.

1-Zones

When the Model object type selected is **1-Zones**, then the list of available loop filter keywords is:

ForAllZones, no filter is applied, i.e. all zones in the building will be listed.

ForAllOccupiedZones, only occupied zones will be listed. See below for how occupancy is assessed.

ForAllUnoccupiedZones, only unoccupied zones will be listed. See below for how occupancy is assessed.

ForAllOccupiedCooledZones, only occupied zones that are also cooled will be listed. See below for how occupancy and cooling are assessed.

ForAllOccupiedHeatedZones, only occupied zones that are also heated will be listed. See below for how occupancy and heating are assessed.

ForAllOccupiedNonHeatedZones, only occupied zones that are also not heated will be listed. See below for how occupancy and heating are assessed.

ForAllOccupiedNonCooledZones only occupied zones that are also not cooled will be listed. See below for how occupancy and cooling are assessed.

Zone occupancy is defined by the zones' [Occupancy density](#), [Floor area per person](#) or [Number of people](#) setting (depending on which [Occupancy method](#) is selected). If there are people in the space and also if the [Zone type](#) is **1-Standard** then the zone is considered to be occupied.

Cooling is defined by the zones' [Cooled](#) setting on the HVAC model data tab.

Heating is defined by the zones' [Heated](#) setting on the HVAC model data tab.

2-Surfaces

When the Model object type selected is **2-Surfaces**, then the list of available loop filter keywords is:

ForAllSurfaces, no filter is applied, i.e. all surfaces in the building will be listed.

ForAllExternalSurfaces, where all external surfaces in the building will be listed.

ForAllInternalSurfaces, where all internal surfaces in the building will be listed.

3-Windows

When the Model object type selected is **3-Windows**, then the list of available loop filter keywords is:

ForAllWindows, no filter is applied, i.e. all windows in the building will be listed.

ForAllExternalWindows, where all external windows in the building will be listed.

ForAllElectrochromicSensorWindows, all windows used as SageGlass sensor windows will be listed.

ForAllShadedWindows, all windows in the building having any sort of "Window shading" on page 439 will be listed.

ForAllSlattedBlindsShadedWindows, all windows in the building having slatted blind "Window shading" on page 439 will be listed.

ForAllDiffusingBlindsShadedWindows, all windows in the building having diffusing "Window shading" on page 439 will be listed.

ForAllElectrochromaticSwitchableShadedWindows, all windows in the building having electrochromic switchable "Window shading" on page 439 will be listed.

ForAllTransparentInsulationShadedWindows, all windows in the building having transparent insulation "Window shading" on page 439 will be listed.

ForAllMultipleStateElectrochromicShadedWindows all windows in the building having multiple state electrochromic (SageGlass) "Window shading" on page 439 will be listed.

Data

Name

Enter the name of the sensor to be used in Erl script to reference the current sensor value.

When a sensor is selected from the list, the sensor **Name** is automatically set to be the name of the sensor but with all illegal characters such as spaces replaced with "_".

Object IDF name

Enter a specific object name such as a zone or surface IDF name, or enter "*" to indicate all relevant objects.

When the [Add loop](#) option is selected and an appropriate sensor is selected from the list, the sensor **Object IDF name** is automatically set to be one of:

<LoopZoneIDFName> when the loop [Model object type](#) is set to **1-Zones** and a "Zone" sensor has been selected

<LoopSurfaceIDFName> when the loop [Model object type](#) is set to **2-Surfaces** and a "Surface" sensor has been selected

<LoopWindowIDFName> when the loop [Model object type](#) is set to **3-Windows** and a "Window" sensor has been selected

<LoopSurfaceZoneIDFName> when the loop [Model object type](#) is set to **2-Surfaces** and a "Zone" sensor has been selected. This can be useful when a surface sensor needs to be associated with data for its zone.

<LoopWindowZoneIDFName> when the loop [Model object type](#) is set to **3-Windows** and a "Zone" sensor has been selected. This can be useful when a window sensor needs to be associated with data for its zone.

Data type

When a sensor is selected from the list, the sensor **Data type** field is automatically set. This is the same as the name of the sensor listed and also the name of the EnergyPlus output report used to feed the sensor with its data.

Outputs

Include outputs

Sensors are based on EnergyPlus outputs so if you would like the output selected for the sensor to also be included in the .eso output file then check this checkbox.

Note: Extra outputs added in this way must be viewed in the [Results Viewer](#) and are not generally available for view in the main DesignBuilder results screen.

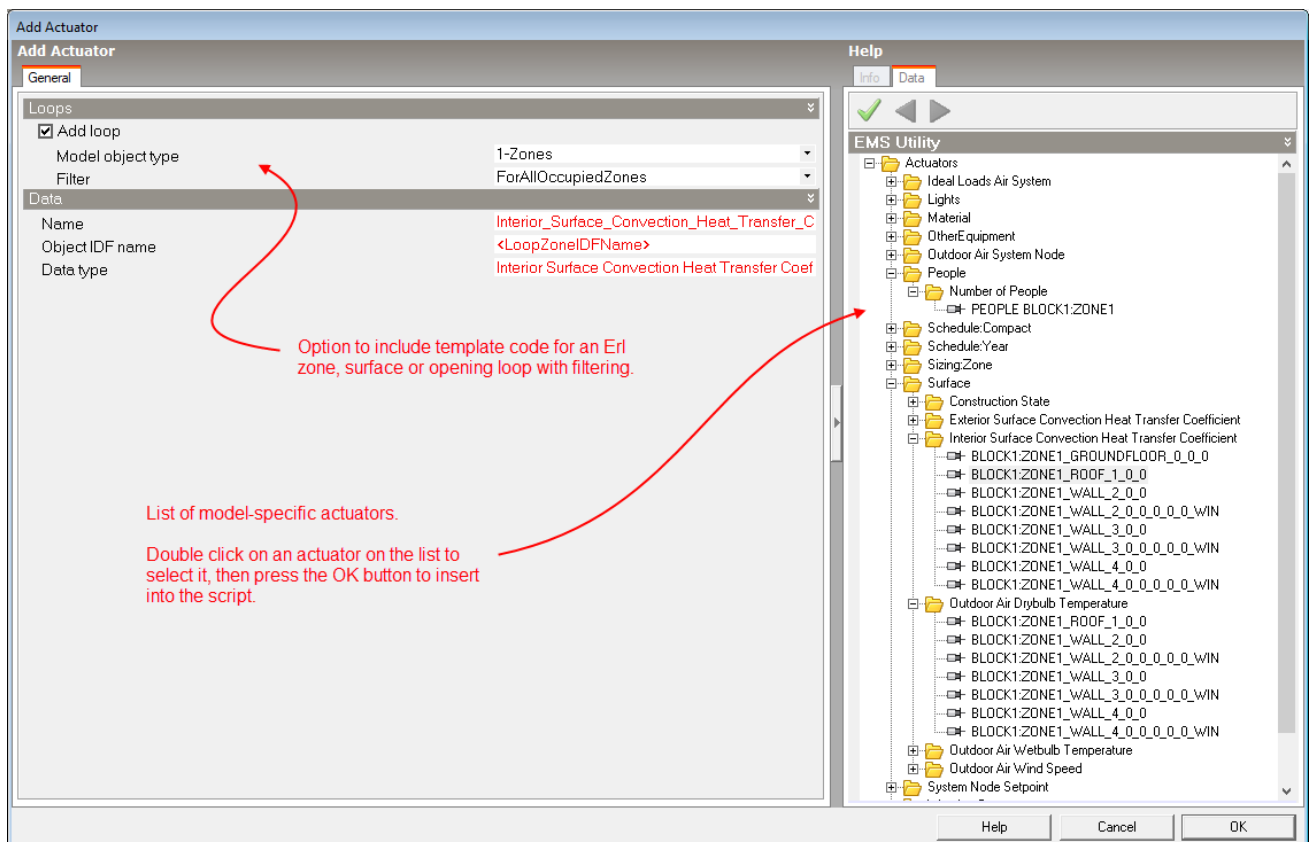
Interval

Select the reporting interval for the selects outputs from the list:

- 1-Use DesignBuilder output options** which means that the selected output reports will be generated for the [intervals](#) selected on the Simulation calculation options dialog.
- 2-Timestep,**
- 3-Hourly,**
- 4-Daily,**
- 5-Monthly,**
- 6-Run period,** totals and averages over the whole simulation period.

Add Actuator

The **Add Actuator** dialog is accessed by clicking on the **Add actuator** link in the Info panel of the [Edit Script](#) dialog when editing an EMS script. With the **Add Actuator** dialog open you can select from a list of actuators based on the actual model.



At the point the dialog is opened DesignBuilder asks you if you would like to update the list of available actuators by automatically running a preliminary simulation to generate an edd file with a list of all available

actuators for the current model. If you agree to this there will be a delay (of several minutes for larger models) while EnergyPlus is run to generate the .edd and .rdd files.

Select the one of the actuators listed in the right hand data panel by either a) double-clicking on the required actuator, or b) by single-clicking on the required actuator and then clicking on the Select  icon.

When you press the OK button Erl code for the actuator is inserted into the script at the cursor position.

Loops

You can get DesignBuilder to set up a DB Erl loop to include script to generate multiple actuators by checking the **Add loop** checkbox. For more information about setting up DB Erl loops see the [Add Sensor](#) dialog help.

Data

Name

Enter the name of the actuator to be used in Erl script.

When an actuator is selected from the list, the actuator **Name** is automatically set to be the name of the actuator, but with all illegal characters such as spaces replaced with "_". If the [Add loop](#) option is activated then one of the following DB Erl keywords is appended to the name, preceded by the "_" character:

```
<LoopZoneVariableName>,  
<LoopSurfaceVariableName>,  
<LoopWindowVariableName>.
```

Object IDF name

Enter a specific object name such as a zone or surface IDF name, or enter "*" to indicate all relevant objects.

When the [Add loop](#) option is selected and an appropriate actuator is selected from the list, the actuator **Object IDF name** is automatically set to be one of:

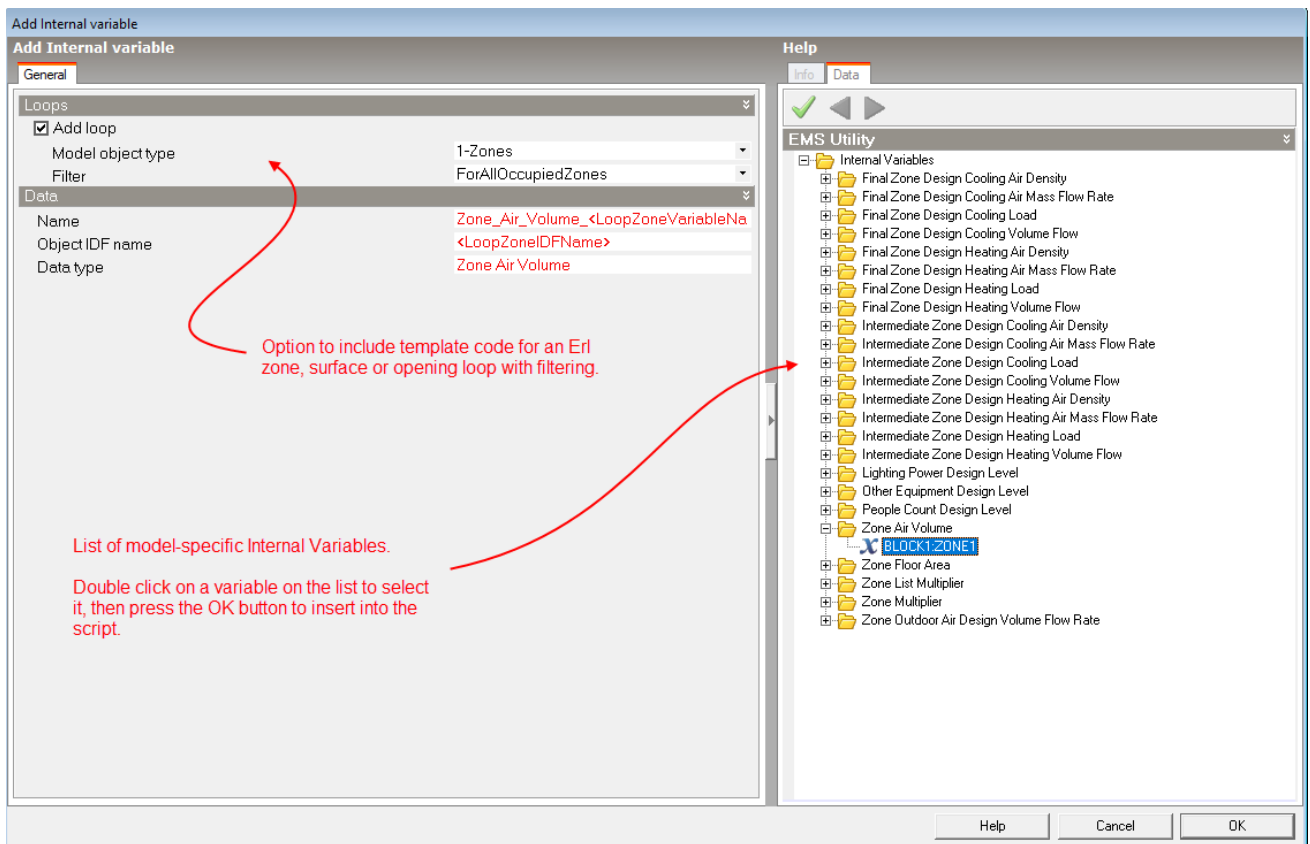
```
<LoopZoneIDFName> when the loop Model object type is set to 1-Zones.  
<LoopSurfaceIDFName> when the loop Model object type is set to 2-Surfaces.  
<LoopWindowIDFName> when the loop Model object type is set to 3-Windows.
```

Data type

When an actuator is selected from the list, the actuator **Data type** field is automatically set. This is the same as the name of the actuator listed.

Add Internal Variable

The **Add Internal Variable** dialog is accessed by clicking on the **Add Internal variable** link in the Info panel of the [Edit Script](#) dialog when editing an EMS script. With the **Add Internal Variable** dialog open you can select from a list of Internal variable based on the actual model.



At the point the dialog is opened DesignBuilder asks you if you would like to update the list of available sensors by automatically running a preliminary simulation to generate an edd file with a list of all available sensors for the current model. If you agree to this there will be a delay (of several minutes for larger models) while EnergyPlus is run to generate the .edd and .rdd files.

Select the one of the Internal Variables listed in the right hand data panel by either a) double-clicking on the required internal variable, or b) by single-clicking on the required internal variable and then clicking on the

Select  icon.

When you press the OK button Erl code for the internal variable is inserted into the script at the cursor position.

Loops

You can get DesignBuilder to set up a DB Erl loop to include script to generate multiple internal variables by checking the **Add loop** checkbox. For more information about setting up DB Erl loops see the [Add Sensor](#) dialog help.

Data

Name

Enter the name of the internal variable to be used in Erl script.

When an internal variable is selected from the list, the internal variable **Name** is automatically set to be the name of the internal variable, but with all illegal characters such as spaces replaced with "_". If the [Add loop](#) option is activated then one of the following DB Erl keywords is appended to the name, preceded by the "_" character:

```
<LoopZoneVariableName>,
<LoopSurfaceVariableName>,
<LoopWindowVariableName>.
```


Object IDF name

Enter a specific object name such as a zone or surface IDF name, or enter "" to indicate all relevant objects.

When the [Add loop](#) option is selected and an appropriate internal variable is selected from the list, the internal variable **Object IDF name** is automatically set to be one of:

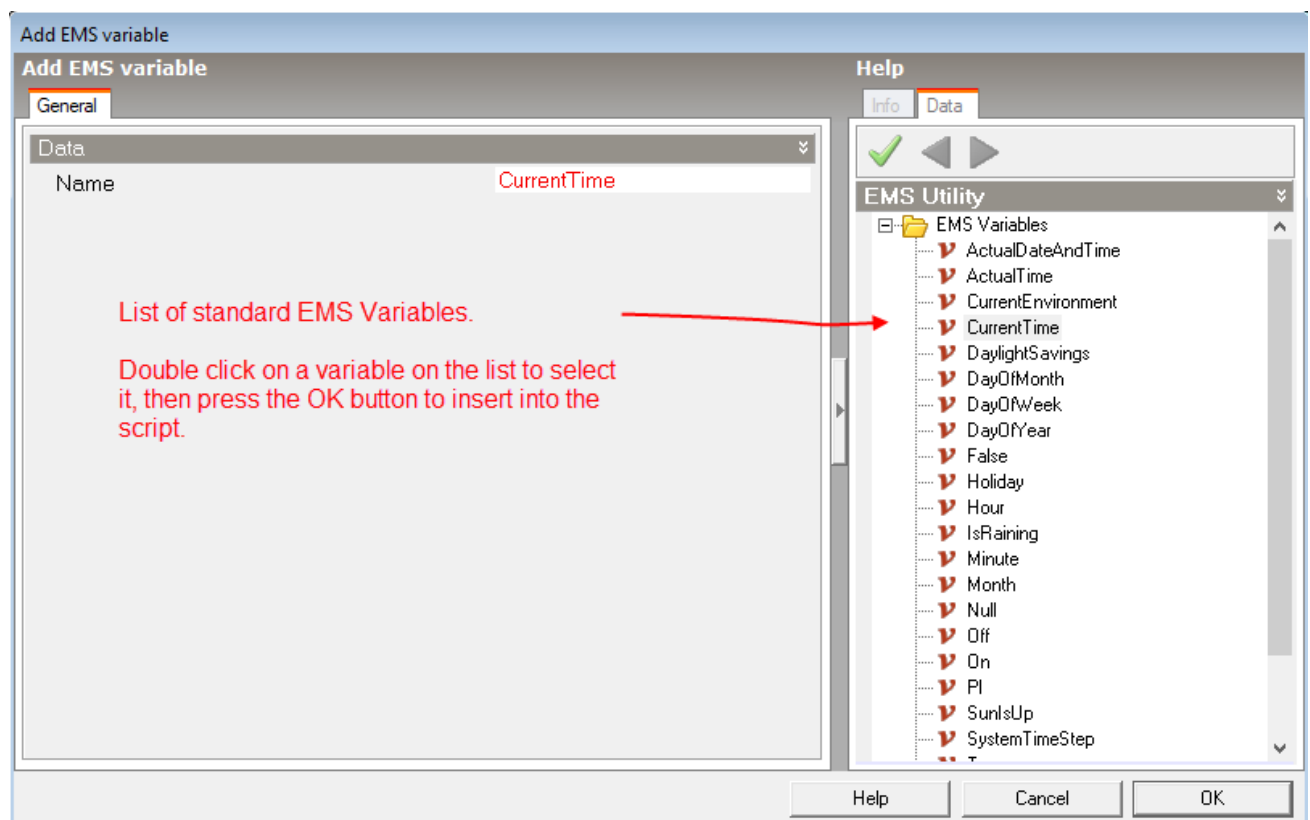
- <LoopZoneIDFName> when the loop [Model object type](#) is set to **1-Zones**.
- <LoopSurfaceIDFName> when the loop [Model object type](#) is set to **2-Surfaces**.
- <LoopWindowIDFName> when the loop [Model object type](#) is set to **3-Windows**.

Data type

When an internal variable is selected from the list, the internal variable **Data type** field is automatically set. This is the same as the name of the internal variable listed.

Add EMS Variable

The **Add EMS Variable** dialog is accessed by clicking on the **Add EMS variable** link in the Info panel of the [Edit Script](#) dialog when editing an EMS script. With the **Add EMS Variable** dialog open you can select from a fixed list of standard EMS variables.



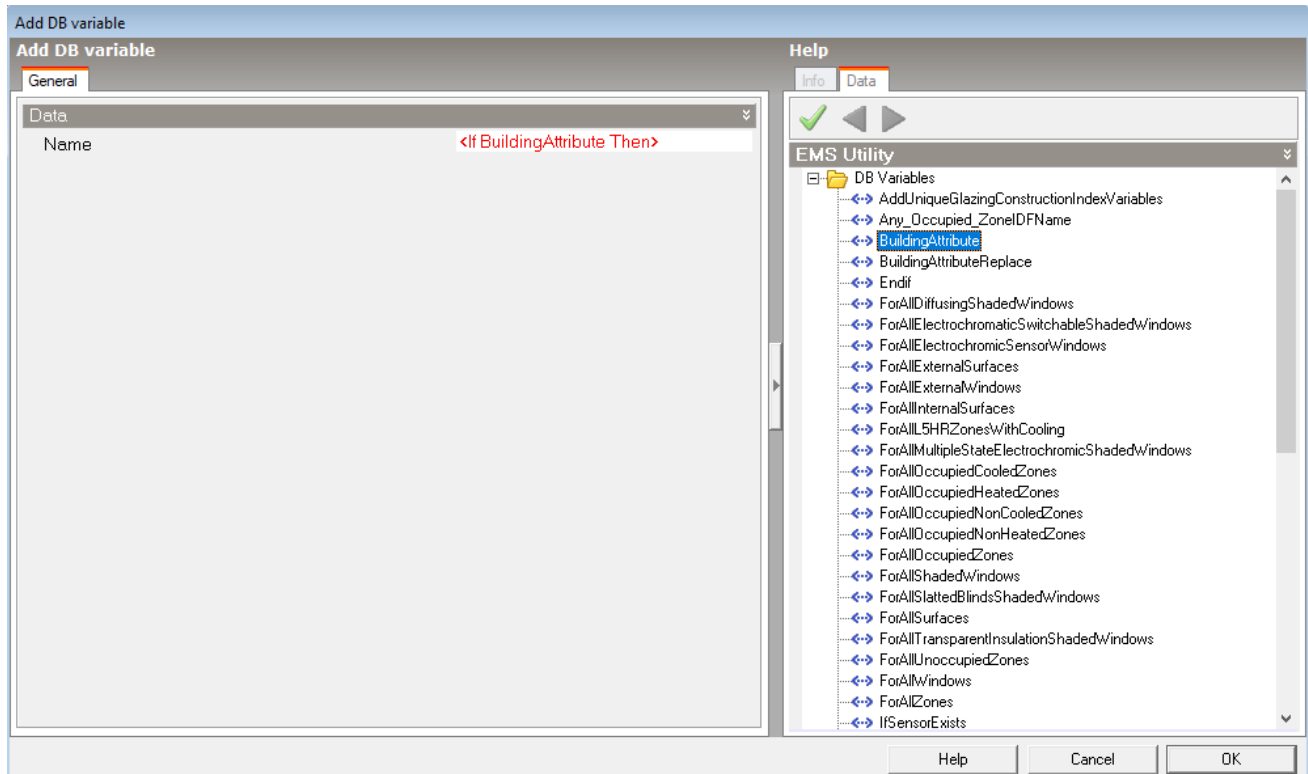
Select the one of the EMS variables listed in the right hand data panel by either a) double-clicking on the required EMS variable, or b) by single-clicking on the required EMS variable and then clicking on the  icon.

When you press the OK button Erl code is inserted into the script at the cursor position.

Add DB Variable

The **Add DB Variable** dialog is accessed by clicking on the **Add DB variable** link in the Info panel of the [Edit Script](#) dialog when editing an EMS script. With the **Add DB Variable** dialog open you can add code for a [DesignBuilder Erl extension](#) keyword by selecting from a fixed list of DB variables.

Note: DB "Variables" are more than variables in the strict meaning of the word as understood by programmers. They are actually a broad set of keywords that can be used as part of [DesignBuilder Erl code](#) for a range of purposes to create loops, access model attributes, add Erl code depending on model settings, including parametric variations etc.



Select the one of the DB variables listed in the right hand data panel by either a) double-clicking on the required variable, or b) by single-clicking on the required variable and then clicking on the Select  icon.

When you press the OK button the code is inserted into the script at the cursor position.

DesignBuilder Erl

DesignBuilder provides access to the full EnergyPlus Runtime Language (Erl) which is described in detail in the [EMS application guide](#). You can enter Erl script to modify your simulations on the [EMS Program](#) dialog.

This page describes a powerful set of extensions to Erl which allow code to be made portable between models. Loops, If logic statements and access to model attributes offer ways to reference the model data without referring to zones, surfaces and windows by name in the Erl scripts.

The combined Erl plus DB extensions language is referred to as **DB Erl** in the documentation.

Note: Erl is case sensitive.

Loops

In models having more than one zone it is usually more convenient to write code using loop statements. It is possible to loop through all zones, surfaces and openings in a model. It is also possible to work with only particular zones, surfaces and openings.

Zone Loops

These have the general format:

```
<ForAllZones>,
IF <LoopZoneVariableName>_DeltaT < 0 ,
SET <LoopZoneVariableName>_DeltaT = 0,
ENDIF,
<LoopNextZone>,
```

When code is written in this way it becomes easier to manage, especially in models having more than a handful of zones or where the code is to be made portable for use in any model.

A summary of all the zone loop-related statements is given in the table below.

DB Erl statement	Description
Loops	
<ForAllZones>	Loop through all zones in the current building.
<ForAllOccupiedZones>	Loop through all occupied zones in the current building. A zone is considered to be occupied if the Occupancy density is > 0 and the Zone type is 1-Standard .
<ForAllUnoccupiedZones>	Loop through all unoccupied zones in the current building.
<ForAllOccupiedCooledZones>	Loop through all occupied zones in the current building that have the Cooled checkbox on the HVAC tab checked.
<ForAllOccupiedNonCooledZones>	Loop through all zones in the current building that do not have the Cooled checkbox on the HVAC tab checked.
<ForAllOccupiedHeatedZones>	Loop through all occupied zones in the current building that have the Heated checkbox on the HVAC tab checked.
<ForAllOccupiedNonHeatedZones>	Loop through all occupied zones in the current building that do not have the Heated checkbox on the HVAC tab checked.
<LoopNextZone>	Signals the end of the zone loop.
In-Loop Place Holders	
<LoopZoneIDFName>	Place holder is replaced with the IDF zone name of the current zone in the loop. E.g. Block1:Zone1
<LoopZoneVariableName>	Place holder is replaced with a version of the the IDF zone name which has been modified to be a legal Erl variable name (":", "." and "?" replaced by "_"). E.g. Block1:Zone1 becomes Block1_Zone1
<LoopZoneDataFromAttribute AttributeName FieldName>	Looks up the component or template stored in AttributeName for the current loop and replaces the place holder with the value of the FieldName field.
<LoopZoneAttribute AttributeName>	Looks up the value of AttributeName and uses it instead of the place holder. E.g. <LoopZoneAttribute ZoneMultiplier> might be replaced with 1 or <LoopZoneAttribute OccupancySchedule> might be replaced with Office_Open_off. See full list of valid attribute names.
Boolean	
<If LoopZoneAttribute a < b Then>	Boolean expression . E.g. <If LoopZoneAttribute MechanicalVentilationRateType = 2 Then>
<If ZoneLoopValue a < b Then>	Boolean expression . Using a value rather than an attribute. Often used in conjunction with <LoopZoneDataFromAttribute AttributeName FieldName>. e.g. <If ZoneLoopValue <LoopZoneAttribute ActivityTemplate AllGainsValue> > 10 Then>

Other	
<Any_Occupied_ZoneIDFName>	Replace <Any_Occupied_ZoneIDFName> with the IDF zone name of any occupied zone in the model. (Actually, the first occupied zone it comes across while looping through.)

Surface Loops

A summary of all the surface loop-related statements is given in the table below.

Loops	
<ForAllSurfaces>	Loop through all surfaces in the current building.
<ForAllExternalSurfaces>	Loop through all external surfaces in the current building.
<ForAllInternalSurfaces>	Loop through all internal surfaces in the current building.
<LoopNextSurface>	Signals the end of the surface loop.
In-Loop Place Holders	
<LoopSurfaceIDFName>	Place holder is replaced with the IDF surface name of the current surface in the loop. E.g. Block1:Zone1_Wall_2_0_0
<LoopSurfaceVariableName>	Place holder is replaced with a version of the the IDF surface name which has been modified to be a legal Erl variable name (":", ".", and "?" replaced by "_"). E.g. Block1:Zone1_Wall_2_0_0 will become Block1_Zone1_Wall_2_0_0
<LoopSurfaceAttributeName>	Looks up the value of AttributeName and uses it instead of the place holder. E.g. <LoopSurfaceAttribute SillHeight> might be replaced with 0.8
<LoopSurfaceDataFromAttribute AttributeName FieldName>	Looks up the object stored in AttributeName for the current loop and replaces the place holder with the value of FieldName. E.g. <LoopSurfaceAttribute WallConstr NumberLayers> might be replaced with 4
<LoopSurfaceZoneIDFName>	Place holder is replaced with the IDF name of the zone that the current surface is in. E.g. Block1:Zone1_Wall_2_0_0 is in Block1:Zone1
<LoopSurfaceZoneVariableName>	Place holder is replaced with the Erl variable name of the zone that the current surface is in. E.g. Block1:Zone1_Wall_2_0_0 is in Block1:Zone1, which becomes Block1_Zone1
Boolean	
<If LoopSurfaceAttribute a < b Then>	Boolean expression . E.g. <If LoopSurfaceAttribute FixedExternalWallThickness = 1 Then> (Checkboxes: 1 means checked, 0 means unchecked)
<If SurfaceLoopValue a < b Then>	Boolean expression . Using a value rather than an attribute. Often used in conjunction with <LoopSurfaceDataFromAttribute AttributeName FieldName> E.g. <If SurfaceLoopValue <LoopSurfaceDataFromAttribute WallConstr NumberLayers> = 4 Then>

Window Loops

A summary of all the window loop-related statements is given in the table below.

DB Erl Statement	Description
Loops	
<ForAllWindows>	Loop through all windows in the current building.
<ForAllExternalWindows>	Loop through all external windows in the current building.
<ForAllShadedWindows>	Loop through all windows in the current building having the Window shading checkbox on the Openings tab checked.
<ForAllSlattedBlindsShadedWindows>	Loop through all windows in the current building having the Window shading checkbox on the Openings tab checked and whose selected Window shading system is from the Slatted blinds category.

DB Erl Statement	Description
<ForAllDiffusingShadedWindows>	Loop through all windows in the current building having the Window shading checkbox on the Openings tab checked and whose selected Window shading system is from the Diffusing shades category.
<ForAllElectrochromaticSwitchableShadedWindows>	Loop through all windows in the current building having the Window shading checkbox on the Openings tab checked and whose selected Window shading system is from the "Window blinds - Electrochromic Switching" on page 1453 category.
<ForAllTransparentInsulationShadedWindows>	Loop through all windows with internal shading of type Transparent Insulation in the current building.
<ForAllMultipleStateElectrochromicShadedWindows>	Loop through all windows with internal shading of type SageGlass Electrochromic in the current building.
<LoopNextWindow>	End of the window loop.
In-Loop Place Holders	
<LoopWindowIDFName>	Place holder is replaced with the IDF window name of the current window in the loop. E.g. Block1:Zone1_Wall_2_0_0
<LoopWindowVariableName>	Place holder is replaced with a version of the IDF window name which has been modified to be a legal Erl variable name (":", ".", and "?" replaced by "_"). Block1:Zone1_Wall_2_0_0 would become Block1_Zone1_Wall_2_0_0
<LoopWindowAttribute AttributeName>	Looks up the value of window AttributeName and uses it instead of the place holder. E.g. <LoopWindowAttribute InternalBlindControlSolar> might be replaced with 120 or <LoopWindowAttribute RevealOutsideProjection> might be replaced with 0.5 if the Outside reveal depth for the window is 0.5m. See full list of valid attribute names.
<LoopWindowZoneAttribute AttributeName>	Looks up the value of the parent zone AttributeName attribute and uses it instead of the place holder. A typical application would be to access the zone "Occupancy Schedule" to obtain the availability schedule for "Follow Occupancy" operation), or "Activity Template" to filter windows using the parent zone activity.
<LoopWindowDataFromAttribute AttributeNameOfTable FieldName>	Looks up the component or template stored in the window AttributeName attribute for the current loop item and replaces the place holder with the value of FieldName. E.g. <LoopWindowAttribute GlazingType Layers> might be replaced with 2 if the glazing for the current window has 2 layers.
Boolean Expressions	
<If LoopWindowAttribute a < b Then>	Boolean expression . E.g. < If LoopWindowAttribute InternalBlind =1 Then> (Checkboxes: 1 means checked, 0 means unchecked)
<If WindowLoopValue a < b Then>	Boolean expression using a value rather than an attribute. Often used in conjunction with <LoopWindowDataFromAttribute AttributeName FieldName> E.g. <If WindowLoopValue <LoopWindowAttribute GlazingType Layers> >= 2 Then>

Optional Arguments

Optional arguments can be used with any loop type to further refine the included loops, filtering out data that is not required.

Optional Argument	What it does	Where it works
--------------------------	---------------------	-----------------------

Tag	Excludes any object that does not match the tag . Case sensitive.	All attribute levels.
MaxSlope	Excludes any surface or opening that has a slope value higher than this value.	External surfaces and openings.
MinSlope	Excludes any surface or opening that has a slope value less than this value.	External surfaces and openings.
MaxOrientation	Excludes any surface or opening that has an orientation value higher than this value.	External surfaces and openings.
MinOrientation	Excludes any surface or opening that has an orientation value less than this value.	External surfaces and openings.
ZoneIDFName	Excludes any zone, surface or opening that is in a zone that does not include this text anywhere in its name. Matching is case sensitive.	Zone, surface and opening.
ActivityTemplate	Excludes any objects that do not have the indicated Activity template loaded. The argument value must match exactly with the template but the matching is case insensitive.	Building, zone, surface and opening.
LightingTemplate	Excludes any objects that do not have the indicated Lighting template loaded. The argument value must match exactly with the template but the matching is case insensitive.	Building, zone, surface and opening.
HVACTemplate	Excludes any objects that do not have the indicated HVAC template loaded. The argument value must match exactly with the template but the matching is case insensitive.	Building, zone, surface and opening.
GlazingTemplate	Excludes any objects that do not have the indicated Glazing template loaded. The argument value must match exactly with the template but the matching is case insensitive.	Building, zone, surface and opening.
ConstructionTemplate	Excludes any objects that do not have the indicated Construction template loaded. The argument value must match exactly with the template but the matching is case insensitive.	Building, zone, surface and opening.

The format of an optional argument is: {argument=value}

Note: There must be at least a single space between ">" and "{".

If multiple alternative values are acceptable for an argument, the format is: {argument=value1//value2}. In this case the code is executed if the argument is either equal to value1 OR value2. See examples below.

Multiple arguments can be included, if they are separated by ",".

Examples

< ForAllExternalSurfaces> {MinSlope=89;MaxSlope=91}	to loop through external vertical walls only.
<ForAllWindows> {ZoneIDFName=Floor1}	to loop through windows in zones in block "floor 1".
<ForAllOccupiedZones> {ActivityTemplate=Domestic Bedroom}	to loop through occupied bedrooms.
<ForAllOccupiedZones> {ActivityTemplate=Domestic Bedroom//Domestic Bathroom}	to loop through occupied bedrooms and bathrooms.
<ForAllZones> {LightingTemplate=LED; ZoneIDFName=Floor1//Floor2}	to loop through all zones with LED lights on floor 1 and floor 2

Note: While loops can be nested, it is only possible to use conditional arguments in the outermost loop statement.

Boolean Expressions - Include/exclude Blocks Of Code

When the DB Erl pre-processor encounters an **<If...>** directive, followed eventually by an **<Endif>** directive, it will include the code between the **<If...>** and the **<Endif>** only if the boolean expression following **<If ...>** evaluates to True. Otherwise the block of code is not included in the model IDF file.

DB Erl statement	Description
<If Boolean expression Then>	Where Boolean expression is an expression evaluating to True or False. E.g. <pre><If BuildingAttribute HourlyOutput = 1 Then> Erl statements for hourly output <Endif></pre> Boolean expression must be in the form: SiteAttribute, BuildingAttribute, LoopZoneAttribute or LoopWindowAttribute followed by , Attribute name followed by , an operator which must be one of "=", "<", ">", "<=", ">=", followed by , Test value. Note for numbered list items, only the number part of should be tested for. e.g.: This is correct: <If LoopZoneAttribute MechanicalVentilationRateType = 2 Then> , This is incorrect: <If LoopZoneAttribute MechanicalVentilationRateType = 2-Min Fresh Air (Per Person)
<Endif>	marks the end of the block of text to be included/excluded.

Parametric Variables

[Custom EMS](#) parametric variables can be introduced into the DB Erl script to provide varying settings based on a parametric analysis or optimisation study. The syntax is:

<Var, VarName, DefaultValue>	VarName is the corresponding Variable key used on the Variables dialog. DefaultValue is the value to be used in the script when it used in a standard, non-parametric simulation.
---	--

DB Erl Changes To Standard EnergyPlus Erl

DesignBuilder Erl is identical to the standard Erl language syntax with 1 exception:

for logical OR operations use "_OR_" instead of "OR". e.g. IF (c _OR_ d), instead of IF (c OR d).

CS-Script

With the C# scripting tools you have full access to the DesignBuilder API, model data attributes, data tables, results etc and you can write scripts that will be executed at key hook points in the program execution such as before and after E+ simulations. This can be useful for pre-processing the model, the IDF file or post-processing results to create custom variations in software behaviour.

Tip: The [Extensibility Tools User Guide](#) is the main source of help for scripts, plugin development and the API.

One of the most common reasons to use scripts is to create custom design variables or KPIs for optimisations and some examples of how to do this can be found in the Optimisation Tutorials section:

"Optimisation Tutorial - Custom Script Variable" on page 1745

"Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760

See also:

"EpNet Library" on page 2122

EpNet Library

EpNet is a .Net library which provides utilities for easy manipulation of EnergyPlus input and output files. It enables reading and writing of EnergyPlus idf input files which can be useful in a number of modelling scenarios such as implementation of [Custom IDF](#) optimisation variables and replacing one type of component with another to gain access to EnergyPlus components that DesignBuilder does not provide a user interface for.

EpNet also allows simulation results data to be read from eso results files and from csv table outputs with the minimum of code needing to be written.

EpNet is integrated in DesignBuilder Scripting module where scripts developed with EpNet are usually executed after idf generation and before simulation.

Tip: You can find full documentation on how to use the EpNet library with code examples at: <https://epnet.ecoeficiente.es>

EpNet library is developed and maintained by [Germán Campos at Aurea Consulting](#).

Python Scripting

Python scripting is used in a very similar way to the [CS-Script](#) capability. Python tends to be easier than C# to work with, especially for non-professional software developers, and is a popular language for more adventurous energy modellers needing to customise their workflow.

With the Python scripting tools you have full access to the DesignBuilder API, model data attributes, data tables, results etc and you can write scripts that will be executed at key hook points in the program execution such as before and after E+ simulations. This can be useful for pre-processing the model, the IDF file or post-processing results to create custom variations in software behaviour.

Tip: The [Extensibility Tools User Guide](#) is the main source of help for scripts, plugin development and the API.

One of the most common reasons to use scripts is to create custom design variables or KPIs for optimisations and some examples of how to do this can be found in the Optimisation Tutorials section:

"Optimisation Tutorial - Custom Script Variable" on page 1745

"Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760

"Python Scripting Examples" on page 2124

Python Scripting Libraries

DesignBuilder comes with 2 useful Python libraries built in:

```
db_eplusout_reader
eppy
```

You can learn more about how to use **eppy** and **db_eplusout_reader** to generate custom output reports and custom KPIs in the "Python Scripting Examples" on page 2124 section.

Db_eplusout_reader

EnergyPlus stores its main outputs results in ESO and SQL files and the **db_eplusout_reader** library provided with DesignBuilder allows the [SQL output type](#) to be read and processed. It simplifies the process of exploring and processing the SQLite output of EnergyPlus simulations and can be especially useful for generating custom reports and for implementing [custom KPIs](#) for parametric analysis, optimisation and UA/SA studies.

Also the SQL output processing methods are available.

The latest Python **db_eplusout_reader** libraries can be downloaded from:

<https://github.com/DesignBuilderSoftware/db-eplusout-reader>

Eppy

[eppy](#) is a scripting language specifically developed to process EnergyPlus IDF input files and output files. It is written in Python and takes advantage of the rich data structure and idioms that are available in Python. You can programmatically navigate, search, and modify EnergyPlus idf files using eppy, for example:

- Make a large number of changes in an IDF file with a few lines of eppy code.
- Use conditions and filters when making changes to an IDF file
- Read data from the summary output files of a EnergyPlus simulation run.

Here are some more specific examples of what you can do with eppy:

- Change construction for all north-facing walls.
- Change the glass type for all windows larger than 2 square meters.
- Change the number of people in all the interior zones.
- Change the efficiency and fan power of all rooftop units.
- Find the energy use of all the models in a folder (or of models that were run after a certain date)

Adding Custom Libraries

You can access other 3rd party libraries from your DesignBuilder Python scripts by adding the required library files to the [Python scripting folder](#).

Some important points to note are:

- DesignBuilder Python is based on Python v2.7 and so may not support libraries created for more recent versions of Python.
- All custom package dependencies must be included with the library.
- Pandas** and **Numpy** libraries are not supported at the moment.

We do not recommend using the [esoreader](#) library because it loads the entire ESO file data set into an array and so is inefficient with memory and so will tend to cause out of memory crashes.

Python Scripting Examples

The following tutorials use Python scripts to customise simulation and/or optimisation results.

"Python Scripting Example 1 - db_eplusout_reader Basics" on page 2124

"Optimisation Tutorial - Custom Script Variable" on page 1745

"Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760

"Optimisation Tutorial - Custom Script KPI Example 1 - db_eplusout_reader Python Libraries" on page 1752

Python Scripting Example 1 - Db_eplusout_reader Basics

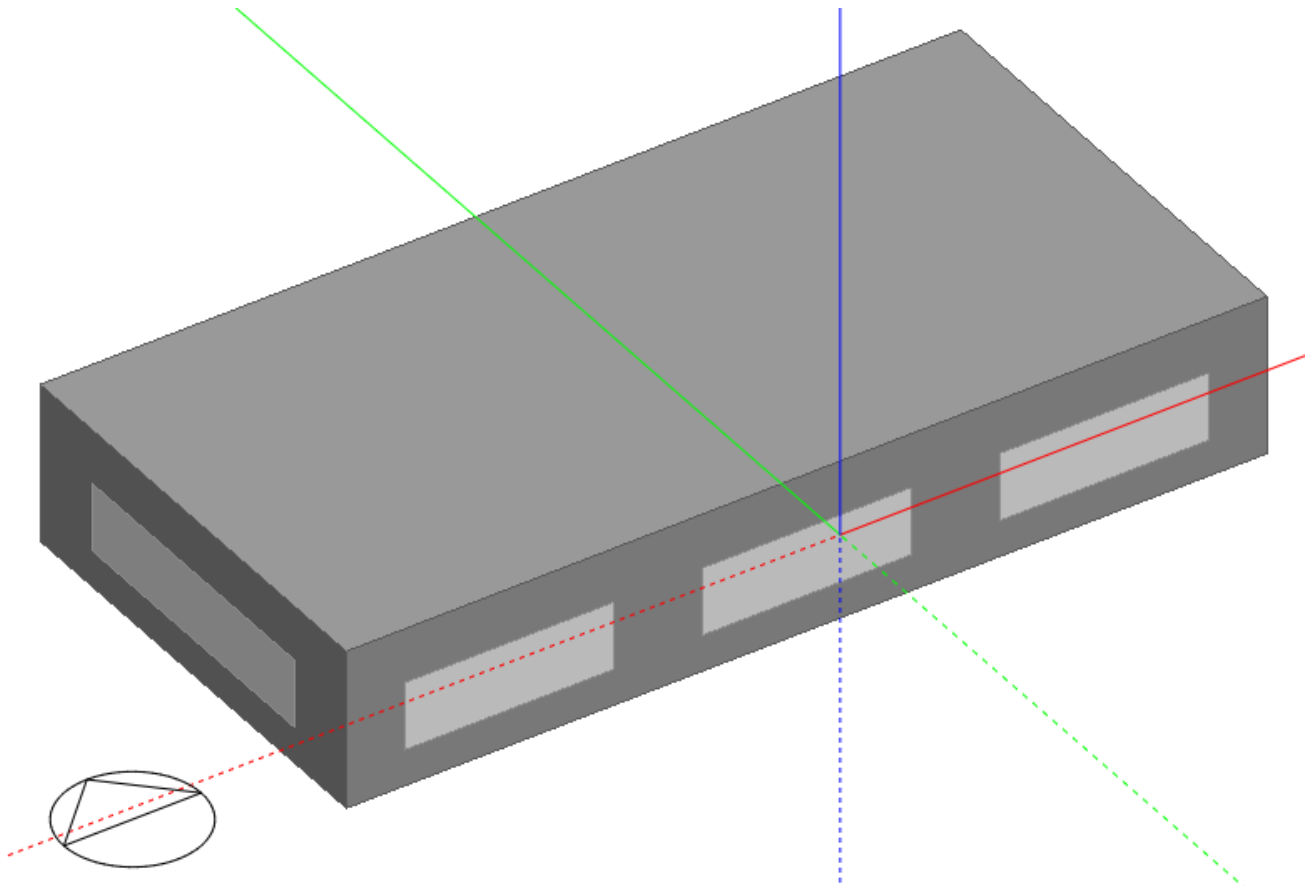
This example shows how to create a simple Python script to write out a basic energy summary report after each simulation by accessing output variables reported in the EnergyPlus SQLite output file. The script could of course be extended to write more sophisticated reports depending on your model and your requirements.

The model is the same base model used in several other custom script tutorials.

The steps involved are as follows.

Step 1 - Create A Base Model

1. Create a new model located at London Gatwick.
2. Add a building using default settings and draw a rectangular block measuring 20m by 10m with block height of 3.5m as shown below.



3. Load the **Generic Office Area** Activity template at building level on the Activity Tab.

Step 2 - Write Custom Script

The next task is to write the script that will read the simulation results from the SQL output file and process and write them out as a summary report file. Steps are as follows.

1. Click on the Scripts  toolbar icon to open the Script Manager dialog and click on the **Enable scripts** checkbox.
2. Click on the Script browse item. A list of the existing scripts appears in the Info panel to the right.
3. Click on the **Python-Script** category and press the **Add new item** Info toolbar icon to open the Edit script dialog.
4. Copy and paste the script below into the script window. In summary, the script works as follows. Before the IDF is generated for the simulation the SQLite output option in each building in the model is forced to be on. Then after the simulation the key heating, cooling, equipment and lighting energy consumption outputs are loaded from the eplusout.sql simulation output file, summed to create a total and written to a text file in the DesignBuilder [EnergyPlus folder](#).

Python example: db_eplusout_reader basics

```
from db_eplusout_reader import Variable, get_results
from db_eplusout_reader.constants import *
from datetime import datetime

# ensure that SQLite outputs are generated
def before_energy_idf_generation():
    site = api_environment.Site
    for building in site.Buildings:
        building.SetAttribute("SSSQLiteOP", "1")

def after_energy_simulation():
```


Support - Getting Help

If you need information that you could not find in this help file you should first check the [Knowledgebase](#) on the Support Desk website.

If this does not give you the information you need you can send us a support request using a [Support Request Form](#) and DesignBuilder team will endeavour to answer your question by email.

See also:

[Online Tutorials](#)

Modelling Guides And Tutorials

This page brings together in one place all of the various modelling guides and tutorials that are available in the program help, arranged by sections.

Modelling Guides

[DesignBuilder LEED and ASHRAE 90.1 Appendix G User Guide](#) - the main source of help for ASHRAE 90.1 analysis.

[Modelling Guide for Unmet Load Hours](#) - helps diagnose and resolve Unmet Load Hours issues through systematic analysis of common causes.

[LEED MEPC report generation](#) - explains how to generate LEED MEPC reports from your ASHRAE 90.1 PRM models.

[DesignBuilder Revit – gbXML Tutorial](#) - explains how to import your Revit models into DesignBuilder.

[DesignBuilder SageGlass Modelling Guide](#) - explains how to model SageGlass Electrochromic glazing systems.

[DX-Assisted Ventilation Unit Modelling Guide](#) - describes how to model a DX-assisted mechanical ventilation system with heat recovery using Detailed HVAC.

Help Tutorials

General Modelling

"Warehouse with Office" on page 2171

HVAC

"Ground Source Heat Pump (GSHP) Example" on page 875

"Air To Water Heat Pump Example" on page 893

[CO2 Control and DCV Modelling Example](#)

Natural Ventilation and Mixed Mode

"Mixed Mode Tutorial" on page 499

Renewable Systems

"How to Model Photovoltaic (PV) Systems" on page 523

[How to Model Wind Turbine Systems](#)

Ground Modelling

"Ground Domain Slab Tutorial" on page 393

CFD

- "Example - CFD External Analysis" on page 1949
- "Example - CFD Internal Analysis Basics" on page 1955
- "Example - CFD Internal Analysis Including Assemblies" on page 1959

Parametric Analysis

- "Parametric Analysis - Basic Tutorial" on page 1679
- "Insights Tutorial - Basic Parametric Analysis " on page 2090
- "Parametric Analysis Tutorial - Future Climate Study" on page 1684
- "Parametric Analysis Tutorial - Custom Script KPI - TM59 Comfort" on page 1690
- "Parametric Analysis Tutorial - Custom Script Variable - Advanced Parametric Facade" on page 1698

Optimisation and Scripting

- "Optimisation Basic Tutorial" on page 1736
- "Insights Tutorial - Basic Optimisation" on page 2095
- "Optimisation Tutorial - Custom EMS Variable" on page 1739
- "Optimisation Tutorial - Custom Script Variable" on page 1745
- "Optimisation Tutorial - Custom Script KPI Example 1 - db_eplusout_reader Python Libraries" on page 1752
- "Optimisation Tutorial - Custom Script KPI Example 2 - Reading the EUI KPI Using EpNet (C#) and eppy (Python)" on page 1760
- "Optimisation Tutorial - Custom Script KPI Example 3 - Using EMS + Python Scripts to Optimise Controls for Comfort and Heating" on page 1770
- "Optimisation Tutorial - Custom Script KPI Example 4 - Adding Python Script for TM52 Comfort to Example 3" on page 1778

Uncertainty and Sensitivity Analysis

- "Uncertainty and Sensitivity Analysis - Basic Tutorial" on page 1868
- "Uncertainty and Sensitivity Analysis - Advanced Tutorial" on page 1878

Outputs and Analysis

- "How to Generate and View Extra EnergyPlus Outputs" on page 2205
- "Insights Tutorial - Basic Optimisation" on page 2095
- "Insights Tutorial - Basic Parametric Analysis " on page 2090

Economics

- "Tariff Examples" on page 565

Weather Data

- "Add New hourly weather data" on page 1420

Video Tutorials

DesignBuilder video tutorials are provided on demand to all registered DesignBuilder users free of charge. They mostly last about 10-20 minutes.

Tip: Most people learn most effectively by "doing" rather than "watching" so please do make sure to try out the techniques shown in the tutorials while they are fresh in your mind before moving on to the next tutorial!

Introductory Tutorials

The Getting Started tutorials provide an overview of how to create DesignBuilder models and run simulations.

- 000 [Introduction to Tutorials: Tutorial concept and freely available resources](#)
- 001 [DesignBuilder User Interface – Modelling: Opening screen, loading a model, screen layout, navigation](#)
- 002 [DesignBuilder User Interface – Model Data: Site level, model data tabs, data hierarchy and inheritance](#)
- 003 [DesignBuilder User Interface – Simulation Screens: Overview of visualisation and calculation screens](#)

Basic Geometry

These tutorials cover the key areas needed to get started, including how to quickly create and edit building geometry, automated adjacency assignment, importing/exporting models from/to other tools, and creating thermal zones using partitions.

- 101 [Starting a new project](#)
- 102 [Blocks and dimensioning protocols](#)
- 103 [Adding blocks and using the measure tool](#)
- 104 [Shading using standard component blocks](#)
- 105 [Setting adjacency conditions using ground and adiabatic blocks](#)
- 106 [Selecting, moving, cloning and deleting objects](#)
- 107 [Stretch, drag face and rotate tools](#)
- 108 [Add surface, increment snaps](#)
- 109 [Adding a roof, construction lines](#)
- 110 [Outline blocks, Boolean operations](#)
- 111 [Cutting blocks, protractor tool](#)
- 112 [Drawing arcs, circles and domes](#)
- 113 [Importing drawing files](#)
- 114 [Partitioning blocks to create zones](#)
- 115 [Drawing holes and merging zones](#)

Basic Model Data

The Basic Model Data tutorials show how to set up data on the Activity, Constructions, Openings, Lighting and HVAC model data tabs for EnergyPlus simulations.

- 201 [Model data hierarchy and inheritance](#)
- 202 [Program and Model Options](#)
- 203 [Data templates](#)
- 204 [Site level data and weather files](#)
- 205 [Activity data templates and time schedules](#)
- 206 [Construction data](#)
- 207 [Create and edit constructions](#)
- 208 [Glazing and openings](#)
- 209 [Drawing windows and doors](#)
- 210 [Creating new glazing types](#)
- 211 [Shading via louvres, overhangs, and side fins](#)
- 212 [Lighting data and daylight controls](#)
- 213 [Simple HVAC Data](#)
- 214 [Timings, schedules and profiles](#)
- 215 [Compact schedules](#)
- 216 [Hourly weather data](#)

Calculations

The Calculations tutorials explain how to set up DesignBuilder EnergyPlus simulations and analyse results.

- 301 [Model Data Grid View & QA checking](#)
- 302 [Heating Design Calculations](#)
- 303 [Cooling Design Calculations](#)
- 304 [Setting up simulations](#)
- 305 [Analysing results in DesignBuilder's interface](#)
- 306 [Simulation Manager and Results viewer](#)

Detailed HVAC

- 401 [Introduction to Detailed HVAC](#)
- 402 [HVAC zone groups and components](#)
- 403 [Adding loops and connecting components](#)
- 404 [Editing HVAC loop and component properties](#)
- 405 [Chilled water loops chillers and fan coil units](#)
- 406 [Condenser loops cooling tower](#)
- 407 [Adding air loops and AHUs](#)
- 408 [Editing Generic air loop and system components](#)
- 409 [Running Detailed HVAC simulations](#)
- 410 [Radiant surfaces](#)

Natural Ventilation

- 501 [Scheduled natural ventilation and infiltration](#)
- 502 [Calculated natural ventilation](#)
- 503 [Mixed mode ventilation](#)

Daylighting

- 601 [Introduction to Daylighting](#)

CFD

- [Introduction to External CFD analysis](#)

Other Video Learning Resources

- [Demo of DesignBuilder Simulation capabilities](#)
- [Getting started tutorials](#)
- DesignBuilder [Webinars](#)

Note: These Tutorials and Webinars are also available on the Training section of the DesignBuilder website and on the [DesignBuilder YouTube Channel](#)

Frequently Asked Questions

An up-to-date Knowledgebase is available in the <http://www.designbuilder.co.uk/content/view/5/48/Support> section of the DesignBuilder web site.

