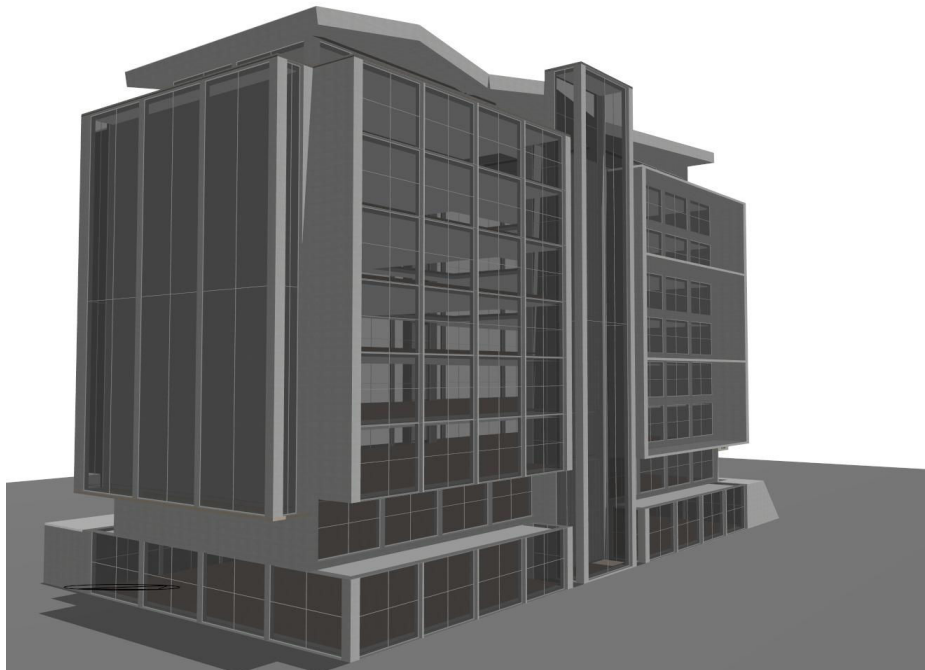




DesignBuilder
Software

DesignBuilder Simulation + CFD Training Guide



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

Contents

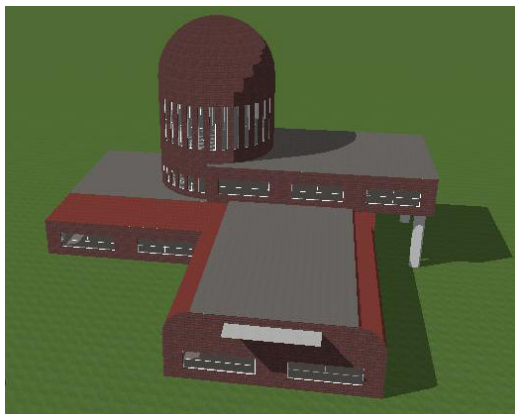
DesignBuilder Simulation + CFD Training Guide	1
Contents.....	3
1 Introduction	1
Overview.....	1
2 DesignBuilder Interface.....	2
2.1 Opening screen	2
3 Create Building Geometry.....	3
Create a new project.....	3
Location tab	3
Template tab.....	3
Add a building.....	3
4 Drawing Options	5
Using the drawing tools	7
Import 2-D Drawing file	13
5 Model Options.....	15
The data tab.....	15
Advanced tab - Lumping and Merging	17
Heating Design tab	17
Cooling Design tab	18
Simulation tab.....	19
Calculation options.....	19
Display tab.....	19
Drawing tools tab	20
Block tab.....	20
Project details tab	20
6 Introduction to Model Data.....	21
Activities.....	21
Internal gains	21
Environmental control	22
The NCM zoning process	22
Dividing the building into zones by drawing partitions.....	23
Data inheritance at block, zone and surface level.....	25
Assigning activities.....	26
Constructions.....	27
Defining constructions in DesignBuilder.....	27
Adjacency	29
Entering construction details in DesignBuilder.....	30
Openings	33
Lighting	34
HVAC.....	36
7 Exercises on Geometry and Model Data	38
Default Model Templates.....	38
Default 20x10 Single Zone.....	38
Default 20x10 2-Zone	39
Exercise 1	39
Exercise 2.....	40
Exercise 3.....	41

Exercise 4	41
Exercise 5	42



.....	43
Exercise 6	43
Exercise 7	44
Exercise 8	44
Exercise 9	45
Exercise 10	45
Exercise 11	45
Exercise 12	46
Exercise 13	47

Use as many of the drawing tools as possible to create a building with more complex geometry such as that illustrated below. Use construction lines to provide snap points to position curved walls and other features. 47



.....	47
Exercise 14	47
Exercise 15	48
8 Data Management	52
Loading data from templates	52
Example – Load data from template	52
Reload data	54
Clear data	54
Export and Import Components and Templates	55
9 Heating Design Calculations	59
Introduction	59
Heating design screen	60

Steady-state.....	60
Summary	61
10 Cooling Design Calculations	62
Introduction	62
Cooling design screen	63
Analysis	63
Summary	63
Parametric Cooling Design	64
11 Simulations using Hourly Weather Data.....	65
Calculation options	65
Model Checking Strategy.....	66
Simulation Analysis.....	66
Parametric Simulations.....	69
Exercise – Parametric simulation - Basics.....	71
12 Timing - Schedules, Profiles, Holidays.....	74
Typical workday	74
Schedules.....	75
7/12 Schedules	75
Profiles.....	76
Compact Schedules.....	78
Setpoint temperature schedules	81
Holidays.....	83
Compact Schedules.....	83
7/12 Schedules	83
Typical workday	83
13 Glazing and Solar Shading.....	84
Modelling Facades.....	84
Drawing openings at surface level	85
Editing Openings at building level.....	86
Solar Shading	88
Window shading devices	88
Local shading devices.....	91
Component blocks	94
14 Daylighting	97
Lighting areas	97
Daylight control procedure	100
Exercise – Daylighting calculations.....	101
Optional Exercise - DElight.....	104
15 Comparing results	105
16 Natural Ventilation.....	108
Scheduled Natural Ventilation.....	109
Introduction	109
Data Entry.....	110
Scheduled Natural Ventilation with Internal Airflow.....	114
Data entry	114
Results analysis.....	114
Calculated Natural Ventilation.....	115
Introduction	115
Data Entry.....	116
Advanced Calculated Natural Ventilation.....	121
Introduction	121

Wind Pressure Coefficients.....	121
Testing the effect of wind.....	121
Display wind speed alongside other results.....	124
17 Simple HVAC	125
Introduction	125
Example.....	125
More Simple HVAC.....	130
Example.....	130
Radiant Heating using Simple HVAC.....	130
Example.....	130
Night Cooling using Mechanical Ventilation.....	132
Example.....	132
18 DesignBuilder Compact HVAC.....	135
Zone-based Systems	135
AHU-based Systems.....	136
Boiler, Chiller, Fans, Pumps Fuel consumption	136
Fan-coil Units.....	136
Example.....	137
Unitary Single Zone - Basic	139
Example.....	139
Unitary Single Zone – Economiser	140
Example.....	141
Unitary Single Zone – Economiser + Lockout.....	143
Example.....	144
Unitary Single Zone – Economiser + Heat Recovery.....	145
Example.....	145
Unitary Multizone – Basics.....	147
Example.....	148
VAV Systems.....	150
Cooling.....	151
Heating	151
Outside Air	151
Outside air sizing	152
VAV Terminal Units.....	152
Heating Data.....	153
Cooling Data.....	154
Exercise.....	154
CAV Systems.....	156
Exercise.....	156
Exercise.....	157
Change-Over Mixed Mode.....	159
Introduction	159
Data Entry.....	160
Extreme Weather Conditions	162
19 Computational Fluid Dynamics (CFD).....	164
What is CFD?	164
CFD Calculations and Convergence.....	164
The Iterative Nature of the Procedure.....	165
Creating Model Geometry for CFD	165
Finite Volume Grid and Geometric Modelling Considerations.....	166
Building Levels and CFD Analyses.....	173

CFD Boundary Conditions	173
Boundary Conditions for Internal CFD Analyses.....	173
Default Wall and Window Temperatures.....	173
Zone Surface Boundaries	174
Component Blocks & Assemblies as CFD Boundaries.....	179
Component Blocks.....	179
Assemblies	180
Setting Up a New External CFD Analysis	181
Default Grid Spacing.....	182
Grid Line Merge Tolerance	183
Setting Up a New Internal CFD Analysis	184
Default Grid Spacing.....	184
Grid Line Merge Tolerance	185
Editing the CFD Grid.....	185
Setting Up CFD Cell Monitor Points.....	187
CFD Grid Information.....	189
CFD Calculations.....	190
Calculation Options Data	192
Residuals and Cell Monitor Graphs	194
Convergence Difficulties and Troubleshooting.....	195
Displaying Results	196
Conducting CFD Comfort Calculations	198
CFD Results Manager	199
Example - CFD External Analysis.....	203
Creating the model.....	203
Creating a new CFD external analysis.....	203
Setting up monitor cells.....	204
Running the CFD calculations	205
Displaying CFD results	205
Display 3-D contours.....	206
Example - CFD Internal Analysis Basics.....	207
Creating the model.....	207
Default wall and window boundary temperatures	208
Adding surface supply diffuser and extract grille.....	208
Creating a new CFD internal analysis.....	209
Running the CFD calculations	210
Displaying CFD results	210
Example - CFD Internal Analysis Including Assemblies	211
Creating the radiator assembly	211
Positioning radiators within the block.....	213
Creating a new CFD internal analysis.....	214
Editing the CFD grid.....	215
Setting up monitor cells.....	216
Running the CFD calculations	218
CFD Display options	218
Comfort calculations	218
Change CFD Analysis.....	219
APPENDIX I - Calculated Natural Ventilation.....	220

1 Introduction

The **DesignBuilder** software is an advanced graphical user interface that has been specially developed to run EnergyPlus simulations. For UK and Republic of Ireland users the DesignBuilder interface can also be used to produce Energy Performance Certificates (EPCs) and to show compliance with the Building Regulations.

Overview

This Training Manual will guide you through the basic of using DesignBuilder with a view to promoting understanding of the software. It is intended that you work your way through the material from the beginning to the end sequentially because some of the explanations and worked examples build on models created in earlier examples. It is not intended to be an exhaustive source of information on the software. You should refer to the program help at www.designbuilder.com/helpv1 for more 'encyclopaedic' documentation.

2 DesignBuilder Interface

2.1 Opening screen

The opening screen has three tabs – **Recent Files**, **Component Libraries** and **Template Libraries**.



Recent Files: This tab lists the recently used files. Double-click on one to open it.

Component Libraries: This tab lists the components available. Components represent individual building elements such as constructions, glazing and materials and can be referenced from other components, templates or model data.

Template Libraries: Templates are used as a source of generic data for loading into building models. For example, construction templates provide data on the construction of external walls, roofs, floors, partitions, roofs etc and air-tightness data.

Library **Components** and **Templates** 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 and templates. Any later change made to the Library components or templates do not affect existing building models because they reference their own set of components and templates.

In general, the contents of the templates, for example constructions, loaded into the model must be edited to represent the systems, structures etc for the actual building.

This is best done by first making a copy of the template using the copy tool  in the tool bar, renaming it and then editing the content using the edit tool .

More information is given on **Templates** and **Components** later.

Learning mode: In Learning mode, the right hand panel displays useful information and common task operations relevant to the current screen. It is recommended that new users leave learning mode on until they are familiar with the program at which point it can be turned off by going to Tools > Program options... and deselecting Learning.

Instructions in the training assume that you have **Learning** mode switched on.

Tutorials: This appears in the right hand panel. Click on it to access the tutorial. It is recommended that users view the tutorial before coming on the training course. You must be connected to the Internet to access the tutorials.

Help: Use the toolbar Help to select one of the help options, or press F1 while in the program to bring up relevant help. There is also a search facility in the index.

By default DesignBuilder uses the **WebHelp** format, which requires an Internet connection, but you change to using a **Local** help system in the Program options. You will need an Internet connection to download the local help the first time and thereafter you can use the help without.

3 Create Building Geometry

This chapter explains how to create a building geometry using drawing blocks.

Create a new project

Select File > New Project to add a new project. This opens a new window with two tabs:

Location tab

- Click in the Untitled box, and enter the project title.
- Click on the arrow by the default location box. Select the location from the list that appears.
- Set analysis type to EnergyPlus

Template tab

Shows a list of existing template models that can be used as the basis for the new project you are creating.

Leave default **Blank** for a completely new site, or select one of the template buildings if you wish to base the new project on an existing model (to load an example building for example).

Click OK. The file loads and opens in a split screen. Note the new tool bar.

Add a building

A building can be created with or without reference to a 2-D drawing file created from a CAD system or by scanning a plan of the building. In either case, the building geometry is defined by entering one or more blocks.

Click on the **Add building** toolbar button or click on **Add building** in the right hand Info pane, to add a new building. A screen will appear requiring data to be entered under a number of headings.

Note that for the **Add building** buttons to be available, you must be in the top **Site** level of the Navigator, **Site** option in Navigator on the left hand side of the screen.

Provide the information required under each of the headings on the Building tab.

- **Model type:** select the appropriate model type from the list in the right hand pane – this can be changed later
- **Project details:** The details required may vary from country to country.
- **Assessor details:** Enter details as required, requirements may vary from country to country
- **Occupier details:** enter occupier details here

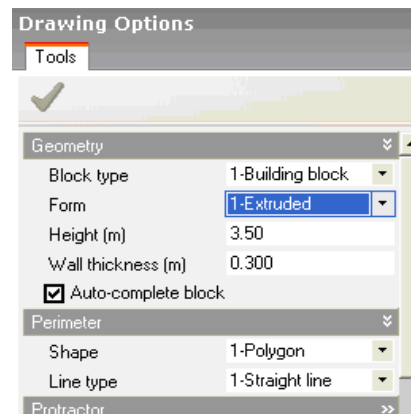
On the Default data tab you can select templates to be used as an initial source of default data to load into the new building. You do not need to make any settings here as you can load data from template later.

Click on the OK button to add the building.

Note that the program automatically activates the Draw Block tool and the **Drawing Option** panel appears in the lower right hand corner of the screen. You can start drawing the first block in the building right away.

4 Drawing Options

Use the Drawing Options panel in the lower left hand corner of the screen to set the block type, form, height and wall thickness. It is important that the wall thickness is set before drawing the block as it is used to calculate the internal dimensions used in the calculations. However wall thickness can be reset after the block has been created by first selecting the block from the navigation pane and then going to model options which will automatically open in the block tab to allow the wall thickness to be changed. We are now ready to add a block using the drawing tool in the middle pane.



There are three **Block types**:

- **Building Blocks** are used to create the model.
- **Outline Blocks** are used to help in the creation of the building and do not affect the model.
- **Component Blocks** are used to create visual and shading structures, ground structures and neighbouring buildings.

See **help** under block type for illustrations.

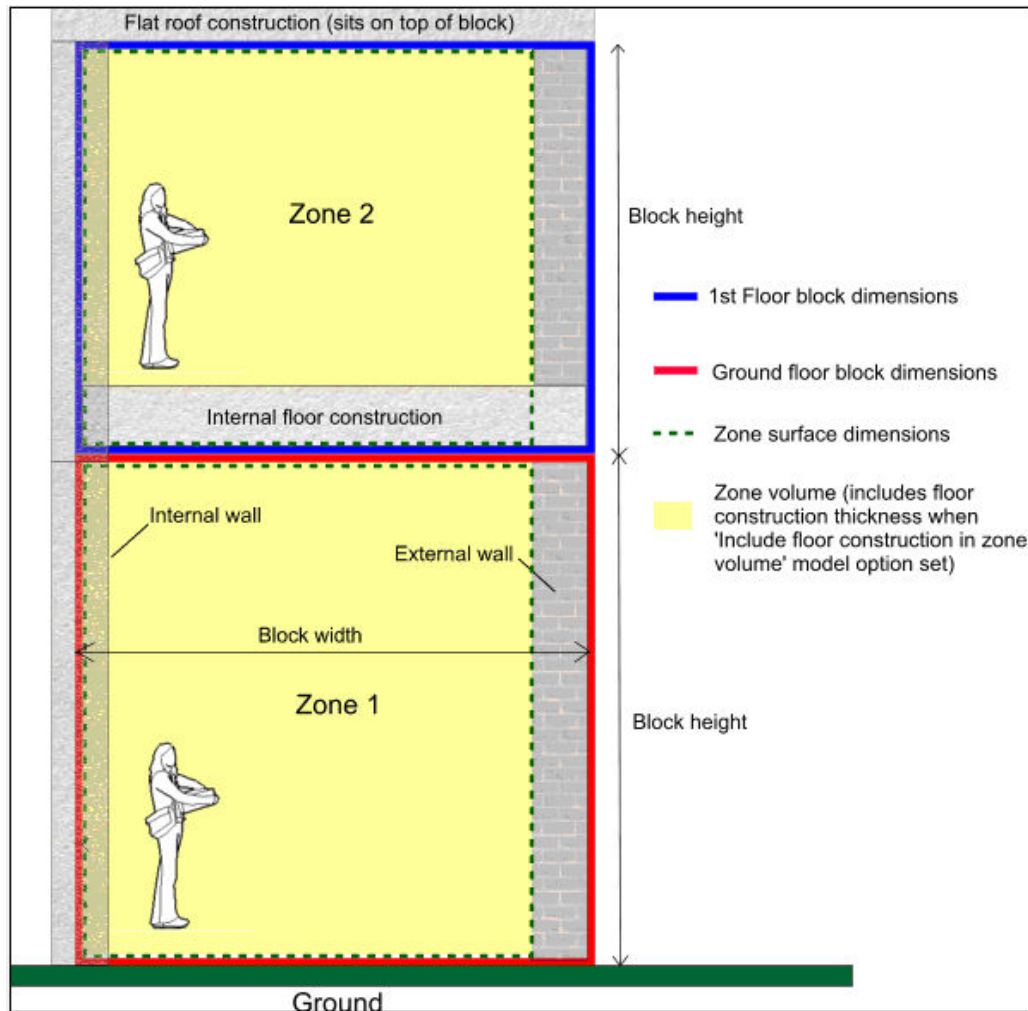
Block form

- **Extruded block form** is probably the most useful. A general extrusion block can be created by drawing a perimeter using a number of pre-defined shapes and then extruding the perimeter by whatever distance is required. Extrusions in both horizontal and vertical planes (or even in sloped planes) can be created using the polygon perimeter shape.
- **Sloping walls block form** is used to create blocks on horizontal planes with uniformly sloping walls. Sloping wall blocks can be created using any perimeter shape (polygon, rectangle or circle).
- **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: when a pitched roof is drawn on top of an existing

block, the overhang or the depth of the roof eaves can be set using the Roof overlap setting.

- **Dome block form** is used to create hemispherical blocks and can only be created using the circular perimeter shape.

Block height – Defining block dimensions



The block height is the same as the floor-to-floor height, also referred to as the zone height. For the top floor, the block height is from the underside of the floor slab to the underside of the ceiling or roof slab. For the ground floor the height should be measured for the ground surface to the underside of the floor slab above as illustrated.

Block widths in DesignBuilder are from an outside surface of an outer wall to the outside surface of another outer wall or to the mid-point of an internal partition wall. DesignBuilder uses the wall thickness to calculate internal dimensions and it is therefore important that the correct wall thickness is specified before creating a block.

Enter the block height in the Height box in the Drawing Options panel. The height of a block can be changed later using the **Stretch** or **Drag Face** tool.

Wall thickness

The wall thickness is used to generate the geometric thickness of the external wall elements of a building block. It is a significant parameter because zones are generated from the internal dimensions of the block which are derived from the external block geometry and the block wall thickness. Therefore, the value of wall thickness used influences the size and shape of zones, especially roof zones (see tip below).

The default setting is automatically generated from the overall thickness of the external wall construction set in the Construction Model data panel but can be overridden by typing in the required value. The constructions that determine the thermal characteristics of the walls can be edited in the Constructions Model data panel without changing the wall thickness used for determining internal dimensions.

The wall thickness is uniform for all surfaces in a particular block, though each block can have a different wall thickness.

The wall thickness does not affect the thermal properties of the wall. Wall thermal properties are based purely on the construction data specified for the wall. The wall thickness is also entered when converting an Outline block to a Building block.

Tip: Enter a low value of wall thickness if:

The block is a roof with a low roof slope angle sloping roof elements. Using a Wall thickness of 0.01m (the lowest allowed) gives **good results** as the roof zone covers most of the block floor area

The block is mainly glazed. A small wall thickness gives rise to a small corner column, allowing the glazing to cover most of the wall.

You wish to work with internal block dimensions.

See examples in **help** under **Wall thickness**.

Perimeter

Blocks are generally created by drawing a perimeter and extruding it to the required height or length.

- **Polygon** allows multi-sided blocks to be drawn and is the most useful general form.
- **Rectangle** allows rectangle to be drawn with a click and drag action.
- **Circle** allows circle to be drawn with a click and drag action.

Line type

There are two line types:

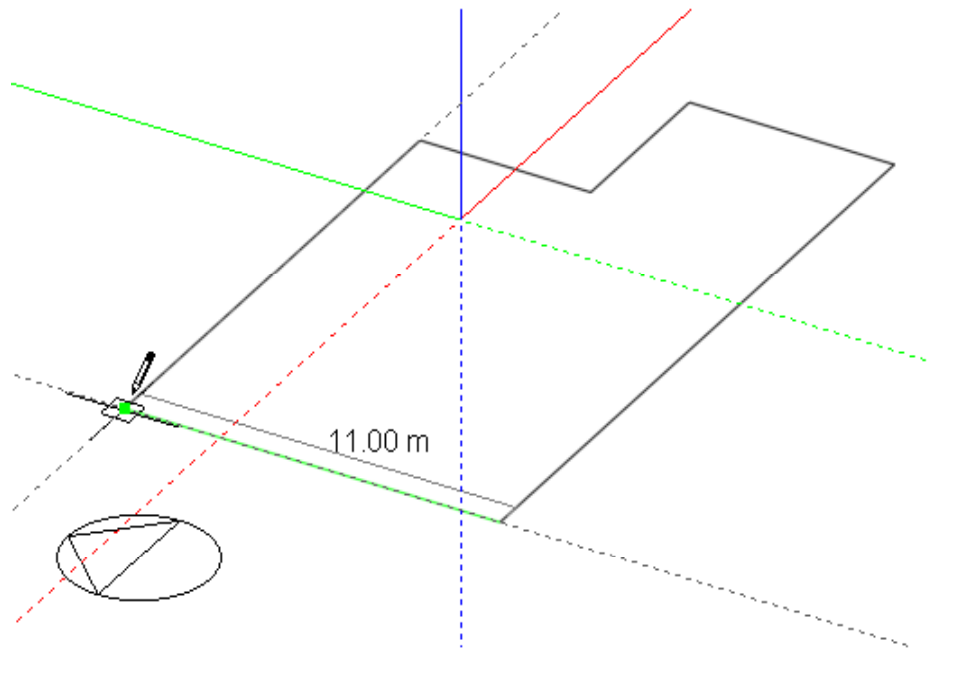
- **Straight line** – default straight line
- **Arc** – can specify sweep angle and number of segments to approximate an arc with a series of straight lines.

Using the drawing tools



To add or create a new block, click on the Add new block tool. The **Add new block** tool is automatically activated, if a new building has just been created. A block is most easily 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-to-floor height. Once the perimeter is completed by snapping on to the start point, (a green square appears as shown below) the block is automatically generated. The block construction, glazing and HVAC will be those set in the default templates based on the age and type of building.



The building model is built up by adding blocks. These can be added using the drawing tool as above but there are other tools which can be used to speed up the process.

Direction snaps

In the Drawing Options panel, open the dialog box for Direction Snaps.



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 changes colour to that of the axis to which it is parallel and stick slightly.

See help on **axis snap** for a full description of all the features.

Point snaps

In the Drawing Options panel, open the dialog box for Point Snaps.



End point snap snaps 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.

Mid-point snap snaps lines to the mid-point of an existing edge. A light blue indicator is displayed when the mouse cursor is within the snap range of an end-point.

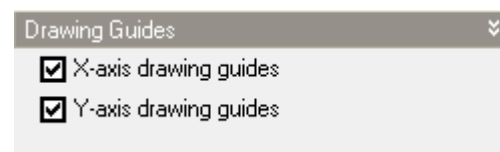
Edge snap snaps to any point on the edge of an existing object. The edge snap can be used in conjunction with the increment snap to enable a point to be located along an edge by moving it in defined increments. A red indicator is displayed when the mouse cursor is within the snap range of an edge and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

DXF snap only applies when **DXF snap** has been imported from an external file. The DXF snap snaps to the end-points of DXF lines.

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 moves a point in a particular direction by defined increments. You can think of the increment snap as acting in the same way as a conventional CAD drawing grid. 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.

Drawing guides

In the Drawing Options panel, open the dialog box for Drawing Guides.



The X- and Y-axes drawing guides are best left selected as they help to complete rectangular blocks by showing a construction line parallel to an axis to place the final wall so that it connects with the starting point.

Cloning tool



- Go to the building level in the navigation menu and then select the block to be cloned by clicking on it – it turns pink.

- Select the clone tool.
The Drawing options menu appears.
- Check that the point snaps are turned on.
- Hover the cross hairs over a convenient corner at the bottom of the selected block until the green snap indicator appears, then click.
- Move the cross hair to the top corner and hover until green snap appears, then click.
A clone of the selected block is added on top of the selected block.

In this way a multi storey building can be quickly built up, but note that if storeys have similar internal partitions it would save time to insert the partitions in the lower storey.

The Clone tool can also be used to copy other objects such as internal partitions, courtyards and surface openings.

See help under **move**.

Stretch tool



Building, outline and component blocks can be stretched (or scaled) at the building level. To stretch a block:

Select the block.

- Click on the Stretch tool or select Stretch from the right-click menu.
Only one block can be stretched at a time. When the Stretch tool is activated, a wire frame is displayed around the extent of the block and as the cursor is moved across the faces of this frame, each face is highlighted in the colour of the axis to which the face is normal.
- Click on one of these highlighted faces, to stretch the block along the axis normal to the face. If the increment snap is switched on, the block can be stretched in increments. This is useful for re-setting the height of a block that has been cloned, and for stretching a circular block into an elliptical one.

See help under **stretch**.

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:

- Select the block to modify.
- set increment snap to 0.1m
- Click on the Drag face tool.

- Move the mouse cursor to the surface to be moved.
As the cursor is moved across the block surfaces, each surface is highlighted 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 and click again or key-in the distance to shift the surface by the required amount.

See help under **drag face**.

Cut block along plane



Blocks can be cut using the Cutting Tool. Blocks are divided using a plane which is defined using one of two methods:

- Draw plane – the plane is defined by clicking on 3 points.
- Select plane – the plane is defined by selecting an existing plane in the model.

See **help** under **cut block** for comprehensive instructions.

This tool is particularly useful for creating complex roof geometries where different roof shapes intersect. As the geometry is complex it is recommended that **outline blocks** are used with **polygon** selected as the perimeter. The **outline block** can be converted to a **building block** when the cutting process is complete. In some circumstances, resulting blocks may not be able to be converted to building blocks due to geometrical limitations. In this case, the block is displayed in a sepia colour to indicate that it cannot be converted.

See also **Using outline blocks** below.

Draw void perimeter (courtyard tool)



To create an external courtyard within a block:

- Go to the required block by clicking on the navigator entry or double-clicking the block in the Edit screen.
- Click on the Draw void perimeter tool.

The perimeter shape and required wall thickness are set in the Drawing Options data panel.

Adding roof block

The four different forms of blocks are listed in earlier. For other than flat roofs, it may be convenient to use either the sloping wall block form or the pitched roof block form to add a roof block. In each case the roof is automatically generated from a perimeter drawn on a horizontal plane. The roof space may be occupied and therefore assume to be conditioned or it may be unoccupied and therefore assume to be unconditioned.

When the pitched roof form is selected in the drawing option, there is an option to tick the box for 'Roofspace occupied' or to leave this blank for an unoccupied space.

Note that this option is only used at the point the block is first created. To change the nature (heated/unheated) of an existing block use the Zone type data on the Activity tab of the block.

If the sloping wall block form is selected it will be necessary to open the activity tab and set the zone type to 'Standard' for an occupied roof space or to 'Semi exterior unconditioned' for an unoccupied roof space.

Tip: In some cases when drawing roof blocks it is best to set wall thickness to 0.01m.

Using outline blocks

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.

Because outline blocks are not part of the model they do not appear in the Navigator panel and in visualisations.

Outline blocks are very useful in the creation of more complicated models especially complex roof geometries. 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. You can remove all outline blocks from a model by selecting Remove all outlines from the **Edit** menu.

For detailed guidance on how to use outline block to create more complex geometries see **help** under Outline blocks.

Using component blocks

There are 3 types of component block:

- **Standard** - used for shading, reflection and visualisation.
- **Ground** - used for setting ground adjacency, shading, reflection and visualisation. Ground component blocks can be used as a convenient way to set the adjacency of any building block surfaces as being adjacent to ground. Particularly useful where part of a building may be below ground level
- **Adiabatic** - used for setting adiabatic adjacency, shading, reflection and visualisation. Adiabatic component blocks can be used as a convenient way to set the adjacency of any building block surfaces to 'adiabatic'. Adiabatic surfaces do not transfer heat beyond their outer surface and are often used for modeling boundaries with other spaces that can be considered to be at roughly the same temperature as the zone. For example a party wall between the building being modeled and an adjoining occupied building.

For more details on component blocks see **help** under Component blocks.

Construction lines



Construction lines have numerous applications in forming temporary attachment points during the creation of precise geometry. 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 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.

See **help** under Drawing Construction Lines for more details

Measure length and angle



Select the measure tool and set the quantity to be measured in the drawing options box. The measure tool allows you measure distance, angles and areas on single planes in the model:

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

The requested data is displayed highlighted in the Drawing Options panel on the left of the screen.

Import 2-D Drawing file

DesignBuilder allows the building geometry to be loaded from 2-D drawing files as DXF files created by CAD systems or scanned images from any other source of building plans. Most common formats are supported. All that is required is the building outline and internal partitions. It is therefore recommended that where possible DXF files are simplified in the CAD environment by turning off unwanted layers before exporting the DXF file. Block heights are also required so the Energy Assessor may find it useful to mark these on the CAD drawing or a print of the 2-D drawing file to refer to when drawing the building blocks.

- Do not click on add building.
- Click on import 2-D drawing file and select file type from drop down menu
- Find the file in the Design Builder Data folder using the browse button ...
It is recommended that 2-D drawing files are saved in this folder as this is the default folder accessed by DesignBuilder, but the browser can be used to load files from any folder.
- For DXF files the units need to be set independently. For scanned images see scaling instructions below
- Click on Next.

- Turn off unwanted layers in DXF file – be sure to keep all walls – unwanted details can be removed later.
- Click on Finish.

It will generally be necessary to align scanned images with the XY axes. Select **Edit > Set axis for imported Bitmap**. In plan view, click on both ends of a line that should be parallel to the X axis. The plan is then automatically orientated to be parallel to the X-axis.

Now select **Edit > Scale bitmap** floor plan. Click on both end of a major dimension in the plan. A blue line appears with a dimension. If the known dimension is different simply type in the known dimension and press enter. The plan is automatically scaled.

Now click Add building or Add block

To add a block trace around the 2-D outline:

- Use the central wheel on the mouse for zoom and orbit
- Use pan control to move around building
- Leave point snaps and direction snaps for DXF files on initially. There are no snap points for scanned images

When the perimeter is complete, the block is generated automatically. If there are small external details on the DXF plan which are not significant, turn off the DXF snap in order to be able to trace across them.

Tip: It can be easier to switch off DXF snap, especially when drawing partitions because it ensures that the partition lines are snapping to the block perimeter and not just to a DXF line close to it.

It is advisable to use the measure tool to check that the dimensions have been properly defined. For example, to check that the measurement are in the same units as on the original drawing.

5 Model Options

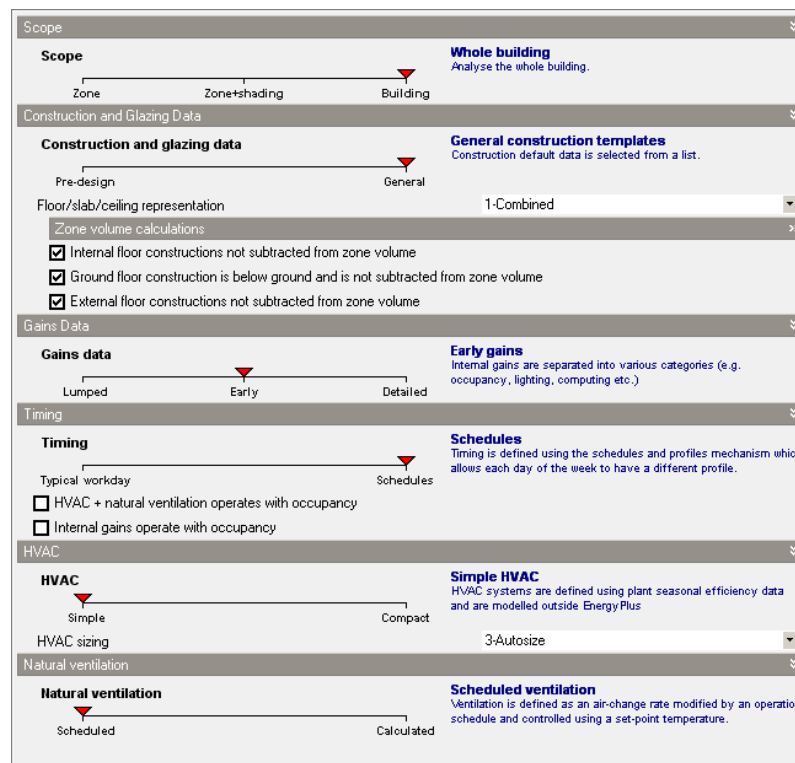
DesignBuilder models can be configured in many different ways and all of the settings related to the level of detail plus many calculation, display and drawing tool options can be found and changed on the Model options dialog. The model options, including the project details referred to above, can be reviewed and edited at any time by clicking on the model option icon (the spanner).

Click on the spanner icon to view the Building model options.



The data tab

Allows the amount of detail in various aspect of the model to be set using a series of slider controls – see below.



Scope

Scope: Zone, Zone+shading, Building (selected). **Whole building**
Analyse the whole building.

Construction and Glazing Data

Construction and glazing data: Pre-design, General (selected). **General construction templates**
Construction default data is selected from a list.

Floor/slab/ceiling representation: 1-Combined

Zone volume calculations:

- ☒ Internal floor constructions not subtracted from zone volume
- ☒ Ground floor construction is below ground and is not subtracted from zone volume
- ☒ External floor constructions not subtracted from zone volume

Gains Data

Gains data: Lumped, Early (selected), Detailed. **Early gains**
Internal gains are separated into various categories (e.g. occupancy, lighting, computing etc.)

Timing

Timing: Typical workday, Schedules (selected). **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 (selected), Compact. **Simple HVAC**
HVAC systems are defined using plant seasonal efficiency data and are modelled outside Energy Plus

HVAC sizing: 3-Autosize

Natural ventilation

Natural ventilation: Scheduled (selected), Calculated. **Scheduled ventilation**
Ventilation is defined as an air-change rate modified by an operation schedule and controlled using a set-point temperature.

Scope control can be used to switch between 'whole building' and 'single zone' modes.

The Construction and glazing model option controls the way construction templates are loaded on the construction model data tab. When the option is set to **Pre-design** you can load construction templates using the Insulation and Thermal mass slider controls. When set to **General**, templates are loaded by selection from a list in the usual way.

Gains options:

- **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.
- **Early** - gains can be defined separately under various categories (see above).
- **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.

Timing options

- **Typical workday** - 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.
- **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.

HVAC options

- **Simple** - the heating/cooling system is modelled using basic loads calculation algorithms. For convective heating systems and all cooling systems the EnergyPlus 'Purchased air' system is used to calculate heating and cooling loads. 'Purchased air' supplies hot/cold air at predefined temperature and humidity for heating and cooling. 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. When using Radiant/convective the heating system type the EnergyPlus 'High Temp Radiant System' system type is used to calculate heating loads.
- **Compact** - the heating/cooling systems are defined in DesignBuilder using Compact HVAC descriptions but modelled in detail in EnergyPlus.

In both cases heat generator (boiler) and chiller fuel energy consumption is calculated from loads as a post-process using efficiency factors.

Natural ventilation

- **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.

You should generally use the **Scheduled** natural ventilation model option because it is quicker and easier to enter the related model data and also because the simulations run more quickly. However if you need to calculate natural ventilation rates which can be achieved with a particular configuration then you should use the 'Calculated' option.

Advanced tab - Lumping and Merging

Simulations can be simplified and speeded up (sometimes quite significantly) by lumping zones, walls and windows. The following options are available:

- Merge zones of same activity
- Merge zones connected by holes
- Lump similar windows on surface
- Lump similar cracks
- Lump walls and windows (developer option)

See DesignBuilder help function for more details on these and the other options on this tab.

Heating Design tab

Key parameters set here are:

Temperature control

- Air temperature
- Operational temperature
- Other

See also Simulation tab.

Design Margin

The Heating Design Safety factor is used to multiply calculated steady state heating loads in each zone to give a recommended heating plant size. It accounts for the additional heat required to bring the building up to temperature in a reasonably short preheat period and allows comfort conditions to be maintained in all but the most extreme winter conditions.

Cooling Design tab

The data on this tab can also be accessed on the Cooling design calculation options dialog.

Summer design day is set up by selecting **day**, **month** and **day of week**

Calculation options

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 room mean air temperature to the heating and cooling setpoint temperatures specified on the Activity tab.
- **2-Operative temperature** - control the room temperature using 0.5 radiant fraction. *This is an advanced option - see notes of operative temperature control under help if you plan to use this option.*
- **3-Other**- you can enter the radiant temperature control fraction.

System Sizing

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 required to cool the building down in a reasonably short precool period and allows you to be confident that comfort conditions will be maintained in all but the most extreme summer conditions (risk depends on the Summer Design Weather Data options selected at site level)

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.
- **3-VAV** - detailed EnergyPlus simulation using VAV equipment.
- **4-Fan coil** - detailed EnergyPlus simulation using Fan coil equipment.

Tip: You should use the default ASHRAE option unless you have experience with Compact HVAC and understand the configuration of these system types.

Solar options

Solar Options allow you to control aspects of the model related to solar gains.

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. Check box **Include all buildings in shading calcs** to include other buildings on the site as shading obstructions.

Model reflections

If using reflections, the program calculates beam and sky solar radiation that is reflected from exterior surfaces and then strikes the building. These reflecting surfaces fall into 3 categories described in the help function.

For Advanced Calculation options and Output options see help

Simulation tab

Simulation Period

Select the start and end days for the simulation, or 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).

Calculation options

Time steps per hour: though many buildings can be successfully simulated with 1 or 2 time steps per hour, EnergyPlus suggest 4 for non-HVAC simulations and 6 for simulations with HVAC.

Temperature control: select from

- **Air temperature** - control the room mean air temperature to the heating and cooling setpoint temperatures specified on the Activity tab.
- **Operative temperature** - control the room temperature using 0.5 radiant fraction. See notes of operative temperature control below if you plan to use this advanced option.
- **Other-** you can enter the radiant temperature control fraction.

For **Advanced Calculation** options and **controlling output data generated** from simulations and **Compact HVAC autosizing** see help.

Display tab

Check boxes to control what is displayed on screen and in visualisation.

Drawing tools tab

Check boxes to turn on direction snaps, point snaps, drawing guides, set rotation increment and protractor as required

Block tab

This tab is only available at block level and is used to change the wall thickness or change roof-wall transition angle if different from default of 70 degrees.

Project details tab

Use this tab to edit project details.

6 Introduction to Model Data

This section takes an introductory look at the data in the Activity, Construction, Openings, Lighting and HVAC tabs. The data on these Model data tabs is further explored in subsequent sections.



Activities

This section explains how to partition the building into zones according to the activities (usage) in the building such as office, meeting room, toilet or plant room. Each activity has the following categories of data associated with it:

- Zone Type (block and zone level only)
- Occupancy
- Metabolic rates
- Holidays (building level only)
- Domestic Hot Water (DHW)
- Environmental control
- Computer gains
- Office equipment gains
- Miscellaneous gains
- Catering gains
- Process gains

For convenience this data is usually loaded from a predefined activity template. DesignBuilder contains a large number of activity templates from the UK NCM and ASHRAE. Some other libraries relevant to different parts of the world are also available from local agents. Also users can add their own templates to the library if necessary.

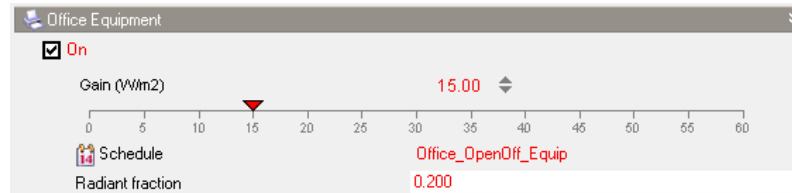
Most of the **Activity** data is associated with either **Internal gains** or **Environmental control**.

Internal gains

There are 7 categories of internal gains in DesignBuilder:

- Occupancy
- Computer gains
- Office equipment gains
- Miscellaneous gains
- Catering gains
- Process gains
- Lighting (data set on the Lighting tab)

You can define the time varying levels of sensible gains and for Occupancy, Catering and Process also latent gains. This is done by setting a maximum gain and applying a schedule to define the way the gains vary with time. You can also set the radiant fraction, i.e. the fraction of the heat emitted that is radiant heat.



The screenshot above shows the data for Office equipment. Here the maximum office equipment gain is set to 15 W/m² using a slider and the schedule is set a standard 'open plan office equipment operation' schedule as specified by the UK NCM. You can change the schedule by making a selection from a list of predefined schedules. You can also add your own schedules and this process is described later.

Environmental control

User comfort requirements associated with the activity can be set:

- Heating setpoint and setback temperatures
- Cooling setpoint and setback temperatures
- Natural ventilation setpoint temperatures
- Mechanical ventilation setpoint temperatures
- Minimum fresh air requirements
- Minimum lighting levels required

This data is explored further in subsequent examples.

The NCM zoning process

The object of the zoning process is to divide the building up into areas of similar activity uniquely defined by activity type plus the HVAC and lighting services, using as few zones as possible.


- Divide the floor into separate physical areas, bounded by physical boundaries such as structural walls or other permanent internal walls (partitions).
- If any part of an Area is serviced by a significantly different HVAC or lighting system, create a separate Area bounded by the extent of those services. This may require the use of virtual partitions (see below) to delineate area that have different access to daylight or where a large open space has perimeter heating with a different heating/cooling system in the central zone.
- Attribute just one Activity (defined by the dataset above) to each resulting Area. If the building is speculative and the activity not fully defined, select the appropriate Speculative Activity for the relevant building type i.e. the activity it is expected to be used for.
- Combine any contiguous Areas which are served by the same HVAC and lighting systems, and which have the same Activity within them, (unless there

is good reason not to). For example a number of cellular offices may be treated as one zone (see also hanging partitions and thermal Mass below).

- Small unconditioned spaces like store cupboards, riser ducts, etc., can be absorbed into the adjacent conditioned spaces. Their floor area is added to that of the adjacent conditioned space. Larger areas should be treated as indirectly conditioned spaces. See Defining indirectly conditioned spaces.
- If any resulting Zones overlap, use discretion to allocate the overlap to one of the Zones.

Dividing the building into zones by drawing partitions

After all blocks have been added and edited, blocks should be divided up according to the activities and zoning rules.

- Select  and then position the pencil on the drawing screen over a wall until the red snap indicator appears (if snap indicator does not appear check snap options)
- Click and draw in the partition in the required position.
- Click where that internal wall ends and continue drawing further internal walls until the partition connects to another wall, when the model automatically updates by adding the partition.

Alternatively trace over DXF internal walls.

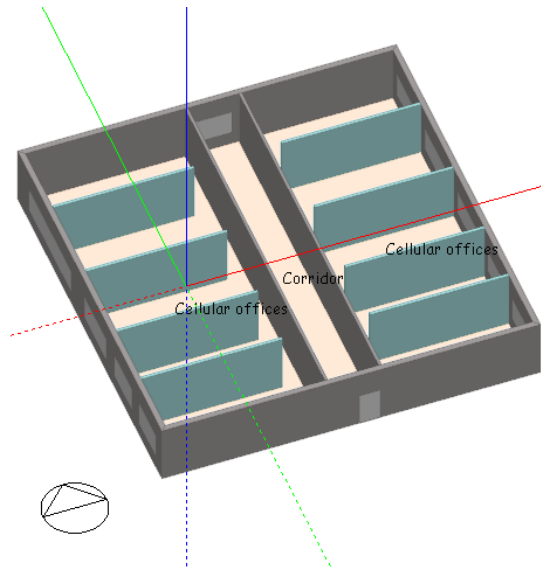
It is best not to create unnecessary zones – see zoning rules. Where a number of zones have the same activity, HVAC, lighting and access to daylight, they can be regarded as one zone as illustrated below for modular offices. However, if the internal partitions are not added the internal thermal mass of the building will be under represented. This is unlikely to have a significant effect on the calculations, but DesignBuilder offers two ways of defining additional internal thermal mass:

- Drawing hanging partitions (i.e. partitions which do not meet other walls at both ends and hence do not create zones), or
- Entering a non-zero internal thermal mass exposed area on the Construction tab under the Internal Thermal Mass header at zone level.

The thermal mass defined using these methods is lumped together for each zone and modelled in a non-geometric way. In both cases, the construction is defined under the **Internal Thermal Mass** header on the Construction tab.

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 blue in the illustration below. These hanging partitions do not need to be accurately positioned, as it does not matter where the thermal mass is added in the zone; it is only the total mass that matters and this is automatically calculated from the area and construction of the hanging partitions. Option 2 requires the user to enter the exposed area of the internal thermal mass. The area should be left as zero if the hanging partition option is used.

See **Help** under **Internal Thermal Mass**.



Virtual partitions allow zones to be subdivided without putting a physical wall into the space. Such partitions can be used to divide a space where there are different services such as lighting or HVAC within an open space to create activity zones with different services but which are not separated by a physical boundary. For example to delineate area that have different access to daylight or where a large open space has perimeter heating with a different heating/cooling system in the central zone.

Daylighting modelling - offsetting electric light with daylight

DesignBuilder uses the EnergyPlus detailed daylight method to model the control of electric lighting according to daylight illumination levels.

Placing Light Sensors

In the Lighting tab, when zone Lighting Control is 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:

The lighting control types available are:

- Linear
- Linear/off
- Stepped

These tools allow the benefits of daylight controls in offsetting electric light where daylight is available to be modelled. **Virtual partitions** can be used to create separate zones where lighting in areas adjacent to windows can be switched independently of lighting in the central zone

Zoning large buildings or blocks

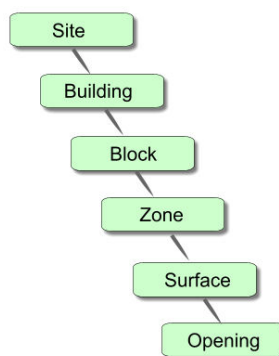
By default, DesignBuilder automatically generates zones as partitions are drawn. This can be time consuming in large blocks with many zones. To uncheck **Automatic Block Zoning**:

- Go to Model Option > Advanced > Miscellaneous Edit and uncheck the box. Partitions are then shown in blue and referred to as **outline partitions**. Outline partitions do not create new zones and are treated in the same way as **hanging partitions**.
- After drawing all the partition for the block, convert outline **partitions** to standard partitions and rezone the block using the **Rezone block** command accessed from the Tools menu.
- Outline partitions that do not connect at both ends, **hanging partitions**, remain in blue even after the Rezone block command.

Alternatively, use the **Rebuild** command from the tools menu, to convert all outline partitions in the building to standard partitions and hence rezone the whole building. This takes longer to process than the **Rezone block** command as it recalculates the surface and adjacency geometry as well as rezoning.

Data inheritance at block, zone and surface level

Once all the blocks and zones making up the building model have been defined, it is necessary to assign Activities, Constructions, HVAC, Lighting and HVAC to each. DesignBuilder uses inheritance rules to make this process simple and efficient. DesignBuilder models are organised in a 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 becomes 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 selecting a block and making a user defined selection (say 'Wall 2') to 'hard set' the data at this level. All the zones in this block will then have external wall construction set to 'Wall 2' as the default.

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 inherited 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).

A further significance of making a user defined selection at a particular level is that the data remains 'hard set' at that level and below even when changes are made to the level above.

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. 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 (<F2> key is useful for this) until the data turns red. This then is the level from which the data has been inherited.

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

Zone type

The zone type can be set at block or zone level on the Activity tab and can be one of:

- **Standard** - considered to be occupied and heated or cooled.
- **Semi-exterior unconditioned** - is an unoccupied zone in the building which is neither heated or cooled. Examples of 'semi-exterior unconditioned' spaces are roofspaces, sunspaces, car parks etc).
- **Cavity** - the zone is a cavity such as the glazed cavity within a double facade or a Trombe wall. This algorithm correctly calculates the convection coefficients for a narrow sealed vertical cavity based on the ISO 15099 standard.
- **Plenum** - the zone is a plenum which is unoccupied and has no heating, cooling, or mechanical ventilation (though of course air will flow through it to meet the needs of the zones it serves).

Assigning activities

When the building model is complete and all the blocks have been zoned, the next step is to assign activities to each zone. It is best to be methodical, especially in large buildings, and to start at the building level by setting the activity which most commonly occurs in the building, for example, classroom in a school or bedroom in a hotel. This sets this activity for all zones. Now work methodically through all the zones changing the activity where it is different from the default. You can also use the block level if most of the zones in a block have the same activity (but different from building default).

- Select a zone from the navigation menu.
- Click on the Activity tab – the screen shows the default activity.
- Click once on Activity – a menu of all activities and sectors appears in the right hand pane.

- Click + to expand the appropriate sector and double-click on the activity. User defined activities can be added to the library of activities using the + or copy functions.

Note that the sector is automatically assigned when an activity for the zone is selected.

- The default zone type, Standard, applies to all zones in the model both conditioned and unheated zones. In most instances spaces with no heating system are assumed to be heated by the HVAC system for the surrounding zones. Such zones are therefore included in the model as heated i.e. standard zones with an HVAC system assigned to them. Where an unheated zone is to be included in the model select “none” from generic list of HVAC templates
- For unheated zones, especially unheated roofspaces, set the zone type to **Semi exterior unconditioned**. See also section below on constructions.

Each zone should be given a meaningful name to make it easy to identify during the analysis. This is important for larger buildings where there may be a large number of zones. To rename a zone:

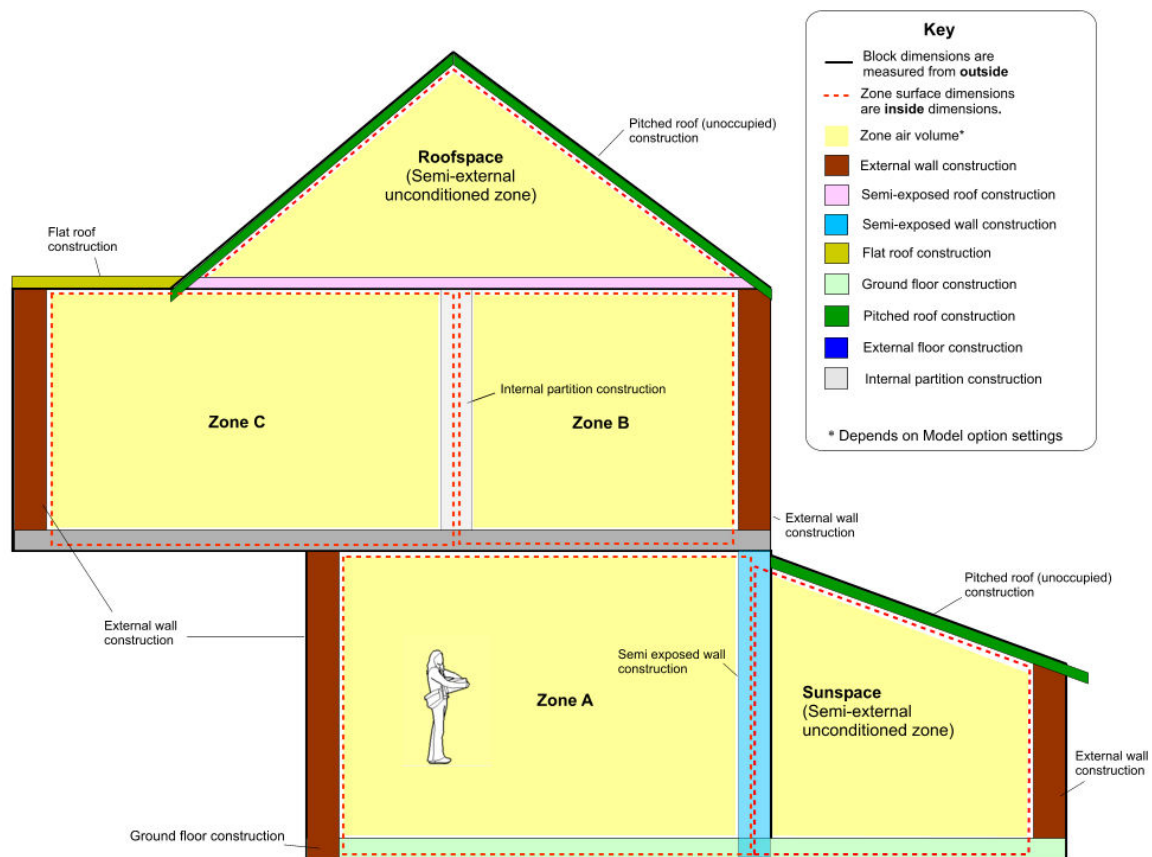
- Click on the zone entry in the navigation list on left of the screen, to highlight the zone.
- Click again to open an edit box and type in a meaningful name such as ‘Office South’. The process is the same as renaming a folder in Windows Explorer.

Constructions

Defining constructions in DesignBuilder

The following definitions apply and are illustrated in the diagram below.

Floor/slab/ceiling constructions



External walls

External wall construction defines the construction of walls adjacent to outside. The definition of walls is given by the roof-wall transition angle of 70 degrees.

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.

Pitched roof (occupied)

The Pitched roof (occupied) construction applies to external sloped surfaces when they appear in occupied zones.

Pitched roof (unoccupied)

The Pitched roof (unoccupied) construction applies to external sloped surfaces when they appear in semi-exterior unconditioned (unheated) zones.

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 (see below).

Semi-exposed walls

The semi-exposed wall construction applies to walls between Conditioned zones and Semi-exterior unconditioned zones. They 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) is the one adjacent to the Semi-exterior unconditioned zone.

Semi-exposed ceiling

The semi-exposed ceiling construction is used between Occupied zones and Semi-exterior unconditioned (unheated) zones (such as roof spaces) above. The construction should include any slabs, insulation and suspended ceiling layers. DesignBuilder determines which of the zones is relatively outside and uses this information to correctly order the material layers in the roof. See discussion under 'Pitched roofs'.

Semi-exposed floor

The semi-exposed floor construction is used between Occupied zones and Semi-exterior unconditioned (unheated) spaces below. The construction should include any slabs, insulation and raised floor layers.

Ground floor

The ground floor construction is used between all internal zones and the ground.

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.

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 does not usually contain insulation. Where the zone above is unoccupied, the Semi-exposed roof construction is used instead. Likewise, where the zone below is unoccupied the Semi-exposed floor construction is used instead.

Adjacency

Adjacency means what is on the other side of a surface. By default DesignBuilder calculates adjacencies between blocks, partitions etc. This default adjacency mechanism can be overridden at surface level using the Adjacency setting on the Construction tab. The default setting for Adjacency is **auto**, but it can be set at surface level to be one of:

- **1-Auto** - the adjacency of the surface is determined automatically by DesignBuilder based on its position. External surfaces below the ground plane are, by default, 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 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 that are between two zones at substantially 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 that 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.

Entering construction details in DesignBuilder

The model is initially created using the default construction data loaded from the project template. When the construction tab is first opened all construction types are labelled as **Project...** and displayed as shown below whether or not they are present in the building model. For example the Pitched roof construction is displayed whether or not there is a pitched roof in the model.



Buildings Constructions

For most buildings the quickest way of setting up constructions is to define them at building level. Go to building level and then open the Construction tab. Double-clicking on any of the constructions such as Wall, Flat roof etc opens a **Construction Edit screen**.

Set the number of layers first, and then select the material and thickness for each layer. Constructions can have up to 6 layers. 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-exposed unconditioned zone). Clicking on material in any layer opens a menu of materials in the right hand pane. Double click on the required material and specify its thickness.

Bridging can be added 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: If the material referenced is defined as a simple R-value then the construction thickness does not have to be entered.



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.

Image tab: This displays an image of the construction layers which is useful to check that the construction has been set up as required.

Calculated tab: This tab shows calculated values and the effect of bridging when present. Note that the U-values etc are not used in simulations but are useful for reference and for use in simplified calculations such as SBEM.

More Complex Construction Mixes

For more complex construction mixes, such as an old building with a number of later extensions of different constructions, it is useful to create a number of wall constructions for example, one for each construction type found. These constructions can then be set in the appropriate blocks and zones.

- Open the Construction tab. All construction types will be displayed showing all possible constructions (external walls, Flat roof, Pitched roof (unoccupied) etc) whether or not they are present in the model.
- Single click on one of the constructions, external walls for example – a menu of wall constructions appears on the right.
- Select a suitable wall construction type, preferably one that will have a similar thermal mass to the actual construction and make a copy using the Create copy icon in the toolbar above .
- To edit select this new copy and then the edit tool. Alternatively use the  function to create your own construction
- An edit screen appears labelled **Model data**. Give it a new meaningful name such as “External wall main block”
- Under ‘Source’ record where the new data has come from as part of the audit trail required. It may be useful to include a brief description and possibly a date or revision number as constructions may change as the design evolves.
- If necessary adjust the number of layers and then edit the materials and thickness data for each layer. Include bridging if appropriate
- Click OK and new construction will be added to list of constructions
- Now click on each of the other constructions present in the model and change the values as required.
- Finally apply the new constructions created to each block and zone as appropriate. First select the block or zone and then in the construction tab select the type of structure i.e. wall roof etc to open the menu of structures. Select the structure required from the list that has been created by double clicking on it to load it into the model.

Sub-surfaces are useful where part of the structure is different from the rest. For example, glazing may have been replaced by insulating panels having a lower U-value

than the original wall construction. **External doors** are also defined at the sub-surface level. These can be drawn in at surface level. See **help** under **Drawing windows**.

Pitched roofs

Heated roof spaces are considered to be occupied and are therefore treated as normal zones. Note that when creating the block for a heated roof space first go to **Drawing options, Geometry** and check box **Roof space occupied**.

Model infiltration

At the bottom of the construction screen is the data for setting model infiltration. Open the infiltration data by clicking on the grey Airtightness header tool bar for setting model infiltration.

If **scheduled** was selected in model options under natural ventilation, the ventilation is defined as an air change rate per hour, modified by an occupation schedule and controlled using a set point temperature.

If **calculated** is selected as the ventilation option, together with compact HVAC, natural ventilation and infiltration air flow rates are calculated based on openings and crack sizes buoyancy and wind pressures. When calculated natural ventilation is active, the Airtightness is defined by a position on a five point scale corresponding to 5 crack templates



Openings

The term opening is used in DesignBuilder to describe any opening in the main building fabric. There are five types of opening:

- Windows
- Doors
- Vents
- Holes
- 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.