Modelling Advice

Modelling advice topics:

[Working with large models](#)
[ASHRAE 90.1 Modelling](#)
[Natural ventilation modelling](#)
[DCV Modelling](#)
[Ground modelling](#)
[Adjacent Buildings](#)
[Electrochromic Glazing](#)
[Trombe Walls](#)
[Double Skin Facades](#)
[Autosizing Heating and Cooling Equipment](#)
[Daylight Saving and Local Time](#)
[Solar Radiation Distribution](#)
[Warehouse with Office](#)
[Startup and Pre-conditioning](#)

Working With Large Models And Speeding Up Simulations

DesignBuilder has been used to successfully model and simulate very large models including buildings with over 2000 zones and huge floor areas. This topic provides advice on how best to go about working with large models and how to reduce EnergyPlus simulation times.

[Creating and Editing Large Models](#)
[Modelling Strategy to Reduce EnergyPlus Simulations Times](#)
[Modelling Strategy for Successful Radiance Simulations in Large Models](#)

Creating And Editing Large Models

You can speed up modelling, and in particular the drawing and deleting of partitions in large models, by switching off [Automatic block zoning](#) option. In this case, after having manually rezoned the block you can also save time when renaming the new zones in the Navigator by:

switching off the [Double-click to navigate site hierarchy](#) Program option and,
 switching off the [Automatically refresh site navigator lists](#) Program option.

Another way to speed up modelling in large models is to zoom into the area of the building or block where you wish to work. This eliminates consideration of objects that are not visible on the screen and allows the screen to refresh more quickly.

While it is possible to create very large, complex building models using DesignBuilder, you should pause to consider what you're trying to achieve by your modelling exercise before diving in and including every detail of the building. Otherwise you may find you have created a beautiful model which is impractical because of long simulation times.

Modelling Strategy To Reduce EnergyPlus Simulations Times

You should consider the following points when setting up models for EnergyPlus simulations.

EnergyPlus simulations are slowed down by:

- Larger numbers of windows
- Zones with many windows and other surfaces (large number of surfaces / zone)
- [Radiant heating systems](#)
- Calculated natural ventilation using the Airflow Network
- Calculating [solar reflections](#)
- Reading hourly or sub-hourly results for extended simulation periods
- Not enough [computer memory](#) (RAM)
- Large volumes of output data
- Large values of [timesteps](#) per hour



You can speed up simulations in a number of ways:

In early stage modelling it is sometimes helpful to reduce the number of windows in the model by increasing the [Window spacing](#). Shading issues aside, modelling a small number of large Windows has the same effect as a large number of small Windows provided you get the frame area right. Reduce the number of surfaces by avoiding unnecessary "wiggles" and indents in block perimeters and partitions - keep it as simple as possible. The larger the model the more important it is to follow this advice. Where an indent in the perimeter gives shading then you could instead model this using a single flat surface without indent and use local shading devices with sidefins to model local window shading.

Avoid very large zones with many surfaces and windows. Contrary to intuition, from the simulation time point of view, it is best to split such zones into more zones with fewer surfaces each.

Use the [Lump similar windows on surface](#) option to speed up simulations with many similar windows per surface. Note that switching this option on has no effect for zones using [Daylight lighting control](#) or if the Full interior and exterior [solar distribution option](#) is selected.

Wherever possible [lump similar adjacent zones](#) together. Only create zones to model areas of the building with specific environmental conditions, and HVAC systems or internal gains schedules. A common mistake made by the beginners is to model each room as a separate zone. This is often not necessary for building energy simulations because many rooms will have reasonably similar operating conditions. In this case you can use one of the DesignBuilder [Merging](#) features to reduce the number of zones.

If there are multiple identical zones then you can use the [Zone multiplier](#) to reduce the number of zones processed.

Use [Simple](#) HVAC descriptions.

Use the [Scheduled](#) Natural ventilation option.

Switching off calculation of [solar reflections](#) in the Model options dialog

If you are interested in analysing a single zone within a much larger model, you could consider carrying out simulations on a single representative or worst case zone, excluding consideration of the rest of the building. You can exclude unnecessary zones by unchecking the [Include zone in thermal calculations](#) option.

Fit as much memory (RAM) to your computer as possible (8 GB or more ideally). Paging memory to disk during simulations of large models can cause EnergyPlus to grind to a halt.

Uncheck the [Allow custom outputs](#) option to speed output variable processing. This change can drastically reduce simulations times in large models. See also the point below.

Only request the output you really need. This will speed up simulation initialisation and reading of output (especially hourly and sub-hourly) at the end. If you are only looking at hourly results then you can get noticeably faster initialisation and simulations by switching off monthly and daily results. For example, simulation times for a test winter week simulation on a real model were:

- With hourly, daily, monthly outputs: **39 min**
- Only hourly outputs: **14 min**
- Only hourly outputs and checking 'Internal surface as adiabatic': **12 min**

If you don't need surface data (Walls, Windows etc. heat transfer) switch the [Surface heat transfer incl solar](#) output option off.

Restricting hourly and sub-hourly results to short simulations for investigation of detailed building operation. Note that if you request large volumes of hourly or sub-hourly data this can result in a program crash when results are being loaded into DesignBuilder. To avoid this you should switch off

[automatic loading](#) of EnergyPlus results to DesignBuilder and view results instead using the [Results Viewer](#).

It is sometimes possible to use less [timesteps](#) per hour to speed up simulations. Some models can give reliable results using 2 timesteps per hour, however when Detailed HVAC or Calculated natural ventilation systems are being modelled, 6, 10 or more timesteps may be required. If you aren't sure, compare the hourly results you obtain when using a range of timesteps per hour values and choose the minimum value that gives results equivalent to the 10 timesteps case.

In some cases, [optimisation](#) studies can be simplified by first running a [sensitivity analysis](#) to [identify and remove less influential variables](#) from the optimisation study. This helps to reduce the complexity and overall optimisation runtime.

If all else fails then you can model very large buildings by splitting them into smaller units and summing results outside DesignBuilder. Note that we haven't seen a need for this since the early days of EnergyPlus development.

Modelling Strategy For Successful Radiance Simulations In Large Models

When running Annual Daylighting calculations, DesignBuilder requires a significant free disk space on the C:\ drive for temporary files. To successfully run large models you should ensure there is at least 50 GB of free space.

Wherever possible, you should avoid attempting to simulate very large zones with many surfaces and windows. Attempting to simulate very large zones can cause the generic **An internal error has occurred. Daylighting simulation has been aborted** error message to be displayed.

If you find DesignBuilder runs out of memory in very large models, you can run the calculations block by block to reduce the memory requirements of each simulation. Note that simulation results are additive in that, previous results are not deleted unless they are overwritten by subsequent simulations. Very large zones can be split up using virtual partitions which allow light to be transferred through them without losses.

If you must simulate very large zones, you should use the largest grid size possible allowed by the calculation methodology you are following.

ASHRAE 90.1 Appendix G PRM Reference

This section outlines the procedures involved in setting up proposed and baseline building models in accordance with ASHRAE 90.1 Appendix G PRM. In the text below "Appendix G" refers to ASHRAE 90.1 Standard Appendix G.

The main sources of documentation on using DesignBuilder for ASHRAE 90.1 Appendix G PRM work are:

[LEED and ASHRAE 90.1 Appendix G User Guide](#)
[Modelling Guide for Unmet Load Hours](#)
[LEED MEPC Report Generation](#)

Note: you will need the DesignBuilder [LEED / ASHRAE 90.1 module](#) to access this functionality.

Add new site

To access ASHRAE 90.1 functionality the [Mandatory energy code](#) must be set to ASHRAE 90.1 2007, 2010, 2013 or 2016 This can be done manually for existing models from the Regions tab at site level, or for new models, follow the instructions in the next step.

When creating a new ASHRAE 90.1 project check the **ASHRAE 90.1 App G PRM** checkbox on the Add new site dialog and choose one of the **1-ASHRAE 90.1-2007** or **2-ASHRAE 90.1-2010**, **3-ASHRAE 90.1-2013** or **4-ASHRAE 90.1-2016** options.

DesignBuilder uses the [ASHRAE climate zone](#) site level setting to identify the climate zone for generating baseline constructions and glazing according to ASHRAE 90.1 building envelope requirements. The *default* climate zone loaded will have been loaded from the Locations template and that value was originally derived from the value shown on the [Hourly weather data](#) dialog which in turn

was derived by analysing the .epw hourly weather data. In some cases the climate zone may not be the same as defined in ASHRAE 90.1 Standard Appendix D for the model location. In this case simply select the correct [ASHRAE climate zone](#) at site level and the baseline constructions and glazing will be automatically updated to reflect the new climate zone.

Add new building

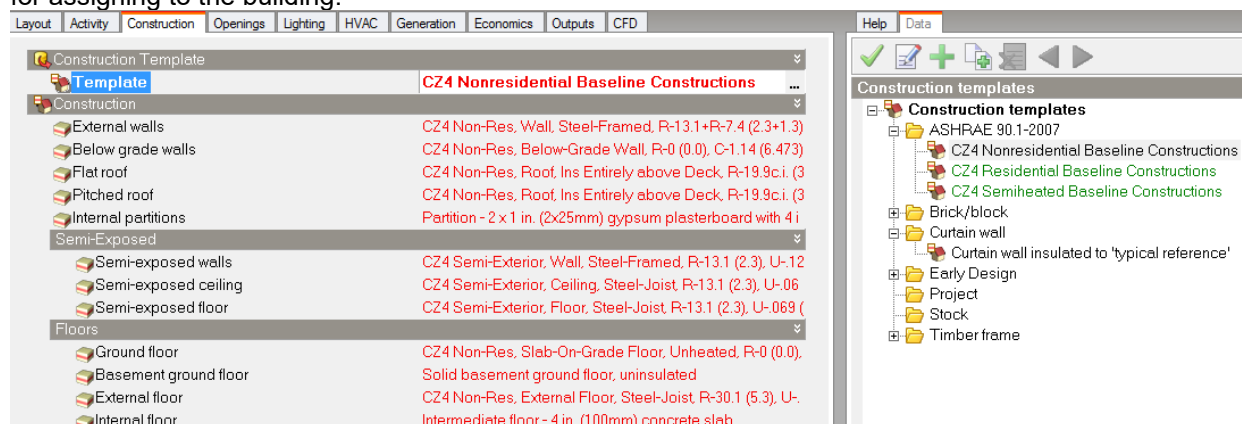
When the [Mandatory energy code](#) is one of the ASHRAE 90.1 standards then on the [Add new building](#) dialog the **Model type** template shows **ASHRAE 90.1**. This loads some appropriate model data settings such as Detailed HVAC and 6 timesteps per hour to the building model.

On the [Add new building](#) dialog you can select either **1-Proposed** or **2-Baseline** for the ASHRAE 90.1 building type. DesignBuilder does not currently generate different settings based on this selection, however it always generates baseline constructions and construction templates in accordance to building envelope requirements defined in 90.1 Constructions from Standard chapter 5 (detailed in Tables 5.5-1 through 5.5-8) can be selected as required for both proposed and baseline buildings. For your proposed design you can improve the baseline construction (by copying and editing) or simply replacing them with definitions shown on architectural drawings or as built for existing building envelopes.

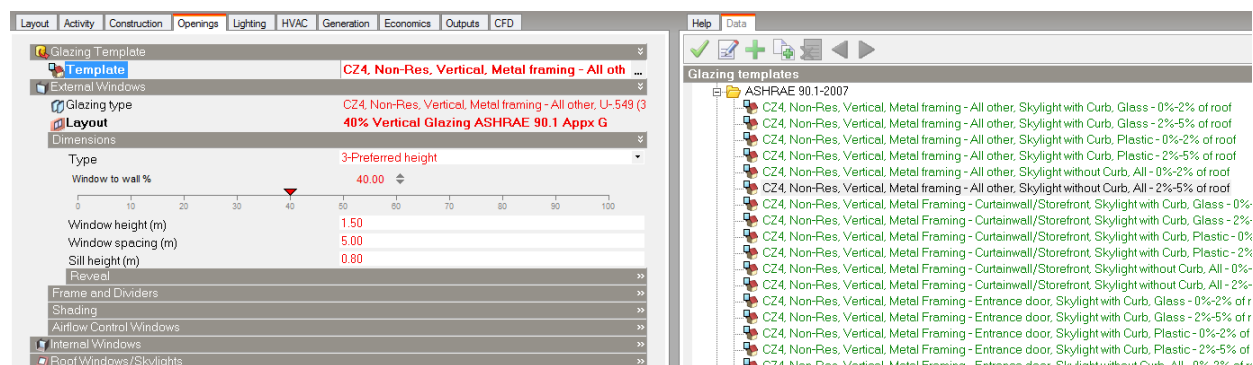
Following formal ASHRAE 90.1 PRM modelling procedure, you are strongly advised to create the proposed building first and to create the baseline building from that.

Constructions and Glazing

When a site is created or when the climate zone is changed, baseline constructions and glazing systems as well as construction templates are generated for the selected [ASHRAE climate zone](#) ready for assigning to the building.



Opaque construction templates



Glazing templates

Two special opaque construction types required by the ASHRAE 90.1 Standard, are F-factor ground floors and C-factor below-grade walls. These are different from conventional layered construction in that they don't use layers of materials but instead the overall thermal properties are defined as an F-factor (for slab on grade floors) or a C-factor (for below grade walls). EnergyPlus uses these factors to generate equivalent layered constructions behind the scenes. During the simulation these surfaces exchange heat with the [FCFactor ground temperatures](#) only.

When running Heating or Cooling design calculations, if the [FCfactorMethod ground temperatures source](#) data at site level has **1-Hourly weather file** selected, a weather file containing ground temperatures in the header should be selected and will be downloaded (if not yet in your weather file

folder on your computer) prior to the calculations. If **2-User defined** is selected, the Heating and Cooling design calculations do not require the weather file.

The terminology used in the ASHRAE 90.1 constructions list gives the climate zone, a description of the construction, insulated level(s) in IP units and the U-value in IP units and in brackets the values in SI units. For example construction **CZ4 Non-Res, Wall, Steel-Framed, R-13.1+R-7.4 (2.3+1.3), U-.064 (.365)** indicates a non-residential steel framed wall for climate zone 4 (CZ4) with internal insulation levels of two layers of minimum 13.1 and 7.4 ft²·°F/(Btu/hr) in IP units (2.3 and 1.3 m²·K/W in SI units) and a U-value of 0.064 Btu/hr·ft²·°F in IP units (0.365 W/m²·K in SI units).

Create 3-D building model

The proposed building model geometry should be set up with reference to ASHRAE 90.1 Standard Appendix G Table G3.1-1.

Define different thermal zone types using the [Zone type](#) option on the Activity tab. For ASHRAE 90.1 models additional Zone types can be selected per 90.1 requirements including:

5-Semiheated, for zones that are heated but not to normal comfort levels.

When the **2-Unconditioned Zone type** is selected choose whether the unconditioned zone is ventilated or unventilated using the [Unconditioned zone ventilation](#) setting below the Zone type setting.

Define each zone type as the model requires (checking with rules from Appendix G Table G3.1-7,8,9), and then assign relevant construction/fenestration templates according to their "Space-Conditioning Categories" (construction default is Non-Residential). Activity, lighting templates for each block/zone should be updated with relevant templates as well.

Geometry convention

It is important for ASHRAE 90.1 based-work that the proposed and baseline buildings have identical floor areas and zone volumes. The easiest way to ensure this is to use the [ASHRAE 90.1 Geometry convention](#) for both buildings. With this template the block geometry drawn gives the exact surface geometry and zone volumes to be used in the simulation. This means that when you copy the proposed building to create the baseline building the zone geometry will be correct even after loading the baseline constructions.

If instead you used the [External measurements](#) template for example you would find that the baseline building surface areas, floor area and zone volumes would be different after loading the baseline constructions because these generally have a different thickness to those selected for the proposed building.

Lighting

Lighting settings shall be either building area method or space-by-space method, but shall not the mix these two schemes. The Building Area Method is generally used for early stage analysis where the exact function of every zone in the building has not yet been determined. The Space-by-Space Method is used more often as it is more flexible, allowing the user to use more detail/specific lighting settings.

Note: When loading an activity template, it will update the lighting schedule.

HVAC

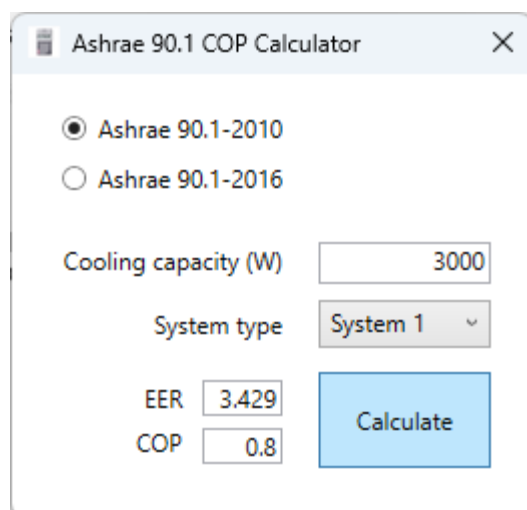
A set of Appendix G HVAC systems for loading to baseline building models is provided in the [Detailed HVAC templates](#) list. The proposed building HVAC system should meet the requirements shown in Appendix G Table G3.1-10. A modified copy of a baseline HVAC system can be used for the proposed building.

For the proposed building design, where no heating or cooling systems exist or have been specified, the heating or cooling systems shall be identical to the systems modelled in the baseline building design.

Define the DHW system based on Appendix G Table G3.1-11.

Baseline DX Cooling Coil Efficiency Input

From the ASHRAE 90.1 Standard, all HVAC equipment in the baseline building design shall be modelled at the minimum efficiency levels, both part load and full load. The efficiency rating that includes the supply fan energy shall be adjusted to remove the supply fan energy so that supply fan energy can be modelled separately. To help meet this requirement, for baseline systems that use DX cooling coils, i.e., systems #1 through #6, the COP input can be calculated using a simple ASHRAE COP calculation tool.



You can [download the COP calculation tool](#) from our website. Thanks to [Germán Campos from Ecoeficiente](#) for providing it!

Note: The calculation requires a sizing run as the cooling capacity is a mandatory input to the program.

Utility costs

Define the utility costs for the model to generate energy cost outputs based on purchased energy rates using settings on the [Tariff analysis](#) dialog on the Economics tab at building level. LEED/ASHRAE 90.1 App G PRM method may only require tariffs object to calculate operational costs.

Outputs

EnergyPlus provides a "LEED Summary" report which is displayed on the Summary tab after the simulation if the appropriate option is checked on the Output tab of the Simulation options dialog under the Summary Tables header.

Convert An Existing Simulation Model To An ASHRAE 90.1 Model

To convert a standard EnergyPlus model to be treated as a proposed building go to the Regions tab at site level and set the Mandatory energy code to one of the ASHRAE 90.1 energy codes. You will be warned to check that the ASHRAE climate zone is correctly set. The default climate zone can be overridden at site level under the [Location](#) header.

Unmet Hours

"Unmet hours" of a building are the summation of the number of hours when the heating or the cooling set point temperature of a zone is not met either by the HVAC system or by the plant. The data is provided in the Summary report as:

Time Setpoint Not Met During Occupied Heating

Time Setpoint Not Met During Occupied Cooling

Understanding/Interpreting/Calculating the number of unmet hours:

There will be an "unmet hour" for "Time Setpoint Not Met During Occupied Cooling" in a particular zone when the zone indoor temperature is higher than the cooling set point specified for that hour. Likewise for heating when the zone heating setpoint temperature is not met.

The number or the percentage of unmet hours in a building is usually given as one of the outputs of the simulation.

Zone wise, unmet hours can be read from the EnergyPlus: Output Variable, **Time Cooling Set point Not met**. When two zones are unmet at the same hour, this will count to one unmet hour for the building.

When two zones have unmet hours during different non overlapping times of a day, the total number of unmet hours in that day is the summation of these unmet hours of each zone. This total for the year should be considered as the total unmet hours of the building.

Example

When each zone is unmet in the specified hours as beside

Zone 1 unmet during hours: 6 8 14 16

Zone 2 unmet during hours: 6 8 12 16

Zone 3 unmet during hours: 7 8 12 13

Total number of unmet hours of the building: 7 hrs and not 12 hrs.

6 7 8 12 13 14 16

When percentage of unmet is specified, then this is the percentage of total number of hours (1 year- 8760 hours) for which the simulation is performed (not just the occupied hours).

As per ASHRAE 90.1-2004, the unmet load hours of the total building should be less than or equal to 300 hours and unmet load hours for the proposed design shall not exceed the number of load hours for the baseline building design by more than 50.

If unmet load hours in the proposed building exceed the unmet load hours in the baseline building by more than 50, then the size of equipment in the baseline building shall be reduced incrementally, until the condition is satisfied.

ASHRAE and USGBC say that it is OK if the baseline unmet load hours exceed the proposed unmet load hours by more than 50, provided they are both below 300.

The above information on Unmet hours is sourced from a post by Ashu Gupta (and subsequent posts) on the bldg-sim mailing list.

Diagnosing Excessive Unmet Hours

Possible causes of unmet load hours are as follows:

- Undersized HVAC Equipment.
- Improper temperature setpoints.
- Inaccurate internal load calculations.
- Incorrect zone sizing.
- Incorrect system sizing.
- Incorrect control of HVAC components (e.g. SPM type).
- Low performance envelope.
- Using weather data that does not represent typical conditions or does not account for climate variations.
- Incorrect modelling of Air Distribution Units.
- Improper AHU and zone OA / supply airflow rates.

To help identify and fix the issue, the following checks can be made:

- Check design temperature for each plant loop.
- Check the operational temperature in the plant loop's SPM.
- Check the operational temperature in the air loop's SPM.

Check airside economisers configuration.
Check HVAC operation schedules (e.g. make sure that zone thermostat schedules agree with schedules of HVAC system operation).
Check whether the zone is supplied with enough cold/warm airflow.
Check the weather file for hot/cold periods that are more extreme than the design day conditions.
Check whether the performance curves of an equipment (HP,VRF, coil) are reducing the available capacity.

Natural Ventilation Modelling

There are two general approaches to natural ventilation and infiltration modelling in DesignBuilder depending on the setting of the [Natural ventilation model option](#):

Scheduled in which the natural ventilation change rate is explicitly defined for each zone in terms of a maximum ACH value and a schedule and infiltration air change rate is defined by a constant ACH value. A range of control options is provided.

Calculated where natural ventilation and infiltration are calculated based on window openings, cracks, buoyancy and wind driven pressure differences crack dimensions etc. Control options are provided.

You can choose [Scheduled natural ventilation](#) if you are able to make a reasonable estimate of the [natural ventilation rates](#) and [infiltration rates](#) in the building. Other airflow scenarios are more difficult to estimate and in this case it may be necessary to calculate airflows.

Scheduled Natural Ventilation And Infiltration

Scheduled [Natural Ventilation](#) flow rate and schedule data is accessed on the HVAC tab. The flow is switched off if the zone air temperature falls below the [Ventilation setpoint temperature](#) set under [Environmental Control](#) on the [Activity](#) tab.

Infiltration is defined by a constant ACH value under [Airtightness](#) on the [Construction](#) tab.

Airflow through *Exterior* Windows, Vents and Doors is considered to be included in the Natural ventilation outside air ac/h value set on the HVAC tab.

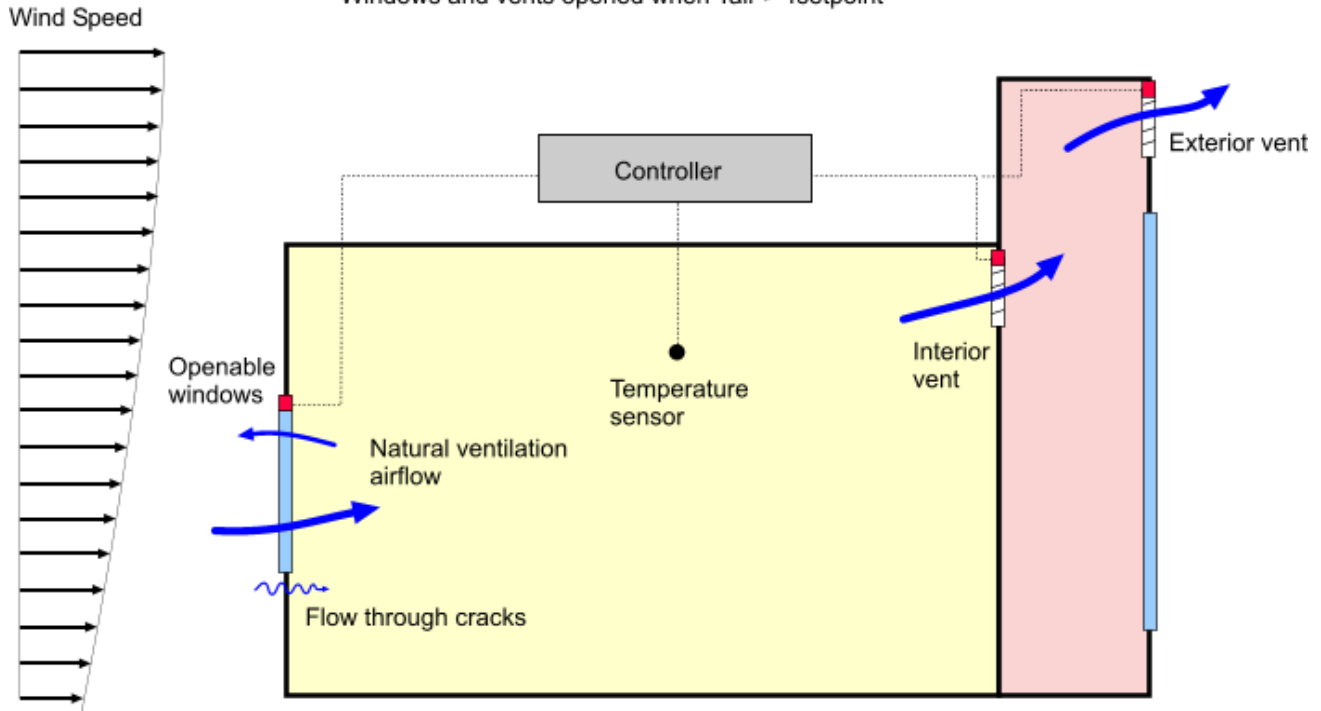
Airflows between zones can be defined using the [Interzone Airflow](#) dialog.

Also, airflow through *Interior* Windows, Vents and Doors can be modelled using the concept of mixing where equal amounts of air are transferred from one zone to another and vice-versa. The flow rate is directly proportional to the opening area using [Airflow rate per opening area](#) data accessed from the [Model options](#) dialog.

Calculated Natural Ventilation And Infiltration

Calculated Natural Ventilation Flow and Control

- * Multizone airflow model solved simultaneously with HVAC and building
- * Wind and Buoyancy pressure causes flow through openings and cracks
- * Windows and vents opened when $T_{air} > T_{setpoint}$



By default [Calculated natural ventilation](#) is controlled by timer (window and vent data on the [Operation header](#) on the Openings tab) and by [Ventilation setpoint temperature](#) on the Activity tab.

The [Crack data](#) under [Airtightness](#) on the Constructions tab is used in the EnergyPlus Airflow Network to calculate infiltration.

Note 1: Using the **Calculated** option increases the complexity of the model and slows simulations down. You should aim to [simplify the model](#) as appropriate to the requirements of the calculations. For example if you are carrying out a summertime simulation of a naturally ventilated building you could consider [switching off infiltration](#) modelling if infiltration heat flow is only a very small part of summertime heat balance.

Note 2: In Version 1 the calculated airflows through external windows, vents, doors, holes and cracks are all lumped into '[External air](#)' data on the Simulation results screen.

Calculations

The ventilation rate (q) through each opening and crack in the model is calculated based on the pressure difference using wind and stack pressure effects:

$$q = C.(DP)^n$$

Where:

q is the volumetric flow through the opening.

DP is the pressure difference across the opening/crack.

n is the flow exponent varying between 0.5 for fully turbulent flow and 1.0 for fully laminar flow.

C is the flow coefficient, related to the size of the opening/crack.

Wind-Driven Ventilation

When wind impinges on the surface of a rectangular building, a positive pressure is induced on the upwind face. The flow separates at the corners resulting in negative pressure regions on the side of the building and a negative pressure distribution on the leeward facade. The pressure distribution on the roof varies according to pitch - the pressure on the upwind face being negative for roof pitches of 30° and positive for steeper pitches.

The pressure on any point on the surface of a building facade can be represented by:

$$P_w = 0.5 \cdot \rho \cdot C_p \cdot v_z^2$$

Where:

P_w is the surface pressure due to wind,

ρ is the density of air,

C_p is the wind pressure coefficient at a given position on the surface and

v_z is the mean wind velocity at height z .

The wind pressure coefficient, C_p , is a function of wind direction, position on the building surface and side exposure. Some typical approximate values for buildings subjected to varying degrees of shelter and wind directions is given in an AIVC publication A guide to energy-efficient ventilation. This data is also quoted in the CIBSE A Guide. DesignBuilder uses this data to populate the [Pressure Coefficients templates](#) and provide default pressure coefficients suitable for use in basic design calculations for buildings having no more than three stories.

For more detailed analysis, or for buildings having 4 or more stories you should obtain specific pressure coefficient data from CFD analysis or from wind tunnel measurements and enter it for each surface under the [Pressure Coefficients](#) header on the [Openings tab](#).

Excluding Wind-Effects

When carrying out conservative design calculations you may prefer to exclude (or reduce) the effects of wind from Calculated natural ventilation. To exclude wind-driven airflow from the analysis altogether set the [Wind factor](#) to 0. For full treatment of wind effects set it to 1 and for intermediate treatment of wind set to a number between 0 and 1.

Limitations

DesignBuilder uses the EnergyPlus Airflow Network method to calculate air flow rates. You should be aware Airflow Network simulations can [take considerably longer](#) than the equivalent simulations with Scheduled natural ventilation.

Modulation

You can modulate the opening sizes to reduce the large temperature swings that can occur if the windows/doors are too far open when they are venting, especially when there is a large inside-outside temperature difference. Use the [Modulation settings](#) on the [Advanced tab](#) of the [Model Options dialog](#) to control this. *By default modulation is on* and opening sizes are reduced to 5% of full values when the outside is 15°C or more colder than inside. This models the fact that occupants are unlikely to throw the windows wide open when it is very cold outside (for manual control systems).

Modelling Infiltration

Normally when [Calculated](#) natural ventilation is set DesignBuilder includes a single crack in each surface in the simulation to account for infiltration. The size and properties of this crack depend on the setting of the Airtightness slider.

Tip: The effects of infiltration in some areas of the building and in some calculation types can be minor and so in order to reduce simulation time for calculated natural ventilation models infiltration can be switched off. This can be done by unchecking the **Model infiltration** checkbox for the areas of the building where infiltration is not to be modelled. For example you may wish to switch off infiltration in roof blocks (and include the roofspace ventilation using a smaller number of vents or holes instead) or you could switch it

off in the whole building for Summertime simulations where the effects of infiltration would be minimal relative to the much larger flow rates due to open windows, vents, holes and doors.

Airtightness slider

The setting of the Airtightness slider indicates the Crack template applied when setting crack properties:

Very poor
Poor
Medium
Good
Excellent

See also [Mixed mode](#).

Technical details

You can find more technical details of the Airflow network used in EnergyPlus including background, algorithms, limitations and validation results in a paper by Lixing Gu at:

https://www.ibpsa.org/proceedings/BS2007/p398_final.pdf

DCV Modelling

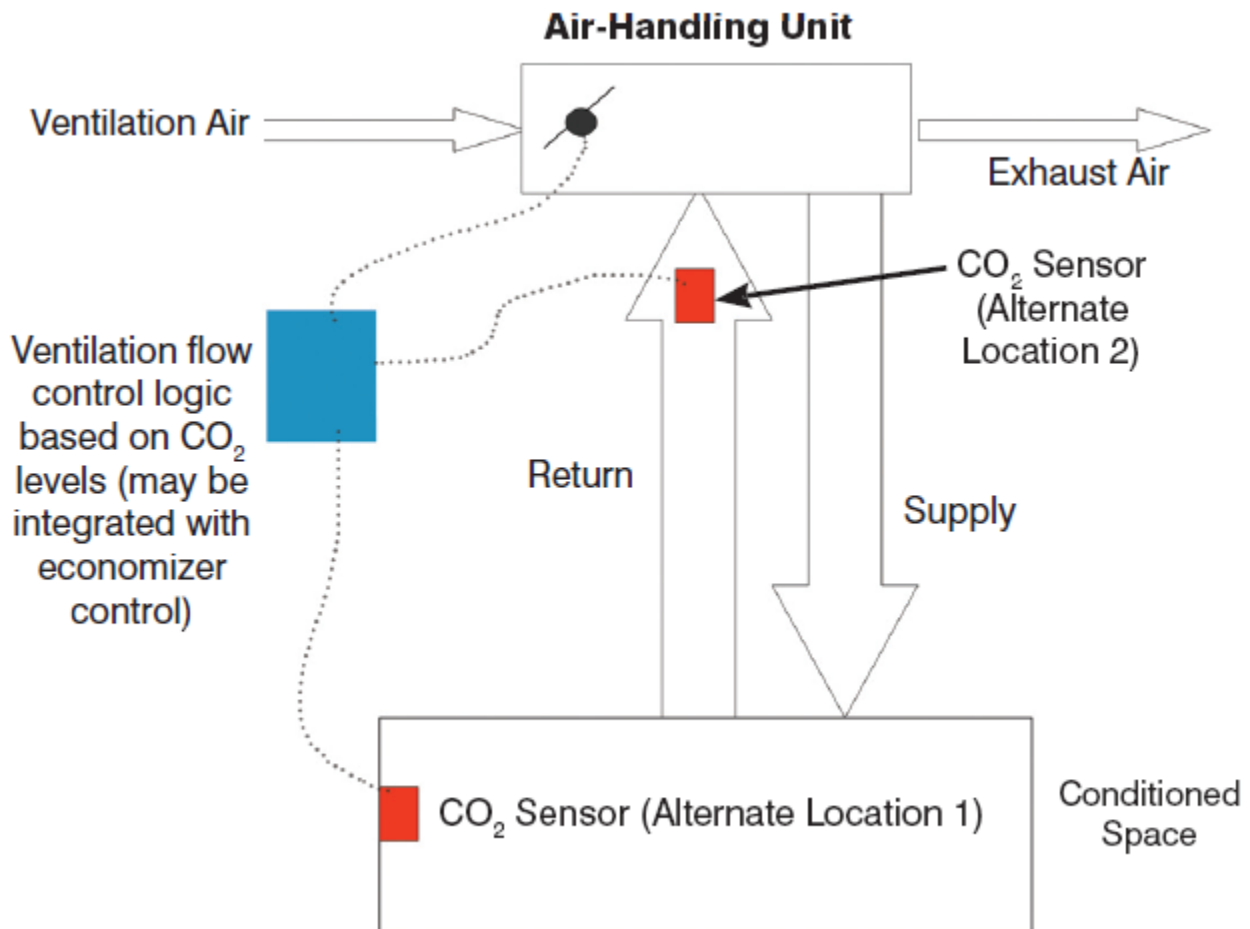
Background

Demand-controlled ventilation (DCV) is an energy-saving control strategy that reduces the rate at which outdoor air is delivered to a zone during periods of partial occupancy. In most commercial buildings ventilation is provided to deal with two types of indoor pollution: odours from people, and off-gassing from building components and furniture. When a space is vacant, it has no people pollution so the people-related ventilation rate is not needed. Many types of high-occupancy spaces, such as classrooms, multi-purpose rooms, theatres, conference rooms, or lobbies have ventilation designed for a high peak occupancy that rarely occurs.

Demand-controlled ventilation can provide significant HVAC energy savings by conditioning only the amount of ventilation air necessary to maintain good indoor air quality. It can be particularly effective in buildings or zones that have widely varying occupancy.

Application

DCV is most successful in reducing ventilation rates, and so reducing fan energy use, when meeting the demands of indoor air quality (IAQ). It is not necessarily so successful, in energy terms, for high design cooling loads, where airflows will increase above the required IAQ minimum to meet space cooling loads. In such cases, the HVAC system would need to increase the supply air flow rate of the conditioned air to remove the excess heat from the space and so maintain the desired indoor setpoint temperature. This will happen regardless of the current IAQ requirements and/or occupancy levels. Therefore, the amount of outside air brought into the building could exceed the minimum airflow rate required for maintaining the indoor IAQ. In other words, even though DCV is designed to reduce ventilation rates to save energy, in this case, the need to cool the space down will be prioritised, and ventilation rates may increase regardless. Practically, this typically means that IAQ-driven DCV is most effective in ASHRAE climate zones 4 and above.



Schematic of a typical DCV system from [Demand-Controlled Ventilation and Sustainability](#) with permission from author Tom Lawrence and ASHRAE.

CO2 Control / DCV Modelling Example

DesignBuilder provides various options for modelling DCV. The examples below show how to model two common types of IAQP (Indoor Air Quality Procedure) system:

AHU IAQP, where the amount of fresh air supplied to zones is controlled by varying the recirculation within the AHU.

Zone IAQP, a common approach for 100% fresh air units where outdoor air is supplied from an AHU, with no recirculation, and the supply air flow rates are controlled within the zone terminal units.

The examples below walk you through the steps involved in modelling both of these systems.

AHU IAQP

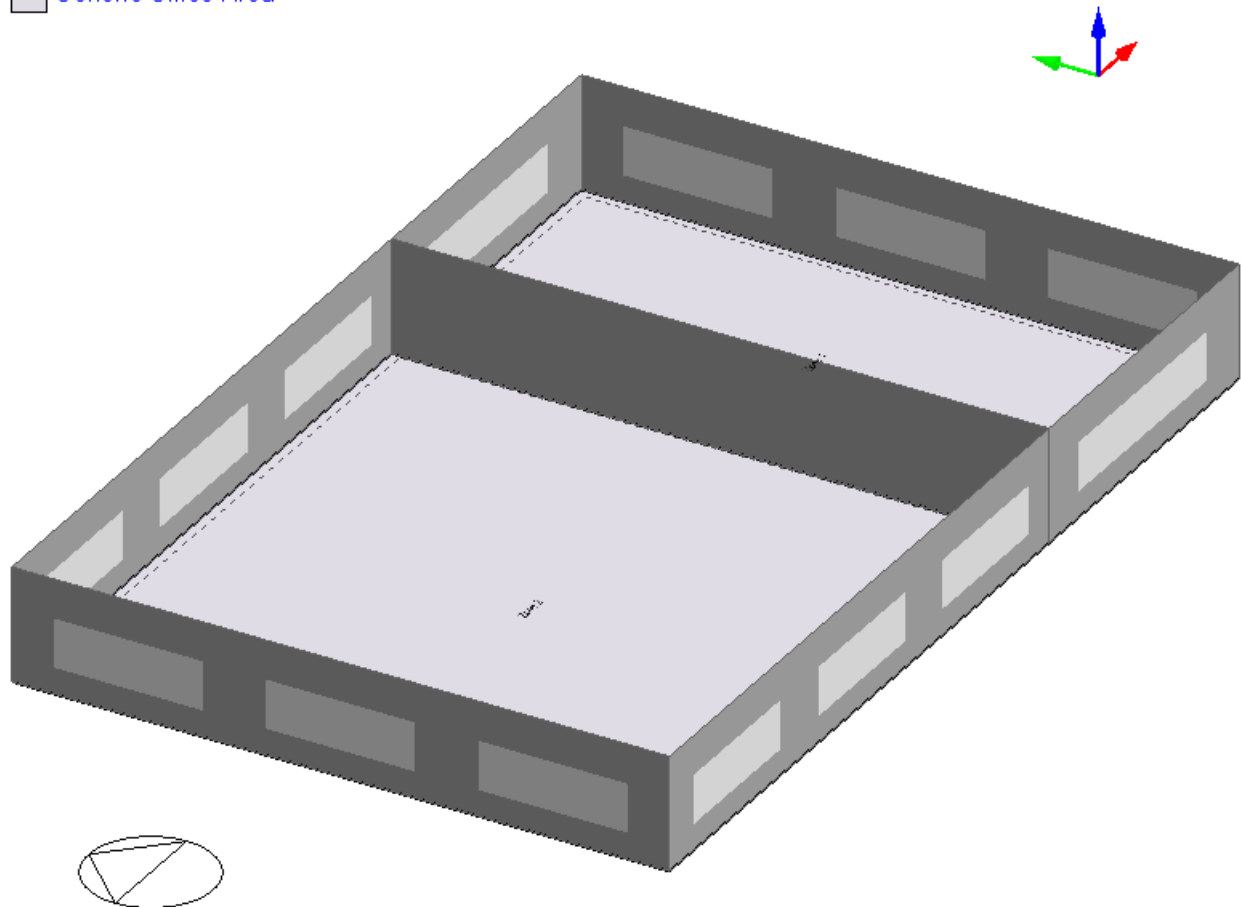
Model Set up

Create a base model

Step 1 - Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m.

Sub-divide the block into 2 zones, an East and a West zone as shown below. The East zone should have East-West length of 10m and the West zone a length of 20m.

 Generic Office Area

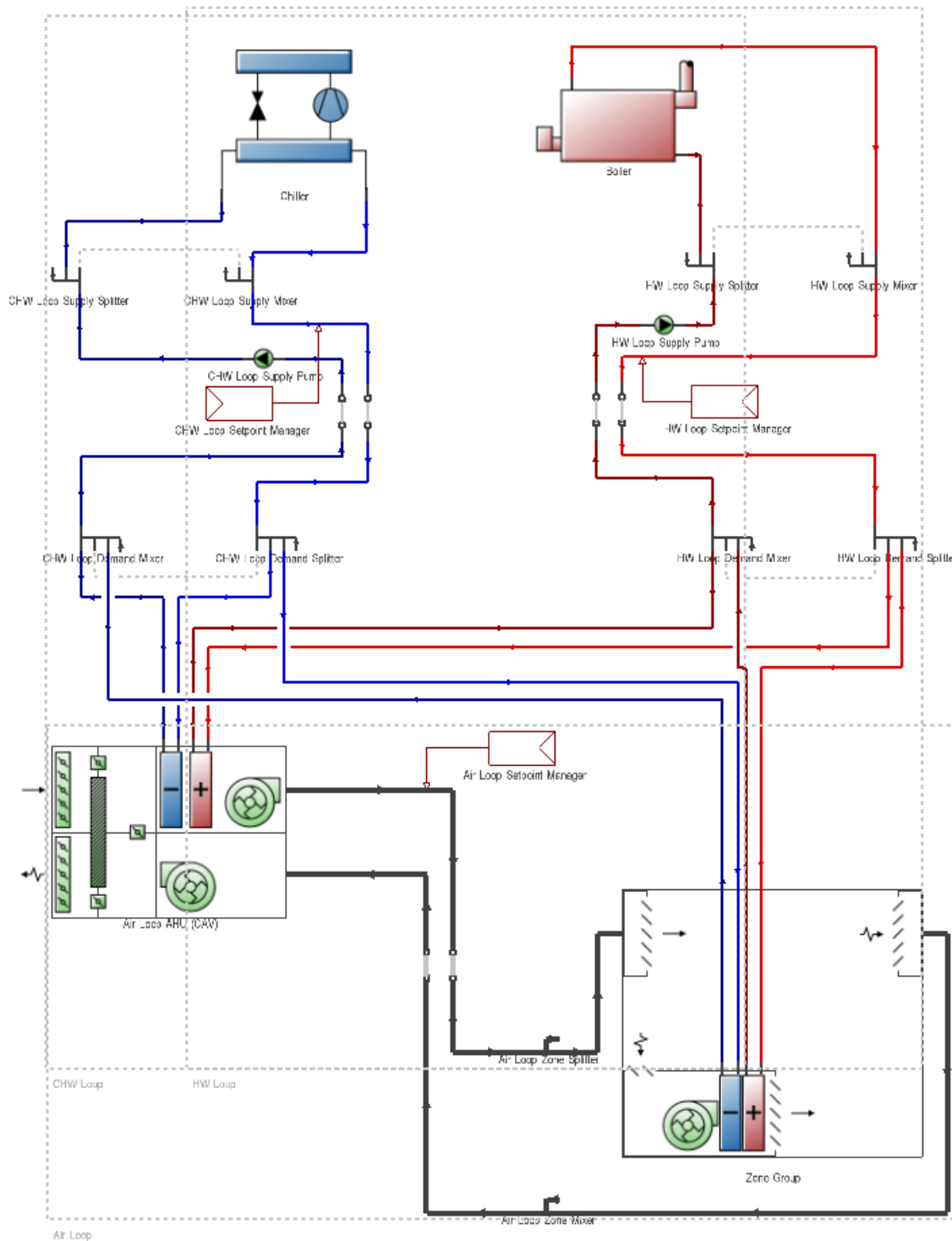


Use default model data settings.

Detailed HVAC layout

Step 2 - Set the HVAC model option to “Detailed”.

Step 3 - Navigate to HVAC System level and load the “DOAS with DCV and FCU” Detailed HVAC template.



AHU

Step - 4 - All relevant IAQP settings should now be applied through the HVAC template, but to highlight the differences between a generic unit and DOAS, check the settings in the steps below.

Navigate to the AHU and open the Edit dialog. On the [Outdoor Air System](#) tab of the [Air Handling Unit](#) dialog:

Under the **Recirculation** header, set [Minimum outdoor air flow rate](#) to 0.

Under the **Demand Controlled Ventilation** header, ensure that the [Demand controlled ventilation](#) checkbox is checked.

Check that [System outdoor air method](#) is set to **4-Indoor Air Quality Procedure (IAQP)**.

Press OK.

Air handling unit

General Outdoor Air System

DOAS

☐ Dedicated outdoor air supply

Recirculation

☒ On

Minimum outdoor air flow rate (m3/s) 0.00000

Maximum outdoor air flow rate (m3/s) Autosize

Minimum limit type 2-Fixed minimum

Economiser (Free Cooling)

Economiser control type 1-No economizer

Outdoor Air Schedules

☐ Apply minimum outdoor air schedule

☐ Apply minimum fraction of outdoor air schedule

☐ Apply maximum fraction of outdoor air schedule

Demand Controlled Ventilation

☒ Demand controlled ventilation

System outdoor air method 4-Indoor air quality procedure

 Availability schedule On 24/7

HVAC zones

Step 5 - As above, the relevant IAQP settings should already be applied through the HVAC template, but you should still check the settings in the steps below to help understand how the system is set up.

Navigate to one of the HVAC zones in the HVAC zone group and open the [HVAC zone](#) dialog.

On the **General** tab, under the **CO2 and Contaminant Control** header, ensure that the [CO2 and contaminant control](#) checkbox is checked.

HVAC Zone

General Sequence Target

General

Name BLOCK1:ZONE1

Humidistat Control

CO2 and Contaminant Control

☒ CO2 and contaminant control

Zone Air Distribution Effectiveness

Sizing

Heating Sizing

Heating design supply air temperature input method 1-Supply air temperature

Heating design supply air temperature (°C) 50.000

Heating design supply air humidity ratio 0.0040

Zone heating sizing factor 1.250

Heating design air flow method 1-Design day

Cooling Sizing

Cooling design supply air temperature input method 1-Supply air temperature

Cooling design supply air temperature (°C) 14.000

Cooling design supply air humidity ratio 0.009

Cooling minimum air flow fraction (turndown ratio) 0.000

Zone cooling sizing factor 1.150

Cooling design air flow method 1-Design day

Outside Air Sizing

Outside air method 7-Indoor air quality procedure

Dedicated Outdoor Air System (DOAS)

☒ Include DOAS system?

Control strategy 1-Neutral supply air

Low temperature setpoint for design (°C) Autosize

High temperature setpoint for design (°C) Autosize

If you've made any changes, on the **Target** tab, apply the changes to all zones by:

- Selecting the **3-Multiple (modified settings only)** option.
- Selecting both zones.
- Press OK

Zone terminal units

Step 6 - Update the terminal units:

- a. Navigate to one of the [CAV No Reheat](#) terminal units and open the Edit dialog.
- b. If the [Control on outdoor air flow](#) checkbox is not already checked, check it.

Note: you can ignore the [Per person ventilation mode](#) child setting of the [Control on outdoor air flow](#) checkbox as it is only used when outdoor air ventilation rates are calculated based on a per person rate, and not when using the IAQP.

- c. If you've made any changes, on the **Target** tab, apply the changes to all zones by:

- Selecting the **3-Multiple (modified settings only)** option.
- Selecting both zones.
- Press OK

Note: the [CAV No Reheat](#) terminal unit used in this example provides pure CO2 control. Replacing this default CAV terminal with a [VAV No Reheat](#) terminal adds free cooling functionality. In this case you would have to change the AHU type to VAV to connect the VAV terminal.

Model Data Settings

Step 7 - Go to Site level and ensure that the [Simulate CO2 concentration](#) site level option is enabled.

The screenshot shows the 'Location' tab in a software interface. The 'Location Template' is set to 'LONDON/GATWICK ARPT'. The 'Site Details' section includes fields for 'Elevation above sea level (m)' (62.0), 'Exposure to wind' (2-Normal), and 'Site orientation (°)' (0.0). The 'Simulate CO2 concentration' checkbox is checked and highlighted with a red box. The 'CO2 concentration schedule' is set to 'Default outdoor CO2 levels 400 ppm (Source: NOA)'. The 'Simulate contaminant concentration' checkbox is unchecked. Other settings include 'Time and Daylight Saving', 'Simulation Weather Data', 'Winter Design Weather Data', and 'Summer Design Weather Data'.

Step 8 - At building level, on the **HVAC** tab, ensure that the [Mechanical ventilation, On](#) checkbox is checked.

In this case, the zone [Outside air definition method](#) setting is used for sizing using fresh air per person, area and ac/h.

Step 9 – At building level, confirm the [CO2 setpoint](#) settings on the Activity tab:

Environmental Control		⌵
Heating Setpoint Temperatures		⌵
🔧 Heating (°C)	22.0	
🔧 Heating set back (°C)	12.0	
Cooling Setpoint Temperatures		⌵
🔧 Cooling (°C)	24.0	
🔧 Cooling set back (°C)	28.0	
Heating Comfort PMV Setpoints		»
Cooling Comfort PMV Setpoints		»
Humidity Control		»
Ventilation Setpoint Temperatures		»
Minimum Fresh Air		»
CO2/Contaminant Setpoints		⌵
CO2 setpoint (ppm)	900	
Min CO2 concentration (ppm)	600	
Max CO2 concentration (ppm)	900	
Contaminant concentration setpoint (ppm)	0.0000000	

Step 10 – Run a simulation over the typical winter period with sub-hourly outputs requested.

Results

When the simulation has finished, check the following sub-hourly results in the [Results Viewer](#):

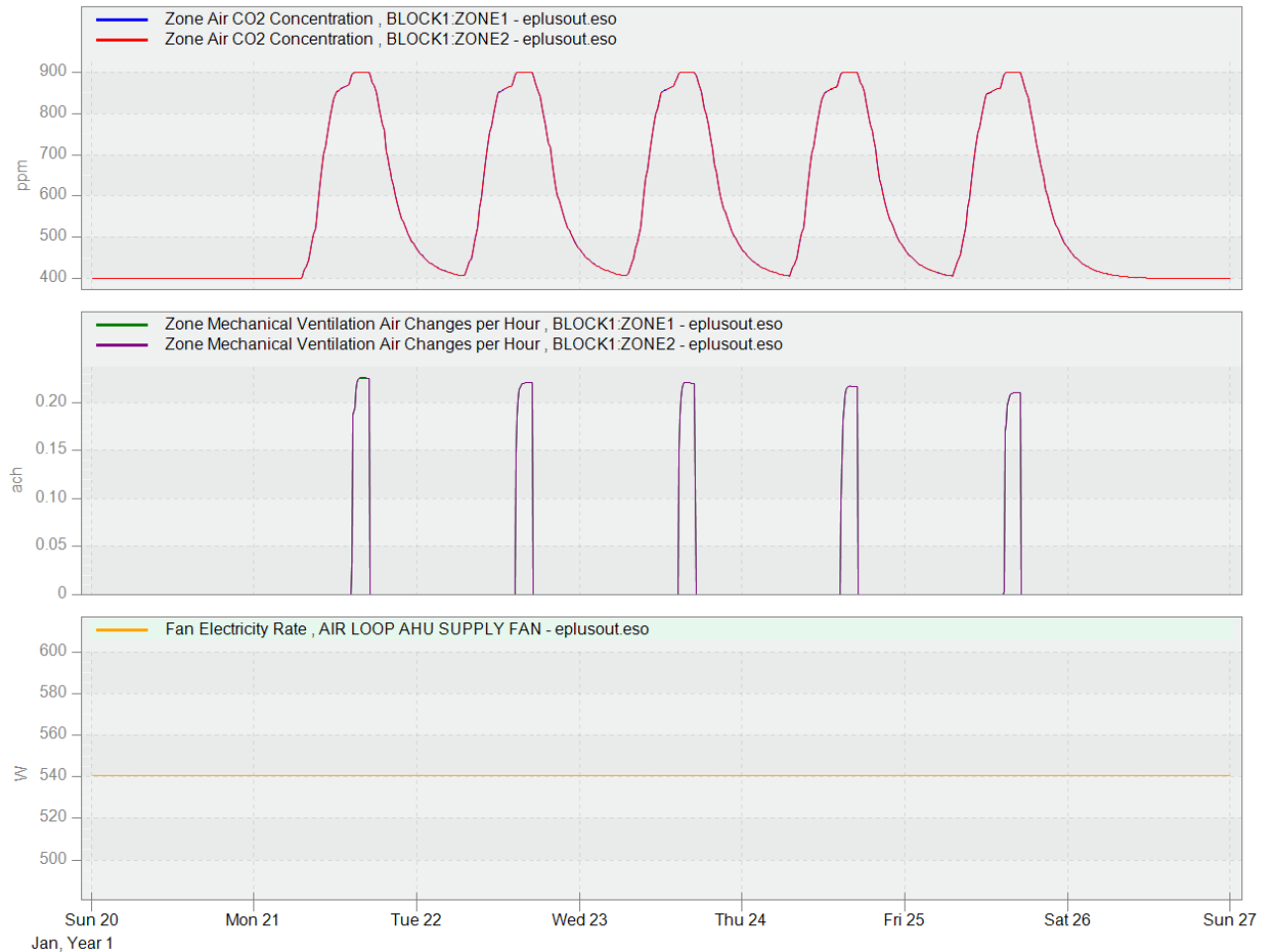
Zone Air CO2
Zone Mechanical Ventilation
Fan Electricity Rate, AIR LOOP AHU SUPPLY FAN

These will allow you to check the indoor CO2 levels, the fresh air delivery rate in ac/h and the associated fan energy consumption.

You should see results similar to those below, with zone CO2 levels controlled to 900 ppm and the fan operating constantly.