In any model, go to building level. Then select the **Openings** tab to bring up a screen headed **Project glazing template** which shows '**Glazing type**' and **Layout** which are initially set as the Project default To introduce alternative values:

- Single click on **Glazing type** to bring up a library of glazing types in the right hand pane (or alternatively click and use the browse button ...). Double click on the required glazing type to load it into the model. To introduce user defined glazing types use the Add new or copy and edit tools to create a new glazing type and then load it into the model.

- Single click on **Layout** to open a library of facade types. Select a façade type that will produce the right appearance for the model. If required the copy or add and edit tools can be used to create user defined facades.
- Open the **Dimensions** header and set the slider to define the Window to wall %. The opening area includes frames. The model will automatically update with the new value. If the changes are made at building level this % is applied to all facades, provided data has not been previously 'hard set' at the lower level. If made at a wall surface level the set % applies to that wall only – see section on Inheritance.
- Open the **Shading** toolbar and select widow and/or local shading and make appropriate selections for the drop down menus
- Go through the same procedure for **Roof glazing** if present
- Open the **Doors** toolbar. Checking the auto generate box in a surface will automatically generate a door. Set up the operation and schedule for the doors.
- Open the **Vents** toolbar. Select the vent type and operation schedule
- The values set in the previous steps will be applied to all surfaces if they were entered at building level. Where a surface requires different data you can select that surface from the navigation menu and repeat the steps as necessary to set the alternative data.

You can use the **copy** and **edit** tools to create more than one Glazing template, which may be useful for example where Southerly facing facades have shading systems which are not present on the Northerly facing facades.

Lighting

Opening the lighting tab brings up the default lighting system from the reference template. To introduce alternative lighting systems start by entering data for the most common system at building level by selecting an appropriate **lighting template**. These templates are data sets that can be used as a source of generic lighting systems for loading into the building data. A single click on 'template' will bring up the library of lighting templates. Select the most appropriate, review and if necessary edit the data under the 3 main headers:

- General
- Task and Display Lighting
- Control

Note that the lighting gain in the default **Reference Lighting** template is **5 W/m² per 100 lux**. For an office activity having **500 lux** target illuminance this results in **25 W/m²** lighting gain, which will be much too high for most modern buildings being designed now.

When you load lighting data from template, the actual lighting internal gain is calculated by multiplying the W/m²/100 lux lighting efficacy data in the template by the required illuminance level as set on the Activity tab under Environmental

Conditions. Consequently you should make sure the correct activity data has been loaded in each zone before loading lighting templates.

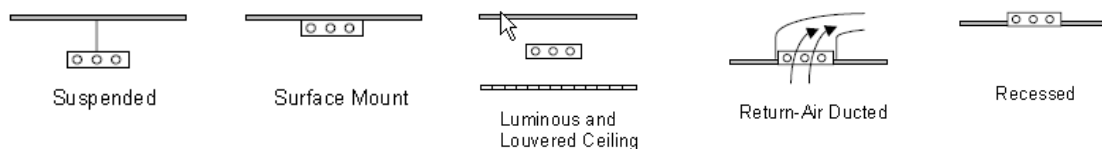
It can be useful to explicitly load data in each zone by checking each zone in the Target tab of the **Load data from template** dialog. This ensures that the correct lighting load for each zone is set. If the activity for one zone is different from that at building level the wrong lighting energy will be calculated for this zone if loading the lighting template at building level only.

Lighting data generates simulation data at the Zone level.

Turn on general lighting and set the installed lighting load including control gear using the slider.

Note the lighting schedule comes from the activity for the zone

Select the **luminaire type** from the drop down menu. This loads typical values for radiant and visible fractions for the luminaire type. You can override these if required. The convective fraction is calculated and an error is generated if the radiant + visible fractions ≥ 1 .



Field Name	Luminaire Configuration, Fluorescent Lighting				
	Suspended	Surface mount	Recessed	Luminous and louvered ceiling	Return-air ducted
Return Fraction Air	0.0	0.0	0.0	0.0	0.54
Fraction Radiant	0.42	0.72	0.37	0.37	0.18
Fraction Visible	0.18	0.18	0.18	0.18	0.18
<i>f_{convected}</i>	0.40	0.10	0.45	0.45	0.10

If appropriate turn on display/task lighting and specify gains and schedule. These gains are assumed not to be linked to any lighting control system and the lighting gain value does not come from the lighting template. In fact if you load a lighting template the Task and display lighting value is reset to zero.

Where lighting controls are installed turn on lighting control and select from the 3 lighting control options. Daylight control is covered in more detail on Day 2.

The lighting specified at building level will be loaded for all zones. Where zones have a different lighting specification, select that zone and edit the lighting data.

HVAC

HVAC can be defined in DesignBuilder using:

- **Simple HVAC** where the heating/cooling system is modelled using basic load calculation algorithms. For convective heating systems and all cooling systems the EnergyPlus 'Purchased air' system is used to calculate heating and cooling loads. 'Purchased air' supplies hot/cold air at predefined temperature and humidity for heating and cooling. 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. When using Radiant/convective the heating system type the EnergyPlus 'High Temp Radiant System' system type is used to calculate heating loads.
- **Compact HVAC** where the heating/cooling systems are defined in DesignBuilder using fairly simple parametric HVAC descriptions but modelled in detail in EnergyPlus.

In both cases heat generator (boiler) and chiller fuel energy consumption is calculated from heating and cooling loads as a post-process using efficiency factors.

Simple and Compact HVAC system will be covered in detail on Day 2.

Selecting and editing HVAC templates

When this option is selected in the model options screen, data can be loaded from a list of generic HVAC systems. (in the UK there is also a list of HVAC systems than can be used for the UK National Calculation Model). While at building level, click on HVAC and use the browse button to open list of generic HVAC systems. Select the most appropriate for the majority of the building.

The auxiliary energy consumed by fans and pumps will be set automatically for the system selected. For both **heating and cooling** the fuel type, the system Coefficient of Performance, the supply air temperature and humidity will be loaded from the template

Note the schedules will be inherited for the activity.

In general it will be necessary to edit the data on the template and in many cases there will be more than one HVAC system. Clicking on the + button in the right hand pane will open an edit screen. Give the HVAC system a recognisable name and set the category to project.

For **Simple HVAC** system the auxiliary power can be edited

For **Compact HVAC** systems the type is selected from the drop down menu.

Tabs for ventilation, heating and cooling can then be opened and appropriate values set.

The final tab allows the air temperature distribution to be set to mixed or dynamic gradient.

Click OK when a prompt will appear 'select named system as template' selecting yes will load the data from this template at building level so that it is inherited down to all zones in the building.

Where a building has a different HVAC system for some blocks or zones, use the + button again to create a second HVCA template, give it a new name and load this into the relevant blocks and zones.

DHW generator

A DHW generator needs to be selected for all occupied zones. Depending on the activity and building type selected for the zone, a standard DHW demand is assumed. 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 tearoom. 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 DHW generator that needs to be specified and described should be the generator which accounts for the majority of this demand, from those defined in the DHW generator tab.

- The default is Project DHW
- Select Type from drop down menu
- If Same as HVAC is selected then the HVAC system is automatically selected as DHW generator.
- Specify the delivery and supply temperature and select a schedule for DHW demand
- A single click on DHW template followed by selecting browse button gives access to other generic types of DHW.

7 Exercises on Geometry and Model Data

Carry out these simple exercises to help consolidate your understanding of DesignBuilder geometry and model data.

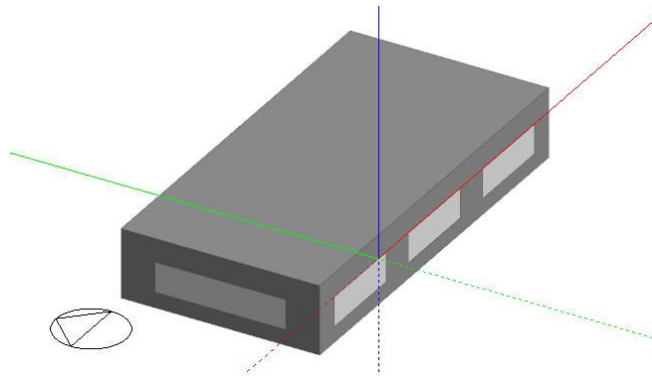
Default Model Templates

Many of the subsequent worked examples will be based on simple single and 2-zone models. These will be created in this exercise and saved as templates for future use.

Default 20x10 Single Zone

First create the single zone model:

1. Create a **New project** using any location and the **Blank** template.
2. Create a **New building** using default options and templates
3. Draw a block 20m x 10m with the longer facades facing North and South as shown in the screenshot below. Use all the default drawing options (Wall thickness = 0.293m and Block height = 3.5m).



Tip: You can achieve the same result with even less work!

4. Delete the block you have just drawn (Ctrl-Z undoes the add block command or select it and delete it)
5. Select the Perimeter shape to be **2-Rectangle** then move the mouse over the edit screen and type

20 10 and press **<Enter>**.

This 'key in' shortcut means "Draw a rectangle 20x10 (x direction first, y second)" Again delete the block and try typing:

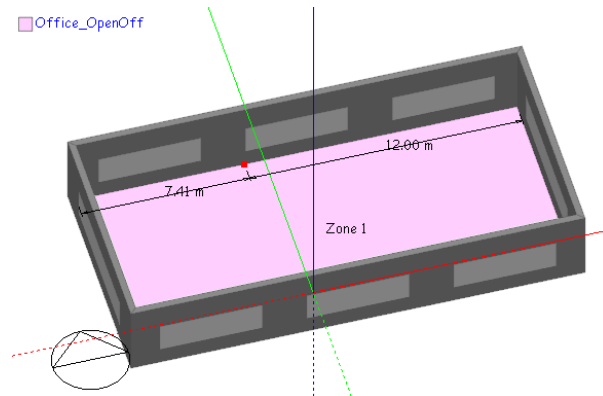
0 0 <Enter>	- go to origin
20 10 <Enter>	- draw the 20x10 block

Now save this single zone model as a template using the File > Save > Save as template menu option. Call the file **Default 20x10 Single Zone.dsb**

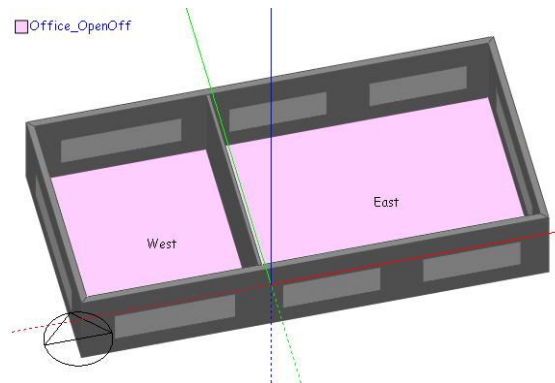
Default 20x10 2-Zone

Now create the 2-zone model. Continuing with the previous Default 20x10 Single Zone model,

1. Go down to the block level and click on the **partition tool** to start drawing a partition.
2. Switch on **Increment snap** to allow a precise positioning of the partition.
3. Move the mouse until you are 12m from the East end of the block and 7.41m from the West as shown in the diagram below. Then use the mouse to **draw the partition**.



4. Rename the zones **East** and **West** and go back up to the block level. The model should look as shown below.



Now save this 2-zone model as a template using the File > Save > Save as template menu option. Call the file **Default 20x10 2-Zone.dsb**

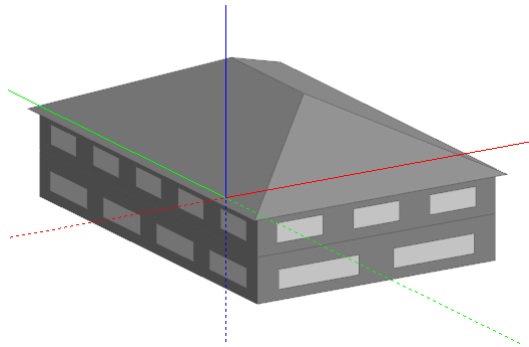
Exercise 1

Basic 2-storey building

Create rectangular building as follows:

External dimensions:	30 x 20m (20m along East-West X-axis)
Wall thickness:	0.35m
Block height ground	3.5
Block height 1 st floor	3.0

Hipped roof - slope 25 degrees with 0.5 m overhang



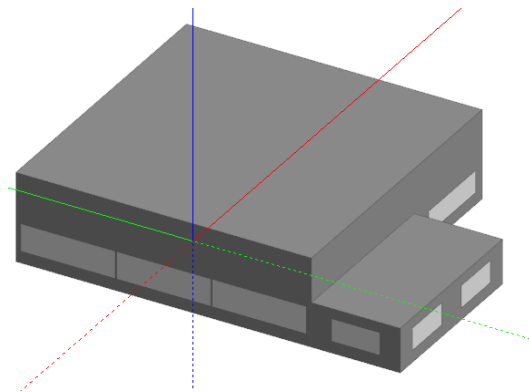
Exercise 2

Building with blocks side-by-side

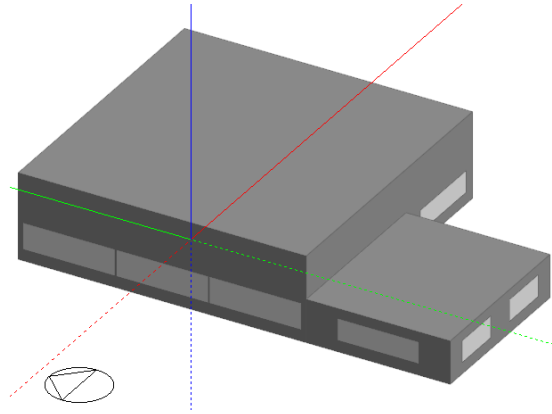
Create a building as follows:

- Draw a block 20 x 20m and 6m high
- Starting at the South west corner add a block that is 3m high and extends 12m along the South wall of the original block and stands out from that block by 6m

Building should look like:



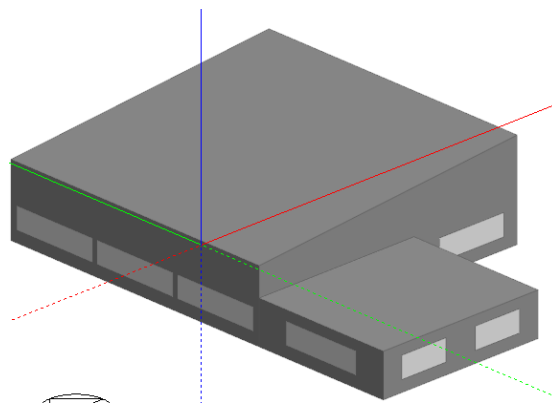
Now use the Drag face tool to pull the smaller block out to a distance of 10m. Note the dimensions shown when dragging the face are incremental (i.e. they show the increase/decrease in the dimension, not the total dimension).



Exercise 3

Sloped roof

- Using cutting and protractor tools to create a single slope roof (useful where occupied space extends to roof skin)
- Continuing from exercise 2, use the drag face tool to increase the height of the larger block by 2m.
- Turn on the protractor and set the increment to 5 degrees
- Then use the **Cut Block** tool to cut the block across the South face at an angle of 5 degrees starting at a height of 5m from the South west corner.
- Delete the cut block to create a single slope roof as shown below

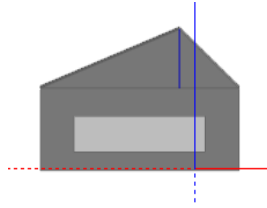


Exercise 4

Extruded blocks

Using **extruded blocks** to create **more complex shapes** such a more complex roof

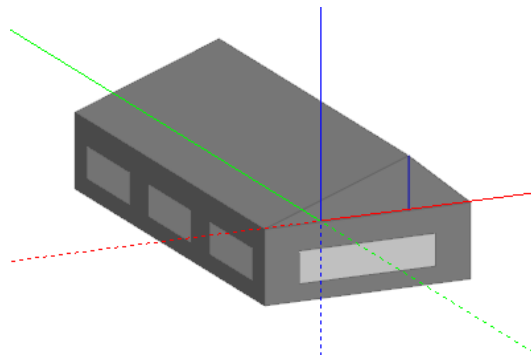
- Draw a simple block 10m x 20m and 3.5m high
- Add a gable end roof with the geometry shown below



- Ridge is 2.5m above eaves and is 7m from left hand corner

Tip: Use a construction line (vertical line in blue) to locate ridge position

Appearance of building



Exercise 5

Using Outline blocks

- Create a simple 2 storey building 10 x 20m and add a 30 degree pitched roof
- Add dormer window in middle of South facing sloping roof using an outline block. Align vertical face of dormer outline block with wall below.
- The vertical height of the dormer walls is 1.5m and the sloping roof sections are 1.5m long and at 45 degrees.
- Cut outline block using roof as cutting plane (cutting method = **Select plane**)
- Convert the dormer window outline block to a building block Wall thickness is 0.1m
- Add a window 1m x 1m.
- Cut a hole in the sloping roof and merge the dormer zone with the main roofspace zone using 'Merge zones connected by holes' Model option.
- Copy the dormer block twice to give a dormer window over each window of the floor below. The hole connections are copied with the block.
- Visualise the building and look inside – explore visualisation options.

Visualisation of building with dormer window created using outline blocks



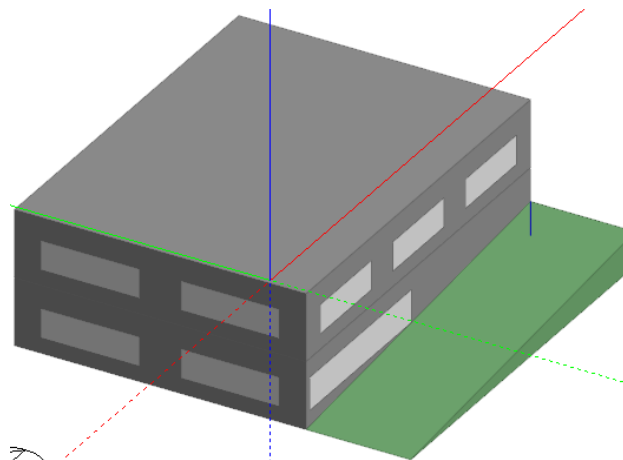
Exercise 6

Using ground component blocks to set ground adjacency

- Create a two-storey building with flat roof 15 x 15m with default 3.5m block heights.
- The building is on a slope such that the ground rises along the South wall from floor level at the west corner to a height of 1.75m at the East corner.
- Use a ground component block to set the adjacency for the South wall as shown below.

Tip: use a construction line to provide the snap point for ground block at South East corner

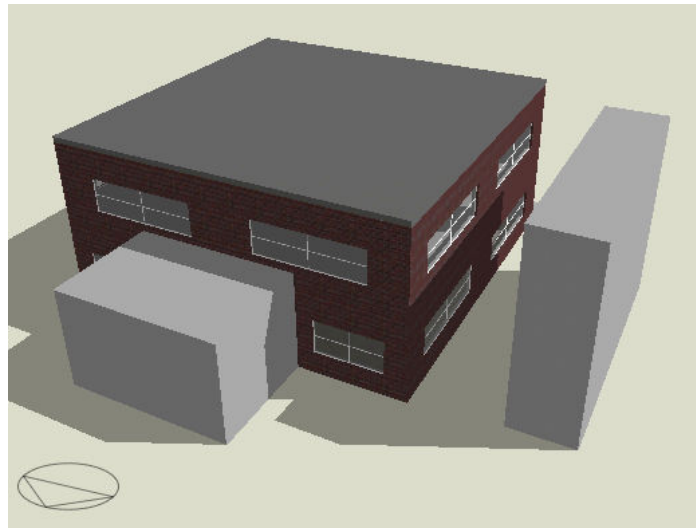
Ground adjacency set by using a ground component block (green)



Exercise 7

Using *standard* and *adiabatic* component blocks

- Rotate model from Exercise 6 to show North face
- Add a 4m high adiabatic block to middle of North face. This will be shown in red.
- Add a standard block same height as building in front of west face. This will provide shading (and reflections if this Solar calculation option is selected).

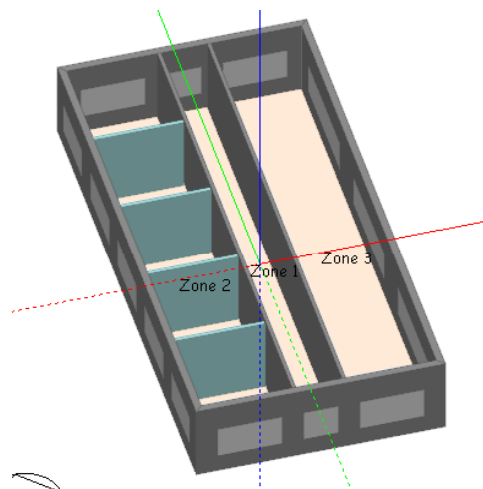


Exercise 8

Zoning the building

- Create a building having 10 x 20m external dimensions
- Add 2m wide corridor down middle parallel with the longer wall
- Insert partitions to create 5 equal cellular offices along west wall - assume all have same HVAC system

Tip: You should have created just 3 zones. Check this by looking in the Navigator.



Exercise 9

Specify lighting

- Draw simple 10 x 30m building with long dimension running North to South.
- Divide building into 4 zones
- Set the activity at building level as Office_OpenOff
- Set activity for Zone 3 as Office_Reception
- Set activity for Zone 4 as Office_Store
- At building level select the lighting template as T8 Fluorescent, triphosphor high frequency control gear.
- Set the lighting energy as 16 W/m²
- Note schedule Office_OpenOff_Light has been set from the activity loaded
- Select luminaire type as surface mount
- Turn on lighting control and set control type to linear and % area covered by lighting area1 to 40%.
- Now go to Zone 3, Reception
- Select lighting template as Fluorescent compact
- Accept default 23 W/m² and set luminaire type as recessed
- Turn off lighting control
- Now go to Zone 4, Store
- Select lighting template as Metal halide
- Set luminaire type to Suspended
- Turn off lighting control
- Data inherits from building level in Zones 1 & 2

Exercise 10

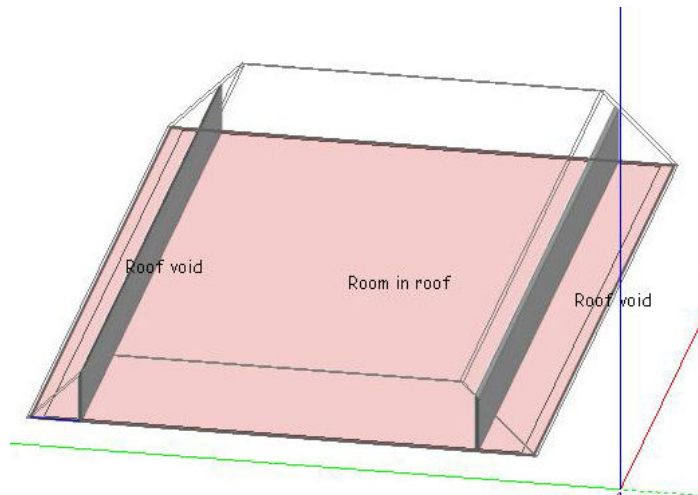
Editing default constructions

- Draw simple building 10 x 15 m
- Add second floor and unheated pitched roof
- Modify construction wall for second floor to:
 - Outer leaf metal cladding steel 2 mm thick (look under metals)
 - Insulation polyurethane board diffusion tight 0.1m thick
 - Inner leaf concrete 1800kg/m³ concrete block, medium density inner leaf 0.1m thick

Exercise 11

Occupied roof space – room in the roof

- Draw a 20 x 30 two storey building with a 45 degree pitched roof
- Use construction lines to position partitions 2m for each edge of the sort dimension as illustrated below.

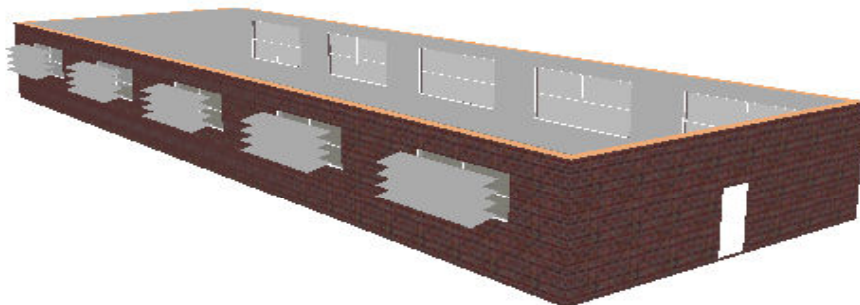


- Name the zones as 'Room in the roof' and 'Roof void' as shown
- Go to building level a cut the roof block at a height of 3m to create a ceiling.
- Select room on roof zone and set zone type to standard and set activity to hotel bedroom.

Exercise 12

Defining openings

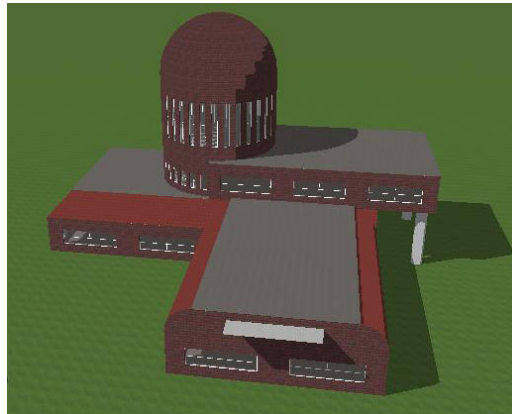
- Draw simple 10 x 30m building with long dimension running North to South
- Change default glazing to **4-20-4 low coated air filled**.
- Edit openings to create building with:
 - 20% glazing in west wall with window height of 1.3m and sill height of 1.5m. Apply local shading with 1m overhang louvre.
 - 40% glazing in East wall with window height of 2m and sill height of 1m. Apply window shading using venetian blinds - light, internal, always on.
 - No other glazing but door in South wall.



Exercise 13

More complex geometries

Use as many of the drawing tools as possible to create a building with more complex geometry such as that illustrated below. Use construction lines to provide snap points to position curved walls and other features.



Exercise 14

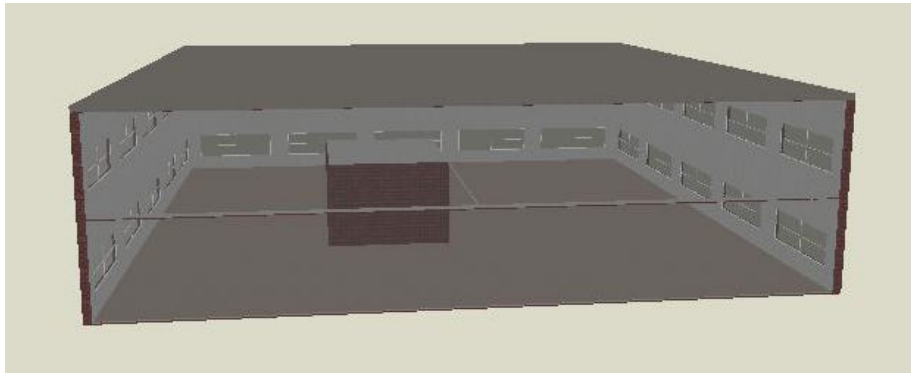
Free standing office in warehouse

Sometimes it is necessary to create a smaller zone such as an office within a larger zone (e.g. a large warehouse space).

The steps involved are:

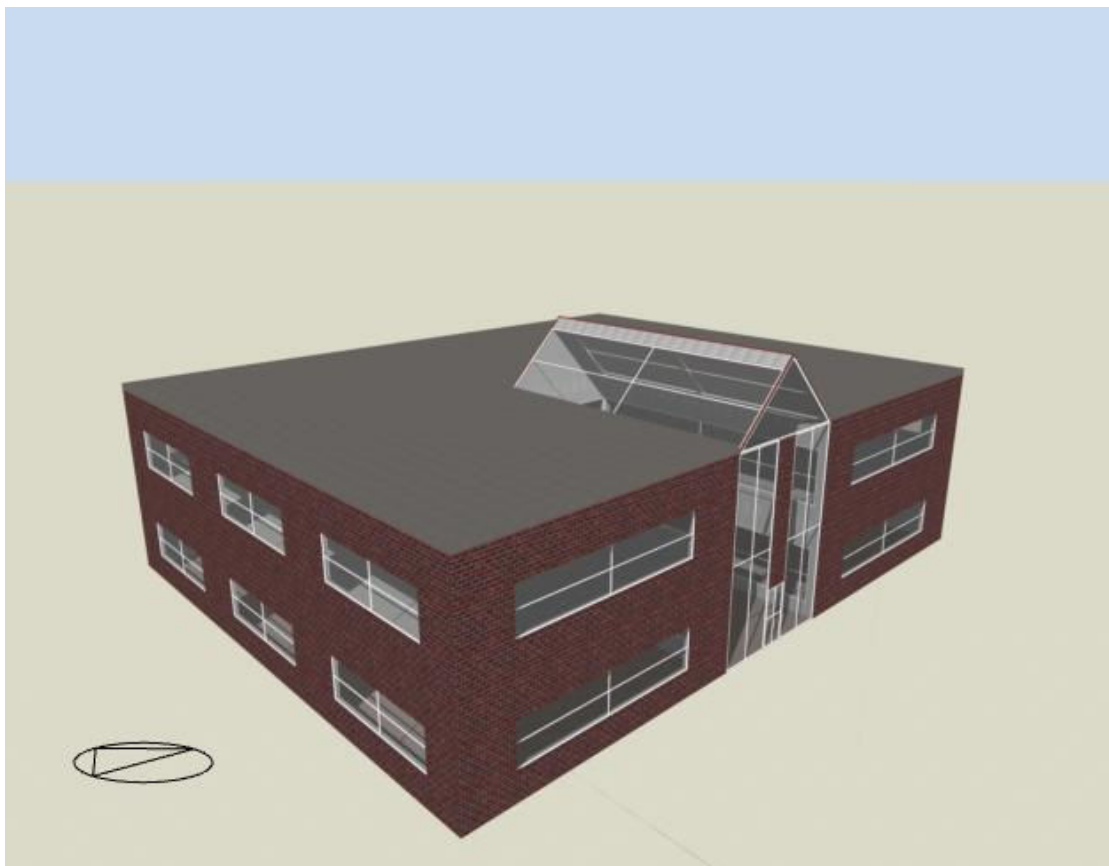
1. Draw the ground floor block including both office and warehouse. The block should be the height of the office space as the top of the block will form the ceiling of the office.
2. Draw the 1st floor block representing the high-level warehouse space.
3. Draw holes (two will be required) so as to cut away all the floor of the 1st floor block except that around the top of the office.
4. Go to model options and on the Advanced tab switch on the 'Merge zones connected by holes' option. This will merge the upper and lower warehouse zones into a single zone. The upper warehouse zone should look something like the screenshot below.
5. From the navigation tree select the first floor block and then floor element corresponding to the ceiling of the office. In the construction tab, open the Adjacency toolbar and tick "Exclude this surface area from the total zone floor area"

See also www.designbuilder.co.uk > Support > Knowledge base > How to include a single story office within a full height warehouse. It is also available as a DesignBuilder example file.

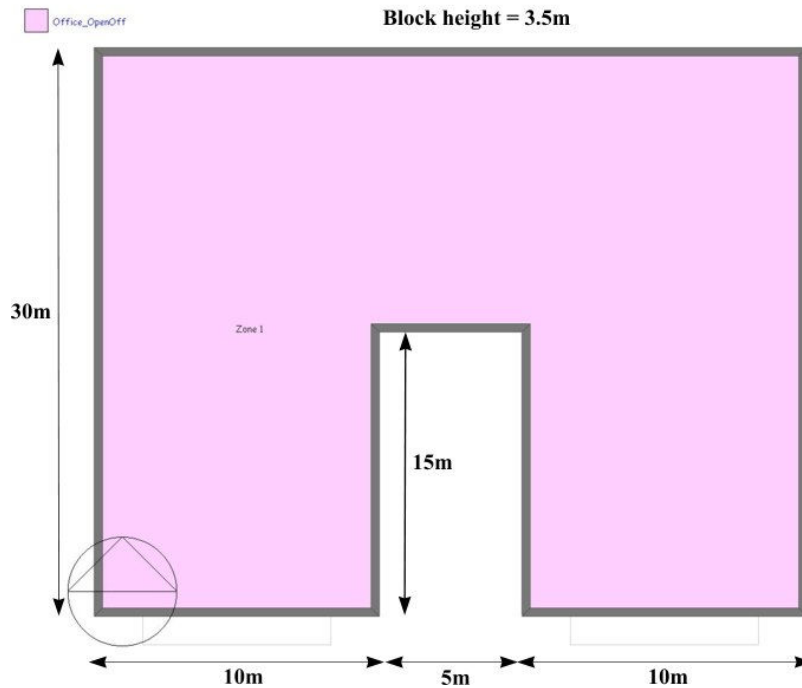


Exercise 15

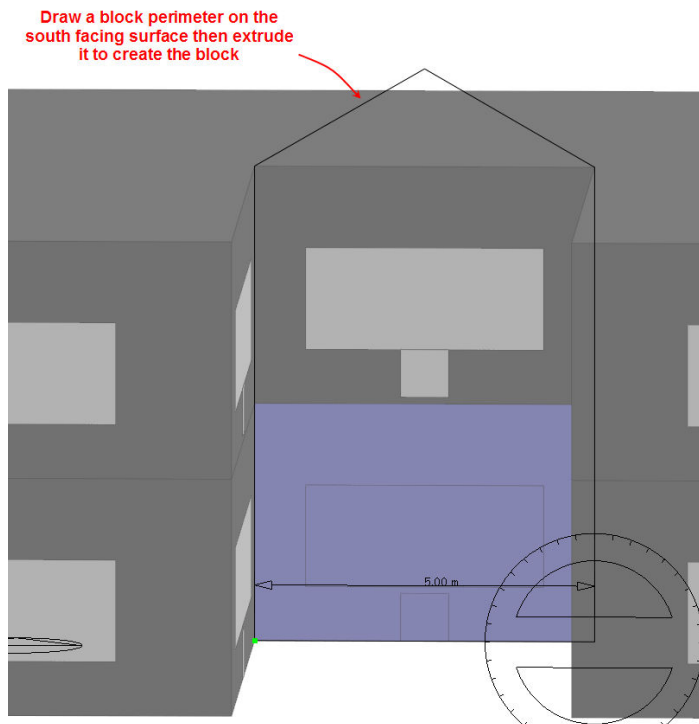
Model used as basis for Day 2 Exercises



- Draw the U-shaped ground floor block using the dimensions below.

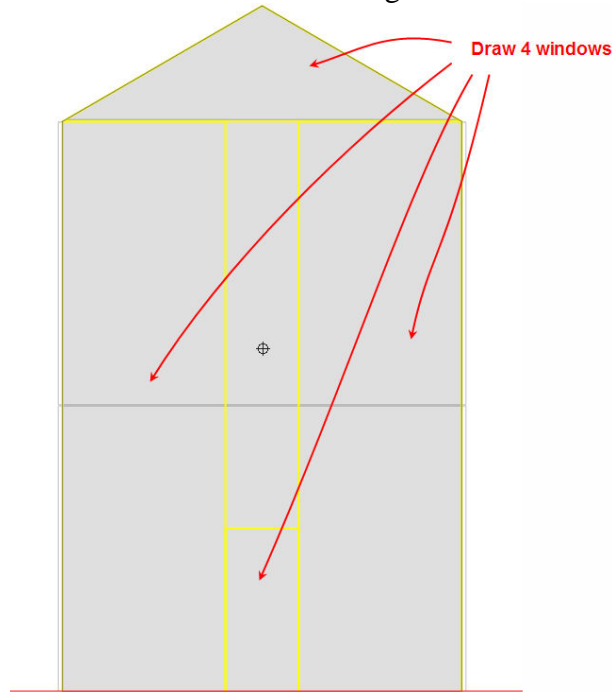


- Copy the ground floor block up to create the first floor block.
- Change the Block type of the Roofspace block to Pitched roof and set the block wall thickness to 0.05m.
- Draw the atrium block by drawing a block perimeter on the indented south face as shown below and extrude to create the block. Use the protractor to create a 30° roof slope. The drawing guides from the mid-point of the adjacent surface will help position the apex of the perimeter.

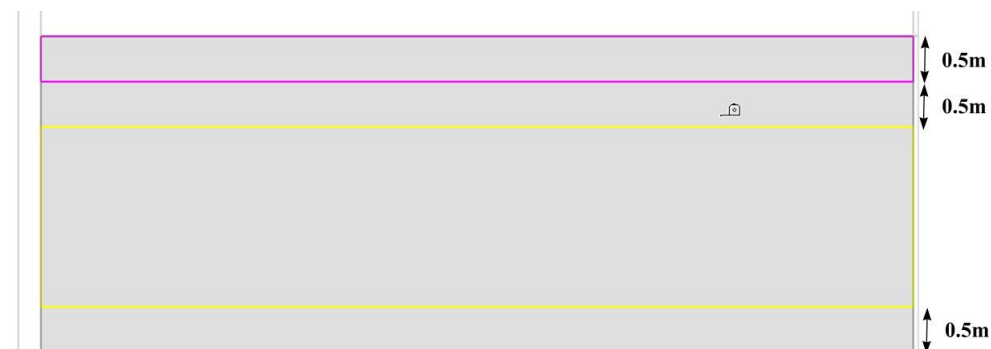


- The block wall thickness should be 0.01m

- Define the atrium block glazing by setting the % glazed = 0 or use the **No glazing** façade template. Draw large roof vents at the top of the roof surface covering the width with a height of 0.5m. Fill the rest of the surface with a window. Repeat for the other roof surface. *Switch increment snap on to help with this.*
- Draw 4 windows on the atrium South-facing external surface as below:

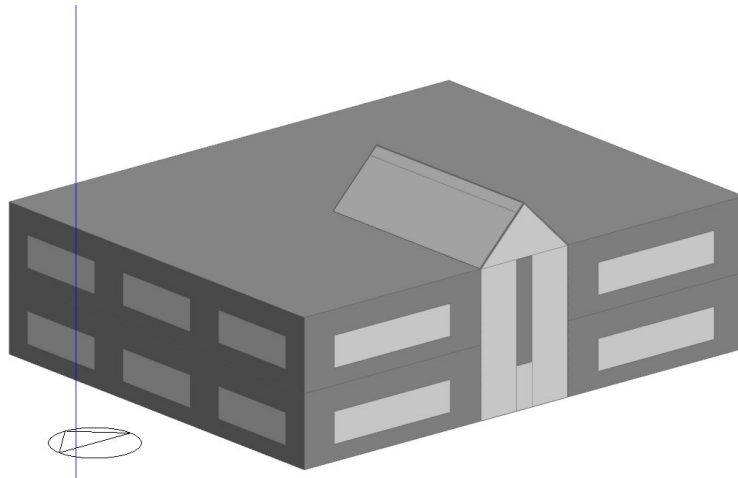


- The atrium is to be built using a lightweight construction so choose the Part-L 2006 Lightweight construction template at block level (optional).
- For visual appearance sake, set the atrium **pitched roof (occupied)** construction to use a lighter coloured material (optional).
- Set the activity for the atrium block as 'Office_Circulation' and use the default 'Office_OpenOff' activity for the rest of the building.
- Draw windows and vents in each of the inter-block partitions as below, the vent is the purple opening at the top of the surface below.



- The external windows for the ground floor and first floor blocks can be set up using the default façade mechanism. At the building level, opening tab, make sure the default 30% glazed windows with 5m spacing and 0.8m sill height are set.
- Name the blocks Ground floor, First floor and Atrium using the Navigator.

Save the model as a template called **Atrium example base**. The model should look like the screenshot below.



This model is completed in the next section by loading appropriate templates.

8 Data Management

This section covers tools you can use to speed and ease data management tasks.

Loading data from templates

DesignBuilder Templates are collections of data that are useful for quickly loading data *en masse* into the model. They can be especially useful if you frequently model similar types of building or if you often wish to model a particular HVAC or lighting system for example. Create the template once and load it multiple times!

Each template contains sufficient data to load the complete set of data displayed under each of the Activity, Construction, Opening, Lighting and HVAC tabs. New templates made up of different sets of data can be created by using the copy or add and edit tools when in the templates tab on the opening screen. These data sets can then be loaded into the model from the new templates.

Templates can be created:




- with a model open, in which case the templates are stored with the model, or
- on the opening screen, in which case the templates are stored in the DesignBuilder Library.

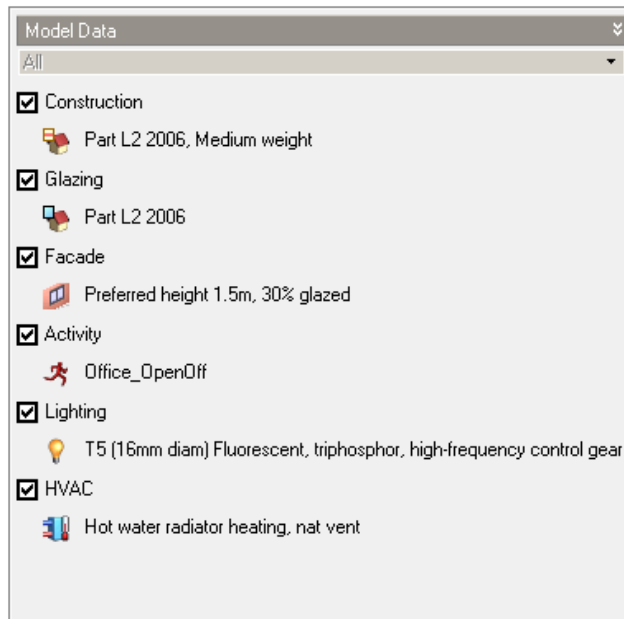
You can copy templates and components from one project to another and to/from the Library using the **Import/Export library data** commands accessed from the File menu. **Export** transfers the data to a .DDF file and **Import** allows a previously created DDF file to be loaded.

Templates do not form part of the model even if they are stored in the model so *an edit to a template will not affect simulation results until the template is loaded to the model.*

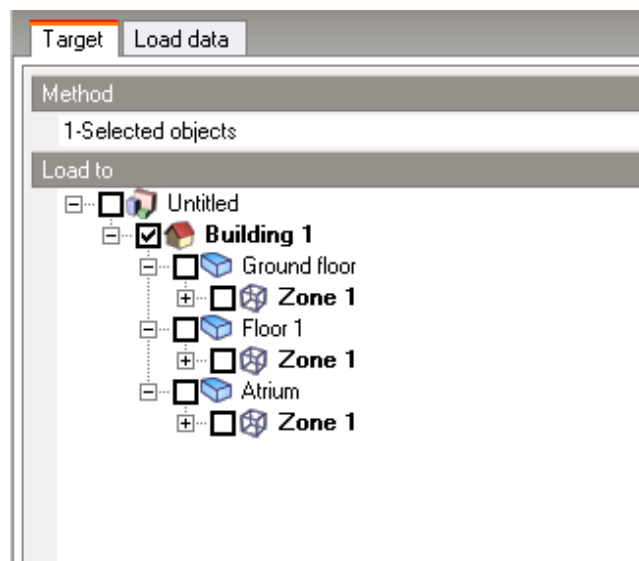
Example – Load data from template

In this example you will complete the **Atrium example base** model started in Exercise 15 and learn how to use the Load data from template function.

- Go to the Building level  and Clear all data to default using 
- Use the Load data from template tool  and on the **Load data** tab select the data to be loaded. The dialog should be as shown below.

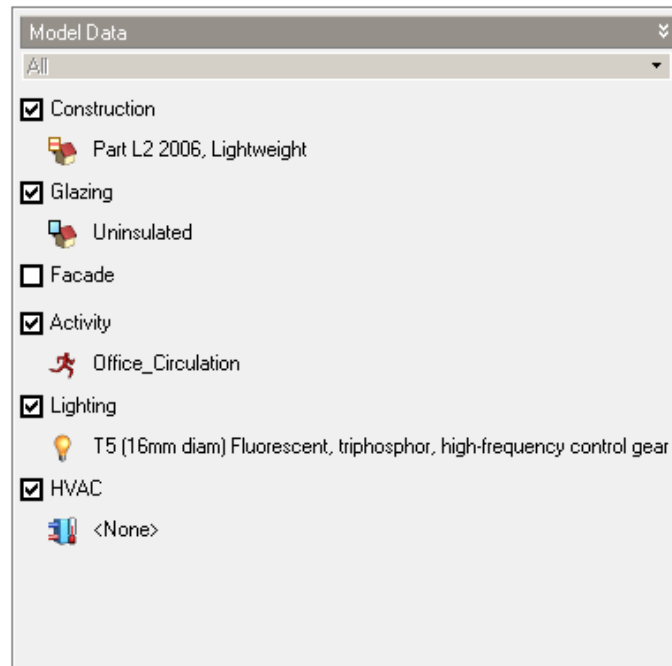


- On the **Target** tab Set the target which is the building only. This will have been selected as the default target for you (because you are at building level in the model).



Clicking OK loads the templates displayed to the parts of the building selected. The templates and the information in them are shown in red in the model data at the level in which they are loaded, but in blue at any lower level.

Then load the templates for the atrium:



In larger models this process is a convenient way to load templates to parts of the building having similar building attributes.

The same effect could have been achieved by going to the atrium zone and loading the template options at the top of the respective tabs. The most efficient targeting strategy for loading the data varies from building to building. In small models you may find it easier to use the template in the Model data tabs but in large buildings with many zones the **Load data from template** function can be an efficient way to load the templates for multiple zones of the same attributes in one go. This works best if you label the zones in the Navigator with a name that recalls the activity. For example 'Store small' would be more useful than 'Small room'.

Reload data

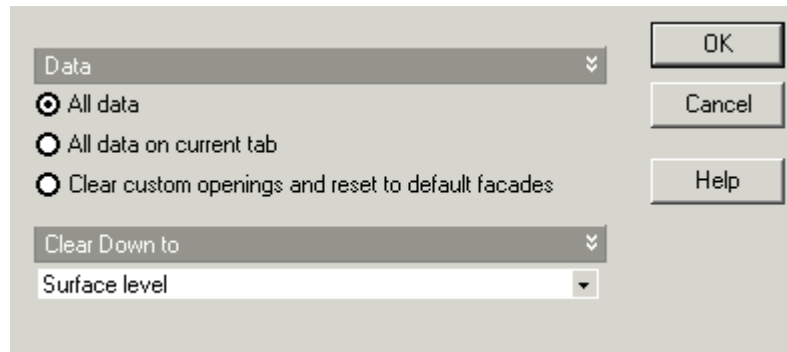
Use the reload option to reload all hard-set templates (i.e. template selections shown in red). This will remove all red bold data where template defaults have been overridden.

Clear data

Is important to understand the concept of 'clearing data' in DesignBuilder because it is the mechanism by which previously hard-set user is cleared and the data instead is inherited from the level above in the hierarchy. This function is accessed from the **Clear Data to Default** dialog, which allows you to:

- **Clear Model data** 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 (shown in blue). This can be done for individual model data items, all model data on the current tab or for all model data on all tabs.
- **Clear custom openings** and revert back to default facades.

You can control the extent of the clearance by setting the **Clear Down to** data at the bottom. Leave this data as 'Opening level' if you want all data in the currently selected model object (including child objects) to be returned to a default state. Depending on the data to be cleared you may only be able to clear down to zone or surface level.



You can clear individual items of model data by right clicking on them and selecting **Clear to default** from the menu.

When used at **building level** the clear model data function actually clears all levels *below* the building level – it is not allowed the clear the building data itself because this is the top of the inheritance hierarchy and there is nothing else to inherit from.

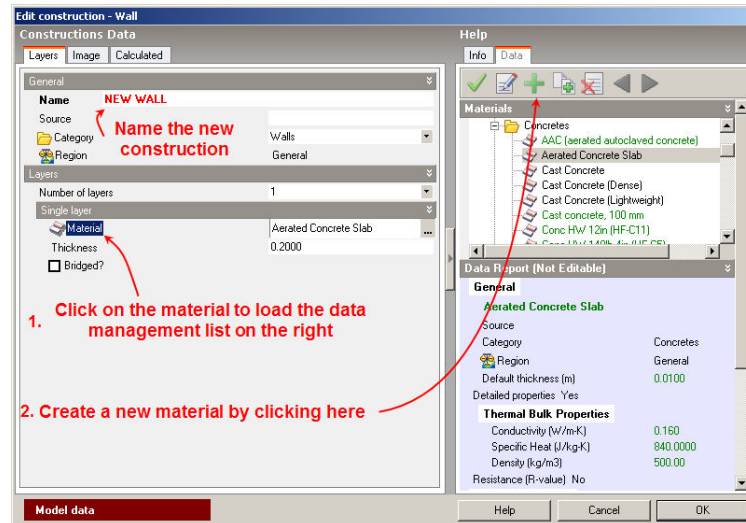
Export and Import Components and Templates

You can export component and templates in **DDF** form and these can subsequently be:

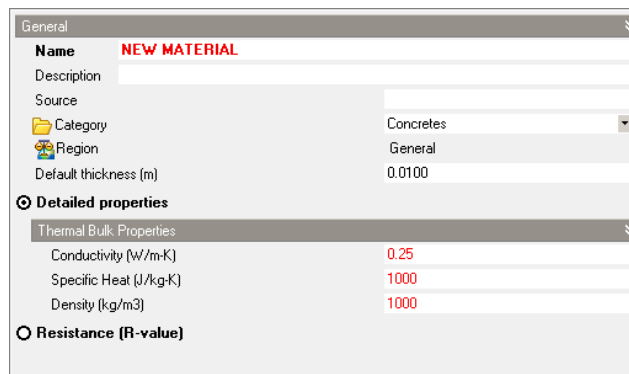
- Imported into other projects,
- Loaded into the DesignBuilder library and also
- Loaded onto other computers, shared with colleagues etc.

This example takes you through the stages in the process.

- Create a new project using the Default 20x10 Single Zone model template.
- Go to the building level and the Constructions tabs. Click on **External wall** to load the Data management panel on the right of the screen.
- Create a new external wall construction by clicking on the + **Add new item** toolbar icon. The new wall is to use a single layer of a new material which we will also define now.



In the materials dialog create the new material, looking something like:

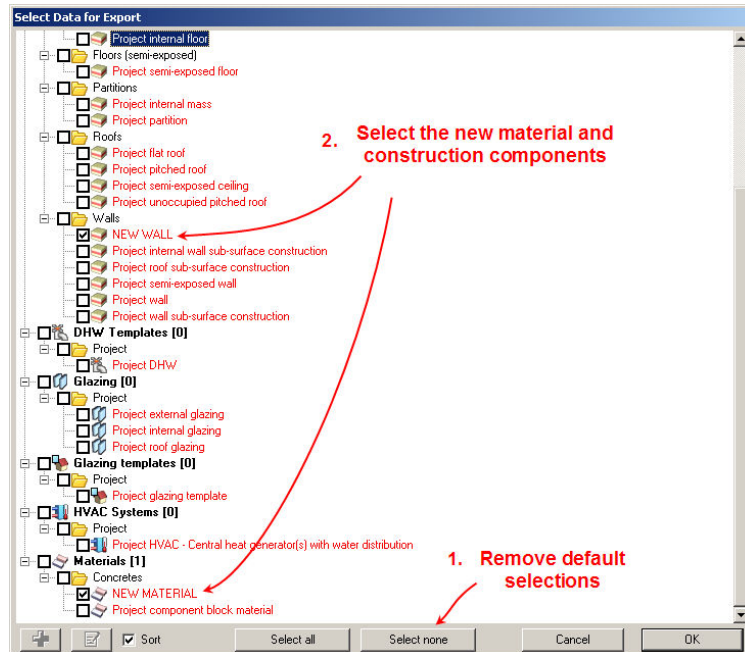


- Click OK to create the material NEW MATERIAL.
- Then back in the Construction dialog, click the Tick icon to select the new material.
- Click OK to save the new construction.
- Select the NEW CONSTRUCTION as the building level External wall construction.

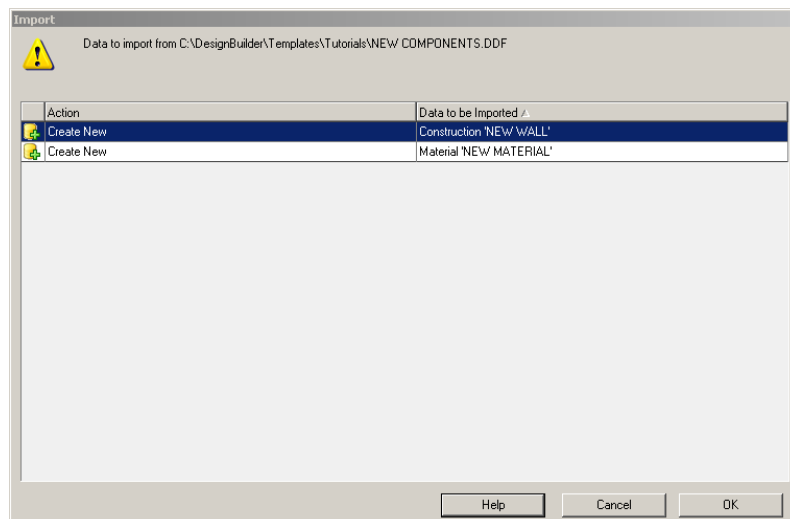
You have now created a new project construction component called **NEW CONSTRUCTION** and this references the new material component **NEW MATERIAL**.

- Now export these 2 components as a DDF file using the File > Export > Export library data.
- This opens the **Select Data for Export** dialog. Clear all the default selections by clicking on the **Select None** button then select the 2 new components under Materials > Concretes and Constructions > Walls categories and Click OK to create the DDF file. Save the file to

somewhere you can find it easily. The default **DesignBuilder data** folder is a good place, as DesignBuilder will look there first when importing later. Save using the filename **NEW COMPONENTS**.



- Now save this project and **create a new blank project**. Add a new building and draw any block quickly
- **Import the DDF file** you just created by using the **File > Import > Import library data** menu option.



DesignBuilder displays a dialog showing the components that have been imported.

- Now in the Constructions tab, click on External wall to list the constructions in the Data panel on the right. You should be able to find the **NEW CONSTRUCTION** that has just been imported to the project.
- Select **NEW CONSTRUCTION** as the default external wall for this building. As an experiment once you have selected NEW CONSTRUCTION as the external wall, try deleting it from the project. You will be prevented from doing so because it is used in the model. Likewise you would not be allowed to delete **NEW MATERIAL** because it is used in **NEW CONSTRUCTION**.

You can also **import** the **DDF** file into the DesignBuilder Library so it will be included with all new projects created. To do this, follow the same Import process but on the **Opening** screen.

9 Heating Design Calculations

Introduction

Heating design calculations are carried out to determine the size of heating equipment required to meet the coldest winter design weather conditions that are 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 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.

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.

The steady-state 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)

Zones can be unheated in which case they will 'float'.

The total heat loss in each zone is multiplied by a Design margin (safety factor) of 1.8 by default to give a recommended heating Design Capacity. This heating capacity will be written directly into the Heating capacity HVAC model data to be used for simulations, depending on the setting of the Plant sizing model option on the Data tab under HVAC.

The Heating Design margin is used to multiply calculated steady state heating loads in each zone to give a recommended heating plant 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.

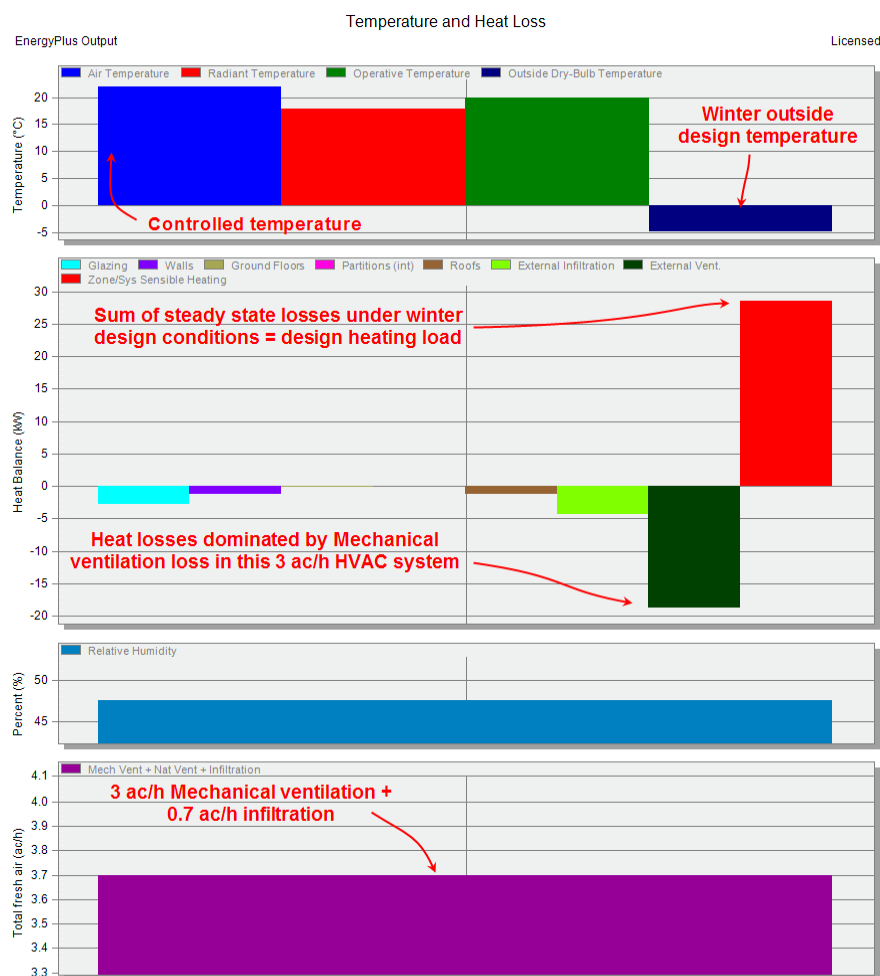
Why use such a high default safety factor? To ensure that the plant is large enough to warm the building up soon after start up on cold winter Monday mornings when the building has cooled down over the weekend. Use a lower margin if the preheat periods will be long or if some underheat on cold winter mornings is acceptable.

Heating design screen

Click on the Heating design tab at the bottom of the screen to run heating design calculations. The first data tab is **Steady-state** showing detailed steady state design calculation results for the currently selected object in the navigator.

Steady-state

The graph below shows the breakdown of the heat losses graphically which can help to identify any particularly poor aspects of fabric or systems. In this case the mechanical ventilation system dominates the heat loss.



Summary

The Summary tab shows the key results for each zone. The most important heating design result, Design Capacity is shown in red and it gives the size of the heating equipment required to heat the zones and building to the inside design temperatures under outside winter design conditions.

The totals for blocks and for the whole building are shown in the grid headers.

Untitled, House , Entrance block, Entrance			
Steady State		Summary	
Zone	Comfort Temperature (°C)	Steady-State Heat Loss (kW)	Design Heating Capacity(kW)
[-] House (Total design heat loss = 28.160 (kWh))			
[-] Entrance block (Total design heat loss = 6.840 (kWh))			
Entrance	18.34	4.56	6.84
[-] Roof 1 (Total design heat loss = 0.000 (kWh))			
Zone 1	7.05	0.00	0.00
[-] Roof 2 (Total design heat loss = 1.740 (kWh))			
Zone 1	10.07	1.16	1.74
[-] Ground floor (Total design heat loss = 19.580 (kWh))			
Toilet	18.35	0.88	1.32
Kitchen	18.73	1.52	2.28
Bedroom	18.16	3.93	5.90
Sitting room	20.07	5.38	8.07
Cupboard	18.45	0.67	1.00
Bathroom	18.79	0.67	1.01

The table can be exported to spreadsheet **csv** format using the **export** toolbar icon.

10 Cooling Design Calculations

Introduction

Cooling design calculations are carried out to determine the capacity of mechanical cooling equipment required to meet the hottest summer design weather conditions that are likely to be encountered at the site location. In zones that 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.

Cooling 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.

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 located 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.

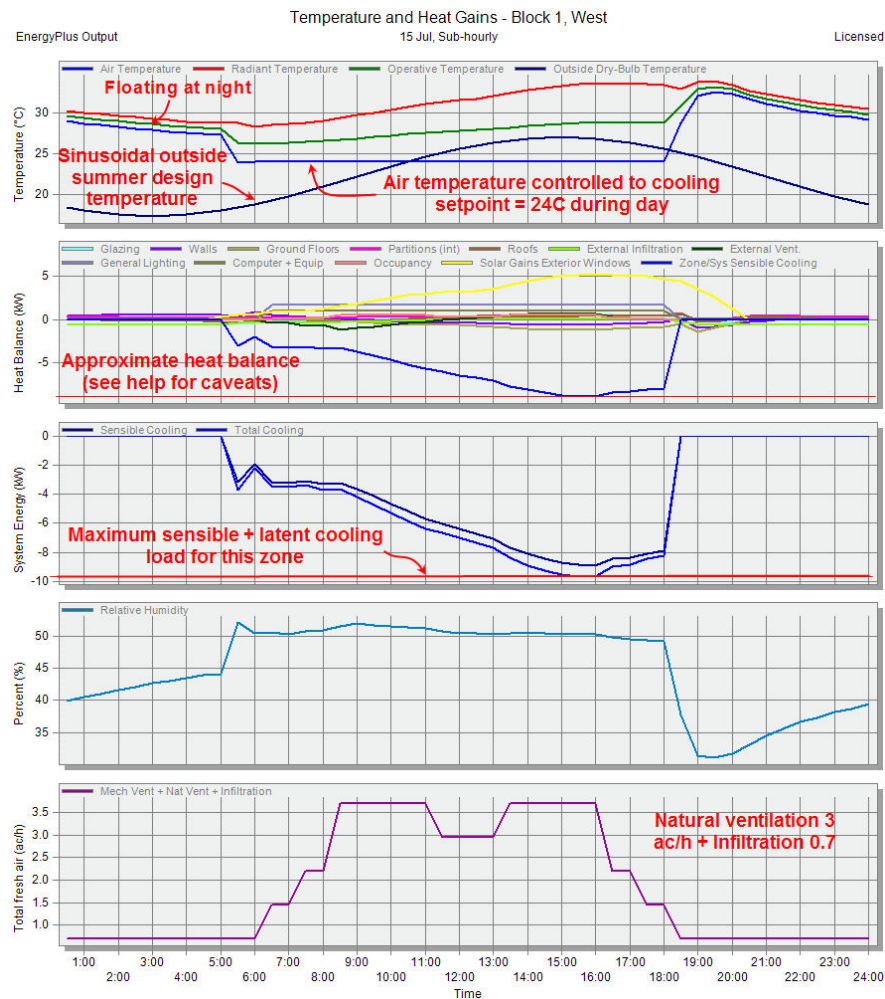
The maximum cooling load in each zone is multiplied by a Safety factor (1.3 by default) to give a Design Cooling Capacity. This design cooling capacity can be written directly into the Cooling capacity model data depending on the setting of the Plant sizing model option on the Data tab under HVAC.

Cooling design screen

Click on the Cooling design tab at the bottom of the screen to run cooling design calculations. The first data tab is **Analysis** showing detailed **periodic steady state** design calculation results for the currently selected object in the navigator.

Analysis

The data on the Analysis tab gives a good overview of the heat flows, fresh air and comfort conditions in each zone.






Summary

The Summary tab shows the key results for each zone. The most important cooling design result, Design Capacity is shown on the left and it gives the size of the cooling equipment required to cool the zones and building to the inside design temperatures under outside summer design conditions.

The totals for blocks and for the whole building are shown in the grid headers.

Analysis Parametric Summary												
Zone	Design Capacity (kW)	Design Flow Rate (m ³ /s)	Total Cooling Load (kW)	Sensible (kW)	Latent (kW)	Air Temperature (°C)	Humidity (%)	Time of Max Cooling	Max Op Temp in Day (°C)	Floor Area (m ²)	Volume (m ³)	Flow/Floor Area (l/s-m ²)
Building 1 Total Design Cooling Requirement = 12.590 (kW)												
Block 1 Total Design Cooling Requirement = 12.590 (kW)												
East	0.00	0.00	0.00	0.00	0.00	-	-	-	39.1	113.0	395.4	0.00
West	12.59	0.80	9.68	8.81	0.77	24.0	50.3	16:00	33.2	68.8	244.3	11.50

 Cooled zone requires cooling equipment of size 12.59 kW
  Uncooled zone
  Reality check

Parametric Cooling Design

Use the Parametric facility for Cooling design as a tool for identifying the impact of changes in a maximum of 2 design variables on cooling load and maximum temperature. The use of parametric simulations is described in detail later under the Simulation section.

11 Simulations using Hourly Weather Data

You can generate detailed building energy performance data based on simulations using measured hourly weather data. Simulations using EnergyPlus have the following characteristics:

- Weather data comes from Hourly weather data file.
- Includes consideration of heat conduction and convection between zones of different temperatures.
- Includes solar gain through windows and shading from other buildings and self shading by the building itself..
- Simulation of HVAC equipment
- Includes one or more **warmup** days to ensure correct distribution of heat in building thermal mass and 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.

Click on the **Simulation** tab at the bottom of the screen to view simulation data. If the necessary data has not yet been generated, a simulation is automatically started to generate the data.

Calculation options

Before the simulation starts you can change calculation options.

Simulation Period

Select the start and end days for the simulation, or 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).

The typical periods accessed from the Info panel refer to statistically derived data for the current hourly weather file.

Interval

Selecting Monthly is required to obtain annual output. Monthly, daily, hourly and sub-hourly data can be selected by checking the appropriate boxes.

Number of time steps per hour

The time steps per hour is the number of times the building thermal network is solved per hour in the simulations. When the HVAC portion of the simulation begins it's solution for the current time step, it uses the basic time step as its maximum but then can reduce the time step, as necessary, to reach the solution. Though many buildings can be successfully simulated with 1 or 2 time steps per hour, EnergyPlus suggest 4 or 6 for non-HVAC simulations and 6 or 10 for simulations with HVAC.

Note:

- 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).
- When using 1 time steps per hour you will not be able to access Temperature distribution results

Temperature control – stick with **Air temperature** control unless you fully understand the consequences of the alternative options.

The other options are less frequently used. Full descriptions are given in the help.

Model Checking Strategy

Simulation models should go through a minimum check procedure including the following:

- Check model by ensuring that data is being inherited correctly so that changes at building and block levels have the desired effect.
- Run **summer design week** simulation generating hourly results. Study hourly results to check correct operation of the building, equipment operation, lighting, natural ventilation, mechanical ventilation, fresh air.
- Run **winter design week** simulation checking same data.
- Run annual simulation generating monthly results, requesting distribution data and view distribution data for key zones.

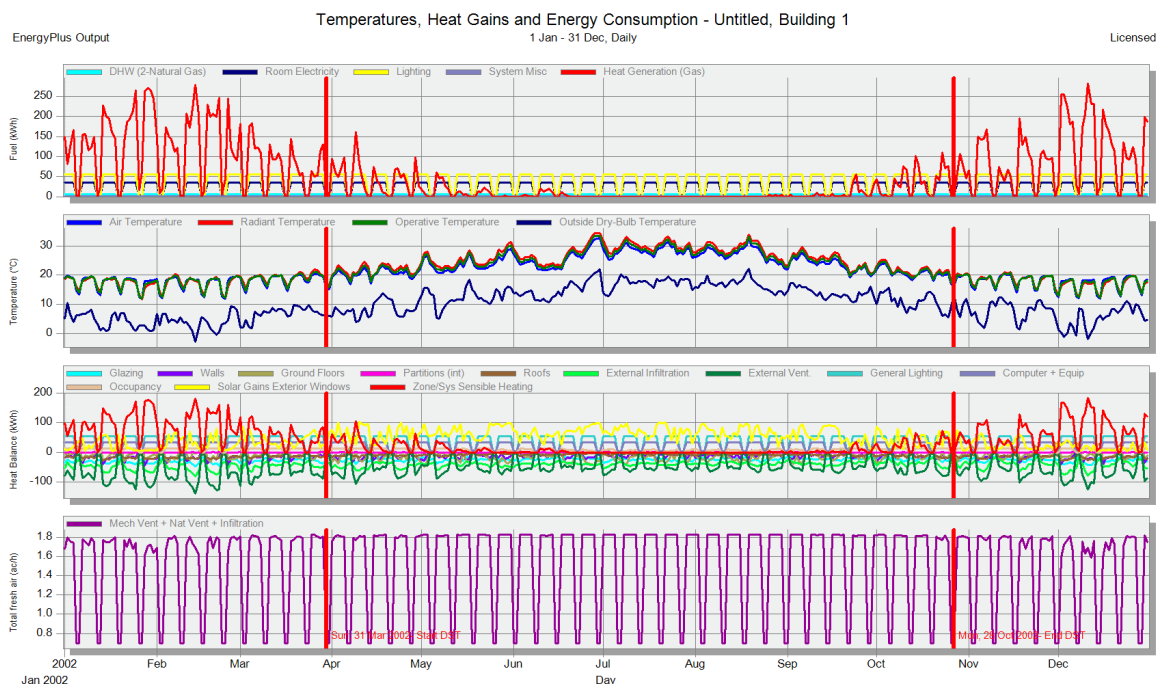
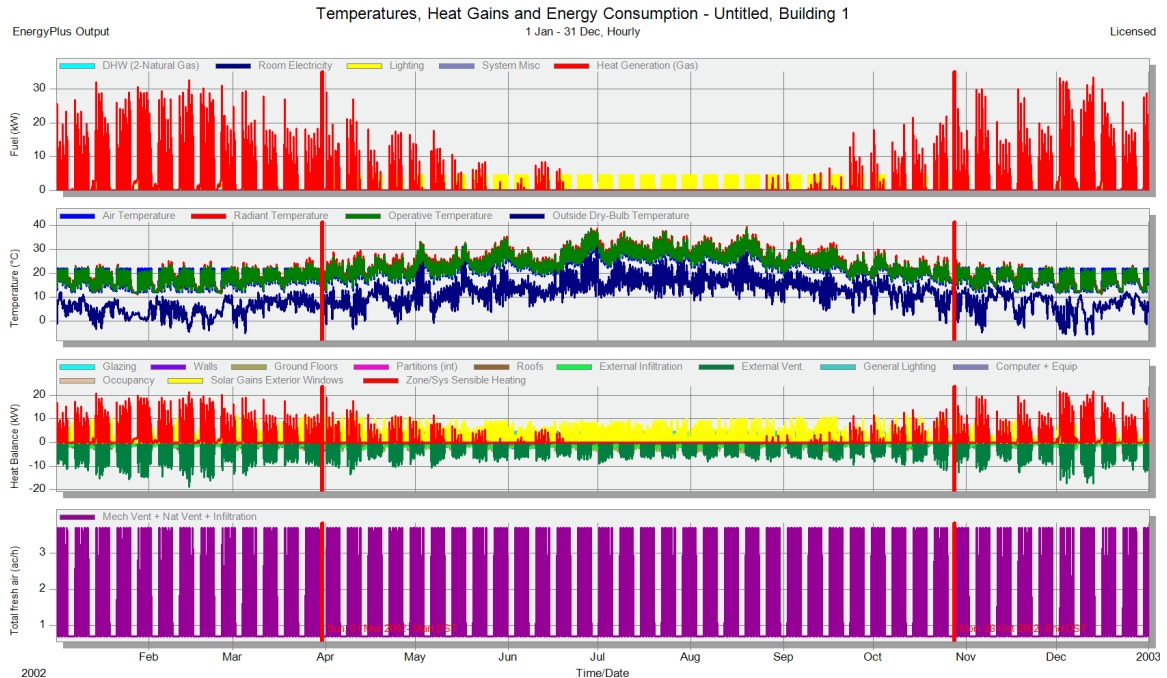
Simulation Analysis

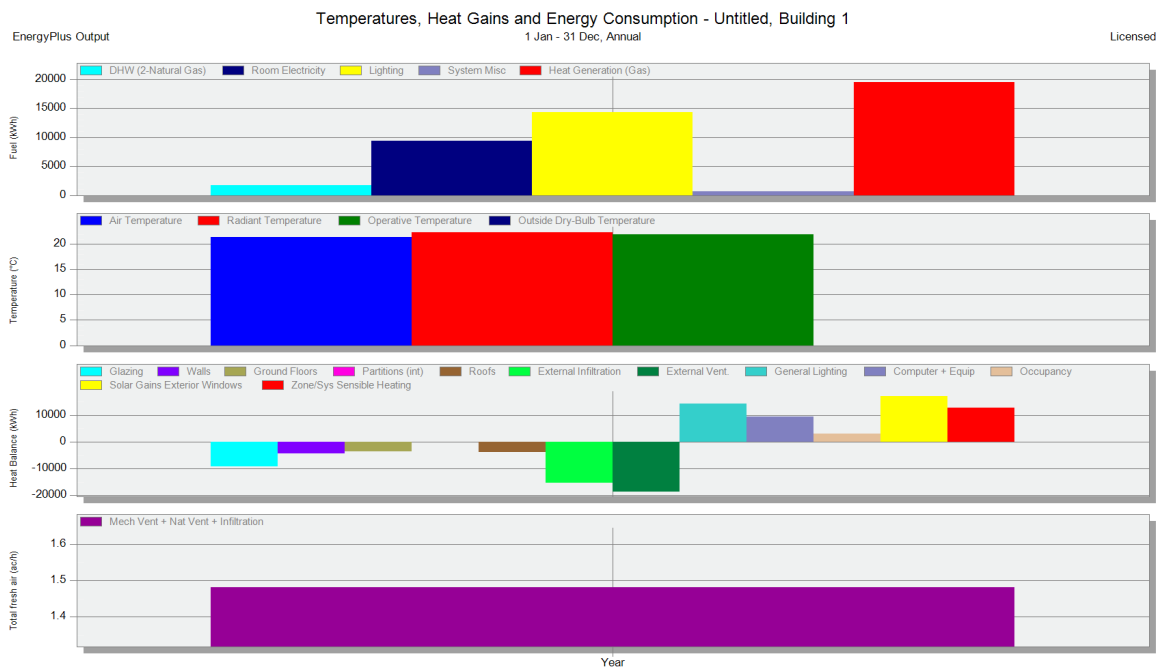
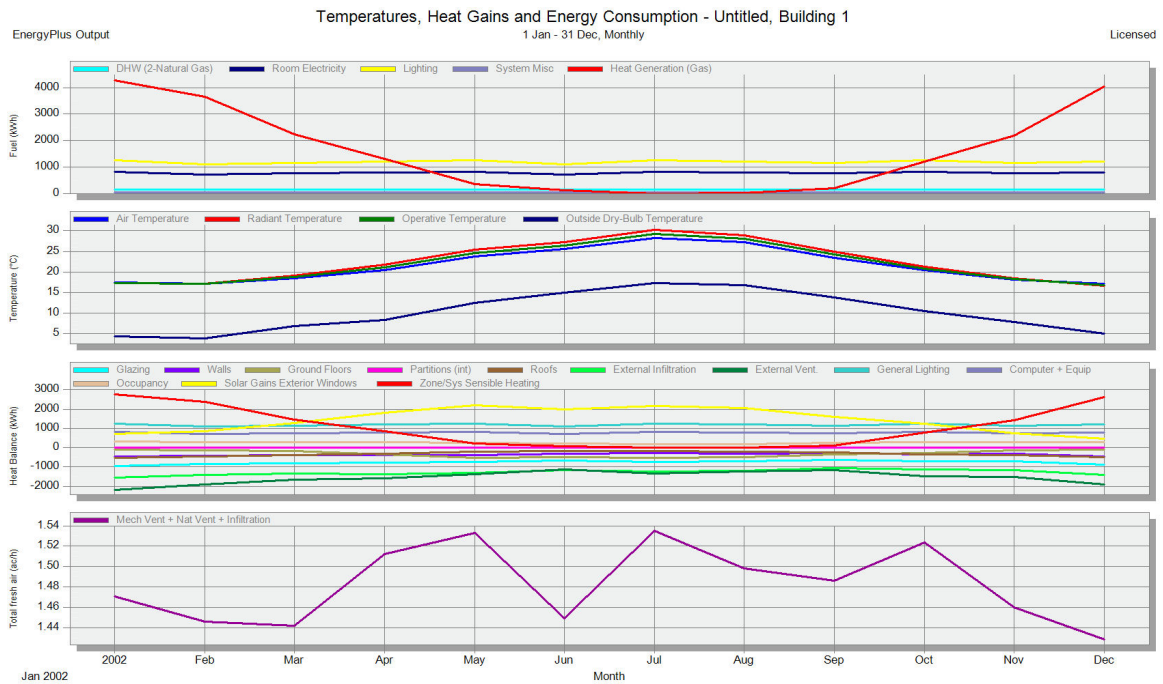
- Create a **new project** based on the **Default 20x10 2-Zone** model with **London Gatwick**.
- Run a simulation on this model as is changing the default simulation output options by requesting **monthly**, **daily** and **hourly** results and some additional output:
 - **Comfort**
 - **Temperature distribution**

Choose an **annual simulation period** (i.e. the simulation runs for the whole year) You should not generally request hourly results when the simulation is

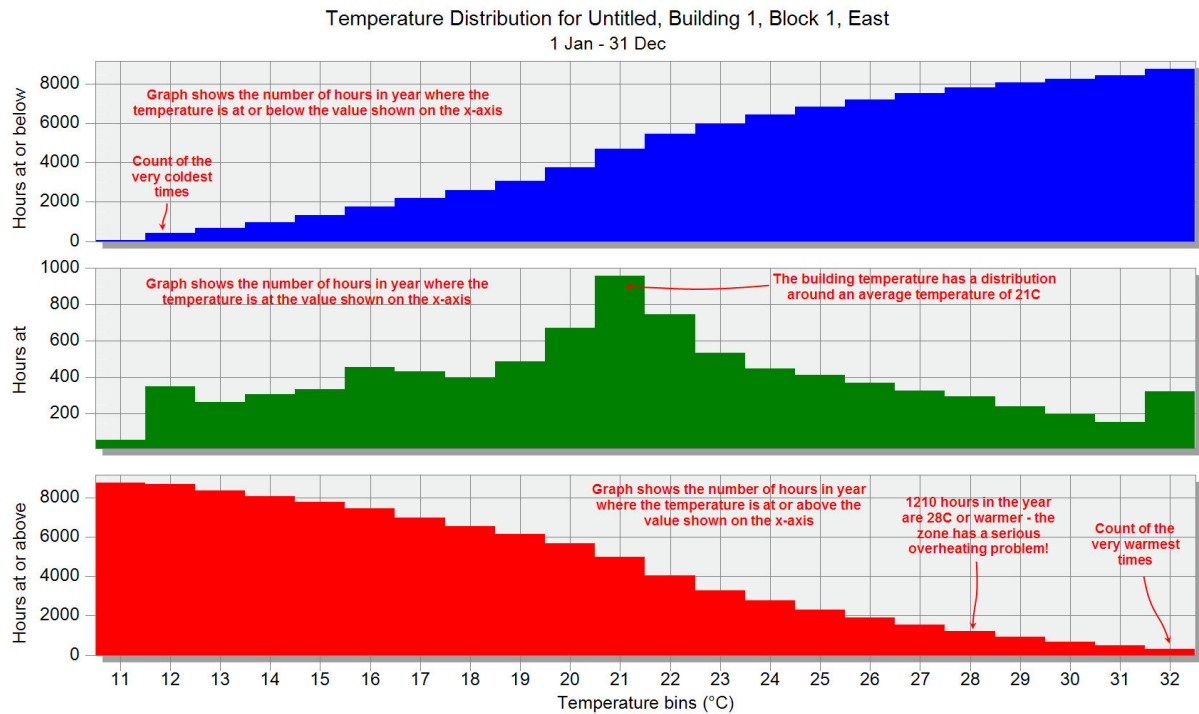
for a whole year because large amounts of data are generated, but in this case it is a small model so hourly results are fine.

- You should see results like those below when at building level you select All data for **Hourly**, **Daily**, **Monthly** and **Annual** intervals.





Also go to the East and select Distribution as the interval and you should see the graph below.

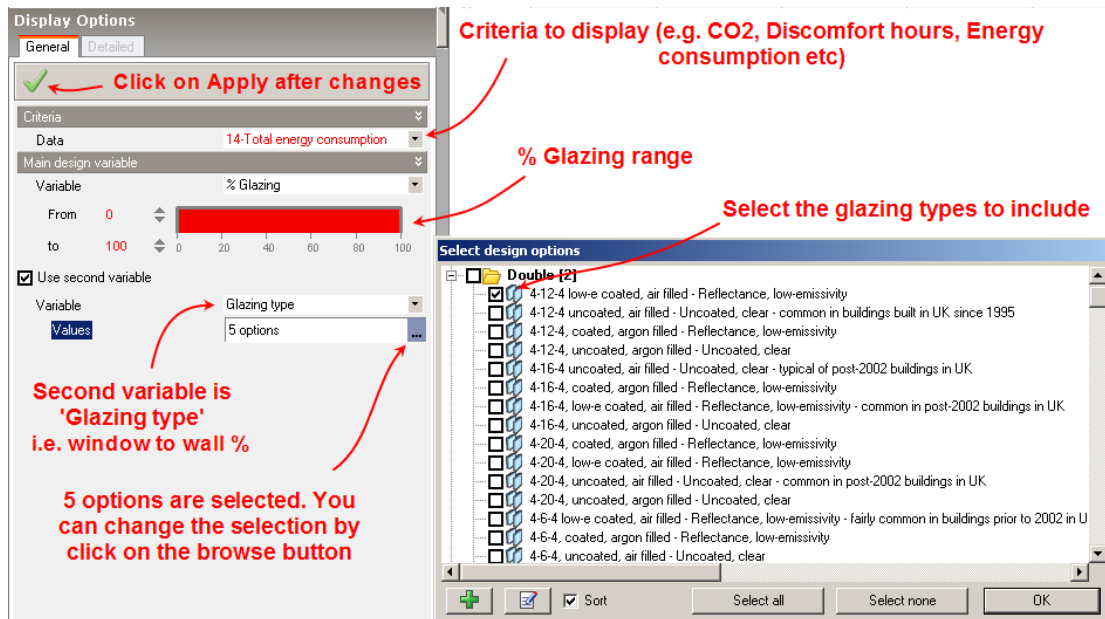


Temperature distribution graphs can provide a useful ‘at a glance’ picture of the range of temperatures experienced by the virtual buildings’ occupants. Data such ‘number of hours at or over 28°C appear in many national codes.

Parametric Simulations

Design curves can be very useful at conceptual stages of the design process to understand how building performance is affected by variations. DesignBuilder allows you to generate curves to show how performance parameters like CO₂ emissions and comfort vary as key elements of the building design vary. You can select up to 2 design variables. For example you might want to plot the cooling energy consumption for the base design with a range of **window to wall %** and **glazing types** variations).

The selection of the design variables is done in the Display options panel in the bottom left of the screen. See below.



The design curves are generated by automatically running multiple simulations of the base design (i.e. the model defined on the edit screen) with variations applied.

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 above example would be different combinations of Glazing type and Window to wall %. In all, $6 \times 5 = 30$ simulations would be required to generate the design curves in this example.

*Note in the early betas v.1.9.0 the standard base case design must be simulated including **monthly** data before going on the Parametric screen.*

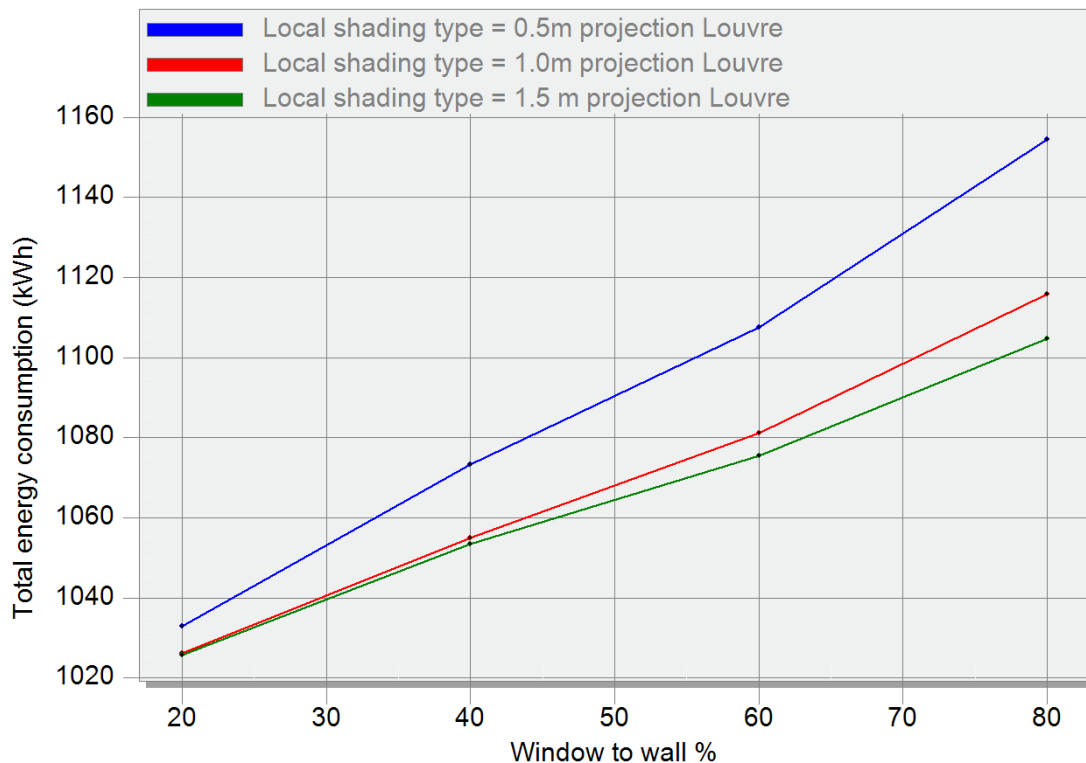
Tip: Parametric simulations can take a long time to complete and DesignBuilder can keep grabbing focus away from the window you are working in as it refreshes the progress bar. To avoid this you can minimise the main DesignBuilder window. If you have a dual core processor, you should now be able to continue working while the simulations continues.

Total energy consumption (kWh) - Untitled, Building 1

EnergyPlus Output

17 Aug - 23 Aug, Parametric Analysis

Licensed



The graph shows the diminishing returns with extra louvre projection from 1m to 1.5.

Typical uses for parametric simulation are:

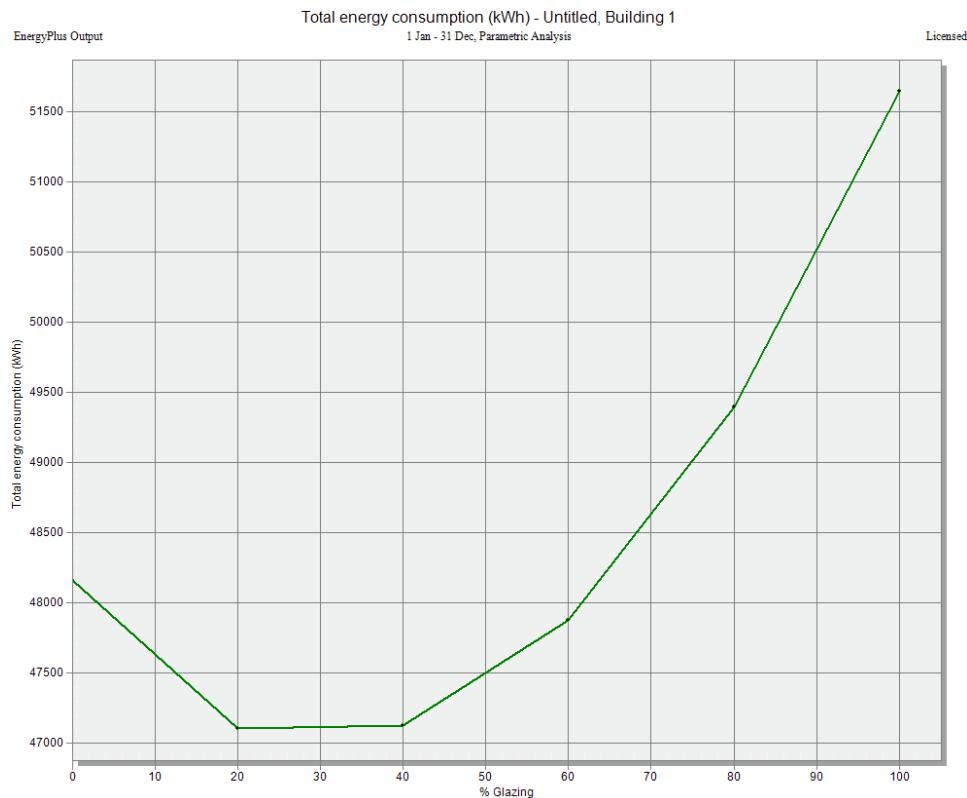
- Generating **design curves** to assist with decision making at early design stages.
- As a **teaching aid** to help **students** understand the impact of variations in the design.
- To help **communicate** concepts with non-experts in building simulation, clients, architects etc. **Architects** particularly appreciate the visual nature of the parametric data.
- Design **optimisation** with 2 variables. The exercise below shows an example of this. The DesignBuilder developers are part way through a research project to investigate the potential for true multi-variable optimisation in building design.

Exercise – Parametric simulation - Basics

- Create a new project using **London Gatwick** location and the **Default 20x10 Single Zone** template.
- At building level go to the HVAC tab and switch on **cooling**.
- Click on the **Simulation** tab and run an annual simulation. *Be sure to request monthly results to avoid a bug in the v.1.9.0 beta.*
- When the annual simulation has finished click on the **Parametric** tab. The parametric simulation will start automatically and by default you should find that **Window to wall %** is selected as the first variable and the second

variable is not selected. If so, allow the parametric simulation to complete. If not cancel the simulation, set the parameters as they should be and restart the parametric simulations by clicking on the Apply toolbar button in the Parametric Display options.

- Set the criteria data to be **Total energy consumption**.
- Click on the Apply toolbar icon to start the simulations. When they are finished should see the graph below.



It shows how the total energy consumption varies with changing window to wall %. With very large areas of glazing, winter heating and summer cooling are both required which causes high energy consumption (top right of graph). Zero glazing fails to make best use of free winter solar gains and it doesn't give the best solution either. Total energy consumption is at it's minimum when there is between 20-40% glazing where the best trade off between getting some winter solar heating and avoiding summer cooling loads is found.

You should understand how parametric simulations work to get the most meaningful results and avoid common pitfalls.

Firstly, model variants are generated internally within DesignBuilder by creating a copy of the base model and overwriting the model data with the variant data at the **building level**.

It is important to take into account how this data will be inherited to down to 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 in the above Parametric example model then changes in this data at building level would not influence the surfaces of

the design variants and the results will indicate that the Window to wall % does not influence results.

Because there is no visual check on the state of the design variants the parametric function would normally be carried out either on:

- Well structured and checked models that have been set up to be compatible with parametric analysis, that is allowing the appropriate data to inherit from the building level parametric changes. Or,
- Smaller models where it is easy to see how data inherits.

Parametric simulation results are stored with a record of the change made to the base case to create it and this allows new variants to be added to the study without invalidating previous results.

12 Timing - Schedules, Profiles, Holidays

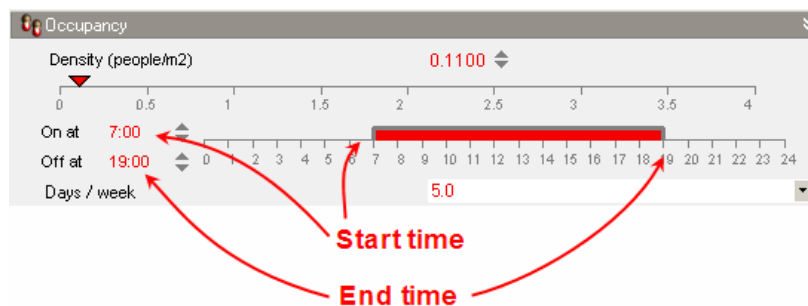
This section explains how to model the time-varying nature of building occupancy and operation. It also covers how holidays are modelled.

The timing of internal gains, heating and cooling systems and ventilation can be defined in one of 2 ways depending on the setting on the **Timing** setting on the Model options dialog:

- **Typical workday** – quick and easy but less flexible
- **Schedules** - default option – takes more time to set up but is more flexible

Typical workday

This can be a very fast and convenient way to define the timing using just a start time, end time, working days per week with seasonal variation available for heating/cooling related data. The **typical workday** approach is intended for easy early stage modelling of non-domestic buildings.

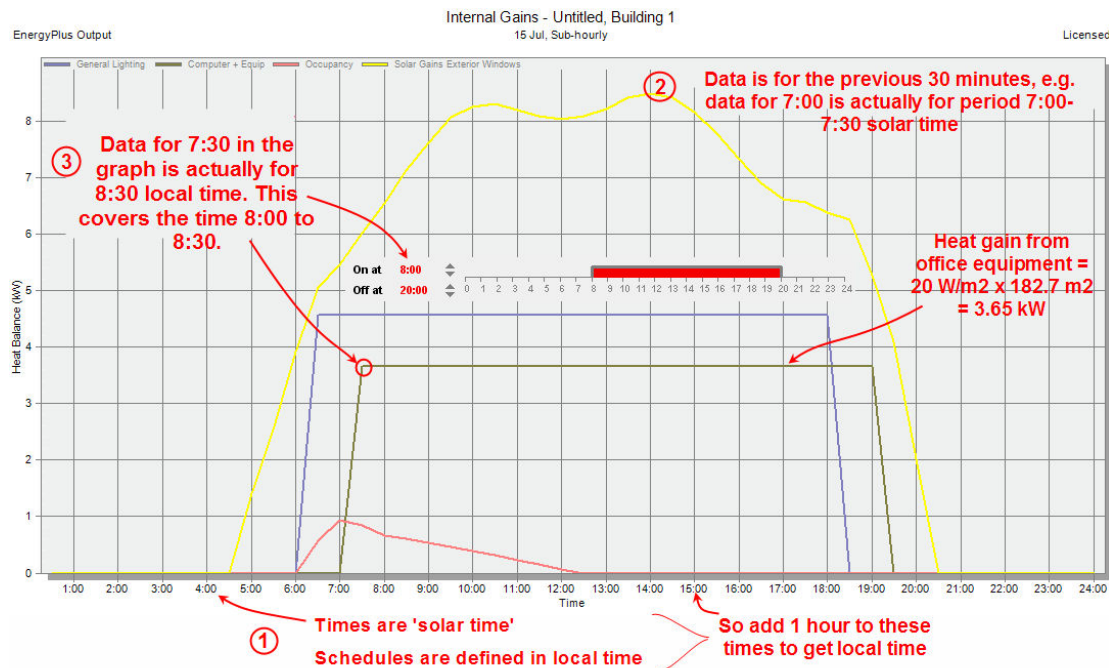


This approach can be a fast way to set up timing but it does not offer much flexibility. For cases where the variations are more complex, you should use schedules instead.

Exercise –Typical workday

- Create a new project using London Gatwick location and the Default 20x10 Single Zone template.
- On the Model options dialog and set the timing option to **Typical workday**.
- At building level on the Activity tab, under **Office Equipment** change the **On** time to **8:00** and **Off** time to **20:00**.
- Now run a Cooling design calculation by clicking on the **Cooling design** tab at the bottom of the screen.
- Use default calculation options, select just **4-Internal gains** results and look at results for **Computer + Equip**.
- Notice that the maximum heat gain is the 20 W/m2 requested x floor area of $182.7\text{m}^2 = 3.65\text{ kW}$.
- The equipment timing of the Cooling design and Simulation graphs requires some explanation. The data displayed on the graphs is for solar time, i.e. with no modification for daylight saving. So at midday the sun has maximum altitude. Also data is displayed for the previous 30 minutes in Cooling design results (or previous 1 hour when viewing hourly simulation results). So

Computer + Equip results at 7:30 actually represent the heat gain to the space over the time 7:00-7:30 solar time or 8:00-8:30 local clock time. Check that this is correct based on the input.



Schedules

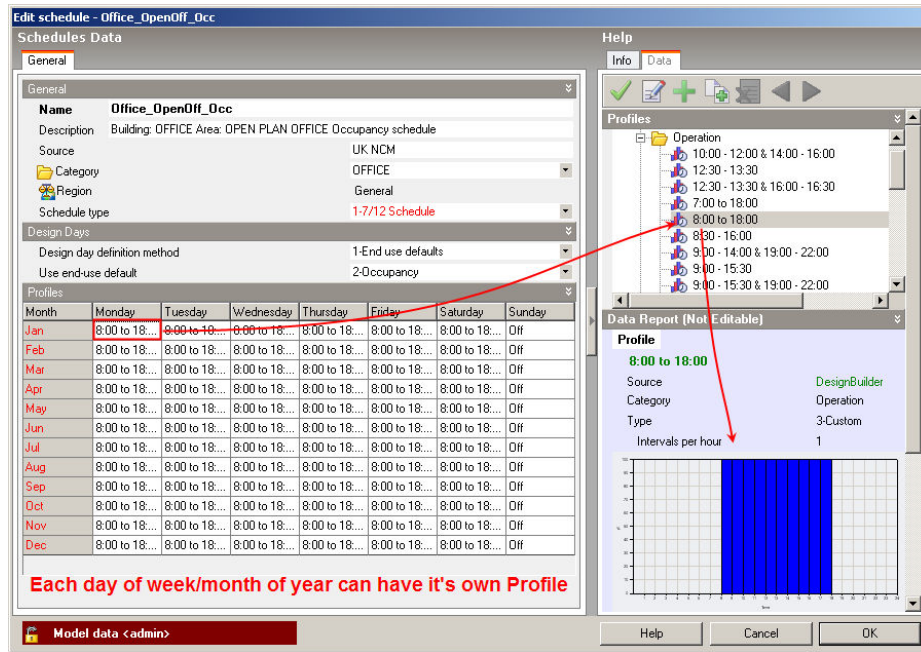
Schedules are more flexible than Typical workday and because they are separate components they can be exported for use in the Library and in other projects.

Schedules are defined in one of 2 ways:

- 7/12 Schedules – graphical interface but with limited flexibility.
- Compact schedules – text interface with much flexibility.

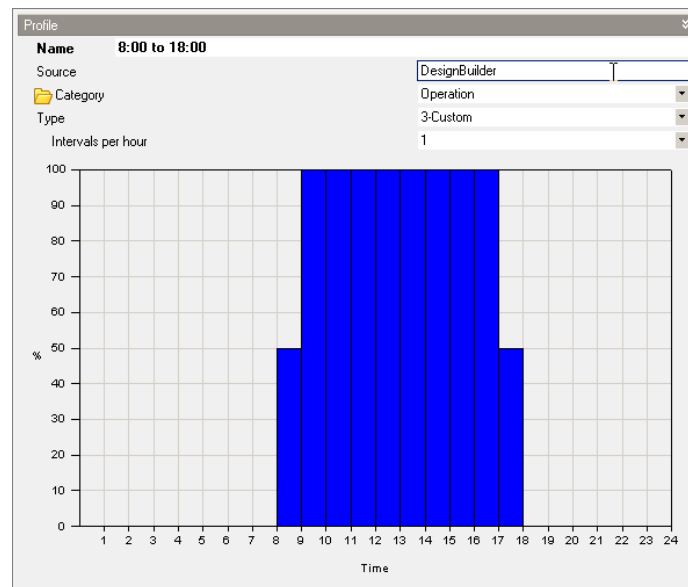
7/12 Schedules

7/12 schedules allow you to select a different daily **profile** for each day of the week and each month of the year.



Profiles

Profiles are separate components allowing a daily profile value between 0 and 100 to be defined over the day. This example profile show variation from 0 at night to 100% during the day with the times 8:00-9:00 and 17:00-18:00 being at 50% operation. The value can be interpreted as the % operation of the equipment (if the profile is used to control equipment) or % occupancy if it is occupancy being modelled etc. The profile can only be used as a component in a 7/12 schedule.



There are 3 types of profile:

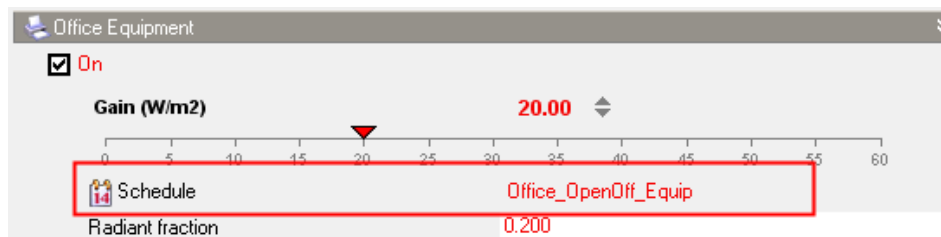
- Constant – a constant value all day.
- Single period – a single on period define using start and end times (controls are like typical workday).

- Custom – define any variation using the mouse (instructions for editing the data are shown on the info panel).

7/12 Schedules are the easiest way to define schedules but lack some flexibility because every Monday in January must be the same (for example).

Example – 7/12 Schedules

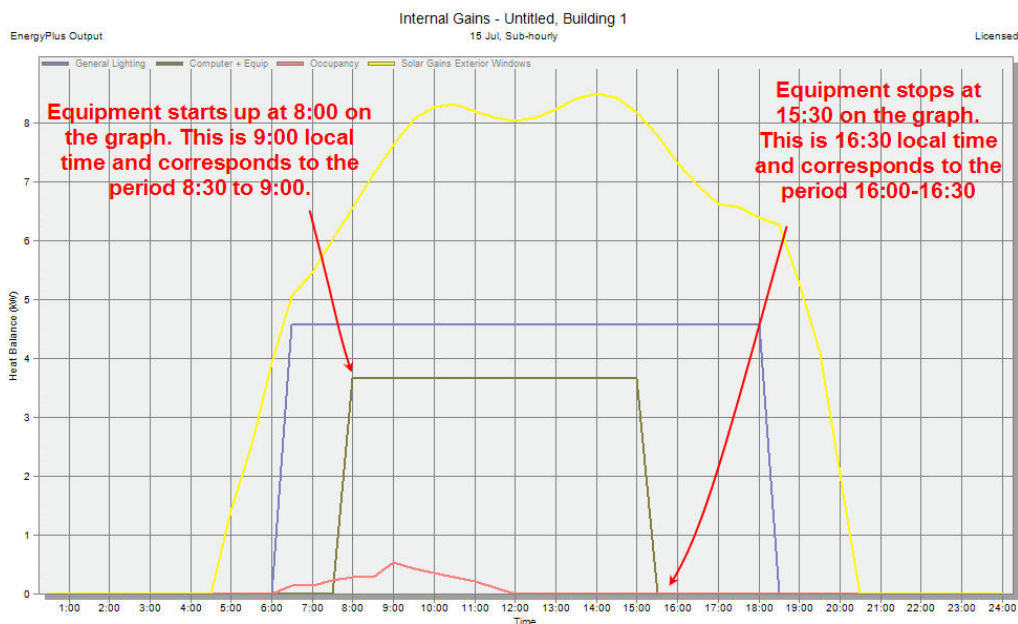
- On the Model options dialog set the **Timing** option back to **Schedules**. And go back to the Activity tab, building level. Notice how the data has changed to reflect the Schedules option.



- We are going to create a new schedule to control the Office equipment.
- Click on the Office equipment Schedule to open the list of schedules in the Data panel on the right then click on the + **Add new item** toolbar icon to create a new schedule.
- Give the new schedule a name e.g. **Occupancy schedule**. If the schedule may also be used to describe the timing of other equipment or occupancy, give it a correspondingly general name (e.g. My Project Occupancy Schedule).
- Leave the type as 7/12 Schedule. Leave the Design days data too. This data describes how the schedule should behave if you use the (default) **SummerDesignDay** Day of week in Cooling design calculations and when autosizing Compact HVAC equipment before the main simulation. You can read up on this topic in the program help.
- The Profiles data has a grid with days of the week along the top and months down the left. This grid of days and months gives rise to the name **7/12** -7 days per week and 12 months per year.
- Profiles are selected for cells in the grid by first selecting the cells to be defined then by clicking on the **Edit selected cells** link on the right.
- So first click on the Month cell in the top left to select all cells, i.e. all weekdays in all months.
- Then click on **Edit selected cells** to open a list of existing profiles. Select the profile **8:30 - 16:00** and press OK to select this as the profile for all cells in the grid.
- Now set the weekends to be switched off. To do this select the weekends by clicking on the **Select all weekends** link on the right. You could also have done this by clicking on the column headers for Saturday and Sunday. Now click on **Edit selected cells** and set the profile to be **Off**. The schedule grid of profiles should look like this:

M...	Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday
Jan	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Feb	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Mar	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Apr	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
May	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Jun	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Jul	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Aug	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Sep	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Oct	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Nov	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off
Dec	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	8:30 - 16:00	Off	Off

- Press OK to create the Schedule.
- The program should ask you if you would like to select this new schedule. Respond 'Yes'. If it does not ask then select it in the usual way.
- Now run another Cooling design calculation by clicking on the **Cooling design** tab at the bottom of the screen. Note that because we have edited the model since the last set of results were generated the results are deleted and the Calculation options dialog is shown.
- You should see results like those below.



- Check that these results correspond with your input.

Compact Schedules

Compact Schedules directly use the EnergyPlus **Schedule:Compact** data format to provide a very flexible way to define time-varying quantities.

Compact Schedules are defined using a slightly modified version of the standard EnergyPlus Schedule:Compact format documented below. It can be easy to make a mistake in format so Compact schedule data is checked for correct syntax in the dialog.

Using the Compact Schedule definition all the features of the schedule components are accessed in a single command. 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.

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 do 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 DB. There is no problem with this, just a methodological concern that if the 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.

Field: 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 that only Month-Day combinations are allowed for this field.

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,
```

Field: 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'.

Field: 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).

Field: 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 WinterDesignDay,  
Until: 07:00, 0,  
Until: 19:00, 1,  
Until: 24:00, 0,  
For: Weekends,  
Until: 24:00, 0,
```

```
For: Holidays AllOtherDays,  
Until: 24:00, 0;
```

This schedule could be used to describe the lighting operation in an office. It indicates:

- Schedule is a fraction (0-1).
- No seasonal variation.
- Weekdays and cooling and heating design days the schedule is 1 (i.e. 'on') between 7am and 7pm and 0 (i.e. 'off') other times.
- Weekends, holidays and all other days the schedule is 'off'.

Example 2 - Temperature Setpoint Schedule

```
SCHEDULE:COMPACT,  
Bedroom_Cool,  
Temperature,  
Through: 31 Dec,  
For: Weekdays SummerDesignDay WinterDesignDay,  
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 AllOtherDays,  
Until: 24:00, 0;
```

This schedule is could be used to describe the cooling temperature setpoint and setback in a bedroom. It indicates:

- Schedule is a temperature (degC).
- No seasonal variation.
- Weekdays and cooling and heating design days 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 and all other days the cooling is off.

Setpoint temperature schedules

Setpoint temperature can vary with the time of the day and setpoints can take on 3 values: **On**, **Off** and **Setback**.

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)
$0 < \text{Value} \leq 0.5$	Equipment is controlled using the setback temperature setpoint data (Activity tab)

Notes

- When using 7/12 Schedules, for 0.5 read 50% etc.
- 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.

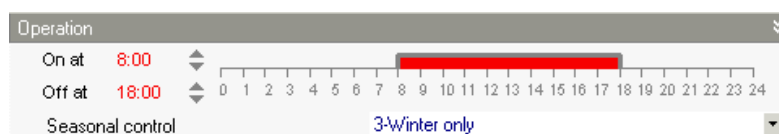
Tip: If the setback temperature set does not appear to affect results, this may be because the heating/cooling operation schedule selected uses only 0 and 1, not the 0.5 required to access setback temperature (see table above).

Typical Workday

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 example below shows the heating system operation data and you can see that the equipment is switched on between the On and Off times (8am and 6pm). During this time the heating equipment is controlled using the main temperature **setpoint** as set on the 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. Days that are not working days are weekends.

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.



Holidays

You can add holidays to the simulation 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 and are additional to any holidays, which may have been defined, in Compact schedules (Schedules Timing model option).

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.

If there are insufficient holidays in the holiday schedule you will receive an error message to this effect prior to a simulation.

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. The value of Compact schedule output during holidays can be defined explicitly using the Holidays day type. 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.

13 Glazing and Solar Shading

This section examines the processes involved in modelling glazing and solar shading with a particular emphasis on avoiding overheating and reducing cooling loads.

The data for all openings is set on the Openings tab. There are essentially 2 ways to define the layout of openings on a façade:

- Use the Default facades mechanism.
- Draw the openings at surface level.

Modelling Facades

Default facades provide a fast and easy way to define some typical window layouts. The options are:

- **None** - there is no glazing.
- **Continuous horizontal** - glazing is generated in a continuous horizontal strip using Sill height, Window spacing and Percentage glazing.
- **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 % (WWR) but prioritises window height.
- **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 %.
- **Fixed width and height** - windows have fixed width and height. This option uses Window to wall % but prioritises Window Width, Height, Spacing and Sill height.
- **Fill surface (100%)** - the entire surface is filled with glazing and there is no frame.

You can use the Window to wall % (WWR) data and Window height, Window spacing and Sill height to easily change the default amount of glazing and its layout on the walls.

The **Preferred height** option is very versatile because it will prioritise the glazing area over the height of the windows, whereas **Fixed height** always gives a fixed window height and does not guarantee giving the requested % WWR.

This approach to defining facades is very useful at early design stages when the exact configuration of the façade may not be known and the design team are talking in terms of WWR. When the design is more established you might wish to include the detailed positions of windows/doors etc and this usually involve drawing individual openings at the surface level.

Drawing openings at surface level

When at the surface level drawing tools become enabled to allow you to draw:

- Windows
- Vents
- Doors
- Sub-surfaces
- Holes (this is the only way to add holes)

The process involved with drawing openings at surface level is described in the help.

Once a custom opening exists (even if it then is deleted) default glazing, vent and door layout model data is ignored. You can clear non-default façade data by using the **Clear data** command.

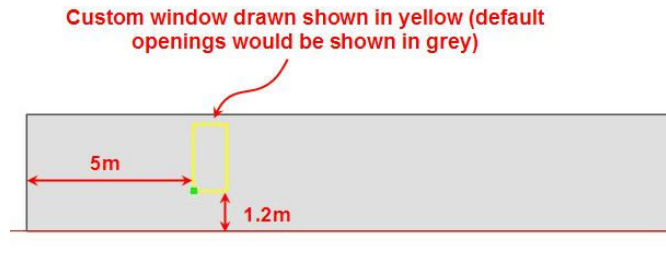
Flat roofs do not have default glazing set up using Roof glazing layout model data. To create openings on a flat roof you should go to the surface level and draw them manually there.

Example – Draw windows at surface level

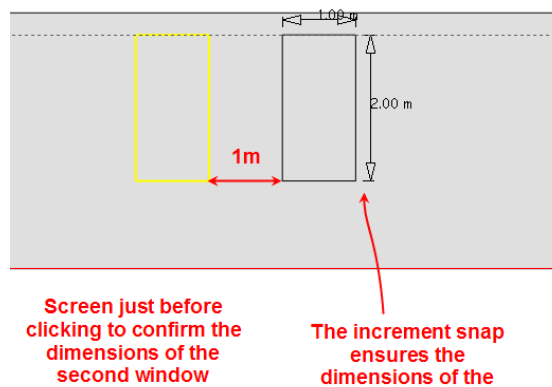
- Create a new project based on the **Atrium example base** template and using London Gatwick location.
- At building level on the Openings tab remove all of the default openings in the main building (Ground floor and First floor blocks) using one of these methods:
 - Set the **Window to wall %** to **0**, or,
 - Load the **No glazing** external glazing template

Note that the custom openings already set in the atrium will be unaffected by changes in façade settings made at the building level.

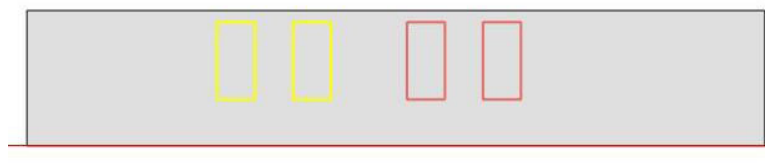
- Go to the West surface of Ground floor > Zone 1,
- Click on the **Normal** button to take the Normal view looking from outside.
- Create a window using key ins. First move the mouse so it is near the lower left corner of the surface and is still over the edit screen (it must be in the edit screen to allow key ins to work)
- Ensure the default **2-Rectangle** shape is selected.
- Type **5 1.2 <Enter>** to position the lower left corner 5m in the x-direction and 1.2m up.
- Type **1 2 <Enter>** to create a window that is 1m wide and 2m tall. Your screen should look like this:



- Now draw another window. With **Increment snap** switched on move the mouse cursor so it hovers over the lower right corner of the first window and move to the right. You will see a dimension line appear. Start the next window 1m to the right of the first window same height of 2m.



- Now select both these windows together and clone them creating the copy 2m from the rightmost edge of the second window (5m from the left edge of the first window).



- Now repeat to create a third pair of windows. The surface should look like:

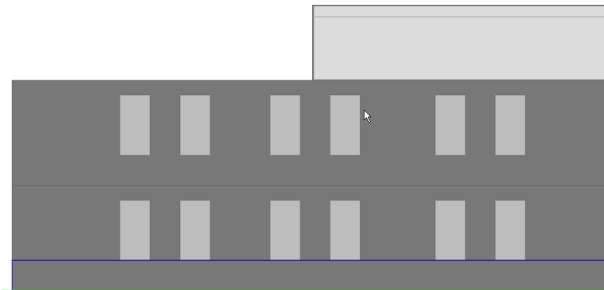


Editing Openings at building level

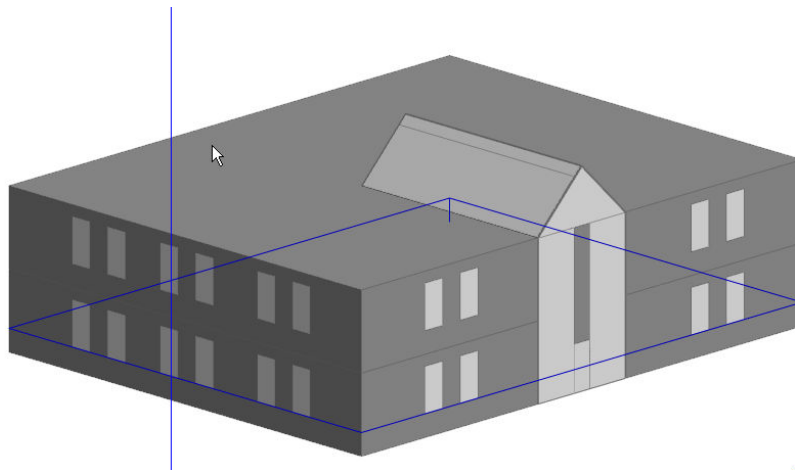
You can **copy**, **move** and **delete** existing openings when at building level. You can't draw openings at building level in v.1.

Exercise – Copy openings at building level

- Continuing from the previous exercise. Go to the **building level** and copy the 6 windows from the Ground floor to the equivalent positions on the First floor West-facing zone, 3.5m up. The West façade should now look like this:



- The next task is to copy these 12 windows to the North and East Facades. As a guide to help position in the z direction, first draw a construction line at 1m all the way round the building (see above).
- Then select all these 12 windows and start the **Clone** command and copy them.



Then copy a single pair on each south-facing surface as shown above.

Save this model as a template calling it **Atrium example custom glazing**

Tips

- When at building level you can select blocks or openings. If you want to go directly to a surface you can double-click on one of the openings to take you there.
- When trying to select a block sometimes it is difficult to avoid selecting openings instead. To force selection of a block hold the **<Shift>** down before clicking.
- You can select all of the windows on one of the facades and move them together.
- You can also **delete** and **move** openings at the building level.

Solar Shading

DesignBuilder allows most of the EnergyPlus solar shading mechanisms to be modelled including:

- Window shading devices placed directly in front of, behind or within the panes of the windows. These types are provided:
 - Internal blinds (diffusing and blades)
 - External (diffusing and blades)
 - Mid-pane (diffusing and blades)
 - Electrochromic shading systems where a pane in the window darkens when activated by a control system.

All of these have a range of methods to control window shading device activation: schedule, solar, temperature, lighting control etc.

- Local shading devices model obstructions local to the glazing system including:
 - Overhangs
 - Sidesfins
 - Louvres

You can define the surface reflection properties of the local shading devices but they are fixed and cannot currently activate based on controls.

- Different glazing types, including solar control glazing
- Component blocks with time-varying transmission (can allow trees with seasonal foliage to be modelled)
- Assembly blocks allow custom shading systems to be easily set up and modified even in large buildings.
- Building self-shading and reflection (if the reflection solar option is selected).

Window shading devices

Window shading data can be accessed on the Openings tab under the External Windows > Shading and Roof Windows/Skylights > Shading headers.

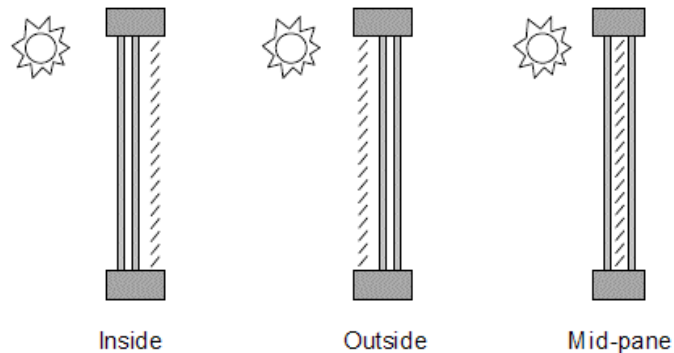
Internal windows cannot be shaded using window shading devices.

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 window-shading device is positioned outside.

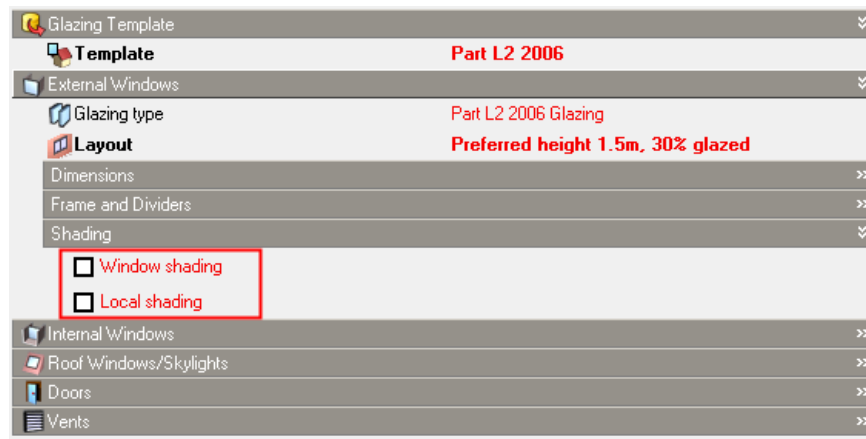
- **4-Switchable** - electrochromic glazing in which the outer pane is darkened based on the shading control.