AHU IAQP Example

20 January, Year 1 [00:10] - 27 January, Year 1 [00:00]



Zone IAQP

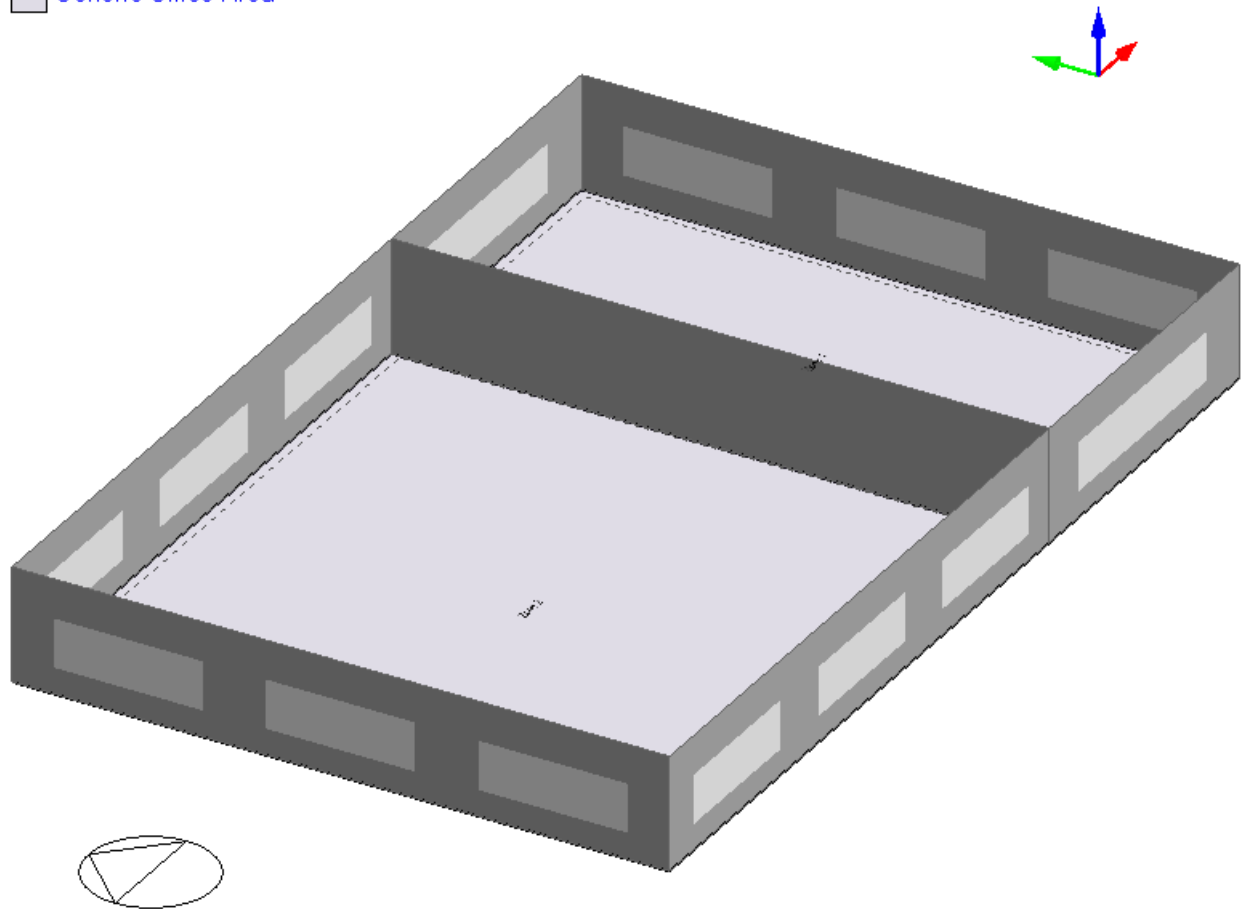
Model Set up

Create a base model

Step 1 - Create a new file located in London Gatwick and add a building to the site with a simple rectangular block having dimensions 30m x 20m.

Sub-divide the block into 2 zones, an East and a West zone as shown below. The East zone should have East-West length of 10m and the West zone a length of 20m.

 Generic Office Area

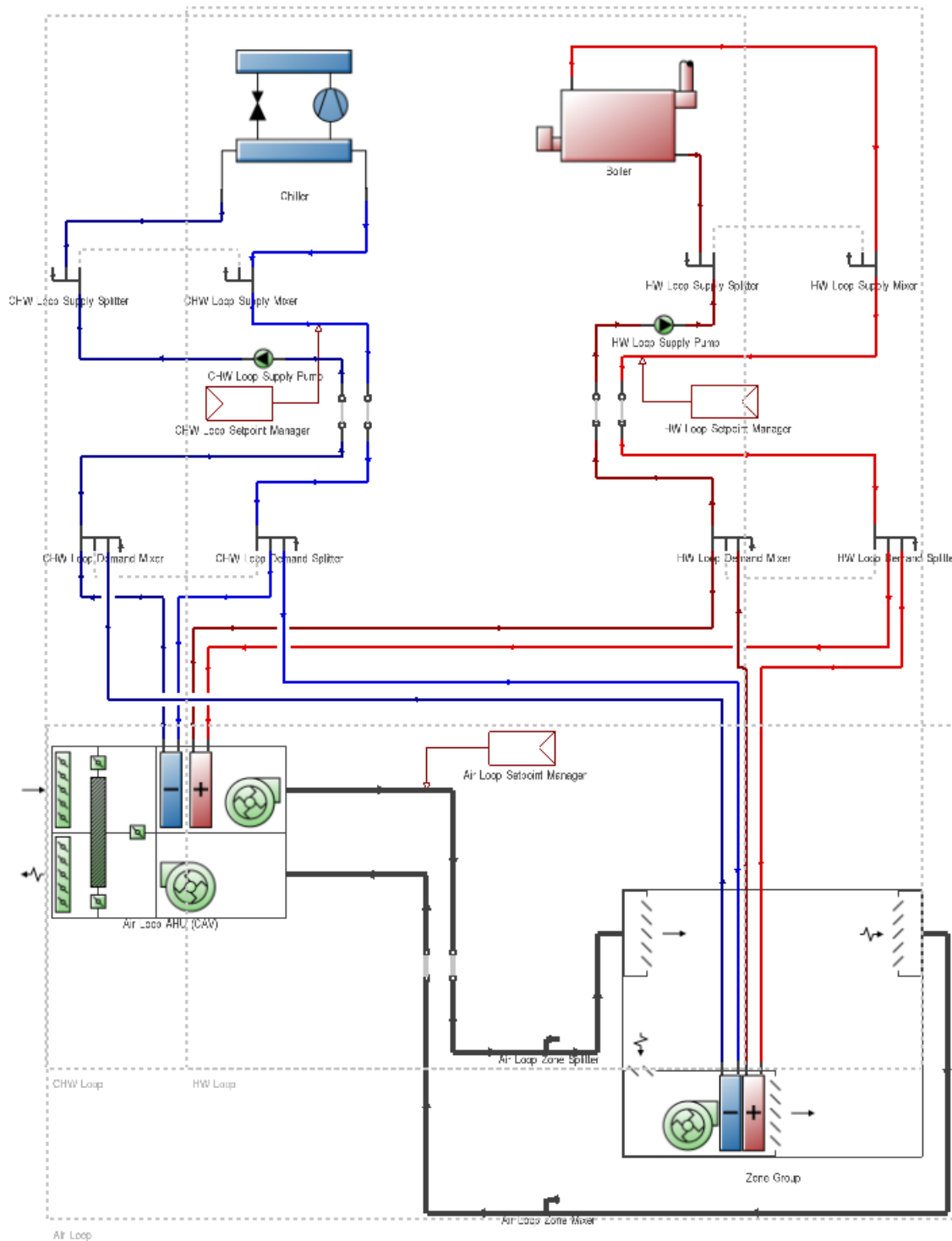


Use default model data settings.

Detailed HVAC layout

Step 2 - Set the HVAC model option to “Detailed”.

Step 3 - Navigate to HVAC System level and load the “DOAS with DCV and FCU” Detailed HVAC template.



AHU

Step - 4 - Navigate to the AHU and open the Edit dialog. On the [Outdoor Air System](#) tab of the [Air Handling Unit](#) dialog:

Under the **Recirculation** header, switch off recirculation by unchecking the **Recirculation, On** checkbox.
Press OK.

HVAC zones

Step 5 - Navigate to one of the HVAC zones in the HVAC zone group and open the [HVAC zone](#) dialog.

On the **General** tab, under the **CO2 and Contaminant Control** header, check that the [CO2 and contaminant control](#) checkbox has been checked.

HVAC Zone

General | Sequence | Target

General

Name: BLOCK1:ZONE1

Humidistat Control

CO2 and Contaminant Control

☒ CO2 and contaminant control

Zone Air Distribution Effectiveness

Sizing

Heating Sizing

Heating design supply air temperature input method: 1-Supply air temperature

Heating design supply air temperature (°C): 50.000

Heating design supply air humidity ratio: 0.0040

Zone heating sizing factor: 1.250

Heating design air flow method: 1-Design day

Cooling Sizing

Cooling design supply air temperature input method: 1-Supply air temperature

Cooling design supply air temperature (°C): 14.000

Cooling design supply air humidity ratio: 0.009

Cooling minimum air flow fraction (turndown ratio): 0.000

Zone cooling sizing factor: 1.150

Cooling design air flow method: 1-Design day

Outside Air Sizing

Outside air method: 7-Indoor air quality procedure

Dedicated Outdoor Air System (DOAS)

☒ Include DOAS system?

Control strategy: 1-Neutral supply air

Low temperature setpoint for design (°C): Autosize

High temperature setpoint for design (°C): Autosize

If any changes were made, on the **Target** tab, apply the changes to all zones by:

Selecting the **3-Multiple (modified settings only)** option.
 Selecting both zones.
 Press OK

Zone terminal units

Step 6 - Update the terminal units:

- Navigate to one of the [CAV No Reheat](#) terminal units and open the Edit dialog.
- Check the [Control on outdoor air flow](#) checkbox if it is not already checked.

Note: you can ignore the [Per person ventilation mode](#) child setting of the [Control on outdoor air flow](#) checkbox as it is only used when outdoor air ventilation rates are calculated based on a per person rate, and not when using the IAQP.

- On the **Target** tab, apply the changes to all zones by:

Selecting the **3-Multiple (modified settings only)** option.
 Selecting both zones.
 Press OK

Note: the [CAV No Reheat](#) terminal unit used in this example provides pure CO2 control. Replacing this default CAV terminal with a [VAV No Reheat](#) terminal adds free cooling functionality. In this case you would have to change the AHU type to VAV to connect the VAV terminal.

Model Data Settings

Step 7 - Go to Site level and enable the [Simulate CO2 concentration](#) site level option.

The screenshot shows the 'Location' tab of a software interface. The 'Template' is set to 'LONDON/GATWICK ARPT'. Under 'Site Details', the 'Elevation above sea level (m)' is 62.0, 'Exposure to wind' is '2-Normal', and 'Site orientation (°)' is 0.0. The 'Simulate CO2 concentration' checkbox is checked, and the 'CO2 concentration schedule' is set to 'Default outdoor CO2 levels 400 ppm (Source: NOA)'. The 'Simulate contaminant concentration' checkbox is unchecked. Other options like 'Time and Daylight Saving', 'Simulation Weather Data', 'Winter Design Weather Data', and 'Summer Design Weather Data' are also visible.

Layout	Location	Region
Location Template		
Template		LONDON/GATWICK ARPT
Site Location		
Site Details		
Elevation above sea level (m)	62.0	
Exposure to wind	2-Normal	
Site orientation (°)	0.0	
Site Height Variation	>>	
Ground	>>	
Horizon	>>	
Water Mains Temperature	>>	
Precipitation	>>	
Site Green Roof Irrigation	>>	
Outdoor Air CO2 and Contaminants	>>	
☒ Simulate CO2 concentration ☐ Simulate contaminant concentration		
☐ CO2 concentration schedule Default outdoor CO2 levels 400 ppm (Source: NOA)		
Time and Daylight Saving	>>	
Simulation Weather Data	>>	
Winter Design Weather Data	>>	
Summer Design Weather Data	>>	

Step 8 - At building level, on the **HVAC** tab, ensure that the [Mechanical ventilation, On](#) checkbox is checked.

Step 8.b - Check the [Indoor air quality procedure](#) checkbox.

Step 9 – At building level, confirm the [CO2 setpoint](#) settings on the Activity tab:

Environmental Control		⌵
Heating Setpoint Temperatures		⌵
🔧 Heating (°C)	22.0	
🔧 Heating set back (°C)	12.0	
Cooling Setpoint Temperatures		⌵
🔧 Cooling (°C)	24.0	
🔧 Cooling set back (°C)	28.0	
Heating Comfort PMV Setpoints		»
Cooling Comfort PMV Setpoints		»
Humidity Control		»
Ventilation Setpoint Temperatures		»
Minimum Fresh Air		»
CO2/Contaminant Setpoints		⌵
CO2 setpoint (ppm)	900	
Min CO2 concentration (ppm)	600	
Max CO2 concentration (ppm)	900	
Contaminant concentration setpoint (ppm)	0.0000000	

Step 10 – Run a simulation over the typical winter period with sub-hourly outputs requested.

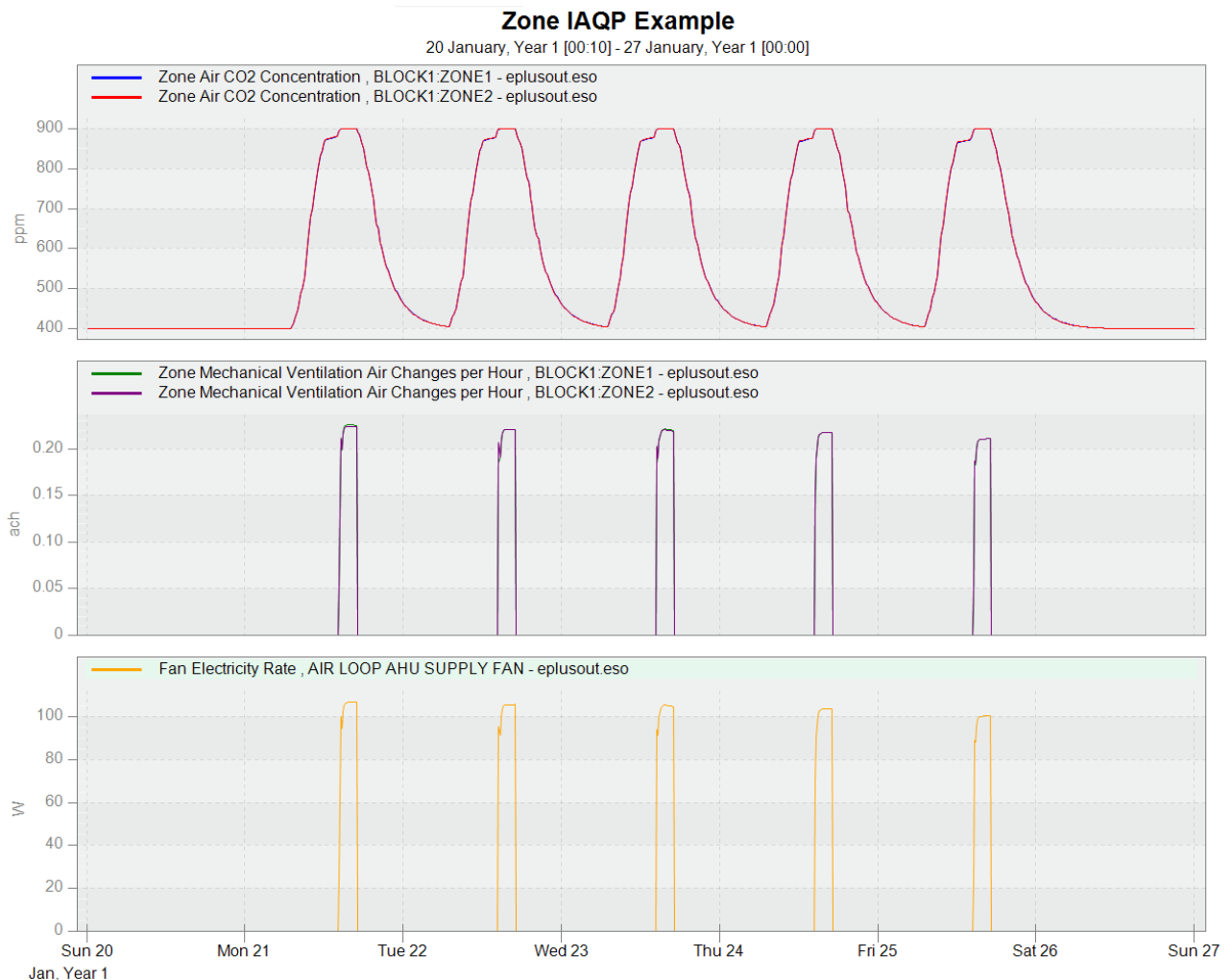
Results

When the simulation has finished, check the following sub-hourly results in the [Results Viewer](#):

Zone Air CO2
Zone Mechanical Ventilation
Fan Electricity Rate, AIR LOOP AHU SUPPLY FAN

These will allow you to check the indoor CO2 levels, the fresh air delivery rate in ac/h and the associated fan energy consumption.

You should see results similar to those below, with zone CO2 levels controlled to 900 ppm and the fan operating for only a few hours in the afternoon to achieve this.



Comparing the results for the two types of IAQP system, it is clear that the Zone IAQP method uses much less fan energy than the AHU IAQP method to achieve similar levels of comfort. This is because the fan only operates to provide fresh air to the zones to meet the IAQP CO₂ setpoint of 900 ppm.

Adjacent Buildings

Adjacent buildings (i.e. other buildings that are touching the modelled building) usually modify the boundary conditions relative to external and also provide shade.

There is no explicit way in DesignBuilder to model temperatures and heat flows in 2 buildings simultaneously and account for energy transfer between them. Future versions may allow this, but in the meantime there are two options:

Model the adjacent building using [component blocks](#) (for shading/reflection) and [adiabatic surfaces](#) between the buildings. This is the usual way to treat an adjacent building not requiring full simulation and is accurate if the temperature of the adjacent and actual buildings are similar or if there is insulation between the two buildings.

If the adjacent building does require simulation then model it as part of the actual building and separate simulation results at the end.

You can set [adiabatic adjacent condition](#) data under Constructions on the Constructions tab at surface level.

Merging buildings

Where two separate buildings have been created inadvertently, the blocks of one building can be transferred to the other building using the [Transfer blocks](#) command from the Edit menu.

Airflow

If there is airflow between the two buildings and the adjacent building is at the same temperature as the modelled building then you could probably ignore its effect. To model the effect of the airflow through an opening between buildings in detail you should model both buildings as one DB building using the [Calculated](#) natural ventilation model option. If you do it this way and the adjacent building uses energy you must separate results for the two buildings at the end. You can use different boiler and chiller fuels to differentiate fuel consumption between the two buildings.

If you use the [Simple](#) natural ventilation instead of Calculated, DesignBuilder uses an assumed mixing of air between the two buildings based on 'flow per opening area' as set on the Model options dialog on the Advanced tab under [Natural Ventilation](#).

Electrochromic Glazing

Electrochromic windows are part of a new generation of technologies called switchable glazing-or 'smart' windows. Switchable glazing can change the light transmittance, transparency, or shading of windows in response to an environmental signal such as sunlight, temperature or an electrical control. Electrochromic windows change from transparent to tinted by applying an electrical current. Potential uses for electrochromic technology include daylighting control, glare control, solar heat control, and fading protection in windows and skylights. By automatically controlling the amount of light and solar energy that can pass through the window, electrochromic windows can help save energy in buildings.

A variety of electrochromic technologies and media have been developed. One type is darkened by applying a small electrical voltage to the windows and lightened by reversing the voltage. Light transmittance during operation varies from five to 80 percent. Once the change in tint has been initiated, the electrochromic glazing has 'memory' and does not need constant voltage to maintain the tinting. Further, the film can be tuned to block certain wavelengths, such as infrared (heat) energy.

Another switchable technology, the liquid crystal suspended particle device (SPD), contains molecular particles suspended in a solution between plates of glass. In their natural state, the particles move randomly and collide, blocking the direct passage of light. When energized, the particles align rapidly and the glazing becomes transparent. This type of switchable glazing can block up to about 90 percent of light.

You can easily model electrochromic glazing in DesignBuilder using the Window shading data and defining the properties of the outer Window Pane to be used when the shading is active. The procedure is:

Specify the glazing type to be used when the electrochromic glazing system is clear (bleached) in the standard way on the Openings tab under External glazing.

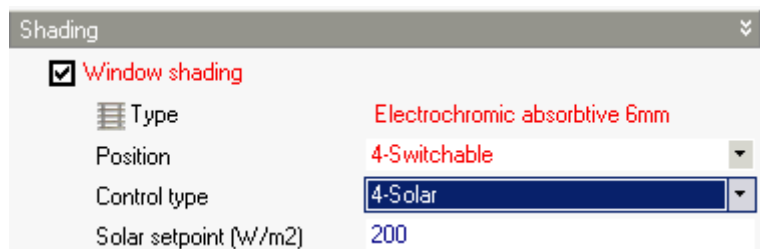
Under External Glazing and Shading headers, check the 'Window shading' option.

Select the Type as 'Electrochromic absorptive 6mm' (for example).

Select the Position as 4-Switchable.

The Shading Type defines the properties of the pane which is to be used in place of the standard outer glazing pane according to the control data in the Control type.

Set the Control type together with any supplementary setpoint data.



A model using the data in the above screenshot would enter its' darkened state whenever the solar radiation incident on the outside of the window is greater than 200 W/m2..

Trombe Walls

Trombe walls are passive solar devices designed for thermal storage and delivery. A Trombe wall consists of a thick wall (150mm to 300mm) [8" to 16"] faced with a selective surface solar absorber, air gap, and high transmissivity glass pane. Trombe walls are usually South facing (in the Northern Hemisphere) for maximum sun exposure. An overhang above the wall is used to decrease exposure in the summer when the sun is high in the sky and heating is not required, yet still allows for full exposure in the winter when the sun is low in the sky and heating is desirable. In DesignBuilder, there is no Trombe wall object per se; rather, it is composed of other standard building components. This approach provides flexibility in specifying the various wall parameters and allows the freedom to explore unusual configurations.

To simulate the Trombe wall, a narrow zone is coupled to the desired surface via an interzone partition. The depth of the zone corresponds to the size of the air space usually 18mm to 150mm ($\frac{3}{4}$ " to 6"). In most cases the Trombe zone will be a sealed zone with no ventilation. The exterior wall of the Trombe zone contains a single or double-pane window. Optimally, the window covers nearly all of the wall area and has a very high transmissivity to allow the maximum amount of solar flux into the Trombe zone. Frames and dividers can be defined as usual for the window. The interior wall is usually constructed of very thick masonry materials with a solar absorber surface as the innermost layer of the wall. The absorber is a selective surface material with very high absorptivity and very low emissivity, e.g. copper with a special black surface treatment. It is important to make sure the [Solar Distribution](#) is set to **3-Full Interior and Exterior** so that the majority of the solar flux is directed on the absorber surface and not just on the very small area of the Trombe zone floor.

The [Zone type](#) on the [Activity tab](#) should be set to **3-Cavity** at zone level in the Trombe wall cavity zone and this causes the following further changes to be made to the model:

The zone is set as unoccupied by loading **<None>** Activity, HVAC and Lighting template data.

Suspended floors and ceilings are switched off when using Separate constructions.

The **5-Cavity** [Internal convection algorithm](#) is set for Cooling design, Heating design and simulation calculations to correctly model the air space.

The **3-Full interior and exterior** [Solar distribution algorithm](#) is set for Cooling design and simulation calculations because it allows solar radiation to be accurately distributed onto zone internal surfaces.

The default option **2-Full exterior** assumes all solar radiation falls on floor which is not good for modelling solar absorption by a Trombe wall.

Finally, an [overhang](#) should be attached to the Trombe zone to control the amount of seasonal sun exposure. There is considerable freedom to experiment with different materials, sizes, and configurations.

Passive Trombe wall

Passive Trombe walls perform without the assistance of any additional mechanical equipment. Most Trombe walls are passive Trombe walls. They can be either sealed or naturally ventilated. For a sealed or unvented Trombe wall, the Zone Inside Convection Algorithm should be set to **5-Cavity**. This algorithm correctly calculates the convection coefficients for a narrow sealed vertical cavity based on the ISO 15099 standard.

The EnergyPlus modeling approach for the sealed passive Trombe wall has been validated with experimental data (Ellis 2003). For a naturally ventilated Trombe wall, there is no built-in algorithm for calculating the correct convection coefficients on the inside of the cavity walls. One option is to use the **1-Detailed** [Inside convection algorithm](#). This algorithm takes into account some natural convection effects but is intended for a normal sized room.

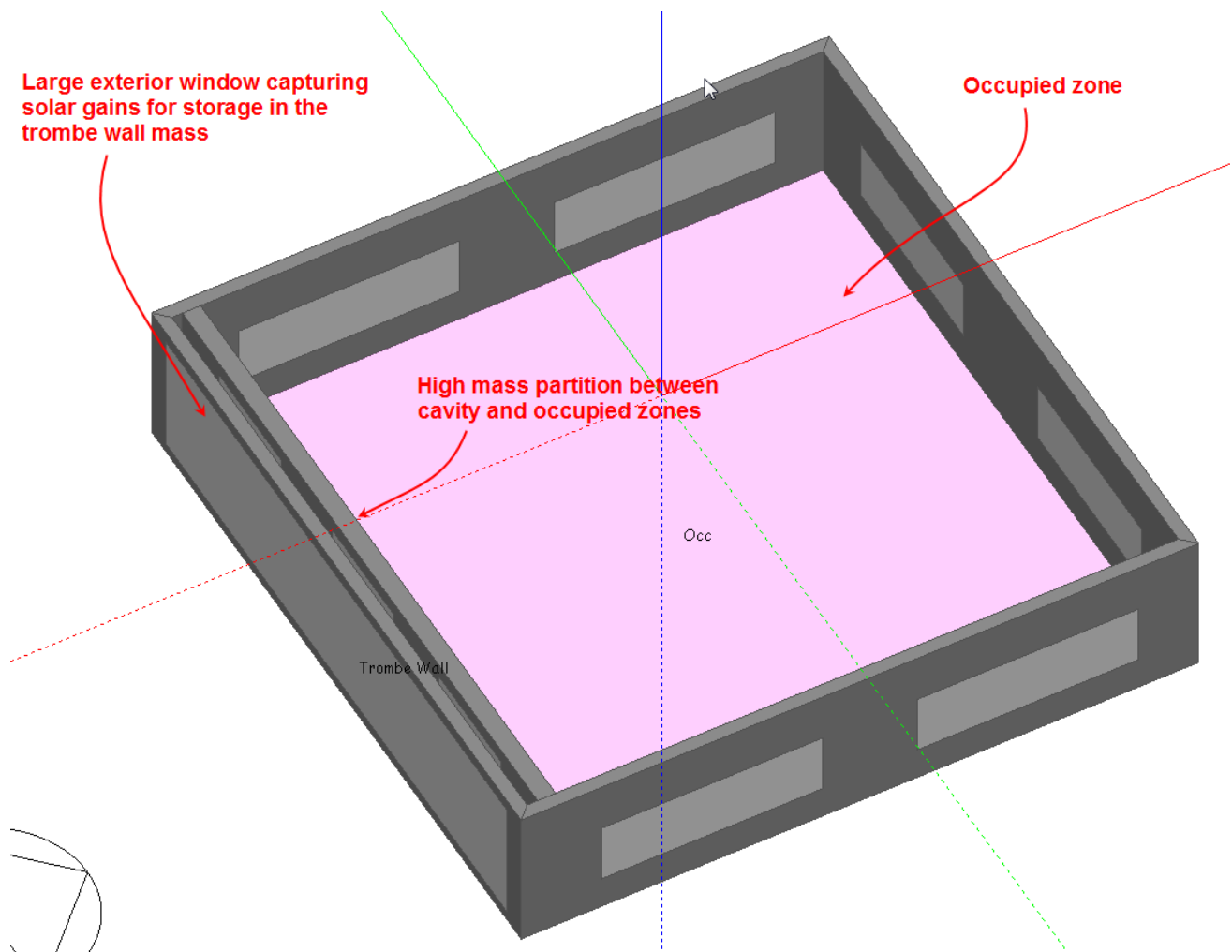
You can define holes and vents through the Trombe wall by [drawing them on at surface level](#). Vent openings can be [scheduled](#) and controlled by [internal temperature](#).

Mechanically ventilated Trombe wall

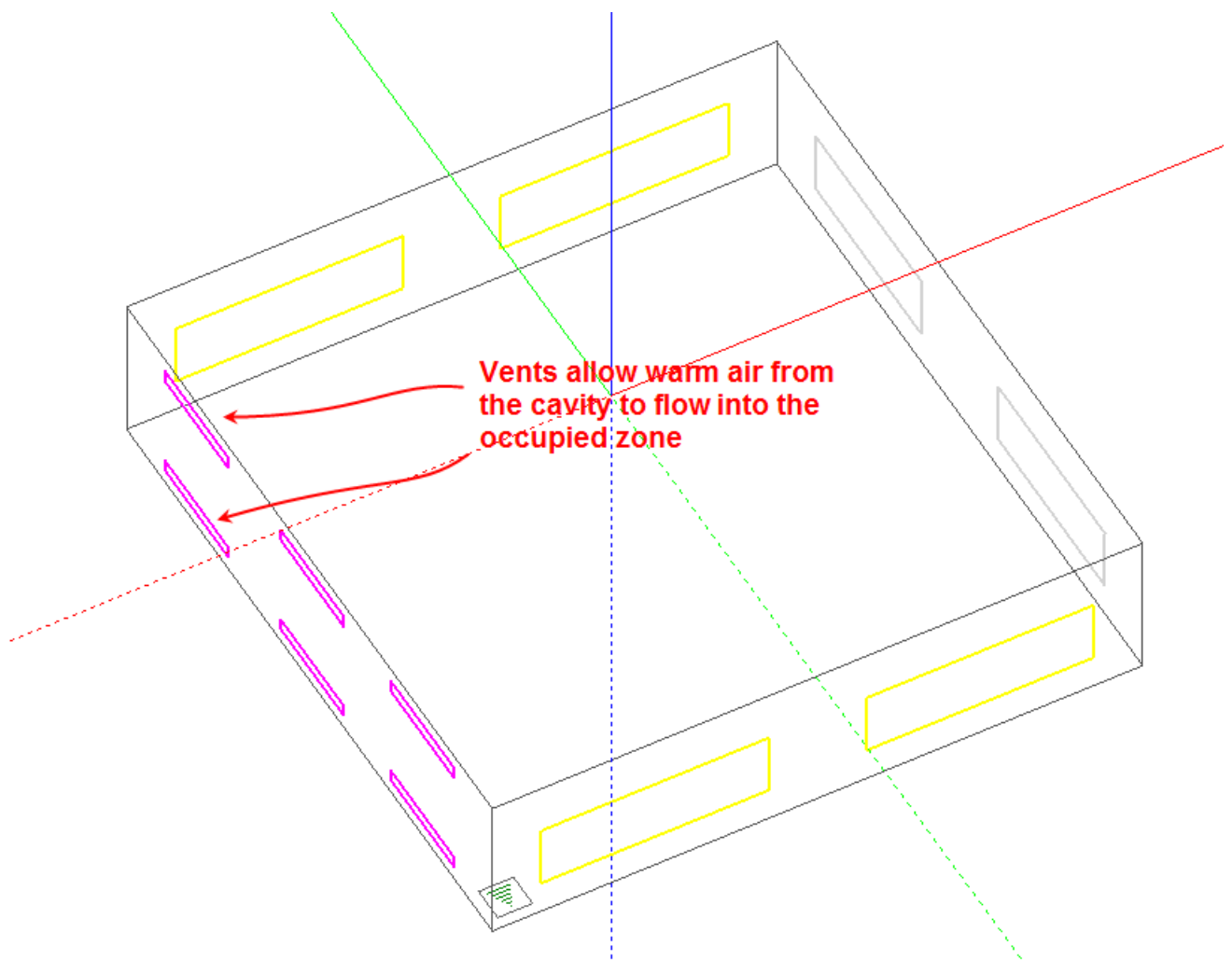
You can also model mechanically ventilated Trombe walls by using Detailed HVAC and specifying the Trombe wall cavity zone as a plenum zone.

Trombe Wall Example

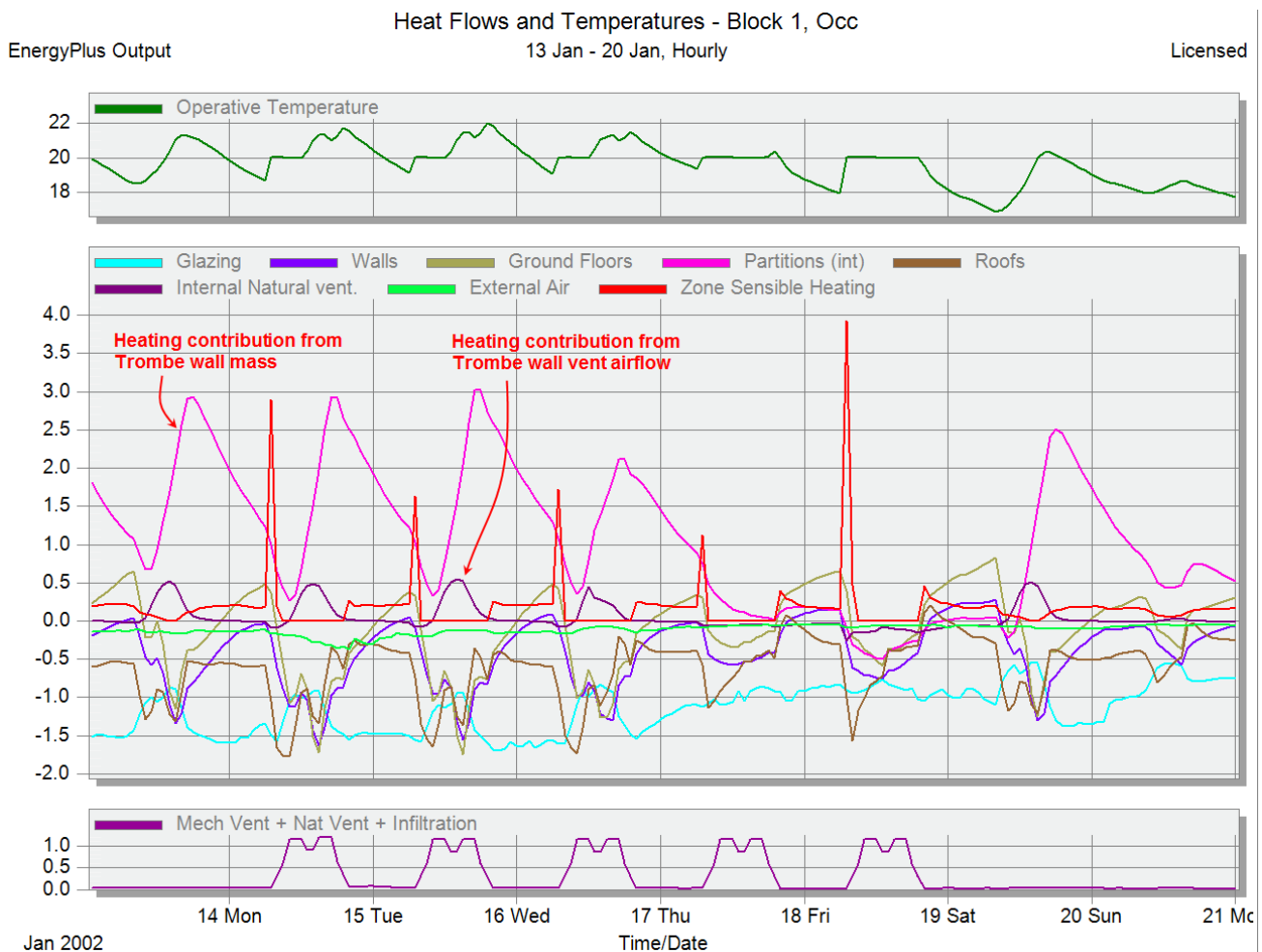
A Trombe wall example model is provided with DesignBuilder. This is a simple 2 zone building, one occupied zone, the other zone representing the cavity. The screen shot below of the example building shows the 2 zones and the high mass partition between them.



The screenshot below shows the configuration of vents in the partition between the 2 zones allowing warm air to move into the occupied space. The calculated natural ventilation model option has been used to allow detailed calculation of air movement between the 2 zones.



The hourly results below are for the occupied zone and illustrate the delayed heating effect of the Trombe wall mass on the occupied zone during the evening and also the lesser but more instantaneous heating effect of the warm air entering through the vents during the daytime.



Double Skin Façades

Background

A ventilated Double Skin Façade (DSF) can be defined as a traditional single façade doubled inside or outside by a second, essentially glazed façade. Each of these two façades is commonly called a skin . A ventilated cavity - having a width which can range from several centimetres at the narrowest to several metres for the widest accessible cavities - is located between these two skins. In some DSF designs the ventilation of the cavity is controllable by fans and/or openings, and other concepts where this ventilation is not controllable . The indoor and outdoor skins are not necessarily airtight (see, for example, the "louvre" type façades). Automated equipment, such as shading devices, motorised openings or fans, are most often integrated into the façade. The main difference between a ventilated double façade and an airtight multiple glazing, whether or not integrating a shading device in the cavity separating the panes, lies in the intentional and possibly controlled ventilation of the cavity of the double façade .

The standard prEN 13119, specifies the terminology associated with curtain walls and of a double-skin façade . These are defined as:

Curtain walling	External building façade produced with framing made mainly of metal, timber or PVC-U, usually consisting of vertical and horizontal structural members, connected together and anchored to the supporting structure of the building, which provides, by itself or in conjunction with the building construction, all the normal functions of an external wall, but does not contribute to the load bearing characteristics of the building structure.
Double-Skin façade	A curtain wall construction comprising an outer skin of glass and an inner wall constructed as a curtain wall

that together with the outer skin provide the full function of a wall.

This definition includes only references to the constructive and structural aspects of the façade. No reference is made to the ventilation of the cavity. Thus formally, the term "double-skin façade" does not designate a ventilated double façade. It is for this reason that the term 'Ventilated Double-Skin Façade' has been proposed as the generic term to designate these façades.

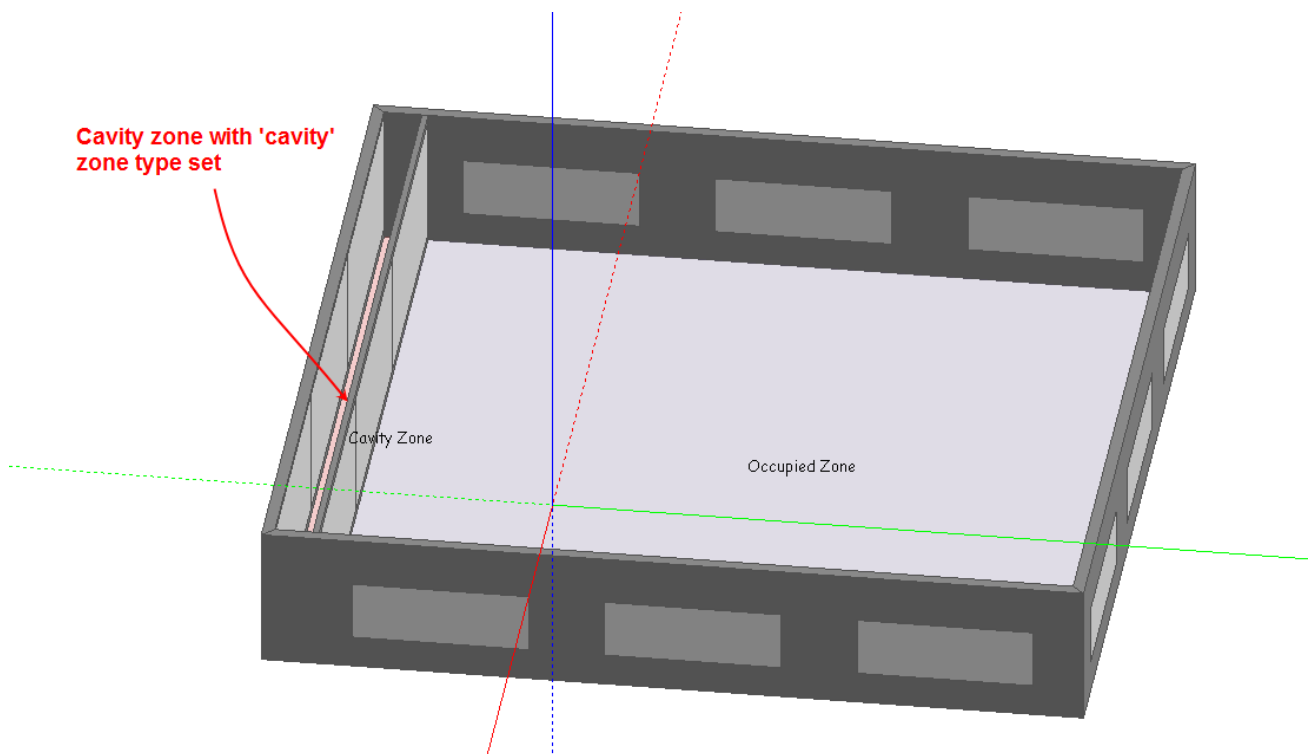
DSFs can be categorised according to the type of ventilation.

Type of ventilation	Name of the façade concept
Natural	Passive façade
Mechanical	Active façade
Hybrid	Interactive façade

This table contains the commonly used names referring (only) to the type of ventilation and thus giving no indication about the partitioning of the façade or the ventilation modes. In the literature, the double façades adopting the *air supply* ventilation mode are called *airflow window* or also *ventilated window*. The façades adopting the air exhaust ventilation mode are also called *extract air* or *exhaust airflow*.

Double Skin Façade Example

A simple example DSF model is provided with DesignBuilder (Double Skin Facade Example.dsb). A screenshot of the model at block level is shown below.



The model was created using a single block with a partition to divide it into 2 zones, one zone to represent the occupied space and the other for the ventilated cavity. The glazing for the partition was set up at surface level.

Modelling Considerations

There are some important considerations when using DesignBuilder to model double façades. Firstly, the [Zone type](#) on the [Activity tab](#) should be set to **3-Cavity** at zone level in the cavity zone and this causes the following further changes to be made to the model:

The zone is set as unoccupied by loading **<None>** Activity, HVAC and Lighting template data.

The **5-Cavity** [Internal convection algorithm](#) is set for Cooling design, Heating design and simulation calculations to correctly model the air space.

The **3-Full interior and exterior** [Solar distribution algorithm](#) is set for Cooling design and Simulation because it allows solar radiation to be accurately transmitted through the interior glazing in the partition.

In the example model a simple ventilation control system has been defined in the cavity zone to provide 10 ac/h of outside air into the cavity when the air temperature in the cavity is greater than 24°C. This helps to ensure that the cavity does not overheat, reducing cooling loads in the occupied zone. The schedule and setpoint could be adjusted to fine-tune the ventilation. The model assumes no transfer of cavity air into the occupied space. This could be modified using an approach similar to that in the [Trombe wall example](#) model where vents in the partition between occupied and cavity zones are used to transfer air between the 2 zones.

Finally, an [overhang](#) has been added to control the amount of summer sun exposure.

Because the model uses a zone to model the cavity in the double façade, the inner glazing is 'interior' and so requires the [Full interior and exterior](#) solar distribution option to allow solar radiation to pass through exterior and then through interior windows. You should read the information about the restrictions associated with this option before using it.

The **2-Pixel counting** [Shading calculation method](#) can be used with the [Full interior and exterior](#) solar distribution option to avoid the convex geometry requirement.

Take care when comparing the exterior solar radiation transmitted into the double façade zone with the interior solar radiation transmitted into the occupied zone. These do not include the re-reflection of solar back out of the external windows. Read the [definitions](#) for the output data.

Autosizing Heating And Cooling Equipment

Simple HVAC Autosizing

The [Heating Capacity](#) and [Cooling Capacity](#) Model Data is found on the [HVAC tab](#) at zone level when the [Plant sizing](#) building model option is set to **2-Manual** or **4-Autosize when not set**.

When the [Plant sizing](#) building model option is set to **4-Autosize when not set** these heating and cooling capacities can be automatically calculated by the software.

Note: if you change heating or cooling capacities by hand you should be aware that the data will be overwritten if [Model Options](#) change.

To keep the heating and cooling capacities fixed, change the [Plant sizing](#) model option to **2-Manual**. You may choose to use this option if you are already satisfied with the autosized capacities and wish to keep them as they are regardless of future changes to the model.

Detailed HVAC Autosizing

Winter and summer design day simulations are automatically carried out as required for autosizing heating and cooling equipment prior to a Detailed HVAC simulation. There are some considerations you should be aware of regarding autosizing.

For DOAS systems which introduce air into zones that is not fully preheated, there are 2 ways to ensure that zone equipment is adequately sized to meet the extra ventilation load:

The recommended way is to make appropriate settings on the HVAC zone dialog to [Account for Dedicated Outdoor Air System](#).

Change [Mechanical ventilation load](#) setting to **1-Met by zone equipment** or to **3-Partially met by AHU**.

Internal gains should normally be excluded from Winter design day calculations. You therefore shouldn't use the same schedule for equipment, occupancy and HVAC operation because equipment and occupancy should be off for heating autosizing and HVAC should be on.

Summer design day schedules are normally the same as the general ones so are easier to deal with.

If you are using [Compact schedules](#) then you should ensure that a specific line of **WinterDesignDay** data is included for all schedule periods.

If you are using [7/12 Schedules](#) then there are 2 options: **End-use defaults** and **Profiles**.

You can include the effect of intermittent operation on heating equipment sizes by using a time-varying Heating demand or HVAC schedule. This would normally provide heating equipment sizes with higher capacity than a steady-state calculation where the heating is on all day.

Daylight Saving And Local Time

DesignBuilder allows you to model the effect of daylight saving time where the clocks are moved forward one hour in the summer months in most non-equatorial countries. The timing of this is handled automatically using DesignBuilders' time zones database when you load data from one of the [Location templates](#) supplied with the program.

You can set the Time zone at the Site level on the [Location tab](#):



There are however some factors you should be aware of:

[Simulation results](#) are displayed in 'local mean time'. This means that during the summer, occupancy will appear to start 1 hour earlier in the day than it does in the winter. To help you pinpoint this, DesignBuilder prints the times when daylight saving starts and ends on the graph.

[Visualisations](#) use local mean time when setting the position of the sun for site shading analysis. Again you should add 1 hour to the time to see shading for local time.

Solar Radiation Distribution

This topic summarises the methods used in calculation of solar radiation in EnergyPlus.

Sky Radiance Model

In EnergyPlus the calculation of diffuse solar radiation from the sky incident on an exterior surface takes into account the anisotropic (non-uniform) radiance distribution of the sky. For this distribution, the diffuse sky irradiance on a surface is given by:

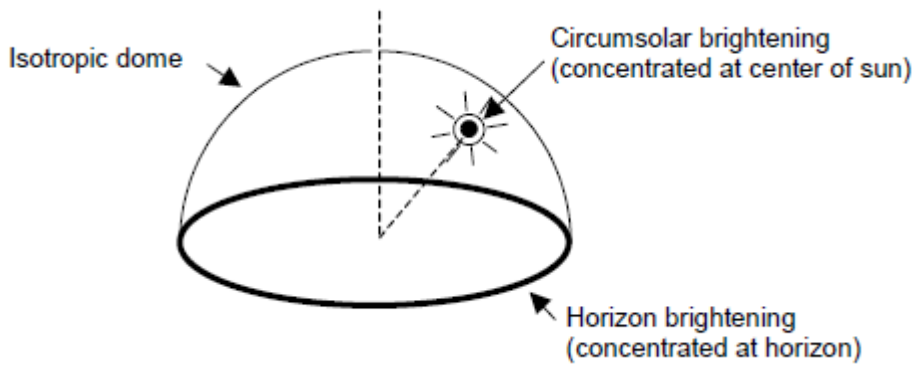
$$AnisoSkyMult * DifSolarRad$$

where:

DifSolarRad is the diffuse solar irradiance from the sky on the ground and *AnisoSkyMult* is determined by surface orientation and sky radiance distribution, and accounts for the effects of shading of sky diffuse radiation by shadowing surfaces such as overhangs. It does not account for reflection of sky diffuse radiation from shadowing surfaces.

The sky radiance distribution is based on an empirical model based on radiance measurements of real skies, as described in Perez et al., 1990. In this model the radiance of the sky is determined by three distributions that are superimposed (see Figure 33)

- An isotropic distribution that covers the entire sky dome;
- A circumsolar brightening centered at the position of the sun;
- A horizon brightening.



Schematic view of sky showing solar radiance distribution as a superposition of three components: dome with isotropic radiance, circumsolar brightening represented as a point source at the sun, and horizon brightening represented as a line source at the horizon.

The proportions of these distributions depend on the sky condition, which is characterized by two quantities, clearness factor and brightness factor, defined below, which are determined from sun position and solar quantities from the weather file.

The **circumsolar brightening** is assumed to be concentrated at a point source at the center of the sun although this region actually begins at the periphery of the solar disk and falls off in intensity with increasing angular distance from the periphery.

The **horizon brightening** is assumed to be a linear source at the horizon and to be independent of azimuth. In reality, for clear skies, the horizon brightening is highest at the horizon and decreases in intensity away from the horizon. For overcast skies the horizon brightening has a negative value since for such skies the sky radiance increases rather than decreases away from the horizon.

Shadowing of Sky Long-Wave Radiation

EnergyPlus calculates the sky long-wave radiation incident on exterior surfaces assuming that the sky long-wave radiance distribution is isotropic. If obstructions such as overhangs are present the sky long-wave incident on a surface is multiplied by an isotropic shading factor. The long-wave radiation from these obstructions is added to the long-wave radiation from the ground; in this calculation both obstructions and ground are assumed to be at the outside air temperature and to have an emissivity of 0.9.

Solar Gains

The total solar gain on any exterior surface is a combination of the absorption of direct and diffuse solar radiation given by

$$Q_{so} = \alpha \cdot \left(I_b \cdot \cos \theta \cdot \frac{S_s}{S} + I_s \cdot F_{ss} + I_g \cdot F_{sg} \right)$$

where:

α = solar absorptance of the surface

θ = angle of incidence of the sun's rays

S = area of the surface

S_s = sunlit area

I_b = intensity of beam (direct) radiation

I_s = intensity of sky diffuse radiation

I_g = intensity of ground reflected diffuse radiation

F_{ss} = angle factor between the surface and the sky

F_{sg} = angle factor between the surface and the ground

If the surface is shaded the program modifies F_{ss} by a correction factor that takes into account the radiance distribution of the sky (see "Shadowing of Sky Diffuse Solar Radiation"). Shading of ground diffuse solar radiation is not calculated by the program. It is up to the user to estimate the effect of this shading and modify the input value of F_{sg} accordingly.

Solar Distribution

As discussed in the Solar Model options section, the [Solar Distribution](#) determines how EnergyPlus will treat beam solar radiation entering a zone through exterior windows. The three choices: **1-Minimal shadowing**, **2-Full exterior** and **3-Full Interior and exterior** are discussed in the [Solar options](#) topic.

EnergyPlus calculates the distribution of short-wave radiation in the interior of each thermal zone. This radiation consists of beam solar radiation, diffuse solar radiation, and short-wave radiation from electric lights. The program determines the amount of this radiation that is (1) absorbed on the inside face of opaque surfaces, (2) absorbed in the glass and shading device layers of the zone's exterior and interior windows, (3) transmitted through the zone's interior windows to adjacent zones, and (4) transmitted back out of the exterior windows. The effects of movable shading devices on the exterior windows are taken into account.

Initial Distribution of Diffuse Solar Transmitted through Exterior and Interior Windows

The treatment of diffuse solar transmitted first through exterior windows and subsequently through interior windows has been improved. Diffuse solar (from sky and ground sources) transmitted through exterior windows is first distributed to the interior heat transfer surfaces in the zone containing the exterior windows. This initial distribution apportions the transmitted diffuse solar to interior surfaces using the approximate view factors described above in "LW Radiation Exchange Among Zone Surfaces." The amount of this initially distributed diffuse solar absorbed by each interior surface, and each window material layer, is calculated and later added to the "short-wave radiation absorbed" values described below. The amount of this initially distributed diffuse solar that is reflected is accumulated for each zone and redistributed uniformly to the other surfaces. The amount of this initially distributed diffuse solar that is transmitted by interior windows to adjacent zones is initially distributed to the interior heat transfer surfaces in the adjacent zone in the same manner as just described.

This new treatment of diffuse solar is intended to more accurately account for the initial absorption, transmittance, and reflection of short-wave radiation prior to the uniform distribution as described in the [Engineering reference](#).

Ground surface reflectance

[Ground reflectance](#) values are used to calculate the ground reflected solar amount. This fractional amount (entered monthly) is used in the following equation:

$$\text{GroundReflectedSolar} = (\text{BeamSolar} \times \text{COS}(\text{SunZenithAngle}) + \text{DiffuseSolar}) \times \text{GroundReflectance}$$

The Ground Reflected Solar is never allowed to be negative. The Snow Ground Reflectance Modifier can further modify the ground reflectance when snow is on the ground. If the user enters 0.0 for each month, no ground reflected solar is used.

Ground Reflectances (Snow)

When [snow](#) is on the ground, ground reflectances may change. The user can specify two values, [Snow reflected solar modifier](#) and [Snow reflected daylight modifier](#).

Snow reflected solar modifier

A number between 0.0 and 10.0 which is used to modify the basic ground surface reflectance when snow is on the ground. Note that the value of $\text{GroundReflectance}_{\text{Used}}$ (below) must be ≤ 1 .

$$\text{GroundReflectance}_{\text{Used}} = \text{GroundReflectance} \times \text{Modifier}_{\text{Snow}}$$

During simulations, the ground is considered to be snow-covered when the SnowDepth data in the hourly weather file is > 0 .

Snow reflected daylight modifier

A number between 0.0 and 10.0 which is used to modify the basic ground surface reflectance when snow is on the ground. Note that the value of $\text{DaylightingGroundReflectance}_{\text{Used}}$ (below) must be ≤ 1 .

$\text{DaylightingGroundReflectance}_{\text{Used}} = \text{GroundReflectance} \times \text{Modifier}_{\text{Snow}}$

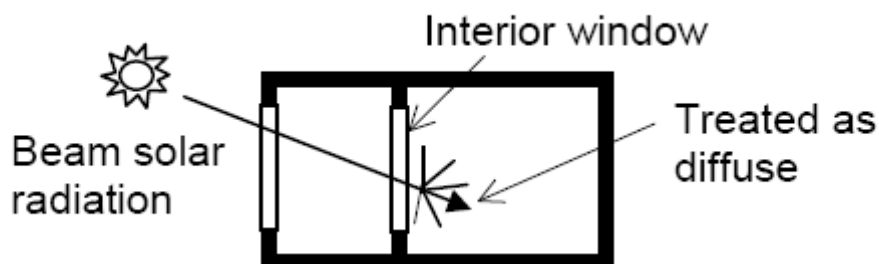
During simulations, the ground is considered to be snow-covered when the SnowDepth data in the hourly weather file is > 0.