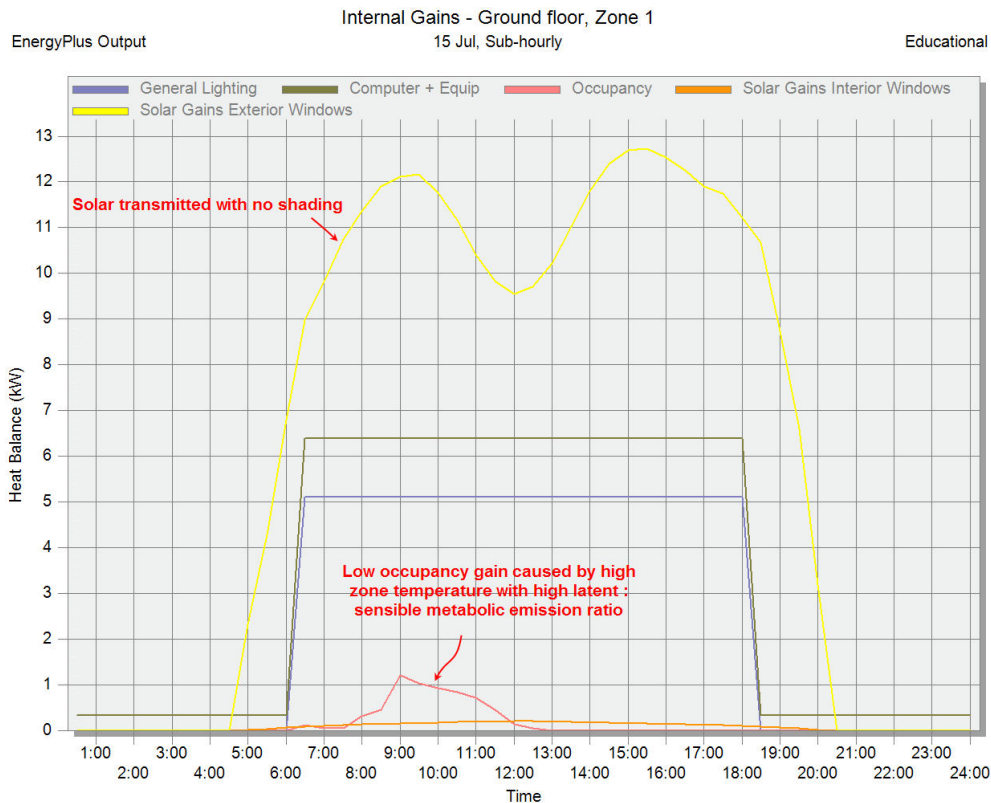
For blinds and shades, when the device is **on** it is assumed to cover all of the window except its frame; when the device is **off** it covers none of the window. The shading device does not affect the wall that the window is on.

Example – Window shading

- Create a new project based on the **Default 20x10 Single Zone** and **London Gatwick** location.
- First run a Cooling design calculation on the base model to see the solar radiation transmitted *without any shading*. See building level screenshot below.

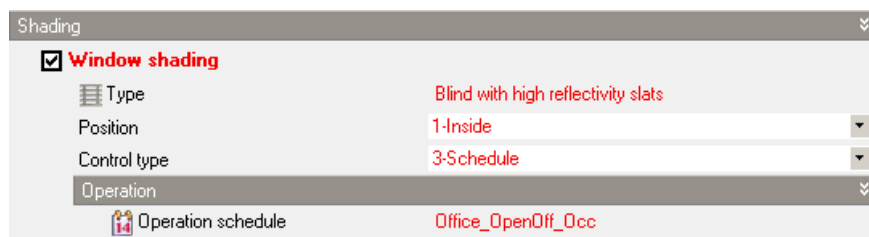


You should see the graph below after having selected Data to be **4-Internal gains** and navigated to Ground floor > Zone 1.



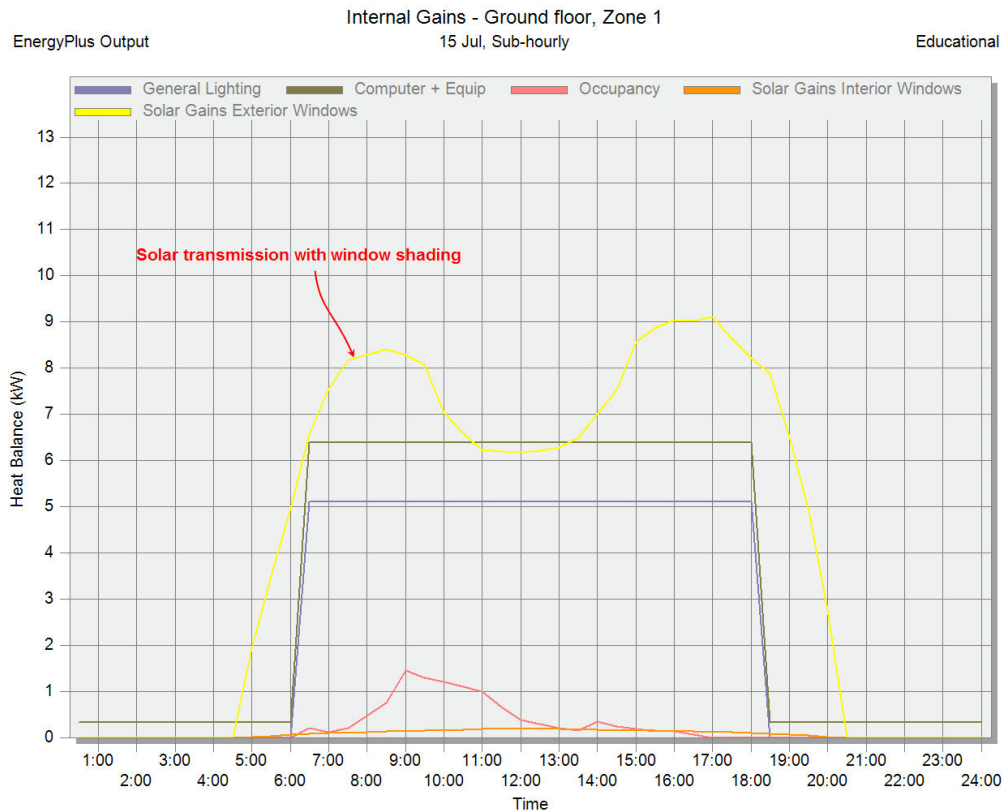
Did you notice how small the occupancy gain is? Also that it not a constant like you might expect from the occupancy schedule used to define occupancy on the Activity tab. The reason is that the zone temperature is up in the range 30-40°C where most human metabolic heat is given off as latent heat from sweating. The data shown in the heat balance is sensible heat only and will fall right down to zero when the zone operative temperature rises up to 35°C and above.

- Also **lock the Y-axis** so you see subsequent results on the same scale.
- Go back to the edit screen (press <F7>). At building level on the Openings tab open the Shading header and switch on **Window shading**. You should see:



- Click on type and see the list of blinds and shades. Note that you can create your own shading devices if necessary.
- The control type is **Schedule** so the Operation schedule is defined. In this model the blind is set to operate according to occupancy.
- Now change the control type to **4-Solar** and leave the setpoint at **120 W/m2**. This means that when the solar radiation on the window is greater than 120 W/m2 the blind operates (i.e. provides shading).

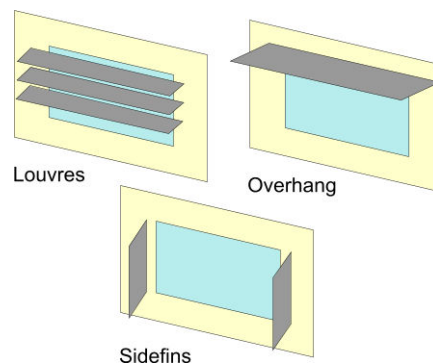
- Run a new Cooling design calculation to see the effect of this shading devices on solar radiation transmitted. You should see the graph below for Ground floor > Zone 1.



The window shading has reduced solar transmission into the zone from about 13kW to about 9kW.

Local shading devices

To include overhangs, sidefins and/or louvres check the **Local shading** checkbox. This shows the Local shading component data.



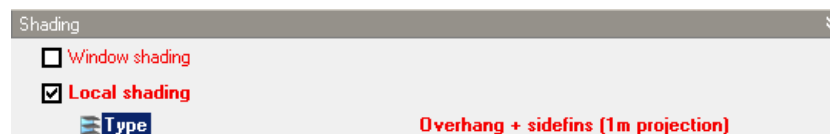
DesignBuilder comes with a small database of Local shading devices, but if you cannot find a suitable component for your model, you can create a custom local shading component in the usual way.

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** and/or **Assembly blocks**.

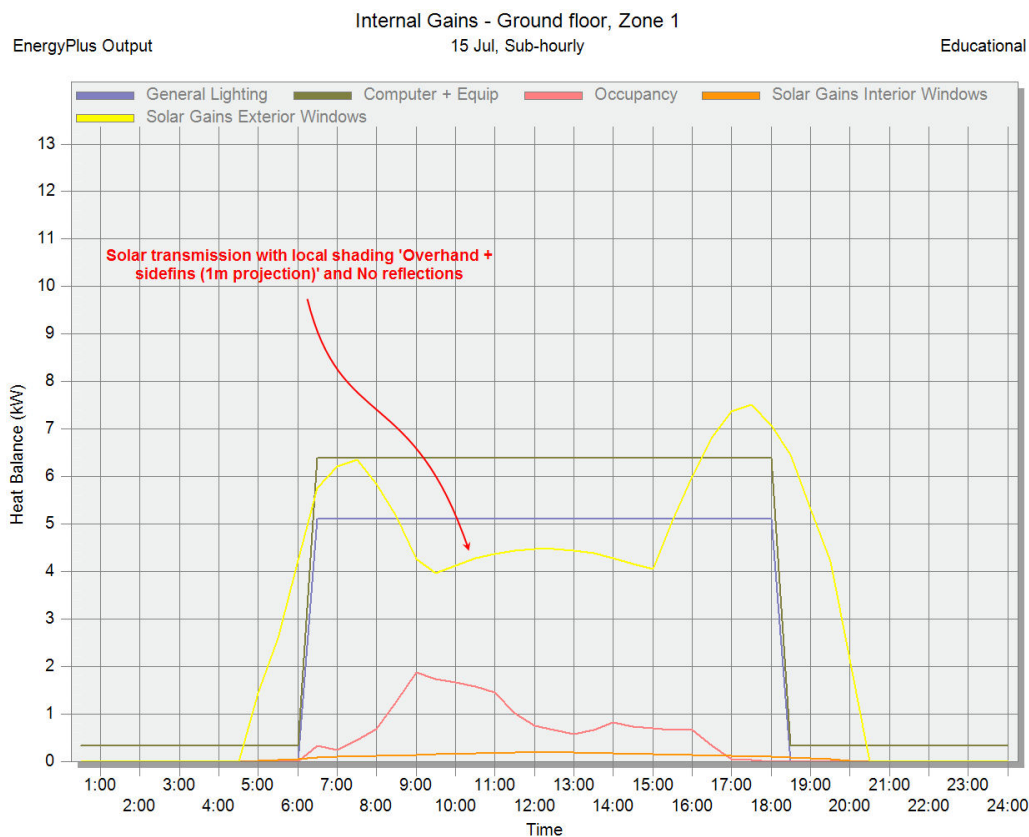
Example – Local shading

Following on from the last example, now look at the effectiveness of local shading devices.

- At building level, switch off **Window shading** and switch on **Local shading** and select the Overhand + sidefins (1m projection) local shading type. You should see:



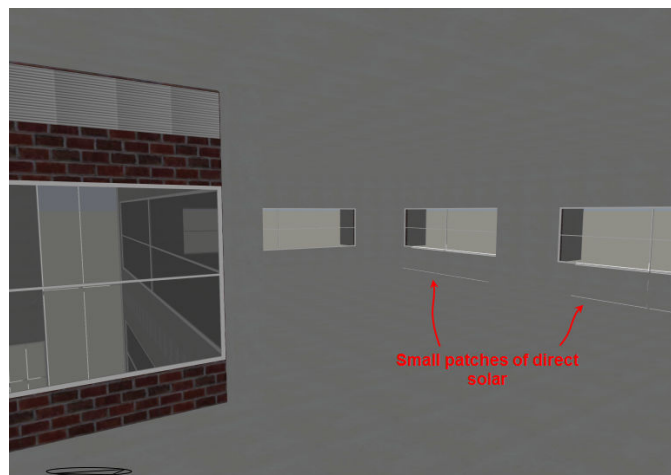
- Run a new Cooling design calculation and view **Internal gain** results for Ground floor > Zone 1:



- The graph, with y-axis still locked from the Window shading tests, shows that the Local shading device is more effective at blocking solar radiation than the window shading. Take a look on the Visualisation screen below to see why. The shading devices effectively protect the glazing from solar radiation during the hottest part of the day.

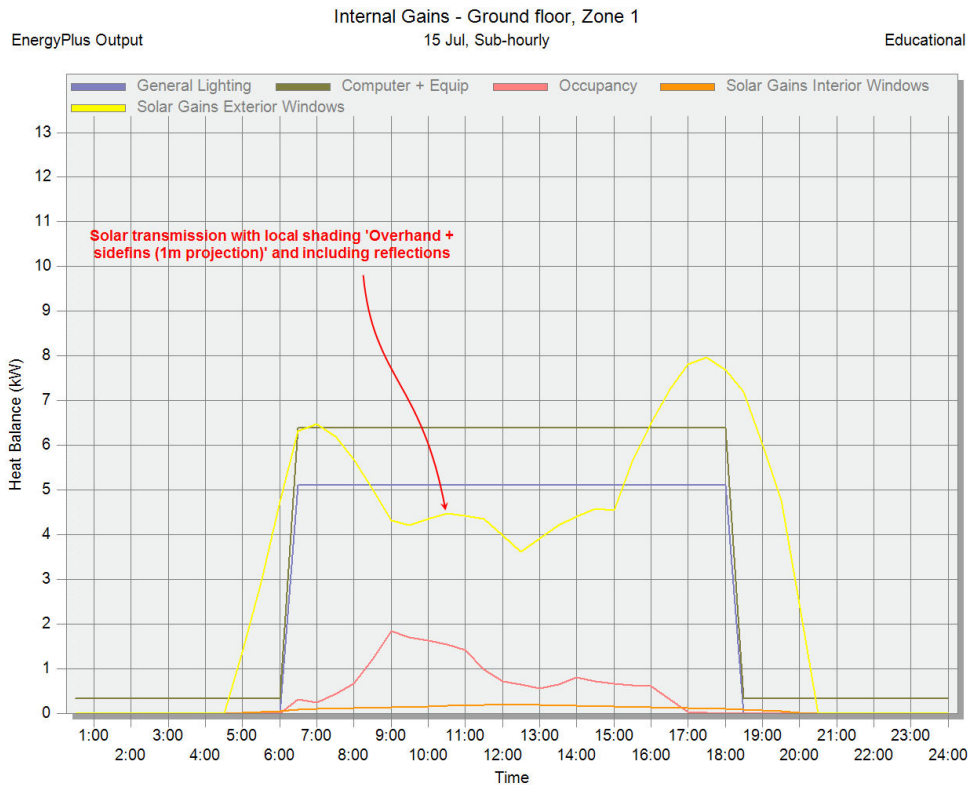


- The shot below shows that no direct solar radiation is finding it's way into the zone at 3pm when looking towards the South West (where the sun would be at the time).



A model with local shading devices or component blocks can be made more accurate by including reflections. These are excluded by default. The exception is ground reflected solar that is fully included without interception by local shading devices or component blocks when reflection is switch off. When reflections are included the shading effect of local shading devices and component blocks on ground reflected solar is included. So select this in the Model options > Cooling design tab > Solar options header > **Model reflections** data.

- Does the inclusion of reflections and blocking of ground reflected solar affect results?



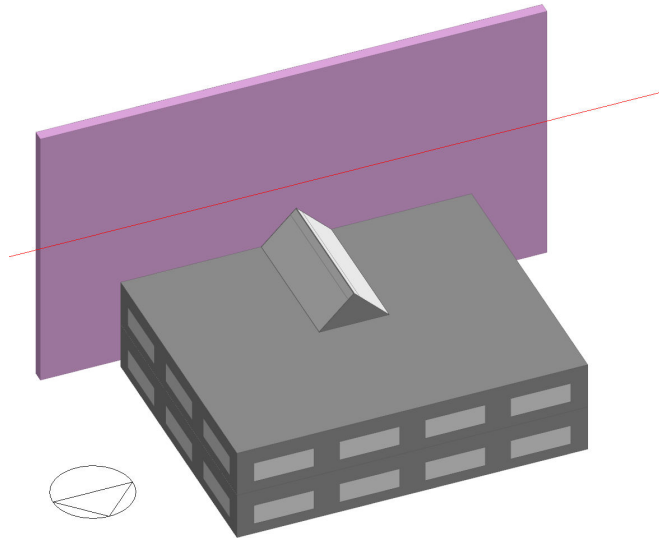
The graph when compared with the previous, suggests that the **Model reflections** option affects solar gains quite a lot. So you should ideally use this option unless your building model is large, in which case, the extra geometric calculations involved in including reflections may slow the simulations too much.

Component blocks

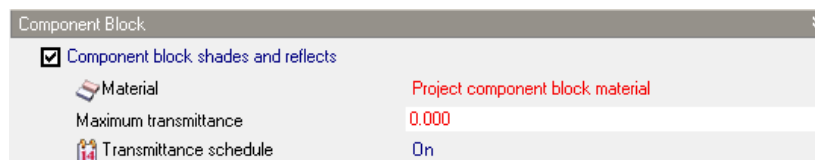
Component blocks for shading the outside of a building should be added at the building level.

Exercise – Component blocks as shading devices

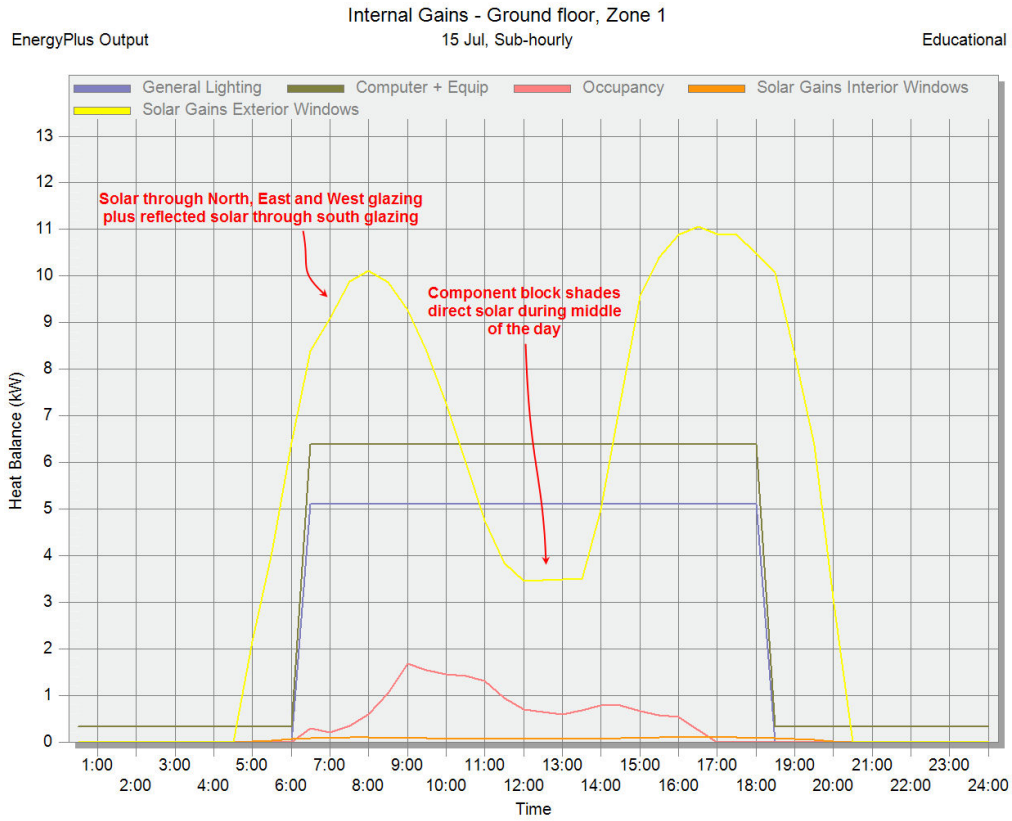
- Starting from the previous example, switch off all Local and Window shading.
- On the Layout tab draw a large component block in front of the South-facing windows as shown below. The component block should be 1m from the South wall and 20m high.



- At building level on the Construction tab open the Component Block header. You should see:



- The transmittance of 0 means that the component blocks in the model are by default opaque. You can allow a fractional opacity by entering a number between 0 and 1 for Maximum transmittance. The material defines the texture and surface properties of the component block for solar and visible reflection. Leave the default values.
- Go to the component block by single clicking on it in the Navigator. You will see that the above is the only data held for it. Go back up to the building level (press <F2>).
- Set the **Model reflections** option (if it is not already set). It is important to use the reflections option for this model because the component block should block ground reflected solar.
- Compare your results with the graph below.



14 Daylighting

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 the simulation 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: Daylight control set in zones without any windows will generate errors preventing calculations from running.

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.

If the zone has a different sensor for the back half (for example) you should check the Second light sensor option.

You may wish to use 2 Lighting Areas for these situations:

- 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.
- 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.

Important note: 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 uncontrolled core lights 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 1 or 2 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. Also 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

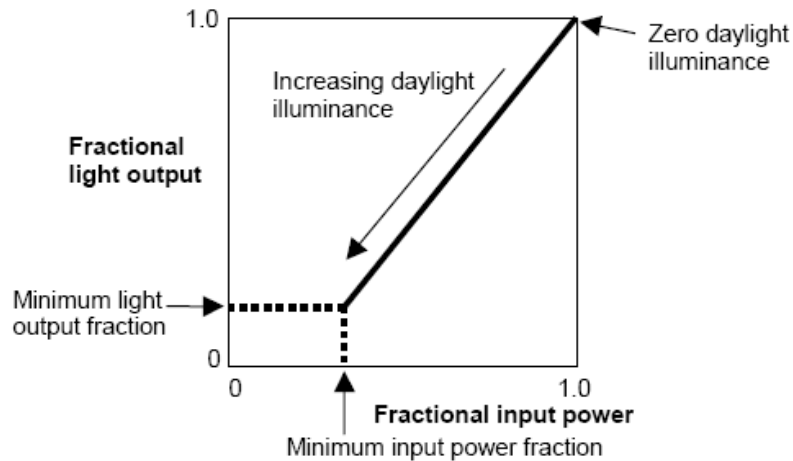
$$100 - [\% \text{Zone Controlled by Lighting Area 1}] - [\% \text{Zone Controlled by Lighting Area 2}]$$

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).

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 under-estimated (daylight savings over-estimated).

Continuous Control

With 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 lights 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 lights switch off completely.

Note: Continuous control provides an idealised lighting control mechanism which can be useful for calculating upper limits on the potential for savings using natural daylight.

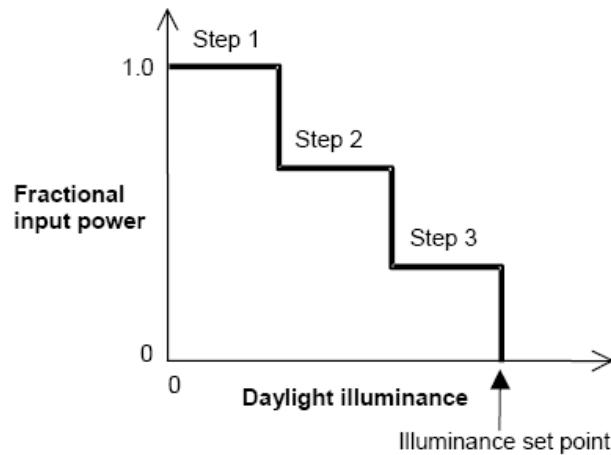
Continuous/Off Control

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

Stepped control allows you to switch lighting on/off according to the availability of natural daylight in discrete steps. Whereas the Continuous 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.



Glare

Maximum allowable glare index

If a daylit 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).

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

DesignBuilder offers a choice of EnergyPlus daylighting algorithms:

- **1-Detailed** is the default option and the most practical for most purposes.
- **2-DELIGHT** does a full radiosity lighting calculation, i.e. it bounces light beams around each zone to calculate daylight distribution. It has some limitations though.

These can be selected on the Model options dialog on the Advanced tab under Lighting.

Daylight control procedure

The procedure for introducing the lighting control is:

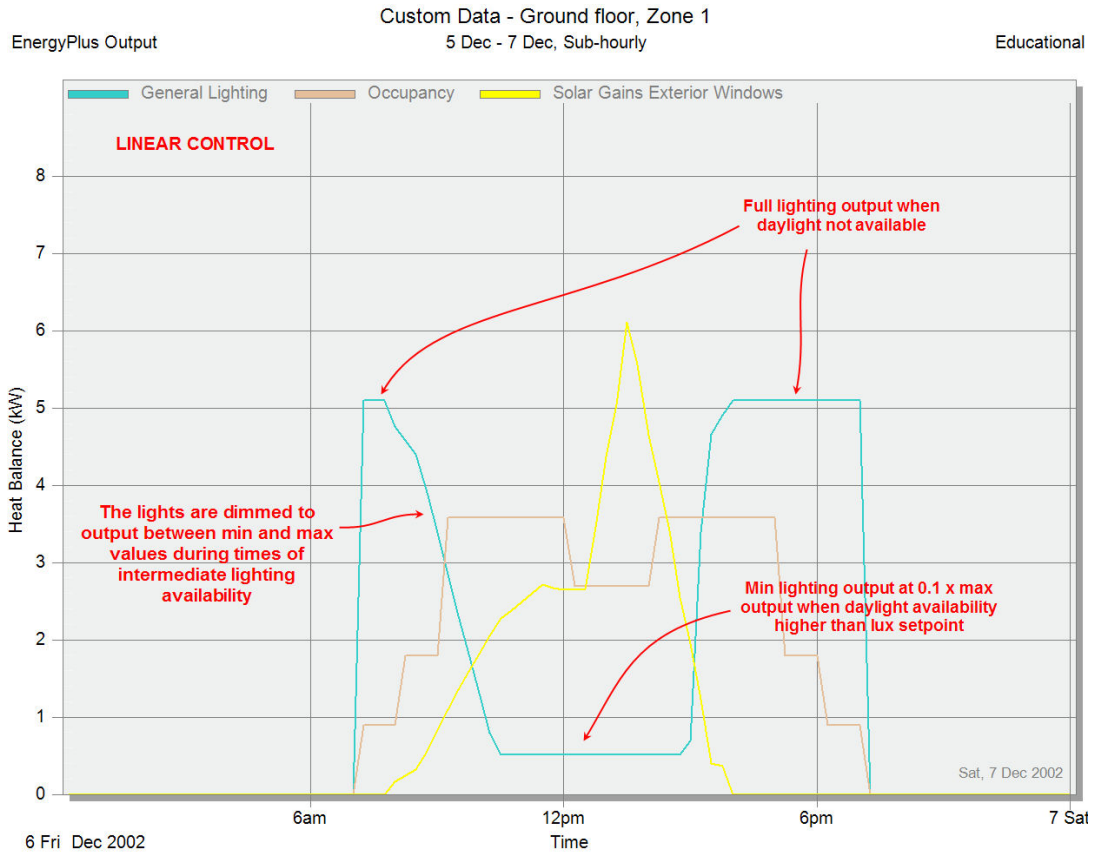
- Check the Lighting Control On checkbox (at building level if all zones are to have lighting control).
- Select the type of control as described below.

- 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.
- 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%.
- If there are two lighting areas then position the second sensor.
- 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 - %Zone covered by Lighting Area 1.
- Check for zones with lighting control specified but having no natural daylight.
- Check hourly results and make sure they make sense!

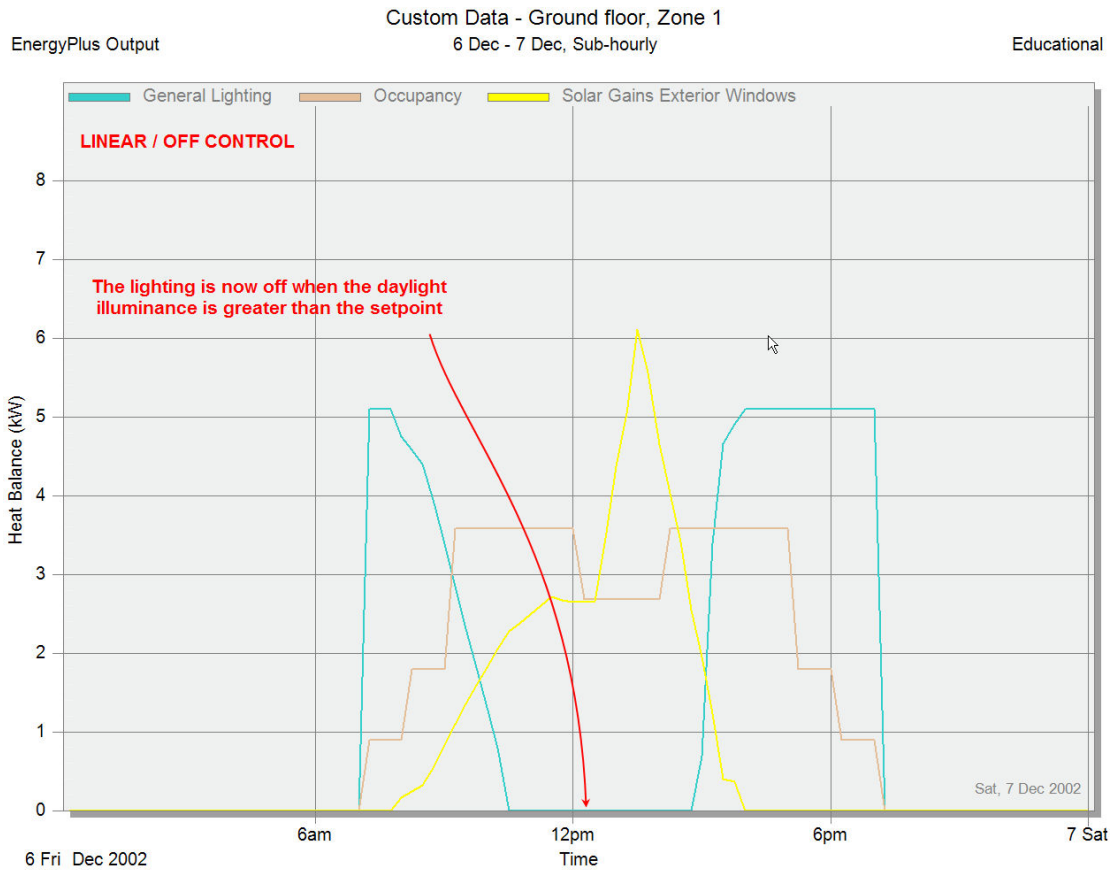
Exercise – Daylighting calculations

To test the various daylighting settings:

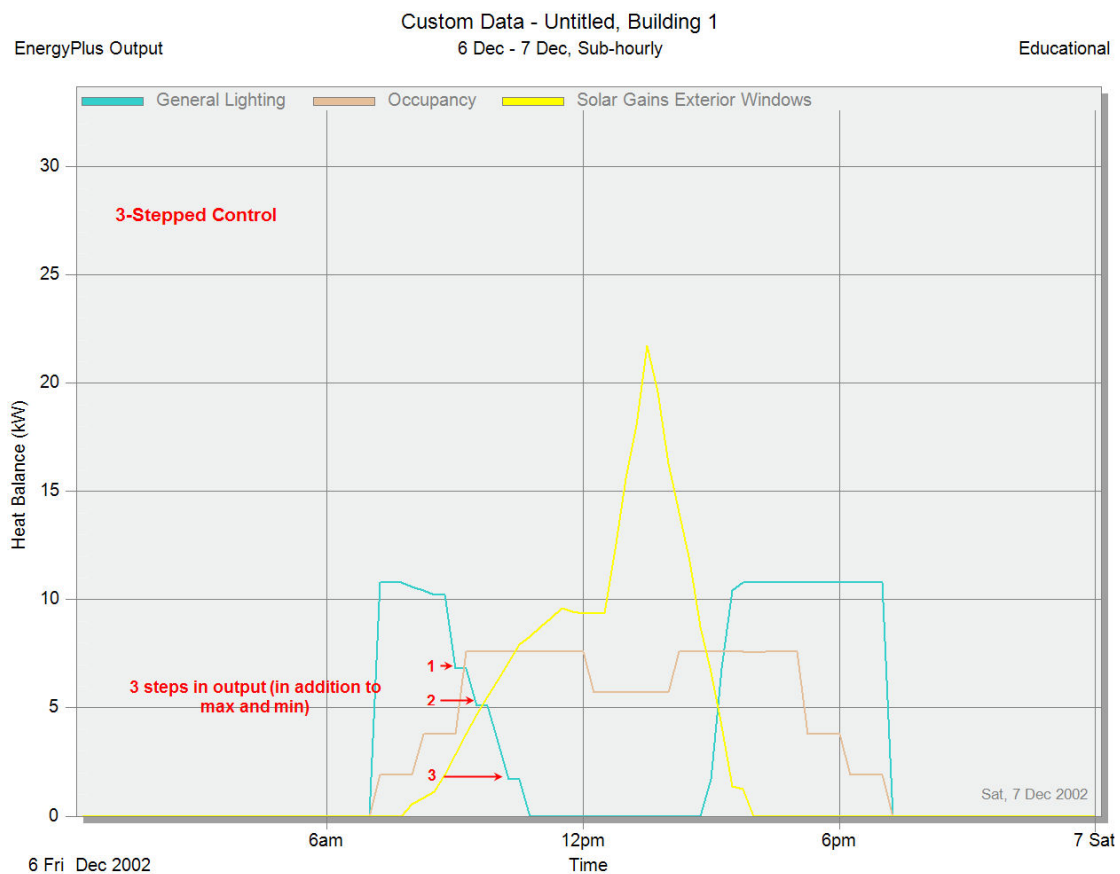
- Following on from the last exercise, at building level go to the **Lighting** tab and switch on **Lighting Control**.
- Note that the default control method is **1-Linear**. With this method you can also define the minimum output fraction of the lights when fully dimmed and the corresponding power fraction. These these in their default states.
- Now go to the **Layout** tab and notice that the **sensor position** is shown.
- By default the sensor is placed in the middle of the zone. You can move it to a specific location by selecting it then **moving** it using the move toolbar icon.
- Also keep the default the **Detailed** daylighting method and **0.8 m** working plane height (Model options > Advanced tab > Lighting header).
- Click on the **Simulation** tab and run a simulation **for 6 December** only. Select **Sub-hourly** results and set the Time steps per hour to **4**.
- Go to Ground floor > Zone 1, select the **sub-hourly** interval and show **4-Internal gains** data. Open the Detailed Display options tab and switch off the display of the **Zone/Sys sensible heating** system under the Internal gains header and save the selection as a template for later use by pressing the **Save** Display options toolbar button. The results should look like:



- Now go back to the edit screen (press **F7**), go to **building** level and change the light control settings.
- Use the **2-Linear/off** Lighting control type this time and run a simulation choosing the same calculation and display options as before, you should see:



Now test the Stepped control method using 3 steps.



Optional Exercise - DELight

Use the EnergyPlus tab of the Program options dialog to change the version of EnergyPlus used. If you have been able to get the exe version, go to the Advanced tab of the model options dialog, open the Lighting header and change the Daylighting method to **2-DELight**. This will use the advanced DELight calculation method using ray-tracing to calculate the daylight illuminance.

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 glazing systems (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 **> 2 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 modelling 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.

Compare the results using **DELight** with those calculated using the **Detailed** daylighting method.

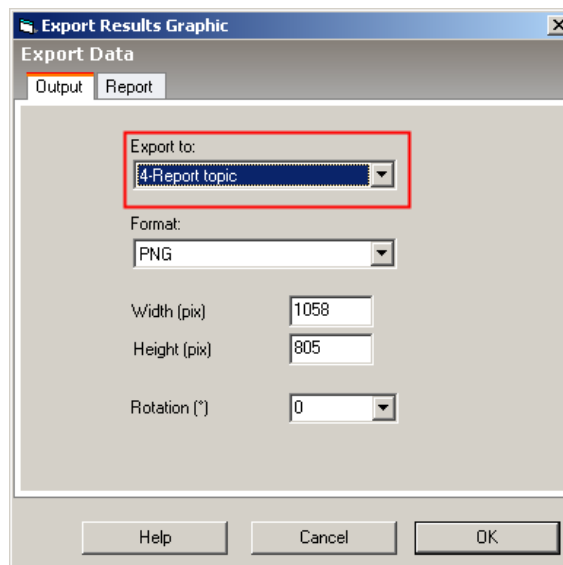
15 Comparing results

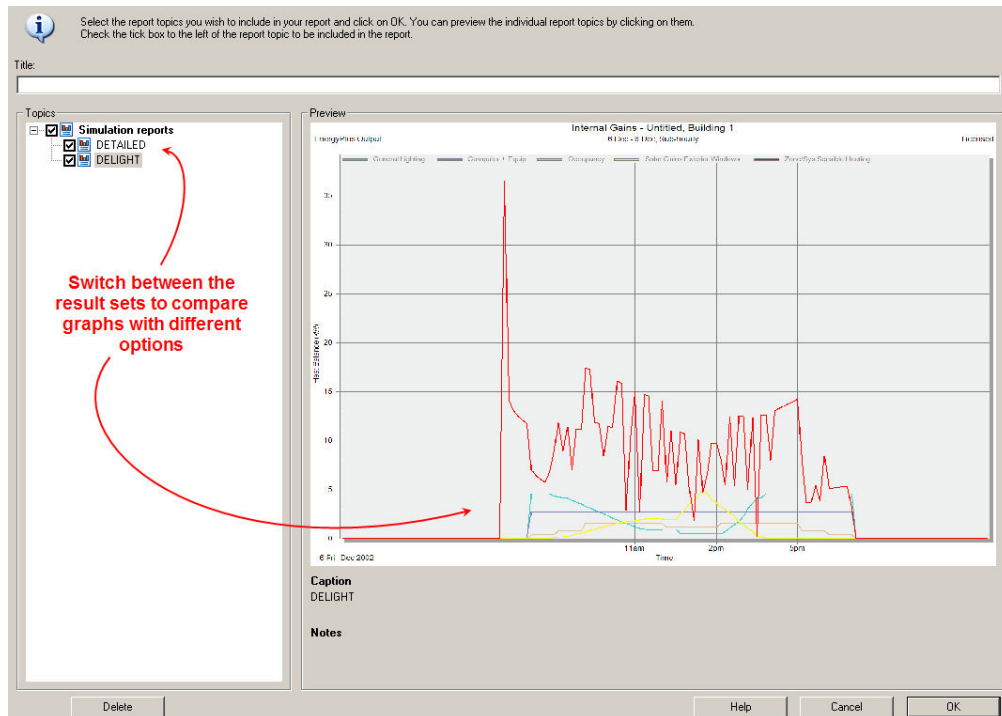
There are 2 ways to compare results between different design variants and they are illustrated by comparing between the results for the 2 Daylighting calculation methods discussed earlier. The 2 comparative reporting methods currently available are:

- **Report topics** and
- **Different buildings**, one for each variation.

Report topics

The Reporting mechanism allows you to collect multiple result sets using the Report topic export option. Export graphs shown on the screen as **report topics** and compare results later in the **Compile reports** dialog. See screenshots below.

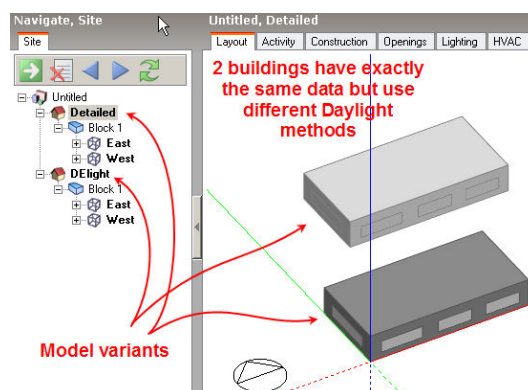




The Compile reports dialog above includes a list of all the report topics that have been exported in the project. You can generate a .doc or .pdf containing the selected report topics or you can use the mechanism to compare results. This approach is good from the memory point of view as the results for the different runs are stored in png or jpg format. But there is not much flexibility – just a record of results with a caption.

Multiple buildings

Alternatively you can set up identical buildings by copying a base building multiple times at site level and making edits to them to create the design variants. Note that most model options are stored with the building so each of the building variants could be exactly the same but with a different model option.



You can compare results on the Simulation screen by switching between the 2 buildings using the navigator.

- This approach works well for small easily managed models but you must make sure that the models really are the same apart from the intended differences.
- Also, it will not work well for very large models because the models and their results are stored in memory simultaneously and this will cause DesignBuilder to slow down especially if you don't have much free memory on your computer.

Some new design comparison tools are in the pipeline for v.2.

16 Natural Ventilation

Natural ventilation is the process by which the building is ventilated to provide cooling and fresh air driven through the building by natural forces (as opposed to use of fans which is called Mechanical ventilation in DesignBuilder). Two forces drive natural ventilation in buildings:

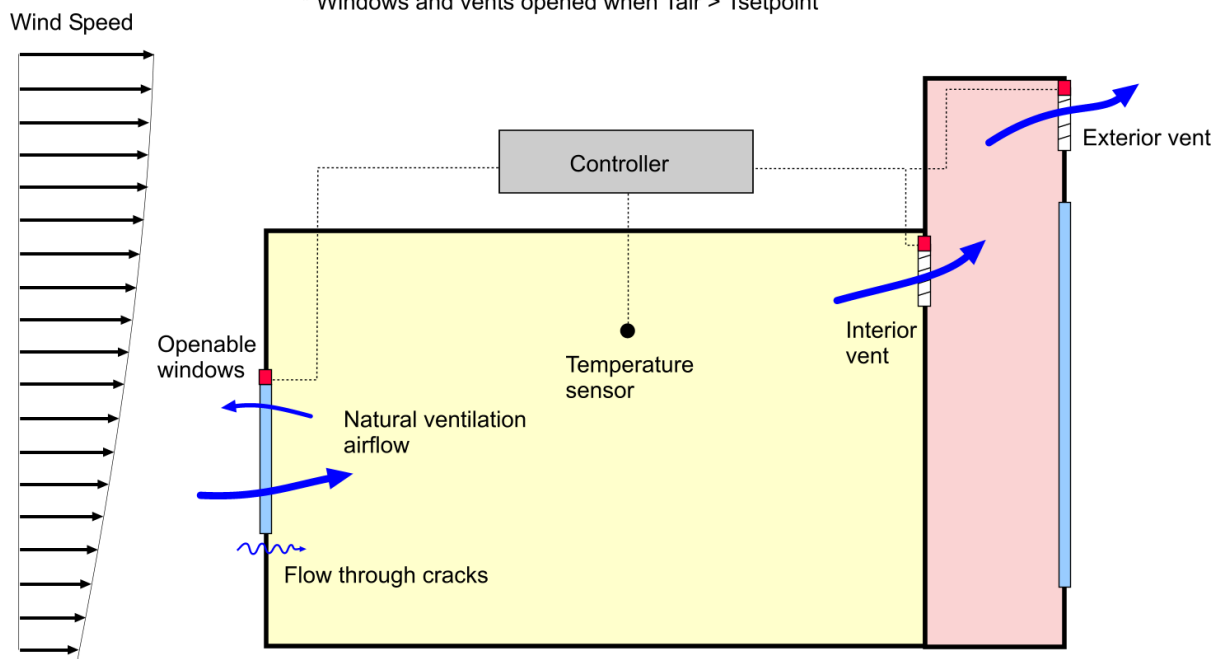
- **Stack effect**, in which relatively warm air is lighter than cooler air and will tend to rise, leaving a region of low pressure behind it.
- **Wind**, which is often the dominant of these two forces.

DesignBuilder can be used to model natural ventilation airflows due to these effects through windows, vents, holes and cracks in the building fabric.

Airflow through an opening can be calculated by determining the pressure differential that exists across the opening. The pressure differential comes from pressure inside the building that is mainly caused by temperature and wind pressure on the opening. More information on this is provided in Appendix I.

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 $T_{air} > T_{setpoint}$



In real buildings these natural ventilation airflows depend on many parameters such as the size, shape, position and orientation of the openings and cracks in the building fabric as well as the meteorological conditions such as wind-speed and outside temperatures.

Natural ventilation can be used in many climate zones to effectively cool buildings without over-reliance on electric driven fans, which can be expensive and polluting to run. Occupants of naturally ventilated buildings will often be expected to experience wider swings in temperature than in air-conditioned buildings but the benefit is often a more pleasant internal environment. Natural ventilation is not normally adequate to maintain good comfort conditions in very hot and humid climates or where very high levels of heat gain are present.

Scheduled and Calculated natural ventilation

DesignBuilder allows you to carry out sophisticated analyses including consideration of wind and buoyancy-driven flows through windows, vents, doors, cracks and holes. Bulk airflow movement between zones is calculated within EnergyPlus using the AirflowNetwork option. This is activated using the **Calculated** Natural ventilation option (described below in Examples 3-5).

Internal thermal comfort can be assisted through control mechanisms ranging from manual window opening to sophisticated automatic vent opening systems that optionally link with a mechanical HVAC system.

For early design stage analysis you can take a simpler approach where you specify a previously estimated or calculated natural ventilation rate using the **Scheduled** Natural ventilation option (described below in Examples 1 and 2). The worked examples that follow take you through the steps involved with using both of these approaches.

Scheduled Natural Ventilation

Introduction

If you have an approximate idea of the natural ventilation rates in the building, or if the natural ventilation heat flows are expected to be small relative to other heat flows in the building you can use **Scheduled** natural ventilation for a fast and easy calculation.

To do this set the Natural ventilation model option to **Scheduled**.

In this case simulated ventilation airflows are defined for 3 categories of flow path: external ventilation, internal ventilation and infiltration.

- Airflow through exterior windows, vents and doors is defined directly in air changes per hour. You can also set a schedule if the flow rate varies according to time.
- Infiltration (flow through cracks and fabric porosity) is defined using a constant air change rate per hour.
- Airflow through the interior windows, vents and doors is modelled using the concept of **mixing** where equal amounts of air transferred from one zone to another and vice versa. The flow rate is directly proportional to the opening area. There is no way in version 1 to set specific airflow rates between zones.

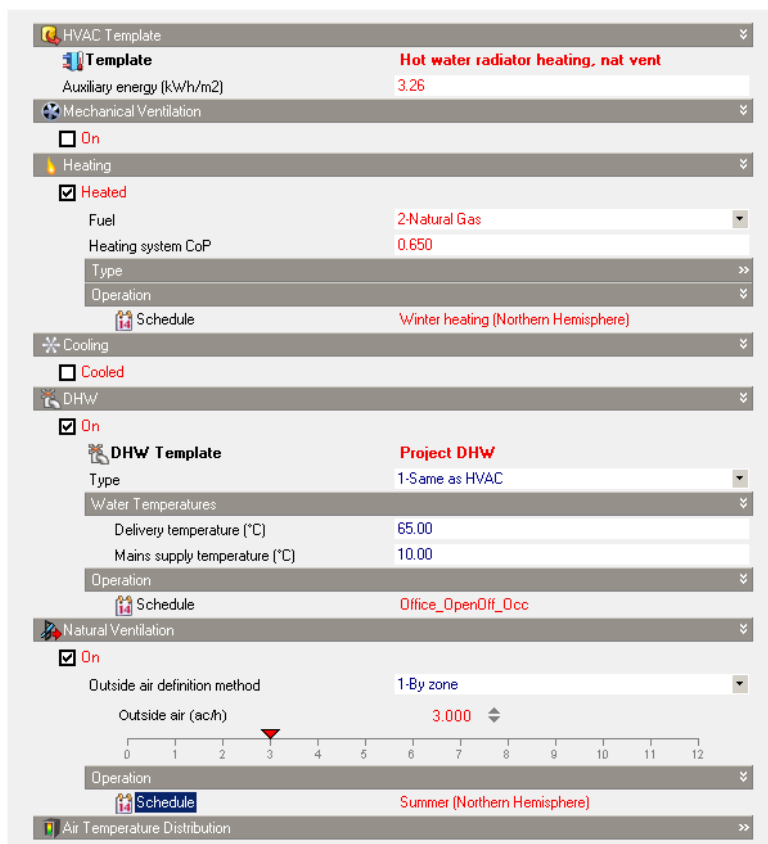
Note that in DesignBuilder **infiltration** is unintentional, uncontrolled airflow entry through the fabric, cracks etc, whereas **natural ventilation** is controlled and can have schedules, setpoints etc.

Bear in mind that although the building you are modelling may not actually have **thermostatic** controlled windows and vents, the building occupants may open and close windows to remain comfortable. So it is often appropriate to use setpoint temperatures even for **manually** operated windows.

Data Entry

This first natural ventilation worked example is based on the Atrium example model template created earlier.

- First create a **new project** using London Gatwick as the location and select the **Atrium example base** model.
- Go to the building level and click on the HVAC tab. If you have used the default **Hot water radiator heating, nat vent** HVAC template you should see something like:



HVAC Template

Template Hot water radiator heating, nat vent

Auxiliary energy (kWh/m2) 3.26

Mechanical Ventilation

☐ On

Heating

☒ Heated

Fuel 2-Natural Gas

Heating system CoP 0.650

Type >>

Operation >>

Schedule Winter heating (Northern Hemisphere)

Cooling

☐ Cooled

DHW

☒ On

DHW Template Project DHW

Type 1-Same as HVAC

Water Temperatures

Delivery temperature (°C) 65.00

Mains supply temperature (°C) 10.00

Operation >>

Schedule Office_OpenOff_Occ

Natural Ventilation

☒ On

Outside air definition method 1-By zone

Outside air (ac/h) 3.000

Operation >>

Schedule Summer (Northern Hemisphere)

Air Temperature Distribution >>

- If you have a different HVAC template loaded then load the **Hot water radiator heating, nat vent** template.

- Load the ‘Summer northern hemisphere’ natural ventilation operation schedule meaning that natural ventilation is available all summer, day and night.
- The heating operation schedule is ‘Winter heating (Northern Hemisphere)’ preventing heating operation in the summer.
- Also check that the Natural ventilation model option is set to **Scheduled** as shown below. It will probably be already set as this is the default option.



- At building level set some effective shading for external windows and rooflights. Use the Window shading option, select Blind with high reflectivity slats and use the 4-Solar control type option. Use the same options for rooflights.
- Now enter the following data (if the data is already set to the required value then leave it):

Model Data	Building	Comment
Construction tab > Airtightness		
Infiltration constant rate (ac/h)	0.5	
HVAC tab > Natural Ventilation		
Outside air definition method	1-By zone	

Outside air (ac/h)	5	Most important data and most difficult to estimate accurately. Also set this in the Atrium zone.
Schedule	Summer (Northern Hemisphere)	
Activity tab > Environmental control > Ventilation Setpoint Temperatures > Natural Ventilation		
Nat vent cooling (°C)	21	
Max in-out delta T (K)	2	
Activity tab > Environmental control > Heating Setpoint Temperatures		
Heating (°C)	20	

Note: Making these settings at building level does not affect the atrium zone as it has the <None> HVAC template loaded at the block level.

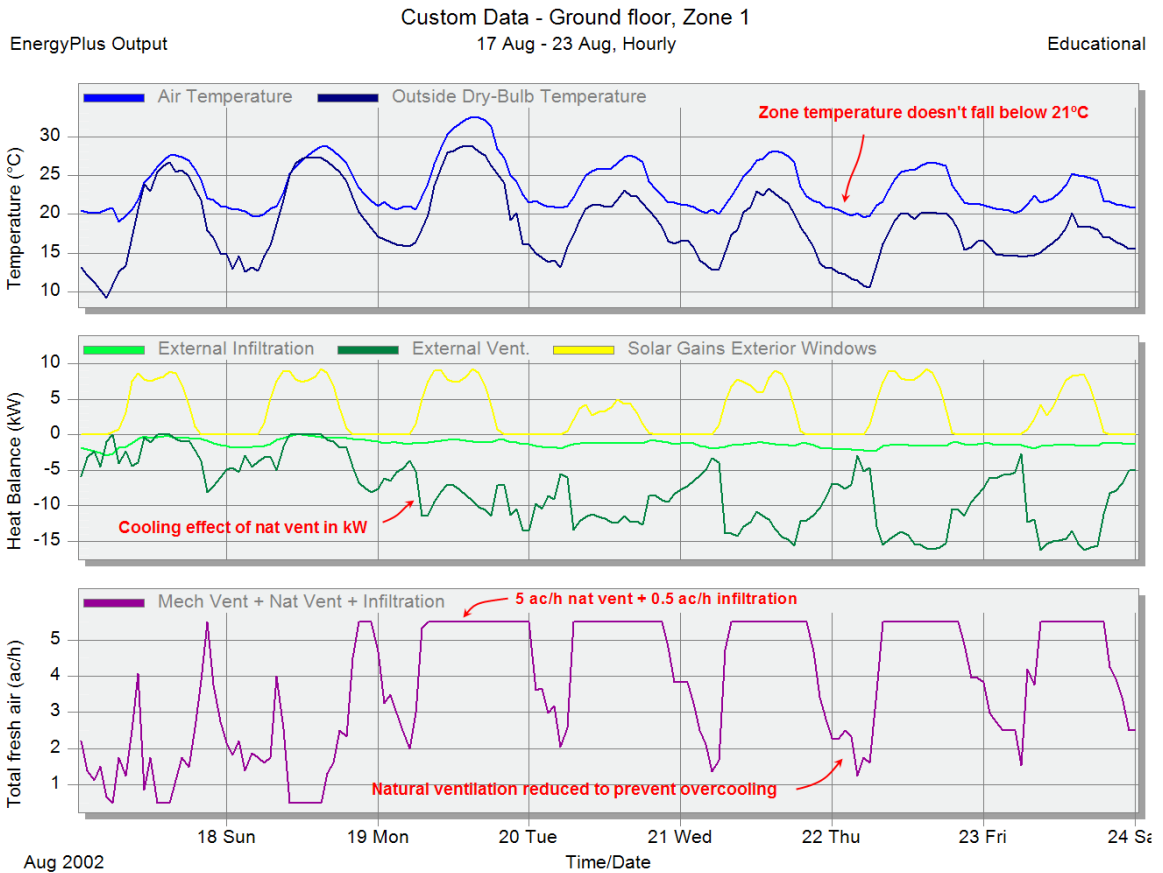
The **Max in-out delta T** sets a maximum temperature difference to avoid warm air being introduced. A value of 2 means that the outside air must be at least 2°C colder than the inside air for the windows to be opened.

If the schedules allow natural ventilation and heating to occur during the same periods, the natural ventilation and cooling setpoint temperature should be higher than the heating setpoint temperature to avoid simultaneous operation and wasted heating energy. This is why the natural ventilation cooling setpoint temperature is set to 21°C and free heating setpoint temperature is set to 20°C and heating is switched off in the summer.

Start a Simulation to test the settings. Click on the **Simulation tab** at the bottom of the screen and when the Calculation options dialog is open select the **Summer typical week** period by clicking on the corresponding link on the Info Panel on the right of the dialog. Select only **Hourly** results.

Results Analysis

When the simulation is complete select **Hourly results** in the Display options panel and go to **Ground floor > Zone 1** by clicking on the Navigator panel. You should see results as below.



Some points to note about these results:

- Looking at the purple “Mech vent + Nat vent + Infiltration” data, the *maximum* airflow rate is 5.5 ac/h. This is made up of 5 ac/h due to natural ventilation and 0.5 ac/h due to infiltration.
- The natural ventilation is switched on all the time in the summer to get free night cooling (windows and vents are left opened) but the natural ventilation air-change rate is not always 5 ac/h. This is because the natural ventilation is being controlled to avoid overcooling the space. You can see that the minimum air temperature in the zone is approximately 21°C. The temperature is being controlled by the natural ventilation cooling setpoint temperature set on the activity tab. This simulates the operation of thermostatic-controlled vents. It also gives an idea of likely building operation for manually operated windows where occupants will tend to open and close windows to stay comfortable.
- The dark green **External Vent** line in the heat balance graph shows the thermal cooling effect due to natural ventilation. Note that natural ventilation supplies up to 15 kW of cooling.

What happens if you change the natural ventilation operation schedule to switch off at night?

Now save the file as a template (File > Save > Save as template) calling it something like **Nat Vent Example 1**.

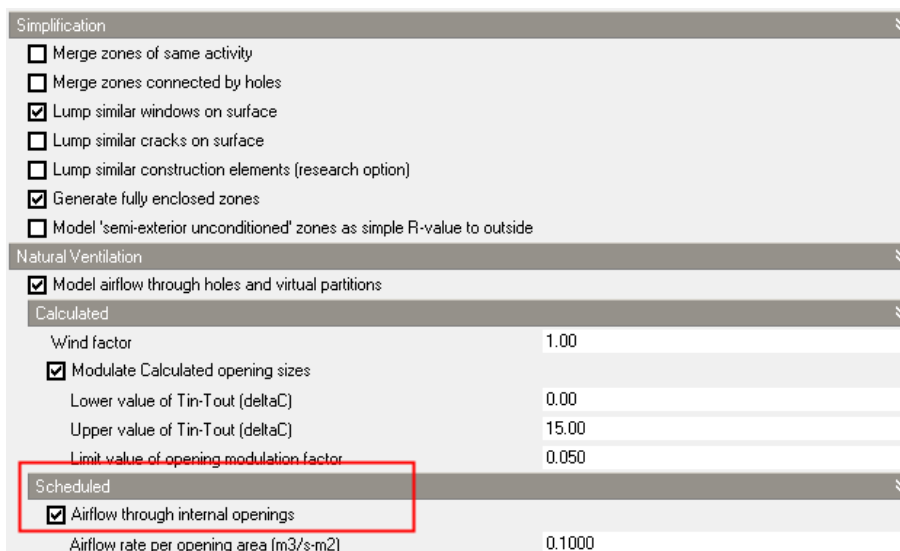
Scheduled Natural Ventilation with Internal Airflow

This worked example follows on from the previous one but by allowing internal airflow to occur between the occupied zones and the atrium.

Data entry

Starting with the model you created in Example 1, follow these steps:

- Open **Model options** dialog and click on the Advanced tab and the Natural ventilation header. Switch on the “**Airflow through internal openings**” option under the Scheduled header to allow airflow to pass through the internal opening we have just drawn. Leave the airflow rate per opening area as the default value of 0.1 m³/s-m². This will cause 0.1 x the area of each opening m³/s airflow mixing between the 2 adjacent zones. The Model options data should now look like the screenshot below.

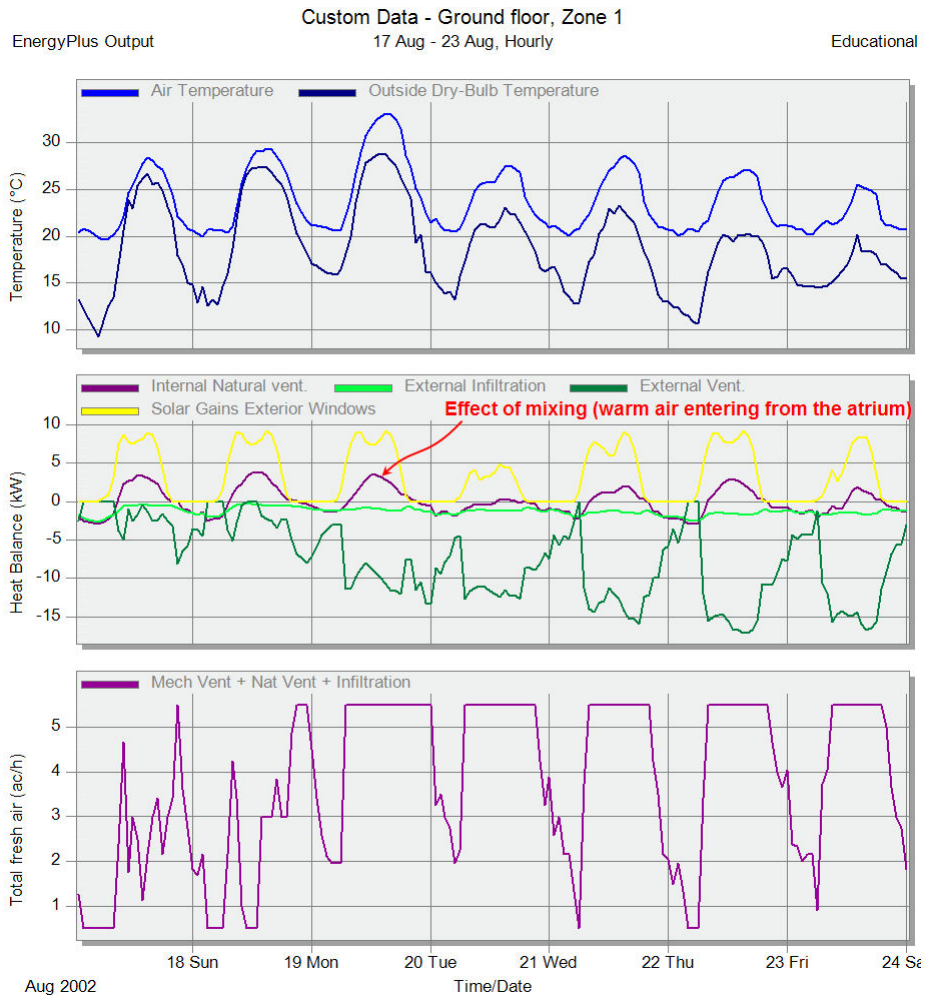


Simplification	
<input type="checkbox"/>	Merge zones of same activity
<input type="checkbox"/>	Merge zones connected by holes
<input checked="" type="checkbox"/>	Lump similar windows on surface
<input type="checkbox"/>	Lump similar cracks on surface
<input type="checkbox"/>	Lump similar construction elements (research option)
<input checked="" type="checkbox"/>	Generate fully enclosed zones
<input type="checkbox"/>	Model 'semi-exterior unconditioned' zones as simple R-value to outside
Natural Ventilation	
<input checked="" type="checkbox"/>	Model airflow through holes and virtual partitions
Calculated	
Wind factor	1.00
<input checked="" type="checkbox"/>	Modulate Calculated opening sizes
Lower value of Tin-Tout (deltaC)	0.00
Upper value of Tin-Tout (deltaC)	15.00
Limit value of opening modulation factor	0.050
Scheduled	
<input checked="" type="checkbox"/>	Airflow through internal openings
Airflow rate per opening area (m3/s-m2)	0.1000

Results analysis

Rerun the simulation for the same period by clicking on the Simulation screen and clicking on the OK button of the simulation options dialog when prompted.

The results for Ground floor > zone 1 should look something like the screenshot below. Note that the purple **Internal Natural Vent** data shows the ‘mixing’ heat transfer through the windows and vents - positive values in Ground floor > Zone 1 show a heat gain from internal ventilation and corresponding negative values in the atrium show heat loss.



Note that this does not model the internal flow in the actual building very well because the atrium would really act as a stack and draw air in from the adjacent offices. In this case the effect of internal natural ventilation would be a benefit to the offices as the negative pressure created will induce cooler outside air to flow in. To model this we need to use Calculated natural ventilation.

Now save the file as a template (File > Save > Save as template) calling it something like **Nat Vent Example 2**

Calculated Natural Ventilation

Introduction

With the **Scheduled** natural ventilation option used in Examples 1 and 2 a known value is used to define natural ventilation flow rate in the model. In many cases however the actual natural ventilation flow rate will not be known and the user may prefer to allow EnergyPlus to *calculate* flow rates based on the size, configuration, position and orientation of the openings and cracks in the fabric as well as the minute by minute simulated physical conditions of the air including wind-speed, pressure, inside and outside temperatures etc. This is possible in DesignBuilder by using the **Calculated** natural ventilation option.

Before starting, you should understand some important differences between Scheduled and Calculated natural ventilation:

- Calculated natural ventilation requires **more input data**, some of which can be quite hard to obtain, in particular wind pressure coefficients. For this reason, while calculated natural ventilation is theoretically more accurate, you should be aware of the need for care in data entry to avoid inaccurate results (garbage in = garbage out). If the building is of fairly typical construction and configuration it can sometimes be more accurate to use empirically obtained ventilation rates for typical buildings if you do not have the time to spend carefully setting up the calculated natural ventilation model.
- People new to building energy simulation can be surprised to find that by switching from Simple natural ventilation to Calculated natural ventilation the **results are very different**. This should not be surprising when you think about it because the data entry is completely different. The key parameters in Calculated natural ventilation being window and vent opening sizes and operation, whereas the key parameters in Scheduled natural ventilation are the directly input ventilation rates and operation. There is not necessarily any relationship between the default window opening areas used with the Calculated option and the default natural ventilation flow rate used with the Scheduled option.
- Calculated natural ventilation **increases simulation time** significantly because of the extra calculations required at each time step in the simulation to calculate airflow rates.

Data Entry

In this worked example we will convert the Scheduled natural ventilation model from Example 1 to use Calculated natural ventilation and explore the most important data required to achieve a realistic natural ventilation simulation.

- First open the model created for Example 1.
- Open the Model options dialog and switch to using **Calculated** natural ventilation. When you press OK on the Model options dialog you will be asked if you wish to use Compact HVAC to allow mechanical ventilation to be modelled. Respond “No” because we will not model mechanical ventilation in this Example.



Scope

Zone → Zone+shading → **Building**

Whole building
Analyse the whole building.

Construction and Glazing Data

Pre-design → **General**

General construction templates
Construction default data is selected from a list.

Floor/slab/ceiling representation: 1-Combined

Zone volume calculations

- ☒ Internal floor constructions not subtracted from zone volume
- ☒ Ground floor construction is below ground and is not subtracted from zone volume
- ☒ External floor constructions not subtracted from zone volume

Gains Data

Lumped → **Early** → Detailed

Early gains
Internal gains are separated into various categories (e.g. occupancy, lighting, computing etc.)

Timing

Typical workday → **Schedules**

Schedules
Timing is defined using the schedules and profiles mechanism which allows each day of the week to have a different profile.

HVAC

Simple → **Compact**

Simple HVAC
HVAC systems are defined using plant seasonal efficiency data and are modelled outside EnergyPlus

HVAC sizing: 3-Autosize

Natural ventilation

Scheduled → **Calculated**

Calculated ventilation
Natural ventilation and infiltration air flow rates are calculated based on opening and crack sizes, buoyancy and wind pressures.

- Go to building level and click on the HVAC tab and check that Natural ventilation is switched on.
- Still at the building level, click on the Openings tab and open the Operation header under External Windows. This data defines the maximum extent to which the windows are able to open. The data is only available when using Calculated natural ventilation.
- The operation schedule can also be defined here. This is not the schedule controlling the operation of the windows but rather an availability schedule describing the times when the windows are *allowed* to open, associated controls permitting. Leave the default ‘Summer (Northern Hemisphere)’ operation schedule. This means that the window opening can open any time when this operation schedule has a value greater than zero (any time in the summer in the case).
- Now go to the Building level and under External Windows > Operation enter “% Glazing area opens” to 5% meaning that each window in the building can open a maximum of 5% of its total area.
- For the Atrium enter “% Glazing area opens” to 0% because atrium external windows do not open.
- Answer “No” when asked if you would like to use Compact HVAC when closing the Model options dialog. The Natural Ventilation section of the dialog should look like this:

Natural Ventilation	
<input checked="" type="checkbox"/>	Model airflow through holes and virtual partitions
Calculated	
Wind factor	1.00
<input type="checkbox"/>	Modulate Calculated opening sizes
Scheduled	
<input type="checkbox"/>	Airflow through internal openings

This same data can be edited under the Natural Ventilation and Options headers on the HVAC tab and this is the preferred way to set this data now. Be sure to be at building level when making the setting in this way so the edit is inherited to all the surfaces in the building.

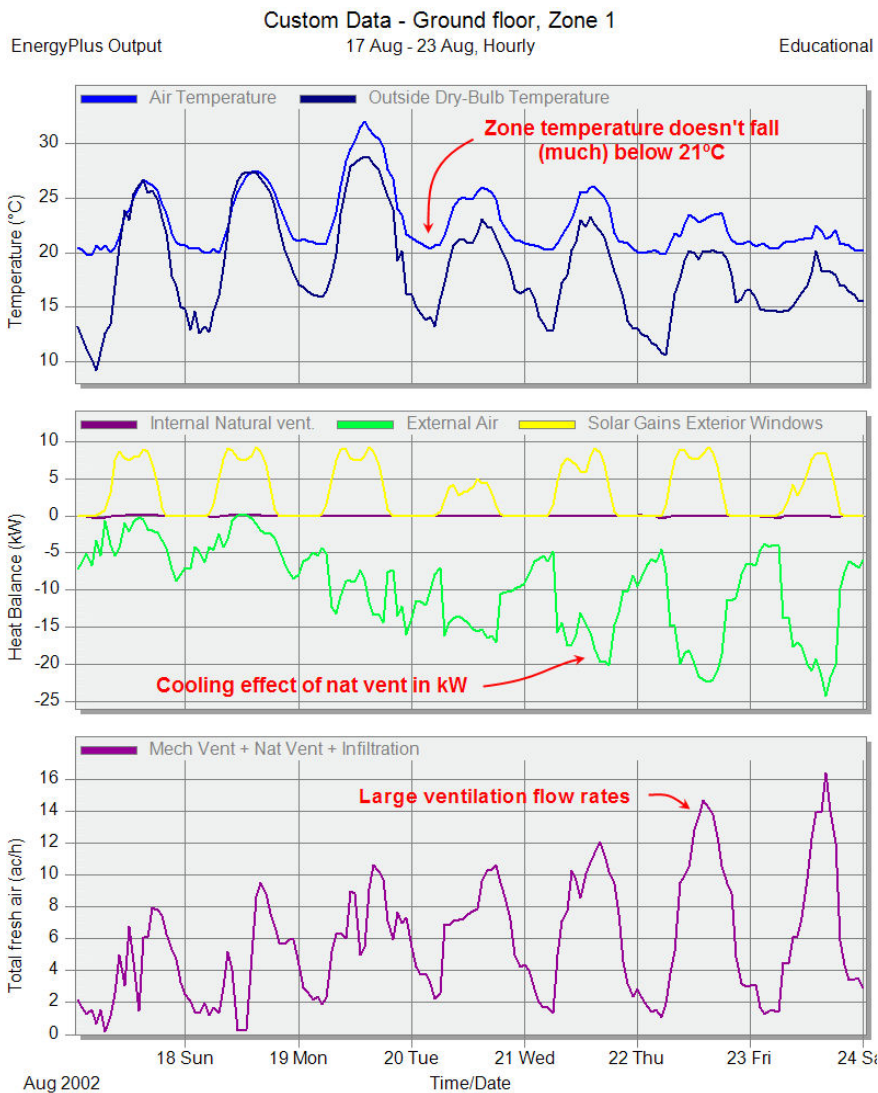
- Go to the building level and Click on the Openings tab and set all vent operation schedules to operate during the summer only. The openings tab should look like:

Glazing Template	
Template	Project glazing template
External Windows	
Glazing type	Project external glazing
Layout	Preferred height 1.5m, 30% glazed
Dimensions	>>
Frame and Dividers	>>
Shading	>>
Operation	>>
% Glazing area opens	5.0
Operation schedule	Office_OpenOff_Occ
Internal Windows	
Glazing type	Project internal glazing
Layout	No glazing
Dimensions	>>
Frame and Dividers	>>
Operation	>>
% Glazing area opens	0
Operation schedule	Summer vent work days (Northern Hemisphere)
Roof Windows/Skylights	
Glazing type	Project roof glazing
Layout	No roof glazing
Dimensions	>>
Frame and Dividers	>>
Shading	>>
Operation	>>
% Glazing area opens	0.0
Operation schedule	Office_OpenOff_Occ
Doors	
Vents	
External	
Vent type	Grille, small, light slats
<input type="checkbox"/>	Auto generate
Operation	>>
Operation schedule	Summer (Northern Hemisphere)
Internal	
Vent type	Grille, small, light slats
<input type="checkbox"/>	Auto generate
Operation	>>
Operation schedule	Summer (Northern Hemisphere)
Roof	
Vent type	Grille, small, light slats
<input type="checkbox"/>	Auto generate

- Go to the building level and Click on the Constructions tab and under the Airtightness header check that **Model infiltration** is switched on and check that the **Crack template** slider is set to **Medium**. This means that all crack sizes in the building will be taken from the “Medium” crack template. You can currently select from these options:
 - Very poor
 - Poor
 - Medium
 - Good
 - Excellent

Note that although the infiltration rate in ac/h can be set this is only used in Heating and Cooling design calculations.

- Click on the “Show crack data” link in the Info panel to see the definition of the currently selected crack template.
- The model is now ready for the simulation so click on the **Simulation tab** and using the same options as before (Hourly results, **Summer typical week**) press OK to start the simulation.
- Check that your hourly results for Ground block > Zone 1 are similar to those below.



The results show:

- The zones have air temperature approximately controlled to the natural ventilation setpoint temperature of 21°C.
- Typical total air change rates (purple Mech Vent + Nat Vent + Infiltration data) are between zero and 16 ac/h.

Try running the simulation again but with the building external windows closed at night. In this case much less ventilation is possible at night and less nighttime cooling occurs. Infiltration rates at nighttime when the windows are closed are between 0.3 and 0.05 ac/h. The variations are caused by variations in wind speed. More on this in the next Example.

Now save the file as a template (File > Save > Save as template) calling it something like **Nat Vent Example 3**.

Advanced Calculated Natural Ventilation

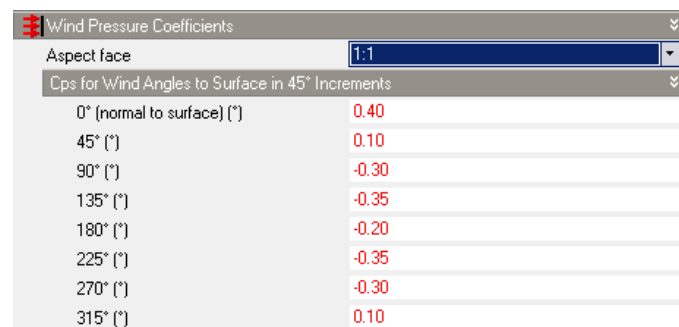
Introduction

In this Example we will explore some more advanced calculated natural ventilation options including wind pressure coefficients, wind effects vs temperature effects, internal airflow and how to check airflow rates through individual openings.

Wind Pressure Coefficients

Building geometry and surroundings play a major role in determining wind effect on opening. The pressure caused by wind blowing on an external surface is calculated using wind data from weather file and the wind pressure coefficient for current particular wind incidence angle on the surface. This information is entered as wind pressure coefficient values, one for each angle of incidence in increments of 45°.

Go to the North facing Wall surface in Ground floor > Zone 1 and click on the Openings tab and open the Wind Pressure Coefficient (WPC) header. You should see something similar to the screenshot below.



Wind Pressure Coefficients	
Aspect face: 1:1	
Cps for Wind Angles to Surface in 45° Increments	
0° (normal to surface) (°)	0.40
45° (°)	0.10
90° (°)	-0.30
135° (°)	-0.35
180° (°)	-0.20
225° (°)	-0.35
270° (°)	-0.30
315° (°)	0.10

This default data is automatically generated and loaded when the surface is first created from standard AIVC tables for building aspect ratios of 1:1. The data is approximate and suitable for early stage natural ventilation analysis only.

You can see that when the wind is normal to this surface the Wind Pressure Coefficient is 0.4 i.e. the pressure, P_w , on the outside of the surface at any time in the simulation is given by the equation.

$$P_w = 0.5 \cdot (\text{air density}) \cdot (0.40) \cdot (\text{wind velocity})^2$$

Likewise similar pressure coefficients are given for other wind angles relative to the surface normal. See **Appendix I** for more details on this.

In this example, leave the Wind Pressure Coefficient data in its default state.

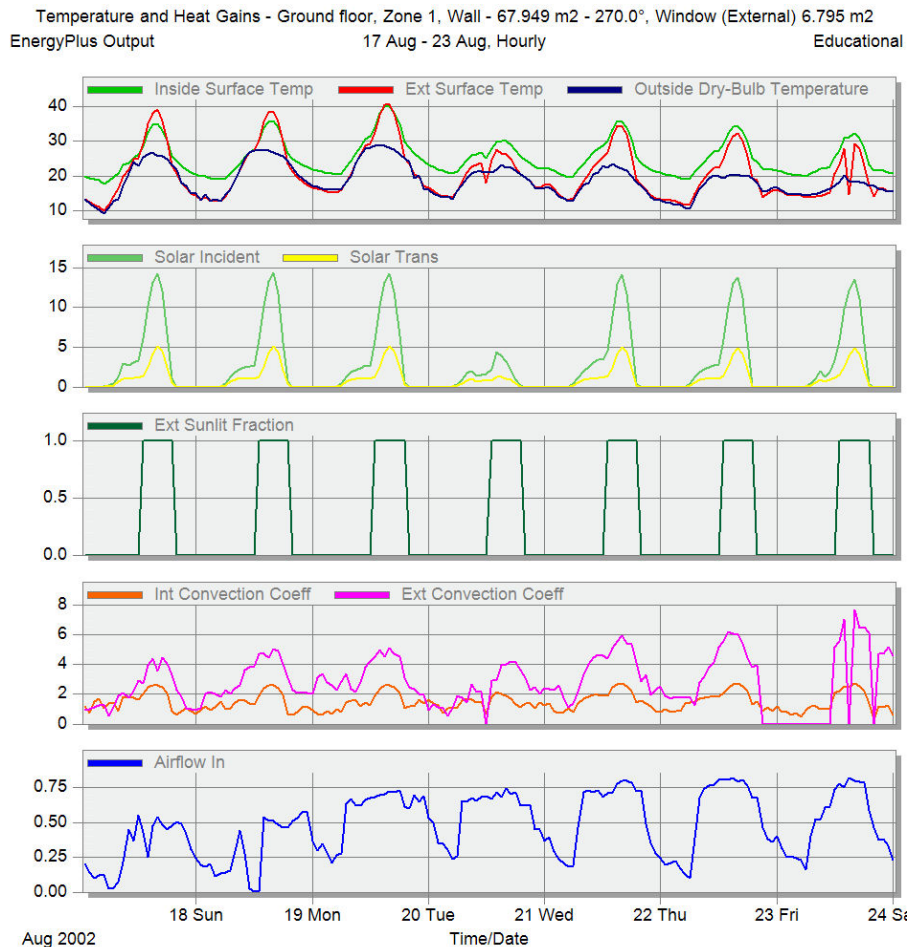
Testing the effect of wind

- Try changing the **Wind factor to 0** (HVAC > Natural Ventilation > Options at building level to effectively switch off all wind effects in the model and rerun the simulation.
- Also switch on output of airflow in/out of openings by going to the Misc tab at building level. Select these options:

- ☒ Store surface and opening output
 - ☒ Store opening output
 - ☒ Inside Surface Temp
 - ☒ Ext Surface Temp
 - ☒ Int Convection Coeff
 - ☒ Ext Convection Coeff
 - ☒ Surface Gain
 - ☒ Solar Incident
 - ☒ Solar Trans
 - ☒ Ext Sunlit Fraction
 - ☒ Airflow In
 - ☒ Airflow Out

Making this setting at building level will mean that large amounts of data could be stored for *each surface and each opening* in the model, especially if hourly or sub-hourly results are requested and long time periods are simulated. For the design week we are looking at you should not have any problems, though simulations will take longer when more output is requested.

Reviewing hourly results for the Ground floor > Zone 1 > Wall 67.795 - 270° > Window (External) 6.795m2 you should see results similar to the screenshot below:



Make sure to select the first window as the other 2 were ‘lumped out’ and don’t have any data. The results for the 1st window have been multiplied by 3 to account for the other 2 windows.

Has the switching off of the wind reduced the fresh air delivery very much in the offices?

You should find that air is mainly flowing into the building through the operable office windows giving cooling in the main occupied part of the building and is exiting from the vents in the roof of the atrium. Check you understand the nature and magnitude of the flows through all openings and surfaces.

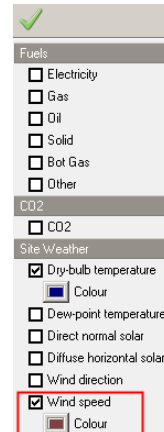
Since there are no wind effects, the natural ventilation is driven entirely by buoyancy effects, which in turn are driven by temperature differences between inside and outside. Because hour-to-hour variations in these temperature differences are less marked than variations in meteorological wind speed the ventilation ac/h rates have less hour-to-hour variations.

Switching wind off in this way can be a useful design tool to ensure that a building can be reasonably well ventilated even if there is no wind. It also illustrates the importance of harnessing natural wind pressures to ensure effective natural ventilation.

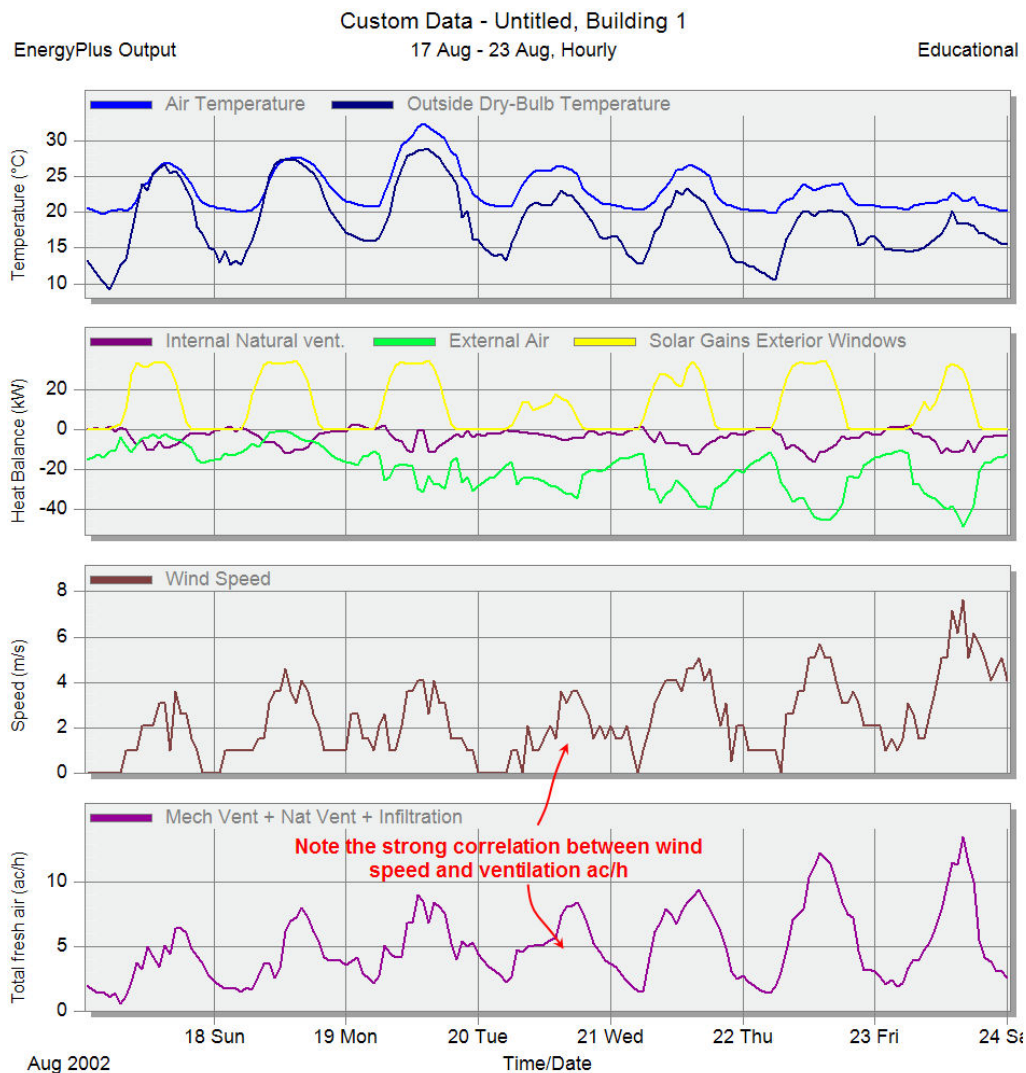
Display wind speed alongside other results

Set the building **Wind factor back to 1**, rerun the simulation and view the results for the Block 1, which gives averages and totals for the whole block.

On the Detailed tab of the Display options, scroll down to the bottom of the list, check the Wind speed option and click on the Apply button. Screenshot right. This includes Wind speed data alongside other results just for this graph (you can save the selection by using the Save toolbar icon on the Detailed tab).



The screenshot below shows that there is quite a strong correlation between Wind speed and airflow rate (bearing in mind that the windows are shut at night and that temperature control also plays a major part in determining external airflow rates). You can see the influence of wind speed by observing that the peaks in wind speed tend to occur at the same time as the peaks in airflow rate.



17 Simple HVAC

Introduction

In DesignBuilder 'HVAC' refers to all elements of the building whose function is to provide heating, cooling and ventilation to provide comfort for the occupants. This includes natural ventilation, but this topic is covered in a separate section.

DesignBuilder offers a range of HVAC system types and also 2 levels of detail:

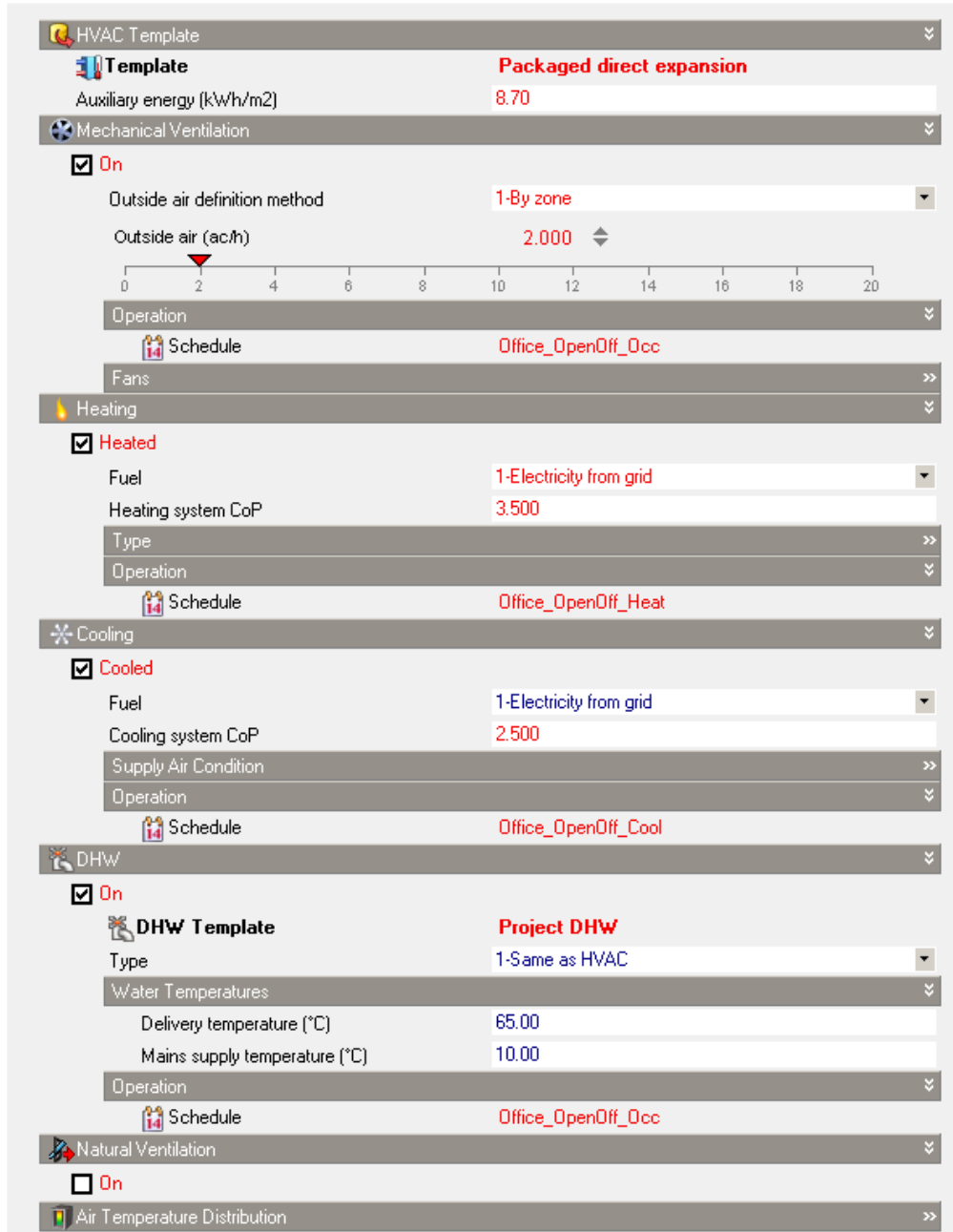
- **Simple** where the system is modelled by calculating the loads required to meet the heating and cooling setpoints. HVAC system energy consumption is calculated from these loads using seasonal system CoPs.
- **Compact** using the EnergyPlus Compact HVAC. This allows a much more detailed model of some HVAC system types to be developed but using simple parametric input data that can be loaded from templates.

In many cases, especially at early stages of the design process when looking at fabric, glazing, solar shading options, you may not want to model the details of the HVAC system. In these cases the heating and cooling energy consumption can be calculated from loads using assumed seasonal system Coefficient of Performance (CoP) values.

Example

This Example will introduce you to the basics of entering data for **Simple HVAC**. Starting from where you left off with the Atrium example model, you will create a simple model with heating and cooling, run a simulation and check daily and hourly results.

1. Ensure Simple HVAC and Schedule natural ventilation model options are selected.
2. Window shading using 'High reflectivity slats' and 4-Solar control with a 150 W/m2 setpoint, T5 fluorescent lights, no daylight control.
3. At Building level go to the HVAC tab and load the **Packaged direct expansion** HVAC template.
4. Set the Mechanical ventilation **Outside air definition method** to **1-By Zone**. This means that the **Outside air** ac/h rate entered on the slider below will be used directly. Other options will be explored in the next Example.
5. Set the Mechanical ventilation **Outside air** delivery to be **2 ac/h**.
6. Leave the Mechanical ventilation **Operation schedule** to be **Office_OpenOff_Occ**.



HVAC Template

Template Packaged direct expansion

Auxiliary energy (kWh/m2) 8.70

Mechanical Ventilation

☒ On

Outside air definition method 1-By zone

Outside air (ac/h) 2.000

Operation

Schedule Office_OpenOff_Occ

Fans

Heating

☒ Heated

Fuel 1-Electricity from grid

Heating system CoP 3.500

Type

Operation

Schedule Office_OpenOff_Heat

Cooling

☒ Cooled

Fuel 1-Electricity from grid

Cooling system CoP 2.500

Supply Air Condition

Operation

Schedule Office_OpenOff_Cool

DHW

☒ On

DHW Template Project DHW

Type 1-Same as HVAC

Water Temperatures

Delivery temperature (°C) 65.00

Mains supply temperature (°C) 10.00

Operation

Schedule Office_OpenOff_Occ

Natural Ventilation

☐ On

Air Temperature Distribution

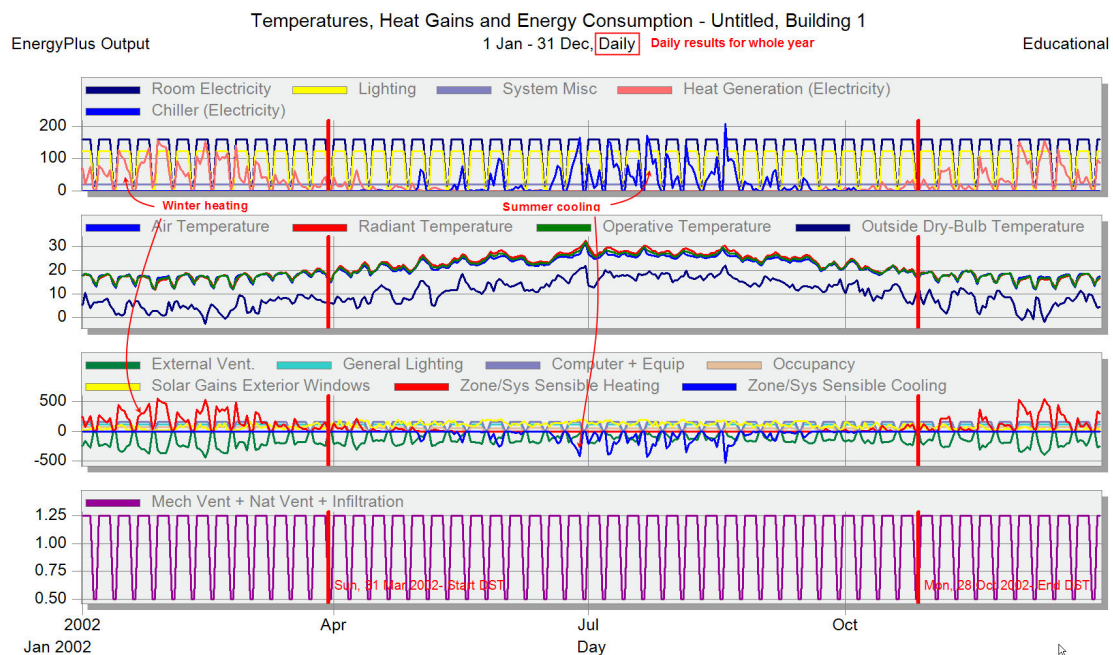
The HVAC tab should look something like the above screenshot. Note the simple definition of the HVAC system. The Heating and Cooling system **CoP** values are used to calculate the HVAC energy consumption data from the heating and cooling loads calculated in the simulation.

The Fan and pump energy consumption is modelled using the **Auxiliary energy** in kWh/m2 (default data for this comes from the UK NCM).

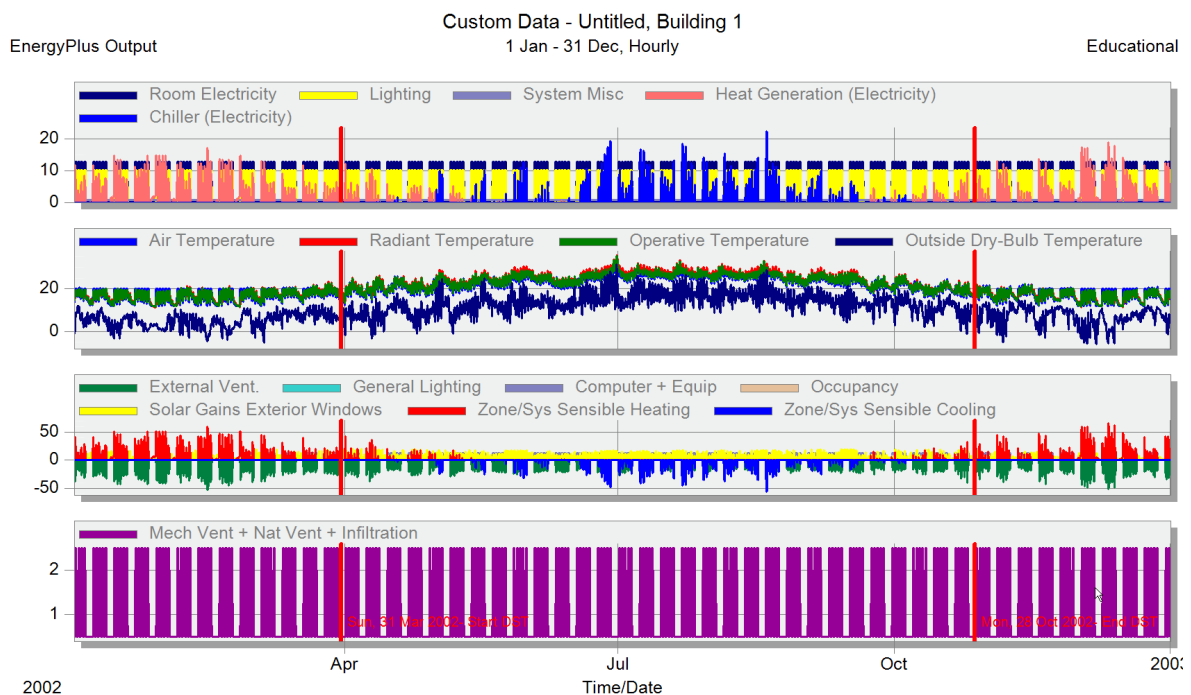
7. Heating and cooling setpoint temperatures are defined on the Activity tab because they relate more to the activity taking place within the space (usage) than to the HVAC system. So still at Building level go to the **Activity** tab and open the **Environmental Conditions** header to show the setpoints. Set the heating setpoint to **20°C** and the cooling setpoint as **26°C**. Leave the Heating

and cooling setback temperatures in their default states – we'll modify those in the next Example.

The model is now ready for simulation. Click on the **Simulation tab** and when the Simulation options dialog is displayed select **Hourly** and **Daily** results, and **Annual simulation** and press OK to start the simulation. After a few minutes, when the simulation is finished and the results have been loaded you should see the daily results below.

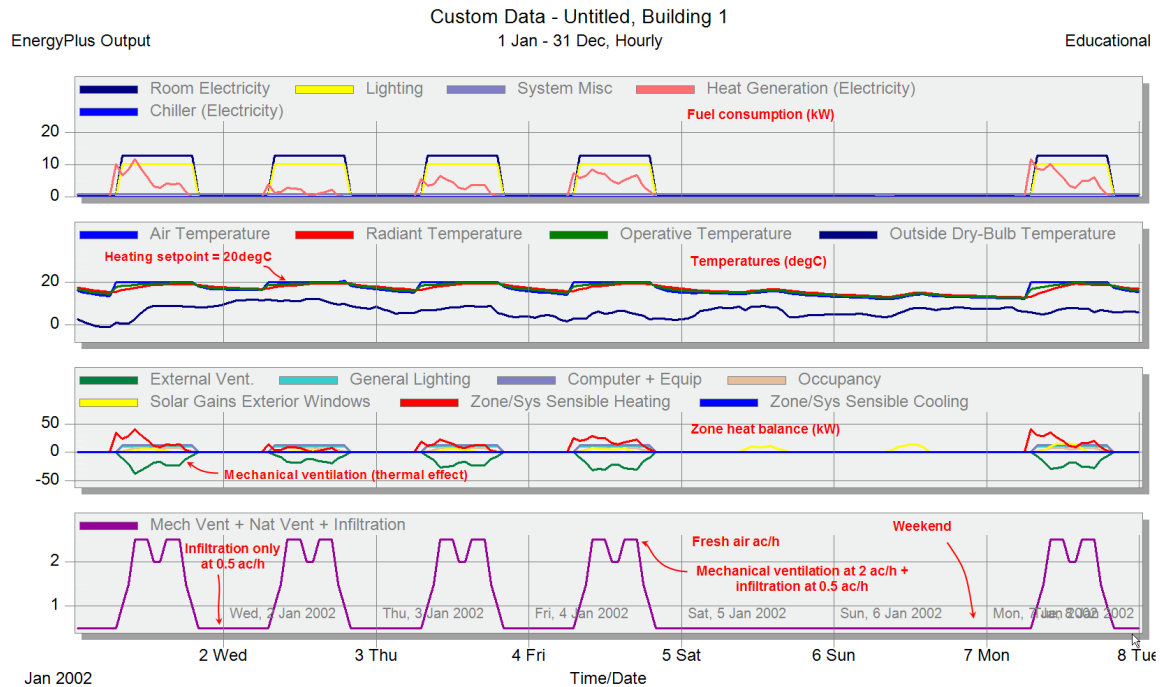


Now select hourly results by selecting the **Interval** as **3-Hourly** in the Display options panel in the bottom left of the screen. You should see:



Hourly results are much more useful for understanding the operation of the HVAC system than daily results because you can check that setpoint temperatures are met, see periods of the day when peak demands occur etc.

The above hourly results for the year are interesting for an overview, but to get a clearer view it is best to display less days at a time. Try setting the **Days per page** to 7 in the Display options panel and you will see results for the first 7 days in the year.

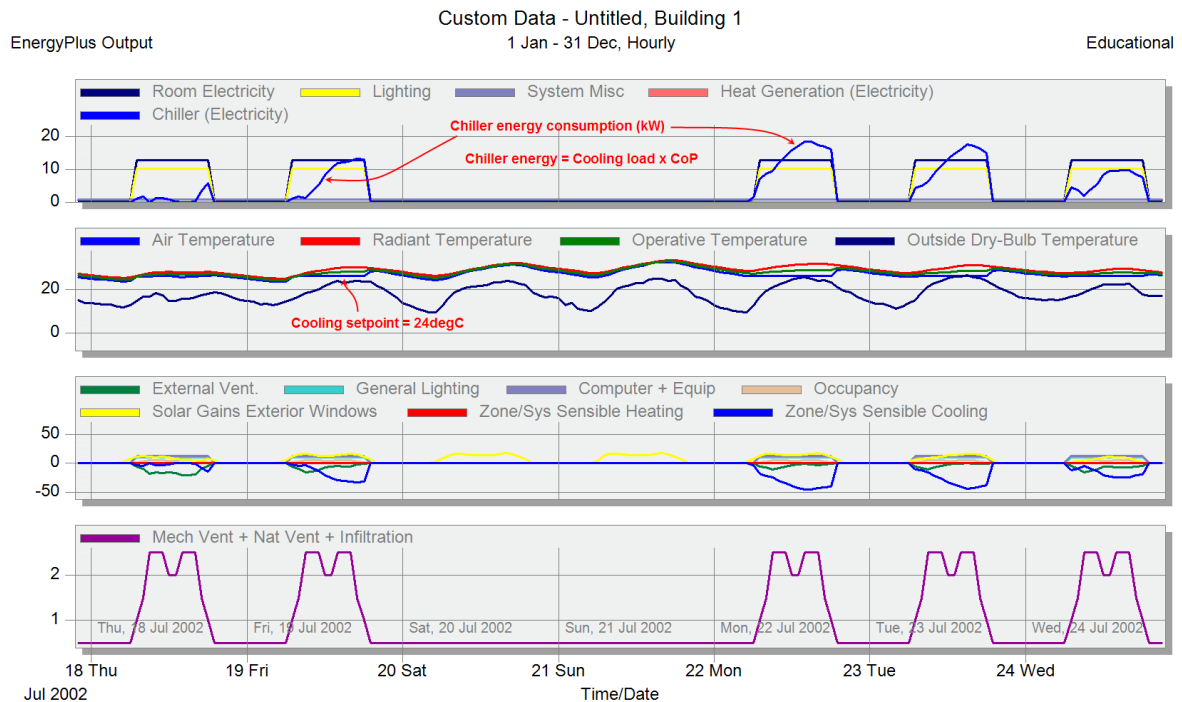


The results show:

1. The heating system maintains the heating setpoint of 20°C during the daytime.
2. The peak heating load takes place on winter mornings
3. The HVAC system is switched off during weekends and at night and the temperature in the zone is allowed to 'float' during these times.
4. Infiltration is the only source of fresh air when the HVAC system is switched off – during these time the fresh air rate is 0.5 ac/h.
5. When the HVAC system is switched on, the fresh air rate is the 0.5 ac/h constant infiltration + the mechanical ventilation at 2 ac/h x Office_OpenOff_Occ schedule value.
6. The thermal effect of the mechanical ventilation is shown in the heat balance graph separately from the cooling loads.
7. The heat generation energy consumption is calculated as heating load (Zone/Sys Sensible Heating on the graph) x CoP.

The thermal effect of the mechanical ventilation (**External Vent**) is shown separately from the cooling system effect (**Zone/Sys Sensible Cooling**).

You can use the scrollbars along the bottom of the graph to move through the hourly results for the year. Scroll until you reach the summer period starting with June 18th as shown below.



This graph shows the cooling system meeting cooling setpoint temperatures and illustrates how the Chiller energy consumption = Cooling load x CoP. Check for your self that this is the case. Chiller CoP is 2.5. Do the same for heating. Heat generation CoP is 3.5.

Also notice how it is the **air temperature** that is being controlled and that in the summer the **radiant temperature** can still rise much higher even while the air is being controlled to 26°C. This leads to higher operative temperatures of 27°C and higher (mean of air and radiant temperatures). The operative temperature is often used as a good measure of occupant thermal comfort. It is a better measure than air temperature because it includes radiant effects.

Warning: *There is an option to control to the **operative temperature** instead of the air temperature but we don't encourage the use of this option because it requires an advanced understanding of the way the model works to understand limits of applicability and error messages that can arise when these limits are not respected.*

For zones with very high radiant temperatures, in order to control the operative temperature to a particular value the supply air must be very cold to bring the zone air temperature down the value required. Sometimes the required supply temperature is lower than the supply air temperature requested by the user and in these cases an error message is generated. If you do use operational control you should look out for error messages from EnergyPlus related to this effect.

More Simple HVAC

Example

In this Example you will modify the model created in Example 1, increasing the **night time setback** temperature for heating and using **minimum fresh air per person** mechanical ventilation.

1. At building level on the HVAC tab set the Mechanical ventilation **Outside air definition method** to be **2-Min fresh air per person**. This means that the maximum mechanical ventilation fresh air is calculated from the occupancy of the zone and the minimum fresh air requirement per person. Both these are defined by the activity.
2. So on the **Activity** tab note that the **occupancy density** is set as **0.11 people/m²**. Also under the Environmental Control > Minimum Fresh Air header. Note the **Fresh air requirement per person** is set to **10 l/s per person**. The Mech vent per area data would also be used if we had selected 4-Min fresh air (Sum per person + per area) option.
3. Still on the Activity tab at building level change the **Heating set back** under the Heat Setpoint Temperatures header to **16°C**. This will be the setpoint whenever the Heating Operation Schedule has a value of 0.5.
4. Go back to the HVAC tab and single-click on the Heating Operation Schedule. The schedule Office_OpenOff_Heat should be selected. The data for the schedule is displayed in the Data report panel in the bottom right of the screen. You will see the schedule has values of 0.5 between 7pm and 6am and a value of 1 between 6am and 7pm. Weekends and holidays have 0.5. This means that the heating system will use setback temperature (16°C) during the night and at weekends and holidays.
5. Likewise for cooling, the main cooling setpoint of 26°C applies during the occupied period. For AllOtherDays (i.e. Cooling design days) a value of zero is used, that is heating is not required for cooling design.

Run the simulation for the Winter design week and check the heating operation at nighttime. The heating system should operate to heat the building to 16°C during the night.

Now save the file as a template (File > Save > Save as template) calling it something like **Simple HVAC Example 2.dsb**.

Radiant Heating using Simple HVAC

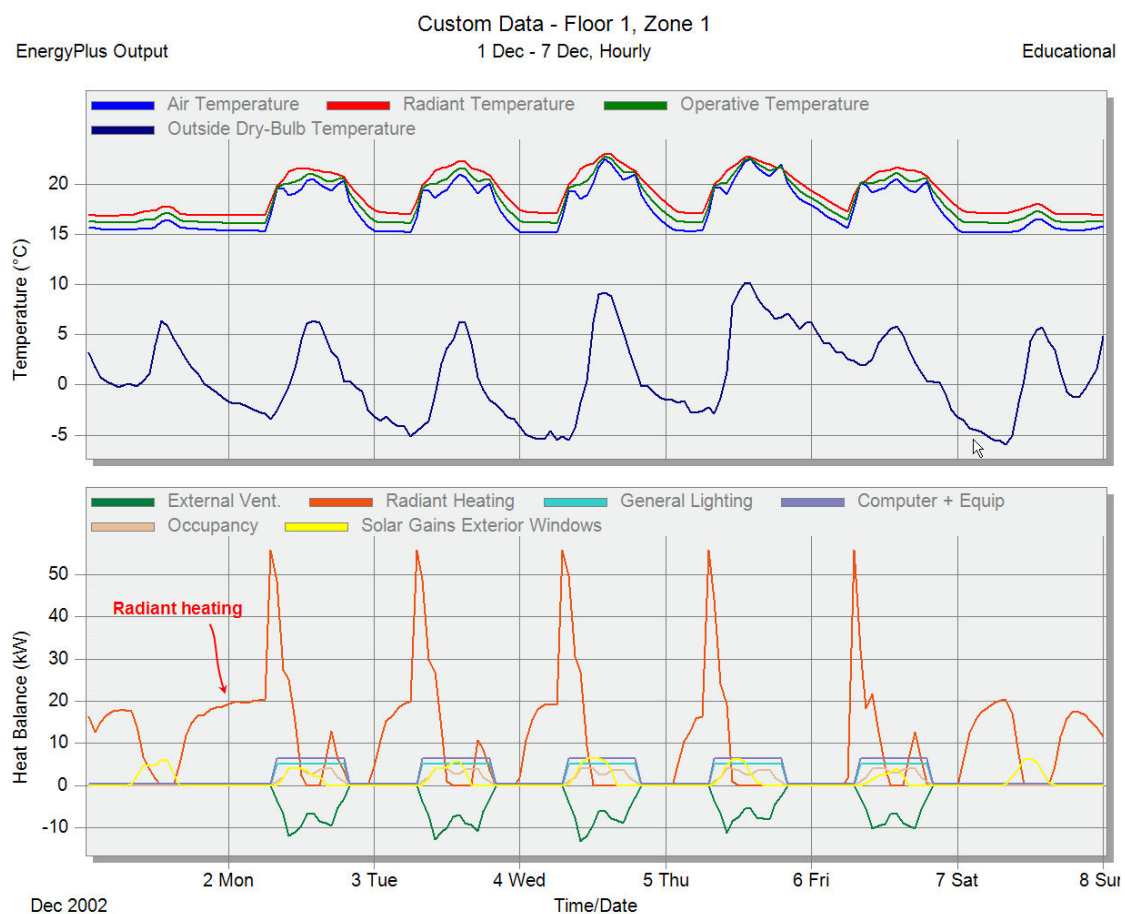
Radiant heating systems supply radiant heat to surrounding surfaces and to building occupant. **Convective** heating systems heat the air only. Radiant systems are often considered to be more comfortable than convective systems and can be more efficient if the occupants directly receive the radiation. Also convective systems can dry the zone air.

Example

In this example we look at how to model radiant heating systems using Simple HVAC.

1. Following on from the last example, at the building level on the HVAC load the **Underfloor heating system, nat vent** HVAC template.
2. Open the **Heating >Type** header and note that the **Heating type** is **2-Radiative/convective**. The Heating radiant fraction is 1 because all of the heat from the radiant system is to be applied in radiant form to the floor. The **Radiant distribution** is set to **2-Floor** because all the heat is to be radiated to the floor surfaces in the zone. The net effect is that all heat from the radiant/convective heating system is applied via the surface of the floor giving a good approximation to a floor heating system.
3. Set the Natural ventilation > Outside air definition method to **2-Min fresh air (Per person)**.
4. On the Activity tab set the natural ventilation cooling setpoint to **10°C** to ensure natural ventilation continues to provide fresh air when the building is occupied irrespective of the temperature in the zone.
5. Check the heating setpoint is **20°C** and the heating setback temperature is **16°C**.

Run a simulation for the design week requesting hourly results. Check that the building is being heated to at least 20°C during the occupied period and to 16°C during the night and weekend.



If you experiment further with this system type, you may notice that, as with real underfloor heating systems, the room temperature response is slower than with convective systems. This is because the heating takes place through surface of the

relatively massive floor construction and not directly to the air as with the Convective option.

Note that simulations with **Radiant/convective** heating systems **run** noticeably more **slowly** than the standard convective only Simple HVAC option. This can be a problem when these system types are used in buildings with hundreds of zones.

Version 2 of DesignBuilder will have other radiant heating options including radiant baseboards in Compact HVAC.

Night Cooling using Mechanical Ventilation

You can use DesignBuilder Simple HVAC to model thermostatic night cooling. This example takes you through the steps involved in setting up a basic early stage design study of a building using daytime natural ventilation to provide fresh air and night mechanical ventilation to pre-cool the building fabric before the start of occupancy.

Example

1. Starting from the Atrium example base file, at the building level on the HVAC tab, check that the Mechanical ventilation, Heating and Cooling are all switched on.
2. To ensure the mechanical ventilation occurs during summer nights and weekends only, **create a new schedule**. A quick way to do this is to copy the **Summer cooling workdays (Northern Hemisphere)** schedule and modify the copy (you can't modify DesignBuilder library components directly).
 - a. Click on **Schedule** under Mechanical Ventilation > Operation to open the list in the Info Data panel on the right.
 - b. Select **Summer cooling workdays (Northern Hemisphere)** under the general category.
 - c. Click on the **Create copy** icon above and immediately on the **Edit** icon to edit the copy.

Make these changes to the schedule:

- d. Change its **name** to something like **Summer night vent**.
- e. Then change the data to. The items marked in bold have been changed.

```
SCHEDULE:COMPACT,  
Summer vent,  
Fraction,  
Through: 31 Mar,  
For: AllDays,  
Until: 24:00, 0,  
Through: 30 Sep,  
For: Weekdays Weekends SummerDesignDay,  
Until: 06:00, 1,  
Until: 24:00, 0,  
For: AllOtherDays,  
Until: 24:00, 0,
```

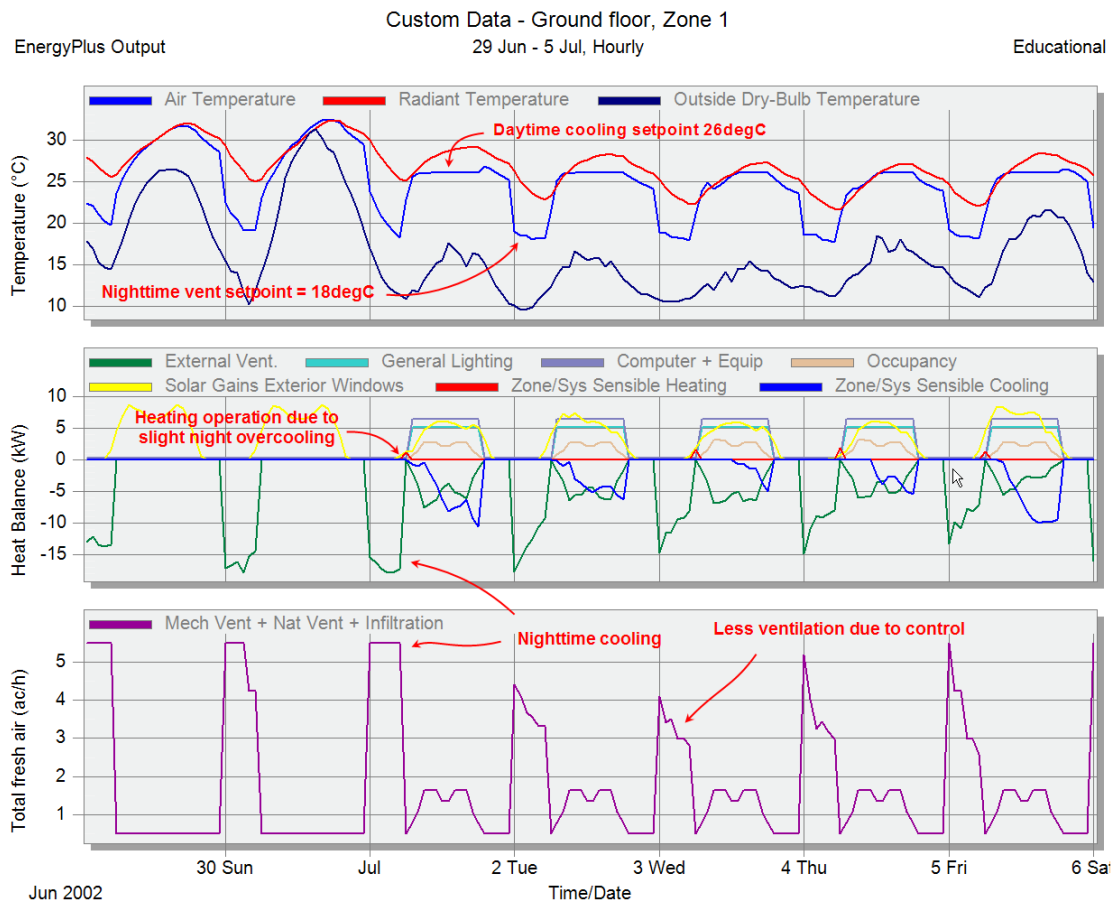
Through: 31 Dec,
For: AllDays,
Until: 24:00, 0 ;

This Compact schedule has values of 1 between 24:00 and 6:00 during weekdays, weekends and summer design days and zero at all other times. The timing will allow the fans to operate between midnight and 6am making use of cheap nighttime electricity tariffs.

3. Select the **Summer night vent** schedule you have just created as the Mechanical ventilation operation schedule.
4. Set the **Mechanical ventilation > Outside air flow** rate to **5 ac/h**.
5. Still at building level on the HVAC tab **switch on Natural ventilation** and select the **Min fresh air per person** option to provide fresh air for the occupants during the day. Leave the schedule as the default Office_OpenOff_Occ.
6. Now on the Activity tab at building level under the **Environmental Control** header make these changes:
 - a. Cooling setpoint = 26°C.
 - b. Natural ventilation cooling setpoint = 18°C.
 - c. Mechanical ventilation setpoint = 18°C. Night ventilation will switch off if the zone air temperature gets below 18°C (to prevent overcooling).
 - d. Mechanical ventilation Max in-out delta T = 2K. This ensures mechanical ventilation is only activated when the outside air is at least 2K colder than inside air temperature.

The model is now ready to start simulations to click on the **Simulation** tab and on the Simulation options dialog select **Hourly data** and **Summer typical week**. Press OK to start the simulation.

Select hourly results using the Interval drop-down list and you should see something like the screenshot below.



Note the following:

1. The night-time ventilation cools zone air temperature down to 18°C when possible.
2. The heating system operates first thing in the morning to raise the temperature of the zone from 18°C to the heating setpoint temperature of 20°C. An improved mode of operation for this building would be for the heating system to be switched off in the summer.

As an extra exercise you may like to investigate the effect of mechanical ventilation rate on the load reduction benefit due to night cooling. Try using this model as a basis for a parametric study. You could carry out annual simulations and use the annual CO₂ emissions as the criteria.

18 DesignBuilder Compact HVAC

In DesignBuilder 'HVAC' refers to all elements of the building whose function is to provide heating, cooling and ventilation to provide comfort for the occupants. This includes natural ventilation, but this topic is covered in a separate Tutorial.