Beam distribution

If the Solar Distribution option is **Minimal shadowing** or **Full exterior**, it is assumed that all beam solar from exterior windows falls on the floor. If Solar Distribution is **Full Interior and Exterior** the program tracks where beam solar from exterior windows falls inside the zone, and wall as well as floor surfaces can receive beam radiation.

Diffuse distribution

Diffuse solar transmitted through exterior and interior windows is distributed according to the approximate view factors between the transmitting window and all other heat transfer surfaces in the zone. This variable is the amount of transmitted diffuse solar that is initially absorbed on the inside of each heat transfer surface. The portion of this diffuse solar that is reflected by all surfaces in the zone is subsequently redistributed uniformly to all heat transfer surfaces in the zone, along with interior reflected beam solar and shortwave radiation from lights. The total absorbed shortwave radiation is given by the next variable.



Solar radiation reflected from exterior surfaces

EnergyPlus has an [option](#) to calculate beam and sky solar radiation that is reflected from exterior surfaces and then strikes the building. For zones with detailed daylighting, these reflections are also considered in the daylight illuminance calculations. The reflecting surfaces fall into three categories:

Shadowing surfaces. These are surfaces like overhangs, component blocks or surfaces of the building. Examples are shown in [Solar Options](#). These surfaces can have diffuse and/or specular (beam-to-beam) reflectance values that are specified with the based on the [material surface properties](#).

Exterior building surfaces. In this case one section of the building reflects solar radiation onto another section (and vice-versa). See Figure 47. Opaque building surfaces (walls, for example) are assumed to be diffusely reflecting. Windows and glass doors are assumed to be specularly reflecting. The reflectance values for opaque surfaces are calculated by the program from the Absorptance:Solar and Absorptance:Visible values of the outer material layer of the surface's construction. The reflectance values for windows and glass doors are calculated by the program from the reflectance properties of the individual glass layers that make up surface's construction assuming no shading device is present and taking into account interreflections among the layers.

The ground surface. Beam solar and sky solar reflection from the ground is calculated even if the Reflections option is not used (the default). But in this case the ground plane is considered unobstructed, i.e., the shadowing of the ground by the building itself or by obstructions such as component blocks is ignored. This shadowing is taken into account only if the Reflections option is used. In this case the ground view factor calculated by DesignBuilder is not used.

See [Solar Options](#),

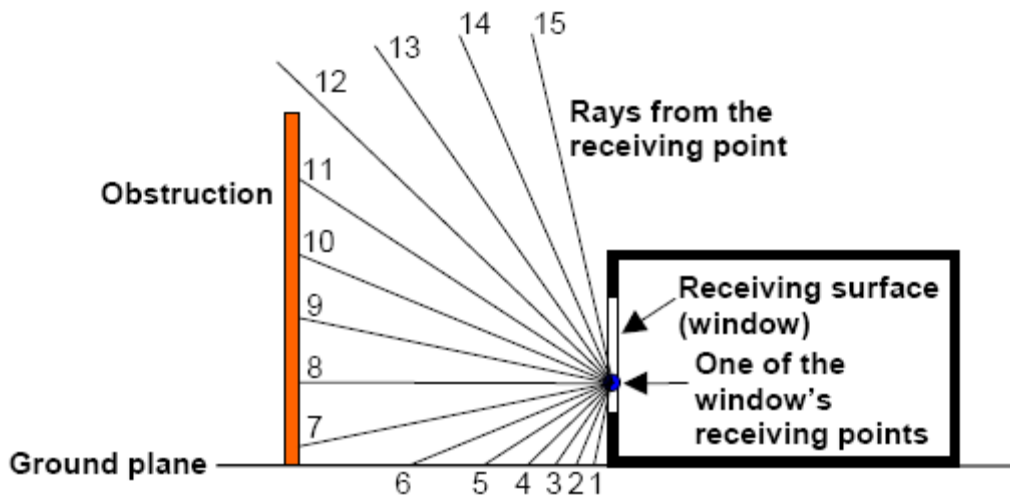
Diffuse reflection of beam solar and sky solar radiation

A ray-tracing method is used to calculate beam solar and sky solar radiation that is diffusely reflected onto each of a building's exterior surfaces (walls, roofs, windows and doors), called here "receiving surfaces". The calculation begins by generating a set of rays proceeding into the outward hemisphere at each receiving point on a receiving surface. Then it determines whether each ray hits the sky, ground or an obstruction. The

radiance at the hit point from reflection of incident beam or sky solar is determined and the contribution of this radiance to the receiving surface is calculated, added to the contribution from other hit points, and averaged over the receiving points. Separate calculations are done for beam-to-diffuse and sky solar reflection from all obstructions and beam-to-diffuse and sky solar reflection from the ground. (For beam-to-beam reflection see [Beam solar radiation specularly reflected from obstructions](#) below.)

Rays

A total of 90 rays are sent out into the exterior hemisphere surrounding each receiving point. An upgoing ray may hit an obstruction or the sky. A downgoing ray may hit an obstruction or the ground. See diagram below.



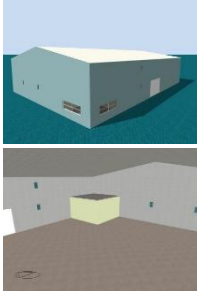
Two-dimensional schematic showing rays going outward from a point on a receiving surface. Rays 1-6 hit the ground, rays 7-11 hit an obstruction, and rays 12-15 hit the sky.

Sky Solar Radiation Diffusely Reflected from the Ground

If a downgoing ray from a receiving point hits the ground (for example, rays 1-6 in diagram below), the program calculates the radiance at the ground hit point due to sky diffuse solar reaching that point. To do this, rays are sent upward from the ground hit point and it is determined which of these rays are unobstructed and so go to the sky (for example, rays 6-10 below). For this calculation it is assumed that the radiance of the sky is uniform.

Warehouse With Office

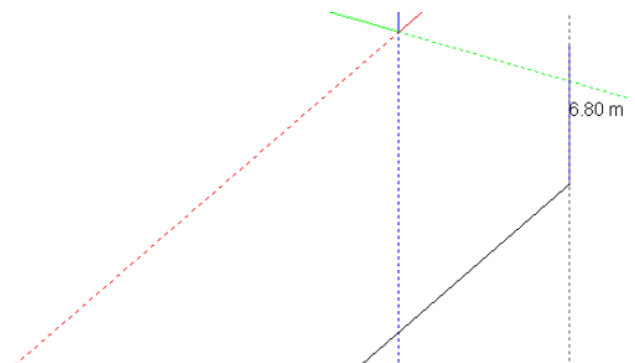
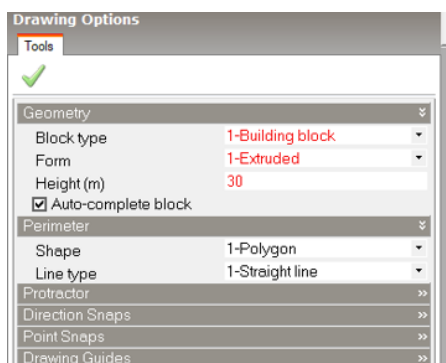
Sometimes it is necessary to create a smaller zone such as an office within a larger zone (e.g. a large warehouse space). This tutorial shows 2 different ways to set up a simple 2-zone model such as that illustrated in the screenshots below where a small office is located in bottom corner of a larger warehouse space.

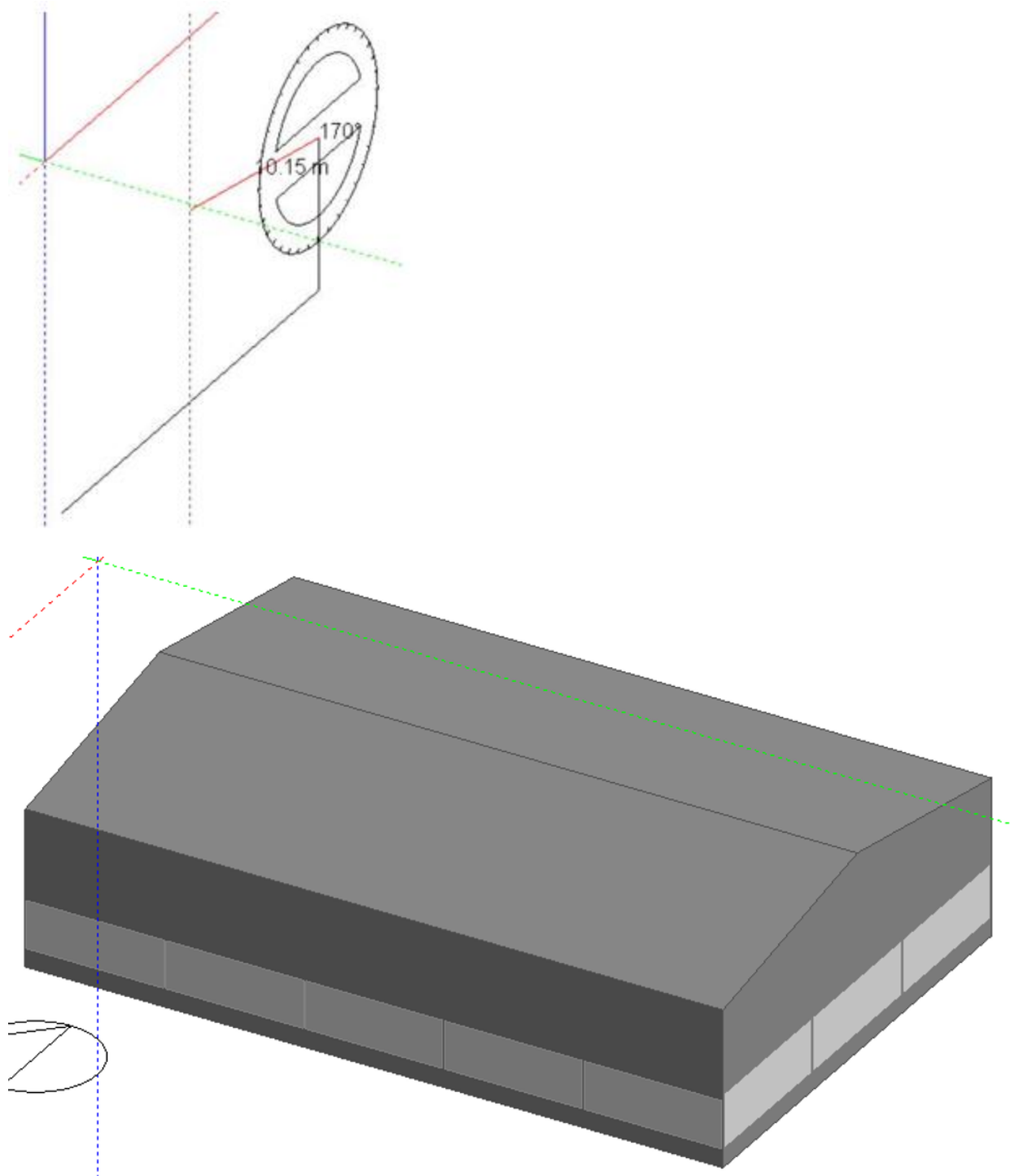


Method 1 - Using the Add Surface Tool

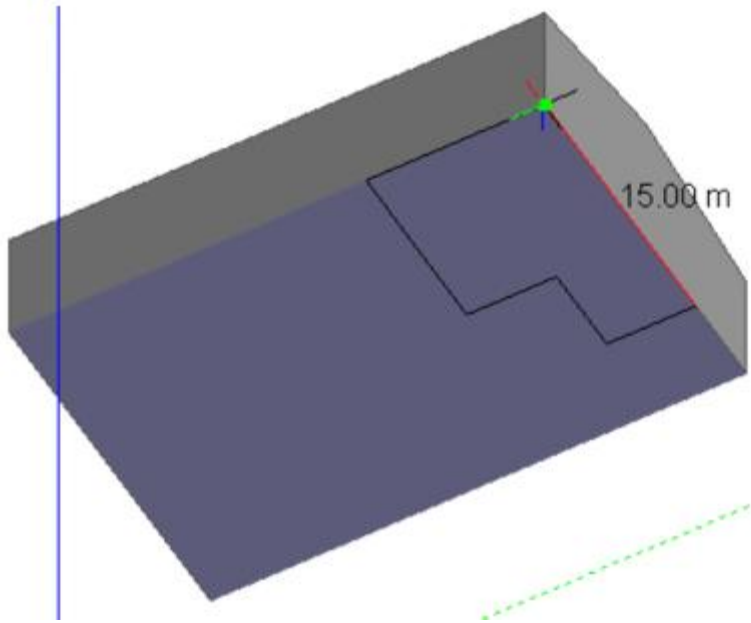
In general this type of building can most easily be drawn using the [Add Surface](#) tool to remove the single height block from the double height space of the warehouse. This method avoids the need to draw a tall space using multiple blocks and merging the blocks using holes drawn between them (see Method 2 below). This section shows how to do this using two blocks one representing the complete building, the other representing the single height parts of the building.

The main warehouse block may be created by extruding from a horizontal or vertical perimeter, depending on whether the plan or the gable surface is more complex. For a rectangular plan it is generally easiest to draw a vertical perimeter and extrude horizontally as it allows any pitched or curve roofs to be produced without multiple operations. The easiest way to draw a vertical perimeter is to draw the horizontal base line drawn first and then snapping the second vertical perimeter line to the vertical axis. Protractor and centre snap tools help when drawing a simple pitched roof in this way.

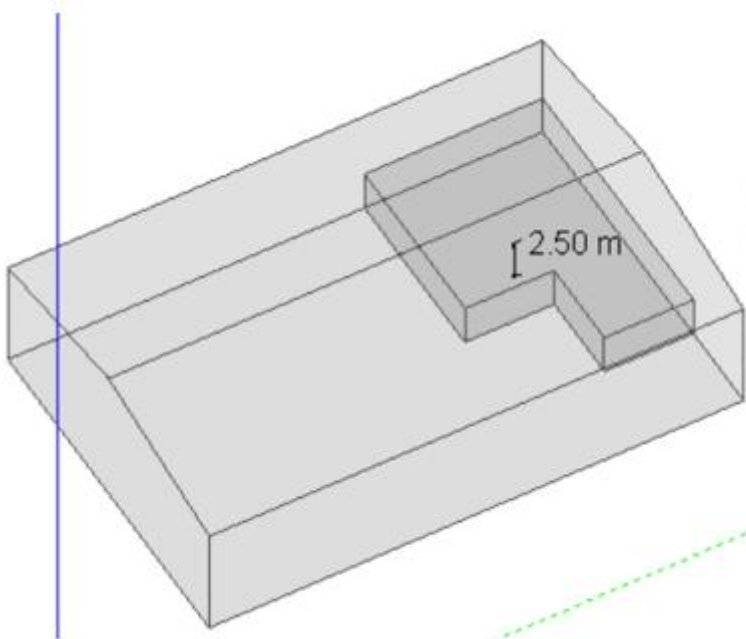




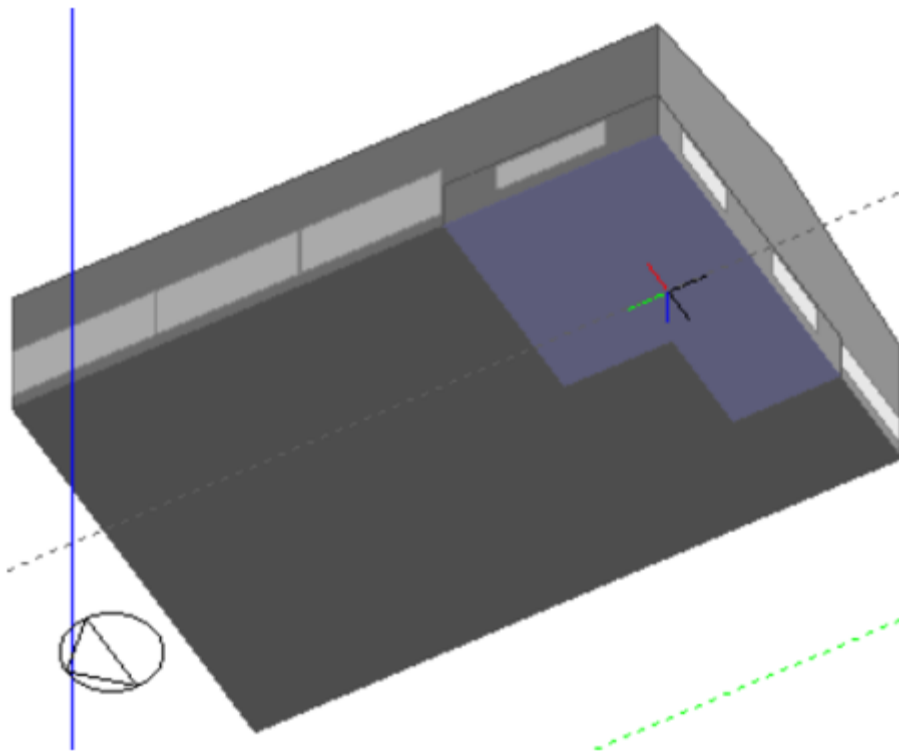
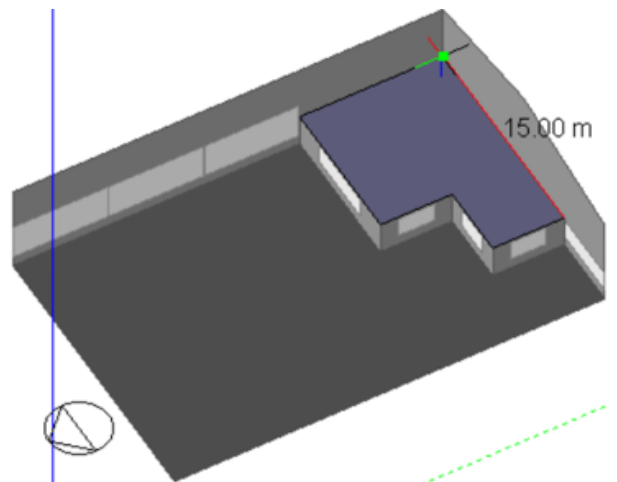
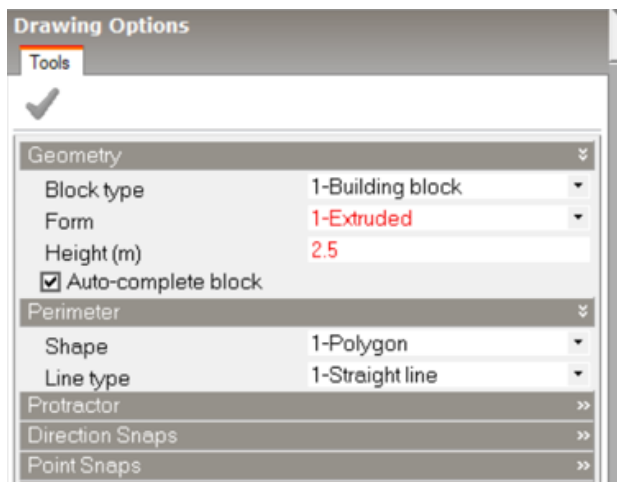
The added surface (really a removed section) will correspond with the geometry of the single height spaces and is best drawn with its perimeter on the base of the complete block if the perimeter is polygonal. Click on the [Add Surface](#) tool and draw the perimeter.



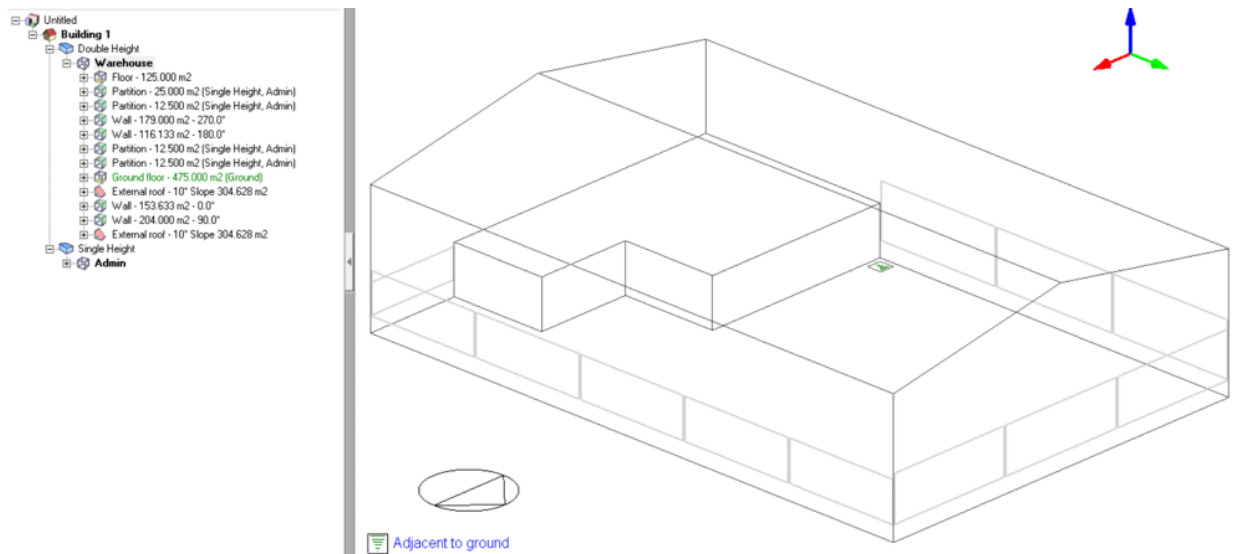
When complete, drag upwards to define the height, entering the height manually or using increment snap.



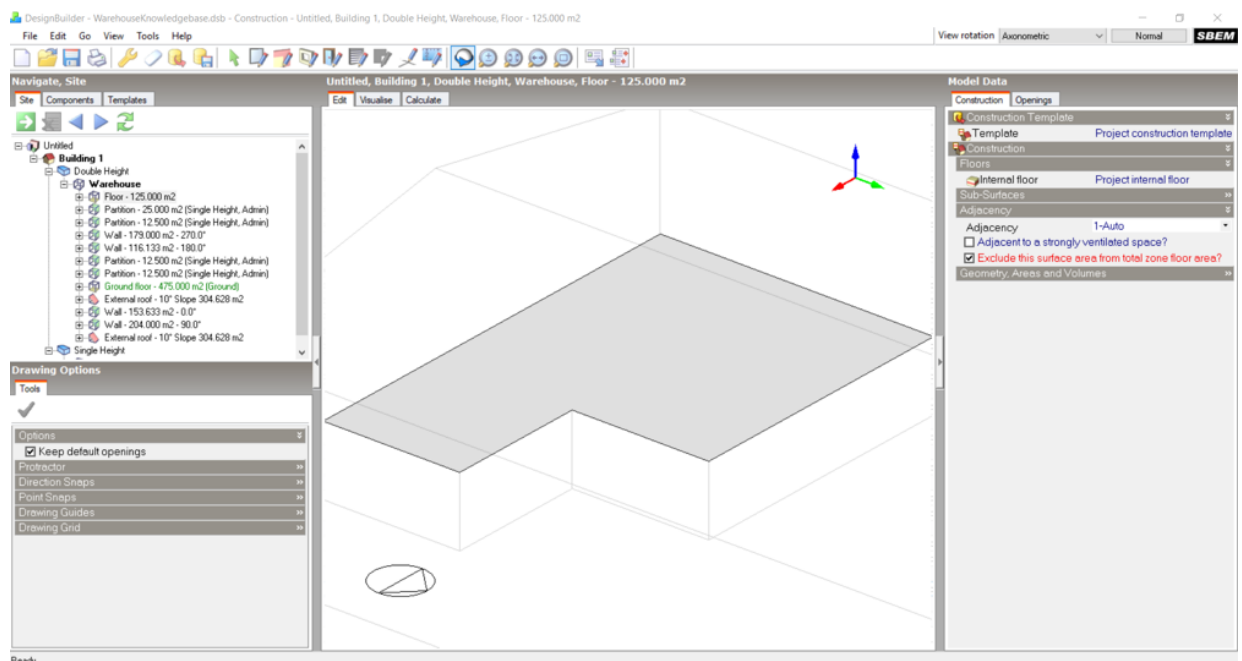
The surface that has been removed needs to be replaced by a separate block. Make sure that the base is visible and start the [Add Block](#) command.



The main geometry is now complete.



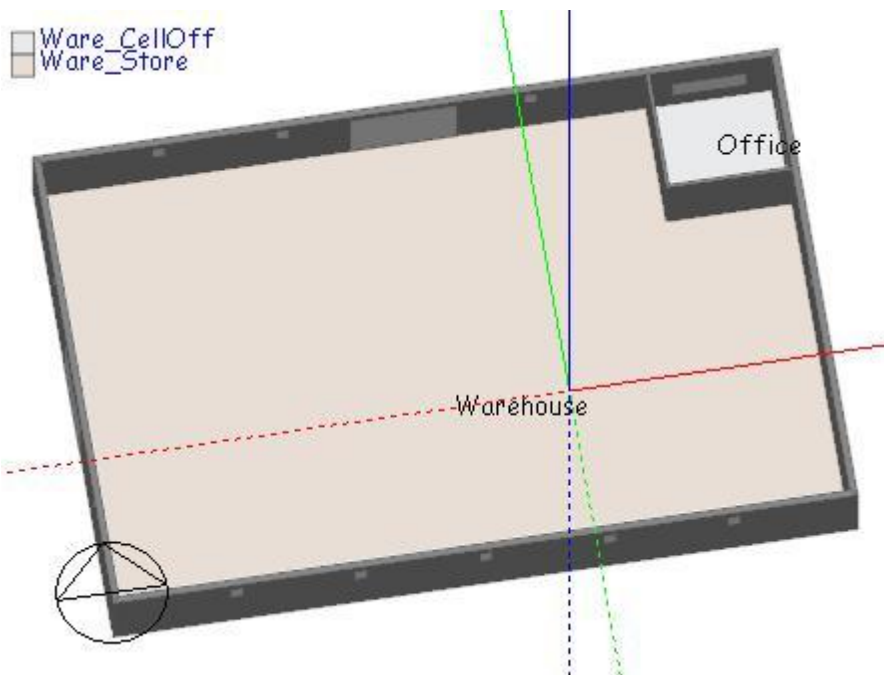
If the floor above the office space is not accessible space then navigate to that floor element and on the construction tab select **Exclude this surface from total zone area** see screenshot below. If the floor is accessible then you should draw virtual partitions around it's perimeter.



Method 2 - Using Merge Zones Connected by Holes

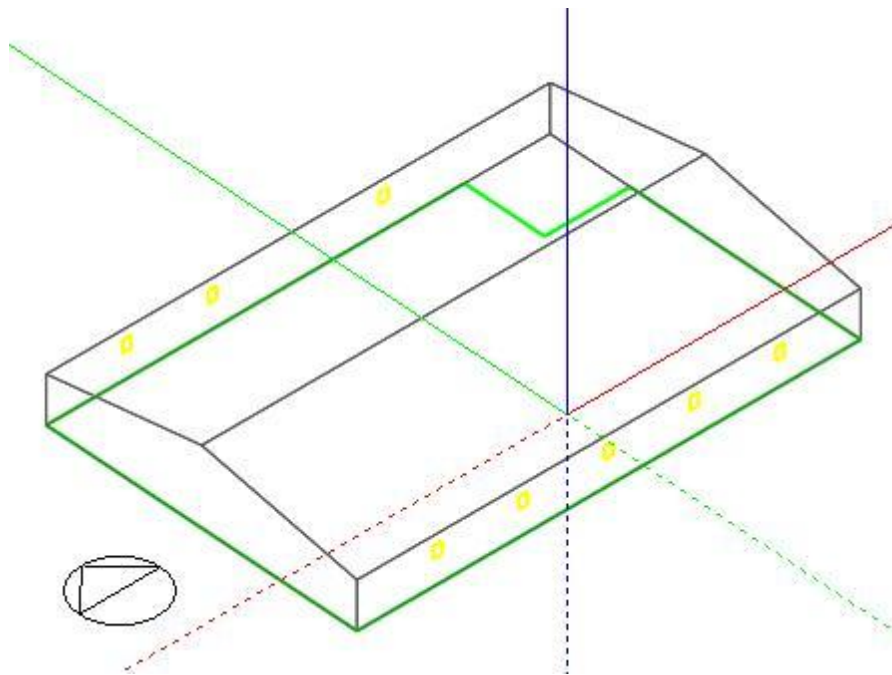
The steps involved are:

1. Draw the ground floor block including both office and warehouse. The block should be the height of the office space as the top of the block will form the ceiling of the office.



2. Draw the 1st floor block representing the high-level warehouse space.

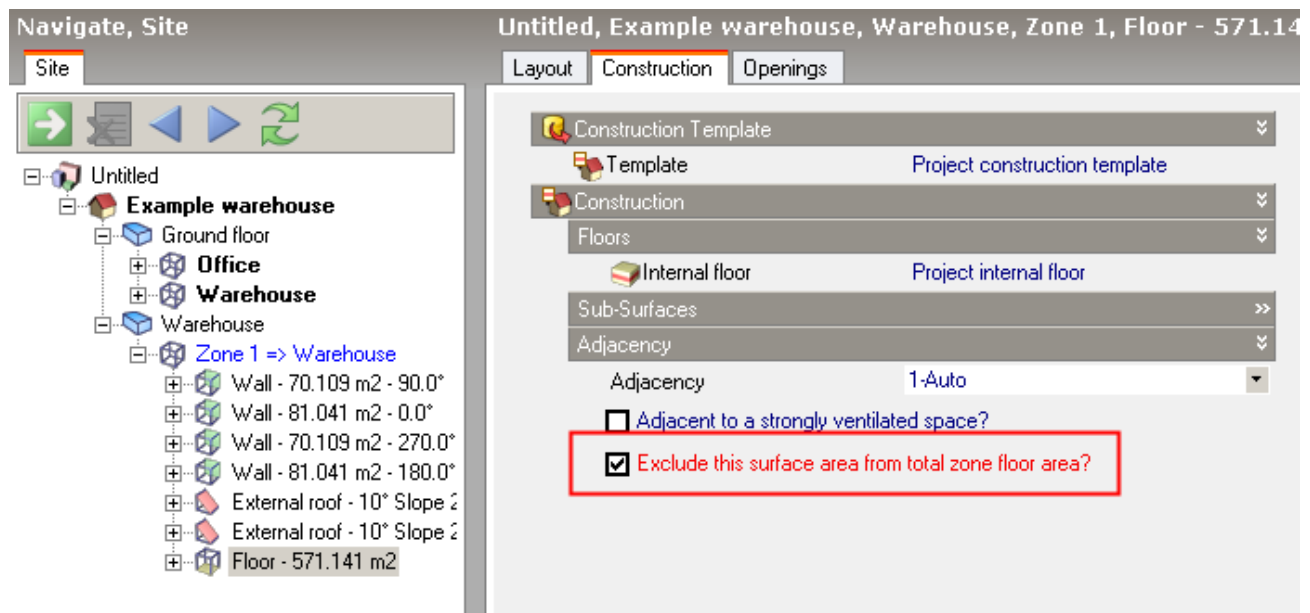
3. [Draw a hole](#) in the floor of the 1st floor block to connect the upper and lower warehouse zones together. The upper warehouse zone should look something like the screenshot below. The green line shows the perimeter of the hole.



4. Select the [Merge zones connected by holes](#) model option to merge the upper and lower parts of the warehouse into a single zone.

5. The 'floor' on top of the office zone is not usable floor area so go to this floor surface and select the [Exclude this surface from total zone floor area](#) option. This indicates that the surface is not part of the building useable floor area. This step ensures that this floor area is not counted in the floor area provided to certification tools such as SBEM. This step is not currently required for EnergyPlus simulations.

Note: if the excluded floor surface contains some external adjacencies as well as internal ones these **external areas will not be subtracted from the zone floor area.**



So the building now consists of 2 zones - an Office and a Warehouse zone. This model is available as one of the DesignBuilder example templates.

The process involved with merging zones connected by holes is illustrated in the [Drawing holes and merging zones tutorial](#)

Start Up And Pre-conditioning

Start up (Warmup in EnergyPlus terminology or pre-conditioning) is the process of repeatedly simulating the first day before the simulation proper starts to ensure that the temperatures in the building fabric are realistic. Warmup continues until temperatures and heat flows in each zone have converged. If convergence does not occur then simulation continues for the maximum number of days as specified in the calculation options.

Warmup And Starting Simulations On A Weekend

Preconditioning of the building by repeated simulation of the first day until convergence is an important part of the simulation but it can cause issues if the first day in the simulation is extreme in any way. For example if the first day in the simulation is a Sunday and the building is not heated or cooled at all on that day and extreme overheating occurs then the fabric of the will be "charged" up with heat as if those conditions had been occurring repeatedly for many days (or even weeks in some heavyweight buildings). This can mean that on the next day, the Monday the building model is far more liable to overheat than would have been the case if the simulation had started on a different day. In cases where this may be an issue, it may be worth making sure that simulations start on an occupied day rather than over a weekend or holiday.

EnergyPlus Background Information

EnergyPlus is the U.S. DOE building energy simulation program for modelling building heating, cooling, lighting, ventilating, and other energy flows. It builds on the most popular features and capabilities of BLAST and DOE-2 but also includes many innovative simulation capabilities such as time steps of less than an hour, modular systems and plant integrated with heat balance-based zone simulation, multizone air flow, thermal comfort, and photovoltaic systems.

EnergyPlus is a stand-alone simulation program without a 'user friendly' graphical interface – which is where DesignBuilder comes in. We have integrated EnergyPlus tightly within the DesignBuilder environment to allow you to carry out simulations without any fuss – just define your building model, request data and let the EnergyPlus simulation engine take care of the details.

DesignBuilder has been specifically developed around EnergyPlus allowing most of the EnergyPlus fabric and glazing data to be input. Databases of building materials, constructions, window panes, window gas, glazing units and blinds are provided.

You can use DesignBuilder to [generate IDF files](#) and work with these outside DesignBuilder to access system functionality not provided by DesignBuilder.

There is a range of EnergyPlus simulators to choose from including the current DOE executable release, the current DLL release and any interim releases made available by DOE. You can carry out DesignBuilder simulations via [EP-Launch](#).

See also:

- [EnergyPlus Documentation](#)
- [Exporting EnergyPlus IDF Files](#)
- [EnergyPlus Version Compatibility](#)
- [EnergyPlus IDF Window Dimensions](#)
- [Calculation of DesignBuilder Output from EnergyPlus Report Variables](#)
- [EnergyPlus Errors and Warnings](#)
- [EnergyPlus Daylight Calculations](#)
- [EnergyPlus Thermal Comfort](#)
- [EnergyPlus Date Definition](#)
- [When DesignBuilder output is different from unprocessed EnergyPlus Output](#)
- [EnergyPlus Daylight Map Output](#)
- "EnergyPlus Curve Fit Tool" on page 923
- [Results Viewer](#)
- "How to Generate and View Extra EnergyPlus Outputs" on page 2205

EnergyPlus Documentation

You can download all of the EnergyPlus documentation at no cost from:

<https://energyplus.net/documentation>

Exporting IDF Files

You can use DesignBuilder to generate [EnergyPlus idf files](#) for use in external simulation environments such as EP-Launch. There are 3 alternative procedures for doing this:

- From the File > Export menu option (*preferred method*).
- By copying the intermediate in.idf file in the EnergyPlus folder.
- Tools > Display simulation input script from Simulation or Heating/Cooling design screens.

DesignBuilder will automatically generate [compatible IDF files](#) with the target version of EnergyPlus as set on the [Program options dialog](#).

Note: It is important to set the [EnergyPlus version](#) you intend to use before exporting the IDF data.

1. File > Export menu option (recommended)

Open your DesignBuilder dsb model file.

In the Program options dialog on the EnergyPlus tab, ensure that the recommended executable DOE version of the most current version of EnergyPlus available on the list is selected (e.g. "Version 7.0.0.036 (Recommended DOE)").

Make sure that you have that version of EnergyPlus downloaded and installed from the DOE website.

Make sure you are at building level or below.

Click on the **File > Export > Export EnergyPlus or DBSim input file > Simulation** menu option.

2. Copy in.idf in the EnergyPlus folder

An alternative way to generate IDF files is to use the intermediate file [in.idf](#) in the [EnergyPlus folder](#) which is generated prior to each DesignBuilder EnergyPlus simulation. You should copy this file to a different filename/location before using it.

Important: You should not generally make any changes to **in.idf** itself because this file will get overwritten before the next DesignBuilder EnergyPlus simulation and your changes will then be lost.

3. Tools > Display menu option

You can view the input data script used to calculate these results by selecting the Tools > Display simulation input script menu option. If you do not have the EnergyPlus IDF editor installed, then you should associate the [.idf file extension](#) with a text editor to make this option work.

Note: this shows the script stored with the model just before the simulation *not* the [in.idf](#) file stored in the [EnergyPlus folder](#).

Technical Details

Geometry is exported to IDF using the following IDF object types:

Building surfaces as BuildingSurface:Detailed

Windows as FenestrationSurface:Detailed

Local shading as Shading:Building:Detailed or Shading:Site:Detailed

Site context surfaces ([component](#) and [assembly](#) blocks) as Shading:Site:Detailed or

Shading:Building:Detailed

Reading ESO Results Files Back Into DesignBuilder

Provided you haven't changed the zone and surface names in the IDF data you can read [ESO](#) results files generated from your modified data back into DesignBuilder for display and analysis.

Tip: If you are already familiar with the EnergyPlus data structure you may find it most convenient to edit DesignBuilder IDF data using a text editor as it shows more clearly the comments included to help reference IDF data back to the DesignBuilder model.

EnergyPlus Version Compatibility

You should ensure that you have the correct version of EnergyPlus set before [exporting IDF](#) data for use outside DesignBuilder.

DesignBuilder will automatically generate compatible IDF files with the target version of EnergyPlus as set on the EnergyPlus tab [Program options dialog](#). If you are [generating IDF files](#) for simulation outside DesignBuilder you should note that, by default, DesignBuilder targets an internal version of EnergyPlus which has an extended IDD data definition file to allow:

Surfaces of up to 330 vertices.
Shading surfaces of up to 33 vertices.

So if you attempt to use IDF files generated for internal versions of EnergyPlus in a version of EnergyPlus supplied by DOE, you may find some surface or HVAC data errors related to the above changes. To avoid this situation you should either:

Explicitly target your DOE EnergyPlus installation by making the appropriate setting on the [Program options dialog](#) (preferred option), or
Carry the DesignBuilder IDD extensions over to your standalone version of EnergyPlus by copying Energy+.idd from the DesignBuilder EnergyPlus folder (use the File > Folder > EnergyPlus folder menu command) to your standalone EnergyPlus folder (e.g. C:\EnergyPlusV2-0-0).

EnergyPlus IDF Window Dimensions

The window dimensions supplied to EnergyPlus are the total opening size (window size including the frame), but deflated by an amount equal to the width of the window frame. The deflation is required because EnergyPlus expects window dimensions *excluding* the frame whereas DesignBuilder openings *include* the frame. DesignBuilder takes this approach because it is usually more convenient to enter the size of the opening in the fabric.

The default window areas generated by DB should correspond with the [Window to wall % \(WWR\)](#) setting on the Openings tab, provided you are using either **Continuous horizontal** or **Preferred height** facade types.

Tip: switching off frames on the Openings tab causes EnergyPlus windows of the expected dimensions to be generated given the simplistic 'window area divided by total wall area' calculation.

Calculation Of DesignBuilder Output From EnergyPlus Report Variables

At the end of an EnergyPlus simulation DesignBuilder reads the EnergyPlus output file [eplusout.eso](#). The format of this file is described in EnergyPlus documentation. The table below explains how the raw EnergyPlus output is processed to create DesignBuilder output. The **Environmental/Comfort Output** column refers to the data shown in [DesignBuilder output](#) screens. EnergyPlus Report variable column data shows report variable names with the units in square brackets. The units should be omitted when using the report variables in IDF data. The **Area** column indicates the zone, surface, component etc.

Environmental/ Comfort Output	EnergyPlus Report Variable	Area	Notes*
Internal air temperature	Zone Mean Air Temperature [C]	Zoneld	
Internal radiant temperature	Zone Mean Radiant Temperature [C]	Zoneld	

Internal operative temperature	Zone Operative Temperature [C]	Zoneld	
Relative Humidity	Zone Air Relative Humidity [%]	Zoneld	
Fanger PMV	FangerPMV	PEOPLE Zoneld	
Pierce PMV ET	PiercePMVET	PEOPLE Zoneld	
Pierce PMV SET	PiercePMVSET	PEOPLE Zoneld	
Pierce Discomfort Index (DISC)	PierceDISC	PEOPLE Zoneld	
Pierce Thermal Sens. Index (TSENS)	PierceTSENS	PEOPLE Zoneld	
Kansas Uni TSV	KsuTSV	PEOPLE Zoneld	
Discomfort hrs (summer clothing)	Time Not Comfortable Summer Clothes [hr]	PEOPLE Zoneld	
Discomfort hrs (winter clothing)	Time Not Comfortable Winter Clothes [hr]	PEOPLE Zoneld	
Discomfort hrs (all clothing)	Time Not Comfortable Summer or Winter Clothes [hr]	PEOPLE Zoneld	

Fabric and ventilation

Glazing heat gain	Window Heat Gain [W] - Window Heat Loss [W] - Window Transmitted Solar [W]	Surfaceld	Window Transmitted Solar is subtracted so that Glazing heat gain represents the transfer of all heat through the window <i>excluding beam and diffuse short-wave solar heat</i> effects. This makes it more comparable with the opaque surface conduction results. NB The subtraction of solar from glazing heat gain is done as a <i>post process</i> on the output results and does not affect the simulation itself.
Walls, Roofs, Ceilings (int), Floors (int), Floors (ext), Ground floors, Partitions (int), Doors and vents	Surface Inside Face Conduction Heat Transfer Rate [W]	Surfaceld	
External infiltration	Zone Infiltration Sensible Heat Loss [J] + Zone Infiltration Sensible Heat Gain [J]	Zoneld	Scheduled Natural ventilation only
External natural ventilation	Zone Ventilation Sensible Heat Gain [J] + Zone Ventilation Sensible Heat Loss [J]	Zoneld	Scheduled Natural ventilation only
External mechanical ventilation	Zone Ventilation Sensible Heat Gain [J] + Zone Ventilation Sensible Heat Loss [J]	Zoneld	Scheduled Natural ventilation only and only when the Mechanical ventilation method model option is set to 1-Room ventilation

Internal Air	AirflowNetwork Mixing Sensible Loss Rate [W] + AirflowNetwork Mixing Sensible Gain Rate [W]	Zoneld	Calculated Natural ventilation only
External Air	AirflowNetwork Infiltration Sensible Loss Rate [W]+ AirflowNetwork Infiltration Sensible Gain Rate [W]	Zoneld	Calculated Natural ventilation only
Airflow			
Mech Vent + Nat Vent + Infiltration	Zone Mechanical Ventilation Air Change Rate [ach]	Zoneld	Summed across blocks and building using volume-weighted average
<i>Calculated as the sum of the 3 EnergyPlus reports in the next column</i>	Zone Ventilation Air Change Rate [ach]	Zoneld	Summed across blocks and building using volume-weighted average
	Zone Infiltration Air Change Rate [ach]	Zoneld	Summed across blocks and building using volume-weighted average
Internal Gains			
Task Lighting	ELECTRIC EQUIPMENT#Zoneld#TaskLights* [J]	-	
General Lighting	ELECTRIC EQUIPMENT#Zoneld#GeneralLights* [J]	-	The 'GeneralLights' ELECTRIC EQUIPMENT EnergyPlus output is a lighting <i>fuel</i> consumption and is summed to give the Lighting fuel consumption (below). The gain to the zone is calculated from this in each zone by multiplying by (1 - Return air fraction).
Miscellaneous	ELECTRIC EQUIPMENT#Zoneld#02:InteriorEquipment* [J]	-	The '02' ELECTRIC EQUIPMENT EnergyPlus output is the miscellaneous internal gains fuel consumption and is summed to give the Room fuel consumption (below). The gain to the zone is calculated from this in each zone by multiplying by (1 - Fraction lost).
Process	ELECTRIC EQUIPMENT#Zoneld#03:InteriorEquipment* [J]	-	The '03' ELECTRIC EQUIPMENT EnergyPlus output is the process internal gains fuel consumption and is summed to give the Room fuel consumption (below). The gain to the zone is calculated from this in each zone by multiplying by (1 - Fraction lost).
Catering	ELECTRIC EQUIPMENT#Zoneld#04:InteriorEquipment* [J]	-	The '04' ELECTRIC EQUIPMENT EnergyPlus output is the catering

			internal gains fuel consumption and is summed to give the Room fuel consumption (below). The gain to the zone is calculated from this in each zone by multiplying by (1 - Fraction lost).
Computer + Equip	ELECTRIC EQUIPMENT#ZoneId#05:InteriorEquipment* [J]	-	
Occupancy	Zone People Sensible Heating Rate [W]	ZoneId	
Solar Gains Exterior Windows	Zone Windows Total Transmitted Solar Radiation Rate [W]	ZoneId	
Solar Gains Interior Windows	Zone Diff Solar from Interior Windows [W] + Zone Beam Solar from Interior Windows [W]	ZoneId	
Zone Sensible Cooling	Zone Air System Sensible Cooling Rate [W]	ZoneId	Overall sensible cooling energy to the zone from the HVAC system (not energy consumption)
Zone Sensible Heating	Zone Air System Sensible Heating Rate [W]	ZoneId	Overall sensible heating energy to the zone from the HVAC system (not energy consumption)

System Heat Flows

Zone Heating	Zone Air System Loads Sensible Heating Rate [W]	ZoneId Ideal Loads Air	Simple HVAC convective
Zone Heating	Total Water Heating Coil Rate [W]	ZoneId REHEAT COIL	VAV/CAV Hot water reheat coils
Zone Heating	Heating Coil Rate [W]	ZoneId REHEAT COIL	VAV/CAV Electric reheat coils
Zone Heating	Heating Coil Rate [W]	ZoneId AHU HEATING COIL	Unitary single zone
Zone Heating	Total Water Heating Coil Rate [W]	ZoneId HEATING COIL	Fan coil
Preheat	Heating Coil Rate [W]	AHU PREHEAT COIL	VAV/CAV Gas/Electric preheat coils
AHU Heating	Total Water Heating Coil Rate [W]	AHU HEATING COIL	VAV/CAV Hot water coils
AHU Heating	Heating Coil Rate [W]	AHU HEATING COIL	VAV/CAV with Gas/Electric AHU heating coils and Unitary multizone
Chiller Load	Chiller Evaporator Cooling Rate [W]	Chiller	Simple HVAC convective and ASHRAE Cooling design
Total Cooling	Ideal Loads Total Cooling Rate [W]	ZoneId Ideal Loads Air	Simple HVAC convective and ASHRAE Cooling design
Total Cooling	Total Water Cooling Coil Rate [W]	AHU COOLING COIL	VAV/CAV with Chilled water coils

Total Cooling	DX Coil Total Cooling Rate [W]	AHU COOLING COIL	Unitary Multizone
Total Cooling	DX Coil Total Cooling Rate [W]	Zoneld AHU COOLING COIL	Unitary single zone
Total Cooling	Total Water Cooling Coil Rate [W]	Zoneld COOLING COIL	Fan coil
Sensible Cooling	Ideal Loads Sensible Cooling Rate [W]	Zoneld Ideal Loads Air	Simple HVAC convective and ASHRAE Cooling design
Sensible Cooling	Sensible Water Cooling Coil Rate [W]	AHU COOLING COIL	VAV/CAV with Chiller water
Sensible Cooling	DX Coil Sensible Cooling Rate [W]	AHU COOLING COIL	Unitary multizone
Sensible Cooling	DX Coil Sensible Cooling Rate [W]	Zoneld AHU COOLING COIL	Unitary single zone
Sensible Cooling	Sensible Water Cooling Coil Rate [W]	Zoneld COOLING COIL	Fan coil

Fuel breakdown (building level)

Heating (Fuel)	Boiler {Electric/Gas etc} Consumption Rate + BaseBoard Electric Consumption + DX Heating Coil Electric Consumption + DX Heating Coil Electric Defrost Consumption + DX Heating Coil Crankcase Heater Consumption + Heating Coil Electric Consumption + Heating Coil Gas Consumption + Heating Coil Rate (electric)	Detailed HVAC	Note: CoP for direct gas coils is assumed to be 0.8 and direct electric coils 1.0.
Heating (Fuel)		Simple HVAC	Heat generator fuel = Heating load / Heating system CoP Note: the Heating system CoP is defined separately for each zone.

Cooling (Fuel)	Chiller Electric Power + DX Cooling Coil Electric Consumption + Chiller Cond Fan Electric Consumption + Evap Cooler Electric Power	Detailed HVAC	
Cooling (Fuel)		Simple HVAC	Chiller fuel = Cooling load / Cooling system CoP Note: the Cooling system CoP is defined separately for each zone.
System Fans	Fan Electric Consumption [J]	AHU SUPPLY FAN	VAV/CAV with Chiller water, Unitary multizone
System Fans	Fan Electric Consumption [J]	Zoneld AHU SUPPLY FAN	Unitary single zone
System Fans	Fan Electric Consumption [J]	Zoneld SUPPLY FAN	Fan coil
System Pumps	Pump Electric Consumption [J]	HOT WATER LOOP HW SUPPLY PUMP + CHILLED WATER LOOP CHW SUPPLY PUMP + CHILLED WATER LOOP CNDW SUPPLY PUMP	
System Fans	Fan Electric Consumption [J]	Simple HVAC with 2-Separate fans and pumps Auxiliary option	
System Pumps	ELECTRIC EQUIPMENT#ZoneId#07:InteriorEquip ment	Simple HVAC with 2-Separate fans and pumps Auxiliary option	

Auxiliary Energy	ELECTRIC EQUIPMENT#ZoneId#07:InteriorEquip ment	Simple HVAC with 1-NCM Auxiliary option	Auxiliary Energy = Auxiliary energy (kWh/m2) x Floor area (m2)
DHW	Domestic Hot Water Consumption [m3]	ZoneId DHW 1	DHW energy (kWh) = 1000 (kg/m3) x 4.187 (KJ/kg-K) x (Domestic Hot Water Consumption [m3]) x (Delivery water temperature - Mains water temperature) / DHW CoP Where: DHW CoP is Heat generator CoP when using 1-Central heating boiler DHW type.
Preheat Energy			(Preheat Heating Coil Rate + Distr loss if appropriate) / Preheat CoP
Lighting			Sum of lighting electricity consumption for all zones.
Room Electricity			Sum of Computer and equipment, Miscellaneous, Process, Catering electricity consumption for all zones.
Room Gas			Sum of Computer and equipment, Miscellaneous, Process, Catering gas consumption for all zones.
Room Oil			Sum of Computer and equipment, Miscellaneous, Process, Catering oil consumption for all zones.
Room Solid			Sum of Computer and equipment, Miscellaneous, Process, Catering solid fuel consumption for all zones.
Room Bottled gas			Sum of Computer and equipment, Miscellaneous, Process, Catering bottled gas consumption for all zones.
Room Other			Sum of Computer and equipment, Miscellaneous, Process, Catering "other" fuel consumption for all zones.
Fuel totals (building level)			
Electricity			1-Electricity from grid
Gas			2-Natural gas
Oil			3-Oil

Solid			4-Coal + 7-Anthracite + 8-Smokeless fuel (including coke)
Bottled gas			5-LPG + 6-Biogas
Other fuel consumption			9-Dual fuel appliances (mineral + wood) + 10-Biomass + 11-Waste heat

CO2 production (building level)

CO2			Calculated from Fuel totals above using locally published CO2 emission factors from the Emissions tab on the Legislative regions dialog accessed from the Site level on the Regions model data tab.
-----	--	--	---

Site Weather data

Outside dry-bulb temperature	Outdoor Dry Bulb	<i>Simulation:</i> Direct from EPW - spot value, <i>Cooling design:</i> Outdoor Dry Bulb - spot value.	For Cooling design, value is initialised from user input before calculation is run but is updated with E+ value after the calculation.
Outside dew-point temperature	Outdoor Dew Point	<i>Simulation:</i> Direct from EPW - spot value, <i>Cooling design:</i> Outdoor Dew Point - spot value.	
Direct normal solar	Direct Solar	<i>Simulation:</i> Direct from EPW - preceding hour, <i>Cooling design:</i> Direct Solar - preceding hour.	
Diffuse horizontal solar	Diffuse Solar	<i>Simulation:</i> Direct from EPW - preceding hour, <i>Cooling design:</i> Diffuse Solar -	

		preceding hour.	
Wind speed	Wind Speed	<i>Simulation:</i> Direct from EPW - spot value, <i>Cooling design:</i> Wind Speed - preceding hour.	
Wind direction	Wind Direction	<i>Simulation:</i> Direct from EPW - spot value, <i>Cooling design:</i> Assumed wind is from North.	
Atmospheric pressure	Outdoor Barometric Pressure	<i>Simulation:</i> Direct from EPW - spot value, <i>Cooling design:</i> Outdoor Barometric Pressure - preceding hour.	Estimated from elevation before cooling design calc is run but E+ value used after calculation.
Solar altitude	Solar Altitude Angle	<i>Simulation:</i> Calculated - spot value, <i>Cooling design:</i> Solar Altitude Angle - preceding hour.	For Cooling design, solar positions are initialised with calculated values before calculation is run but are updated with E+ values after the calculation.
Solar azimuth	Solar Azimuth Angle	<i>Simulation:</i> Calculated - spot value, <i>Cooling design:</i> Solar Azimuth Angle - preceding hour.	Calculated simulation data is based on 'spot values' on the hour so in this case the value for 12:00 is the value exactly at midday.

* Variables shown in **green** represent EnergyPlus output.

'Direct from EPW' means that the values in the hourly weather file itself are displayed.

EnergyPlus hourly output always represents totals and averages over the preceding hour so data for 12:00 is for the period 11:01 to 12:00.

See also: [When DesignBuilder output is different from unprocessed EnergyPlus Output](#)

EnergyPlus Errors And Warnings

A file [eplusout.err](#) is generated following each EnergyPlus simulation in the [EnergyPlus folder](#). It may contain three levels of errors (Warning, Severe, Fatal) as well as the possibility of just message lines. These errors may be duplicated in other files (such as the standard output file) and are described more completely in the [EnergyPlus Output Details](#) and [Examples](#) documents.

Errors

1

```
** Severe  ** Did not find "Until: 22:00" in list of Objects
```

You may have entered a semi-colon character (;) at the end of one of the lines in a [Compact schedule](#) when you meant to enter a comma (,).

2

```
** Severe  ** Problem in interior solar distribution calculation
(CHKBKS)
**      ~~~      **      Solar Distribution = FullInteriorExterior will not
work in Zone=78474
**      ~~~      **      because vertex 3 of back surface=W_73536_2_0_0 is in
front of receiving surface=W_78474_3_0_0
**      ~~~      **      (Dot Product indicator=1.4966)
**      ~~~      **      Check surface geometry; if OK, use Solar Distribution
= FullExterior instead.
```

The [3-Full interior and exterior](#) solar distribution option is selected but one or more zone in the model is non-convex. Note that DesignBuilder is [configured](#) by default to check for this condition and provides more friendly error messages.

3

```
** Severe ** For Ideal Loads ZONE 21438 Ideal Loads serving Zone 21438
** ~~~ ** ..the supply air temperature for cooling [10.00] is greater
than the zone cooling setpoint [8.99].
```

This severe error can occur when using Simple HVAC, or Cooling design calculations with [Operative](#) control in zones with strong radiant heat gains causing high radiant temperatures (e.g. un-insulated roof or zone is highly glazed). The supply air temperature is fixed (to 10°C in the above example) but the zone air temperature must be cooled lower than this (8.99°C in the above example) to give the required operative setpoint (e.g. 24 °C). Operative temperature, T_{op} , is calculated as:

$$T_{op} = 0.5 \times (T_a + T_r)$$

where:

T_{op} is the operative temperature.

T_s is the air supply temperature.

T_a is the zone air temperature (T_a must be $> T_s$).

T_r is the zone radiant temperature.

So there is a requirement:

$$T_{r \max} \leq 2 T_{op} - T_s$$

If the zone radiant temperature is higher than $T_{r \max}$ and Operative temperature control is used then this severe error will occur.

The solution may be to use Air temperature control and to reduce the radiant temperatures using solar shading/insulation as appropriate.

See the discussion on [Operative temperature control](#) under Calculation options for more on this.

4

```
** Severe ** Problem in interior solar distribution calculation
(CHKBKS)
** ~~~ ** Solar Distribution = FullInteriorExterior will not
work in Zone=42272
** ~~~ ** because vertex 6 of back
surface=F_42272_10106_0_10277 is in front of receiving
surface=W_42272_9_0_0
** ~~~ ** (Dot Product indicator=7.3314)
** ~~~ ** Check surface geometry; if OK, use Solar
Distribution = FullExterior instead.
```

One or more zones is [non-convex](#) which makes the model incompatible with the [3-Full interior and exterior](#) solar distribution algorithm. This error can also lead to temperature out of bounds errors.

5

```
** Severe ** Temperature out of bounds (637.89) for
surface=W_43311_4_0_0
** ~~~ ** in Zone=43276
** ~~~ ** Occurrence info=BRUSSELS - BEL IWECA DATA WMO#=064510,
01/04 13:30 - 13:32
** ~~~ ** A temperature out of bounds problem can be caused by
several things. The user
** ~~~ ** should check the weather environment, the level of
internal gains with respect
** ~~~ ** to the zone, and the thermal properties of their
materials among other things.
** ~~~ ** A common cause is a building with no thermal mass --
all materials with
** ~~~ ** Regular-R definitions.
```

The temperature of a surface has become too high/low, during the simulation. This error can be caused by highly glazed zones with holes taking up all or most of the floor. Some examples are double facade cavities, solar chimneys and atria-type zones. In some cases EnergyPlus isn't able to deal with such zones. The issue often isn't really overheating so much as instability in the solution - very low temperatures can result from the same situation. Here are some actions which have helped get round this problem in the past:

Make holes in the floor smaller. Bear in mind that with the default "Full exterior" solar distribution option, all solar gain is received by the floor, but if there is no floor due to a full-sized hole then a very large gain could be absorbed by a very small area of floor surface.

Avoid R-value (or other very low mass materials) on the inside surface of constructions. It may be possible to use zone merging to combine zones which are being modelled individually separated by large holes. This can sometimes help to avoid the error.

Try switching off the [Generate fully enclosed zones](#) Advanced model option. This has helped in a few cases because the construction applied to surfaces used to fill in the gaps in zone geometry is low mass and has very high R-value which could cause a build-up of heat.

Warnings

1

```

** Warning ** GetHTSubSurfaceData: A Multiplier > 1.0 for
window/glass door W_67_4_0_0_0_0
**   ~~~   ** in conjunction with SolarDistribution = FullExterior
or FullInteriorExterior
**   ~~~   ** can cause inaccurate shadowing on the window and/or
**   ~~~   ** inaccurate interior solar distribution from the
window.

```

The [Lump similar windows on surface](#) model option has been selected which causes DesignBuilder to use the EnergyPlus Window Multiplier option to speed up simulations. The message is warning that the Window Multiplier option, when used in conjunction with the [Full exterior](#) or [Full interior and exterior](#) calculation options can lead to inaccurate distributions of solar radiation around the room.

2

```

** Warning ** GetSurfaceData:The total number of floors, walls,
roofs and internal mass surfaces in Zone 59
**   ~~~   ** is < 6. This may cause an inaccurate zone heat
balance calculation.

```

This warning is often generated for pitched roofs and for interior zones separated by virtual partitions and can safely be ignored.

3

```

** Warning ** GetGroundTemps: Some values for "GroundTemperatures"
fall outside the range of 15-25C.
**   ~~~   ** These values may be inappropriate. Please consult
the Input Output Reference for more details.

```

The default ground temperatures used in DesignBuilder is 14degC. This may be inappropriate in some cases and EnergyPlus is warning about this. You should take care to enter realistic ground temperatures bearing in mind the advice given in the [Ground Modelling](#) section.

4

```

** Warning ** DXFOut: Could not triangulate
surface="W_7469_0_0_1", type="Wall", check surface vertex
order(entry)

```

Caused by a minor bug in EnergyPlus when creating a DXF representation of the model - it is not always able to fully triangulate all surfaces and generate a full set of surfaces. You can ignore the message for the purposes of simulation results.

5

```

** Warning ** Base surface does not surround subsurface (CHKSBS),
Overlap Status=No-Overlap
**   ~~~   ** The base surround errors occurred 3 times.
**   ~~~   ** Surface "W_7376_3_0_0" misses SubSurface
"W_7376_3_0_0_0_0"

```

These warnings are usually a reporting bug in EnergyPlus v2.0 and earlier.

6

```

** Warning ** Calculated design cooling/heating load for
zone=78474 is zero.
**   ~~~   ** Check ZONE SIZING and ZONE CONTROL:THERMOSTATIC
inputs.

```

Cooling/heating may have been switched off in this zone or there is no cooling/heating load (i.e. the zone does not require any cooling/heating).

7

```

For a zone sizing run, there must be at least 1 Zone Sizing object
input.

```

Can usually be ignored.

8

```

** Severe ** CAUTION -- Interzone surfaces are usually in
different zones
**   ~~~   ** Surface=F_59604_10071_0_10046, Zone=59604
**   ~~~   ** Surface=C_72917_1_0_0, Zone=59604

```

This warning can be safely ignored. These messages can be prevented by switching on the [Surfaces within zone treated as adiabatic](#) option in Simulation [Advanced Calculation Options](#)

9

```

** Warning ** CAUTION -- Interzone surfaces are usually in
different zones

```

This warning will occur for partitions between 2 merged zones. Because of the merging, the zone is the same on both sides of the partition and EnergyPlus flags this as a potential mistake. To avoid this message, you can select the 'Surfaces within zone' treated as adiabatic model option for Heating design, Cooling design and Simulation calculation options. In this case such partitions will instead be modelled using an adiabatic formulation where the 'Outside face environment' surface is the same as the actual surface and the warning is not generated. The warning can in any case be safely ignored.

Maximum Iteration Errors

Maximum iteration errors are among the most common warnings encountered in EnergyPlus and DesignBuilder simulations. These errors indicate that the simulation is unable to converge on a set of flow rates and temperatures that meet the current load conditions within the specified number of iterations.

What causes maximum iteration errors

Maximum iteration errors are usually related to mass flow or temperature problems around HVAC loops. For example, this can occur when the air loop side is doing something contrary to the zone equipment, creating conflicts in the HVAC system simulation. In simple terms, the simulation is trying to balance several factors:

- The airflow each zone requires
- The temperature of that airflow
- The available heating and cooling capacity
- The timing of equipment operation

When these don't align, the model gets stuck in a loop, repeatedly trying to reach a physically impossible solution.

Common causes

Below is a concise introduction to the common causes of maximum iteration errors:

- Airflow mismatches:** The sum of terminal unit airflows does not equal the supply fan airflow
- Inconsistent equipment schedules:** Conflicting operating schedules between related components
- Inconsistent control setpoints:** Conflicting temperature or airflow control setpoints
- Undersized equipment:** Coil and plant equipment capacities insufficient to meet supply air setpoints
- Complex system types:** Variable Air Volume (VAV) systems with reheat or floating supply air temperatures are especially prone to these warnings.
- Controller convergence issues:** Loose controller convergence tolerances

When to be concerned

The significance of maximum iteration errors depends on several factors. In general, how often the errors occur is more important than the total number reported. The total count of errors can vary greatly depending on the model's complexity - such as the number of zones, air systems, and the length of the simulation period. For example, a thousand errors might be acceptable in a large annual model but could indicate a problem in a small, single-zone design-day simulation.

It is also important to consider when the errors occur. If they appear at every timestep, they likely indicate a persistent HVAC system issue. However, if the errors occur only occasionally, such as during periods of rapid system change like setback or recovery, they are generally less concerning.

Finally, assess how these errors affect system performance. If the zones maintain proper temperature control and the errors occur only intermittently, the results are probably not affected. On the other hand, if the zones consistently fail to meet their setpoints, the errors are significant and will impact the accuracy of the simulation.

See the [Max iterations exceeded](#) section of the [Tips and Tricks for Using EnergyPlus](#) guide.

Diagnostic steps

When convergence warnings arise, follow a structured diagnostic workflow to uncover their root causes: review the detailed error log to pinpoint struggling systems, enable extra warnings to reveal hidden conflicts, and examine key hourly outputs to trace mismatches in temperatures or flow rates.

a. Review the EnergyPlus error file

When maximum iteration errors occur, the first step is to review the EnergyPlus error file (eplusout.err). This file lists all warnings and errors generated during the simulation and is the primary source for identifying convergence issues. Always read the file from the bottom upward. The final message usually describes the most serious problem. Repeated "maximum iteration" messages indicate that certain systems are struggling to find stable operating conditions.

b. Display extra warnings

For a more detailed understanding of the problem, you can enable additional diagnostic messages in EnergyPlus. This is done by checking the [Display extra warnings](#) Program option. These extra warnings can provide valuable insights into the cause of convergence difficulties, such as mismatched setpoints or unstable controller behaviour.

c. Analyse key hourly results

It is also helpful to examine key simulation outputs to understand how the system behaves during the problem periods. Reviewing hourly results for zone air temperatures, system flow rates, and coil or loop outlet temperatures can reveal where the mismatch or instability is occurring. For instance, abrupt temperature swings or unrealistic flow rate changes often indicate issues with system control or component interaction.

If the simulation fails with a fatal error and the output results are not available (like .csv or .eso files) running shorter simulations (e.g., single design day or one-week period) allows to identify the exact day or time when instability occurs, produces results faster, helping with iterative debugging, and makes it easier to visualize system behavior in the output variables. Once the model runs stably over a short period, extending to a month or full-year simulation can be done in confidence.

Solutions and Troubleshooting

Once maximum iteration errors are identified, the next step is to focus on resolving their underlying causes. Most solutions involve refining equipment sizing, improving control logic, or adjusting simulation settings to help the model reach stable operating conditions.

a. Check Equipment Sizing

Undersized coils, fans, or plant components are among the most common reasons for convergence issues. The first step is to confirm that all HVAC equipment can meet the required loads. Whenever possible, use the

autosizing feature rather than fixed capacities. Autosizing ensures that coils, fans, and controllers are proportionally sized to the building's actual heating and cooling demands. If manual sizing is necessary, verify that coil capacities, plant loop flow rates, and fan airflows are consistent with the design intent and that supply airflow matches the sum of the terminal unit flows.

b. Review Control Logic and Setpoints

Inconsistent or conflicting control logic can cause EnergyPlus to oscillate between incompatible conditions. Check all temperature and flow setpoints for consistency across the system. Simplifying control logic and maintaining consistent setpoints between related components often eliminates persistent iteration problems.

c. Adjust Convergence Tolerances

If the system configuration appears correct but iteration errors persist, you may need to relax the simulation convergence tolerances. The temperature and loads convergence values can be found under the [Advanced](#) header on the [Options](#) tab of the [Simulation Calculation Options](#) dialog.

EnergyPlus uses two main settings to determine how closely it must match energy and temperature balances before proceeding to the next timestep:

The temperature convergence tolerance, which defaults to 0.4°C
The loads convergence tolerance, which defaults to 0.04.

For complex models, slightly increasing these values can help achieve stable results without meaningfully affecting accuracy. It is best to adjust these parameters gradually while monitoring simulation stability.

Summary

Maximum iteration errors require systematic investigation focusing on HVAC system configuration, equipment sizing, and control logic. While some intermittent errors may be acceptable, persistent convergence failures indicate underlying modelling issues that can significantly impact simulation accuracy. The key is to identify whether the errors affect system performance and zone conditions, then address the root cause through proper system configuration and convergence settings.

EnergyPlus Daylight Calculations

The EnergyPlus daylighting model, in conjunction with the thermal analysis, determines the energy impact of daylighting strategies based on analysis of daylight availability, site conditions, window management in response to solar gain and glare, and various lighting control strategies.

The daylighting calculations are carried out during the simulation in 3 main steps:

Daylight factors, which are ratios of interior illuminance or luminance to exterior horizontal illuminance, are calculated and stored. The user specifies the coordinates of one or two reference points in each daylit zone. EnergyPlus then integrates over the area of each exterior window in the zone to obtain the contribution of direct light from the window to the illuminance at the reference points, and the contribution of light that reflects from the walls, floor and ceiling before reaching the reference points. Window luminance and window background luminance, which are used to determine glare, are also calculated. Taken into account are such factors as sky luminance distribution, window size and orientation, glazing transmittance, inside surface reflectances, sun control devices such as movable window shades, and external obstructions. Dividing daylight illuminance or luminance by exterior illuminance yields daylight factors. These factors are calculated for the hourly sun positions on sun-paths for representative days of the run period.

A daylighting calculation is performed each heat-balance time step when the sun is up. In this calculation the illuminance at the reference points in each zone is found by interpolating the stored daylight factors using the current time step's sun position and sky condition, then multiplying by the exterior horizontal illuminance. If glare control has been specified, the program will automatically deploy window shading, if available, to decrease glare below a specified comfort level. A similar option uses window shades to automatically control solar gain.

The electric lighting control system is simulated to determine the lighting energy needed to make up the difference between the daylighting illuminance level and the design illuminance. Finally, the zone lighting electric reduction factor is passed to the thermal calculation, which uses this factor to reduce the heat gain from lights. The EnergyPlus daylighting calculation is derived from the daylighting calculation in DOE-2.1E, which is described in [Winkelmann, 1983] and [Winkelmann and Selkowitz, 1985]. There are two major differences between the two implementations: (1) In EnergyPlus daylight factors are calculated for four different sky types—clear, clear turbid, intermediate, and overcast; in DOE-2 only two sky types are used—clear and overcast. (2) In EnergyPlus the clear-sky daylight factors are calculated for hourly sun-path sun positions several times a year whereas in DOE-2 these daylight factors are calculated for a set of 20 sun positions that span the annual range of sun positions for a given geographical location.

EnergyPlus Thermal Comfort

EnergyPlus includes a sophisticated building thermal analysis tool allowing you to determine whether the environmental control strategy will be sufficient for the occupants to be thermally comfortable. This section provides background on thermal comfort and gives an overview of state of the art thermal comfort models.

Background on Thermal Comfort Models

Throughout the last few decades, researchers have been exploring the thermal, physiological and psychological response of people in their environment in order to develop mathematical models to predict these responses. Researchers have empirically debated building occupants' thermal responses to the combined thermal effect of the personal, environmental and physiological variables that influence the condition of thermal comfort.

There are two personal variables that influence the condition of thermal comfort: the thermal resistance of the clothing (Icl), and the metabolic rate (H/ADu). The thermal resistance of the clothing (Icl) is measured in units of "clo." The 1985 ASHRAE Handbook of Fundamentals (3) suggests multiplying the summation of the individual clothing items clo value by a factor of 0.82 for clothing ensembles.

The environmental variables that influence the conditions of thermal comfort include:

- Air Temperature (T_a),
- Mean Radiant Temperature (T_r),
- Relative air velocity (v),
- Water vapor pressure in ambient air (P_a)

The Air Temperature (T_a), a direct environmental index, is the dry-bulb temperature of the environment. The Mean Radiant Temperature (T_r) is a rationally derived environmental index defined as the uniform black-body temperature that would result in the same radiant energy exchange as in the actual environment. The Relative air velocity (v) a direct environmental index is a measure of the air motion obtainable via a hot wire or vane anemometers. The Water vapor pressure in ambient air (P_a) is a direct environmental index.

The physiological variables that influence the conditions of thermal comfort include:

- Skin Temperature (T_{sk}),
- Core or Internal Temperature (T_{cr}),
- Sweat Rate,
- Skin Wettedness (w),
- Thermal Conductance (K) between the core and skin.

Where the Skin Temperature (T_{sk}), the Core Temperature (T_{cr}) and the Sweat Rate are physiological indices. The Skin Wettedness (w) is a rationally derived physiological index defined as the ratio of the actual sweating rate to the maximum rate of sweating that would occur if the skin were completely wet. One more consideration is important in dealing with thermal comfort - the effect of asymmetrical heating or cooling. This could occur when there is a draft or when there is a radiant flux incident on a person (which is what is of primary interest to us here). Fanger (5) noted that the human regulatory system is quite tolerant of asymmetrical radiant flux. A reasonable upper limit on the difference in mean radiant temperature (T_r) from one direction to the opposing direction is 15_ (1). This limit is lower if there is a high air velocity in the zone.

DesignBuilder Thermal Comfort Calculator

You can use the DesignBuilder Thermal Comfort Calculator to evaluate

Mathematical Models for Predicting Thermal Comfort

Many researchers have been exploring ways to predict the thermal sensation of people in their environment based on the personal, environmental and physiological variables that influence thermal comfort. From the research done, some mathematical models that simulate occupants' thermal response to their environment have been developed. Most thermal comfort prediction models use a seven or nine point thermal sensation scale, as in the following tables.

Seven point Thermal Sensation Scale:

3 hot
2 warm
1 slightly warm
0 neutral
-1 slightly cool
-2 cool
-3 cold

Nine point Thermal Sensation Scale:

4 very hot
3 hot
2 warm
1 slightly warm
0 neutral
-1 slightly cool
-2 cool
-3 cold
-4 very cold

The most notable models have been developed by P.O. Fanger (the Fanger Comfort Model), the J. B. Pierce Foundation (the Pierce Two-Node Model), and researchers at Kansas State University (the KSU Two-Node Model). Berglund (6) presents a detailed description of the theory behind these three models.

Note: for all Thermal Comfort reporting: Though the published values for thermal comfort “vote” have a discrete scale (e.g. –3 to +3 or –4 to +4), the calculations in EnergyPlus are carried out on a continuous scale and, thus, reporting may be “off the scale” with specific conditions encountered in the space. This is not necessarily an error in EnergyPlus – rather a different approach that does not take the “limits” of the discrete scale values into account.

The main similarity of the three models is that all three apply an energy balance to a person and use the energy exchange mechanisms along with experimentally derived physiological parameters to predict the thermal sensation and the physiological response of a person due to their environment. The models differ somewhat in the physiological models that represent the human passive system (heat transfer through and from the body) and the human control system (the neural control of shivering, sweating and skin blood flow). The models also differ in the criteria used to predict thermal sensation.

Fanger Comfort Model

Fanger's Comfort model was the first one developed. It was published first in 1967 (7) and then in 1972 (2), and helped set the stage for the other two models. The mathematical model developed by P.O. Fanger is probably the most well known of the three models and is the easiest to use because it has been put in both chart and graph form.

Predicted Mean Vote (PMV) and Predicted Percentage Dissatisfied (PPD)

Fanger developed the model based on the research he performed at Kansas State University and the Technical University of Denmark. Fanger used the seven-point form of a thermal sensation scale along with numerous experiments involving human subjects in various environments. He related the subjects in response

to the variables, which influence the condition of thermal comfort. Fanger's model is based upon an energy analysis that takes into account all the modes of energy loss (L) from the body, including: the convection and radiant heat loss from the outer surface of the clothing, the heat loss by water vapour diffusion through the skin, the heat loss by evaporation of sweat from the skin surface, the latent and dry respiration heat loss and the heat transfer from the skin to the outer surface of the clothing. The model assumes that the person is thermally at steady state with his environment. By determining the skin temperature and evaporative sweat rate that a thermally comfortable person would have in a given set of conditions, the model calculates the energy loss (L). Then, using the thermal sensation votes from subjects at KSU and Denmark, a Predicted Mean Vote (PMV) thermal sensation scale is based on how the energy loss (L) deviates from the metabolic rate (M).

Fanger proposed that the condition for thermal comfort is that the skin temperature and sweat secretion lies within narrow limits. Fanger obtained data from climate chamber experiments, in which sweat rate and skin temperature were measured on people who considered themselves comfortable at various metabolic rates. Fanger proposed that optimal conditions for thermal comfort were expressed by the regression line of skin temperature and sweat rate on metabolic rate in data from these experiments. In this way an expression for optimal thermal comfort can be deduced from the metabolic rate, clothing insulation and environmental conditions.

The final equation for optimal thermal comfort is fairly complex and need not concern us here. Fanger has solved the equations by computer and presented the results in the form of diagrams from which optimal comfort conditions can be read given a knowledge of metabolic rate and clothing insulation.

Fanger extended the usefulness of his work by proposing a method by which the actual thermal sensation could be predicted. His assumption for this was that the sensation experienced by a person was a function of the physiological strain imposed on him by the environment. This he defined as "the difference between the internal heat production and the heat loss to the actual environment for a man kept at the comfort values for skin temperature and sweat production at the actual activity level" (Fanger 1970). He calculated this extra load for people involved in climate chamber experiments and plotted their comfort vote against it. Thus he was able to predict what comfort vote would arise from a given set of environmental conditions for a given clothing insulation and metabolic rate. Tables of PMV are available for different environments for given clothing and metabolic rates. Such tables form the basis of ISO standard 7730 (see summary comfort categories below). Note however that his method for PMV is inconsistent with the basic assumptions of his equation (Humphreys and Nicol 1995).

Fanger realised that the vote predicted was only the mean value to be expected from a group of people, and he extended the PMV to predict the proportion of any population who will be dissatisfied with the environment. A person's dissatisfaction was defined in terms of their comfort vote. Those who vote outside the central three scaling points on the ASHRAE scale were counted as dissatisfied. PPD is defined in terms of the PMV, and adds no information to that already available in PMV. The distribution of PPD is based on observations from climate chamber experiments and not from field measurements.

From ISO 7730:2005 Annex A

Comfort categories:

- A: PPD < 6%
- B: PPD < 10%
- C: PPD < 15%

Pierce Two-Node Model

The Pierce Two-Node model was developed at the John B. Pierce Foundation at Yale University. The model has been continually expanding since its first publication in 1970 (8). The most recent version on the model appears in the 1986 ASHRAE Transactions (9).

The Pierce model thermally lumps the human body as two isothermal, concentric compartments, one representing the internal section or core (where all the metabolic heat is assumed to be generated and the skin comprising the other compartment). This allows the passive heat conduction from the core compartment to the skin to be accounted for. The boundary line between two compartments changes with respect to skin blood flow rate per unit skin surface area (SKBF in $L/h \cdot m^2$) and is described by alpha – the fraction of total body mass attributed to the skin compartment (13).

Furthermore, the model takes into account the deviations of the core, skin, and mean body temperature weighted by alpha from their respective set points. Thermoregulatory effector mechanisms (Regulatory sweating, skin blood flow, and shivering) are defined in terms of thermal signals from the core, skin and body (13).

The model for Standard Effective Temperature (SET) also uses skin temperature as part of its limiting conditions, but uses skin wettedness (w) rather than sweat rate for the other limiting condition. The values for T_{sk} and w are derived from the Pierce 'two-node' model of human physiology (see Nevins & Gagge (1972)). SET relates the real conditions to the (effective) temperature in standard clothing and metabolic rate and 50% RH which would give the same physiological response. Effective temperature can then be related to subjective response.

The latest version of the Pierce model (15) uses the concepts of SET^* and ET^* . The Pierce model converts the actual environment into a "standard environment" at a Standard Effective Temperature, SET^* . SET^* is the dry-bulb temperature of a hypothetical environment at 50% relative humidity for subjects wearing clothing that would be standard for the given activity in the real environment. Furthermore, in this standard environment, the same physiological strain, i.e. the same skin temperature and skin wettedness and heat loss to the environment, would exist as in the real environment. The Pierce model also converts the actual environment into a environment at an Effective Temperature, ET^* , that is the dry-bulb temperature of a hypothetical environment at 50% relative humidity and uniform temperature ($T_a = MRT$) where the subjects would experience the same physiological strain as in the real environment.

In the latest version of the model it is suggested that the classical Fanger PMV be modified by using ET^* or SET^* instead of the operative temperature. This gives a new index PMV^* which is proposed for dry or humid environments. It is also suggested that PMV^* is very responsive to the changes in vapor permeation efficiency of the occupants clothing.

Besides PMV^* , the Pierce Two Node Model uses the indices TSENS and DISC as predictors of thermal comfort. Where TSENS is the classical index used by the Pierce foundation, and is a function of the mean body temperature. DISC is defined as the relative thermoregulatory strain that is needed to bring about a state of comfort and thermal equilibrium. DISC is a function of the heat stress and heat strain in hot environments and equal to TSENS in cold environments. In summary, the Pierce Model, for our purposes, uses four thermal comfort indices; $PMVET$ -a function of ET^* , $PMVSET$ - a function of SET^* , TSENS and DISC.

KSU Two-Node Model

The KSU two-node model, developed at Kansas State University, was published in 1977 (10). The KSU model is quite similar to that of the Pierce Foundation. The main difference between the two models is that the KSU model predicts thermal sensation (TSV) differently for warm and cold environment.

The KSU two-node model is based on the changes that occur in the thermal conductance between the core and the skin temperature in cold environments, and in warm environments it is based on changes in the skin wettedness. In this model metabolic heat production is generated in the core which exchanges energy with the environment by respiration and the skin exchanges energy by convection and radiation. In addition, body heat is dissipated through evaporation of sweat and/or water vapor diffusion through the skin. These principles are used in following passive system equations.

Here, control signals, based on set point temperatures in the skin and core, are introduced into passive system equations and these equations are integrated numerically for small time increments or small increments in core and skin temperature. The control signals modulate the thermoregulatory mechanism and regulate the peripheral blood flow, the sweat rate, and the increase of metabolic heat by active muscle shivering. The development of the controlling functions of skin conductance (KS), sweat rate (E_{sw}), and shivering (M_{shiv}) is based on their correlation with the deviations in skin and core temperatures from their set points.

The KSU model's TSV was developed from experimental conditions in all temperature ranges and from clo levels between .05 clo to 0.7 clo and from activities levels of 1 to 6 mets (6).

References

- ASHRAE, "High Intensity Infrared Radiant Heating", 1984 system Handbook, American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta, GA, Chapter 18, 1984.
- Fanger, P.O., Thermal Comfort-Analysis and Applications in Environmental Engineering, Danish Technical Press, Copenhagen, 1970.

ASHRAE, "Physiological Principles for Comfort and Health," 1985 Fundamental Handbook, American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta, GA, Chapter 8, 1985.

Du Bois, D. and E.F., "A Formula to Estimate Approximate Surface Area, if Height and Weight are Known", Archives of internal Medicine, Vol.17, 1916.

Fanger, P.O., "Radiation and Discomfort", ASHRAE Journal. February 1986.

Berglund, Larry, "Mathematical Models for Predicting the Thermal Comfort Response of Building Occupants", ASHRAE Trans., Vol.84, 1978.

Fanger P.O., "Calculation of Thermal Comfort: Introduction of a Basic Comfort Equation", ASHRE Trans., Vol.73, Pt 2, 1967.

Gagge, A.P., Stolwijk, J. A. J., Nishi, Y., "An Effective Temperature Scale Based on a Simple Model of Human Physiological Regulatory Response", ASHRAE Trans., Vol.70, Pt 1, 1970.

Gagge, A.P., Fobelets, A.P., Berglund, L. G., "A Standard Predictive Index of Human Response to the Thermal Environment", ASHRAE Trans., Vol.92, Pt 2, 1986.

Azer, N.Z., Hsu, S., "The prediction of Thermal Sensation from Simple model of Human Physiological Regulatory Response", ASHRAE Trans., Vol.83, Pt 1, 1977.

Hsu, S., "A Thermoregulatory Model for Heat Acclimation and Some of its Application", Ph. D. Dissertation, Kansas State University, 1977.

ISO., "Determination of the PMV and PPD Indices and Specification of the Conditions for Thermal Comfort", DIS 7730, Moderate Thermal Environment, 1983.

Doherty, T.J., Arens, E., "Evaluation of the Physiological Bases of Thermal Comfort Models", ASHRAE Trans., Vol.94, Pt 1, 1988.

ASHRAE, "Physiological Principles and Thermal Comfort", 1993 ASHRAE Handbook- Fundamentals, American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta, GA, Chapter 8, 1993.

Fountain, Marc.E., Huizenga, Charlie, "A Thermal Sensation Prediction Tool for Use by the Profession", ASHRAE Trans., Vol.103, Pt 2, 1997.

Int-Hout, D., "Thermal Comfort Calculation / A Computer Model", ASHRAE Trans., Vol.96, Pt

EnergyPlus Date Definition

Field Contents	Interpretation	Example
<Number> / <Number>	Month / Day	12 / 31
<Number> / Month	Day and Month	31 Dec
Month / <Number>	Month and day	Dec 31
<Number> Weekday in Month	Numbered weekday of month	1st Saturday in April
Last Weekday in Month	Last weekday of month	Last Sunday in October

Notes

Month can be one of:

January,
February,
March,
April,
May,
June,
July,
August,
September,
October,
November,
December.

Abbreviations of the first three characters are also valid.

Weekday can be one of:

Sunday,
Monday,

Tuesday,
Wednesday,
Thursday,
Friday,
Saturday.

Abbreviations of the first three characters are also valid.

When DesignBuilder Output Is Different From Unprocessed EnergyPlus Output

At the end of an EnergyPlus simulation [DesignBuilder reads](#) the EnergyPlus output file [eplusout.eso](#). Much of the data is stored and displayed without further processing but in some cases the outputs are processed to ensure that they are consistent with the DesignBuilder model data. DesignBuilder output data will differ from the data that can be found in the Simulation Summary tab in these cases:

Simple HVAC Heating and Cooling Fuel

DesignBuilder applies a fuel type and a CoP to the idealised heating and cooling systems used in Simple HVAC, whereas EnergyPlus displays the unprocessed loads under 'District heating'. This is because the idealised systems in EnergyPlus are considered to be met directly without use of HVAC equipment through a district heating/cooling type system.

Simple HVAC DHW Fuel

Like the heating and cooling in Simple HVAC, the Simple HVAC DHW system is modelled as a simple water heater which does not need a heat source so it is considered to be provided by district heating and loads are displayed directly in EnergyPlus output. In DesignBuilder the DHW loads reported by EnergyPlus are multiplied by the DHW CoP value at building level (when the DHW type is the **Same as HVAC**) or the DHW CoP at zone level for all other DHW type settings.

See also: [Calculation of DesignBuilder Output from EnergyPlus Report Variables](#)

Simple HVAC Pump Energy

When using the default Simple HVAC **2-Separate fans and pumps** [Auxiliary energy calculations](#) model option, DesignBuilder calculates auxiliary energy based on the "Pump etc energy" on page 469 setting and corresponding schedule on the HVAC tab. In this case, the auxiliary energy is reported under "Interior equipment" in the "End Uses" Summary tables. Therefore the "Interior equipment" summary output will not agree with the sum of the internal gains reported by DesignBuilder on the Analysis tab.

Parametric Simulations

There are some cases where optimisation, parametric analysis and UA/SA calculations produce different results from the standard EnergyPlus simulations.

LCC Optimisation Results Not Reproduced Manually

Important Advice for using the [Life Cycle Cost \(LCC\) KPI](#) in parametric simulations (optimisations, parametric analysis and UA/SA):

The LCC total present value KPI requires the Detailed HVAC model option to be used as it uses settings on the Economics model data tab which is only available for Detailed HVAC. The LCC total present value values calculated in optimisation and other parametric simulations will only correspond with the value calculated directly by EnergyPlus if the basis for calculation is the same. In practice this means that the following 2 further requirements must be met:

Any additional [Adjustment](#) costs on the [Component cost](#) dialog should be removed. To do this set all costs to zero and the [Regional adjustment factor](#) to 1.

The **1-BeginningOfYear** [Discounting convention](#) on the [Life cycle cost](#) dialog must be used. This should be the case by default for models created in v7 onwards.

Additional Background Information

DesignBuilder calculates building the LCC KPI during parametric simulations by adding:

the sum of the base construction cost line items (as calculated on the Cost and Carbon tab) to the LCC value calculated by E+ without the line item construction costs.

The addition is done by DesignBuilder at the time of generating the result for the LCC KPI, after the E+ optimisation simulation has been completed.

This simplification in the calculation of the LCC KPI is necessary because it is not currently possible for DesignBuilder to feed in parametrically varied cost results to the E+ inputs for the corresponding parametric variant. The rationale behind the approximation used by DesignBuilder in the calculation of the optimisation LCC KPI is that the base cost calculated on the cost and carbon tab is close to the more correct Present Value base construction cost used by E+ and so it can be added to the LCC present value calculated in E+ with no construction costs included. I.e. the “Present Value Factor” for Construction Costs (reported in the “Present Value for Recurring, Nonrecurring and Energy Costs” Summary report table) is close to 1.

Unless conditions 2 and 3 above are met, this simplified approach generates different results from the case where the base construction costs are fed into E+ for a full LCC calculation taking into account the Present Value of the construction costs and a range of other adjustments and factors included in the full E+ financial calculation method, which is based on NIST.

EnergyPlus Daylight Map Output

EnergyPlus generates a file called [eplusmap.csv](#) when one or more zones have [daylight control](#) and the [Daylight map simulation output option](#) has been selected for those zones. This file contains a series of grids for each zone with daylight controls and having daylight map output requested and for each hour of the day in the simulation.

The example output below shows some graphical output for 3 hours for a zone using a spreadsheet with cells shaded based on illuminance values. This was done using a macro as described in a [forum article by Jean Marais](#).

	A	B	C	D	E	F	G	H	I	J	K
1	Date/Time,STA	RefPt1=(76.42:-70.11:23.63)									
2	04/06 07:00	x=0.20	x=6.57	x=12.93	x=19.30	x=25.66	x=32.03	x=38.39	x=44.76	x=51.12	x=57.49
3	y=-111.63	7	7	7	7	7	7	7	7	7	7
4	y=-102.01	7	7	7	7	7	46	27	13	20	7
5	y=-92.39	7	7	7	52	8	23	122	7	7	7
6	y=-82.77	7	7	53	11	68	7	7	7	7	7
7	y=-73.14	7	7	13	49	7	7	7	7	7	7
8	y=-63.52	7	28	18	168	7	7	7	7	7	7
9	y=-53.90	7	18	29	7	7	7	7	7	7	7
10	y=-44.28	7	15	64	7	7	7	7	7	7	7
11	y=-34.66	7	12	107	7	7	7	7	7	7	7
12	y=-25.04	7	8	7	7	7	7	7	7	7	7
13	04/06 08:00	x=0.20	x=6.57	x=12.93	x=19.30	x=25.66	x=32.03	x=38.39	x=44.76	x=51.12	x=57.49
14	y=-111.63	29	29	29	29	29	29	29	29	29	29
15	y=-102.01	29	29	29	29	29	228	99	51	76	29
16	y=-92.39	29	29	29	206	31	86	434	29	29	29
17	y=-82.77	29	29	193	43	249	29	29	29	29	29
18	y=-73.14	29	29	51	183	29	29	29	29	29	29
19	y=-63.52	29	107	69	624	29	29	29	29	29	29
20	y=-53.90	29	69	109	29	29	29	29	29	29	29
21	y=-44.28	29	56	238	29	29	29	29	29	29	29
22	y=-34.66	29	45	405	29	29	29	29	29	29	29
23	y=-25.04	29	29	29	29	29	29	29	29	29	29
24	04/06 09:00	x=0.20	x=6.57	x=12.93	x=19.30	x=25.66	x=32.03	x=38.39	x=44.76	x=51.12	x=57.49
25	y=-111.63	69	69	69	69	69	69	69	69	69	69
26	y=-102.01	69	69	69	69	69	569	240	128	188	69
27	y=-92.39	69	69	69	556	73	213	1058	69	69	69
28	y=-82.77	69	69	471	106	620	69	69	69	69	69
29	y=-73.14	69	69	125	464	69	69	69	69	69	69
30	y=-63.52	69	269	172	1618	69	69	69	69	69	69
31	y=-53.90	69	171	276	69	69	69	69	69	69	69
32	y=-44.28	69	139	618	69	69	69	69	69	69	69
33	y=-34.66	69	110	1075	69	69	69	69	69	69	69

More sophisticated [daylight analysis tools](#) using Radiance are also available on the [Daylighting screen](#).

How To Generate And View Extra EnergyPlus Outputs

EnergyPlus provides literally thousands of different output report types and DesignBuilder allows you to access the most useful of these through the main Simulation results screen. Sometimes however it can be useful to access some the other reports and this help topic explains how to go about generating and viewing any EnergyPlus outputs. The process is illustrated through a [step by step tutorial](#) at the end.

Requesting The Extra Outputs

The first step is to request the extra outputs. Ultimately, this involves adding extra lines of IDF script to the EnergyPlus IDF input file and there are 2 ways to do this:

The easiest way is by using the [Add Output](#) Scripting dialog. This approach requires a licence for the [Scripting module](#).

Alternatively if you don't have access to the Scripting module you can use the more basic [Include IDF Data](#) option.

These 2 methods are described below in more detail.

Adding Extra Outputs Using the Scripting tools

If you have a licence for the Scripting module then the method below is the easiest way to add extra outputs and ensures that the extra outputs will be generated even if the model is sent to a colleague to simulate.

Click on the Scripts toolbar icon to open the Script Manager dialog and then Enable scripts.

Add a new EMS script, give it a name, enable the program and click on the blank script page.

Click on the Add Output link on the Info panel and, if asked about re-generating the list of available outputs, actuators and sensors, click Yes.

There will be a pause while EnergyPlus generates the list of available reports customised for the current model, after which the [Add Output](#) dialog is displayed. Here you can enter the object name (use * for all), select the interval and then select the output reports you need by double-clicking on the appropriate items in the list. Note that if your model is not in a state that allows simulations to run at this point, or if it is a large model and you don't want to wait for EnergyPlus to generate the RDD file, you can access the list of available outputs from a RDD file that was manually generated in a previous simulation and copy/paste report names to the script manually. The manual process is outlined in the tip in the [Adding Extra Output Reports using "Include IDF Data"](#) section below.

Once the required list of outputs has been set up press OK to confirm the settings and to create the script.

Press OK on the EMS Script dialog and the selected outputs will be included in the .eso results file after any subsequent simulations.

Note: You can find detailed descriptions on all of the available EnergyPlus outputs from the [Input Output Reference](#) document.

Adding Extra Output Reports Using "Include IDF Data"

If you don't have access to the Scripting module you can still request extra output reports using the [Include IDF Data](#) Calculation option. In this case you will need to write the IDF script manually, save it to an IDF file and reference that from the [Include IDF Data](#) Advanced setting on the Simulation Calculation Options dialog.

In more detail:

Generate the full list of EnergyPlus outputs available for your simulation by selecting the [RDD/EDD file](#) output option and running a test simulation. Note that if you used the Simulation Manager to run the simulation you must load results to DesignBuilder to be able to access the RDD file in the EnergyPlus folder.

After the simulation has completed, access the RDD file from the [EnergyPlus Folder](#) and open it. Search within the RDD file for the required variable names and copy them. If you are not sure exactly what a EnergyPlus variable name means, you can copy it and search for it within the relevant version of the [EnergyPlus InputOutput manual](#).

Save a text file using the idf extension with a list of all the new required variable names using the required IDF syntax. **The IDF file must be saved in the EnergyPlus Folder.**

Now select your custom IDF file from the Simulation Calculation Options dialog on the Options Tab, under the Advanced header. When you run your next simulation your custom IDF will be included in the input data provided to EnergyPlus.

After the simulation has completed, view the results using [Results Viewer](#).

Use Results Viewer's [filter](#) function to find the custom results of interest.

IDF Report Variable Format

The format used by EnergyPlus to define the "reports" (output data) required is described below.

A single "report" can be thought of as a single set of output data or alternatively as a line of data on a graph and is defined using a single line of text using this syntax:

```
Output:Variable, <Object name>, <Report Name>, <Interval>
```

`Output:Variable` is an IDF keyword which indicates that what follows is a request for output data.

`<Object name>` is the name of the object for which data is required, e.g. the zone or the HVAC component name. In some cases it is possible to enter "*" to indicate that the data is required for all relevant objects (e.g. all zones).

`<Report Name>` is the name of the report (output data) required. Thousands of reports are available in all and you should refer to the EnergyPlus help file for details on what is available.

`<Interval>` should be one of the following keywords:

Monthly

Daily

Hourly

Timestep - corresponds to the **sub-hourly** interval in DesignBuilder terminology.

Detailed - a special interval provided by EnergyPlus where detail down to the level of the HVAC timesteps can be provided. the actual intervals reported will often vary during the simulation, but it can at times provide useful debugging information.

For example to generate a new hourly data set for the "Time Heating Setpoint Not Met While Occupied" report for the zone having IDF name "Block1:Zone1" add this line of text to the IDF file:

```
Output:Variable, Block1:Zone1, Time Heating Setpoint Not Met While Occupied, hourly;
```

To generate sub-hourly data on the activity of the ideal loads economiser for *all* zones use:

```
Output:Variable, *, Ideal Loads Time Economizer Active, timestep;
```

Note the use of the "*" character to mean all objects and the semi-colon at the end of the line. For further clarification on how to specify requests for EnergyPlus outputs you can refer to the Output:Variable data provided by DesignBuilder in previously generated IDF files.

The above is just one way of obtaining outputs. Once familiar with EnergyPlus reports you can be creative with use of additional IDF to request further outputs.

Viewing Extra Output Reports

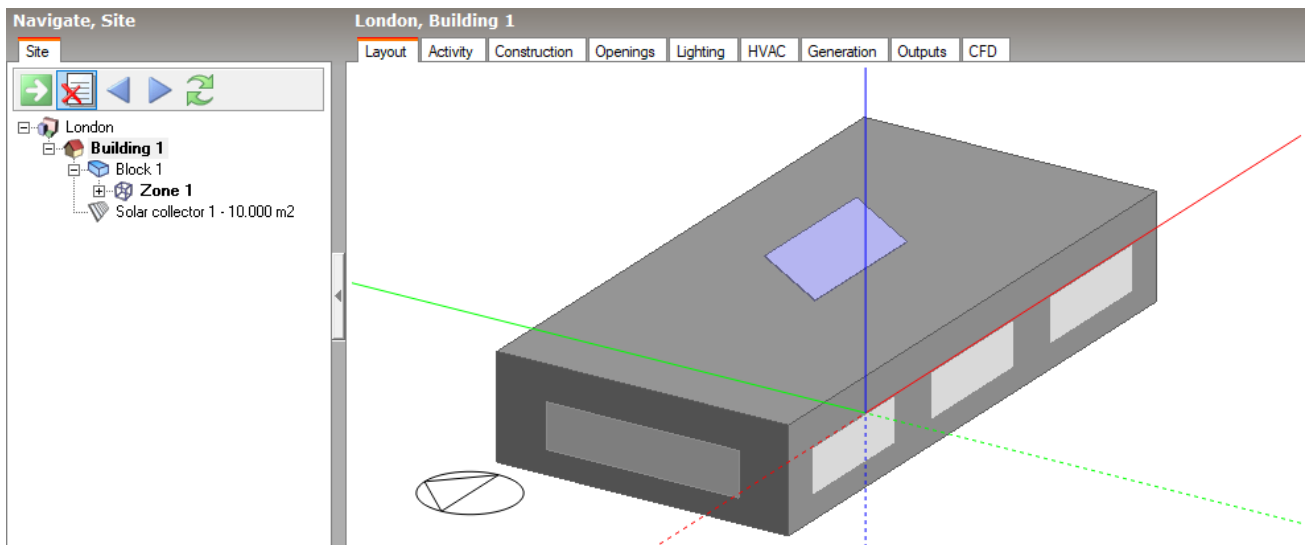
Regardless of the way you used to generate the extra outputs, the method used to view them is the same, namely through DesignBuilder's standalone [Results Viewer](#) application. If you are using the Simulation Manager you can do this by selecting the simulation(s) of interest and clicking on the **View results in DesignBuilder Results Viewer** toolbar icon. Otherwise you can use the **View EnergyPlus results** toolbar icon on the Simulation screen to open the most recently simulated results into the Results Viewer. More details on how to load eso results files into the Results Viewer can be found in the [Results Viewer help](#).

Tutorial - Viewing Direct And Diffuse Solar Radiation Incident On A PV Panel

This tutorial provides details on how to view extra outputs for PV panels that are not provided on the standard Simulation results screen. In particular it shows how to generate and view a breakdown of the direct and diffuse solar radiation incident on a PV Panel.

The tutorial assumes that you have access to the [Scripting module](#). If you do not then you can achieve the same results through the [Include IDF Data](#), though in this case the process is a little less convenient.

The screenshot below shows the simple model to be used to illustrate the method in this tutorial. It consists of a 20x10m rectangular block with a single zone and a 5x2m South-facing PV solar collector panel sitting on the roof at an angle of 30° from horizontal. The building is located in London/Gatwick UK.



Add Extra Outputs



Start by clicking on the Scripts toolbar icon to open the Script Manager dialog.

Check the **Enable scripts** checkbox and click on the **Script** selector option to open the list of existing scripts in the Info panel browse list on the right.

Click on the **Add new item** Info panel toolbar icon.

On the **Edit Script** dialog that pops up provide a name for the script, such as "Extra Solar Outputs" and check the **Enable program** checkbox to ensure that the script will be executed at Simulation run time.

Click on the **Add Output** link in the Info panel.

At this point you will see a message to say that no RDD data is stored in the model and would you like to regenerate the list. Answer Yes and there will be a short pause while EnergyPlus generates the list, after which the **Add Output** dialog is displayed.

Enter the object name. You could enter "*" to select all surfaces in the model, or to be more efficient in this case it would be better to enter "Solar collector 1", which is the IDF name of the surface used to represent the PV panel in the model.

Select **3-Hourly** as the interval.

Reading the relevant sections of the [EnergyPlus Input Output Reference](#) guide shows that the required outputs are:

Surface Outside Face Incident Sky Diffuse Solar Radiation Rate per Area

Surface Outside Face Incident Beam Solar Radiation Rate per Area and

Add these 2 reports to the list of selected outputs on the Add Output dialog by double-clicking on the appropriate items in the Info panel list. At this stage the Add Outputs dialog should look something like the screenshot below. When you have selected the required outputs press OK to confirm and add the

requested script to the main EMS scripting window.

Add Output

General

Data

Object IDF name: Solar Collector 1

Interval: 3-Hourly

Output

Surface Outside Face Incident Beam Solar Radiation Rate per Area

Surface Outside Face Incident Sky Diffuse Solar Radiation Rate per Area

The script should look like this:

Output:Variable, Solar Collector 1, Surface Outside Face Incident Beam Solar Radiation Rate per Area, Hourly;

Output:Variable, Solar Collector 1, Surface Outside Face Incident Sky Diffuse Solar Radiation Rate per Area, Hourly;

Press OK to confirm that and then also press OK on the Script Manager dialog to return to the main DesignBuilder window.

Run an annual simulation using the [Simulation Manager](#) by checking the **Use Simulation Manager** checkbox on the Simulation Manager tab of the Simulation Calculation Options dialog.

If all has gone according to plan you will now have generated the extra outputs.

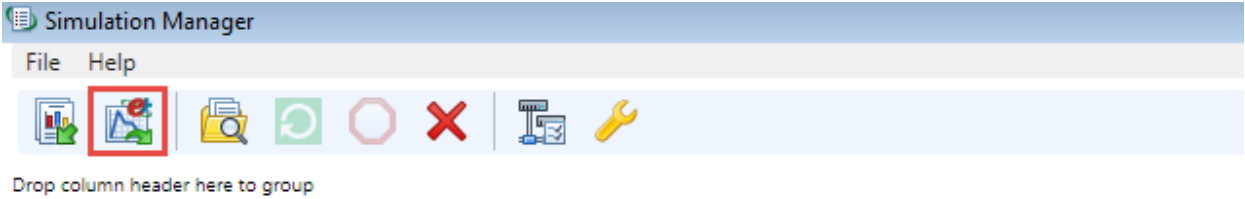
View the Results

If you have not already done so, [download](#) and install the latest version of the [Results Viewer](#) from the DesignBuilder website.

If the Simulation Manager dialog isn't already open from when you ran the simulation, click on the

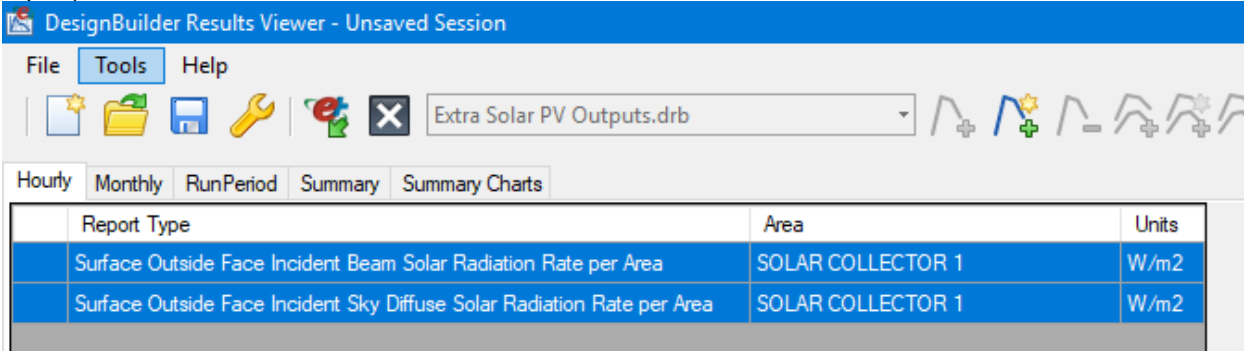
Simulation Manager toolbar icon  to open it.

Select the simulation you have just run and click on the **View results in DesignBuilder Results Viewer** toolbar icon as shown in the screenshot below.

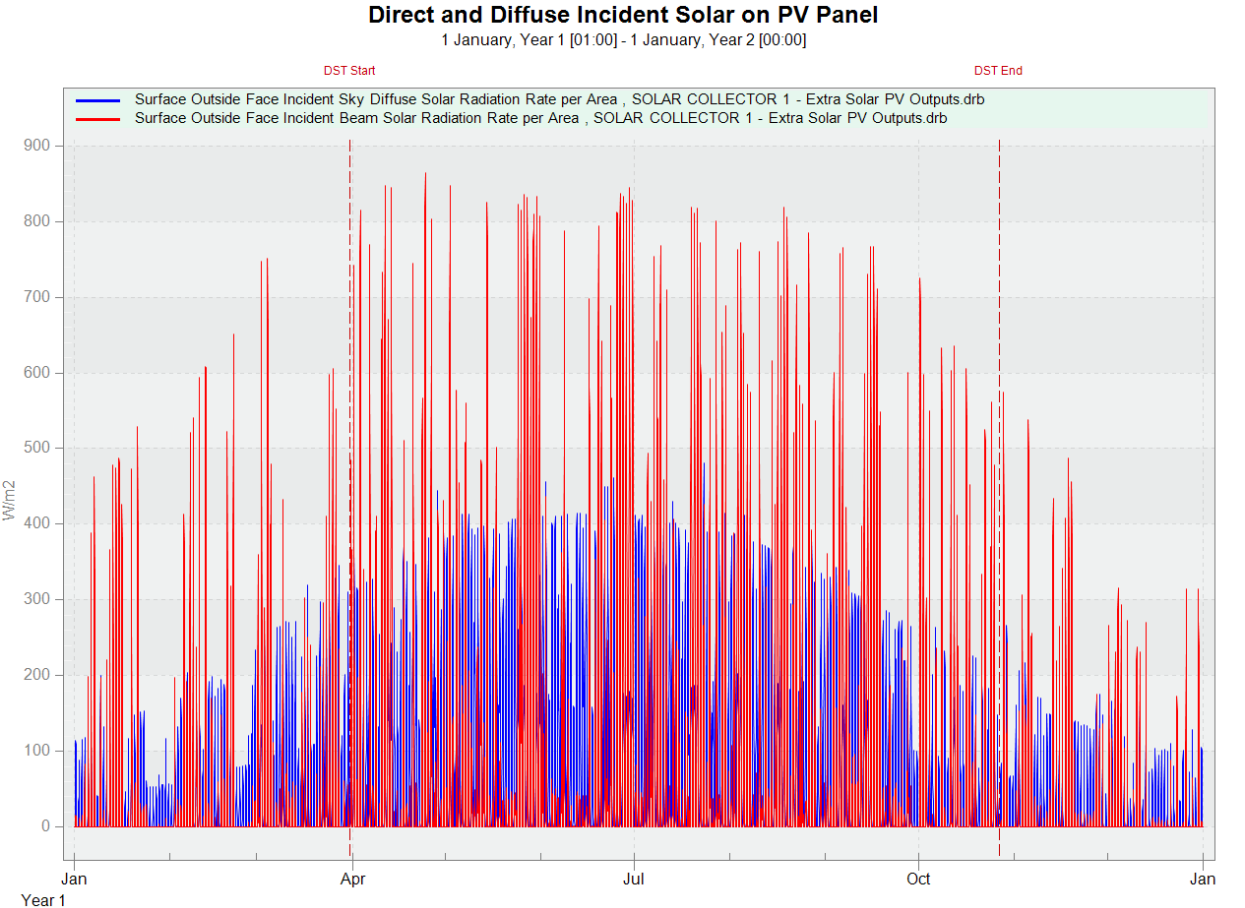


	Description	File Name
	Extra Solar PV Outputs	PV Panel Extra Outputs Example.dsb

In Results Viewer, click on the **Hourly** tab, select both the 2 extra outputs "Surface Outside Face Incident Beam Solar Radiation Rate per Area" and "Surface Outside Face Incident Sky Diffuse Solar Radiation Rate per Area" (hold the <Ctrl> key down and click with the mouse to select more than 1 report).