DesignBuilder offers a range of HVAC system types and also 2 levels of detail:

- **Simple** where the system is modelled by calculating the loads required to meet the heating and cooling setpoints and multiplying heating and cooling loads by system CoPs to calculate energy consumption.
- **Compact** using the EnergyPlus Compact HVAC (called 'HVACTemplates' in EnergyPlus v3 and later). This allows a much more detailed model of some HVAC system types to be developed but using simple parametric input data that can be loaded from templates.

Compact HVAC allows you model a wider range of different HVAC types and control options than Simple HVAC and it can be useful in the design of the HVAC system itself.

There is no need to draw networks of ducts and pipes. The expansion of simple parametric definitions in the DesignBuilder interface to detailed EnergyPlus HVAC description is done in the background automatically using the EnergyPlus Compact HVAC data.

The sizing of the detailed components is carried out automatically before the simulation using heating and cooling design data and **WinterDesignDay** and **SummerDesignDay** schedule data. Summary results for these sizing calculations can be found in the EnergyPlus output file **eplusout.eio** and details in the files **eplusssz.csv** and **epluszsz.csv**.

The 5 EnergyPlus Compact HVAC system types currently available are:

- **Fan coil units** – for modelling fan coil units.
- **Unitary Single Zone** – for modelling constant volume packaged DX systems, split systems.
- **Unitary Multizone** – for modelling multizone DX systems where a single AHU supplies multiple zones.
- **CAV**- Constant Air Volume systems with central AHU.
- **VAV** - Variable Air Volume systems with central AHU.

Zone-based Systems

The Compact HVAC zone-based system types are:

- Unitary single zone
- Fan coil units

Zone-based systems have separate data for each zone and there is no central AHU. Data for these systems is defined at the zone level (though you can of course use inheritance and define data at the building and block level if most zones in the building/block have the same type). These HVAC system types can be used in much the same way as Simple HVAC in the sense that the system for each zone is self-contained without reference to AHU data.

AHU-based Systems

The Compact HVAC AHU-based system types are:

- Unitary multizone
- VAV
- CAV

These system types require data to be set at the building level to define the characteristics of the Air Handling Unit (AHU). Data for zone terminal units, fresh air requirement, heating/cooling requirement and setpoints etc is set at the zone level (or inherited from building or block levels above in the hierarchy).

More data is required to define the AHU-based Compact HVAC systems, especially the VAV and CAV system types, and these systems do involve more complexity than with Simple HVAC. Some care is therefore required in setting the model up to avoid simultaneous heating and cooling, under/over heating/cooling etc. It is essential to check the HVAC system operation by looking at hourly or sub-hourly results for short summer and winter periods.

Boiler, Chiller, Fans, Pumps Fuel consumption

DesignBuilder Compact HVAC models heating and cooling generation (i.e. boilers, chillers, heat pumps etc) in a similar way to Simple HVAC using seasonal CoP values. However unlike Simple HVAC CoP values are entered specifically for the heat generator (typically a boiler or heat pump) and for the chiller. They are not whole system CoPs as in Simple HVAC.

For example, in a Compact HVAC fan-coil unit provided with heat from a boiler, EnergyPlus calculates the heating requirement of the heating coil and DesignBuilder calculates the boiler energy consumption based on the heating coil load, a distribution loss factor and the Heat generation CoP for the boiler entered by the user.

Fans are modelled within EnergyPlus based on the supply airflow rate, fan pressure drops and efficiency values.

Pumps are also modelled in Compact HVAC systems using hot or cold-water coils but there is no user access to the pump data. Default characteristics are assumed.

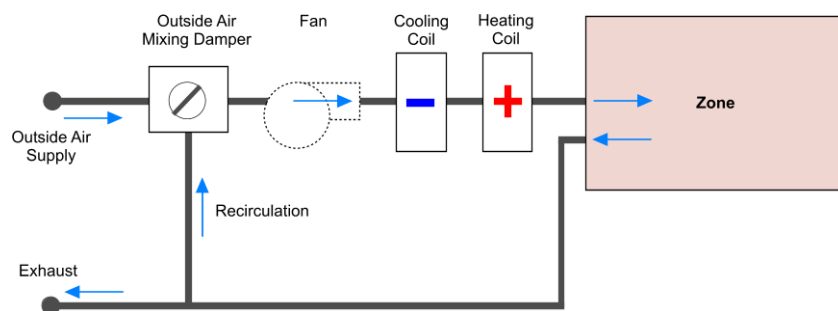
Fan-coil Units

The Fan coil unit Compact option simulates a 4 pipe fan coil unit with hot water heating coil, chilled water cooling coil, and an outside air mixer. Fan coils contain an

outdoor air mixer, a fan, a simple heating coil and a cooling coil. 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. The unit is controlled to meet the zone (remaining) heating or cooling demand. If there is a heating demand, the cooling coil is off and the hot water flow through the heating coil is throttled to meet the demand. If there is a cooling demand from the zone, the hot water coil is off and the chilled water flow through the cooling coil is throttled to meet the load.

You can model Fan coil unit systems with or without outside air. If you include Mechanical ventilation with your Fan coil unit system then heating and cooling will only operate when the Mechanical ventilation operation schedule is on.

If you do not want to include outside air in your system, you should uncheck the Mechanical ventilation 'On' check box. In this case heating/cooling availability is determined entirely from the heating/cooling operation schedules under the Heating and Cooling headers.



Example

To get started with Compact HVAC in an existing Simple HVAC model the first thing to do is to open the Model options dialog and set the HVAC model detail to Compact. In this example we will start using the Atrium example base template.

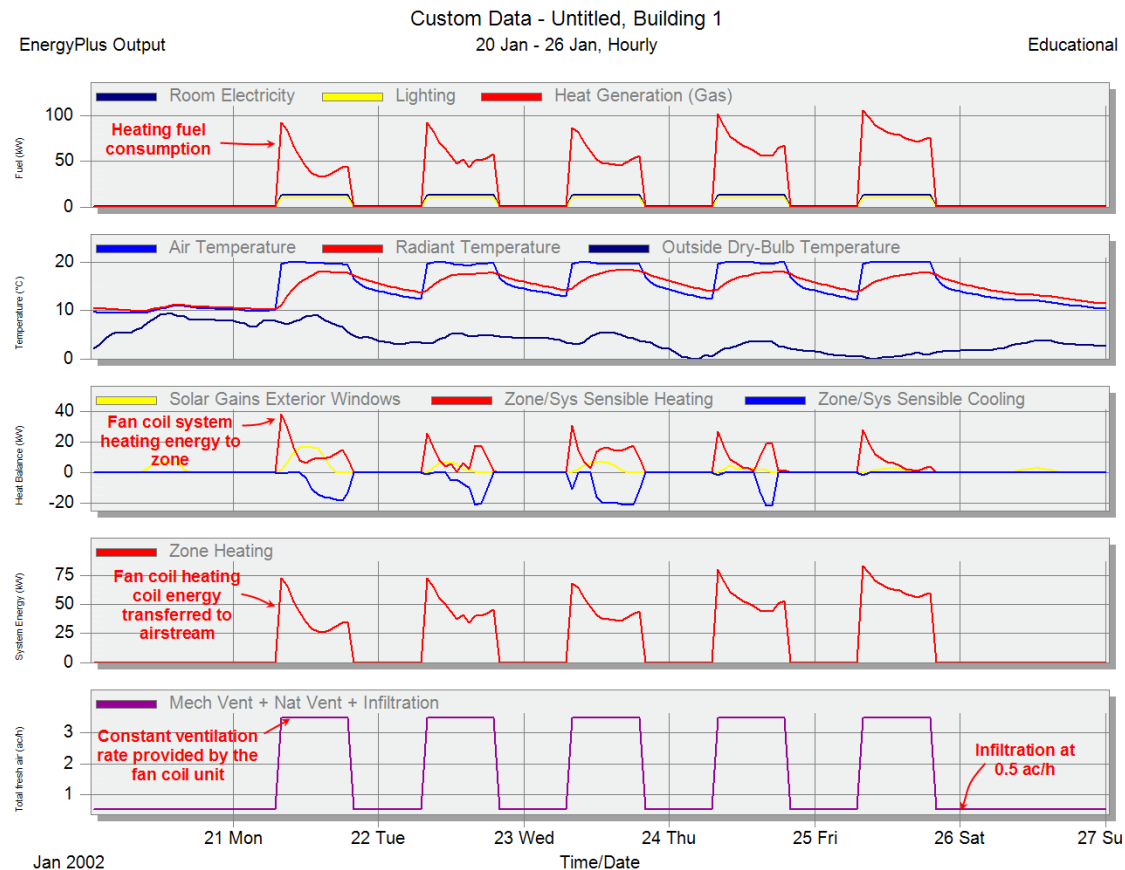
1. Load Create a **new project** and select **London Gatwick** as the location and select the **Atrium example base** template.
2. Open the Model options dialog and set the HVAC model detail to **Compact** and confirm. The model now uses Compact HVAC.
3. At the building level go to the HVAC tab and load the **Fan-coil unit** HVAC template.

You should notice at this point that Compact HVAC Fan-coil units have very little extra data to enter relative to Simple HVAC. The only difference in data input is use of Heat generator CoP, Heating and Cooling Distribution losses and the Cooling coil type.

You will also see below the HVAC template the **Type** of the Compact HVAC system last loaded. This cannot be edited and it is important to understand that the correct template must be used to load the appropriate Compact HVAC type.



Now start a simulation by clicking on the **Simulation** tab, selecting **hourly** results and **Winter typical week**. When the simulation finishes, select **Hourly** results in the Display options panel and you should see something like the screenshot below.



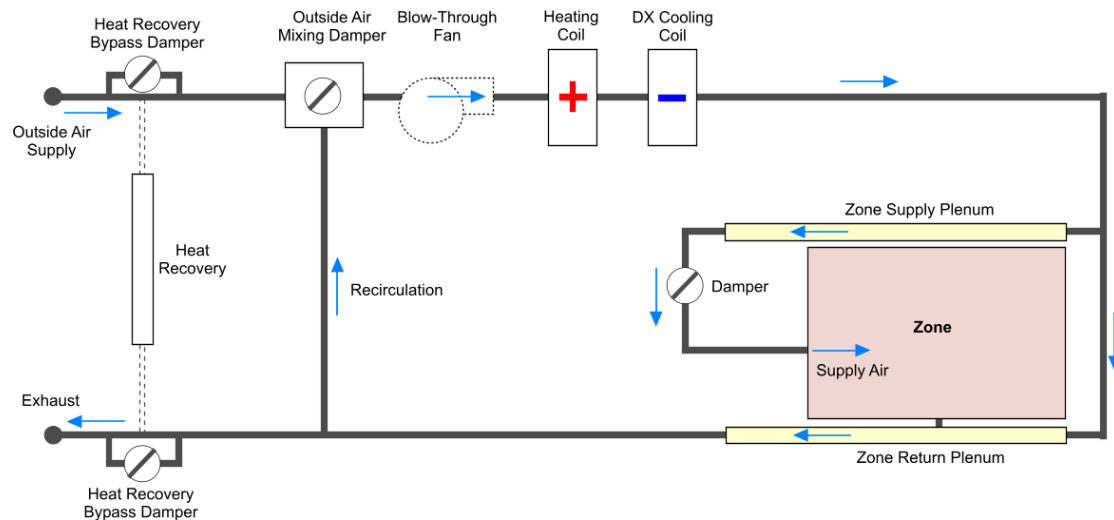
The important things to notice about these results for fan-coil units are:

- Unlike Simple HVAC, in Compact HVAC EnergyPlus calculates loads on the coils (i.e. heat transferred to air stream) and this data is displayed in the System energy graph (above). Heating fuel consumption is calculated from this by applying distribution loss and heat generation CoP factors.
- The outside fresh air rate provided by fan-coil units is 'full on' when the Min AHU Outside Air Requirement Schedule has values greater than zero. There is no modulation between no outside airflow and full outside airflow.

Now save the file as a template (File > Save > Save as template) calling it something like **Fan Coil Units.dsb**.

Unitary Single Zone - Basic

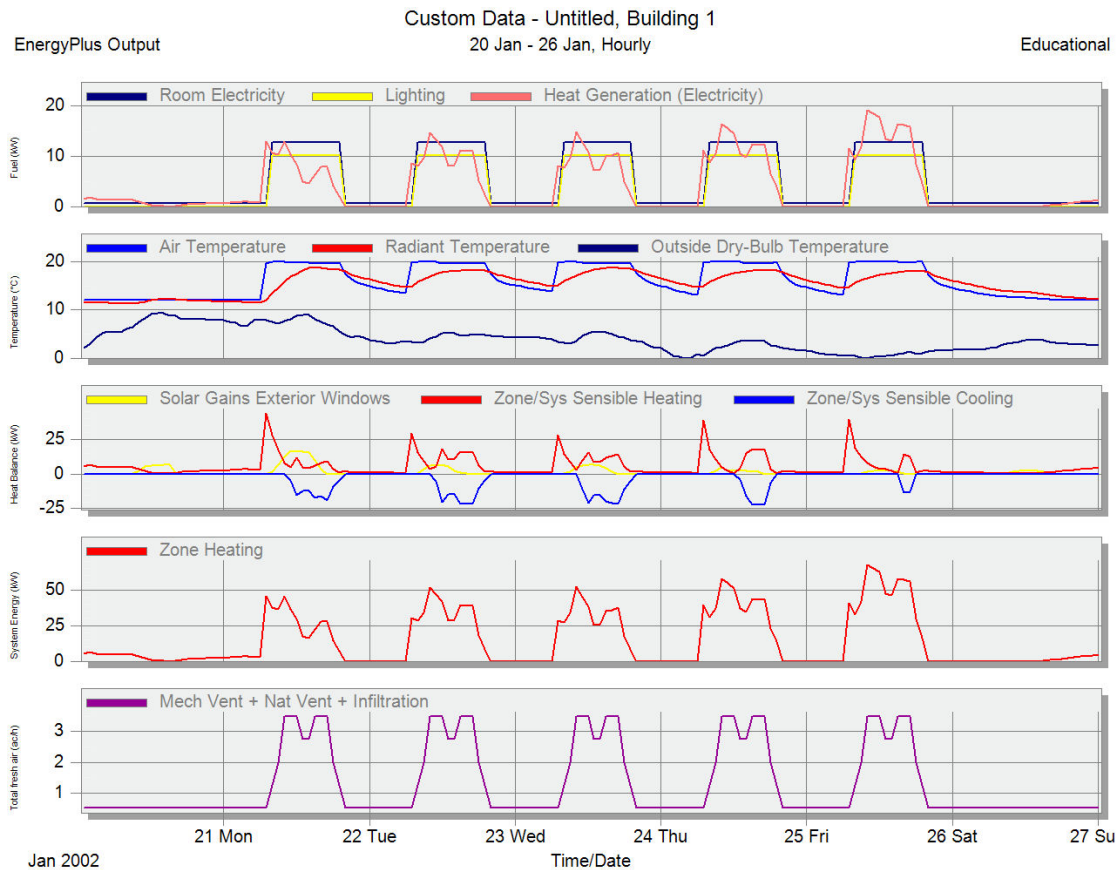
The Unitary single zone option allows you to model simple single constant volume direct-expansion (DX) based HVAC configurations with several different heating options. Direct expansion includes single packaged rooftop systems commonly seen in commercial buildings and split systems commonly seen in residential buildings. In this system, local cooling coils are used to condition air, which is supplied to the zone through a local air delivery system. Air delivery can be set up to ensure minimum fresh air supply and there are options for economisers to provide free cooling and heat recovery.



Example

Starting where you left off in the Fan coil unit example, go to the HVAC tab at building level and load the **Packaged direct expansion** HVAC template. This uses the Compact HVAC Unitary Single Zone type.

Take a look at the data on the HVAC tab. You will notice that in addition to the data previously mentioned for Fan coil units, Unitary single zone systems also allow Economisers, Fan operation mode and Heat recovery data to be entered. For now accept these default settings and run the simulation again to compare the results with the Fan coil system.



The most obvious differences between these results for the Packaged direct expansion system relative to the Fan coil results are that the mechanical ventilation fresh air rate is modulated according to the mechanical ventilation schedule and also that the heating fuel energy consumption is much lower because the packaged direct expansion system uses a heat pump with a high heating CoP (3.5) whereas the Fan coil system uses a boiler with CoP of 0.83.

Now save the file as a template (File > Save > Save as template) calling it something like **Unitary Single Zone – Basic.dsb**

Unitary Single Zone – Economiser

Unitary single zone systems also allow an economiser to be specified. An economizer is a damper + opening that allows up to 100% outside air to be supplied to the air-handling unit when the outside air is cooler than the return air from the zones, thus providing free cooling. Economisers can reduce energy use quite significantly, especially for buildings liable to overheat when the outside air is relatively cool.

EnergyPlus automatically sizes the maximum outside airflow rate and the economiser can progressively open the outside air damper to maximise use of free outside air cooling.

There are four Economiser options:

- **1-None** where no economizer operation will be simulated and outside fresh air is based on the Mechanical ventilation Outside air rate and associated operation schedule.
- **2-Return air temperature** establishes a limit control on the return air temperature so that when the temperature of the air entering the mixer on the outside air side is greater than the return air temperature the outside airflow rate is set to the minimum.
- **3-Return air enthalpy** establishes a limit control on the return air enthalpy so that when the enthalpy of air entering the mixer on the outside airside is greater than the return air enthalpy the outside airflow rate is set to the minimum.
- **4-Return air temperature and enthalpy** where both the above limit controls are in operation, i.e. both temperature or enthalpy limits are used so that when either the temperature or the enthalpy of the air entering the mixer on the outside air side is greater than the return air temperature/enthalpy, the outside airflow rate is set to the minimum.

In addition to the above checks in which return air and outside air conditions are compared, the economiser can also have absolute limits on outside air temperature and enthalpy. When the outside air conditions go beyond these limits the EnergyPlus economiser switches to minimum outside air mode, i.e. the outside air damper is closed to its minimum position and heat recovery will be enabled if available (more on this later). When you select one of the Economiser options 2-4 (above), the temperature and enthalpy limits below are shown in the interface.

- **Economiser upper temperature limit** is the outside air temperature high limit for economizer operation. If the outside air temperature is above this limit, the outside airflow rate will be set to the minimum.
- **Economiser lower temperature limit** is the outside air temperature low limit for economizer operation. If the outside air temperature is below this limit, the outside airflow rate will be set to the minimum.
- **Economiser upper enthalpy limit** is the outside air upper enthalpy limit for economizer operation. If the outside air enthalpy is above this value, the outside airflow rate will be set to the minimum.

Also displayed is the **Lockout** control allowing the Economiser to set minimum outside air when heating and/or cooling is in operation.

Example

Now test the effect of the standard Economiser settings.

1. First set the Mechanical Ventilation > Outside air rate to 3 ac/h. This will help us to interpret outside fresh air rate when checking the operation of the economiser.
2. Select the **2-Return air temperature** Economiser type
3. Leave the Economiser settings in their default state.

The HVAC data at building level should look like the screenshot below.

HVAC Template

Template Packaged direct expansion

Type 1-Unitary single zone

System Availability

Schedule On

Mechanical Ventilation

☒ **On**

Outside air definition method 1-By zone

Outside air (ac/h) 3.000

0 2 4 6 8 10 12 14 16 18 20

Min AHU Outside Air Requirement

Schedule Office_OpenOff_Occ

Fans

Fan operation mode 1-Continuous

Pressure rise (Pa) 50

Total efficiency (%) 70

Fan in air (%) 100

Economiser

Type 2-Return air temperature

Lockout 1-No lockout

Upper temperature limit (°C) 25.0

Lower temperature limit (°C) 11

Upper enthalpy limit (J/kg) 50000

Heat Recovery

☐ **On**

Heating

☒ **Heated**

Fuel 1-Electricity from grid

Heat generation CoP 3.500

Heating distribution loss (%) 0.0

Operation

Schedule Office_OpenOff_Heat

Cooling

☒ **Cooled**

Fuel 1-Electricity from grid

Chiller CoP 2.500

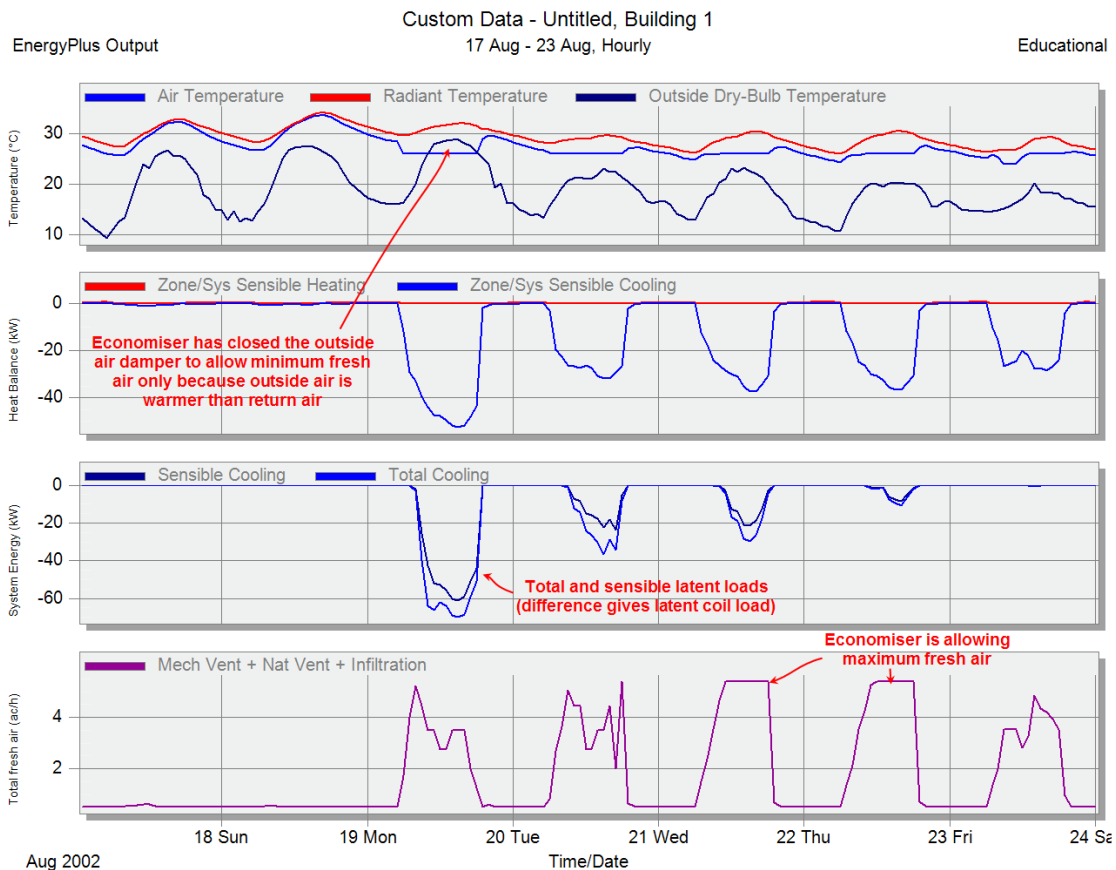
Cooling distribution loss (%) 0.0

Operation

Schedule Office_OpenOff_Cool

Now run a simulation requesting **Hourly** results and select the **Summer design week**.

The results should look something like the screenshot below.



You should see:

1. The effect of the economisers providing free cooling, with large outside air flows being introduced when the criteria are satisfied.
2. The separate display of the Sensible and Total (sensible + latent) coils loads. This is the cooling energy transfer from the coils to the HVAC air stream. The latent energy is essentially the moisture that has condensed out onto the cold coils. Sensible is the heat based on dry-bulb temperature only.

Unitary HVAC systems use direct evaporation (DX) coils, i.e. the refrigerant is evaporated directly in the coils. DX coils usually operate at lower temperatures than chilled water coils served by a chiller and so the latent loads can be higher with more water evaporating out on the cold DX coils.

Now save the example file as a template (File > Save > Save as template) calling it something like **Unitary Single Zone – Economiser.dsb**.

Unitary Single Zone – Economiser + Lockout

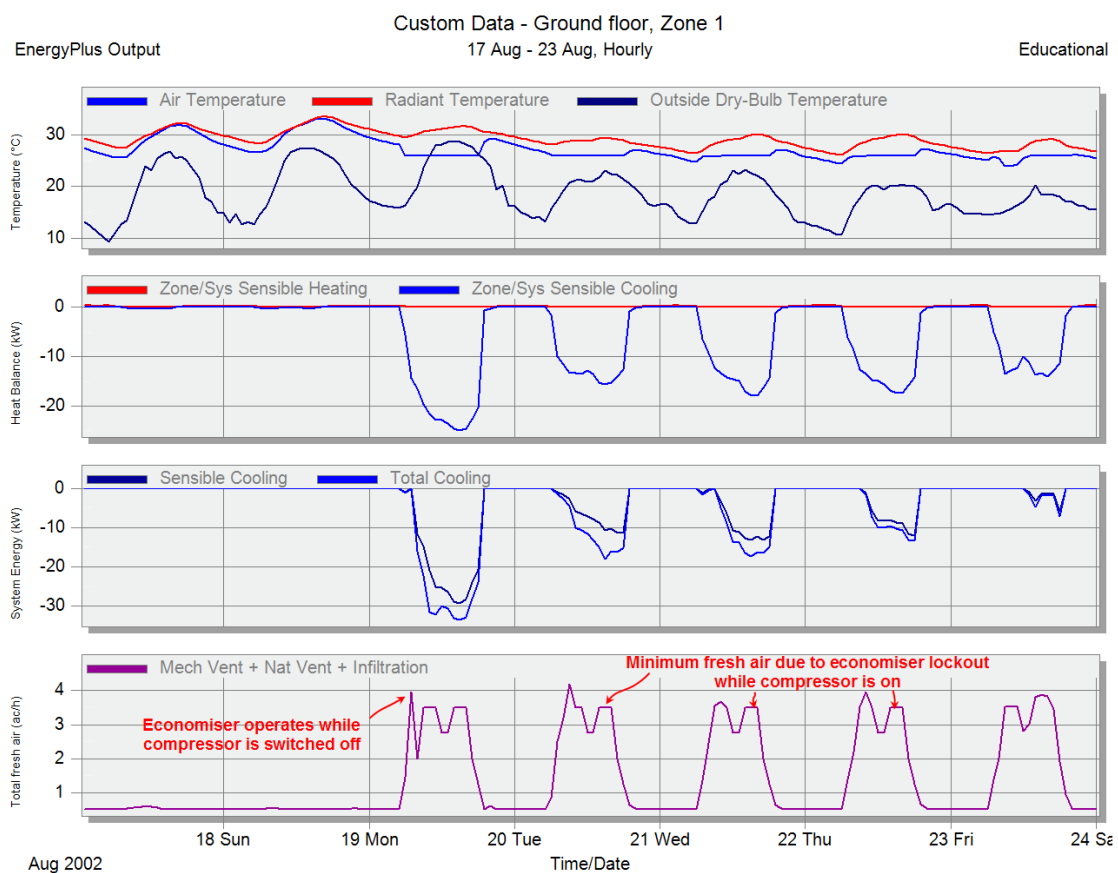
Lockout is where the outside air damper can be set to minimum fresh air when either heating or cooling is in operation. Settings are under the Mechanical Ventilation > Economiser header on the HVAC tab.

The 3 options are:

- **No lockout** where the Lockout control is not active and heating or cooling can occur simultaneously with economiser fresh air.
- **Lockout with heating** allows the Economiser to set minimum outside air when the heating system is in operation. This avoids the need to heat large volumes of mainly outside air when the HVAC system is active.
- **Lockout with compressor** allows the Economiser to set minimum outside air when either heating or cooling is in operation. This avoids the need to heat/cool large volumes of mainly outside air when the HVAC system is active but can restrict the savings possible by economiser action.

Example

Starting from the Unitary Single Zone - Economiser Example model, select the **Lockout with compressor** Economiser option and run the simulation again (hourly results, summer design week). You should see hourly results like those below.



Note how the Lockout control switches off the Economiser while the compressor is switched on.

Save the example file as a template (File > Save > Save as template) calling it something like **Unitary Single Zone – Economiser + Lockout.dsb**.

Unitary Single Zone – Economiser + Heat Recovery

With Compact HVAC, you can specify air-to-air heat recovery, which is available between the relief air stream and the outside air inlet stream. You can also combine economisers with heat recovery.

You can select the type of air-to-air heat recovery. The choices are:

- **1-Sensible** - recovers sensible heat
- **2-Enthalpy** - recovers sensible and latent heat.

Sensible heat recovery effectiveness - is applied as a constant effectiveness at all conditions. The valid range is from 0.0 to 1.0, and the default is 0.70.

Latent heat recovery effectiveness - if Heat Recovery Type is Enthalpy this is applied as a constant effectiveness at all conditions. The valid range is from 0.0 to 1.0, and the default is 0.65.

Heat recovery heating setpoint temperature (Unitary systems only)

When no economiser is selected this data defines the heating setpoint temperature of the heat recovery device for Unitary and Unitary multizone systems. Note that the single value is applied throughout the year. For example enter 15C to model a system which heats outside air up to 15C and no higher. A high setpoint value will provide optimal heating but this may add to cooling loads in the summer.

Heat recovery rules

If an economiser option is selected, the heat recovery is bypassed whenever the economiser operates and the heat recovery is always bypassed if economiser operation would be beneficial. The heat recovery will operate when:

- Heat recovery is selected in the interface **AND**
- Economiser operation is off.

You can have heat recovery and no Economiser action by switching on Heat recovery and setting economiser to 1-None. In this case the heat recovery unit heats outside air using the Heat recovery heating setpoint temperature. It also cools outside air when return air temperature is lower than outside air temperature.

If the economiser is selected together with heat recovery then the Heat recovery heating setpoint temperature data is not shown and the effective heat recovery setpoint temperature is given by the lower economiser limit temperature.

Example

To test this, switch off the Lockout control used in the last example and open the **Heat Recovery** header and switch on heat recovery. Leave the default heat recovery settings (**Sensible** only type with Sensible heat recovery effectiveness of 0.7). The Mechanical ventilation part of the HVAC data should look like the screenshot below.

Mechanical Ventilation

☒ **On**

Outside air definition method: 1-By zone

Outside air (ac/h): 3.000

Min AHU Outside Air Requirement

Schedule: Office_OpenOff_Occ

Fans

Economiser

Type: 2-Return air temperature

Lockout: 1-No lockout

Upper temperature limit (°C): 25.0

Lower temperature limit (°C): 11.0

Upper enthalpy limit (J/kg): 50000.0

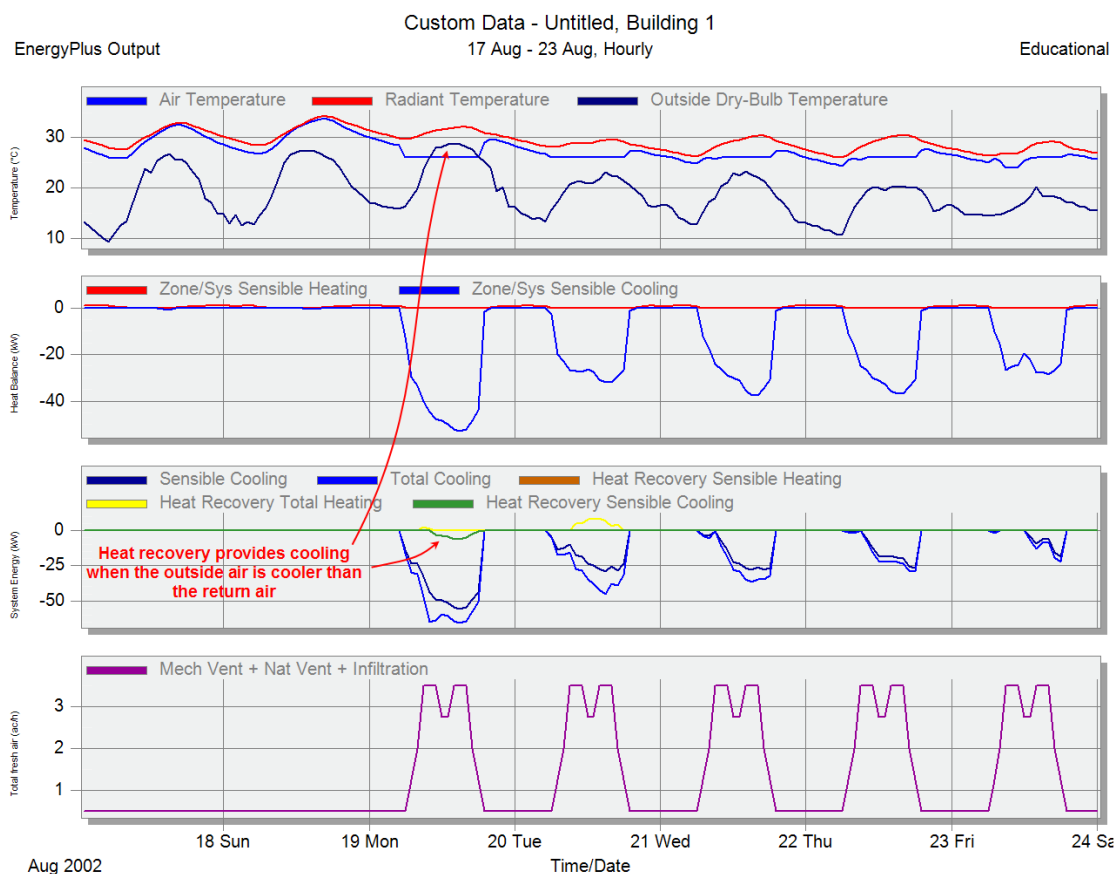
Heat Recovery

☒ **On**

Heat recovery type: 1-Sensible

Sensible Heat Recovery Effectiveness: 0.700

Run the simulation again (hourly results, summer design week). You should see hourly results like those below.



Note that Sensible cooling is provided by the heat recovery on the 19th August during times when the outside air is warmer than the return air from the zone. At other times the heat recovery unit is bypassed.

Now run the winter design week using the same model and check results. You should find that the heat recovery provides very significant savings during the heating season.

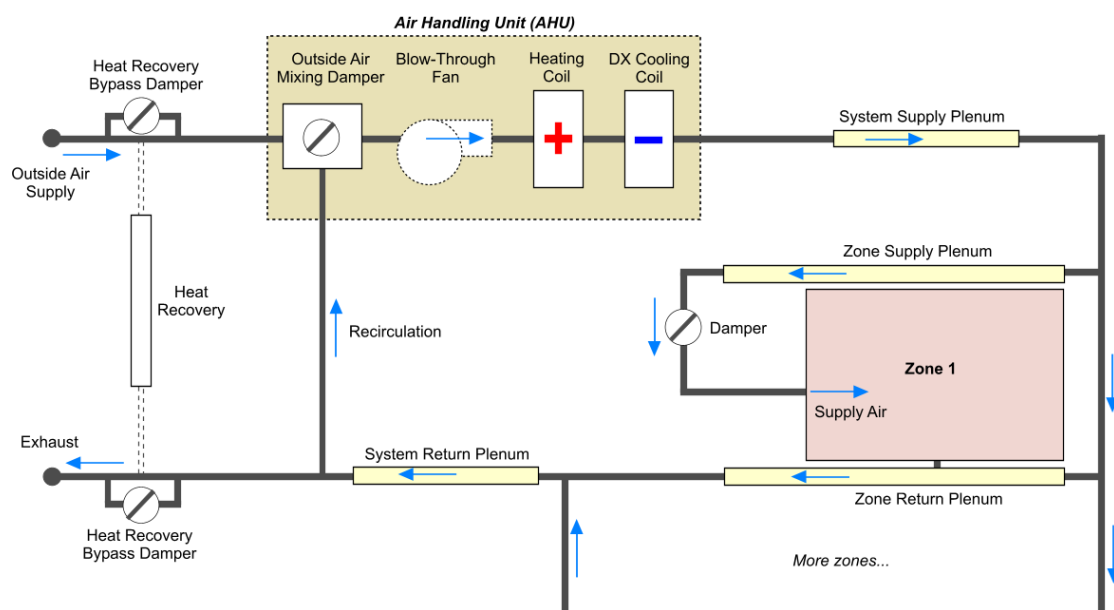
Save the example file as a template (File > Save > Save as template) calling it something like **Unitary Single Zone – Economiser + Heat recovery.dsb**.

Unitary Multizone – Basics

The Unitary multizone option allows you to model constant volume direct-expansion (DX) based HVAC configurations with several different heating options. Direct expansion includes single packaged rooftop systems commonly seen in commercial buildings and split systems commonly seen in residential buildings. In this system, central heating and cooling coils are used to condition air, which is delivered to the each of the zones in the system through an air delivery system. Air delivery can be set up to ensure minimum fresh air supply and there are options for economisers to provide free cooling and heat recovery.

The **Unitary multizone** system is essentially the same as the **Unitary single zone** system except that the same AHU supplies multiple zones, and one of these zones is used as the control zone.

Only one **Unitary multizone** system can be defined per building and it can serve up to **400 zones**.



A **Unitary multizone** HVAC template must be loaded at building level and in all zones which are part of the system (zones that are part of the Unitary multizone system can of course inherit their setting from the building level). Zones that are not part of the Unitary multizone system can have Compact HVAC type **Unitary single zone** or **Fan coil units** but not **VAV** or **CAV**.

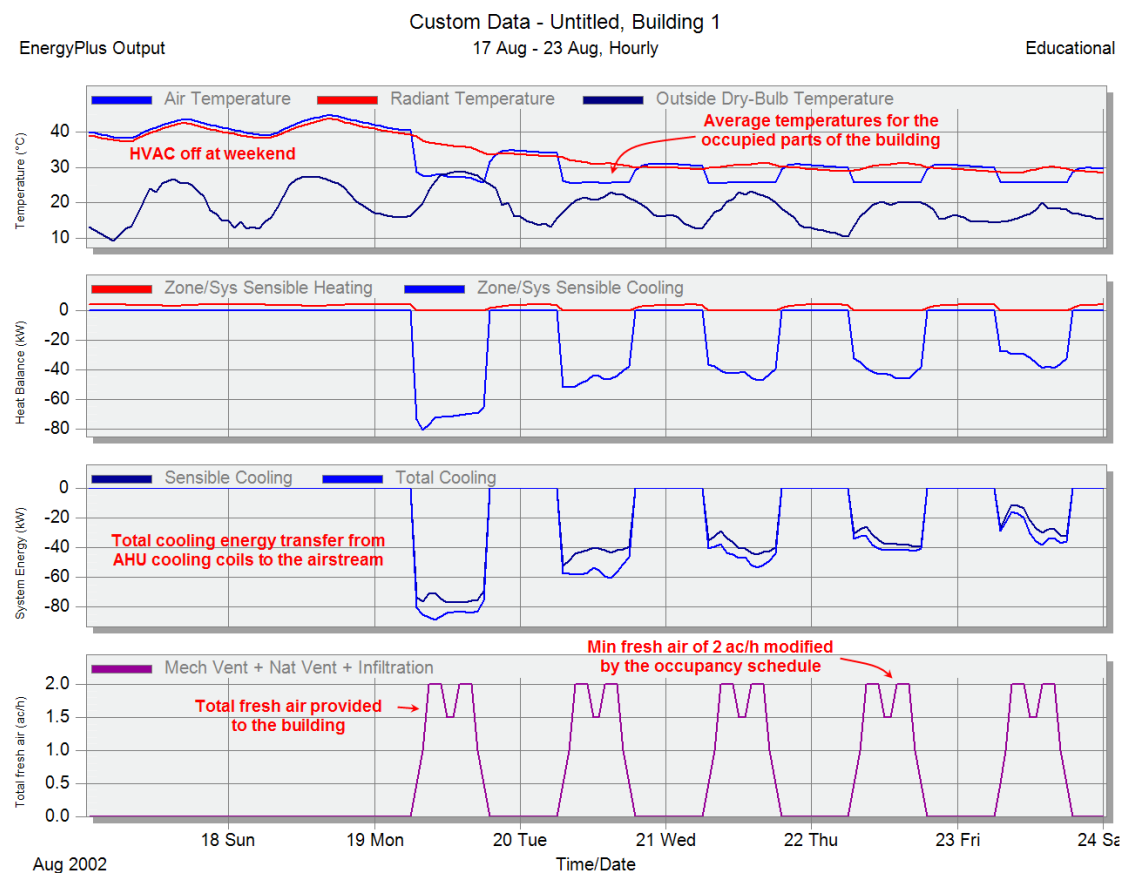
When using this system type you should be careful to locate the thermostat in a **Control zone** which will have a representative temperature for the whole building, otherwise some zones may become over/under heated/cooled.

Example

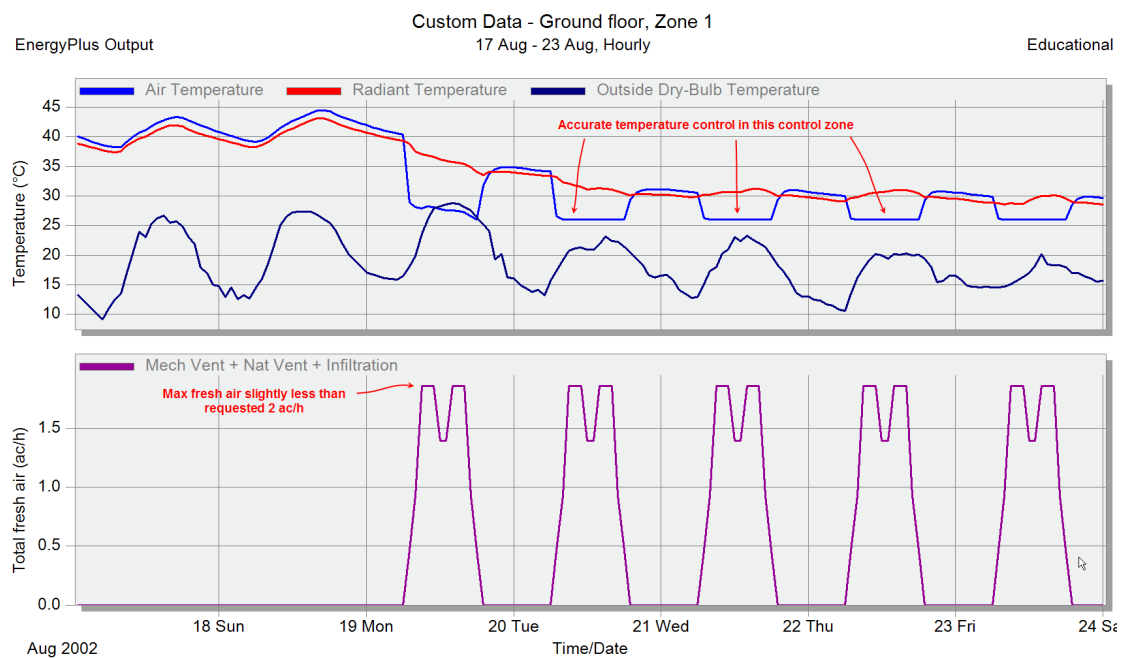
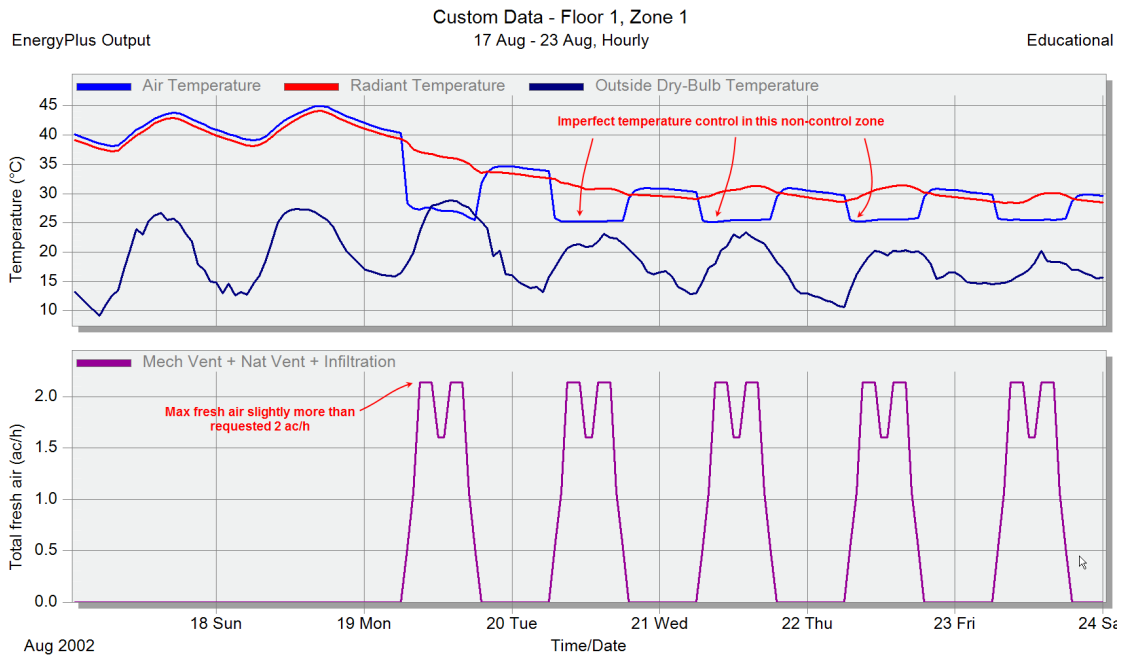
To create a basic 2-zone Unitary multizone model:

1. Starting from where you left off in the last example, at building level on the HVAC tab load the **Constant volume DX (using unitary multizone)** HVAC template. This data should be inherited by the zones below (ie no red hard set data at the block or zone level).
2. Set the Mechanical ventilation **Outside air definition method** to **1-By Zone**. This means that the **Outside air** ac/h rate entered on the slider below will be used directly.
3. Set the Mechanical ventilation **Outside air** delivery to be **2 ac/h**.
4. Leave the Mechanical ventilation **Operation schedule** to be **Office_OpenOff_Occ**.
5. On the Constructions tab at building level set the **Infiltration** to be **off**. This will make it easier to focus on outside fresh air delivery through the HVAC system.
6. Go to **Ground floor > Zone 1** and set this as the Control zone by checking the **Thermostatic control zone for unitary system** checkbox.
7. Run a simulation requesting **hourly** results and use the **Summer design week**.

The building level hourly results should look the screenshot below.



Notice how the total fresh air delivery to the building is 2 ac/h as specified. Now look at results for the Ground and First floor zones.



These show that the maximum values of fresh air delivery in each zone are not 2 ac/h as one might expect. The fresh air delivery in the ground floor zone is slightly higher than 2 ac/h and in the First floor zone it is slightly lower. To understand why this is you must first understand how the EnergyPlus fresh air control works.

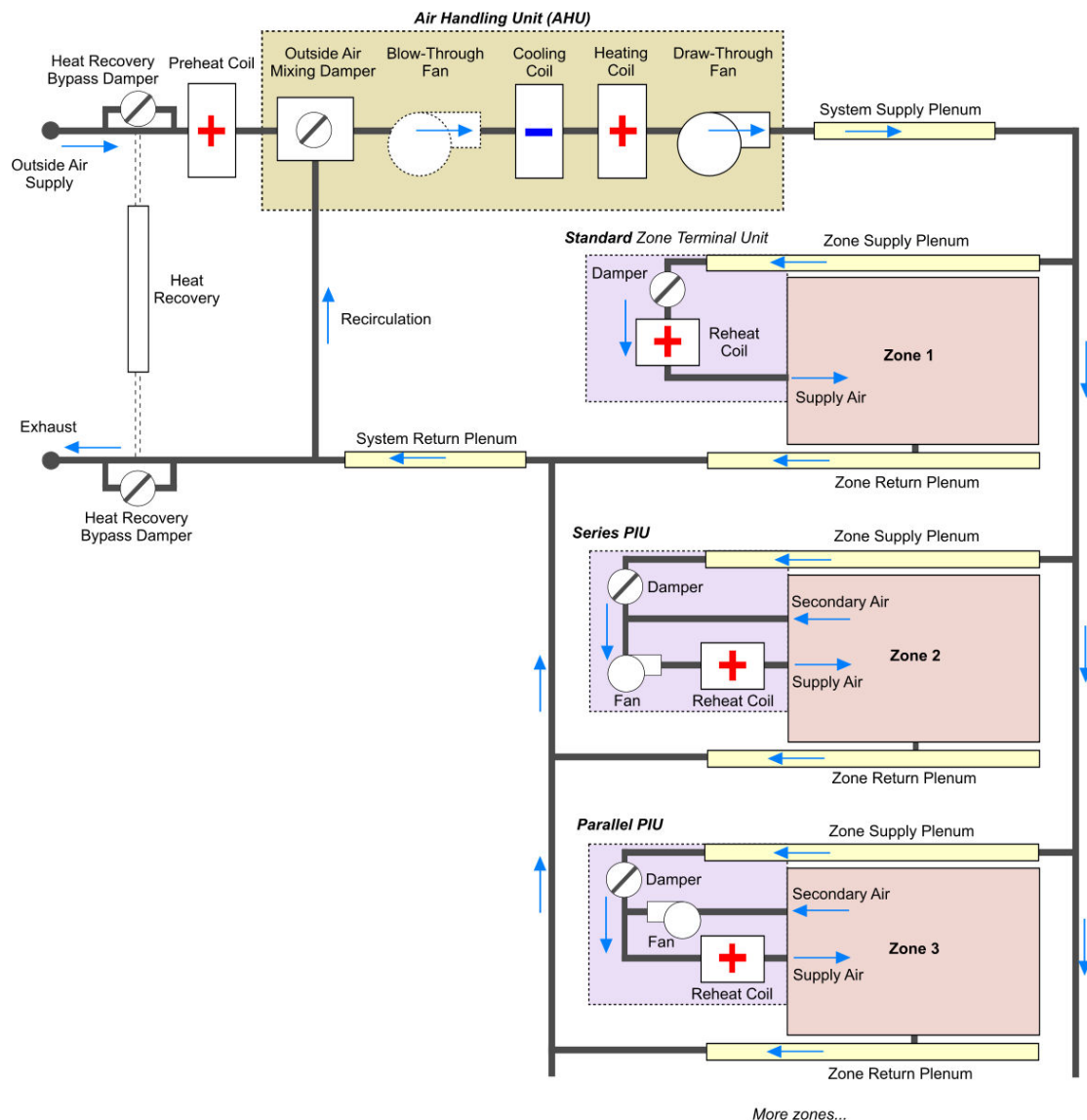
Before the start of the simulation EnergyPlus calculates the **total fresh air** requested for each zone and uses this total as the minimum outside airflow rate entering the AHU inlet. During the simulation when minimum fresh air is requested (i.e. economiser is not in operation) the outside air damper is adjusted to provide this minimum fresh air to the AHU. How this fresh air is distributed to the various zones in the building depends on the component sizing calculations automatically carried out before the simulation based on heating and cooling design days.

Advanced user note: see **eplusout.eio** in the EnergyPlus folder for a summary of EnergyPlus component and system sizing results.

Save the example file as a template (File > Save > Save as template) calling it something like **Unitary MultiZone.dsb**.

VAV Systems

In a typical Variable Air Volume (VAV) system the AHU supplies air to the zone terminal units at a constant temperature. This constant temperature is achieved by blowing air across cooling and heating coils.



VAV systems can be used for core or perimeter zones with a common fan system, air temperature control, and reheating devices. Heating is provided with the zone terminal units by use of reheat coils.

The DesignBuilder VAV option allows you to model VAV systems with optional preheat, AHU heating and cooling coils and zone terminal reheat. You can select

series or parallel fan powered terminal units (also known as powered induction units or PIU). There are options to control the air distribution system in various ways including the ability to ensure minimum fresh air delivery to each zone, set VAV box minimum flow fraction and various outside air options including an economiser to provide free cooling when appropriate. You can also model heating and cooling coil temperature reset.

Only one VAV system can be defined per building and it can serve up to 400 zones.

Cooling

During times when a zone requires cooling the volume of air supplied to the zone is adjusted by opening/closing the zone supply air damper in the terminal unit until the zone temperature setpoint is achieved. If more cooling is required then more cool air from the AHU is supplied. If the provision of minimum fresh air would over-cool the zone then a reheat coil in the zone terminal unit operates to heat the air from the AHU so as to achieve zone thermal comfort. This cooling and reheating can make poorly designed VAV systems inefficient.

Heating

During times when heating is required the zone terminal unit damper normally closes to minimum opening position and the reheat coil in the zone terminal unit operates to maintain comfort in the zone.

Outside Air

Under the Outside Air Mixing header you can specify the outside air mixing type. Choose between:

- **1-Recirculation** - the outside airflow supply to the AHU is mixed with recirculated return air from the zones. This is done to save energy often in conjunction with an economiser to ensure that optimal use is made of outside air. When heat recovery is used with the Recirculation outside air mixing option, the economiser must be selected in the interface for the heat recovery to be by-passed.

For recirculation systems you can select the outside air control minimum flow type to be one of:

- **1-Fixed** - the minimum outside airflow rate is fixed as set above no matter what the actual system flow rate is. In practice even with **Fixed** outside air control minimum it is possible for the outside airflow rate into individual zones to be slightly different from the requested value. This is because the minimum outside air is controlled at the AHU level to ensure that the total fresh air entering the AHU is the total required for the building (the sum of the individual zone outside air requirements). But depending on the heating/cooling loads in each zone, some zones will require more supply air to meet the load than others. So the total fresh air entering the building will be as specified but fresh air distribution around the building may not be perfectly maintained.
- **2-Proportional** - the minimum outside airflow rate varies in proportion to the total system airflow rate.

The minimum outside *airflow* rate through the AHU outside air inlet is automatically calculated as the sum of the individual zone ventilation requirements. This is the 'design minimum'. During the simulation, 'minimum' and 'maximum' represent the limits for economizer operation.

If there is no economizer, then the minimum flow of outside air is delivered. If **1-Fixed** is selected, then this value will remain fixed (unless the total VAV system flow rate falls below this value, in which case, the system will deliver 100% outside air at whatever the system flow rate is).

If, **2-Proportional** is specified, then the operating minimum falls as system flow rate falls. This simulates a typical system with a fixed lower stop on the outside air supply damper.

Notes:

- Delivering **1-Fixed** requires enhanced control systems.
- To ensure minimum outside air is delivered you should use **1-Fixed**.
- **2-Full fresh air** - the airflow into the AHU is 100% outside air, without any recirculation. This is modelled in EnergyPlus by setting the supply airflow rate to the outside airflow rate. Note that when using this option, fixing the supply airflow rate to the outside air rate will in many cases mean that the system will not be capable of meeting the design cooling load. To increase the maximum cooling capacity of a Full fresh air VAV/CAV system you can increase the outside airflow rate (such systems could be very inefficient in energy use without heat recovery). When heat recovery is used with the 2-Full fresh air outside air mixing option, the heat recovery is by-passed when this is advantageous in reducing AHU cooling coil loads. The heat recovery also operates when the outside air temperature is less than 12°C.

Important note for Full fresh air systems: to ensure that there is no recirculation you must also set the HVAC **sizing factor** on the Simulation options dialog under the Advanced header to **1** (the default is 1.2)

Outside air sizing

When the Outside air definition method is 2-Min fresh air per person, sizing for minimum outside air (OA) flow rate is done using the Number of People value with no regard for the schedules. The total OA requirement for each zone on the air handler is summed and used to set the minimum OA flow rate for the air handler. The OA controller sets an OA flow rate for the air handler each time step. After that, each zone receives a share of the total OA flow proportioned by the current supply flow rate to each zone. There is no further zone-by-zone allocation of OA quantities (which is what happens in a real system).

VAV Terminal Units

VAV systems control the dry bulb temperature inside a zone by varying the supply air volume instead of the supply air temperature. At full cooling the VAV terminal unit damper is fully open supplying the maximum airflow rate. As the cooling load decreases, the damper closes until it reaches the minimum stop specified by the turndown ratio (zone minimum airflow fraction).

The Turndown ratio (supply air minimum flow fraction) is the fraction that the zone VAV terminal unit can reduce flow to relative to the supply air maximum flow rate. The minimum flow rate to the zone while the system is operating is specified as a fraction of the maximum airflow rate. The turndown ratio 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 Zone damper heating action is set to 1-Nomal (the default).

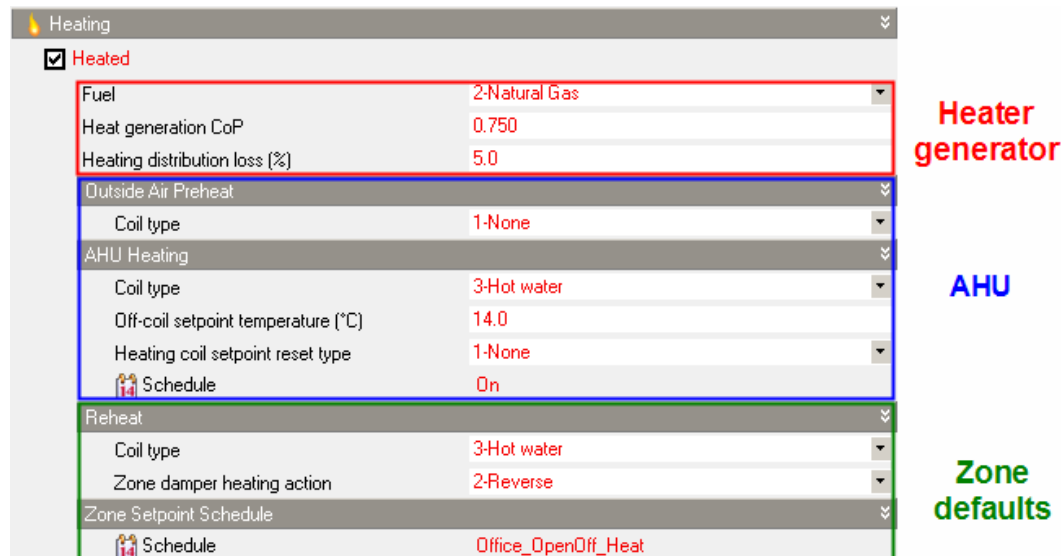
A value of 0 means the terminal unit can completely shut off flow into the zone and a value of 1.0 means that the terminal unit cannot reduce supply air at all (i.e. it is a constant volume system). For CAV systems this data is fixed at 1 and is not available for edit. Turndown ratio is typically in range 0.3 - 0.5. Minimum is 0 and maximum is 1.

VAV Terminal Unit Type – you can select from 3 types but we will only look at the Standard type in this example.

Heating Data

You should switch zone heating on if you want the zone to be heated to the heating setpoint temperature specified on the Activity tab. The Heating option should generally be on at building level for VAV systems.

At the building level you can define the data for the plant, AHU and the zone defaults. Under the Heating header you can specify data for the Heat generator, Preheat coils and the main AHU heating coil as well as setting the defaults for the zone terminal reheat units and zone heating setpoint schedule. See screenshot below.



Heating

☒ Heated

Heater generator

- Fuel: 2-Natural Gas
- Heat generation CoP: 0.750
- Heating distribution loss (%): 5.0

AHU Heating

- Coil type: 1-None
- Off-coil setpoint temperature (°C): 14.0
- Heating coil setpoint reset type: 1-None
- Schedule: On

Zone defaults

- Coil type: 3-Hot water
- Zone damper heating action: 2-Reverse
- Zone Setpoint Schedule: Office_OpenOff_Heat

Zone damper heating action

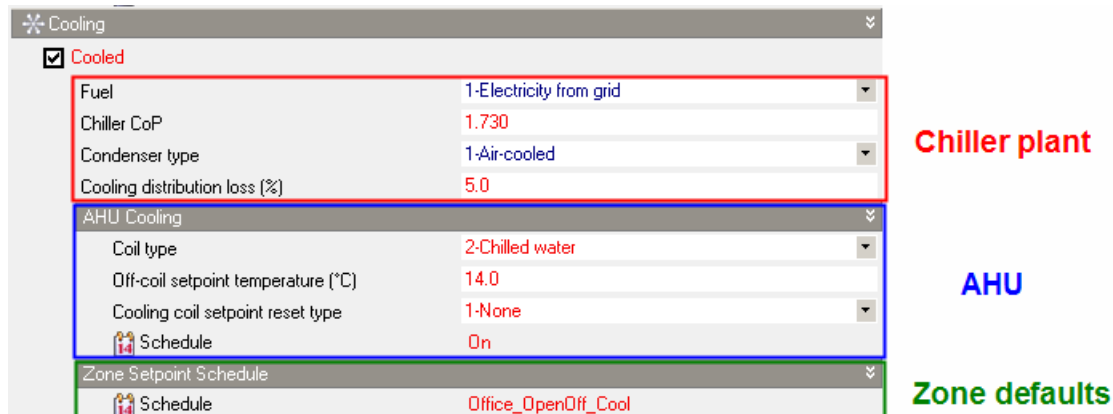
During heating operation, there are two control options for the damper controlling the air flow in the VAV terminal unit as the zone moves above or below the zone setpoint. With both control options, the damper is at the minimum airflow rate whenever the zone temperature is between the cooling and heating setpoints.

- **1-Normal** - the zone damper will remain at the minimum airflow 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. The system essentially acts like a constant volume system during heating.
- **2-Reverse** - the airflow rate through the zone terminal unit box may increase above the minimum when the heating demand cannot be satisfied. As the heating load increases, the unit starts at minimum airflow and minimum hot water flow. The hot water flow is increased until it reaches maximum flow, then the air damper starts to open to meet the load. This option is used if the minimum airflow rate is not adequate to serve the maximum heating load.

Cooling Data

Under the Cooling header you can specify data for the Chiller, main AHU cooling coil as well as setting the defaults for the zone terminal reheat units.

Note that the AHU cooling schedule is **On**. See screenshot below.



Chiller plant

AHU

Zone defaults

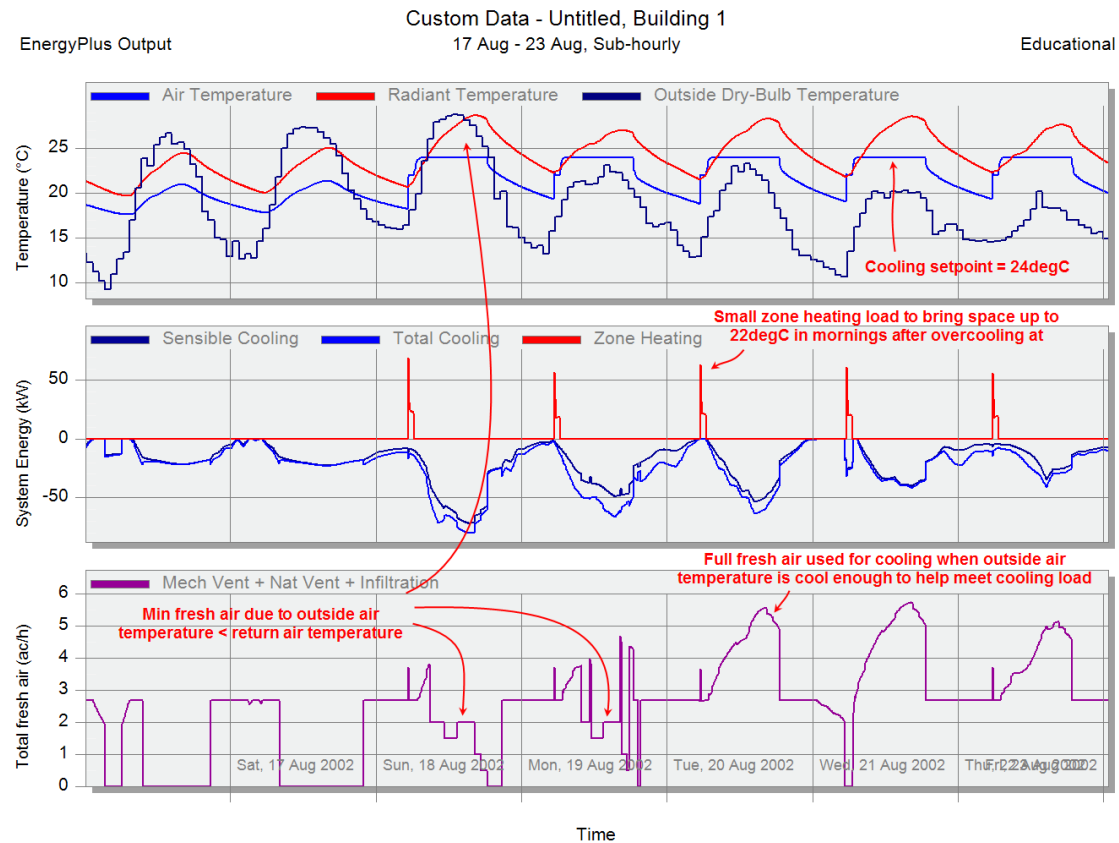
Exercise

The purpose of this exercise is to explore the VAV HVAC data, run a basic simulation and understand the results.

1. The starting point is the Unitary multizone example file from the previous exercise. So create a new project based on that template using London Gatwick as the location.
2. At building level on the HVAC tab load the **VAV with terminal reheat** HVAC template and take a look at the data. Much of the data you will recognise from the previous exercises but there is also some new data for VAV systems.
3. Set the Min AHU Outside air requirement schedule as **On**. This models a system providing constant minimum fresh air, day and night.
4. Set the Outside air definition method to **1-By zone** and set the mechanical ventilation outside rate to **2 ac/h**. These last 2 settings will help us understand HVAC fresh air delivery in the results.
5. Building heating setpoint is 22°C and Cooling setpoint is 24°C.

Now run a simulation using the default **VAV with terminal reheat** options. On the Simulation options dialog select **sub-hourly results** for the **Summer design week**, select **10 timesteps per hour** and start the simulation.

Select **Sub-hourly** results as the Interval in the Display options panel. Viewing Sub-hourly results over short simulation periods can provide get a clearer picture of the processes taking place in the model.



The graph shows the operation of the VAV system under various operation regimes in the summer:

1. When the return air is cooler than outside air and cooling is required the economiser shuts the outside air damper to the minimum position to provide the minimum fresh air (2 ac/h).
2. This also happens when the economiser limits are exceeded (outside air temperature < 11°C or > 25°C or when the outside air enthalpy > 50,000 J/kg).
3. When the return air is warmer than outside air the economiser opens the outside air damper to 2.0 ac/h.
4. When the return air is cooler than outside air and the cooling demand requires more airflow the flow rate is increased to provide beneficial free cooling.
5. There is a short heating operation time at the start of occupancy to bring the zone air temperature up to the heating setpoint temperature of 22°C. This could be avoided by switching off summer reheat.
6. The AHU heating coil operates briefly during 21 August. Otherwise, the mixed air temperature (temperature after the mixed air damper) is above the

AHU heating coil setpoint temperature of 14°C. This could be avoided by scheduling the AHU heating coil to be off during the summer.

7. There is a cooling load when the mixed air temperature is greater than 14°C.
8. Check the effect of the **Economiser** by switching it off and re-running the simulation. You should see that the outside fresh air is now a constant 2 ac/h during the occupied period.

Check system operation for the winter design week.

Save the example file as a template (File > Save > Save as template) calling it something like **VAV.dsb**.

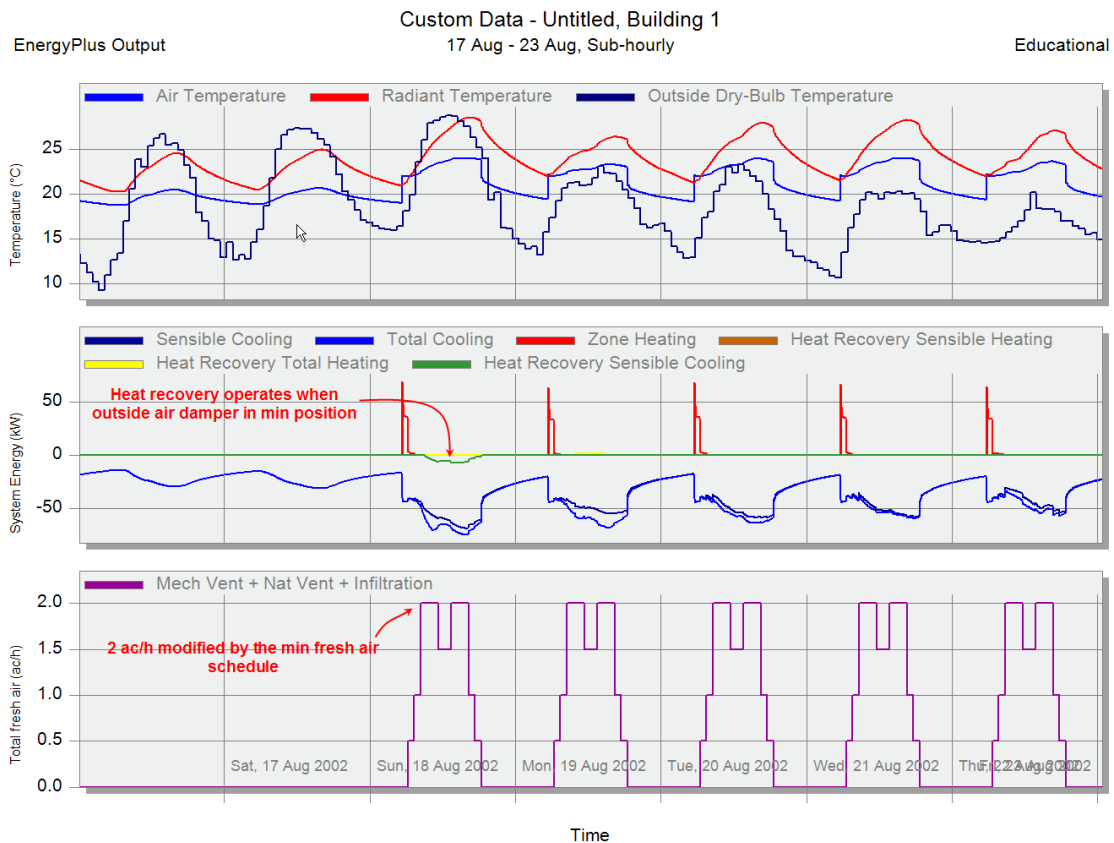
CAV Systems

Constant air volume systems provide a constant volume flow rate through the AHU. The air may be heated and/or cooled at the AHU. DesignBuilder Constant Air Volume (CAV) systems are **identical to VAV** systems with the exception that the **turndown** ratio is set to **1** preventing the zone terminal unit damper from closing.

Exercise

1. Starting from the VAV example, at the building level on the HVAC tab, load the **Constant volume** HVAC template.
2. Set the Mechanical ventilation outside air rate to **2 ac/h**.
3. Switch on the **Heat recovery** and use the default sensible only option at efficiency 0.7.
4. Run a simulation requesting **sub-hourly** results for the **summer design week**.

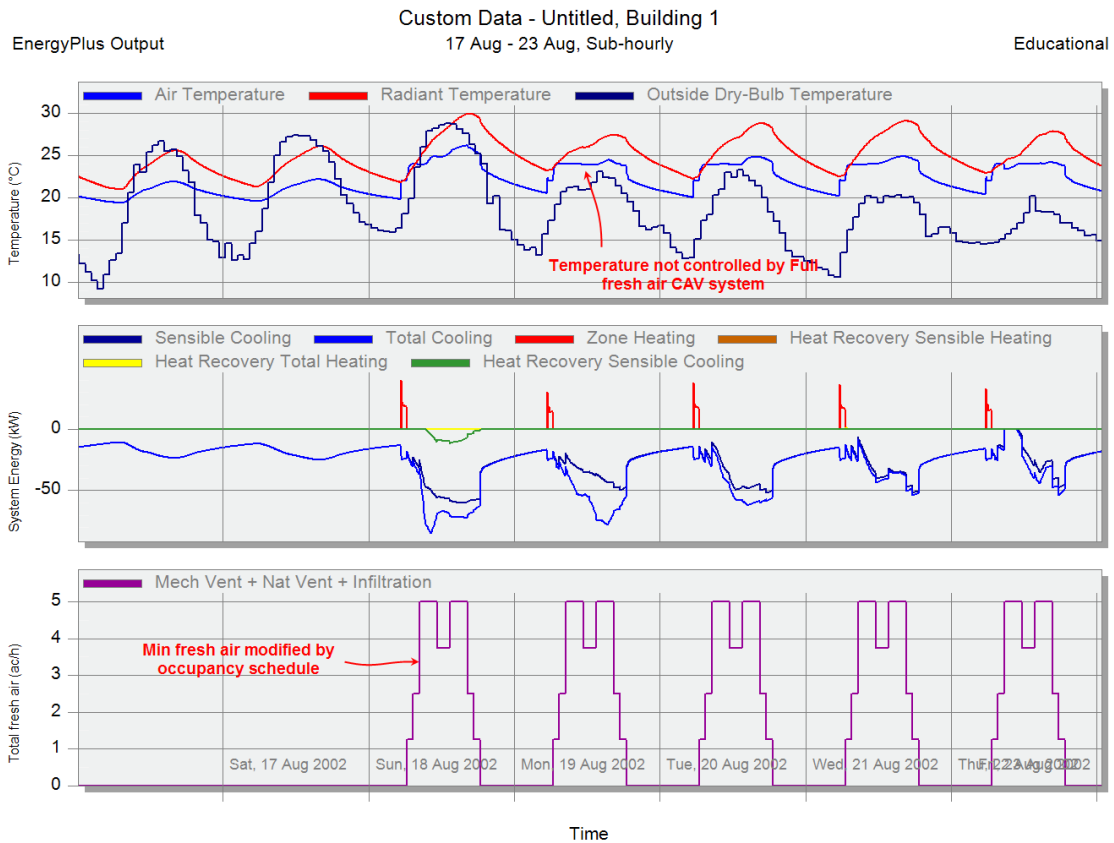
The results should look something like the screenshot below.



Exercise

Now modify the previous example as follows:

1. Select Outside air mixing as **2-Full fresh air**. This means that there is no recirculation.
2. The Min outside air schedule is Office_OpenOcc and the Outside air mechanical ventilation rate is 5 ac/h.
3. To be sure of full fresh air (no recirculation) you must also set the **Compact HVAC Autosizing factor** on the Simulation tab of the Model options dialog to have a value of **1**.
4. Run the simulation again and check results.



1. You should find that the outside fresh air provision is 5 ac/h modified by the schedule without any economiser action or extra flow to boost system capacity.
2. Also notice how the zone air temperature is not controlled – the airflow is not modulated to control the zone temperature in this variant of the CAV system.

Change-Over Mixed Mode

Introduction

This Example introduces *Mixed mode* control, which allows you provide a hybrid natural ventilation and HVAC cooling system in your models. These control systems aim to make best use of natural ventilation availability while maintaining optimal comfort for occupants. DesignBuilder can model all 3 types of mixed mode system defined by CBE at LBL.

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. **Change-over** (*Same space, different times*) - Change-over designs are becoming increasingly common, where the building “changes-over” between natural ventilation and air-conditioning 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 anytime 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).

Concurrent and **Zoned** systems can be modelled without using DesignBuilder mixed mode control. In DesignBuilder 'mixed mode' specifically refers to **Change-over** systems where the HVAC and the natural ventilation do not take place at the same time. The rest of this example focuses on this type of system.

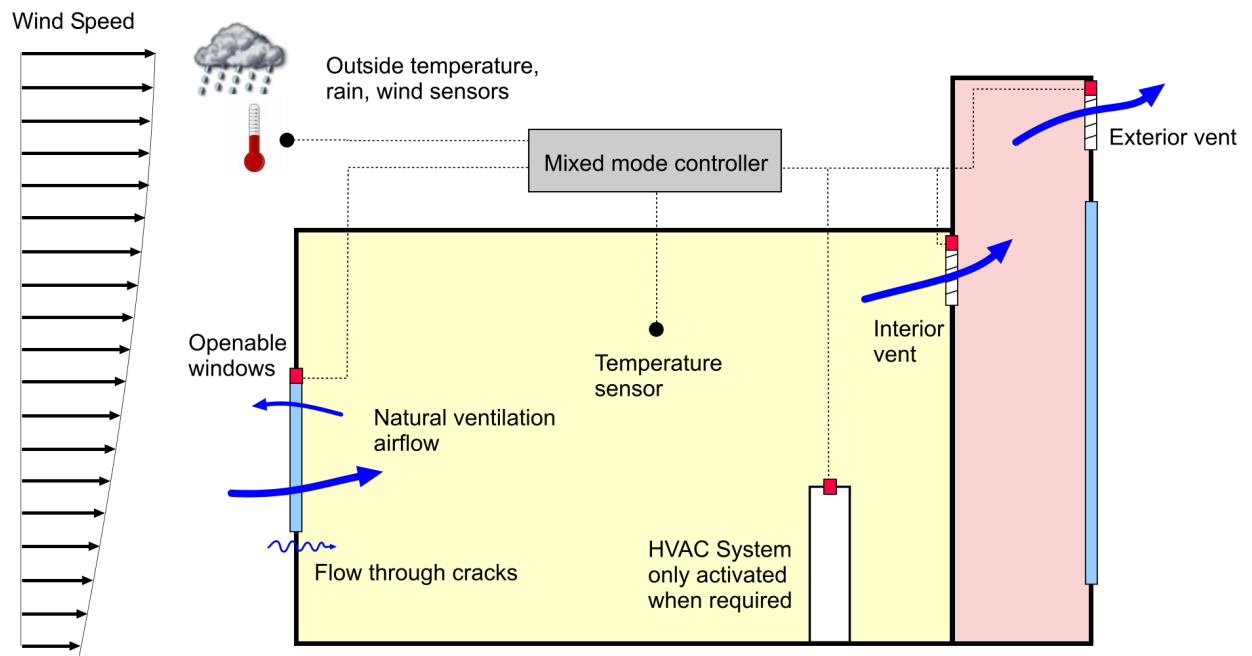
In change-over mixed mode buildings, natural ventilation has priority, but the cooling system is switched on in cases where natural ventilation is unable to cool the space sufficiently to meet comfort requirements. During periods where the mixed mode controller switches off the natural ventilation system, zones served by the HVAC

system have their windows and vents closed automatically to avoid the mechanical HVAC system running while windows and vents are open.

Natural ventilation can also be shut down due to extreme weather conditions such as strong wind (to avoid papers blowing around), rain (so the building does not get wet inside) and temperature (so uncomfortably cold outside air is not introduced). The diagram below illustrates a typical change-over mixed mode system.

Mixed Mode 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 $T_{air} > T_{setpoint}$, closed when HVAC operates
- * Priority given to natural ventilation - HVAC operates only when required
- * Windows and vents can be shut off due to extreme external conditions (wind, rain, temperature etc).



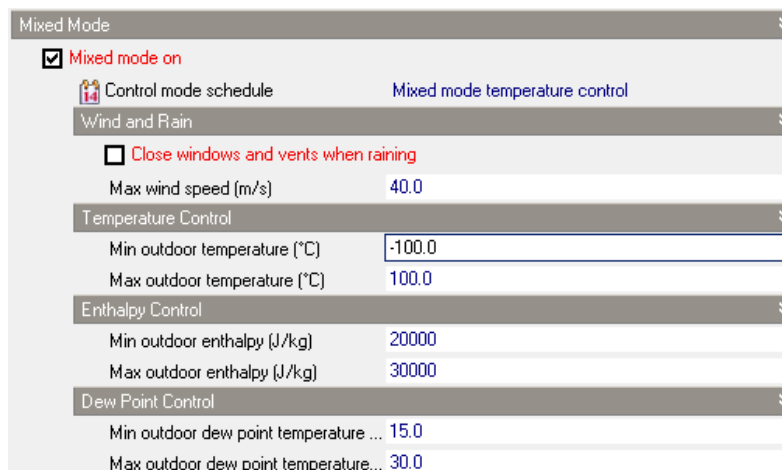
Data Entry

Continue from where you left off in the previous exercise or Start by loading the **Nat Vent Example 3** template file you created earlier.

To start with you will make settings for a simple temperature controlled mixed mode building without protection from wind, rain, low and high temperatures. The associated HVAC system will be VAV.

1. First select **Compact HVAC** and **Calculated Nat Vent** on the model options dialog.
2. Then go to the building level and load the **VAV with HR + outside air reset + mixed mode HVAC** template.
3. Set the Mechanical ventilation flow definition method to be '**By zone**' and set the Mechanical ventilation 'Outside air' rate to be **3 ac/h** (it will probably already have this value).

4. Set the **Min AHU Outside Air Requirement** to use the **On** schedule. For the purposes of this exercise, steps 3 and 4 give a fixed 3 ac/h fresh air demand making it easier to see the operation of the VAV system in the results.
5. Still on the HVAC tab at building level, open the Natural Ventilation and mixed mode headers and check that **Mixed mode on** is selected.
6. Leave the control mode schedule as the default **Mixed mode temperature control**.
7. Switch off the **Close windows and vents when raining** option – we want the natural ventilation to be controlled only by temperature.
8. Leave **Max wind speed** as **40 m/s**.
9. Leave default **Min and Max outdoor temperatures** as **-100°C** and **100°C** respectively. Also leave enthalpy and dew point temperature upper and lower limits - these won't be used because we are sticking with temperature control as per the **Mixed mode temperature control** schedule set in the control mode schedule data above.
10. Go to **Ground floor > Zone 1** and on the HVAC tab select this as the **mixed mode control zone**. This means that the conditions in this zone will be used to control the operation of the mixed mode system. For example, when the temperature in this zone is below the cooling setpoint but above the natural ventilation setpoint then the natural ventilation will be allowed to operate in all zones served by the VAV system. When natural ventilation is unable to satisfy cooling demand in this zone and temperature starts to rise above the cooling setpoint temperature, all the windows and vents in zones served by the VAV HVAC system are closed and the VAV system provides cooling.
11. Go back up to building level and check that the 'Natural Ventilation > Options > **Modulate** opening areas' option is not selected.
Data on the HVAC tab under Mixed Mode should look like the screenshot below.



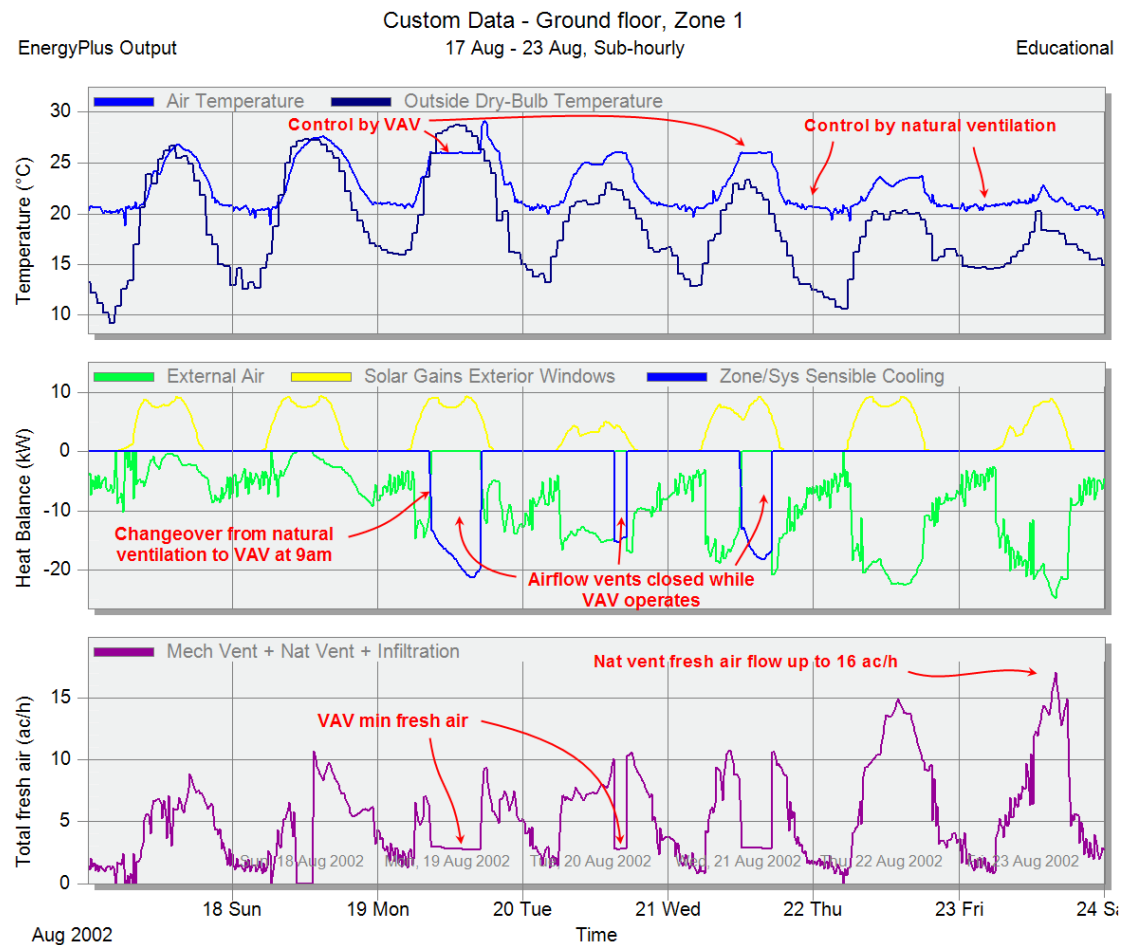
Mixed Mode	
<input checked="" type="checkbox"/> Mixed mode on	
Control mode schedule	Mixed mode temperature control
Wind and Rain	
<input type="checkbox"/> Close windows and vents when raining	
Max wind speed (m/s)	40.0
Temperature Control	
Min outdoor temperature (°C)	-100.0
Max outdoor temperature (°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 ...	15.0
Max outdoor dew point temperature...	30.0

12. On the Activity tab at building level, enter the setpoint temperatures for Heating, Cooling and Natural ventilation. The **Heating setpoint is 20°C**, the **Cooling setpoint temperature is 26°C** and the **Natural ventilation is 21°C**.
13. On the Constructions tab **switch off** the **Model infiltration** for the building. This will make it easier to check fresh air delivery for windows, vents and mechanical ventilation only.

Now start a simulation and on the Simulation options dialog request only **sub-hourly** data. Use the same **Summer typical week** as set from the previous example. Also, on the Options tab set the **Number of timesteps per hour** to **4**.

Make sure to switch off surface and opening results (unless you are happy to wait a minutes or so longer for the results).

When you review **Sub-hourly results** at **building level** you should see results as below:

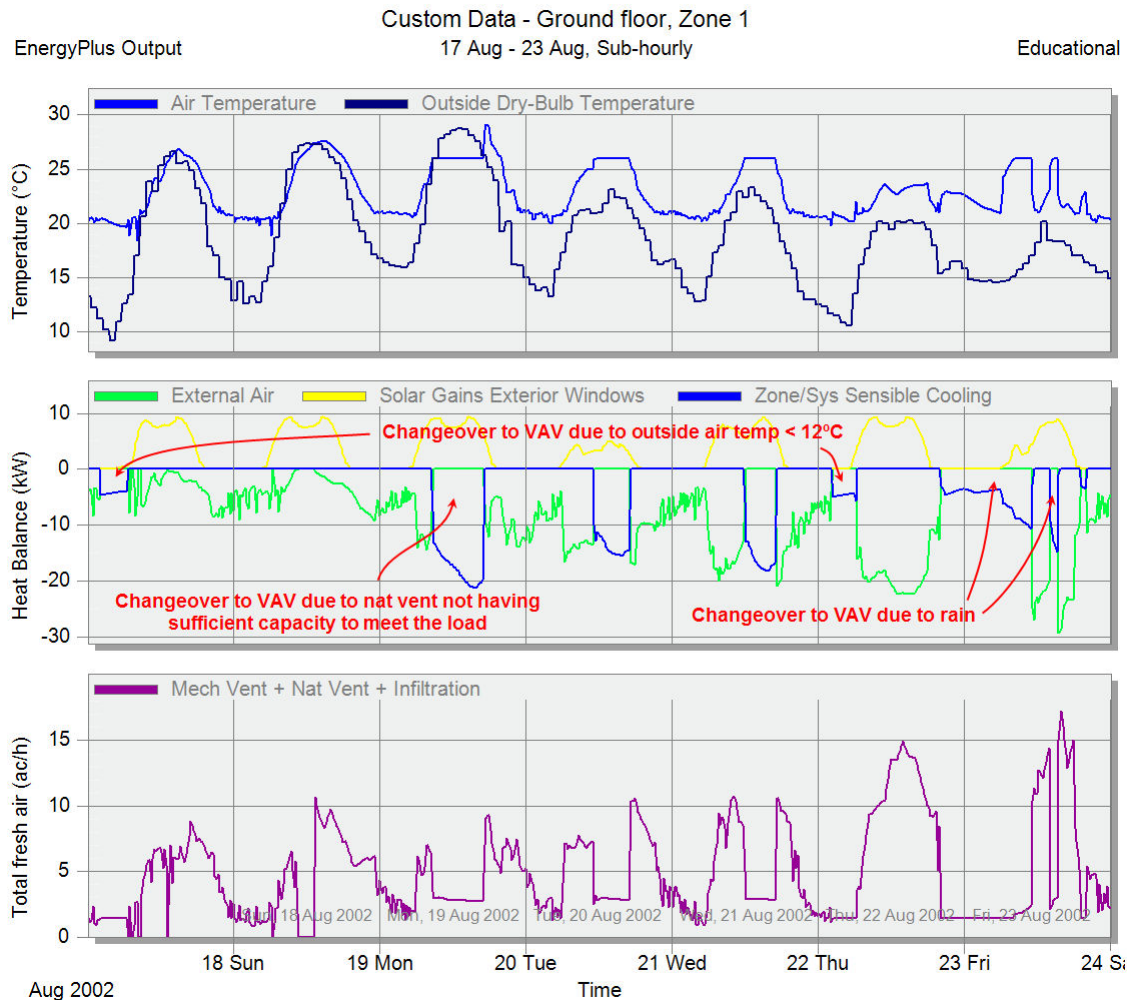


Extreme Weather Conditions

Now explore some options with a view to preventing natural ventilation operation when extreme weather occurs:

1. Go to the building level and switch on the natural ventilation wind protection by checking the **Close windows and vents when raining** checkbox under Natural Ventilation > Mixed Mode > Wind and Rain.
2. Also set the **Minimum outside temperature** to 12°C.

Re-run the simulation with these options to see the effect.



You should see that the windows close when it rains or when the outside air is cooler than 12°C and this requires the cooling to operate more frequently. This problem could be eliminated by either using windows that can open in the rain or by using automatically controlled **vents** that can remain open when it rains.

In many climates a mixed mode HVAC system combined with good solar shading and insulation standards and energy efficient services would be a good starting point when investigating design options.

19 Computational Fluid Dynamics (CFD)

This section describes the various processes involved in conducting calculations using the DesignBuilder CFD module.

Note that this CFD section is different from the rest of the Guide is that it is quite technical and detailed. To continue in the same flavour as the rest of the guide, skip forward to the [CFD Examples](#).

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 throughout a predefined air volume in and around building spaces with specified boundary conditions 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 analyses provide the distribution of air velocity and pressure around building structures due to wind effect and this information can be used to assess pedestrian comfort, determine local pressures for positioning HVAC intakes/exhausts and to calculate more accurate pressure coefficients for EnergyPlus calculated natural ventilation simulations. Internal analyses provide the distribution of air velocity, pressure and temperature throughout the inside of building spaces and this information can be used to assess the effectiveness of various HVAC system designs and to evaluate interior comfort conditions.

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 ϕ represents the dependent variables:

$$\frac{\partial}{\partial t}(\rho\phi) + \text{div}(\rho u\phi) = \text{div}(\Gamma \text{ grad } \phi) + S$$

The $\frac{\partial}{\partial t}(\rho\phi)$ term represents the rate of change, the $\text{div}(\rho u\phi)$ term represents convection, the $\text{div}(\Gamma \text{ grad } \phi)$ term 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.

Creating Model Geometry for CFD

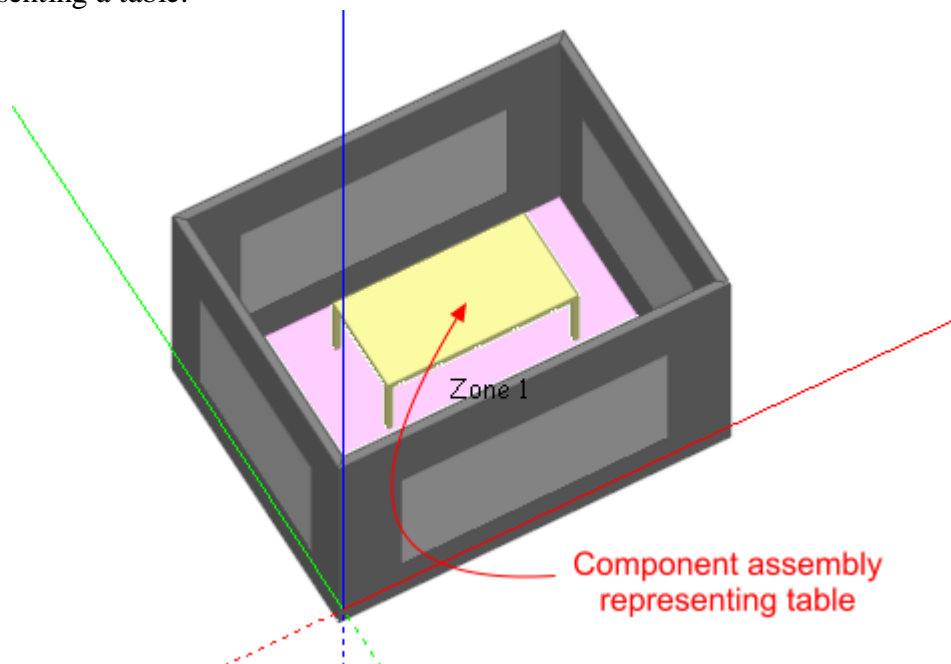
If you have already created a model for the purpose of an additional analysis (e.g. thermal simulation or SBEM calculation), you can use exactly the same model for CFD, although you should read the next section on the finite volume grid and geometric considerations. On the other hand, if you intend to conduct a CFD analysis from scratch, you should refer to the 'Building Models' section of the DesignBuilder Help for information on creating models.

Finite Volume Grid and Geometric Modelling Considerations

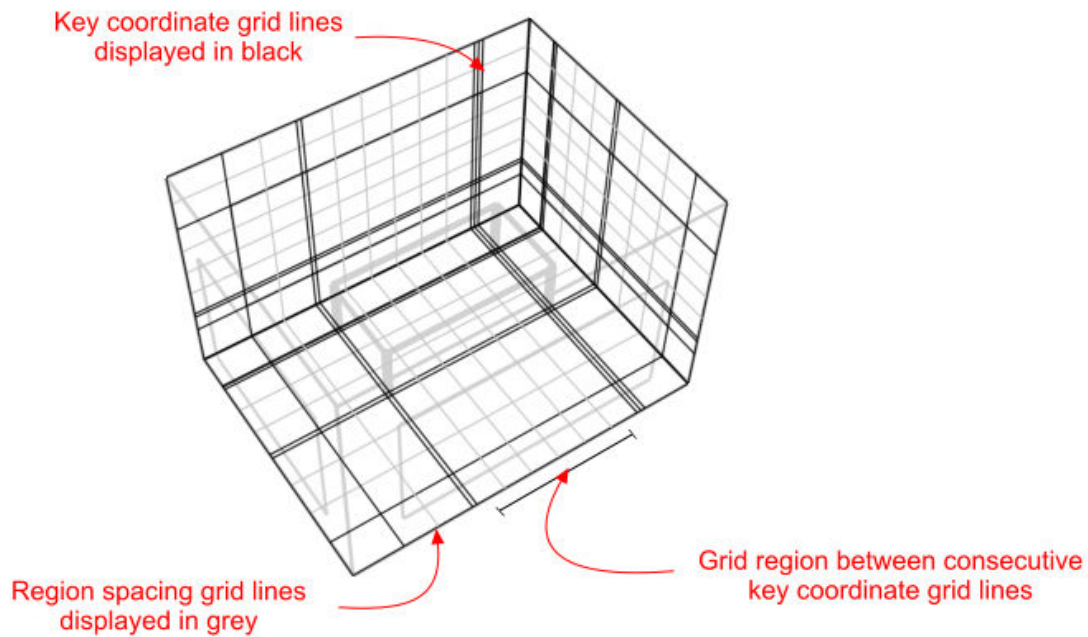
The calculation method requires that the geometric space across which the calculations are to be conducted is first divided into a number of non-overlapping adjoining cells which are collectively known as the finite volume grid.

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.

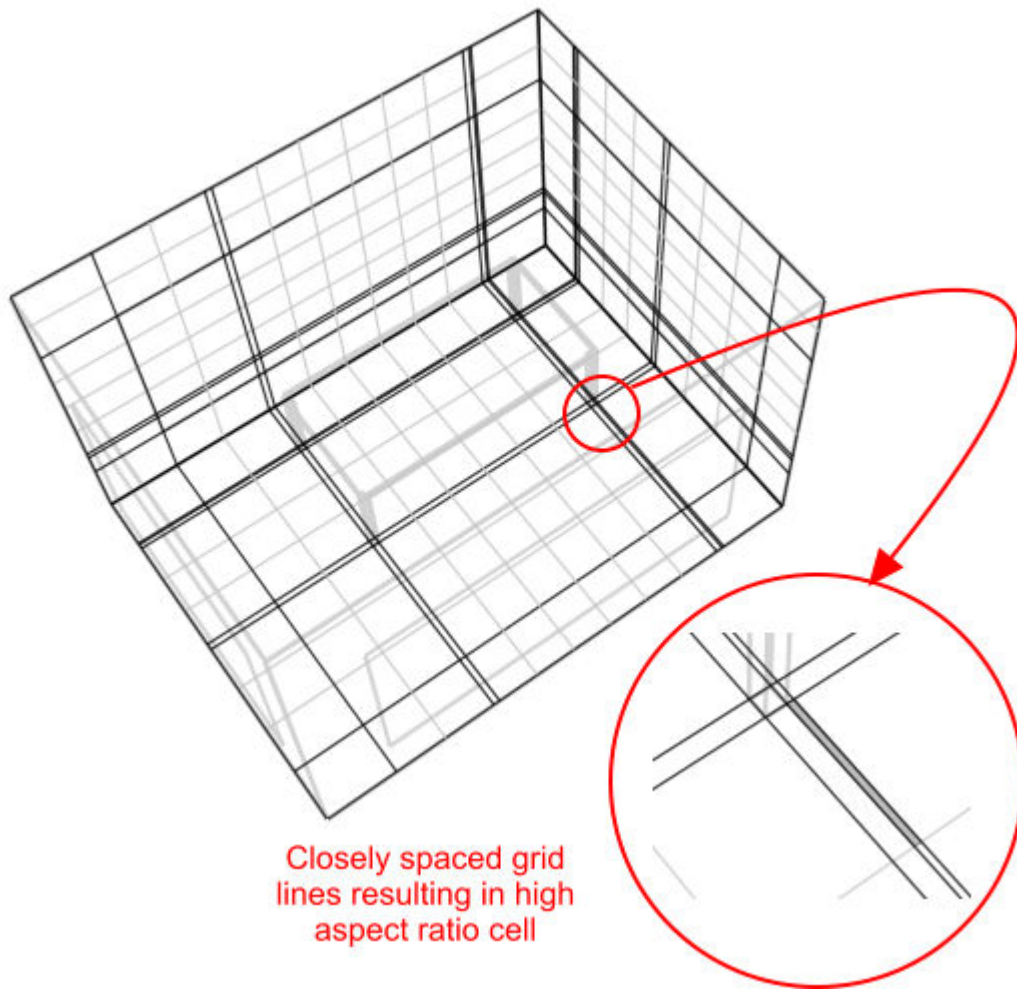
For example, looking at a simple building block with a single component assembly representing a table:



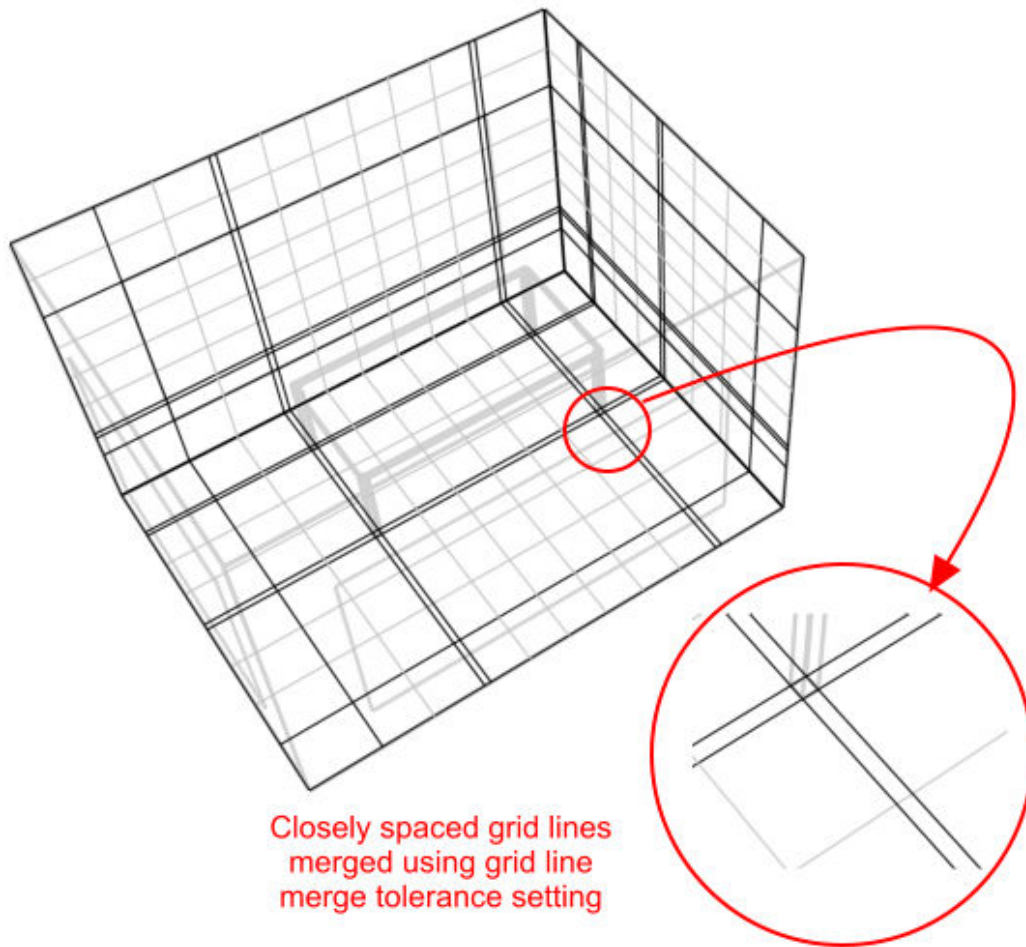
The resulting grid, generated with 0.3m default grid spacing would be as follows:



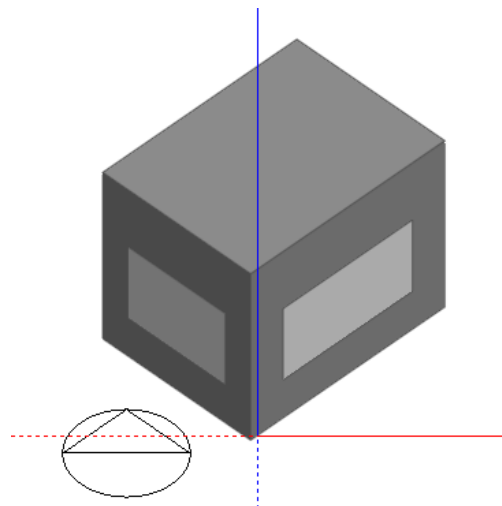
By default, grid regions are spaced uniformly using a spacing that is calculated to be as close to the user-defined default grid spacing as possible. Notice the narrow regions created between key coordinates associated with the tabletop and table legs. In this case, the distance between these key coordinates 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:



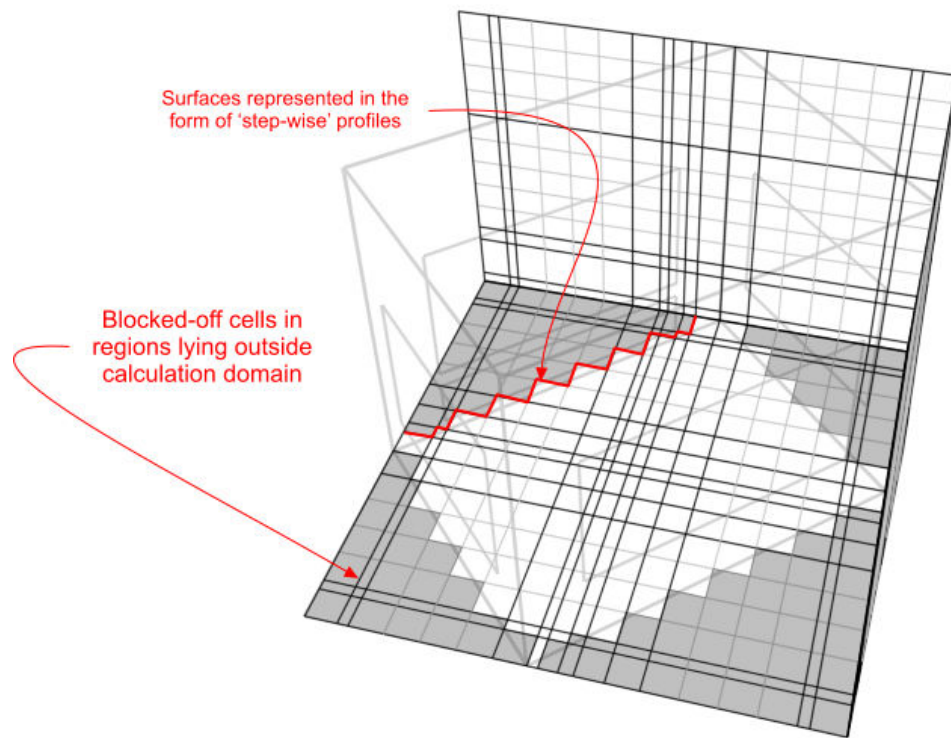
These closely spaced grid lines can be merged by creating a new analysis and increasing the grid line merge tolerance setting to say 0.025m:



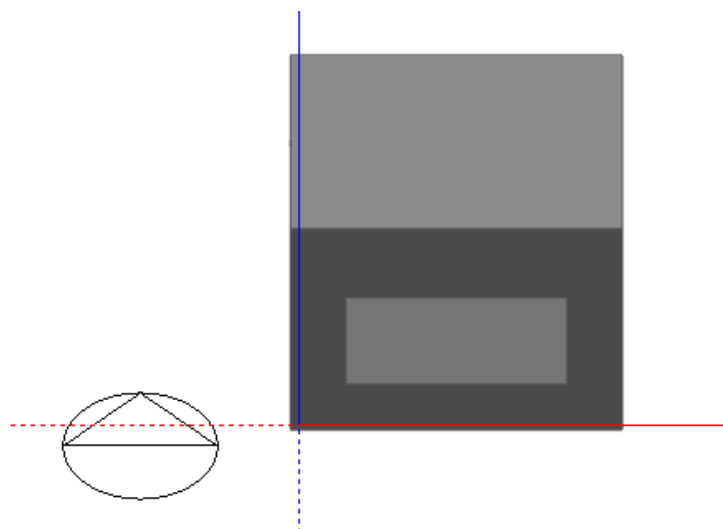
Due to the strict rectilinear nature of the grid, grid cells that lie in regions outside of the domain required for calculation are ‘blocked-off’ in order to cater for irregular geometries. It is important to take this into account when creating a model in order to maximise the efficiency of grid generation 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:



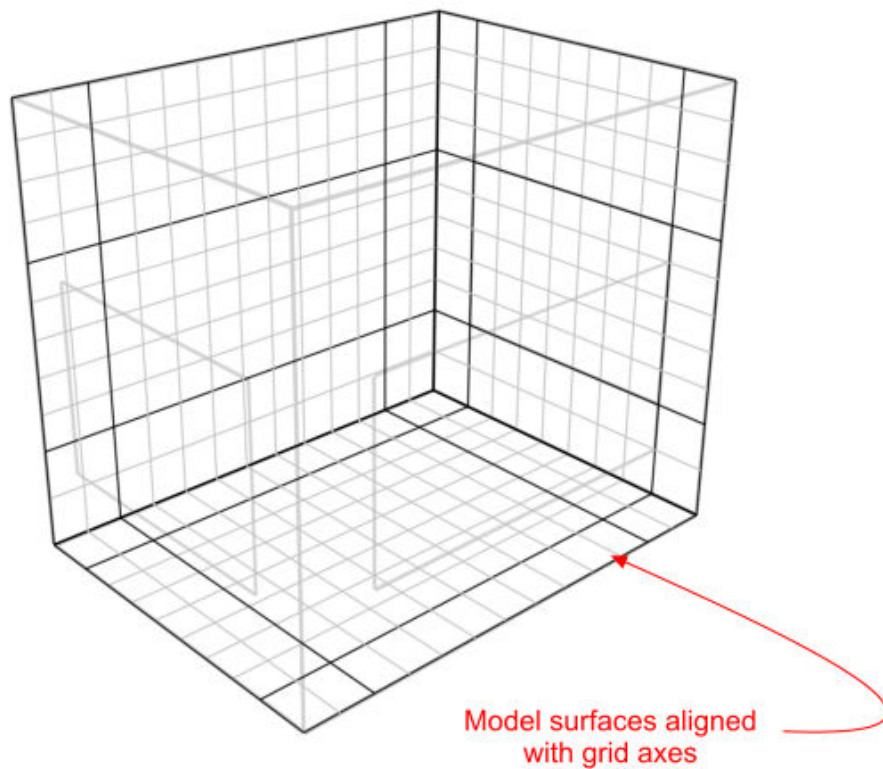
The resulting grid, generated using 0.3m default grid spacing, contains a great deal of redundancy in the form of blocked-off regions and inaccuracy of surface representation:



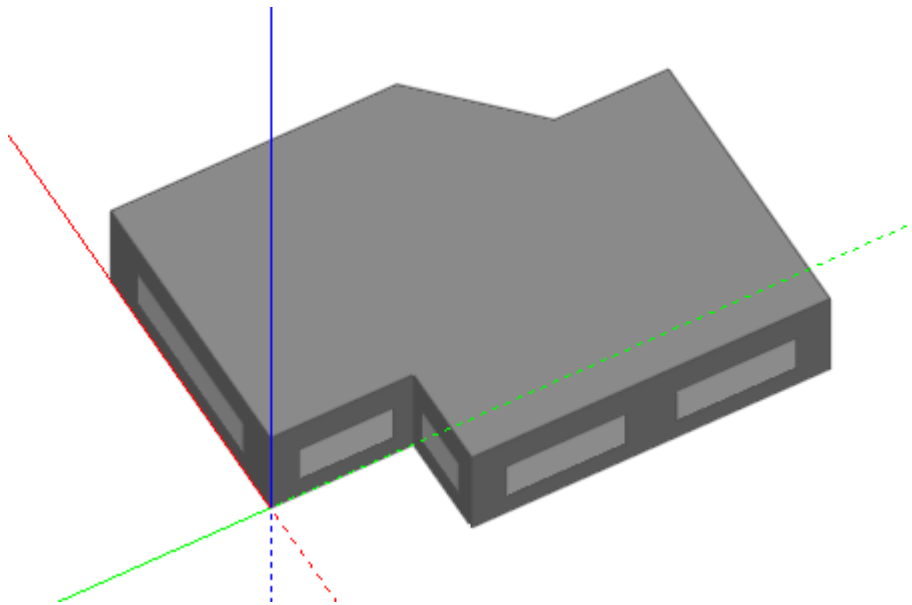
If the block is first rotated, to be aligned with the X-axis:



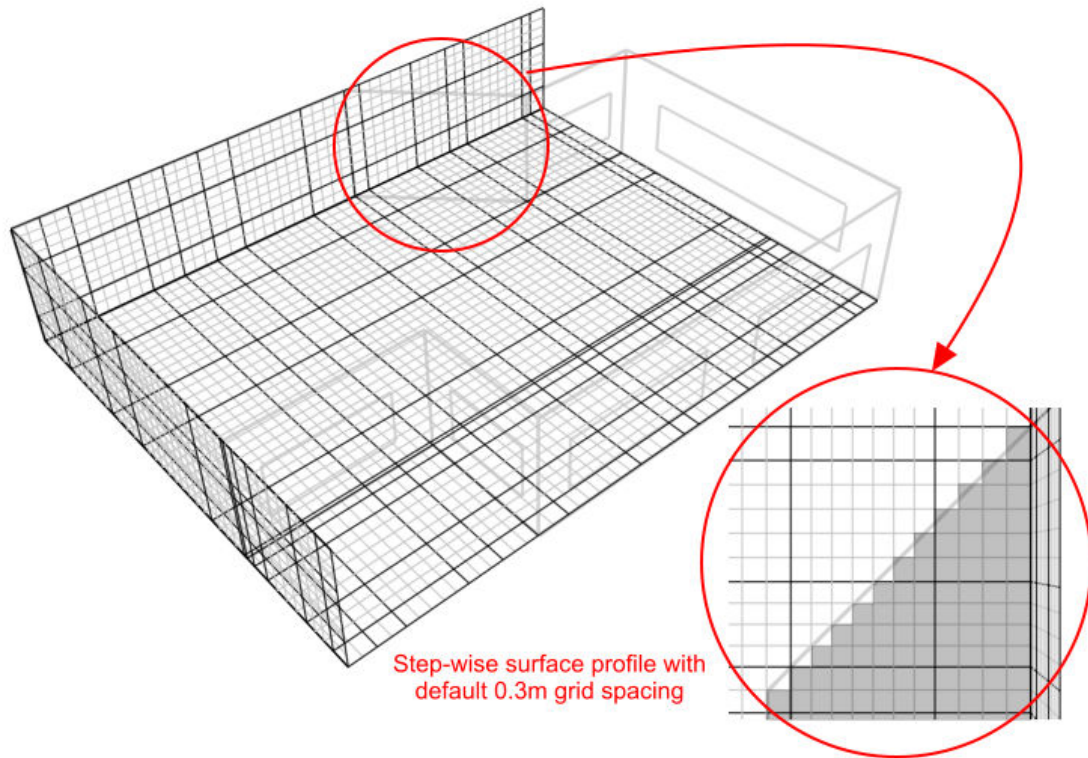
The resulting grid exhibits no redundancy and there is no inaccuracy in surface representation:



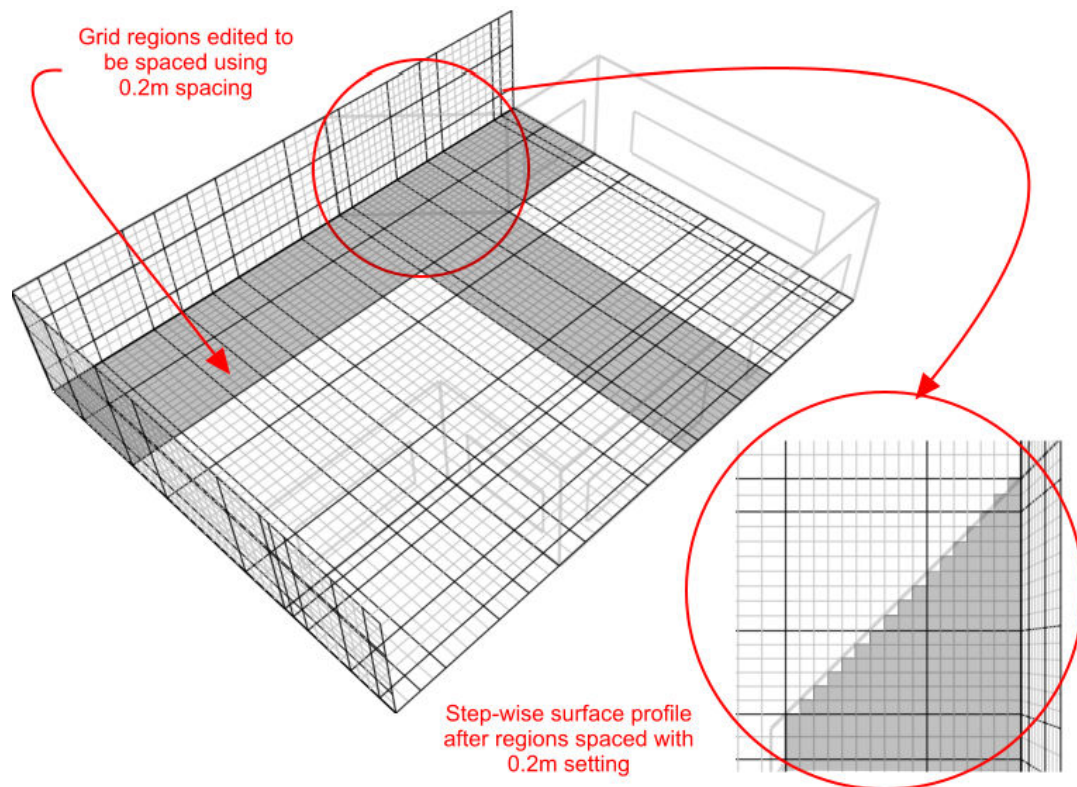
The accuracy of the representation of non-orthogonal surfaces can be improved by using smaller default grid spacing and in some cases specific grid regions can be modified to increase accuracy in a more localised fashion as illustrated by the following example:



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.

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 and just requires setting the building exposure, wind velocity and wind direction and this is done using a single dialog box. However, internal analysis boundary conditions tend to be a bit 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 ones and that you correctly specify any additional boundary conditions required for your project.

Boundary Conditions for Internal CFD Analyses

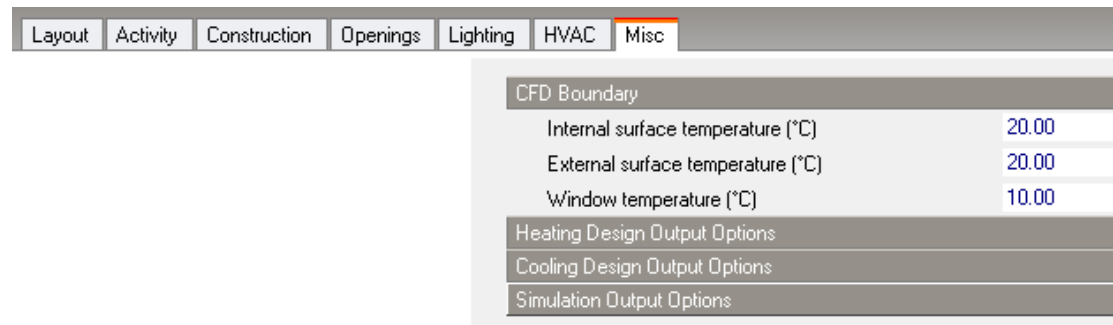
Internal boundary conditions can be defined in three ways:

1. Default wall and window temperatures can be defined for all zone surfaces.
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.

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 (see 'Importing EnergyPlus Output Data').