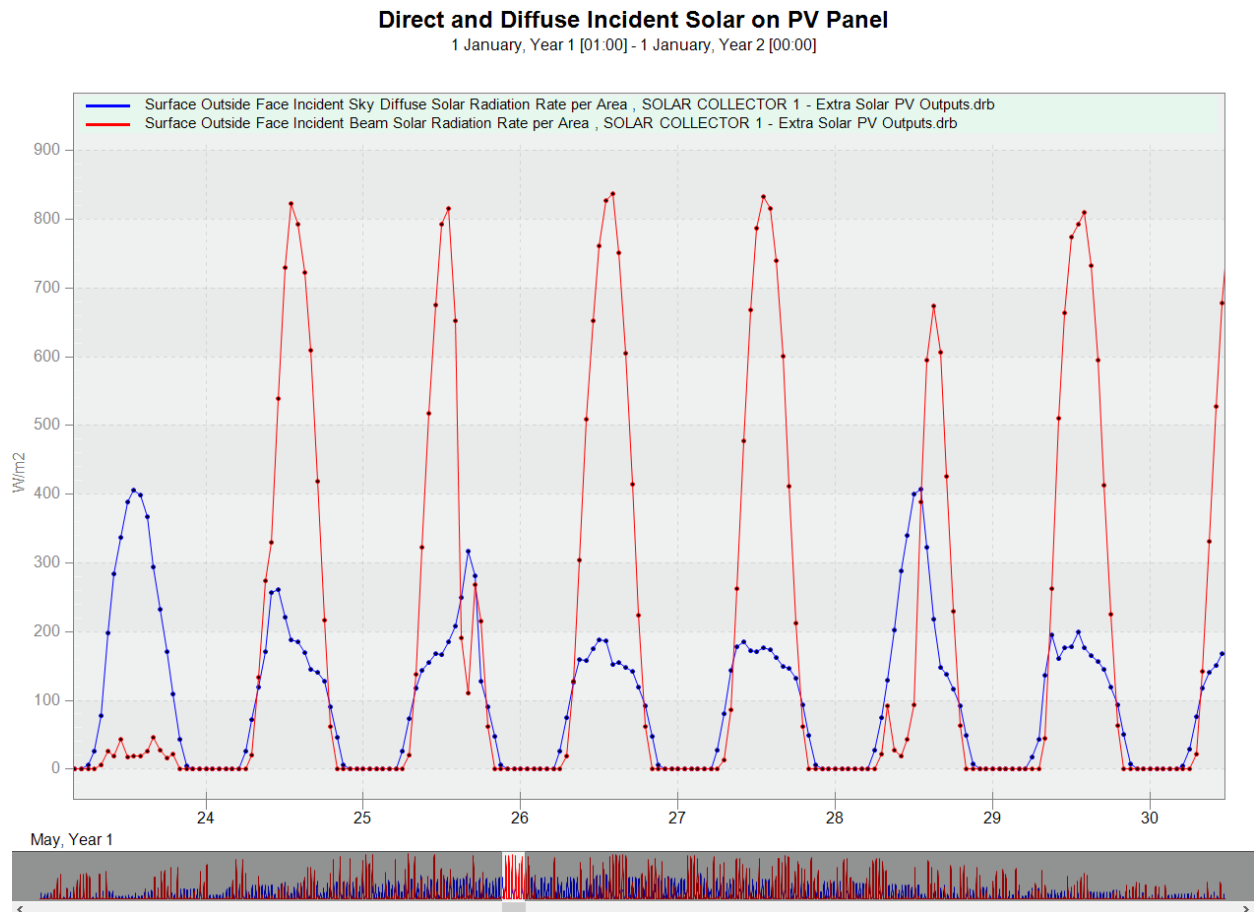


Click on the **Add selected variables to new graph** toolbar icon . Results similar to those below should be displayed.



Using the zoom tool to check hourly values of the direct and diffuse solar incident radiation on the panel

for a summer week:



DesignBuilder Files, Location And Extensions

Filename/Extension	Folder	Description
.dsb	Any local or network folder(DesignBuilder Data folder is default)	DesignBuilder's default model description file format. It contains a file of format .skh (below) and possibly other files that need to be included with the model such as any custom .epw weather files used in the model (below). Note that dsb files are already compressed so don't need to be 'zipped' before sending in emails.
.skh	Any local or network folder(DesignBuilder Data folder is default)	DesignBuilder's native binary model description file.
.xml	Any local or network folder(DesignBuilder Data folder is default)	DesignBuilder model description file in dsbXML form. This is an xml version of the skh file format but unlike skh files, results are not included.
.ddf		DesignBuilder exported component/template data
.dat	Library folder	Construction and Template exported files.
.lic	Any (DesignBuilder Data folder is default)	DesignBuilder Licence file for extended evaluation, activating licences when Internet access is not available or for training.
.idf	EnergyPlus folder	EnergyPlus input script file .
.stat	Weather data folder	EnergyPlus hourly weather data statistics file used by DesignBuilder to automatically load location and weather statistics to the

		Hourly weather data dialog when the epw file is selected. stat files are supplied with DesignBuilder and are automatically generated by the Hourly weather data translation process when creating custom weather data.
.epw	Weather data folder	EnergyPlus hourly weather data. a database of hourly weather data is supplied by automatic download or you can create your own hourly weather data . The Climate Analytics help provides a comprehensive description of the epw file format .
in.idf	EnergyPlus folder	Temporary file generated by DesignBuilder used as input to EnergyPlus - this file is overwritten before each simulation.
Energy+.idd	EnergyPlus folder	EnergyPlus data definition dictionary (see EnergyPlus documentation for information on this).
Eplusout.err	EnergyPlus folder	EnergyPlus error message file .
Eplusout.eso	EnergyPlus folder	Standard eso Output File - contains detailed results to be read in by DesignBuilder. This file can also be read by the Results Viewer .
Eplusout.eio	EnergyPlus folder	EnergyPlus model summary configuration output file.
Eplusout.rdd	EnergyPlus folder	Report Variable Data Dictionary - list all possible EnergyPlus output reports for a particular idf input file.
Eplusout.bnd	EnergyPlus folder	HVAC layout summary file - shows details about the branches, nodes, and other elements of the flow connections. It is intended for use in debugging potential problems and supports the EnergyPlus diagramming tool.
Eplusout.dxf	EnergyPlus folder	DXF file (from Report,Surfaces,DXF;)
Eplusout.mtr	EnergyPlus folder	Similar to .eso but only has Report Meter outputs.
Eplusout.end	EnergyPlus folder	A one line summary of success or failure used by DesignBuilder to indicate whether the simulation was successful or not.
Eplusout.dbg	EnergyPlus folder	From Debug Output object – may be useful to support to help track down problems.
EPlusmap.csv	EnergyPlus folder	For zones simulated with daylighting controls, an illuminance map will be generated with a user defined grid (max 10 x 10) of daylighting reference points. The resulting map is output as a comma delimited text file that can be imported into a spreadsheet program for rapid visualization of the daylighting illuminance patterns in a zone.
Eplusout.csv	EnergyPlus folder	Not directly from EnergyPlus - produced by EnergyPlus utility program that reads the .eso file and produces output files for spreadsheet programs.

Folders

Folder	Typical Location*	Use
--------	-------------------	-----

DesignBuilder Data	Documents\DesignBuilder Data	Default location for storing DesignBuilder dsb data files.
DesignBuilder Data\Backup	Documents\DesignBuilder Data	Default location for DesignBuilder automatic backup files and file copies made when opening existing files.
EnergyPlus	C:\Users\User\AppData\Local\DesignBuilder\EnergyPlus	Storing files related to EnergyPlus simulations.
Radiance	C:\Users\User\AppData\Local\DesignBuilder\Radiance	Storing files related to Radiance simulations.
Library data	C:\Users\User\AppData\Local\DesignBuilder	Library data and .dat files currently being used by DesignBuilder
Diagnostic files folder	C:\Users\User\AppData\Local\DesignBuilder	Diagnostic files
Template projects	C:\ProgramData\DesignBuilder\Templates	DesignBuilder dsb files stored in this folder can be used as templates for future models. Click on the Templates tab on the Create New Site dialog for a list of available template files.
Weather data	C:\ProgramData\DesignBuilder\Weather Data	EnergyPlus hourly weather data is stored in this folder. The data may have been automatically downloaded from the DesignBuilder website or it can be added manually .
Python scripting folder	C:\Program Files (x86)\DesignBuilder\Python27	Add any Python library files you need to access from your Python scripts here

* The actual names of the folders on a particular computer will depend on the language setting and on your Windows user name. The above locations assume English as the language. The Windows user name 'User' is used as an example.

Tip: the above folders can most easily be accessed from the [File > Folders](#) menu commands.

DesignBuilder XML (dsbXML)

DesignBuilder XML (dsbXML) is an alternative model file format that includes the full model description and is an XML equivalent of the SKH binary format that is compressed within standard DSB model files. File format aside, the only difference between dsbXML files and SKH files is that dsbXML files do not include calculation results.

DesignBuilder XML files have extension .xml and can be:

Exported using the [File > Export > Export model as XML](#) menu command, and,
Imported using the [File > Import > Import model from XML](#) menu command.

The dsbXML file format is especially useful in 2 ways:

It supports **forward compatibility**. In other words, you can import a dsbXML file exported in a more recent version of the software into an older version. Note that the earliest version to support dsbXML is v7.2. Like the other model file formats, dsb and skh, it also supports backward compatibility.

It can be read and written by 3rd party applications enabling **interoperability** with DesignBuilder without using the [API](#).

Note: The [Include library data in exported DesignBuilder XML](#) program option setting can be used to control whether library data is included in the exported dsbXML file.

Note: The [DesignBuilder XML User Guide](#) explains how to export and import models in XML, details the schema structure, object hierarchy and geometry definitions, and provides best practice for interoperability with third-party tools. Information is provided on how to include library data for self-contained packages, validate against the official schema, and generate models programmatically for integration with other applications. It also covers advanced topics such as HVAC networks, loops, and components, equipping you to create, edit, and manage XML files effectively.

See also:

[Export DesignBuilder XML \(dsbXML\)](#)
[Import DesignBuilder XML \(dsbXML\)](#)
[Include library data in exported DesignBuilder XML](#)

Hardware Requirements

You can find an up to date list of hardware requirements in the DesignBuilder [Hardware Requirements](#) Knowledgebase article.

Results Viewer

Standalone DesignBuilder Results Viewer v4.3

The Results Viewer is a separate application which can be used to view EnergyPlus results and other data stored in one or more .eso, .csv or .htm files. It can be downloaded from the main [Downloads > Software](#) area of the DesignBuilder website. When installed the application allows you to view any results contained within EnergyPlus .eso and .htm results files. There are various ways to open .eso results files:

From the Simulation screen in DesignBuilder click on the **View EnergyPlus results** toolbar icon. If you are not using the Simulation Manager, this will open the most recently simulated results into the Results Viewer. Alternatively if you are using the Simulation Manager to run simulations, this option will load the most recently loaded set of results for display in DesignBuilder into the Results Viewer.

From within the Simulation Manager, by selecting one or more simulations in the list and clicking on the [View results in DesignBuilder Results viewer](#) icon.

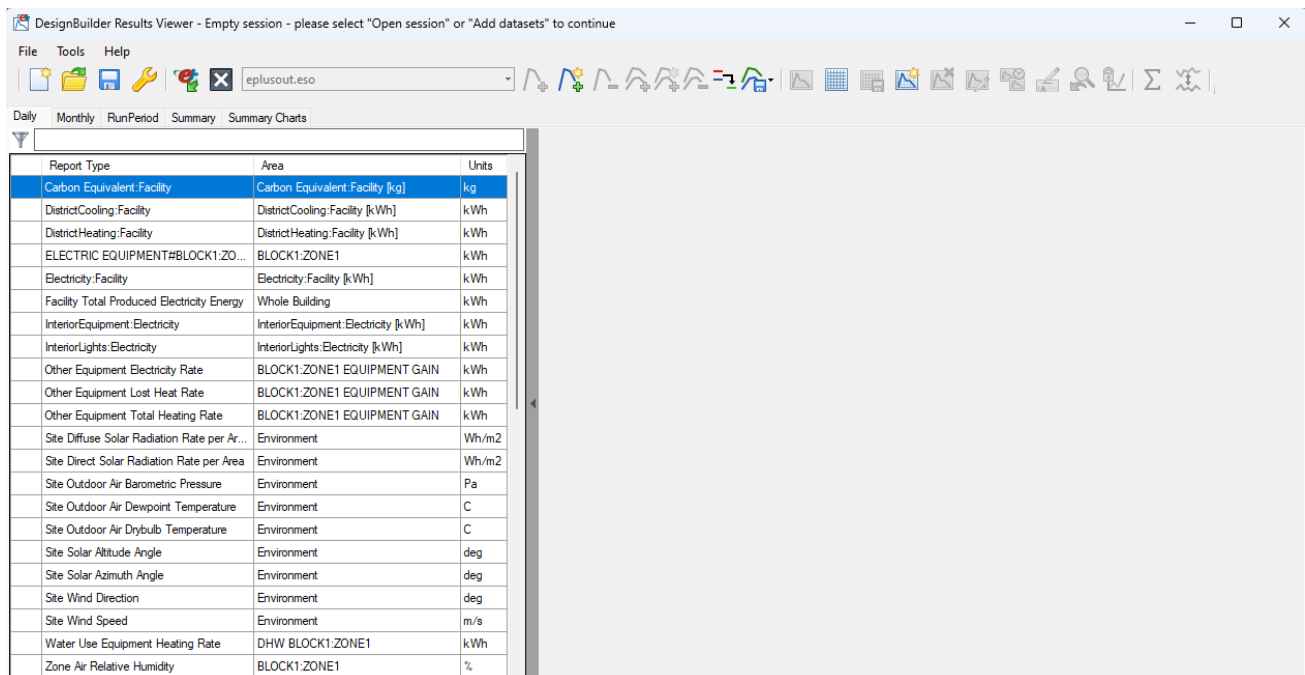
Double-click on an .eso file when in Windows Explorer.

From within the Results Viewer use the **File > Open eso/data set** menu command. You can open Results Viewer from the DesignBuilder folder when clicking on the Windows Start button.

By making the [appropriate selection on the Program options](#) DesignBuilder will offer to open the .eso file at the end of the simulation.

Multiple .eso, .csv .epw and .htm files can be opened at a time. Use the combo box below the toolbar to select the current results set for plotting.

With an .eso file open the first view will be something like that shown below.



DesignBuilder Results Viewer - Empty session - please select "Open session" or "Add datasets" to continue

File Tools Help

Day Monthly RunPeriod Summary Summary Charts

Report Type	Area	Units
Carbon Equivalent: Facility	Carbon Equivalent: Facility [kg]	kg
DistrictCooling: Facility	DistrictCooling: Facility [kWh]	kWh
DistrictHeating: Facility	DistrictHeating: Facility [kWh]	kWh
ELECTRIC EQUIPMENT#BLOCK1.ZO...	BLOCK1.ZONE1	kWh
Electricity: Facility	Electricity: Facility [kWh]	kWh
Facility Total Produced Electricity Energy	Whole Building	kWh
InteriorEquipment: Electricity	InteriorEquipment: Electricity [kWh]	kWh
InteriorLights: Electricity	InteriorLights: Electricity [kWh]	kWh
Other Equipment Electricity Rate	BLOCK1.ZONE1 EQUIPMENT GAIN	kWh
Other Equipment Lost Heat Rate	BLOCK1.ZONE1 EQUIPMENT GAIN	kWh
Other Equipment Total Heating Rate	BLOCK1.ZONE1 EQUIPMENT GAIN	kWh
Site Diffuse Solar Radiation Rate per Ar...	Environment	Wh/m2
Site Direct Solar Radiation Rate per Area	Environment	Wh/m2
Site Outdoor Air Barometric Pressure	Environment	Pa
Site Outdoor Air Dewpoint Temperature	Environment	C
Site Outdoor Air Drybulb Temperature	Environment	C
Site Solar Altitude Angle	Environment	deg
Site Solar Azimuth Angle	Environment	deg
Site Wind Direction	Environment	deg
Site Wind Speed	Environment	m/s
Water Use Equipment Heating Rate	DHW BLOCK1.ZONE1	kWh
Zone Air Relative Humidity	BLOCK1.ZONE1	%

See also the [Simulation Manager and Results viewer](#) tutorial

Details on specific areas of Results Viewer functionality can be found at:

"Results Viewer Data Display" on page 2217

"Results Viewer Menu And Toolbar Commands" on page 2231

"Results Viewer Options Dialog" on page 2235

"Results Viewer Supported Dataset File Formats" on page 2241

Sessions

It can take some time to load .eso files and to select results so DesignBuilder provides methods to save pre-processed results files and session files to speed loading and setting up reports the next time.

.drb results files

When the Results Viewer loads an .eso file it automatically generates a .drb file with exactly the same data but in a form that can be loaded much more quickly. If you need to view the results again in future you can open the drb file instead of the .eso file. The .drb file will have the same filename as the original .eso file (apart from the extension). It does not contain any display settings - just the data.

.drs results files

You can also save a session file which stores all of your display settings for a graphing session as well as the corresponding .drb file(s). Opening the session file will take you back to where you were before saving the session file. Session files provide a very useful way to package up all data and settings for a Results Viewer session in a small file size. They can be sent to colleagues for viewing.

DesignBuilder Program option settings

You can configure DesignBuilder to save .eso files in various ways as described for the [EnergyPlus tab of the Program options](#). If you plan to use the Results Viewer rather than the inbuilt DesignBuilder results display then you might use one of the settings configurations below.

Note: The **Save ESO simulation output, Prompt for filename** and **Auto load ESO** program options only apply when the Simulation Manager is not being used.

Output file ☐ Save ESO simulation output	The default option where the .eso file is automatically generated and named as eplusout.eso in the DesignBuilder EnergyPlus folder. This .eso file will be overwritten each time a simulation or design calculation is run. The eplusout.eso file is automatically loaded to DesignBuilder after the simulation. You may wish to avoid this option if you have generated very large hourly or sub-hourly data sets which could cause long delays or a program crash if loaded.
Output file ☒ Save ESO simulation output ☐ Prompt for filename ☐ Auto load results from ESO file?	The .eso file is automatically generated and named and an option provided to open it in the Results viewer. The results data is not automatically loaded to DesignBuilder, which might be a usual option to avoid very large hourly or sub-hourly data sets from being loaded.
Output file ☒ Save ESO simulation output ☒ Prompt for filename ☐ Auto load results from ESO file?	The .eso file is automatically generated and you are prompted for a filename for the .eso file and asked if you would like to open it in the Results viewer. The results data is not automatically loaded to DesignBuilder, which might be a usual option to avoid very large hourly or sub-hourly data sets from being loaded.
Output file ☒ Save ESO simulation output ☒ Prompt for filename ☒ Auto load results from ESO file?	The .eso file is automatically generated and you are prompted for a filename for the .eso file and asked if you would like to open it in the Results viewer. The results data is also loaded to DesignBuilder for your convenience. You may wish to avoid this option if you have generated very large hourly or sub-hourly data sets which could cause long delays or a program crash if loaded.

Diagnostics

Results Viewer generate 2 reports which can be useful to help solve any software issues and may be requested by the Support Desk in the case of an issue report:

Error / crash reports.
Diagnostics.

These reports can be accessed as described below.

Error reports

In the case where a severe error or crash occurs in Results Viewer an entry is written out to an error log file which can be accessed from the folder:

C:\Users\<Username>\AppData\Local\DesignBuilder\DBResults\Errors\

File names are in the form:

Error_<Date>_<Time>.txt.

For example:

error_03-01-2023_11-47-14.txt

The error logs include details of the error or crash and can be sent to the Support Desk as part of an enquiry. These reports are always generated and do not need a Registry setting to be made.

Diagnostics reports

When the necessary Registry entry has been made (see below) ResultsViewer writes out a log of the main steps it takes in the program execution. Logs files can be found at:

C:\Users\<Username>\AppData\Local\DesignBuilder

File names are in the form:

ResultsViewer<Date>.log.

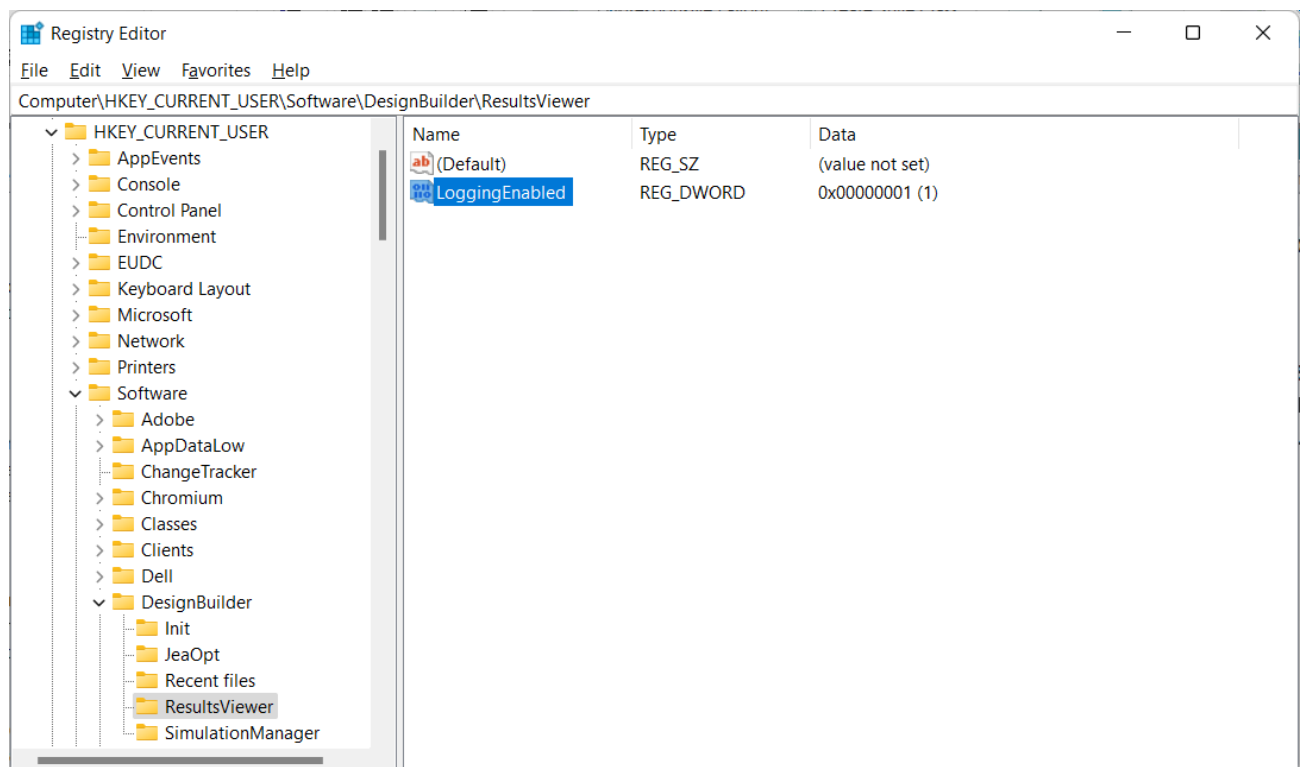
For example:

ResultsViewer230127.log

In order to generate diagnostics logs a manual Registry entry needs to be made in the folder:

Computer\HKEY_CURRENT_USER\Software\DesignBuilder\ResultsViewer

The name of the Registry entry is **LoggingEnabled**, it should be a DWORD type and should have the value of 1 to enable diagnostics logging. A screenshot showing the setting in the Registry Editor is shown below.



Results Viewer Data Display

When a dataset is loaded from a DesignBuilder simulation the associated eso and htm results files are loaded together. Depending on the data that was generated in the simulation, various tabs may be available:

[Timestep, Hourly, Daily, Monthly, Run period](#)

[Hourly Heat Map](#)

[CIBSE TM52](#)

[CIBSE TM59](#)

[Summary](#)

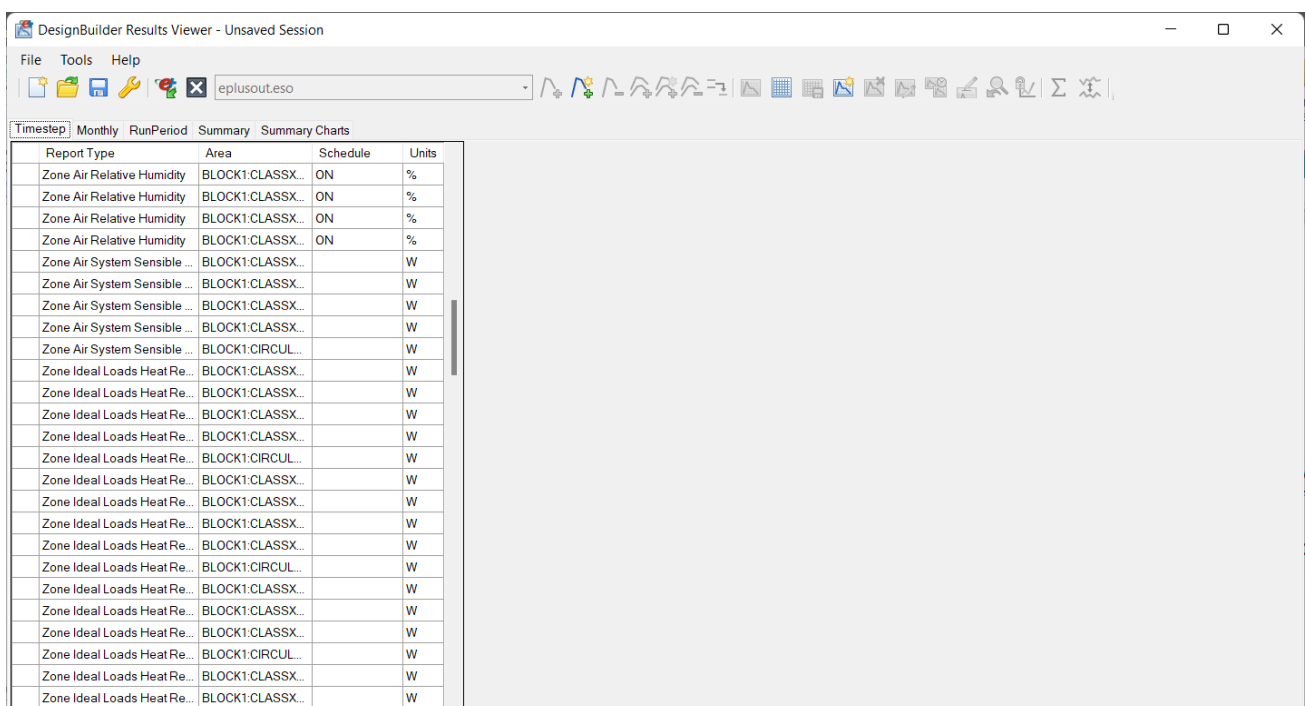
[Summary Charts](#)

Other topics on this page:

[Working with Multiple Data sets](#)
[Zooming](#)
[Create Composite Plot](#)
[Crosshair and Tooltips](#)
[Data Comparison](#)
[Styling](#)

Detailed Results - Timestep, Hourly, Daily, Monthly, Run Period Tabs

To show detailed results for a particular interval click on one of the **Timestep**, **Hourly**, **Daily**, **Monthly** or **Run period** interval tabs. When one of the interval tabs is selected the **Reports grid** is displayed on the left of the screen with a list of the available reports available for the selected interval. On the right of the screen is the area where the graph(s) are displayed. The screenshot below shows the Results Viewer screen after loading an eso file and an associated htm file and before any reports have been selected for viewing in the Reports grid.



Reports grid

The Reports grid provides a list of the available reports for the currently selected dataset. One or more reports can be selected for plotting from the grid.

The grid display includes several columns some of which are always displayed and others that are optional depending on settings on the [Graph](#) tab of the [Options](#) dialog:

Icon - a green icon with a tick is displayed here alongside any reports currently selected for display.

Report Type - the name of the report is displayed in this column. These names are predefined by EnergyPlus and are unaffected by the language settings selected on the Options dialog.

Area - the name of the zone, HVAC component or other building element that the report provides data for.

Schedule - if the report only includes data for certain periods then the name of the schedule that controls the active periods is displayed in this column. This optional column is displayed only when the [Schedules](#) option is set to **On** or **Auto** and one or more reports have a schedule associated with them.

Units - the units that apply to this report.

Source Units - the units of the report originally read in from the eso file. This may be different from the units used in the display, in particular for energy and power reports and for most reports when displaying results in [IP units](#). This optional column is displayed only when the [Display original dataset units in grid](#) option is selected.

Timestep	Hourly	Daily	Monthly	RunPeriod	Hourly Heat Map	Summary	Summary Charts			
	Report Type					Area		Schedule	Units	Source Units
	Zone Infiltration Sensible Heat Loss Energy					GROUND FLOOR: WAREHOUSE			W	J
	Zone Infiltration Sensible Heat Loss Energy					GROUND FLOOR: OFFICE			W	J
	Zone Interior Windows Total Transmitted Beam Solar Radiatio...					GROUND FLOOR: WAREHOUSE			W	W
	Zone Interior Windows Total Transmitted Beam Solar Radiatio...					GROUND FLOOR: OFFICE			W	W
	Zone Interior Windows Total Transmitted Diffuse Solar Radiati...					GROUND FLOOR: OFFICE			W	W
	Zone Interior Windows Total Transmitted Diffuse Solar Radiati...					GROUND FLOOR: WAREHOUSE			W	W
✓	Zone Mean Air Temperature					GROUND FLOOR: OFFICE		ON	C	C
✓	Zone Mean Air Temperature					GROUND FLOOR: WAREHOUSE		ON	C	C
	Zone Mean Radiant Temperature					GROUND FLOOR: OFFICE		ON	C	C
	Zone Mean Radiant Temperature					GROUND FLOOR: WAREHOUSE		ON	C	C
	Zone Mechanical Ventilation Standard Density Volume Flow R...					GROUND FLOOR: WAREHOUSE			m3/s	m3/s
	Zone Mechanical Ventilation Standard Density Volume Flow R...					GROUND FLOOR: OFFICE			m3/s	m3/s
	Zone Operative Temperature					GROUND FLOOR: OFFICE		ON	C	C
	Zone Operative Temperature					GROUND FLOOR: WAREHOUSE		ON	C	C
	Zone People Sensible Heating Rate					GROUND FLOOR: OFFICE			W	W
	Zone People Sensible Heating Rate					GROUND FLOOR: WAREHOUSE			W	W

To plot a report on a graph, first select one or more reports on the grid by clicking with the mouse. Use <Ctrl>+ left mouse click to add more reports to the selection set. Then use one of these methods to display the data:

Select one of the toolbar options **Add selected row to current graph** or **Add selected row to new graph**.

Right-click on the item in the grid to display a context menu and select the **Add selected row to new graph** menu option to add the report to a new graph.

Double-clicking on the report in the grid will add it to the current graph (or if no graph exists it creates a new graph and adds it).

Filtering the reports in the grid can help to find particular data more quickly. To filter the reports displayed, click anywhere on the grid so it has the focus and press <Ctrl-F>. This displays the Filter text box which allows you to type in a criterion to be used to filter the reports. Any rows whose **Report Type**, **Area** or **Schedule** (when displayed) data includes the criterion will be displayed and any rows that do not are hidden.

To hide the Filter box press <Ctrl-F> again or press <Esc>.

Pressing the x button to the right of the Filter box clears the criterion text.

Hourly

Hourly Heat Map

Summary

Summary Charts



air temperature



	Report Type	Area	Schedule	Units
	Zone Mean Air Temperature	WITHINFILTRATION:MECHVENTXWITHHR	ON	C
	Zone Mean Air Temperature	NOINFILTRATION:MECHVENTXWITHHR	ON	C
	Zone Mean Air Temperature	WITHINFILTRATION:MECHVENTXNOHR	ON	C
	Zone Mean Air Temperature	NOINFILTRATION:MECHVENTXNOHR	ON	C
	Zone Mean Air Temperature	WITHINFILTRATION:NATVENT	ON	C
	Zone Mean Air Temperature	NOINFILTRATION:NATVENT	ON	C

Sorting the reports can also help quickly find particular data. To sort the data by a particular column click on the column header that you wish to sort by. For example to see data sorted by Area click on the **Area** header. This will collect together all data for each zone, HVAC component, Environment etc. in the list. In the screenshot below the data is sorted by Area and user has scrolled to view all of the data for the BLOCK1:CLASS8X14 zone:

DesignBuilder Results Viewer - Unsaved Session

File Tools Help

epplusout.eso

Timestep Monthly RunPeriod Summary Summary Charts

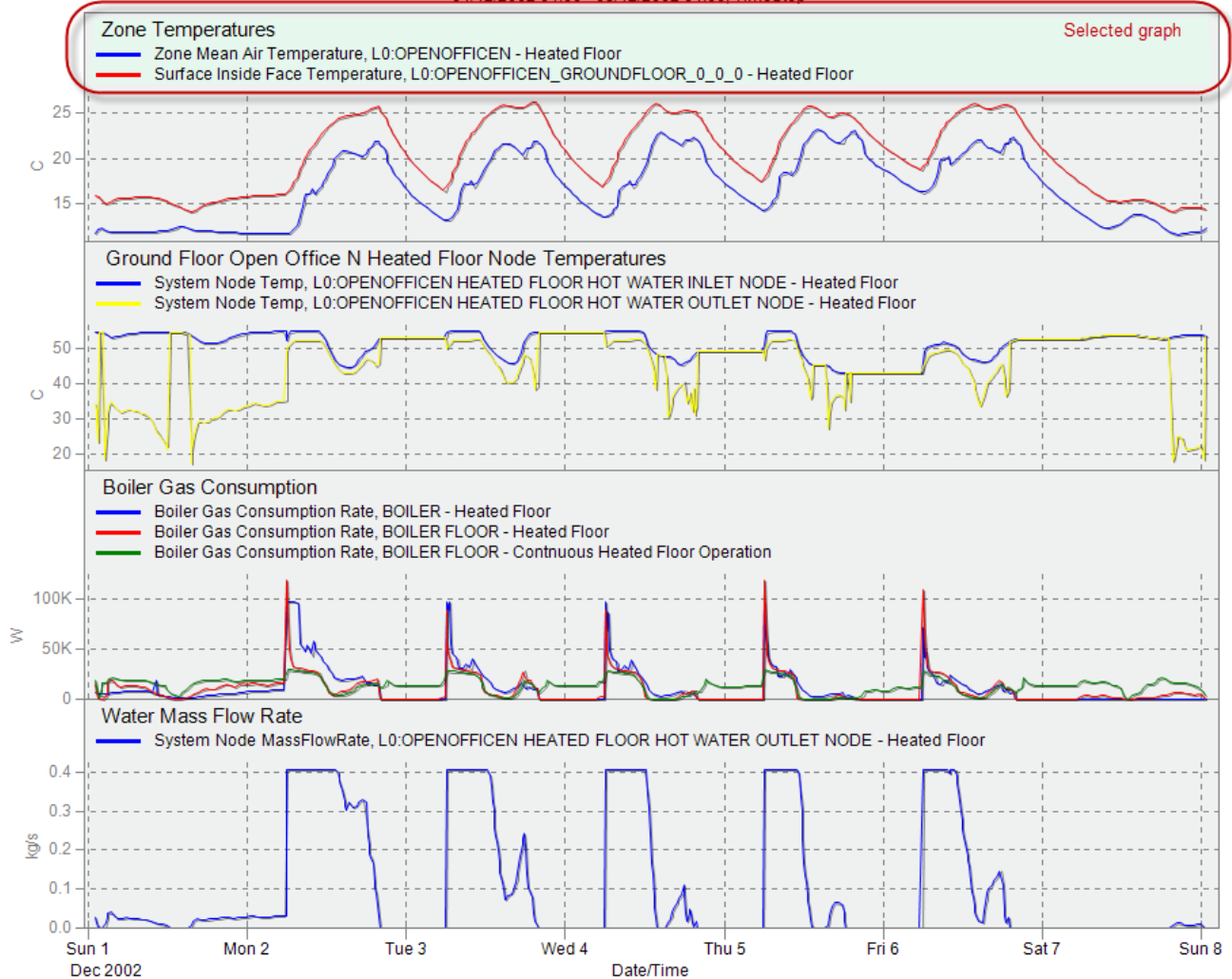
Report Type	Area	Schedule	Units
Zone Ideal Loads Heat Recovery Total He...	BLOCK1:CLASSX6X...		W
Zone Ideal Loads Heat Recovery Sensible ...	BLOCK1:CLASSX6X...		W
Zone Ideal Loads Heat Recovery Sensible ...	BLOCK1:CLASSX6X...		W
Zone Ideal Loads Heat Recovery Total Co...	BLOCK1:CLASSX6X...		W
Zone Ventilation Standard Density Volume ...	BLOCK1:CLASSX8X14		m3/s
Zone Mechanical Ventilation Standard Den...	BLOCK1:CLASSX8X14		m3/s
Zone Infiltration Sensible Heat Loss Energy	BLOCK1:CLASSX8X14		W
Zone People Sensible Heating Rate	BLOCK1:CLASSX8X14		W
Zone Ventilation Sensible Heat Loss Energy	BLOCK1:CLASSX8X14		W
Zone Infiltration Air Change Rate	BLOCK1:CLASSX8X14		ach
Zone Ventilation Sensible Heat Gain Energy	BLOCK1:CLASSX8X14		W
Zone Ventilation Air Change Rate	BLOCK1:CLASSX8X14		ach
Zone Ventilation Fan Electricity Energy	BLOCK1:CLASSX8X14		W
Zone Operative Temperature	BLOCK1:CLASSX8X14	ON	C
Zone Mechanical Ventilation Air Changes p...	BLOCK1:CLASSX8X14		ach
Zone Windows Total Transmitted Solar Ra...	BLOCK1:CLASSX8X14		W
Zone Air Relative Humidity	BLOCK1:CLASSX8X14	ON	%
Zone Interior Windows Total Transmitted Dif...	BLOCK1:CLASSX8X14		W
Zone Mean Air Temperature	BLOCK1:CLASSX8X14	ON	C
Zone Mean Radiant Temperature	BLOCK1:CLASSX8X14	ON	C
Zone Interior Windows Total Transmitted Be...	BLOCK1:CLASSX8X14		W
Zone Air System Sensible Heating Rate	BLOCK1:CLASSX8X14		W
Zone Infiltration Sensible Heat Gain Energy	BLOCK1:CLASSX8X14		W
Other Equipment Lost Heat Rate	BLOCK1:CLASSX8X...		W
Other Equipment Total Heating Rate	BLOCK1:CLASSX8X...		W

Selecting a graph

If you have more than 1 graph set up you can select the current graph by clicking on it. You will see the graph heading highlight in a different colour blue when selected as shown below.

Heated Floor Analysis for Intermittent and Continuous Operation

01/12/2002 01:00 - 08/12/2002 01:00, TimeStep



Summary And Summary Charts Results

Summary tab

On the Summary tab you will find the summary results report generated by EnergyPlus in html format. This report consists of a series of tables containing simulation output results. It is the same report that is displayed on the [Summary tab](#) of the DesignBuilder Simulation screen after results have been loaded. Results tables can be requested in the DesignBuilder Simulation [Output options](#).

For example the screenshot below shows the "Electric Loads Satisfied" summary table for the whole building.

Electric Loads Satisfied

	Electricity [kWh]	Percent Electricity [%]
Fuel-Fired Power Generation	0.000	0.00
High Temperature Geothermal*	0.000	0.00
Photovoltaic Power	19005.828	176.84
Wind Power	0.000	0.00
Power Conversion	-1900.58	-17.7
Net Decrease in On-Site Storage	0.000	0.00
Total On-Site Electric Sources	17105.245	159.16
Electricity Coming From Utility	3716.427	34.58
Surplus Electricity Going To Utility	10074.389	93.74
Net Electricity From Utility	-6357.96	-59.2
Total On-Site and Utility Electric Sources	10747.283	100.00
Total Electricity End Uses	10747.283	100.00

The Summary report includes a useful Table of Contents which you can find at any point by clicking on a Table of Contents link to the right of each major header, e.g. at the top of the report, see below:

Program Version: **EnergyPlus, Version 9.4.0-217a24fc09, YMD=2023.04.04 16:16**

[Table of Contents](#)

Tabular Output Report in Format: **HTML**

Building: **Building**

Environment: **GREENPOWER (01-01:31-12) ** 4450115 - UK WG data WMO#=03862**

Simulation Timestamp: **2023-04-04 16:16:49**

Report: **Annual Building Utility Performance Summary**

[Table of Contents](#)

For: **Entire Facility**

Timestamp: **2023-04-04 16:16:49**

Values gathered over **8760.00** hours

Site and Source Energy

	Total Energy [kWh]	Energy Per Total Building Area [kWh/m2]	Energy Per Conditioned Building Area [kWh/m2]
Total Site Energy	33448.32	109.83	115.82
Net Site Energy	16343.07	53.67	56.59
Total Source Energy	58644.57	192.57	203.07
Net Source Energy	4472.26	14.69	15.49

Site to Source Energy Conversion Factors

	Site=>Source Conversion Factor
Electricity	3.167
Natural Gas	1.084

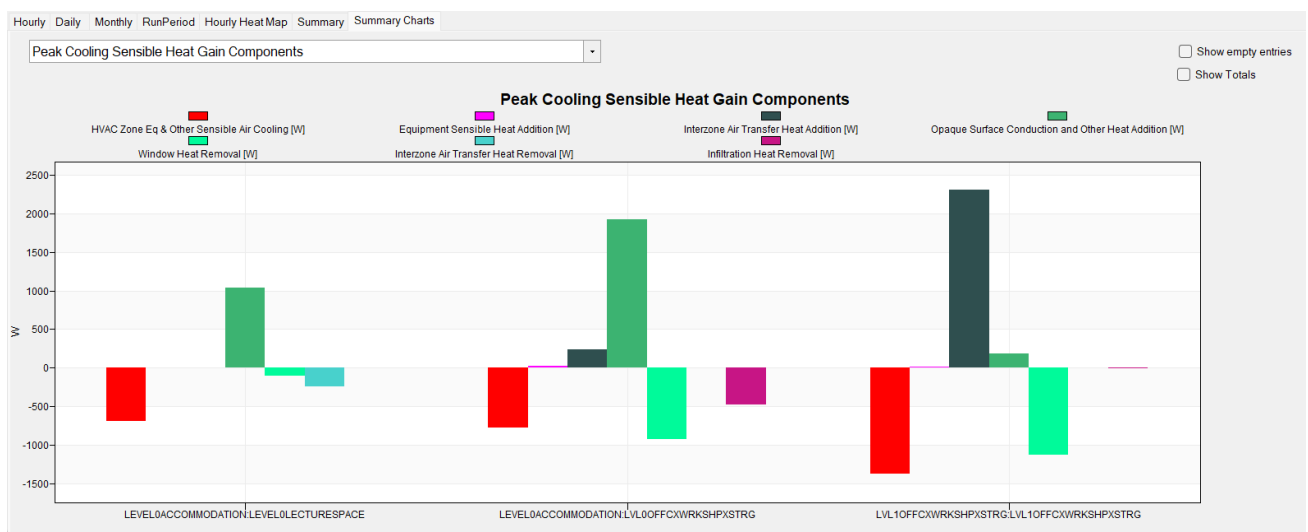
The Table of Contents includes a link to each major section in the report:

Table of Contents

[Top](#)
[Annual Building Utility Performance Summary](#)
[Input Verification and Results Summary](#)
[Demand End Use Components Summary](#)
[Source Energy End Use Components Summary](#)
[Component Cost Economics Summary](#)
[Climatic Data Summary](#)
[Lighting Summary](#)
[Equipment Summary](#)
[HVAC Sizing Summary](#)
[System Summary](#)
[Outdoor Air Summary](#)
[Energy Meters](#)
[Sensible Heat Gain Summary](#)
[Standard 62.1 Summary](#)
[LEED Summary](#)

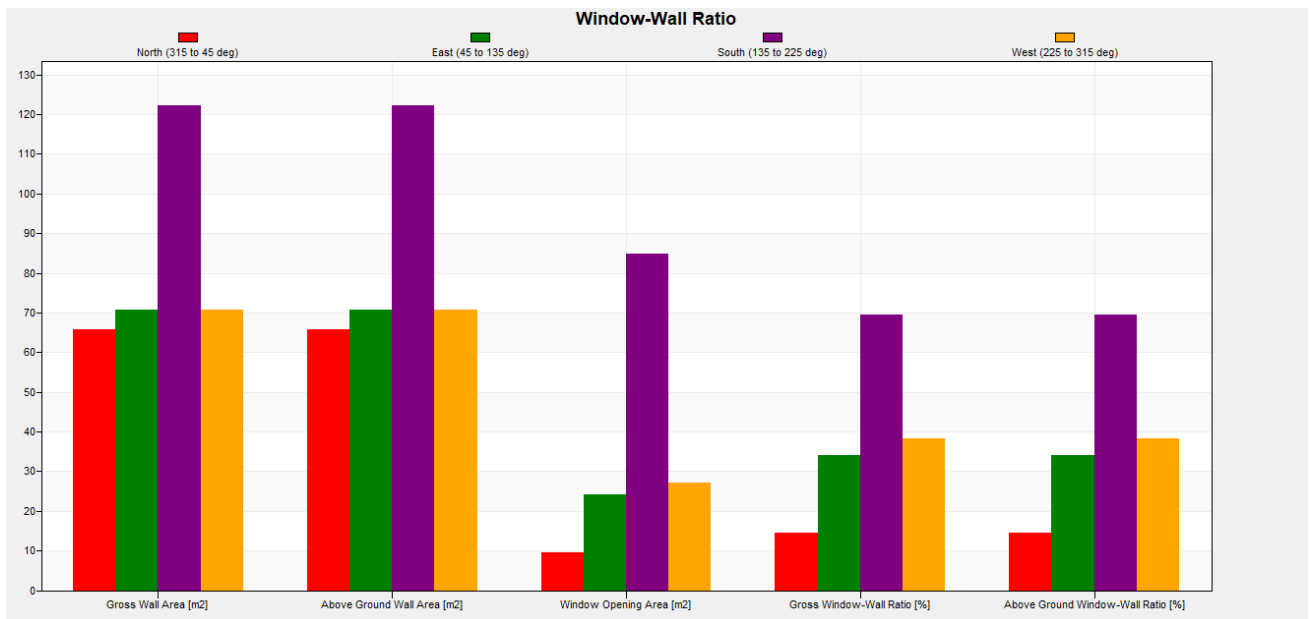
Summary Charts tab

This tab allows most of the annual summary results in the tables on the Summary tab to be plotted graphically in histogram form. For example the screenshot below shows "Peak Cooling Sensible Heat Gain Components" outputs for all cooled zones.



You can choose whether or not to include totals and empty entries by clicking on the appropriate checkboxes on the top right of the screen. Check the **Show totals** checkbox to display a histogram to the right with the building totals. Check the **Show empty entries** checkbox to show empty histograms when all values are zero.

You can also use the this tab to display graphical summaries of some aspects of model input data. For example the screenshot below shows wall and window areas, the window to wall ratio (WWR) for each zone for each of the 4 major directions.



Hourly Heat Map

The **Hourly Heat Map** tab is displayed when 1 or more report with data at hourly intervals have been loaded. It allows you to display a year or more of hourly data on one screen, with each day of the year displayed as a vertical bar and labelled on the x-axis and time of day on the y-axis and with each hour corresponding to a small cell. That cell is assigned a false colour based on the value for that hour. This way of displaying hourly data can be especially useful for quickly identifying diurnal and seasonal trends as well as periods of particular interest that may go unnoticed when only plotted as a time series.

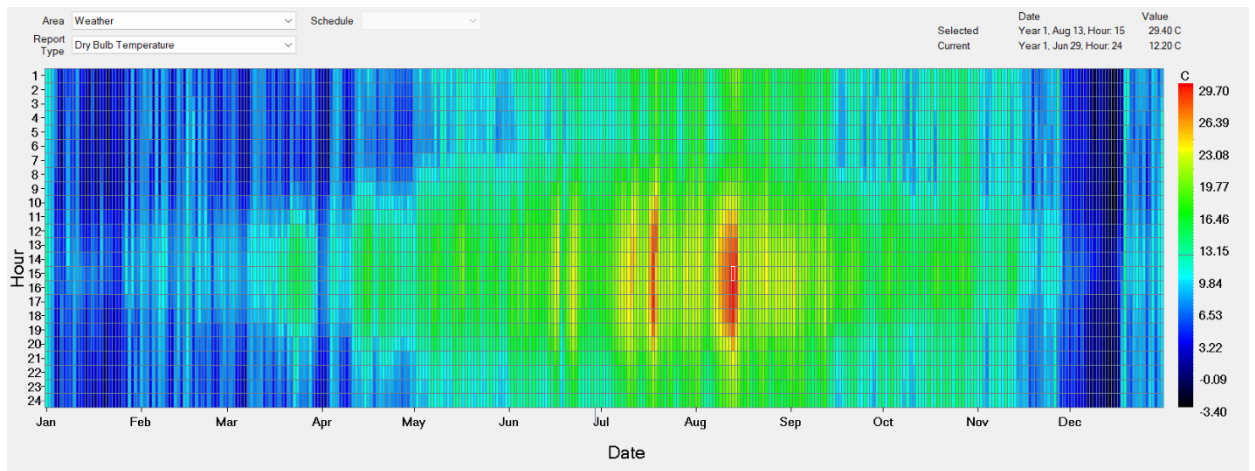
To obtain a readout of the numeric results you can click on a cell and the Selected value is displayed in the top right of the screen. As you move the mouse across the heat map the Current value is also updated.

The Options dialog allows some [hourly heat map display settings](#) to be controlled.

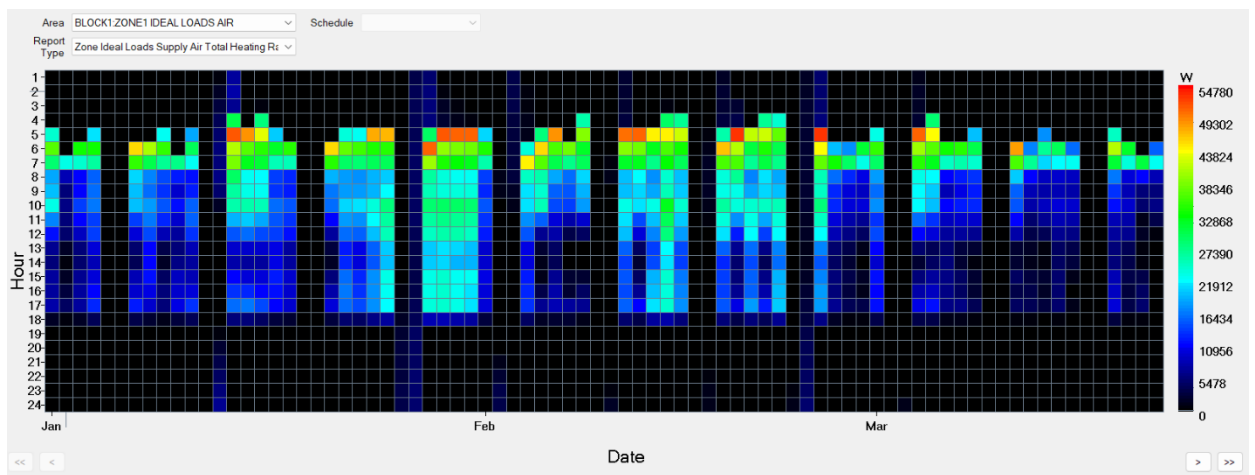
Tip: Any hourly data can be displayed in heat map form including standard outputs, weather data and custom hourly data generated in additional IDF outputs or EMS scripts. See "How to Generate and View Extra EnergyPlus Outputs" on page 2205.

Some example applications of heat maps include:

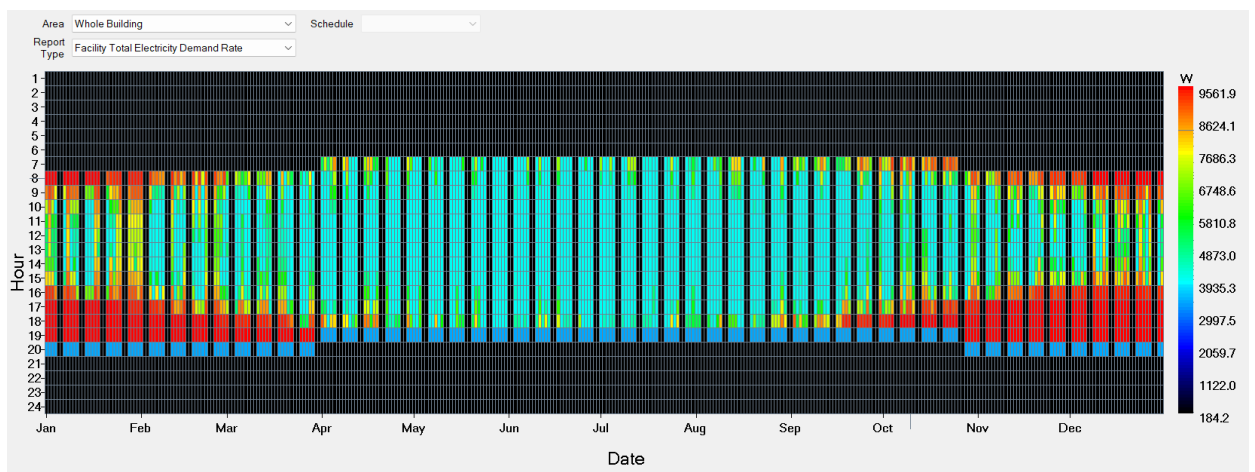
Analysing and comparing hourly weather data sets, quickly identifying particularly hot, cold and/or sunny periods for use as design periods in your simulations. The image below shows the dry-bulb temperature for 2022 for a location in the UK. An extended hot period in mid-August is clearly displayed in red. This could be an ideal series of days to analyse in a natural ventilation study.



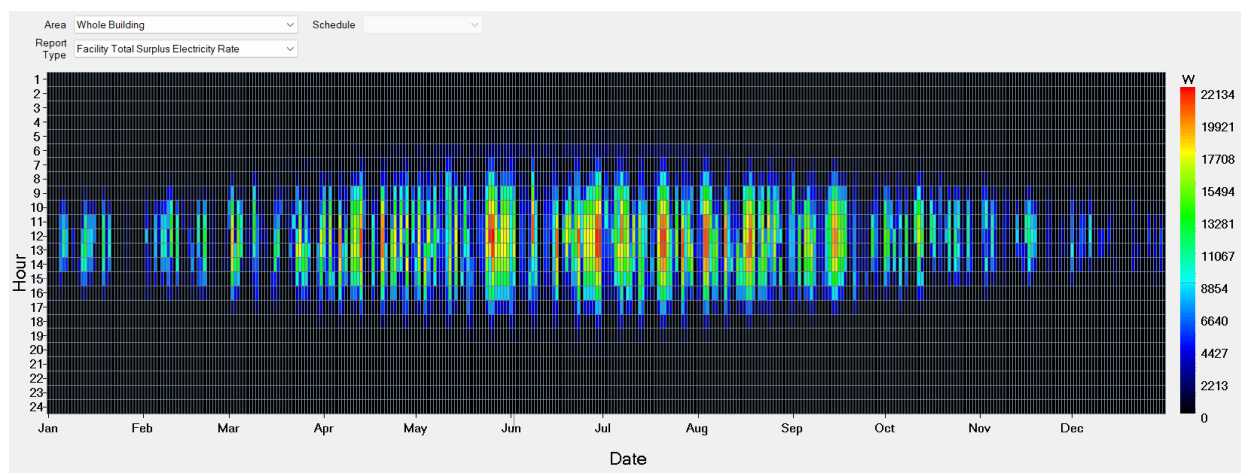
Analysing building heating consumption to identify periods when: optimum start control heating controls start the heating system earlier on cold mornings; when heating energy usage peaks and when set back heating operates etc. The image below illustrates with an example of 80 days of heating system operation.



Checking seasonal and diurnal building total electricity demand patterns. The example output below shows how lighting dimming control can reduce electricity consumption during summer months:



Analysing periods when renewable energy systems supply surplus generated electricity to the grid:



CIBSE TM52 Results

When loading an eso file that includes [CIBSE TM52](#) results an extra **CIBSE TM52** tab is displayed. Clicking on this tab reveals the summary TM52 results grid as shown below.

Daily

Monthly

RunPeriod

CIBSE TM52

TM52 Example Output

CIBSE TM52

Export CSV

Block	Zone	Criterion 1 (%)	Criterion 2 (K.hr)	Criterion 3 (hr)	Pass/Fail
Floor1	Comidor	0.00	0.00	0.00	Pass
Floor1	EOffice	0.00	0.00	0.00	Pass
Floor1	NWOffice	0.20	2.00	0.00	Pass
Floor1	Reception	0.32	4.00	0.00	Pass
Floor1	SWOffice	0.32	3.50	0.00	Pass
Floor1	WC	0.00	1.00	0.00	Pass
Floor2	Comidor	0.28	3.50	0.00	Pass
Floor2	EOffice	0.44	4.50	0.00	Pass
Floor2	NWOffice	0.71	4.50	0.00	Pass
Floor2	Reception	0.83	7.00	0.00	Pass
Floor2	SWOffice	0.87	7.50	0.00	Pass
Floor2	WC	0.32	3.00	0.00	Pass
GroundFloor	Comidor	0.00	0.00	0.00	Pass
GroundFloor	EOffice	0.00	0.00	0.00	Pass
GroundFloor	NWOffice	0.16	1.50	0.00	Pass
GroundFloor	Reception	0.08	1.00	0.00	Pass
GroundFloor	SWOffice	0.24	3.00	0.00	Pass
GroundFloor	WC	0.00	0.50	0.00	Pass

The table shows pass/fail results for each zone and for each of the 3 criteria. Each zone must pass 2 or more of the criteria to pass overall.

CIBSE TM59 Results

When loading an eso file that includes [CIBSE TM59](#) results an extra **CIBSE TM59** tab is displayed. Clicking on this tab shows the summary TM59 results grid as shown below.

Daily	Monthly	RunPeriod	CIBSE TM59	8% glazing opening
-------	---------	-----------	------------	--------------------

CIBSE TM59		Export CSV
------------	--	------------

Naturally ventilated	Coridors
----------------------	----------

Criteria for predominantly naturally ventilated homes

Block	Zone	Criterion A (%)	Criterion B (hr)	Pass/Fail
TypicalFlat	Bedroom1	2.40	8.00	Pass
TypicalFlat	Bedroom2	2.86	4.00	Pass
TypicalFlat	LivingRoomXKitchen	3.08	N/A	Fail

Daily	Monthly	RunPeriod	CIBSE TM59	8% glazing opening
-------	---------	-----------	------------	--------------------

CIBSE TM59		Export CSV
------------	--	------------

Naturally ventilated	Coridors
----------------------	----------

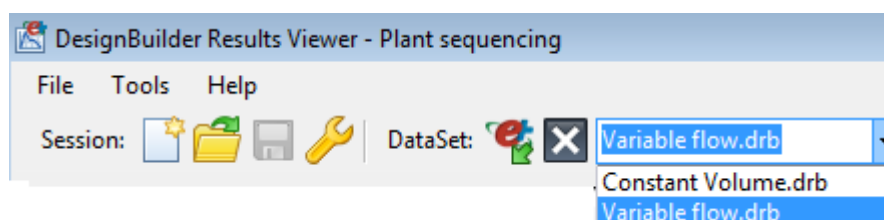
Criteria for corridors

	Block	Zone	% Hours Exceeded	Pass/Fail
▶	TypicalFlat	CommunalCorridor	0.07	Pass
	TypicalFlat	Comidor	0.27	Pass

For more information on how to interpret the results see the [CIBSE TM59 Outputs](#) section.

Working With Multiple Data Sets

You can load as many datasets as required to a single Results Viewer session by using the **Open Datasets** menu or toolbar option. A list is maintained of all data sets currently opened in the **Datasets toolbar drop list** at the top of the window.



When you have more than one data set open it is usually helps to [Include the dataset name in the legend](#) to make it easier to differentiate between them. This option is set by default and can be controlled from the Options dialog.

Note: eso files contain 1 or more "environment", each environment consisting of a set of results for a separate run. For example, it is common for eso files to include results for 2 or sizing runs as well as a set of results for a main simulation run. In this case each environment is loaded to a separate dataset. The current dataset can be selected from the **Datasets toolbar drop list**.

Zooming

In some cases you may find that too much data is displayed on the X-axis at one time and you need to focus on a section (time period) of the results graph. You can use the mouse to do this simply by dragging a time region of interest to zoom in on data for particular days.

To return back to the original "un-zoomed" state, use the [Undo zoom](#) toolbar option.

Create Composite Plot

You can create your own "composite" report by summing or averaging 2 or more existing reports. To create a composite plot first select 2 or more reports of interest by left-clicking with the mouse. Hold the <Ctrl> key down when clicking to add new plots to the selection set. Once you have selected the source reports, click on the **Create composite plot** toolbar icon to open the **Generate Composite Plot** dialog.

Note: The **Create composite plot** toolbar icon is only enabled when: a) 1 report is selected in the left-hand list, or b) 2 or more plots *with the same units* are selected.

Tip: Although it is not possible to rename reports in Results Viewer, you can create a new plot based on an existing one and apply a user-defined name to the copy. This is done by selecting a *single report* in the list, clicking on the **Create composite plot** toolbar icon and entering an appropriate name for the new plot. The result is a copy of the original report with the user-defined name.

Generate Composite Plot

Name

Total Fan Energy

Area

HVAC

Calculation Type

☐ Average ☒ Sum

Options

☒ Create composite plot for all intervals

☒ Create composite plot for all datasets

OK Cancel

Name

Enter the name of the plot to be created. This name will appear in the [Report Type](#) column of the Reports grid.

Area

Enter the "Area" of the plot to be created. The Area represents the area of the building or environment covered by the new plot. This text will appear in the [Area](#) column of the Reports grid. For example if the new plot is the sum of a set of electricity consumption data for all zones then the Area might be entered as "Building".

Calculation type

Data for the selected plots can be combined in one of two ways depending on the selection made in the Calculation type area of the dialog:

Average - in this case the values are averaged. Note that Results Viewer is not able to perform area or volume-weighted averaging as it does not data on the floor area or volume for the area of each of the source plots. If you need to create area or volume-weighted averages or apply some other more sophisticated method then new reports of arbitrary complexity that can be created using [EMS scripts](#).

Sum - in this case the values are summed.

Create composite plot for all intervals

Check this option if you wish your composite plot to be applied to all intervals in the current dataset. For example, if you are working on the **Hourly** tab and create a new composite plot with this option set, the same plot will also be added to the list for any other interval data you may have on the **Daily**, **Monthly**, **Run period**, etc. tabs.

This check box will be greyed out if you only have one interval in the dataset.

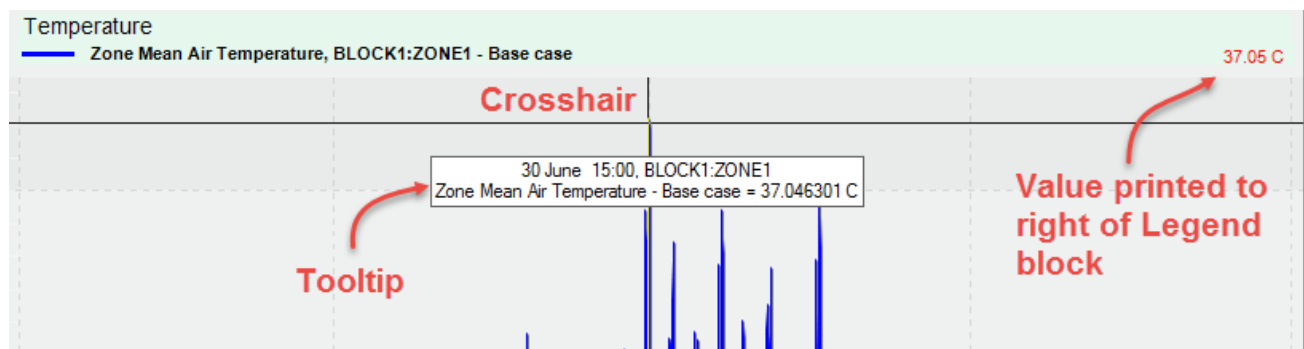
Create composite plot for all datasets

Check this option if you wish your composite plot to be applied to all datasets currently loaded.

Tip: If both the **Create composite plot for all intervals** and the **Create composite plot for all datasets** check boxes are checked, then new plots will be added for all intervals and datasets.

Crosshair And Tooltips

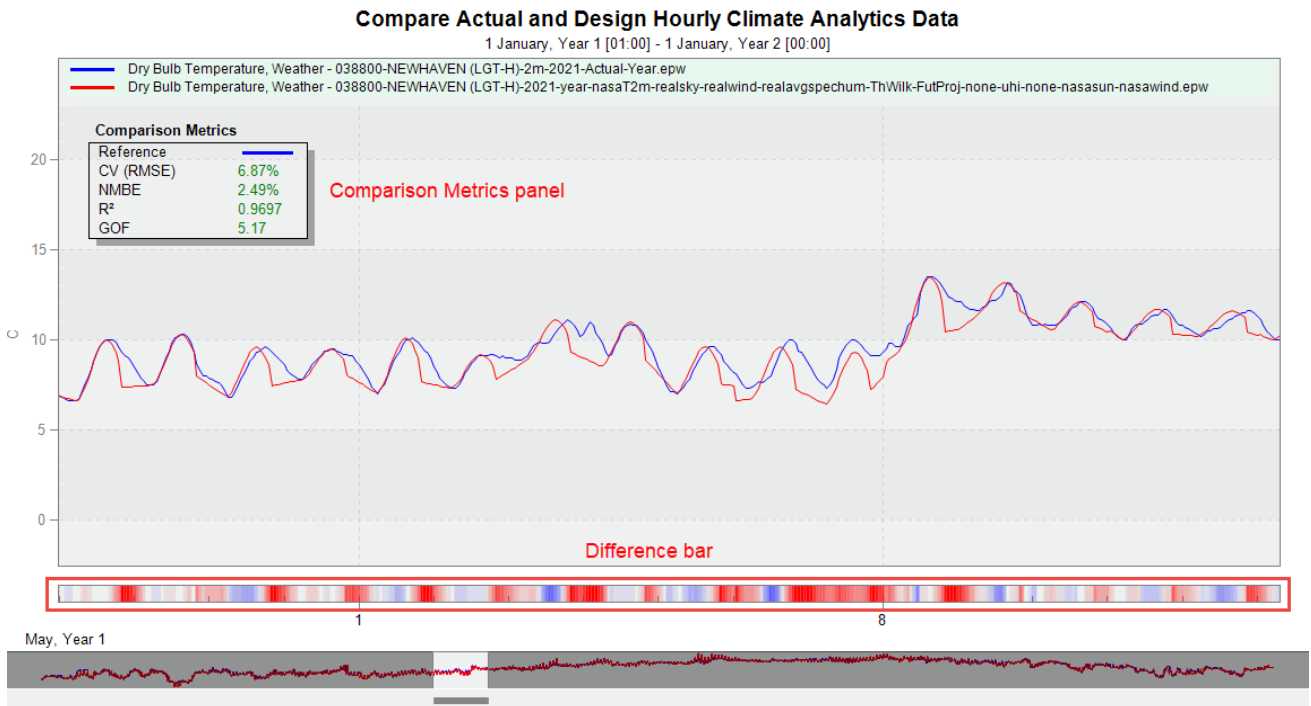
When you move the mouse over a point on a plot, a floating tooltip window appears with data for that point only. Left clicking with the mouse causes the value at that point to be plotted on the right hand side of the graph legend block and a black crosshair is displayed with a vertical and horizontal line centred on the point of interest. The crosshairs can be useful for checking simultaneous values for a range of reports. The value for all other reports currently plotted are also displayed for the current time indicated by the crosshair.



Tip: When the crosshair is displayed you can use the <Left> and <Right> arrow keys to scroll through the data.

Data Comparison

When exactly 2 datasets are displayed on a graph and there is only 1 graph, you can click on the **Compare datasets** toolbar icon  to display the **Comparison Metrics panel** and the **Difference bar**. This provides some summary statistical data indicating how well the sample or modelled data fits with the reference or measured data. The screenshot below gives an example where 2 hourly weather datasets for the same location and year are compared. The blue line is for the most accurate "Actual" data based on hourly recordings and is used as the reference dataset and the red line is the sample "Design" dataset.



Comparison Metrics panel

The Comparison Metrics panel includes the summary statistical data listed below.

Reference - shows the colour of the plot that represents the reference data (typically measured or most accurate). In the above screenshot the blue line represents the reference data.

CV (RMSE) - the Coefficient of the Variation of the Root Mean Square Error, a metric that is used to calibrate models to represent measured building performance. It indicates how well the model is able to predict the target values (accuracy). Lower values indicate better fit and a value of 0 indicates perfect fit. ASHRAE Guideline 14 calibration criteria requires CV(RMSE) to be less than 15% for monthly data and less than 30% for hourly data. These are the [default acceptance thresholds](#) set on the Options dialog.

NMBE - Normalised Mean Bias Error indicates whether the data globally over or under-predicts relative to the reference values. Positive values indicate over-prediction and negative values indicate under-prediction and a value of 0 indicates perfect fit. It normalises the Mean Bias Error by dividing it by the mean of the reference values. ASHRAE Guideline 14 calibration criteria requires NMBE to be less than +/-5 for monthly data and less than +/- 10 for hourly data. These are the [default acceptance thresholds](#) set on the Options dialog.

R² - the coefficient of determination is a standard statistical measure used to indicating how accurate a linear regression model is at predicting the reference data, i.e. how close the model data is to the fitted regression line. It is given on 0 - 1 scale, with a value of 1 indicating a perfect fit.

GOF - a standard statistical measure that determines how well the model data fits a distribution from a population with a normal distribution.

Tip: The Comparison Metrics panel is initially displayed in the top left of the screen, but it can be dragged to a more suitable location as required.

Difference bar

When the **Data comparison** function is active, a Difference bar is displayed above the horizontal scrollbar at the bottom of the screen (see screenshot above). For each time interval on the graph a coloured band is displayed to indicate how close the model data is to the corresponding measured value. A red band indicates that the model data is lower than the reference and blue that it is higher. The shade of red/blue indicates the extent of the differences with darker colours showing greater differences. In large datasets (e.g. a year of hourly values) this display can help you to quickly find data that deviates the most from the reference values.

Styling

When you create graphs with Results Viewer, they are styled (e.g. Title Font, Background colour, etc) using a default styling template. You can change the styling defaults to your own preferences by using the right-hand context menu on the graph pane. The following options are currently available:

- Border Style
- Font Size
- Grid Options
- Customise Dialog (more detailed Font and Colour changes)

If you make some changes and want to revert back to the default styling at any time, select the **Tools > Restore Graph Styling** menu option.

Any styling changes made to the currently open session will be made permanent once the session has been saved.

Results Viewer Menu And Toolbar Commands

You can access a range of options from the top menu bar, toolbar and right-click context menus. Note that some commands are only available from the menu bar, others are only available from the toolbar and still others are only available from the context menu.

Top Menu Bar

The Top menu bar has 3 top level entries:

- [File](#)
- [Tools](#)
- [Help](#)

The following menu bar commands are available:

File

- New session
- Open session
- Save session
- Save session As
- Open datasets
- Close dataset
- Close all datasets
- Save dataset as drb
- RemoveHtml file
- Exit

Tools

- Restore Default Graph Styling
- Options

Help

Help
About

Toolbar

New session

Creates a new session first closing any currently open session.

Open session

Opens a previously saved session file.

Save session

Save the current session to file.

Options

Opens the [Options dialog](#).

Open datasets

Use this option to load one or more new datasets into the current Results Viewer session. When you click on the icon and open Results Viewer, a file dialog appears. From here you can select files of the following types:

- [.eso](#) - EnergyPlus results files in text format
- [.sql](#) - EnergyPlus results files in SQLite format
- [.htm](#) or [.html](#) - EnergyPlus Summary results files
- [.epw](#) - EnergyPlus hourly weather files
- [.csv](#) - Comma-separated value grid of data
- [.drb](#) - Compiled Results Viewer data set
- [.json](#) - JSON format grid of data

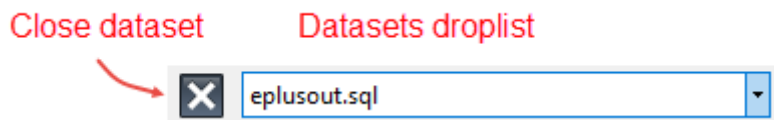
For more information on these file formats, see "Results Viewer Supported Dataset File Formats" on page 2241

Close dataset

Use this option to close the currently selected data set as indicated by the text in the **Selected datasets** drop list immediately to the right. See screenshot below under Selected data set drop list.

Selected dataset drop list

If you have more than one dataset loaded, you can use the **Selected datasets** drop list (aka Dataset drop list) to select the current dataset.



When you use this control to select a new dataset, the available plots for the dataset are loaded into the grid on the left of the dialog.

Add selected variables to current graph

Selecting this option causes the variables currently selected in the grid to be plotted on the current graph. The current graph can be identified by the different blue colour (see [Selecting a graph](#)). Single variables can also be added to the current graph by double clicking on the variable in the grid.

Add selected variables to new graph

The same as [Add selected variable\(s\) to current graph](#) above but creates a new graph and plots the variable there.

Delete selected variables from current graph

Click this option to remove the selected variable(s) from the current graph. Single variables can also be removed from the current graph by double clicking on the variable in the grid.

Add selected variables to current graph for all datasets

The same as [Add selected variable\(s\) to current graph](#) above but plots equivalent variables for all datasets on the current graph. This option is only available when more than one dataset is loaded.

Add selected variables to new graph for all datasets

The same as [Add selected variable\(s\) to current graph for all datasets](#) above but creates a new graph and plots the variable there.

Remove selected variables from current graph for all datasets

Click this option to remove the selected variable(s) from the current graph for all datasets.

Create composite plot

Sometimes it can be useful to create a new plot by combining data from other plots. The [Create Composite Plot](#) dialog allows you to do this for selected plots.

Export selected variables

This tool provides a quick way to export a set of results to a CSV file by directly selecting them from the grid without needing to first display them in a graph or grid.

Note: This gets around the 100 variable limit that applies to plotting data on the graphs and grid.

Display graph

Display the data as a graph instead of a grid.

Display grid

Display the data as a grid instead of a graph.

Save grid to CSV File

Allows you to save the data as a comma separated values (CSV) file for loading into a spreadsheet for further analysis.

Add new graph

Adds a new empty graph.

Delete Selected Graph

Removes the currently selected graph along with any variables that were plotted on it. Note that the underlying data is not deleted, just the graph.

Rename Selected Graph

Allows you to change the title for the currently selected graph. To change the name of the current graph right-click on the graph and from the **DesignBuilder Options**, select the **Rename graph title** option. Enter the title for the graph in the dialog and press OK.

Copy graphs to another frequency

If you have generated similar data for multiple frequencies then use this tool to use settings for the current frequency and display the same reports using a different frequency.

Change main graph title

Change the text to be used for the main title for the graph.

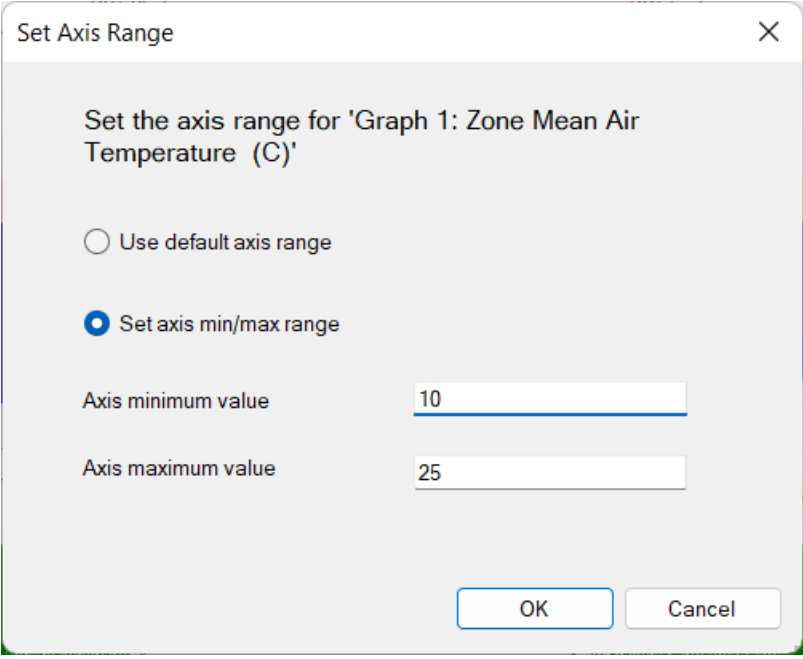
Undo zoom

Undo any [zoom settings](#) previously made.

Change axis range

You can change the Y-axis range for the currently selected report from the **Change Axis Range** dialog which is accessed by clicking on this toolbar icon. By default the **Use default axis range** option is checked, in which case Results Viewer auto-scales the Y-axis based on the maximum and minimum values in all datasets that use the units of the selected dataset.

However the Y-axis range can be changed by clicking on the **Set axis min/max range** option and entering appropriate values for the **Axis minimum value** and **Axis maximum value**.

A screenshot of the 'Set Axis Range' dialog box. The title bar says 'Set Axis Range' with a close button. The main text says 'Set the axis range for 'Graph 1: Zone Mean Air Temperature (C)'. There are two radio buttons: 'Use default axis range' (unselected) and 'Set axis min/max range' (selected). Below the radio buttons are two input fields: 'Axis minimum value' with the value '10' and 'Axis maximum value' with the value '25'. At the bottom are 'OK' and 'Cancel' buttons.

Compare datasets

When exactly 2 datasets are displayed on a graph and there is only 1 graph, you can click on this toolbar icon to display:

the [Comparison Metrics panel](#) with summary statistical data indicating how well the modelled data fits with the reference or measured data, and, the [Difference bar](#).

See [Data Comparison and Calibration](#) for more information.

Change measured plot

When 2 plots are being compared using the [Compare datasets](#) tool, one plot is considered to represent measured or reference data while the other is considered to represent the modelled data. Pressing this toolbar icon toggles the selection so what is currently the modelled data becomes the measured data and what is currently the measured data becomes the modelled data.

Context Menu

Results Viewer Options Dialog

The Options dialog can be accessed either from the  toolbar icon or from the [Tools](#) top menu option.

General Tab

Autosave session

Select this option if you would like the session to be saved automatically when closing the Results Viewer.

Check for save on exit

If you would like Results Viewer to offer to save a [drs session file](#) when exiting then check this option.

Default html text zoom percentage

The zoom level of the html Summary data can be controlled with this setting.

Language

Set the language to be used in the Results Viewer user interface. Options are:

- English - default
- Italian
- Spanish
- French
- Portuguese - not maintained so will be the same as English.
- German

You must close Results Viewer down and re-open it to see any changes made to the language option.

Note: This setting only controls the language used for dialog labels, menus and options. It does not affect the language used for the names of the [Reports](#) or the [Areas](#) displayed on the [Reports grid](#) and graphs.

Schedules

When a report is only generated for certain periods in the simulation, e.g. when the simulation [Reporting period](#) option is set to **3-User-defined schedule**, the name of the schedule used to control the times that data for each report is generated can optionally be displayed in the **Schedule** column in the [Reports grid](#). This setting controls the display behaviour of the Schedule column. The available options are:

- Show** - the **Schedule** column is displayed.
- Hide** - the **Schedule** column is not displayed.
- Auto** - the **Schedule** column is displayed only if one or more reports in the current dataset use a schedule to control the output of the data. This is the default option.

Units Tab

The settings on the Units tab allow you to control the way that data is displayed on the screen, in particular whether SI or IP units are to be used and the units to be used energy and power data. The screenshot below illustrates the various settings that are available on the Units tab.

	Power	Energy
Timestep	☒	☐
Hourly	☒	☐
Daily	☐	☒
Monthly	☐	☒
Annual / Runperiod	☐	☒

☐ Display original dataset units in grid

Units

Use the **Units** drop list to define whether results are to be displayed in SI or IP units.

Energy units

EnergyPlus natively provides energy outputs in J, however this is rarely a convenient unit to work with when analysing building performance results. The **Energy units** drop list allows you to define the units that Results Viewer will use to display all such Energy reports. The available options are:

- kWh (default)
- MWh
- Wh
- J
- kJ
- MJ
- GJ
- Btu
- kBtu
- MBtu

Power units

EnergyPlus natively provides power outputs in W, however this is not always a convenient unit to work with when analysing building performance results. The **Power units** drop list allows you to define the units that Results Viewer will use to display all such Power reports. The available options are:

- W (default)
- kW
- MW
- Btu/h
- kBtu/h
- MBtu/h

Power or Energy for each interval

You can define whether energy and power data should be displayed as either power or energy for each interval by checking radio buttons in the grid as appropriate.

	Power	Energy
Timestep	☒	☐
Hourly	☒	☐
Daily	☐	☒
Monthly	☐	☒
Annual / Runperiod	☐	☒

For example if the eso file generated by the simulation includes hourly energy reports which use energy (J) as the units, it will usually be much more convenient to view such results as power (e.g. W or Btu/h). In which case you should check the Hourly / Power radio button in the grid (it is checked by default).

On the other hand if the eso file you have generated includes monthly energy reports which use power (W) as the units, it will usually be much more convenient to view such results as energy (e.g. MJ, kWh or Btu). In this case you should check the Monthly / Energy radio button in the grid (it is checked by default).

By default all energy and power reports are displayed as power at hourly and timestep intervals and as energy at longer intervals.

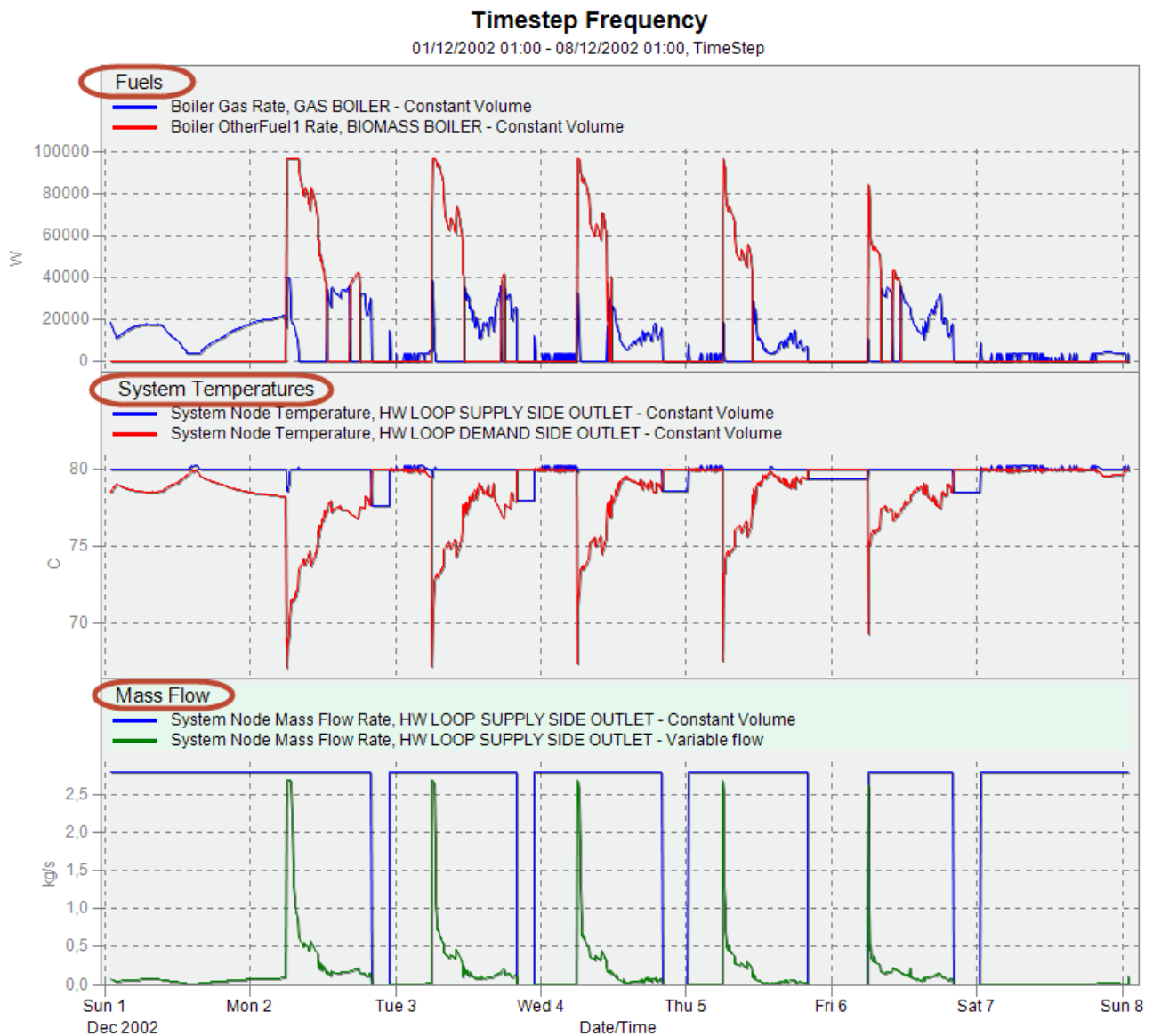
Display original dataset units in grid

Check this option if you would like to see the units originally used in the EnergyPlus results in the [Reports grid](#). In this case an additional **Source Units** column is displayed alongside the standard **Units** column. The source units are most likely to be different from the standard display units for energy and power reports and for most reports when displaying results in [IP units](#).

Graph Tab

Display a title for each graph

Selecting this option causes the title of each graph to be displayed for each graph as shown in highlighted areas in the graph below.



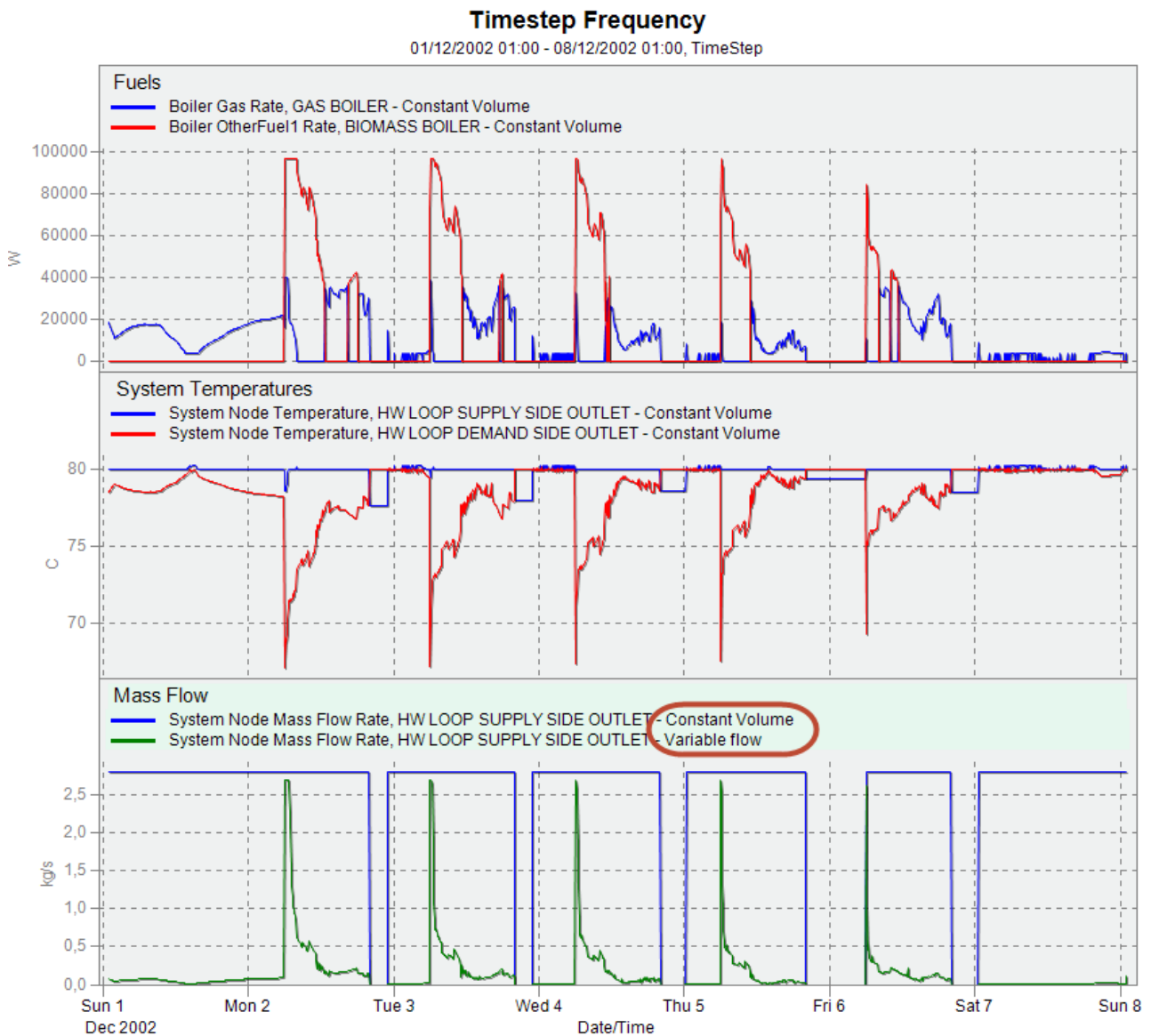
To change the name of the current graph right-click on the graph and from the **DesignBuilder Options**, select the **Rename graph title** option. Enter the title for the graph in the dialog and press OK.

Display Day of week labels

When this option is checked Results Viewer displays the day of week (Monday, Tuesday ... etc) as well as the day of the month.

Include dataset name in legend

If you have more than one data set loaded then you should usually select this option to ensure that the data set name is included in the legend. This can help when comparing results for different simulations.



The output above shows how the dataset name is added to each legend.

Show interpolated values if crosshair is on

When the [Crosshair](#) function is enabled you can choose whether to display interpolated values or not.

Mark data points

Check this setting if you would like a dot to be displayed to represent each data value. The option provides a further level of control by allowing a threshold to be set. When there are more time points on the graph that this value the dots are suppressed.

☒ Mark data points

☒ < data points

Fade non-selected plots

When this option is selected any plots (lines) that are not selected are displayed with a degree of transparency which can be set in the control below as a Percent fade value.

☒ Fade non-selected plots

83 Percent fade

To "unfade" the non-selected plots click on the graph area to unselect all plots.

Include folder name in dataset name

If you include the dataset name in the legend then do you want the folder name included too? If so check this option. This option is only usually used when the result sets are stored in files with the same name but in different folders.

Grid Tab

Copy grid headers with CTRL+C

When copying grid data from the data grid or the report grid using the CTRL+C key press, checking this check box ensures that the grid headers are included when you paste the data into a spreadsheet or other application.

Dataset Comparison Tab

On the Dataset Comparison tab you can set the thresholds used to define whether a [calibration](#) result is considered to be within the acceptable range or not when displayed on the [Comparison Metrics panel](#). Values that are within range are displayed in green and out of range values in red.

The default values for **CV (RMSE)** and **NMBE** are those recommended by ASHRAE.

The screenshot below illustrates the options that are available.

The screenshot shows the 'Dataset Comparison' tab with a 'Metric Thresholds' section. It contains input fields for CV (RMSE), NMBE, R², and GOF, with separate columns for Hourly and Monthly thresholds. R² and GOF only have a single input field.

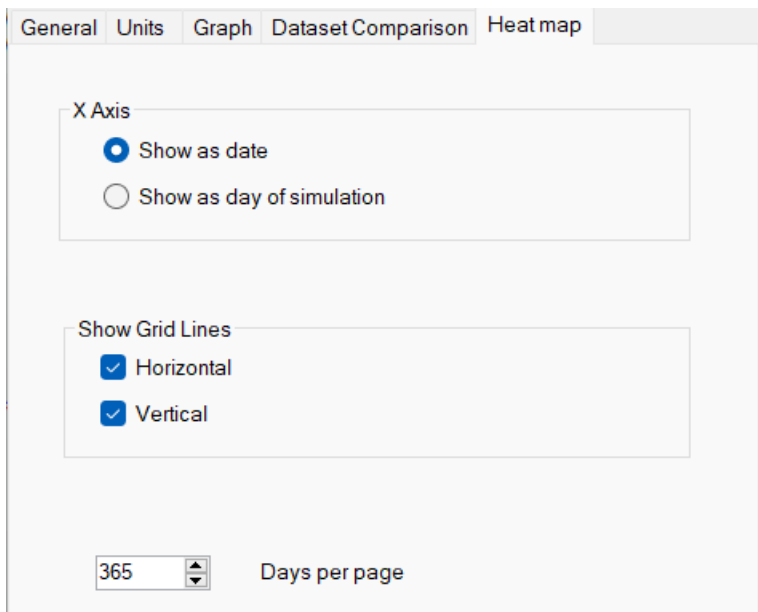
	Hourly	Monthly
CV (RMSE)	30	15
NMBE	10	5
R²	0.75	
GOF	10	

Heat Map Tab

The settings the Heat map tab allow you to configure the appearance of the display on the Hourly heat map tab. The following settings are available:

- Define whether X-axis labels are to be displayed as dates or "day of simulation".
- Display/hide horizontal and vertical grid lines.
- Set the number of days to be displayed at a time on a page. By default 365 days are displayed to allow data for a full year to be seen at once.

The screenshot below illustrates the various options that are available.



Results Viewer Supported Dataset File Formats

You can load dataset files of the following extension and formats to Results Viewer:

- [.eso](#) EnergyPlus simulation results file in text format
- [.sql](#) EnergyPlus simulation results file in SQLite format
- [.htm/.html](#) EnergyPlus HTML summary results file
- [.epw](#) EnergyPlus format hourly weather file
- [.csv](#) spreadsheet file
- [.drb](#) Results Viewer binary dataset file from a previous session
- [\(.drs\)](#) Results Viewer binary session file saved from a previous session. Not a dataset format, but included here for completeness.)

The processes involved with working with these file formats are described below.

Tip: Each You can [load multiple datasets](#) to Results Viewer to allow data comparison. This applies to [.eso](#), [.epw](#), [.csv](#) and [.drb](#) dataset files.

.eso EnergyPlus Simulation Results File

[.eso](#) is the main EnergyPlus detailed simulation results file format supported by DesignBuilder. These files can be loaded into Results Viewer using its [Open Datasets](#) toolbar command. They can also be loaded directly into Results Viewer by double-clicking on an [.eso](#) file in Windows File Explorer if the [.eso](#) extension has previously been associated with Results Viewer.

[.eso](#) files from the previous simulation can typically be found in DesignBuilder's EnergyPlus folder.

When you load simulation results from the DesignBuilder Simulation screen using either the [View EnergyPlus Results](#) toolbar / menu command or from the Simulation Manager using the [View results in DesignBuilder Results Viewer](#) toolbar command then an [.eso](#) file from the simulation will be loaded along with the corresponding html summary results file.

.sql EnergyPlus Simulation Results File

As well as the main .eso output format, EnergyPlus can also write detailed simulation results and summary results into a SQLite database file with extension .sql. These files can be loaded into Results Viewer using its **Open Datasets** toolbar command. They can also be loaded directly into Results Viewer by double-clicking on a .sql file in Windows File Explorer if the .sql extension has previously been associated with Results Viewer.

.sql files from the previous simulation can typically be found in DesignBuilder's EnergyPlus folder.

.htm/.html EnergyPlus HTML Summary Results File

EnergyPlus generates a summary results file in html format. These can be attached to the current dataset from the **Open Datasets** toolbar / menu command.

When you load simulation results from the DesignBuilder Simulation screen using either the [View EnergyPlus Results](#) toolbar / menu command or from the Simulation Manager using the [View results in DesignBuilder Results Viewer](#) toolbar command then an htm summary results file from the simulation will be loaded along with the corresponding eso detailed results file.

Note: only one htm file can be added per dataset.

.epw EnergyPlus Format Hourly Weather File

[EnergyPlus format hourly weather files](#) can be loaded as new datasets to Results Viewer using the **Open Datasets** toolbar / menu command. They can also be loaded directly into Results Viewer by double-clicking on an epw file in Windows File Explorer if the .epw extension has previously been associated with Results Viewer.

.csv Spreadsheet File

Results Viewer can load csv files of specific formats as new datasets. The csv formats that can be loaded are described below. Note that if you would like to load a csv file with different format to those described below it will usually be possible to modify the csv file to conform with one of the supported formats.

Results Viewer Export Format

This is the format exported using the [Save grid to CSV file](#) toolbar / menu command.

These files consist of a dataset header listing the datasets involved followed by a blank line, followed by 2 rows with a column per selected report which define the data in the rows below. Sections A, B and C in the screenshots below illustrate the overall layout.

In a spreadsheet:

	A	B	C	D	E	F	G	H	I	J	K	L
A	1 038800-NEWHAVEN (LGT-H)-2m-2021-Actual-Year.epw											
3	2 038800-NEWHAVEN (LGT-H)-2021-year-nasaT2m-realsky-realwind-realavgspchum-ThWilk-FutProj-none-uhf-none-nasasun-nasawind.epw											
B	Date/Time	[1] Global Horizontal	[1] Wind Speed (m/s)	[1] Relative Humidity	[1] Direct Normal Ra	[1] Dry Bulb Temper	[2] Global Horizontal	[2] Wind Speed (m/s)	[2] Relative Humidity	[2] Direct Normal Ra	[2] Dry Bulb Temperature (C)	
5	Weather	Weather	Weather	Weather	Weather	Weather	Weather	Weather	Weather	Weather	Weather	
C	Year 1 Jan 01 01:00	0	5.4	75	0	4.1	0	4	92	0	2.9	
7	Year 1 Jan 01 02:00	0	5.5	74	0	4	0	4	92	0	2.99	
8	Year 1 Jan 01 03:00	0	5.5	73	0	3.9	0	4	91	0	3.07	
9	Year 1 Jan 01 04:00	0	5.5	73	0	3.8	0	4	91	0	3.16	
10	Year 1 Jan 01 05:00	0	5.4	72	0	3.8	0	4	90	0	3.25	
11	Year 1 Jan 01 06:00	0	5.3	72	0	3.7	0	4	90	0	3.34	
12	Year 1 Jan 01 07:00	0	5.3	71	0	3.7	0	4	89	0	3.42	
13	Year 1 Jan 01 08:00	0	5	71	0	3.7	0	4	88	0	3.51	
14	Year 1 Jan 01 09:00	27	4.7	71	0	3.6	38	4	87	0	3.73	
15	Year 1 Jan 01 10:00	108	4.3	69	353	3.7	118	4	85	471	4.08	

The same data in a text editor:


```

A 1,038800-NEWHAVEN (LGT-H)-2m-2021-Actual-Year.epw,,,,,,,,,
2,038800-NEWHAVEN (LGT-H)-2021-year-nasaT2m-realsky-realwind-realavgspechum-ThWilk-FutProj-none-uh1-none-nasasun-nasawind.epw,,,,,,,,
B Date/Time,[1] Global Horizontal Radiation (Wh/m2),[1] Wind Speed (m/s),[1] Relative Humidity ([1] Direct Normal Radiation (Wh/m2),[1] Dry Bulb Temperature (C),[2] Global Horizontal Radia
Weather.Weather.Weather.Weather.Weather.Weather.Weather.Weather.Weather.Weather
C Year 1 Jan 01 01:00,0,5,4,75,0,4,1,0,4,92,0,2,99
Year 1 Jan 01 02:00,0,5,5,74,0,4,0,4,92,0,2,99
Year 1 Jan 01 03:00,0,5,5,73,0,3,9,0,4,91,0,3,07
Year 1 Jan 01 04:00,0,5,5,73,0,3,8,0,4,91,0,3,16
Year 1 Jan 01 05:00,0,5,4,72,0,3,8,0,4,90,0,3,25
Year 1 Jan 01 06:00,0,5,3,72,0,3,7,0,4,90,0,3,34
Year 1 Jan 01 07:00,0,5,3,71,0,3,7,0,4,89,0,3,42
Year 1 Jan 01 08:00,0,5,71,0,3,7,0,4,89,0,3,51
Year 1 Jan 01 09:00,27,4,7,71,0,3,6,38,4,87,0,3,73
Year 1 Jan 01 10:00,108,4,3,68,353,3,7,118,4,85,471,4,08

```

The format is described in more detail below.

Dataset headers (Section A)

Multiple rows identifying the original datasets relating to the data columns as loaded into the session from which the export file was created. Each dataset label is prefixed by an index number.

Empty line

This is represented by a CR/LF only.

Column headers for exported data (2 rows)

i) Row 1: column identifiers

First column is always Date/Time. This format is custom and specific to Results Viewer.

Other headers are prefixed with dataset identifier in square brackets, and units in suffix in parentheses.

ii) Second row

Area of column data. Row starts with delimiter since Date/Time has no area.

Data rows

Each row starts with the Date/time, followed by delimited values for each variable.

Note: The original dataset names are discarded on import, since they have no meaning. Instead, all columns will be assigned to a new dataset with the name of the CSV file.

DesignBuilder Grid Results Export Format

This is the format exported when viewing results on the main DesignBuilder screen using the [Grid format](#). It is accessed by setting the **Show as** Display option to [2-Grid](#).

These files consist of a header with 2 rows which define the data in the rows below. Subsequent rows contain the data itself. Sections A and B in the screenshots below illustrate the overall layout.

Note: In this format, all values are enclosed in quotes.

In a spreadsheet:

	A	B	C	D	E	F
A	Date/Time	General Lighting	Computer + Equip	Occupancy	Solar Gains Exterior Windows	Total Latent Load
		kW	kW	kW	kW	kW
B	15/06/2000 00:30	0	0.258922	0	0	0
4	15/06/2000 01:00	0	0.258922	0	0	0
5	15/06/2000 01:30	0	0.258922	0	0	0
6	15/06/2000 02:00	0	0.258922	0	0	0
7	15/06/2000 02:30	0	0.258922	0	0	0
8	15/06/2000 03:00	0	0.258922	0	0	0
9	15/06/2000 03:30	0	0.258922	0	0	0
10	15/06/2000 04:00	0	0.258922	0	1.19E-03	0
11	15/06/2000 04:30	0	0.258922	0	3.989201	0
12	15/06/2000 05:00	0	0.258922	0	9.970937	0
13	15/06/2000 05:30	0	0.258922	0	13.35228	0
14	15/06/2000 06:00	0	0.258922	0	15.06431	0
15	15/06/2000 06:30	6.117492	4.800192	0.1272746	15.89688	1.125557

The same data in a text editor:

```
A "Date/Time","General Lighting","Computer + Equip","Occupancy","Solar Gains Exterior Windows","Total Latent Load",
  ,"kW","kW","kW","kW","kW",
B "15/06/2000 00:30:00","0","0.258922","0","0","0",
  "15/06/2000 01:00:00","0","0.258922","0","0","0",
  "15/06/2000 01:30:00","0","0.258922","0","0","0",
  "15/06/2000 02:00:00","0","0.258922","0","0","0",
  "15/06/2000 02:30:00","0","0.258922","0","0","0",
  "15/06/2000 03:00:00","0","0.258922","0","0","0",
  "15/06/2000 03:30:00","0","0.258922","0","0","0",
  "15/06/2000 04:00:00","0","0.258922","0","1.186233E-03","0",
  "15/06/2000 04:30:00","0","0.258922","0","3.989201","0",
  "15/06/2000 05:00:00","0","0.258922","0","9.970937","0",
  "15/06/2000 05:30:00","0","0.258922","0","13.35228","0",
  "15/06/2000 06:00:00","0","0.258922","0","15.06431","0",
  "15/06/2000 06:30:00","6.117492","4.800192","0.1272746","15.89688","1.125557",
  "15/06/2000 07:00:00","6.117492","4.800192","0.1851248","16.39756","1.067707",
```

Line 1 - Column headers (Section A)

Date/Time first, followed by Data column names.

Note that the Date/Time format is local time.

Line 2 – Units (Section A)

The Date/Time column is left blank and so this line starts with a delimiter.

Line 3 to end of file – Data (Section B)

The first column is always Date/Time. This is followed by the data , one item per column to align with the data column headers.

DesignBuilder Grid Results Export Format - Modified

This format is the same as DesignBuilder Grid Results Export Format above, but prefixed with a header. The format has been developed to read files exported by a tool for reading data generated by building data collection systems that is still under development. The header includes a version number and an interval value to define the meaning of the data. Multiple intervals are not allowed to be mixed within a single CSV file.

Version followed by a numeric value for file format version. (Defines remaining header fields)

Interval which can be one of:

 Timestep,
 Hourly,
 Daily,
 Monthly,
 Annual,
 RunPeriod

Example data:

Version:1.0,Interval:Annual

"Date/Time","Mech Vent + Nat Vent + Infiltration","External Infiltration"
,"ac/h","kWh"
"01/01/2002","1.038702","-12397.66"

.drb Results Viewer Binary File

[.drb files](#) are a binary representation of the data format used internally by Results Viewer. A drb file represents the data in a single dataset. They can be saved and loaded more quickly than eso and csv files and are used internally within drs session files, one drb file per dataset in the session.

These files can be saved by using the **File > Save dataset as file** menu command.

Diagnostics

DesignBuilder creates two types of **diagnostics files** on startup, "diagnostic log" files and "installation diagnostics log files":

Diagnostics_dd_mm_yyyy_hh_mm_ss.log - files contain information about the operating system settings, hardware configuration and settings within the program. It also contains internal program information written out as the program runs which can help DesignBuilder support to track down problems.

Installation Diagnostics_dd_mm_yyyy_hh_mm_ss.log - files contain information on the steps carried out during the final stage of software installation and any errors that may have occurred.

The "dd_mm_yyyy_hh_mm_ss" part of the filename is the date and time the file was originally created. For example the file "Diagnostics_01_08_2010_09_26_53.log" was created at 1 August 2010 at 9:26:53 am.

You may be asked to supply these files to DesignBuilder Support if you have problems with program operation on your computer. One typical request from the Support desk is to supply the diagnostics file from a particular program run and this will usually be the most recent one. To find the most recently created diagnostics file, simply sort the files in Windows Explorer by using the "Details" view and clicking on the "Date modified" column to carry out the sort. Alternatively you can scan the files by eye to find the most recently created file using the "dd_mm_yyyy_hh_mm_ss" part of the filenames.

Finding The Diagnostics Files

The diagnostics files are generated in the [Diagnostic files folder](#) which can be accessed from DesignBuilder the **File > Folders > Diagnostic files folder** menu command. When you select this menu option a Windows Explorer window opens and you will see the diagnostics and installation diagnostic log files.

If for any reason you are unable to start DesignBuilder then the diagnostics files can usually be found the folder:

C:\Users**User**\AppData\Local\DesignBuilder

Where **User** is your username on the computer.

Calculation Of Air Density

The air density is used for calculating the air volumetric flow rate from air mass flows as part of the [loads](#) calculation procedure.

The method used in DesignBuilder is the same as that in EnergyPlus for consistency. Air is assumed to be dry and at 20°C. The site elevation is used to calculate standard atmospheric pressure using the equation for 'standard atmospheric' pressure on p 6.1 of the ASHRAE 1997 HOF.

$$\text{Density} = P / (R * T)$$

Where:

R is the Gas constant = 287.05 J/kg-K

T is the temperature in K. Air is assumed to be at 20°C so, T = (20 + 273.15)

P is standard pressure:

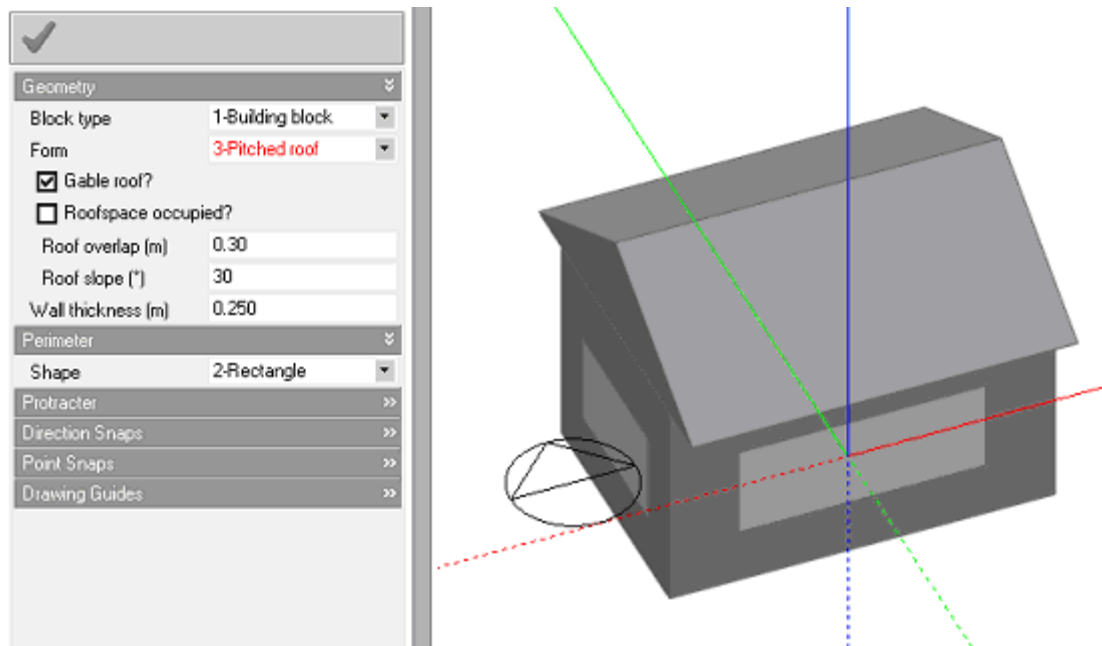
$$P = 101325 \times (1.0 - Z \times 0.0000225577)^{5.2559}$$

where:

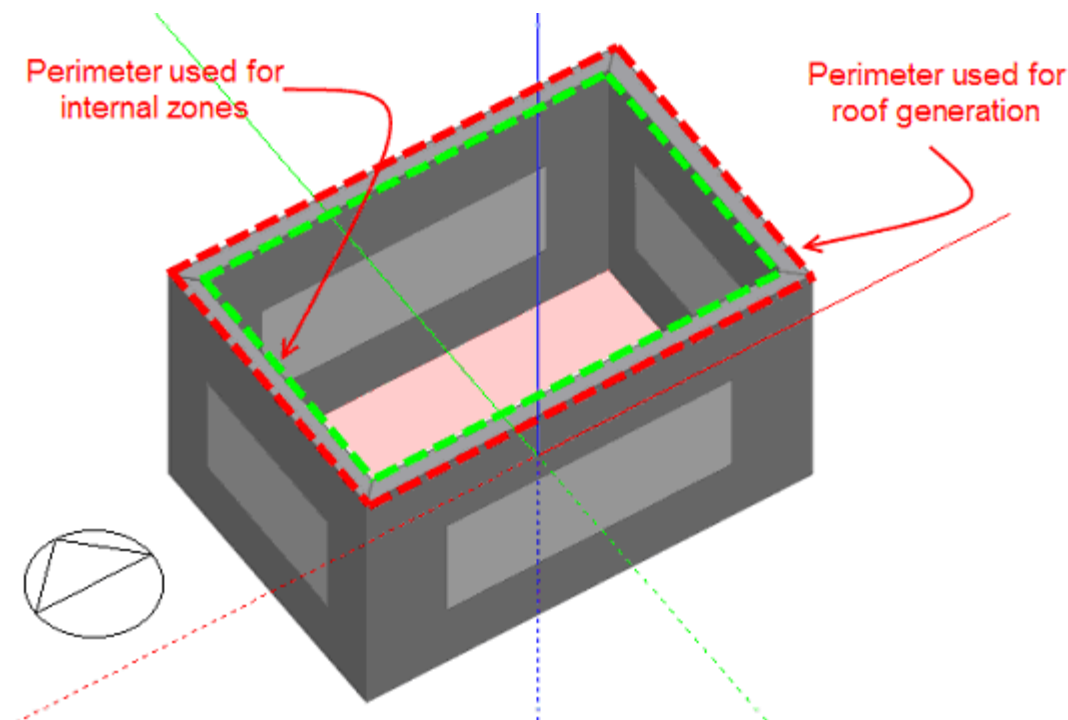
Z = [Elevation](#) above sea level (m)

Pitched Roof Geometry - Technical Description

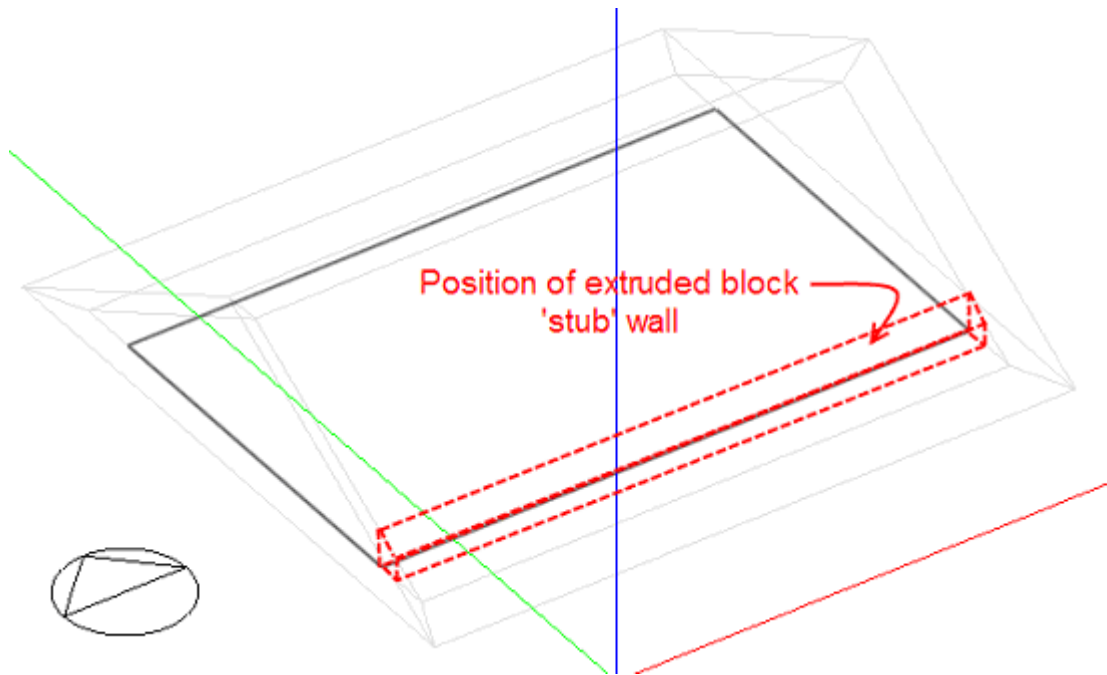
This page describes DesignBuilder pitched roof geometry generation with reference an example roof created by drawing a rectangular perimeter with an offset of 0.3m and an external wall thickness of 0.25m:



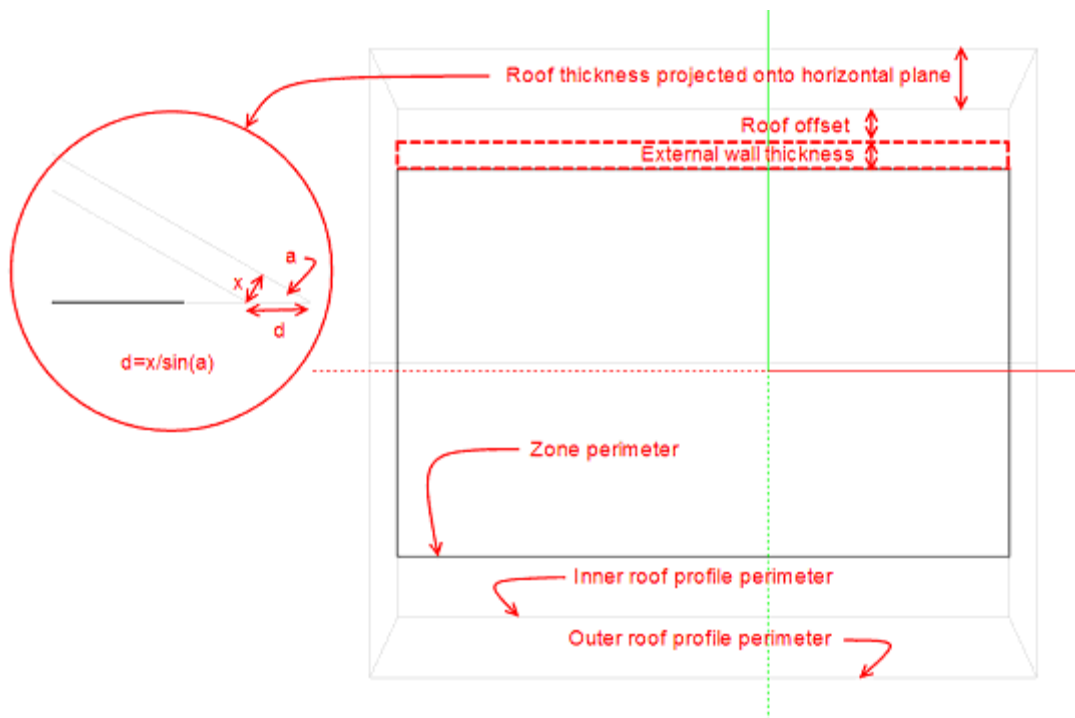
Roof perimeter blocks are created behind the scenes by first generating a conventional plan extrusion using the current external wall thickness and then generating the roof inner profile topology from the resulting block outer perimeter:

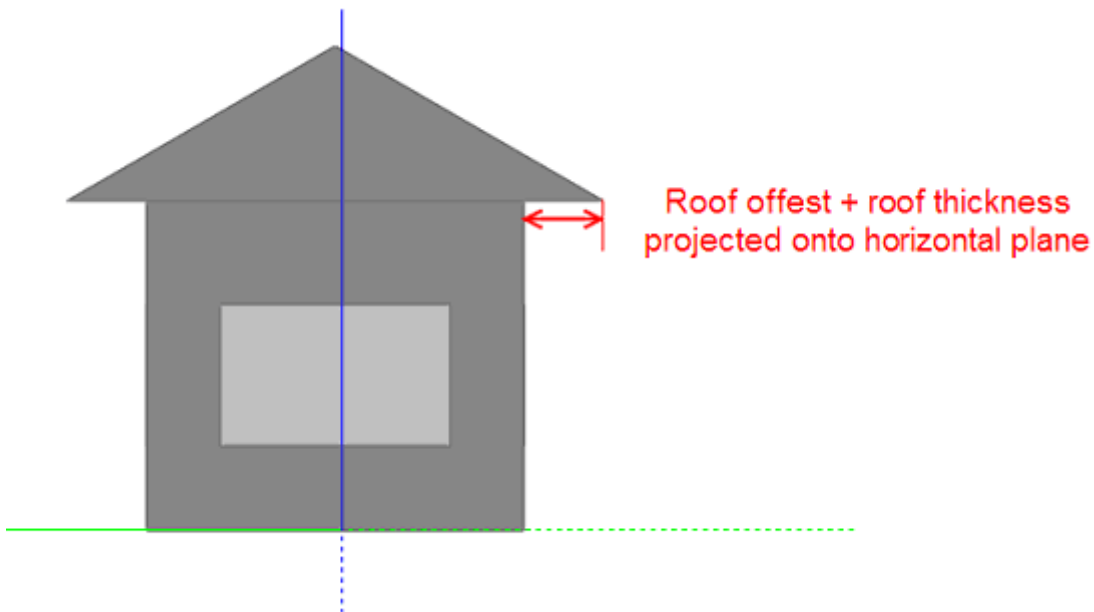


The roof geometry is created as if a stub wall existed from the original extrusion, cut to fit within the roof space and on top of which the roof assembly sits:



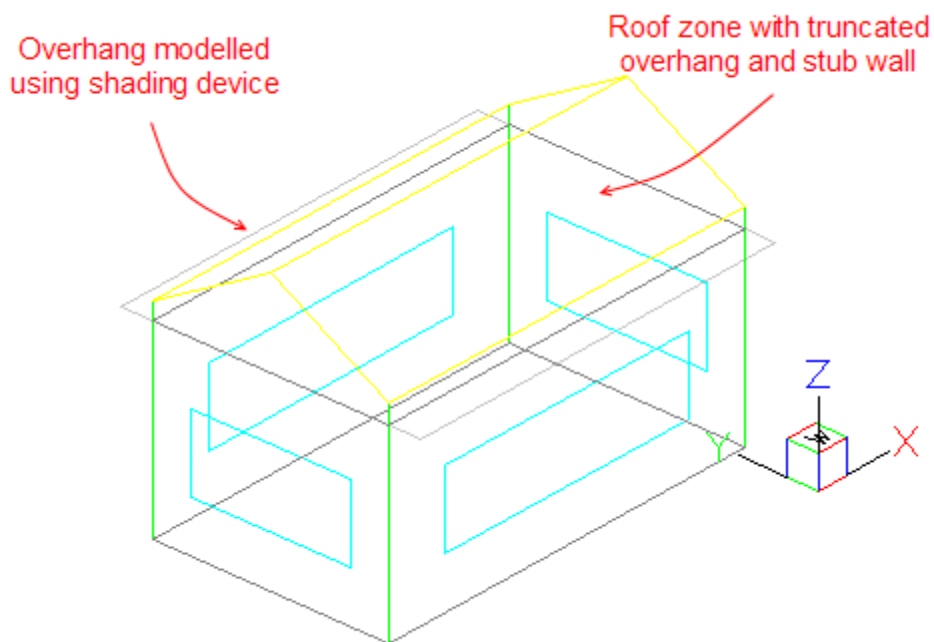
The resulting roof block is composed of a polyhedron representing the inner zone, which is created from the original inner block perimeter and is cut to fit within the roof space and a number of slabs that represent the actual roof construction. The roof slabs are generated from the inner roof profile and an outer profile created using the roof construction thickness.





EnergyPlus model

DesignBuilder automatically generates overhangs for Pitched roof block types using EnergyPlus shading devices. The diagram below shows EnergyPlus DXF output of the building model generated using the above data. It shows the overhangs in grey.



Calculation Of Simple HVAC Energy Consumption From EnergyPlus Loads

With Simple HVAC DesignBuilder calculates boiler, chiller, DHW fuel consumption from the loads calculated by EnergyPlus using the CoP data at the building level on the HVAC tab as follows:

Boiler energy = *EnergyPlus heating loads* / **Heating system CoP**

Chiller energy = *EnergyPlus cooling loads* / **Cooling system CoP**

DHW energy (kWh) = **DHW CoP** x 1000 (kg/m³) x 4.187 (KJ/kg-K) x (*EnergyPlus DHW m3 loads*) x (**Delivery water temperature - Mains water temperature**)

Where:

DHW CoP is **Heating system CoP** when using '1-Central heating boiler' DHW type

Data shown in **blue** is HVAC model data.

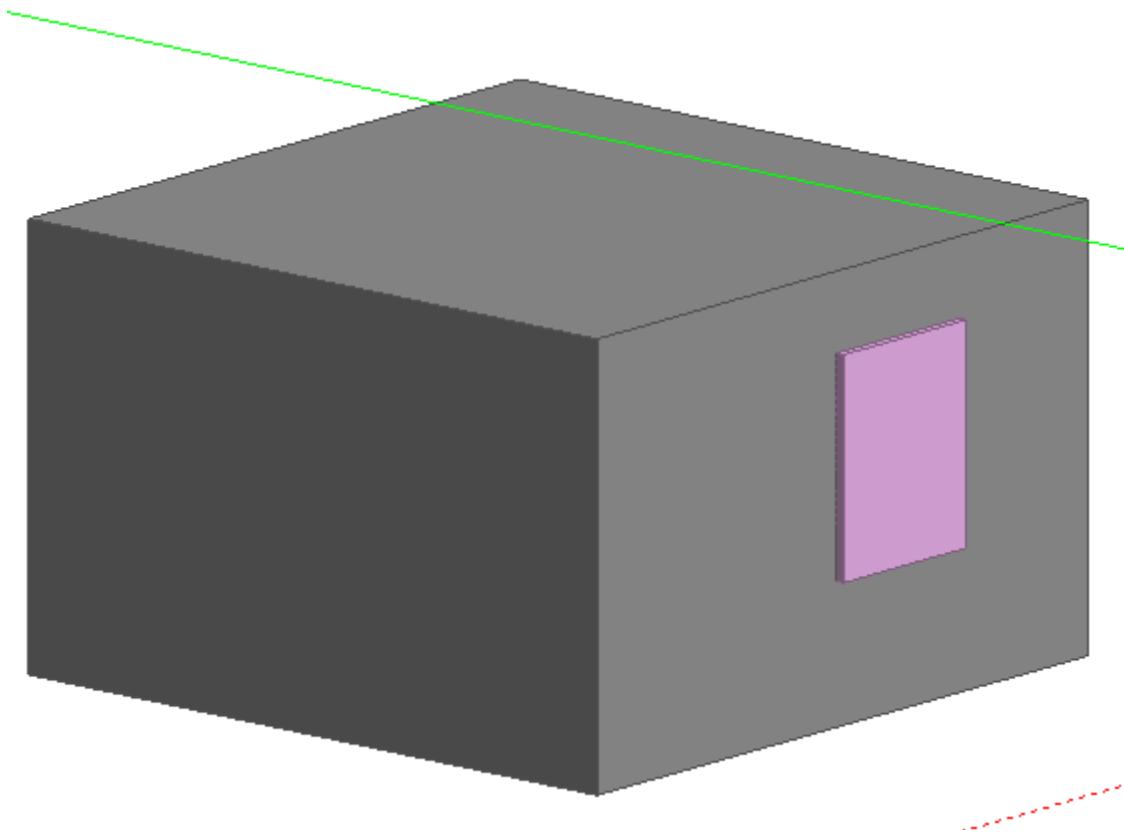
Data shown in *italics* is read from the [EnergyPlus results eso file](#).

Component Blocks Used For Local Shading

[Component blocks](#) are a flexible way to apply shading to any part of the building and can be used to model more complex window shading systems not catered for by the in-built [local shading](#) systems that can be selected on the Openings tab. The issues to consider when using component blocks in this way are discussed below.

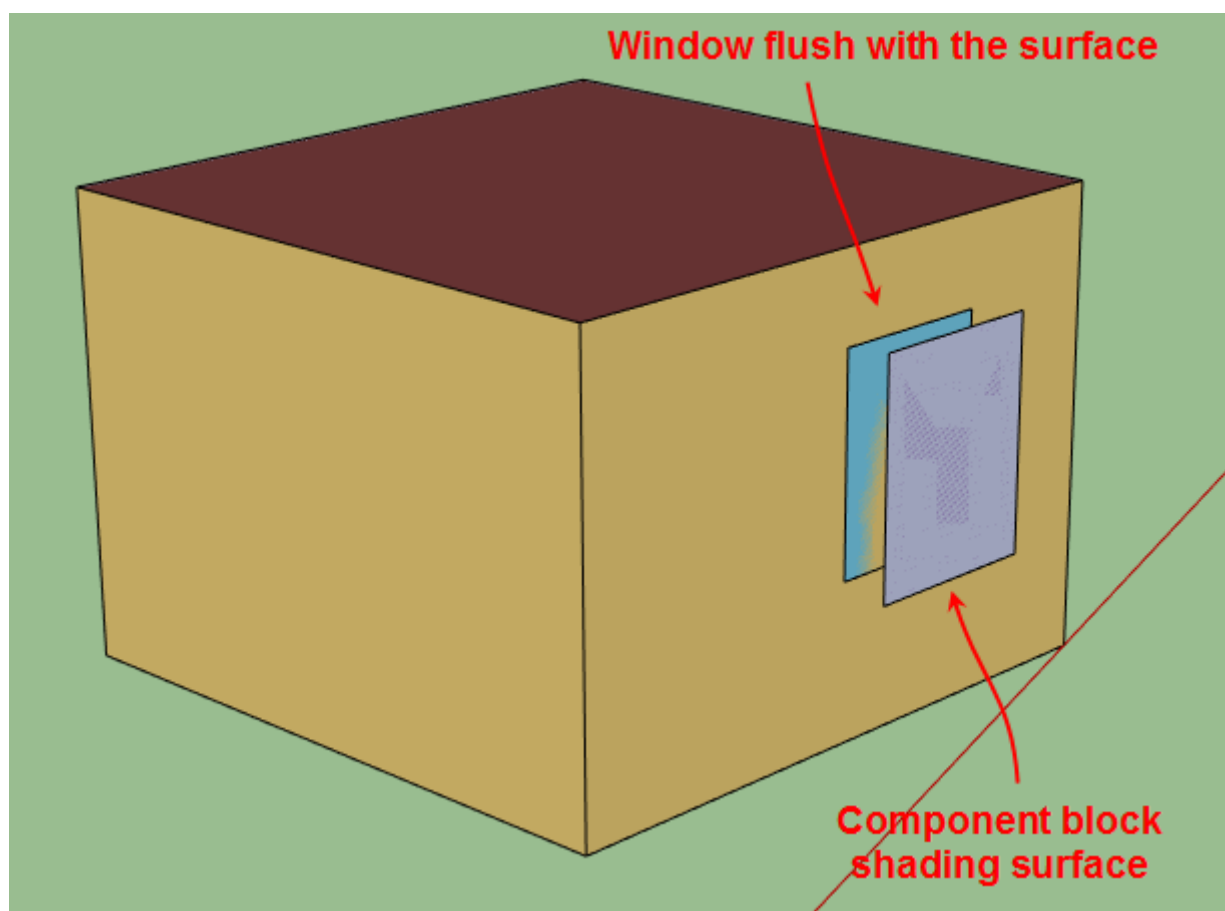
Positioning

When using the [zone inner geometry](#) option for surface geometry, component blocks placed close to the window at building level may not be as close to the window in the simulation as might be expected from the view in the model. For example, in the simple model shown below a component block has been placed directly in front of a window, which cannot be seen as it is hidden by the component block. It is placed so that it touches the outer surface of the block. So one would think that it is touching the window in the simulation model and therefore no solar gain and light could pass round or through the component block shade.



However results from simulations will show quite a lot solar gain entering the window. This is because the window is actually placed on the zone surface which is on the *inner* surface of the block, the difference between the inner zone surface and the block outer surface being due to the [External wall thickness](#). In this model the wall thickness is 0.23m so there will be a gap of 0.23m between the component block shade and

the window in the simulation model. This can be seen in the example DXF output shown below which corresponds to the model above.



The DXF output above shows the gap between the window and the component block shade caused by the External wall thickness.

The solution in this case was simply to use either one of the [External measurement Geometry conventions](#) or a zero External wall thickness which removes the gap between window and shade.

Reflection from ground

As discussed [elsewhere](#), shading of the ground by component blocks is only taken into account if the [Model all external reflections and shading of ground reflected solar](#) option is used. So if you use component blocks to model a local shading device and forget to include reflections in the simulations you will obtain higher than expected solar gains due to unobstructed ground-reflected solar gain.

Data Recovery

There can be times when data is lost due to a program crash or a power failure at a time before you had a chance to save the model. DesignBuilder keeps track of the model in its various stages using 3 different mechanisms:

An optional [incremental backup](#) system where the current working file is automatically saved to the [Backup folder](#) every 10 minutes (by default). These files have names such as **My File Autosave 18_01_2012_14_05_50.dsb**. In this example the original file was called **My File.dsb** and the backup was made at 14:05:50 on 18 January 2012.

When opening an existing file the original file is copied to the backup folder. In this case the file is called something like **My File Archive 18_01_2012_13_49_01.dsb**. In this example the original file was called **My File.dsb** and the file was opened and copied at 13:49:01 on 18 January 2012.

If for any reason DesignBuilder does not close down cleanly and is unable to save data, an skh backup file will remain in the same folder as the dsb file and DesignBuilder will ask whether you want to recover

this data the next time you open the dsb file. The skh file will have the same name as the original dsb file but will have a ~ character prepended. e.g. **~My File.skh**.

Files in the [Backup folder](#) are stored for up to 30 days before being automatically deleted.

To recover data you should check the files in the Backup folder and find possible candidates based on the filename and time of saving. Then copy the files you would like to check up to the DesignBuilder Data folder and open them in DesignBuilder to find the most complete data set.

See also [Program options, Backups](#)

Standards Used In DesignBuilder

DesignBuilder and it's underlying calculation engines reference and apply a wide range of different national and international standards.

Standards used in DesignBuilder calculations		
Standard	Description	How used in DesignBuilder
EN ISO 12831	Heating systems in buildings. Method for calculation of the design heat load.	The infiltration defined in the model is converted to m/s for input to EnergyPlus using the methods described in this standard. See: "Natural Ventilation and Infiltration model Options" on page 1272 "Airtightness (Scheduled Natural ventilation)" on page 400
EN ISO 6946	Building components and building elements - Thermal resistance and thermal transmittance -Calculation methods.	Calculation of R and U-values for Constructions
ISO 13788 2012	Hygrothermal performance of building components and building elements - Internal surface temperature to avoid critical surface humidity and interstitial condensation - Calculation methods.	Condensation analysis calculations for constructions.
ASHRAE 55	Thermal Environmental Conditions for Human Occupancy.	EnergyPlus occupant comfort calculations: Graph Related Outputs Simple ASHRAE Standard 55 Adaptive ASHRAE Standard 55
ASHRAE 62.1	Ventilation and Acceptable Indoor Air Quality.	A range of ventilation calculations including DCV to ensure adequate fresh air is provided to occupants: "CO2 generation rate" on page 335 Zone Air Distribution Effectiveness AHU Demand Controlled Ventilation (DCV)
ASHRAE 90.1 Appendix G 2010, 2013, 2016	Performance Rating Method (PRM).	ASHRAE 90.1 Appendix G is the Energy Standard for Buildings Except Low-Rise Residential Buildings and as a key component of the LEED Optimize Energy Performance credit calculations .

EN 15251	Indoor environmental input parameters for design and assessment of energy performance of buildings addressing indoor air quality, thermal environment, lighting and acoustics.	EnergyPlus occupant comfort calculations: Adaptive CEN Standard 15251 "Comfort" on page 1718 "Control mode" on page 489
ISO 10292	Glass in building - Calculation of steady-state U values (thermal transmittance) of multiple glazing	Calculation method for centre of glass U-value on the Glazing dialog.
EN 673	Glass in building – Determination of thermal transmittance (U value) – Calculation method.	Calculation method for centre of glass U-value on the Glazing dialog.
CIBSE TM52	TM52 The limits of thermal comfort: avoiding overheating (2013).	"CIBSE TM52 Overheating Assessments" on page 1632
CIBSE TM59	TM59 Design methodology for the assessment of overheating risk in homes (2017).	"CIBSE TM59 - Modelling and Analysis Guide" on page 1634
Part-O (England and Wales)	Approved Document O: Overheating mitigation.	CIBSE TM59 - Modelling and Analysis

Standards DesignBuilder complies with

Standard	Description	How used in DesignBuilder
ASHRAE 140 2017 / BESTEST	Method Of Test For Evaluating Building Performance Simulation Software. The tests compare temperatures, energy flows and other outputs with other simulation software. DesignBuilder passes these tests.	Validation tests Download a report.
EN ISO 13790	Energy performance of buildings - Calculation of energy use for space heating and cooling.	Validation tests Download a report.
EN ISO 15265	Energy performance of buildings - Calculation of energy needs for space heating and cooling using dynamic methods - General criteria and validation procedures.	Validation tests Download a report.

Other Standards DesignBuilder can be used for

Standard	Description	How used in DesignBuilder
ASHRAE Standard 209	Define reliable, consistent procedures that advance the use of timely energy modeling to quantify the impact of design decisions	DesignBuilder can be used to comply with all modelling cycles, including sensitivity and optimisation cycles. Watch this webinar , which

		covers the various modelling cycles that DesignBuilder parametric tools can help with.
CIBSE TM54	Evaluating operational energy use at the design stage	See the TM54 Simple HVAC case study and the TM54 Detailed HVAC case study .
NABERS UK Design for Performance (DfP)	A framework to predict and ensure that new office buildings achieve a target energy rating in use.	See NABERS DfP case study
ASHRAE Guideline 14/IPMVP/CIBSE TM63	These are complementary documents providing guidance on developing calibrated models for measurement and verification studies (Option D in IPMVP)	See modelling and calibration case study
ASHRAE Standard 189.1	This standard provides minimum requirements for the design, construction, and operation of high-performance green buildings.	DesignBuilder can model and simulate building performance aspects, such as energy consumption, indoor air quality, and thermal comfort, to ensure designs meet the standards' criteria.
ISO 50001: Energy Management Standard	This standard provides a framework for organisations to systematically manage their energy performance and reduce consumption.	DesignBuilder can provide insights into energy consumption patterns and performance of buildings which is crucial for identifying significant energy users (SEUs) and developing energy performance indicators (EnPIs), as recommended by ISO 50001.
ISO 52016: Energy Performance of Buildings	This standard provides a comprehensive set of rules and methods for evaluating the energy performance of buildings, including heating, cooling, and other energy-related aspects.	DesignBuilder can be used to create detailed energy models that comply with the requirements of ISO 52016.

To see how DesignBuilder can be used for various regulations, certification schemes and green ratings see our web pages:

[Energy Assessors product webpage](#)
[Green rating schemes](#)

Thermal Comfort Calculator

Accessed from the [Tools](#) menu

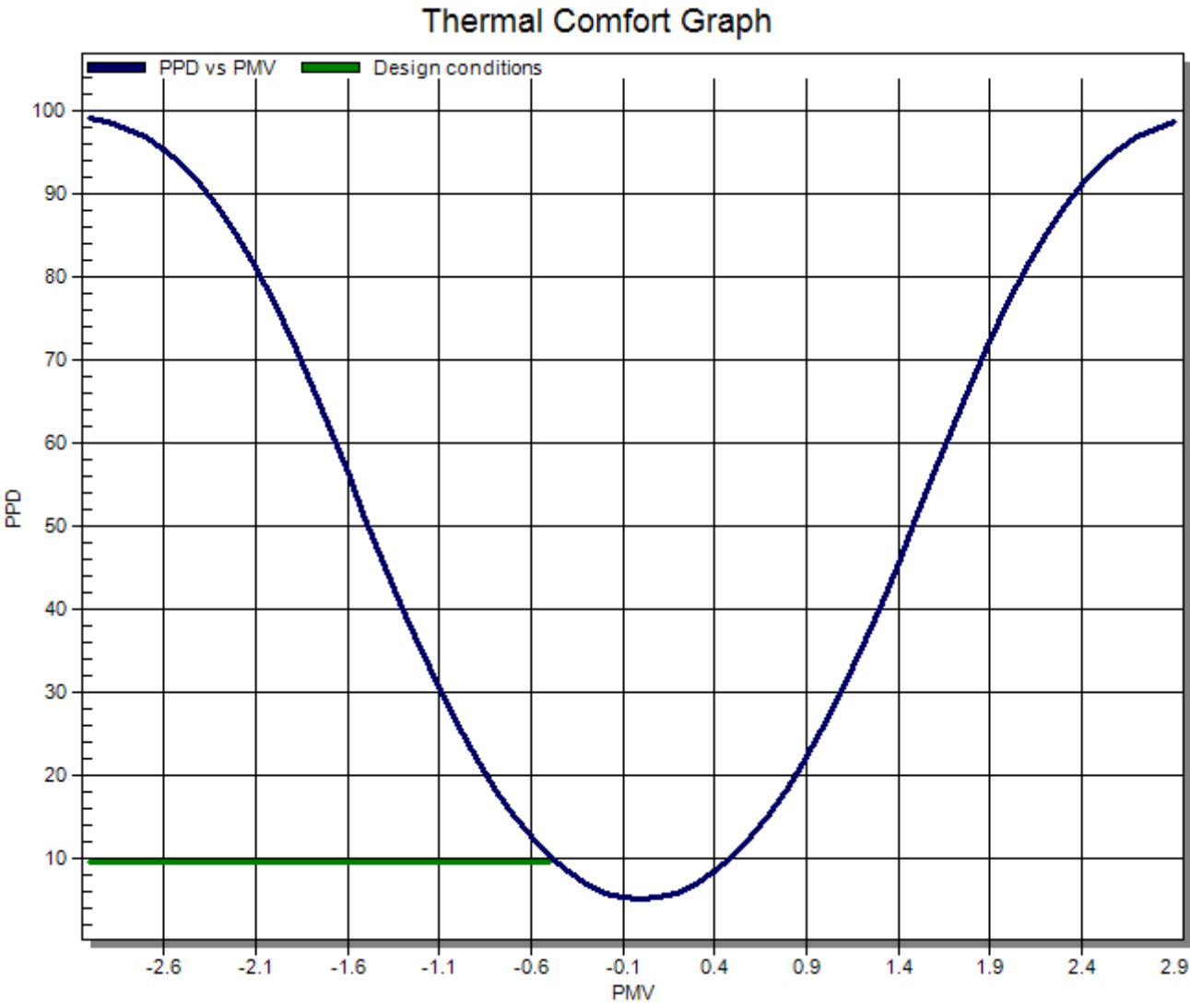
DesignBuilder provides a dialog which allows you calculate thermal comfort based on the Fanger comfort indicators **PPD** (% people dissatisfied) and **PMV** (predicted mean vote). These inputs are required:

Air temperature (°C or °F)
 Radiant temperature (°C or °F)
 Clothing (Clo)
 Activity level method - choose from one of:
 Activity (MET) or
 Activity (W/person or Btu/hr-person)
 Air speed (m/s or ft/min)
 Relative humidity (%)

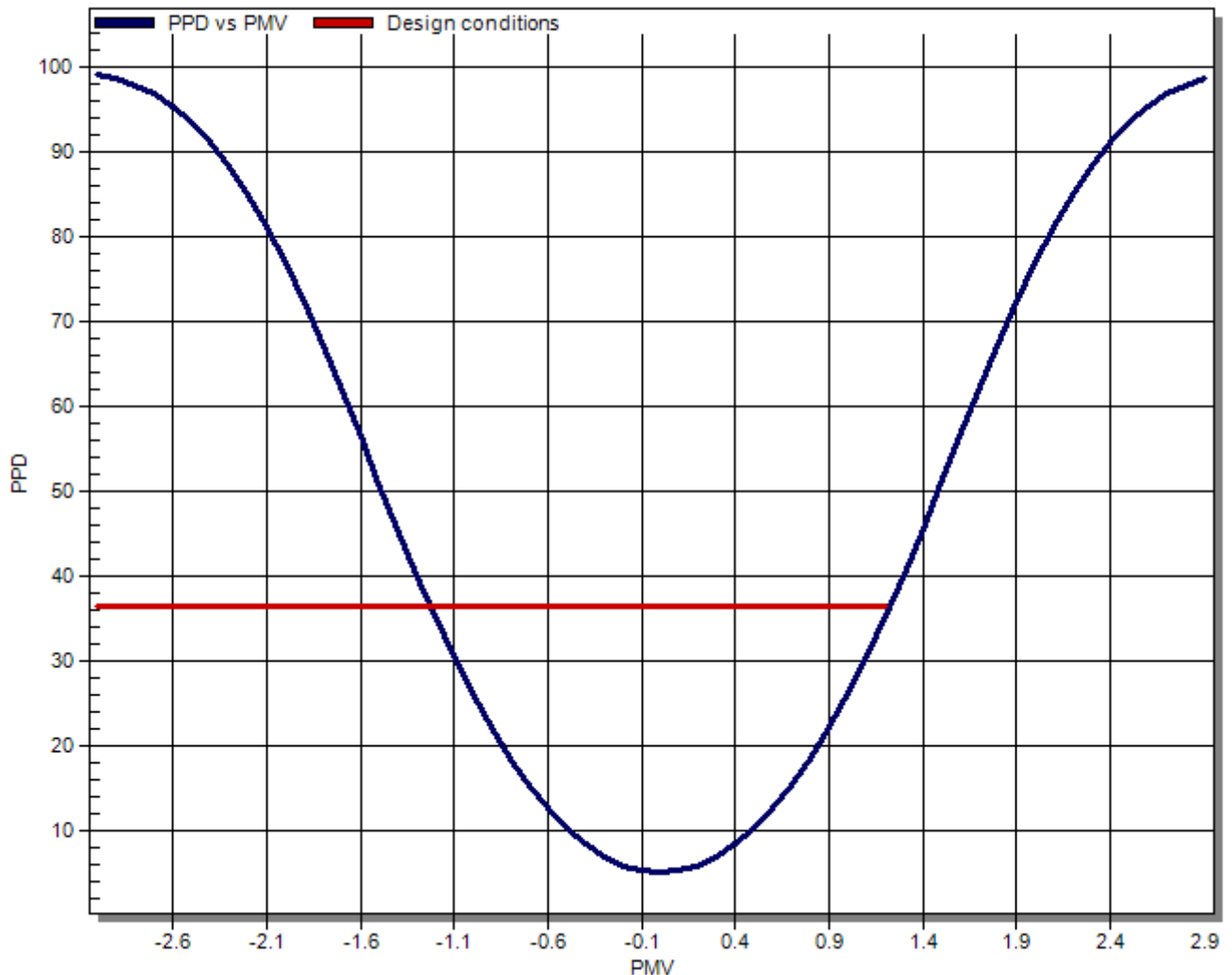
Outputs are:

Operative temperature
Predicted mean vote (PMV)
% people dissatisfied (PPD)

A curve illustrating the relationship between PMV and PPD is displayed to the right on the dialog.



Thermal Comfort Graph



To use the tool, enter the environmental conditions and press the **Update** button at the bottom of the dialog. This causes the graph and PPD/PMV outputs to be updated. If the PMV value is within the comfort region (-0.5 to +0.5) then the outputs and the line between the PPD axis and the curve are displayed in green, otherwise they are displayed in red as shown in the example outputs above.

You can read more about Fanger comfort calculations on the [EnergyPlus Thermal Comfort](#) page.

Calculating Fan Pressure Rise

If the fan pressure rise is not known it can be calculated approximately from Specific Fan Power (SFP) data using:

$$\Delta P = 1000 * SFP * \text{Fan total efficiency}$$

Annex E of ISO 5801 shows that by rearranging the formula it can be derived that the SFP is a function of fan pressure divided by the efficiency of the fan system. Therefore the SFP will increase or decrease with a respective increase or decrease in the system pressure.

The SFP is a function of the volume flow of the fan and the electrical power input and is quoted for a particular flow rate;

$$SFP = P_e / V$$

Where:

V is volume flow (l/s)

P_e is electrical power input (W) to the fan system or complete air movement installation

[Reference FMA, UK, 2006]

Typical values for various system types are shown in the table below.

System Type	Specific Fan Power (W/l-s)
Central mechanical ventilation including heating, cooling and heat recovery	2.5
Central mechanical ventilation including heating and cooling	2.0
All other systems	1.8
Local ventilation units within the local area, such as window/wall/roof units, serving one room/area	0.5
Local ventilation units remote from the local area, such as ceiling void or roof mounted units, serving one room/area	1.2
Fan coil units (rating weighted average)	0.8

Source ESTA.

Note: In reality the pressure rise across the fan has to meet the pressure drop across the index circuit of the actual system (i.e. the circuit with highest resistance to airflow). A more accurate estimate of fan pressure can be obtained by estimating the overall length of the ducting index circuit, assuming it's sized on 1 Pa/m, adding approximately 20% for fittings and pressure drops for specific items such as diffusers, HEPA filters, etc.

Importing Custom Templates And Components

It can be useful to be able to import components and templates set up outside DesignBuilder to models. This page explains how to do it. The process is not fully documented and requires a willingness to experiment, but in most cases it is fairly straightforward. The recommended process is as follows:

[Export a .ddf file](#) containing similar model data to that which you would like to import.

Rename it to have extension .zip so it can be opened in WinZip, WinRAR or similar zip file editor.

Open the .zip file to see a series of .cdt (compressed .dat files).

Extract the .cdt files and open them in a text editor.

Study the format - there is no documentation to explain the format, but it is usually fairly easy to work out by reference to the corresponding DesignBuilder dialogs and the headers. Each .cdt file consists of a line of category ids, followed by a header line which indicates the meaning of the fields below. Each further line is a record with fields separated using the "#" character. See below for an example of ActivityTempates.cdt.

If you need to make more widespread edits then it is usually easiest to import the .cdt file into a spreadsheet. Here the data can be edited and exported again using tab delimiters and replacing tabs with "#" in a text editor. Note you must avoid including adjacent "#" characters in the .cdt and the easiest way to ensure this is to search for "##" and replace with "# #". Note the first column starts with

```

"##".
#86 #351 #352 #353 #354 #355 #356 #357 #358 #359 #360 #361 #362 #363 #364 #365 #366 #367 #368 #369 #370 #371 #372 #373 #374 #375 #376 #377 #378 #
#Id #Name #CategoryId #Default #SectorId #NCMName #ComputersOn #ComputersValue #EquipmentOn #EquipmentValue #ProcessOn #ProcessValue #Fluorescent
#10000 #ASHRAE 62.1 - Office Buildings - Office--Enclosed #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #.05 #10 #1 #20 #26 #5 #0 #
#10001 #ASHRAE 62.1 - Office Buildings - Office--Open Plan #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #.05 #10 #1 #20 #26 #5 #0 #
#10002 #ASHRAE 62.1 - General - Conference/Meeting #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #.5 #9 #1 #20 #26 #5 #0 #300 #1 #
#10003 #ASHRAE 62.1 - Educational Facilities - Classroom (ages 5-8) #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #.25 #11 #.75 #20 #2
#10004 #ASHRAE 62.1 - Educational Facilities - Art classroom - Exh 3.5 L/(s*m2) #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #.2 #9 #1
#10005 #ASHRAE 62.1 - Public Assembly Spaces - Auditorium seating area #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #1.5 #3 #1 #20 #2
#10006 #ASHRAE 62.1 - Misc. Spaces - Bank vaults/safe deposit #1 #False #1 # #0 #5 #0 #5 #0 #0 #1 #0 #.05 #10 #1 #20 #26 #5 #

```

To create a .ddf file for importing to DesignBuilder use the reverse process. Create the .cdt files (by modifying equivalent existing files) paying careful attention to all fields. Then create the .ddf by zipping the .cdt files to be imported and renaming to use the .ddf file extension.

Import the .ddf file to the DesignBuilder model.

A common requirement is to create compact schedules from data from other sources (e.g. measurements or energy code references). You can use the above procedures for this by adding schedule records to Schedules.cdt. Compact schedule data is stored in the field **CompactData**. Note that end of lines are marked by "|^" sequences. So a compact schedule like:

```

Schedule:Compact,
7:00 - 18:00 Mon - Fri,
Fraction,
Through: 31 Dec,
For: Weekdays SummerDesignDay WinterDesignDay,
Until: 07:00, 0
Until: 18:00, 1,
Until: 24:00, 0,
For: Weekends,
Until: 24:00, 0,
For: AllOtherDays,
Until: 24:00, 0 ;

```

Becomes text data in the Schedules field "CompactData":

```

Schedule:Compact,|^7:00 - 18:00 Mon - Fri,|^Fraction,|^Through: 31 Dec,|^For: Weekdays
SummerDesignDay WinterDesignDay,|^Until: 07:00, 0,|^Until: 18:00, 1,|^Until: 24:00, 0,|^For:
Weekends,|^Until: 24:00, 0,|^For: AllOtherDays,|^Until: 24:00, 0 ;|^

```

Lock/Unlock Model

Accessed from the [Tools](#) menu

You can lock the model to ensure that no further changes are possible. This may useful in cases when you wish to send a model on to colleagues or clients to allow them to view inputs and run calculations but not to make any changes to the model itself.

To lock the model you must first enter a password which can be used to later to unlock it again. Please make sure to keep a note of the password when locking the model to ensure that you can unlock it again if necessary.

Command Line Processing

Models can be opened and processed in a variety of ways via the command line. This can be useful for batch processing multiple models.

The command line syntax is:

```
"DesignBuilder" "MyFile" /process=ListOfProcesses
```

Four key points need to be considered when working via the command line:

DesignBuilder is the file path and name of the DesignBuilder executable, e.g. "C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe".

MyFile is the file name and path of the model. It must be surrounded by quotes if it contains any spaces.

There must be a space between MyFile and /process=.

Individual process commands in ListOfProcesses must be separated by a comma.

Available Processes

The table below lists the command line processes that can be executed on an open DesignBuilder model.

Command	What it does
miGSS	Changes the view to the simulation screen
miGHL	Changes the view to the Heating Design screen
miGHG	Changes the view to the Cooling Design screen
miGDY	Changes the view to the Daylighting screen
miGCalculate	Changes the view to the SBEM/DSM Calculation screen
miGCFD	Changes the view to the CFD screen
miTUpdate	Runs the calculation for the current screen. Generally, this command should be placed last on the line, before any ChangeAttributeValue commands.
TabChange_n	Changes the view to tab n on the current screen. e.g. TabChange_2 on the Daylighting screen will change the view to the Annual daylighting tab
SimStartDate d m	Sets start date to the numerically specified day and month. e.g. SimStartDate 4 5 sets the start date to May 4.
SimEndDate d m	Sets end date to the numerically specified day and month. e.g. SimEndDate 11 5 sets the start date to May 11
ChangeAttributeValue <i>AttributeName Value</i>	Changes the value of the specified attribute <i>at building level</i> for the currently selected building. As for attributes set in the UI, the inheritance of building level attributes down to block, zone, surface and opening levels is governed by DesignBuilder's normal inheritance rules. Multiple ChangeAttributeValue commands can be included in a single "/process= string". Multiple commands should be separated by commas, as with other process commands. For example, the following command prompt opens a named model, switches to the Daylighting screen, sets the Zone multiplier to 2 at building level, sets the Daylighting Simulation type to "4" (selecting the 4-LM-83-12 option) and finally, runs the daylight simulation: <pre>"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe" "C:\<somepath>\shoebox.dsb" /process=miGDY, ChangeAttributeValue ZoneMultiplier 2, ChangeAttributeValue RMSimulationType 4, miTUpdate</pre> Syntax for Attribute Values: a. For numeric attributes, DesignBuilder stores values internally as integer codes. For example, to set 0-None in Openings > External Windows > Dimensions > Type, the command is: <pre>ChangeAttributeValue Layout 0</pre>

	b. For booleans, use 1 or 0 to specify True or False. For example, for “True” in Activity > Computers, the correct command is: <pre>ChangeAttributeValue ComputersOn 1</pre> c. For real numbers, just add the number with decimal separator if required. For example, for 0.5 in Activity > Occupancy Density, the correct command is: <pre>ChangeAttributeValue OccupancyValue 0.5</pre> Note: DesignBuilder stores real values using SI units and uses “.” as the decimal separator. ChangeAttributeValue expects <i>Value</i> to be provided in the same native format, regardless of the Windows Region selected on your machine. d. For database table selections such as Schedules, Constructions etc. settings are made using a numeric Id, not the name of the item. Numeric Ids aren’t available directly via the UI, but they can be obtained via the API, iterating over the table records to find the item by name. Once the correct record is returned, the Id can be read (the zeroth field in the record) and used to set the Id of the Schedule, Construction etc. using ChangeAttributeValue. Tip: To ensure you are using the correct <i>AttributeName</i>: 1. Go to Tools > Program Options > Interface. 2. Enable Show attribute names in tooltips. 3. Hover your mouse over the input field in the DesignBuilder interface. The tooltip will display the <i>AttributeName</i>.
ExternalCommand_ <i>arg</i>	Run an external command with optional arguments
ImportModelData_ <i>File path and name</i>	Imports the model data stored in the defined file. The file must be a csv file previously exported from the model.
ImportLibraryData_ <i>File path and name</i>	Imports the library data stored in the defined file
miFExportAsXML	Exports the model as a dsbXML file with the same name as the current model
UseSimManager	Use this argument to ensure the simulation is run using the Simulation Manager. The Server and EnergyPlus server method settings currently set in the model will be used - localhost and EnergyPlus respectively by default.
NoClose	Prevents DesignBuilder from shutting down after all commands have been processed

Example Usage

```
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe" "C:\<somepath>\shoebox.dsb"
/process=miGSS, miUpdate
```

Runs an EnergyPlus simulation and closes.

```
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe" "C:\<somepath>\shoebox.dsb"
/process=miGSS, miTUpdate, ExternalCommand_ProcessResults
```

Runs an EnergyPlus simulation, process the results in a custom standalone application called ProcessResults.exe and closes. In this case ProcessResults.exe must be either in the folder from which the command is executed or in a folder referenced by the Windows path list.

```
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe" "C:\<somepath>\shoebox.dsb" /process= miGHL,
miTUpdate, miGHG, miTUpdate, NoClose
```

Runs both Heating Design and Cooling Design calculations, then stops DesignBuilder from closing.

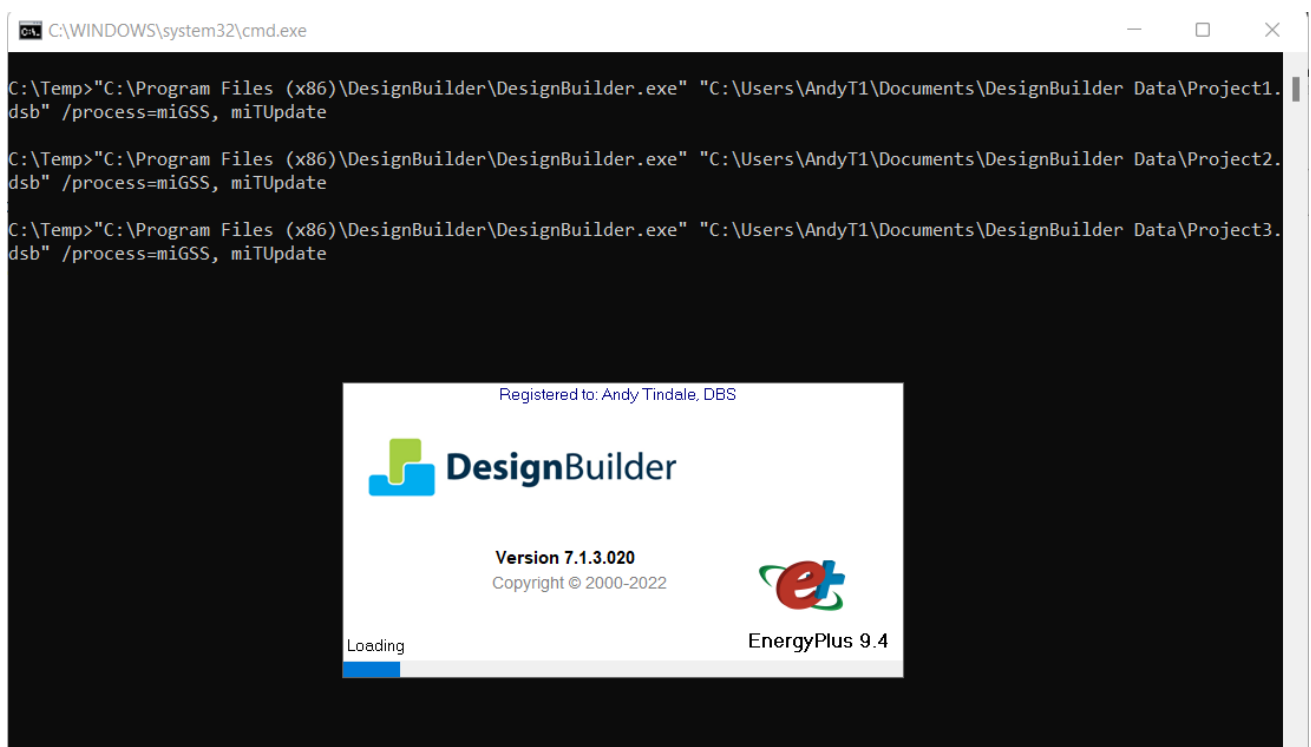
```
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe" "C:\<somepath>\shoebox.dsb"
/process= miGDY, miTUpdate, TabChange_2, miTUpdate
```

Runs both Illuminance and Annual daylighting calculations and closes.

```
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe" "C:\<somepath>\shoebox.dsb"
/process= miGCFD, miTUpdate
```

Runs the CFD simulation and closes. Whether this is simple or advanced CFD simulation depends on the model options previously set in Shoebox.dsb.

Batch Processing



A convenient way to process a set of models is to run a script with multiple commands. This allows several models and/or several calculations to be run unsupervised.

Scripts can be set up in a number of ways including using Windows batch files and PowerShell and Python scripts.

The easiest way to set up a batch process for most users will be through a simple batch file. To do that follow the steps below.

1. Set up the DesignBuilder models you wish to simulate with the required model data and calculation options. Close DesignBuilder.
 2. Create a batch file in a folder that you can easily navigate to from the command line.
 - a. For example create a folder from the root of C:\ drive called "DBBatch".
 - b. Create a blank text file in that folder called something like BatchRunDB.bat.
 3. Copy the commands that you wish to execute into the batch file using a text editor such as Notepad. For example to run simulations for 3 models called Project1, Project2 and Project3 stored in the default Documents\DesignBuilder Data folder, the contents of the batch file might look something like this:

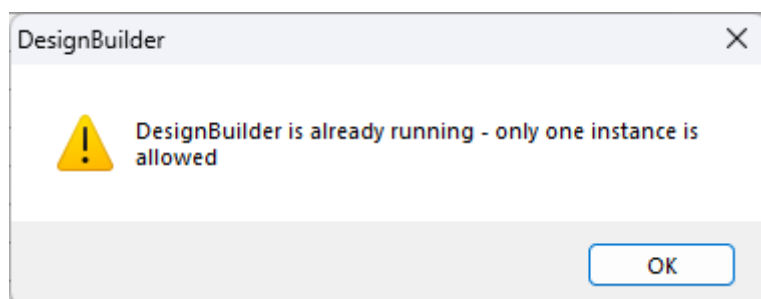
```
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe"  
"C:\Users\MyUsername\Documents\DesignBuilder Data\Project1.dsb" /process=miGSS,  
miTUpdate  
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe"  
"C:\Users\MyUsername\Documents\DesignBuilder Data\Project2.dsb" /process=miGSS,  
miTUpdate  
"C:\Program Files (x86)\DesignBuilder\DesignBuilder.exe"  
"C:\Users\MyUsername\Documents\DesignBuilder Data\Project3.dsb" /process=miGSS,  
miTUpdate
```

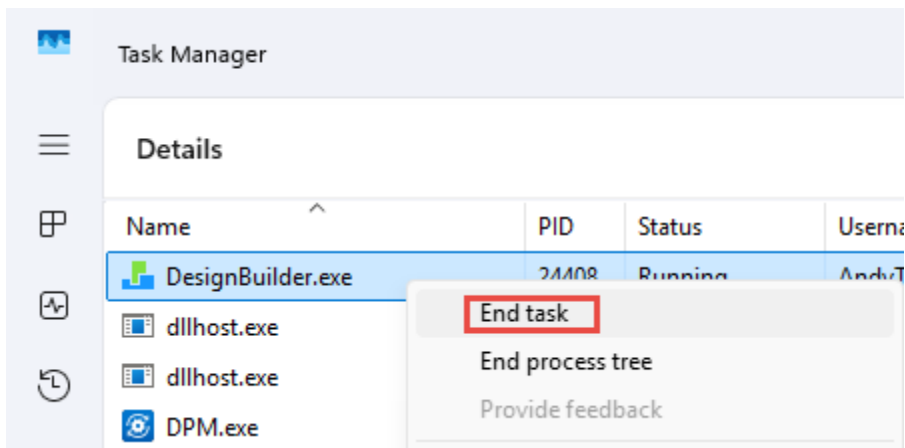
The above commands may wrap onto new lines when viewed in your browser, but in the batch file each command should be on 1 line.
 4. Run the batch file, BatchRunDB.bat, by double-clicking on it in Windows File Explorer, or by typing "BatchRunDB.bat" on the command line and pressing <Enter>
 5. The batch file will load each model into DesignBuilder in turn and execute the commands requested without any user intervention required. When the commands have all been executed for one model, DesignBuilder will save the model and close, returning control to the batch file.
- In the above example, Project1.dsb is loaded, a simulation is run and when complete and the results have been loaded to the model the project is saved to disk. Then DesignBuilder closes. The same process is repeated for Project2.dsb and Project3.dsb. Once all commands have been executed the batch process is complete and the requested results will be stored in each of the models.
6. To retrieve the simulation results, open the processed dsb files in the usual way.

Tip: If you wish to continue to use your computer while the batch process runs, it can help to minimise the DesignBuilder main window so that it doesn't keep grabbing focus each time a new model is loaded.

Limitations

In DesignBuilder v2025.1, after running a command line process, the software remains open and must be closed manually using the Task Manager:





This issue will be addressed in a future version.