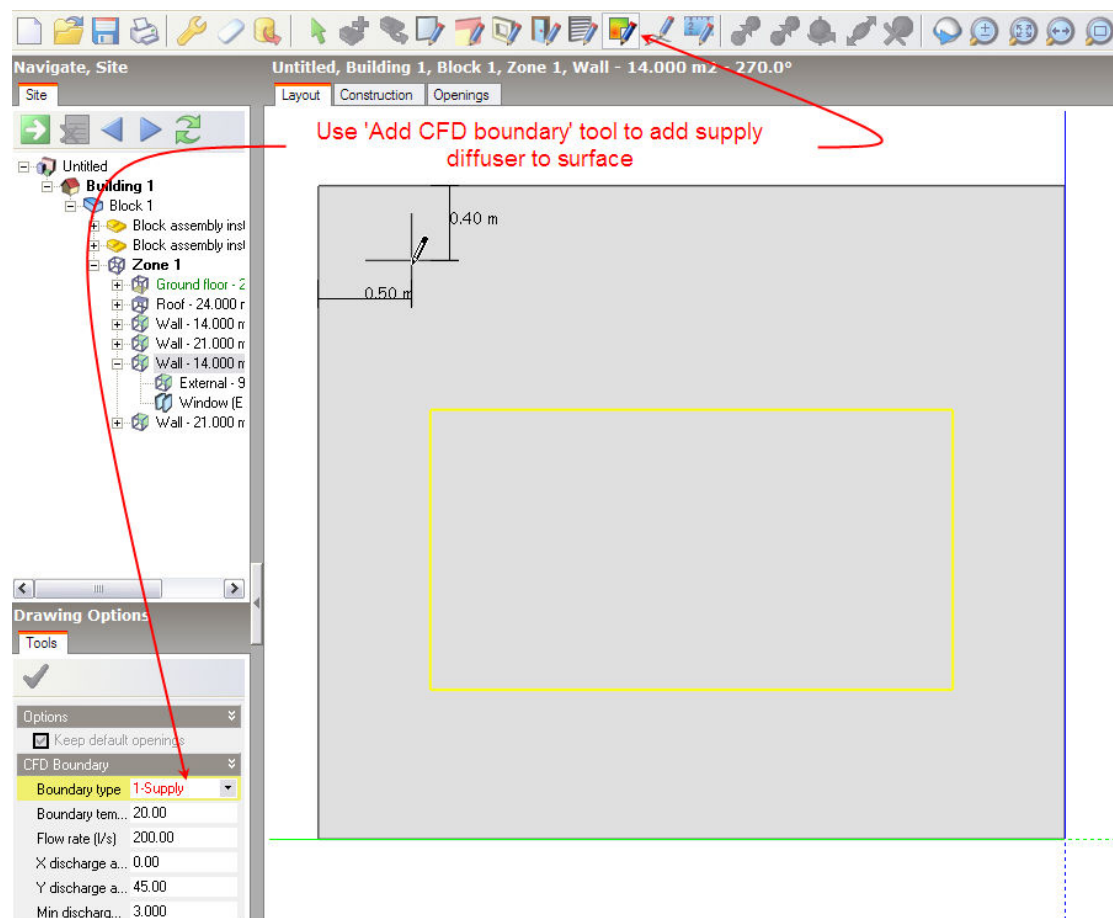
Default inside and outside wall and window boundary temperatures can be accessed under the 'CFD Boundary' header on the 'Misc' tab:



Zone Surface Boundaries

Supply diffusers, extract grilles, temperature and heat flux patches can be added to zone 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:



You can draw a CFD boundary on to the surface in exactly the same way that you would draw a window. Please refer to the 'Add window' Help section for details of drawing windows.

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)
- 6 – Flux (surface heat flux patch)

The following additional boundary types for multi-directional diffusers will appear for ceiling surfaces only:

- 2 – Four-way (four-way supply diffuser)
- 3 – Two-way (two-way supply diffuser)

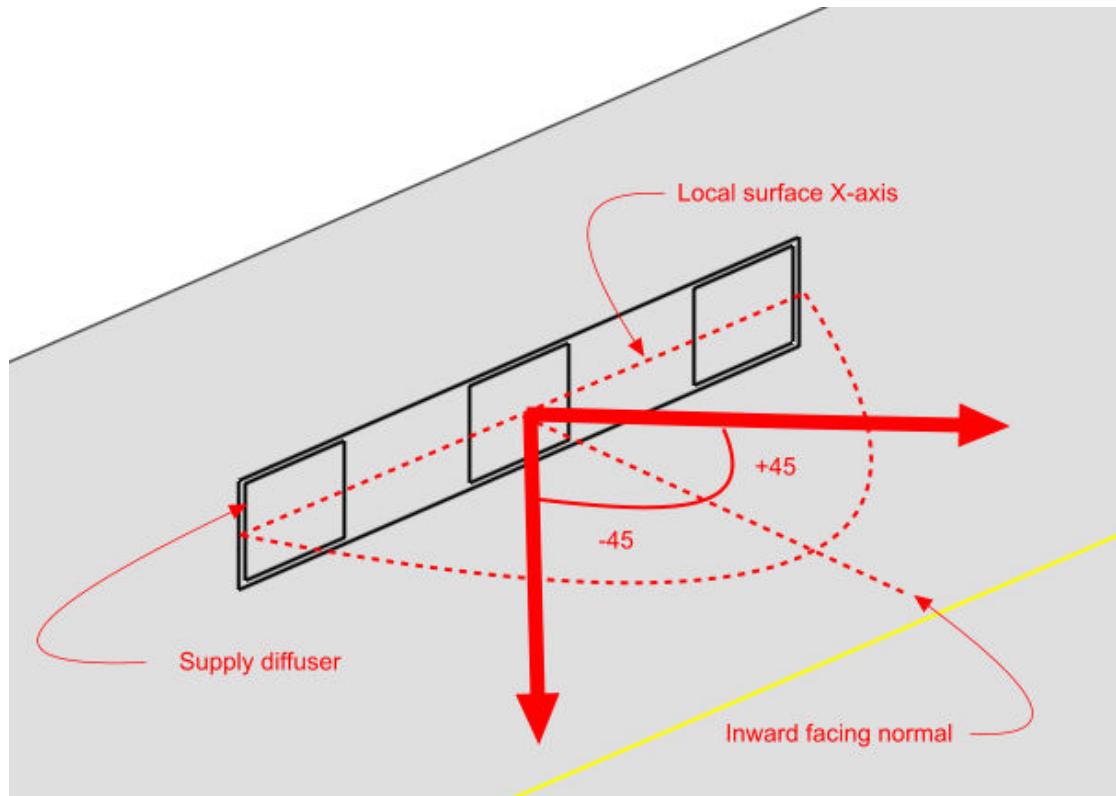
Various settings are available depending on the boundary type:

1-Supply:

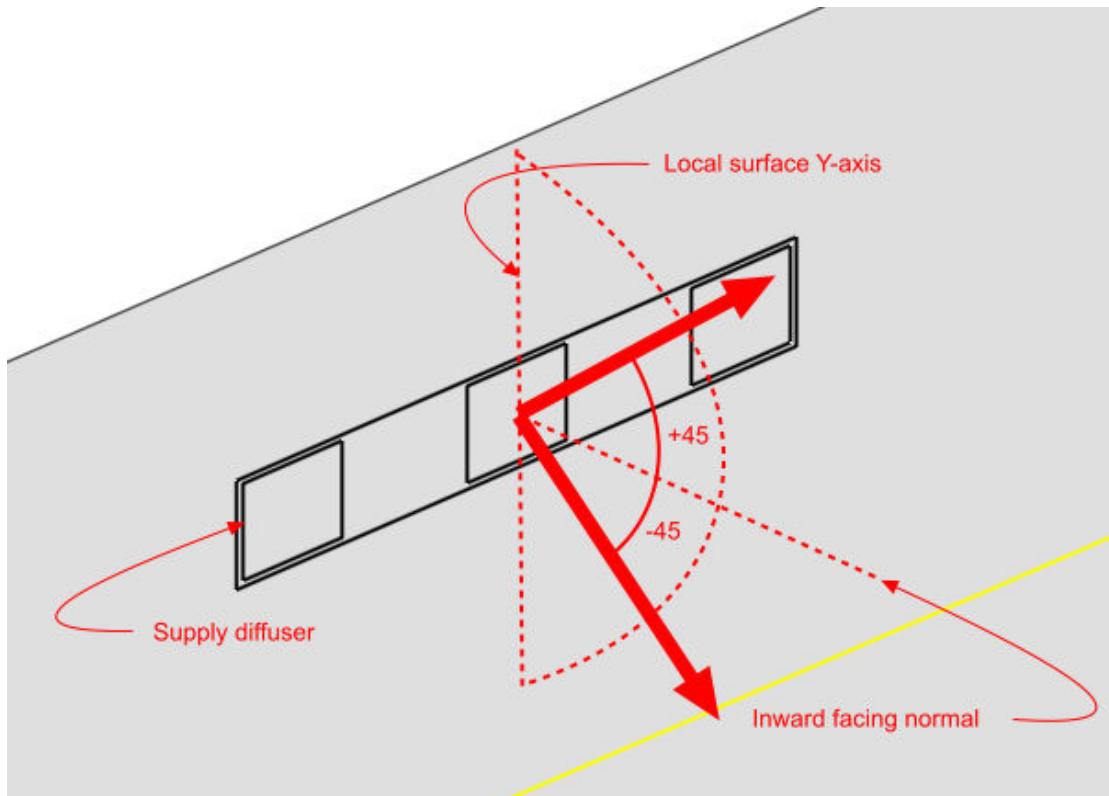
Boundary temperature (°C): Enter the required temperature of the air entering the space.

Flow rate (l/s): Enter the supply volume flow rate.

X-discharge angle (°): This setting allows you to define 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 (°): This setting allows you to define 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 :



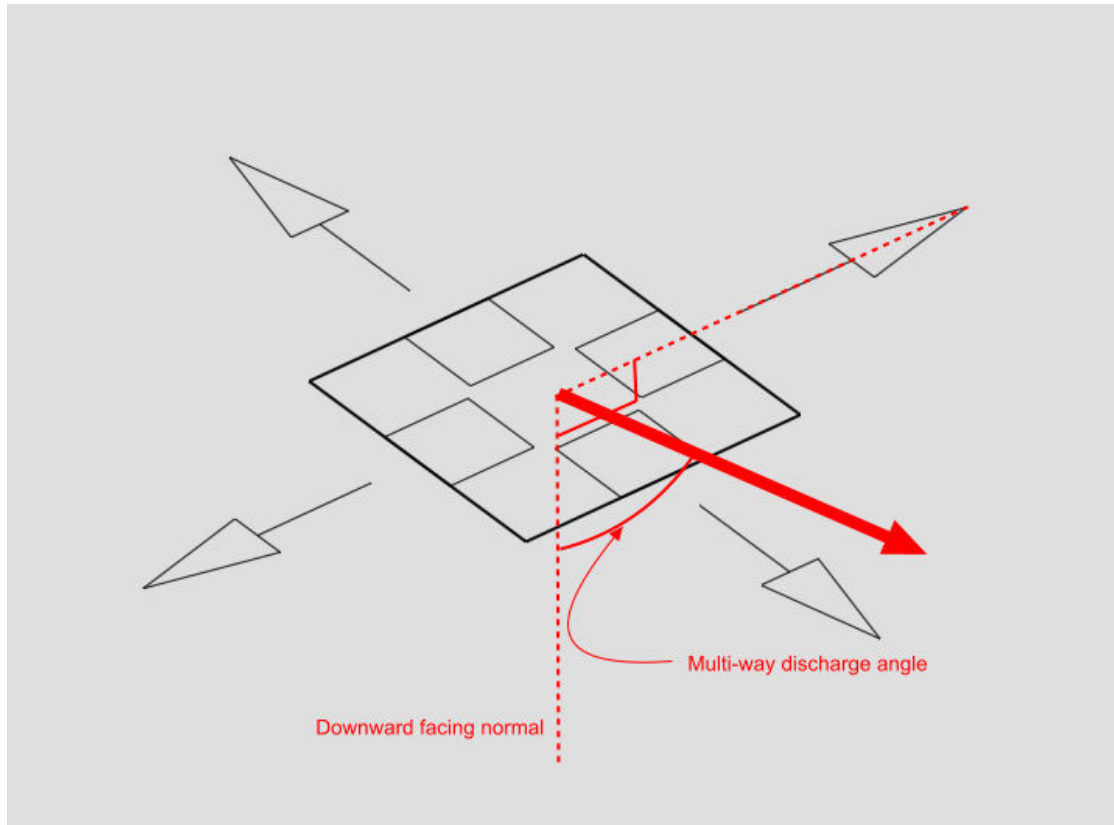
Minimum discharge velocity (m/s): The minimum discharge velocity is the minimum required velocity at the face of the diffuser. Supply diffusers are created using a number of elements, the areas of which are determined by combining the specified volume flow rate together with the minimum discharge velocity. The maximum linear dimension of each component element is currently 'hard-set' at 0.2m.

2-Four-way

Boundary temperature (°C): Temperature of the air entering the space.

Flow rate (l/s): 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 (m/s): The minimum discharge velocity is 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 (°C): Enter the required temperature of the air entering the space.

Flow rate (l/s):

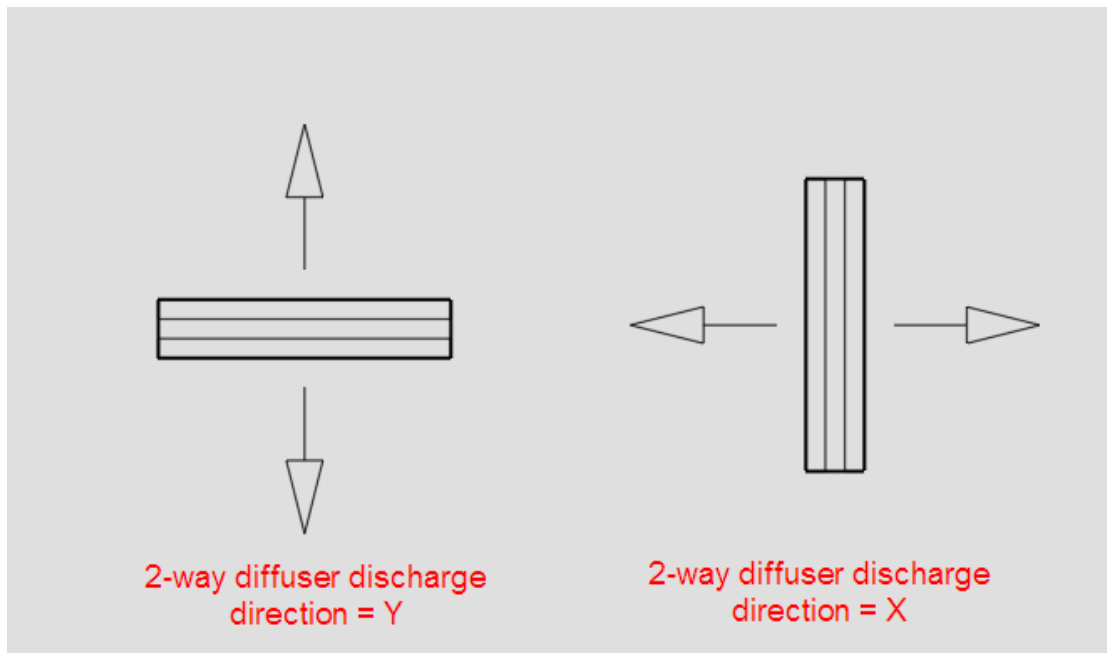
Enter the supply flow rate.

Multi-way diffuser discharge angle (°):

Enter the discharge angle, which is the angle between the downward pointing normal and the diffuser jet (see illustration in ‘Four-way’ section).

2-way diffuser discharge direction:

Enter the local surface axis along which the diffuser is to discharge:



Minimum discharge velocity (m/s): The minimum discharge velocity is 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.

2–Extract

Flow rate (l/s): The extract flow rate.

3–Temperature

Boundary temperature (°C): Temperature for the patch.

4 – Flux

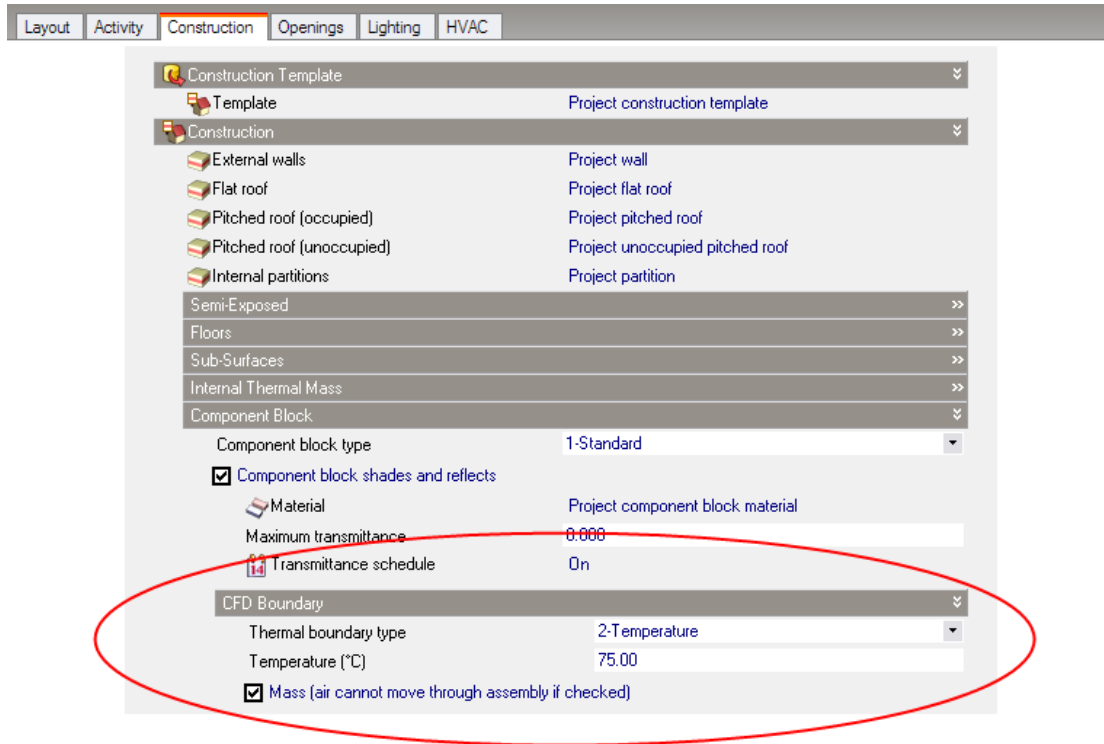
Heat flux (Wm^{-2}): Heat flux for the patch.

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.

Component Blocks

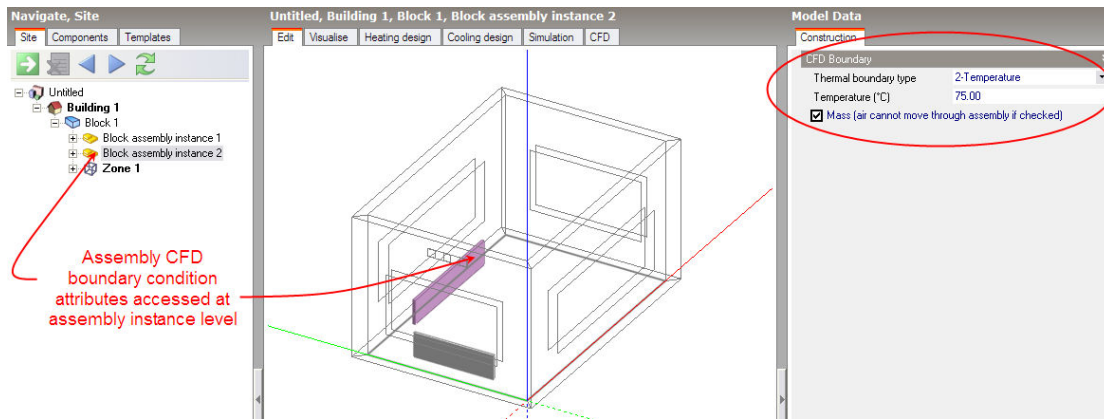
CFD attributes for component blocks are inherited down from the building level to component blocks located at both building and building block levels. CFD attributes for component blocks are accessed under the ‘Component block’ header on the ‘Construction’ tab:



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. occupant assemblies have a defined heat flux of 90W. You can also define your own assemblies for use in CFD analyses and you will find details of how to do this in the 'Assemblies' Help 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 'Construction' 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 are available 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.
- **3 – Flux:** Component block supplies a fixed heat flux to the surroundings.

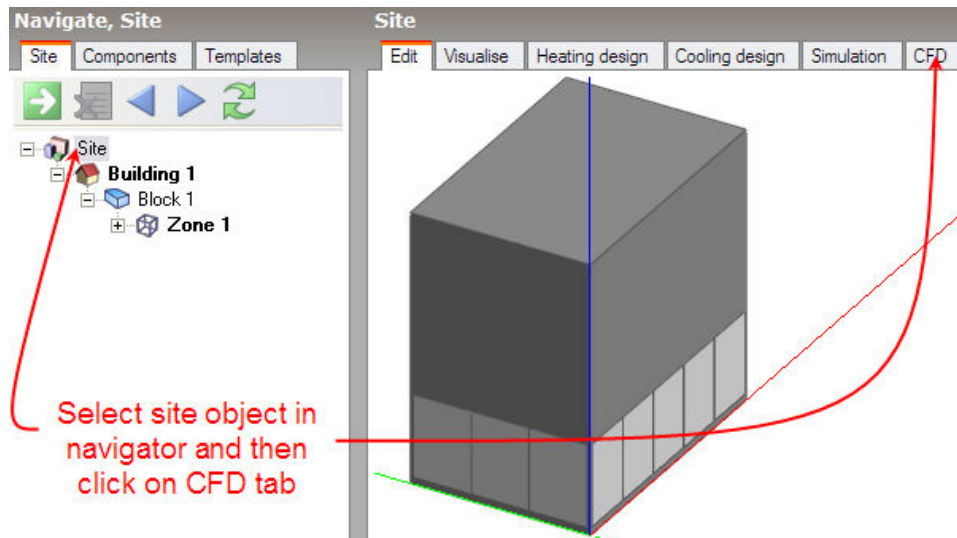
Temperature (°C): If the thermal boundary type has been set to 'Temperature', enter the temperature of the block.

Flux (W): If the thermal boundary type has been set to 'Flux', enter the heat emission of the block.

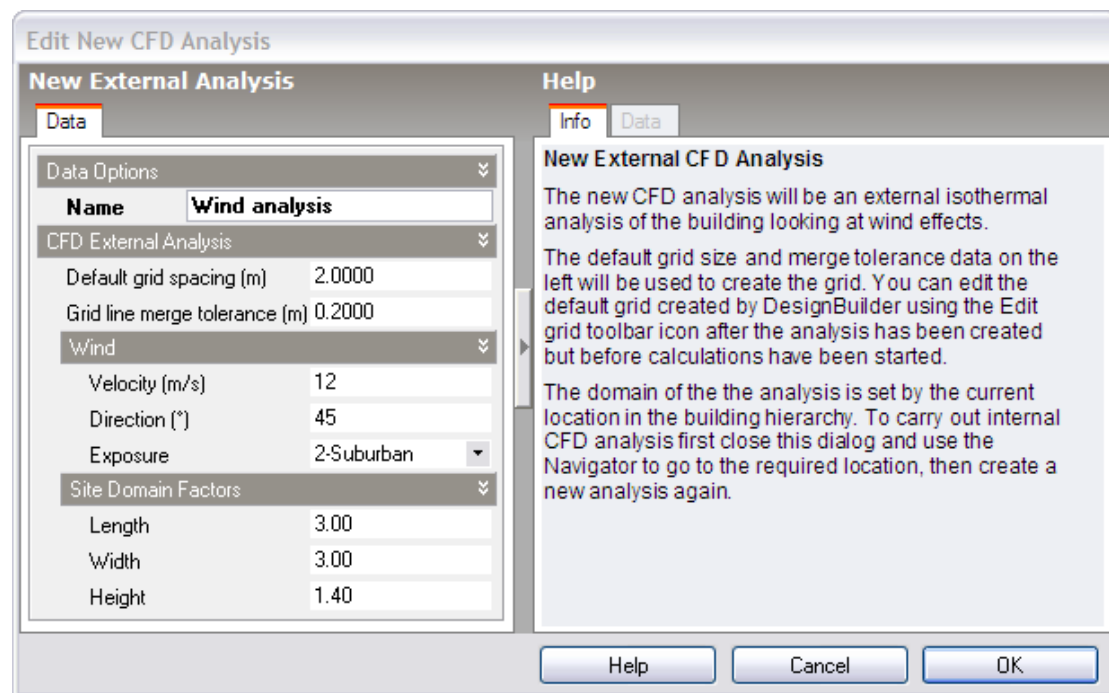
Mass: Determines whether or not the block will act as a physical obstruction to the surrounding flow. In some cases, it may be desirable to fix the temperature or flux of the component but allow air to pass through the block, e.g. to represent an occupancy flux throughout a large volume without needing to locate individual occupants.

Setting Up a 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 tab:



The 'New CFD Analysis Data' 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:



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 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.

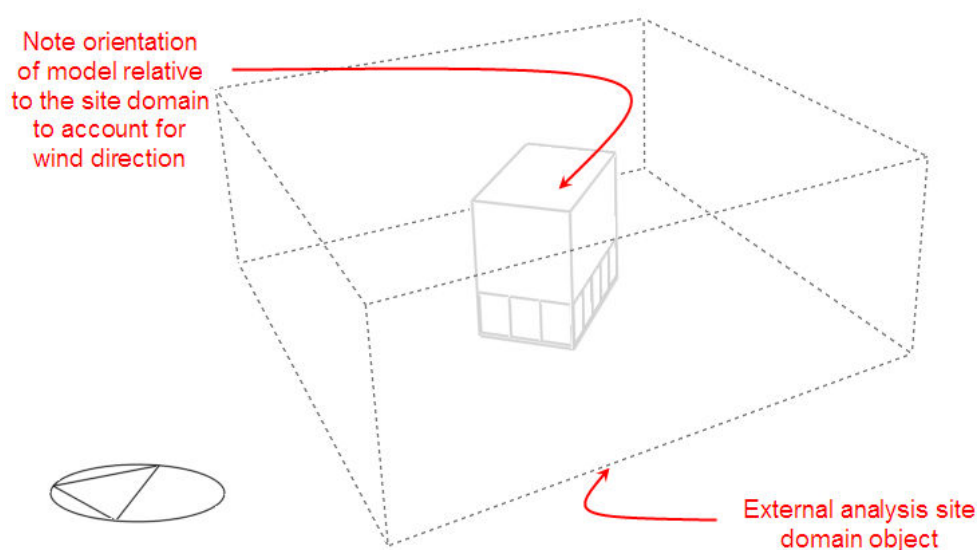
Wind Exposure

The free stream wind velocity is corrected for height above ground and surrounding terrain using an empirical relationship. Urban, suburban and open country exposures can be selected.

Site Domain Factors

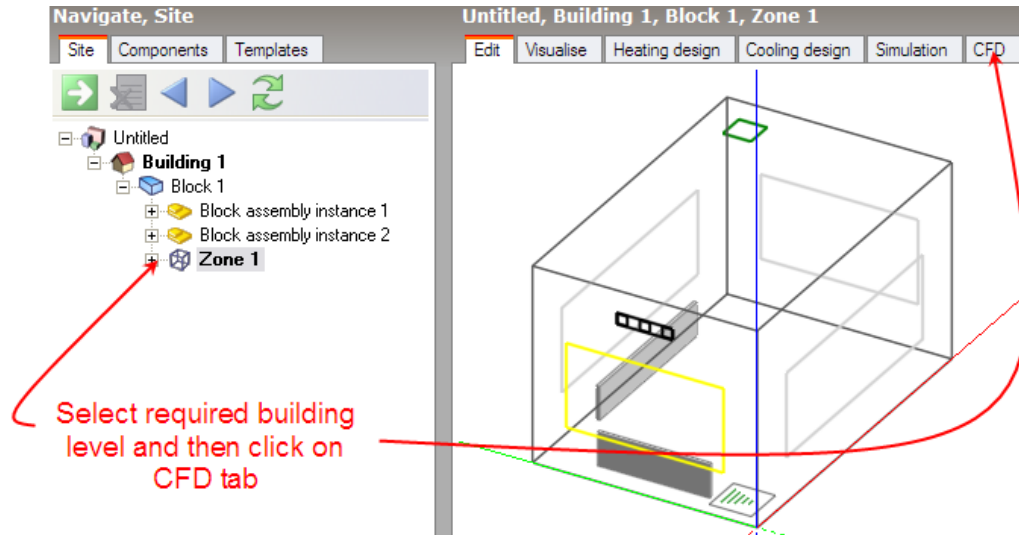
The length, width and height site domain factors are multipliers that are applied to the overall dimensions of the building model in order to arrive at an external volume or domain across which the analysis is carried out. The default factors are 3.0 applied to the length and width and a factor of 2.0 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:

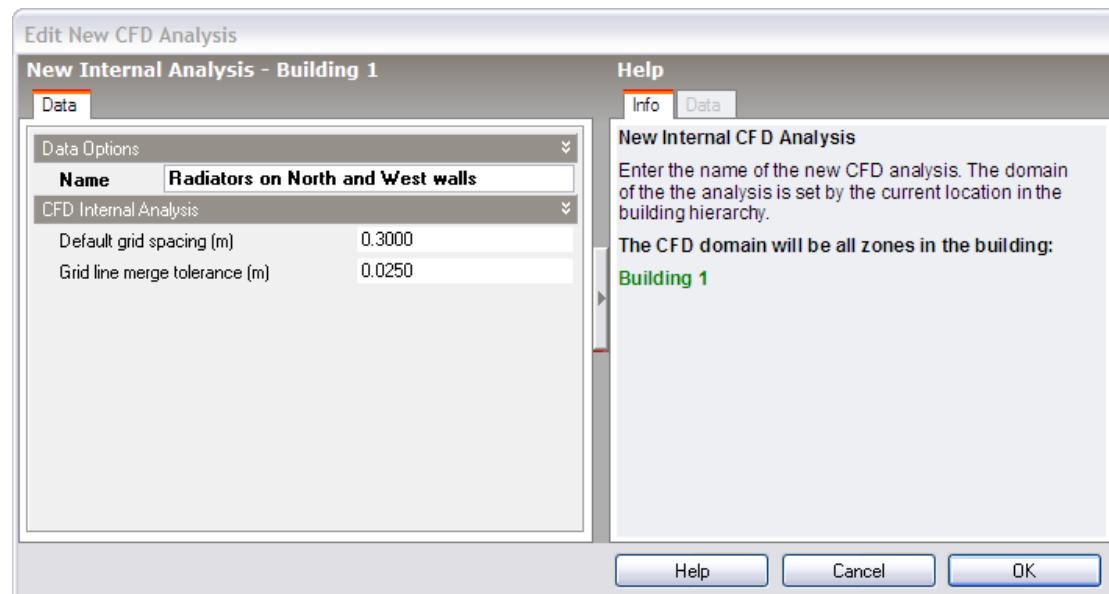


Setting Up a New Internal CFD Analysis

To create a new external CFD analysis, first make sure that you have selected the appropriate level object (building, building block or zone) in the model navigator and then click on the CFD tab:



The 'New CFD Analysis Data' 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 regions. Each CFD grid region along each major axis is automatically spaced using the 'default grid spacing' dimension.

Grid Line Merge Tolerance

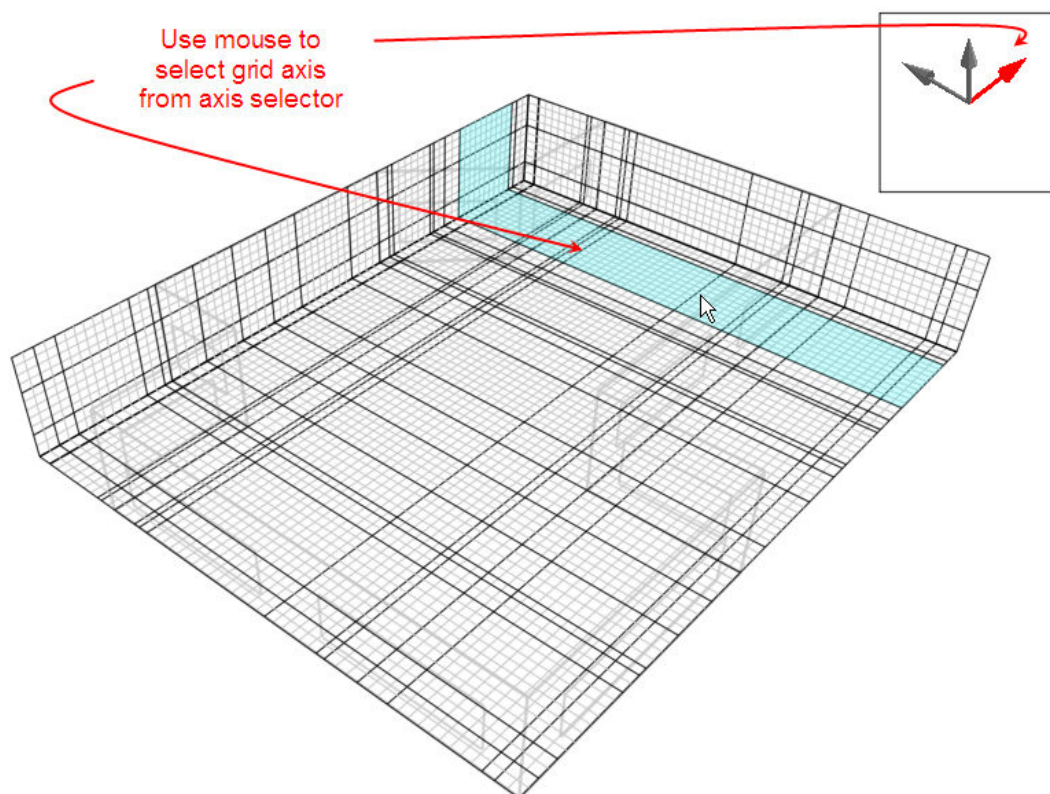
A potential problem when generating the grid is the creation of cells with a high aspect ratio that can lead to instability in the equation solver. In order to avoid such cells, grid lines that are very close together can be merged. The 'grid line merge tolerance' is the maximum dimension that will be used in determining whether or not to automatically merge grid lines.

Editing the CFD Grid

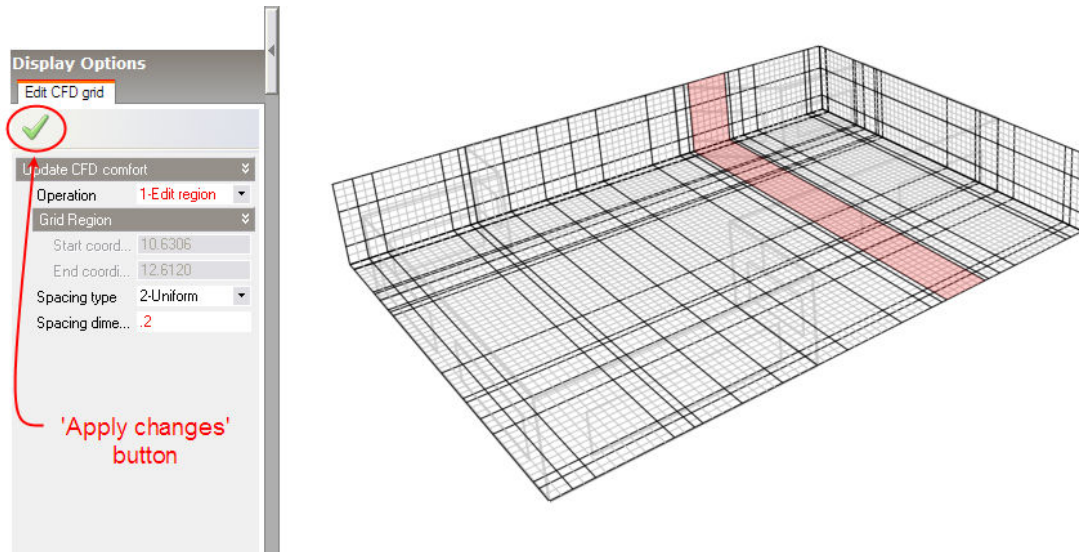


As soon as a new analysis data set has been created, a default CFD grid is generated using the default grid spacing defined as part of the data set. The default grid may be edited using the 'Edit CFD Grid' tool which allows you to change the spacing used for default regions, insert additional regions 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 $i > n/2$: $x_i = [(\text{region dimension})/2][2 - (2i/n)]^{\text{power}} + x_s$

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.

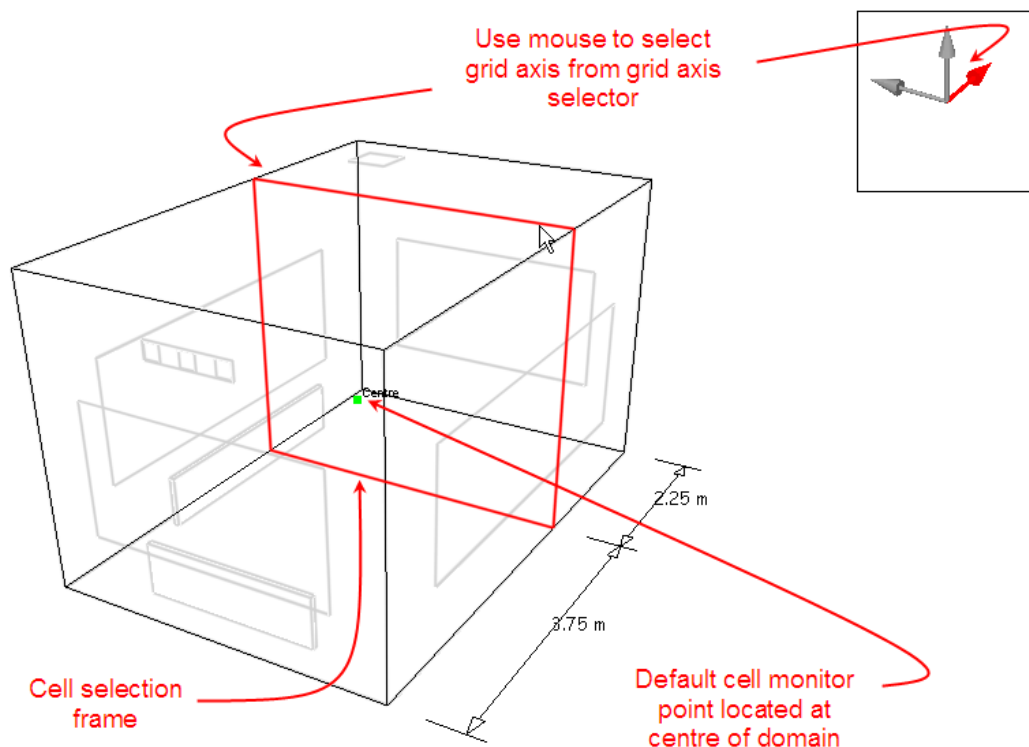
After selecting an operation and adjusting the required settings, you then need to click on the 'Apply changes' button to update the selected grid region with the current settings.

Setting Up CFD Cell Monitor Points

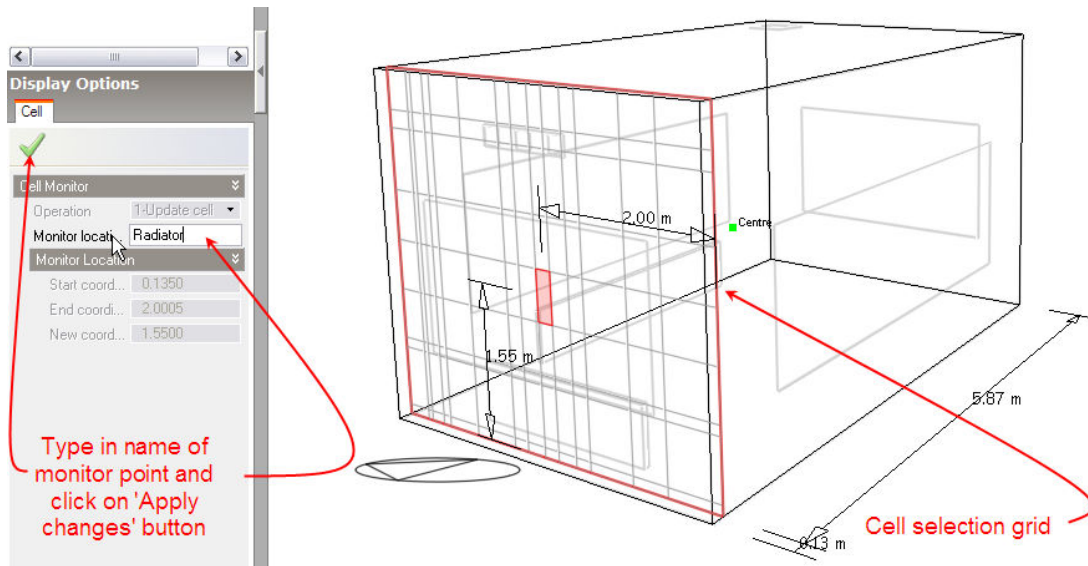


The CFD calculation process is iterative and is considered to have completed once 'convergence' has been achieved. The main indicator for convergence is the point at which the finite difference equations for all cells are satisfied by the current values of the appropriate dependent variables. However, 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 nam

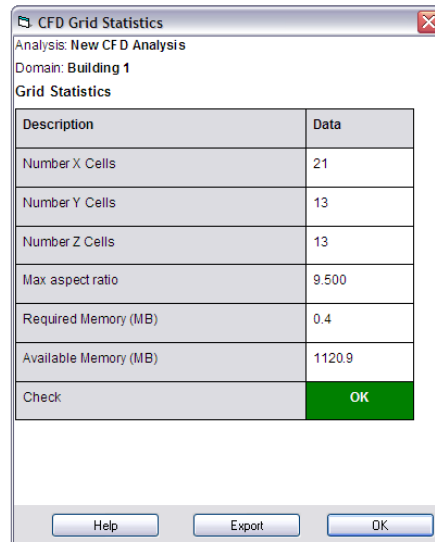
Enter name for monitor point.

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.

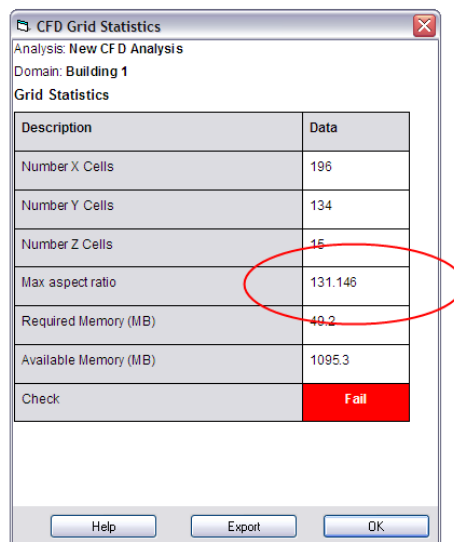
CFD Grid Information



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 50 or the required memory exceeds the physical memory available, the 'Check' will indicate a failure:



CFD Calculations



Please refer to the CFD Calculations and Convergence section for an overview of the calculation methodology that will help in understanding the concepts used to initialise, control and monitor the calculations.

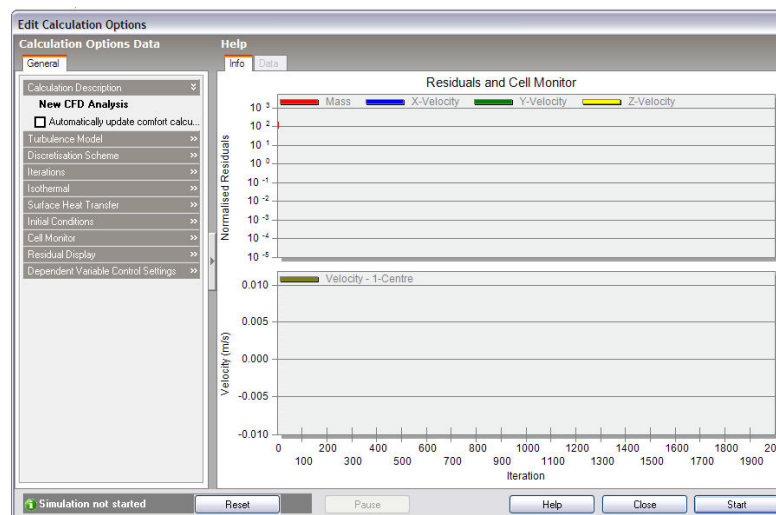
To run the calculations, click on the **Update calculated data** tool. If a problem is found with the grid generation, the CFD Grid Statistics dialog is displayed showing the error. There are three possible error conditions:

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 Results Manager' section).

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.

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. 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.

Calculation Options Data

The calculation settings panel incorporates the following settings:

Turbulence Model:

The following turbulence models are available:

- **1-k-ε:** 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:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$$

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.

- **2 – Constant effective viscosity:** The constant effective viscosity model is a much simpler approach and involves 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 - ε model and can be numerically much more stable.

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

Temperature is assumed to be constant throughout the calculation domain and the energy equation is removed from the calculations.

Surface heat transfer

The following surface heat transfer options are available:

- **1–Calculated** - Surface heat transfer coefficients are calculated by the CFD calculation scheme if the k- ϵ turbulence model has been selected.
- **2–User-defined** - Surface heat transfer coefficients can be defined for ceilings, wall and floors.

Initial conditions

In some cases, a faster solution can be achieved by setting the initial conditions closer to the final expected conditions.

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

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.

Dependent Variable Control Settings

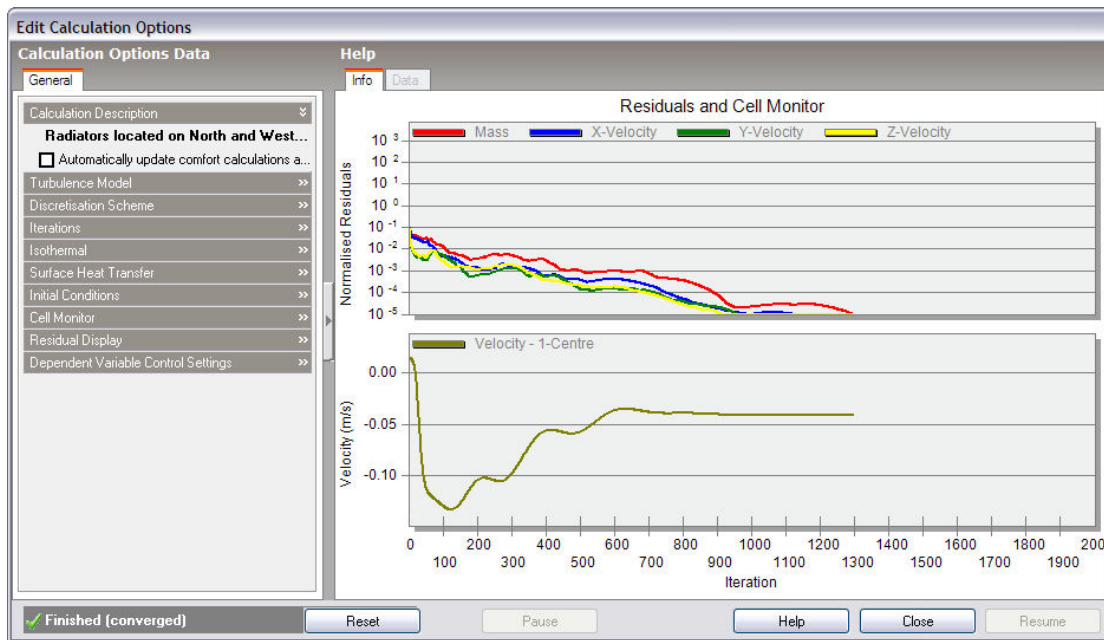
The calculations involve a nested iterative scheme whereby dependent variable equations are solved iteratively within an overall outer iterative loop (see ‘CFD Calculations and Convergence’ section). The dependent variable control settings enable control inner iterative dependent variable calculations.

- **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 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.

Residuals and Cell Monitor Graphs

The value of each 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 next '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 ‘Calculation Options Data’ section above). 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

If the plotted residuals are found to diverge or fluctuate significantly, in many cases you will find that 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.

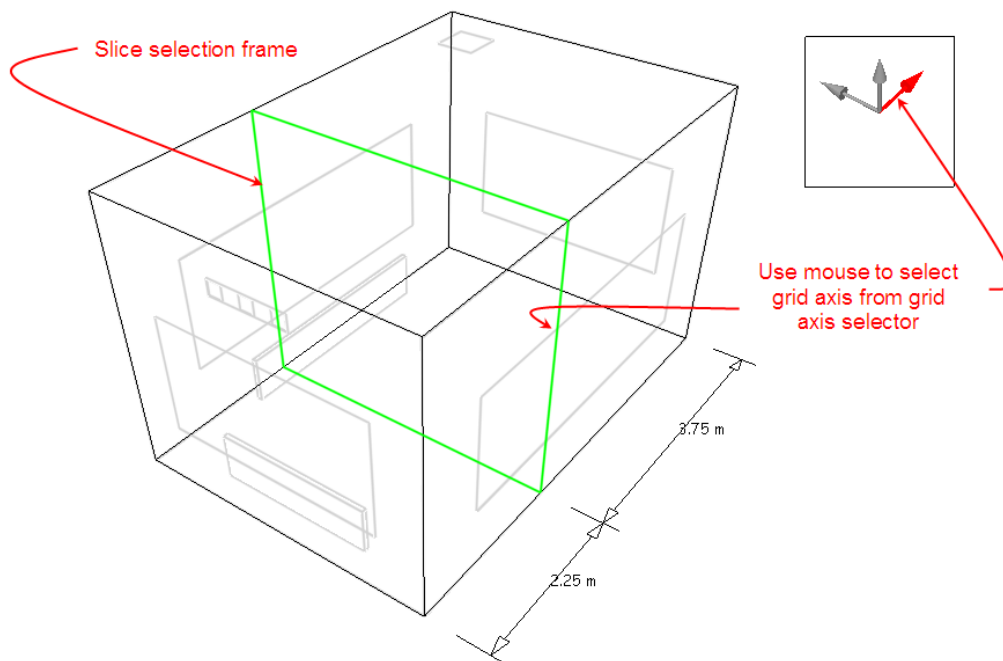
However, in some cases a solution can be very difficult, if not impossible, to achieve using the standard $k-\epsilon$ turbulence model. Particularly difficult problems involve very low velocity buoyancy-driven flows where diffusion is dominated by convection and this can cause velocities to fluctuate 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.

Displaying Results

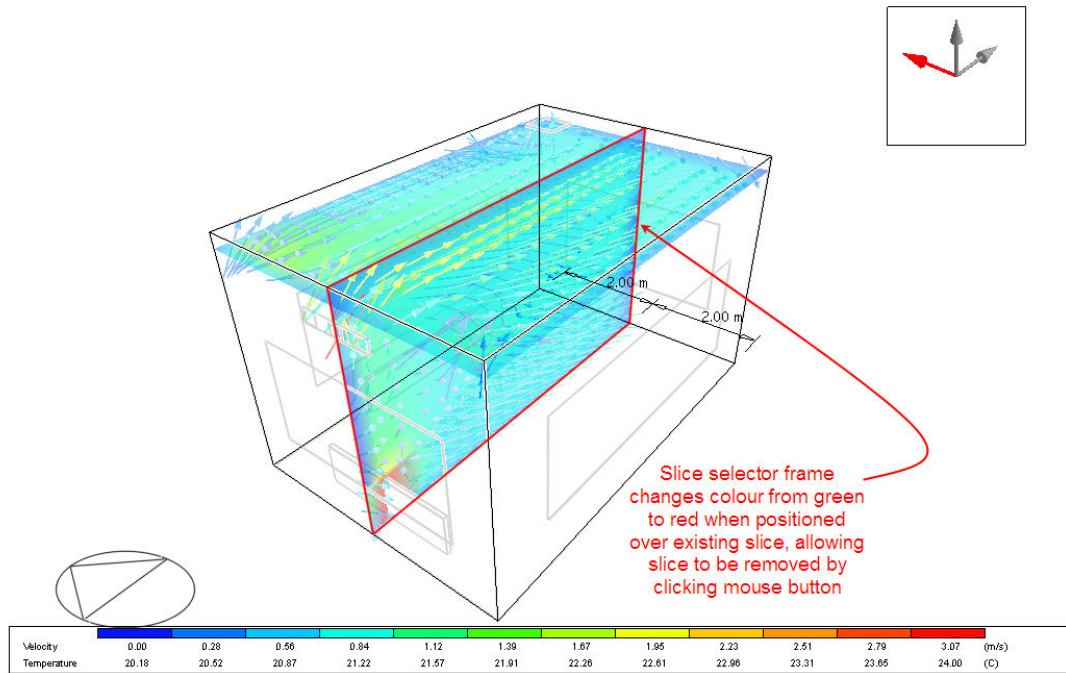


After completing or pausing the calculations, the CFD Display Options data panel and the 'Select CFD slice' tool become active.

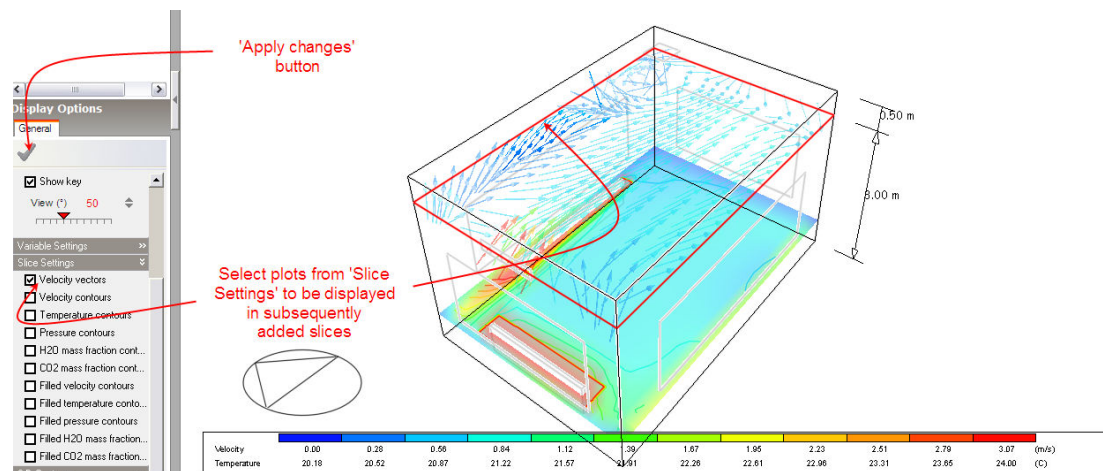
The main mechanism for displaying results is the slice tool that allows you to select a slice, along one of the main grid axes and perpendicular to it, within which can be displayed any of the selected results display plots. After selecting the 'Select CFD slice' tool, the CFD results screen is displayed 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.

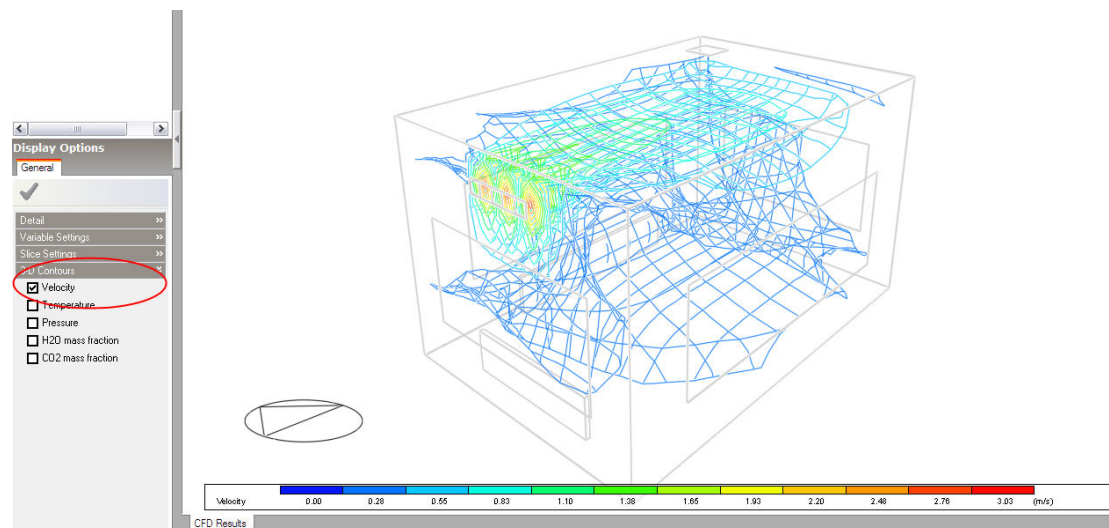
The 'Variable Settings' group contains groups of variable banding data for velocity, temperature and pressure variables. Also, under the variable settings 'Velocity' header, there are two velocity vector settings, 'Maximum vector length' and 'Velocity scale factor'. Velocity vectors are displayed as 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:



Conducting CFD Comfort Calculations

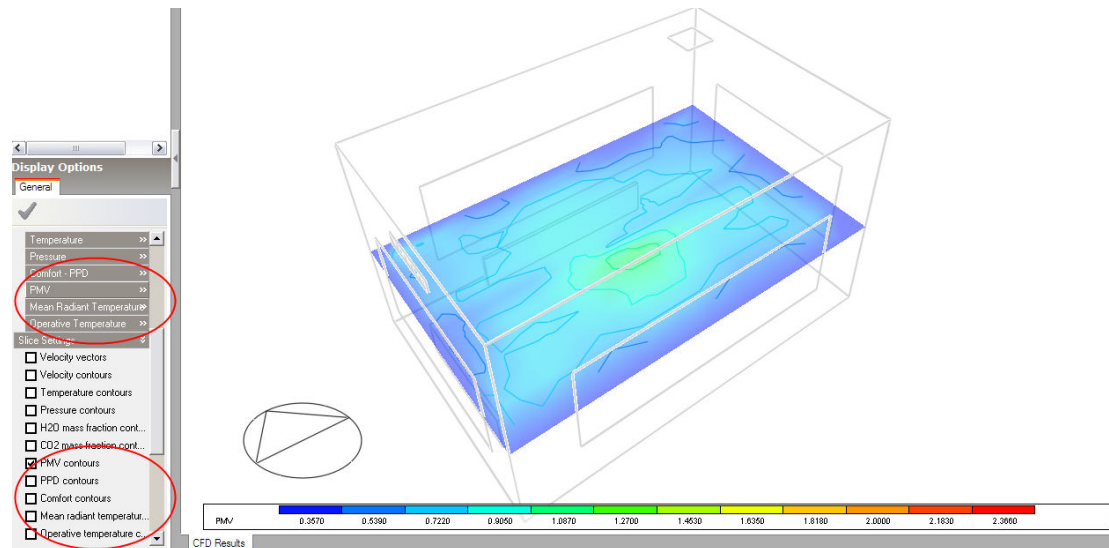


After completing or pausing the CFD the 'Update CFD comfort' tool will become enabled. After selecting the 'Update CFD comfort' tool, a progress bar will be displayed indicating that mean radiant temperature (MRT) calculations are in progress, followed by another progress bar for the comfort calculations themselves.

Comfort calculation metabolic rates and clothing levels are obtained from the data entered under the 'Metabolic' header on the 'Activity' tab of the model data panel.

After completing the calculations, some additional comfort display variables are available on the Display Options data panel:

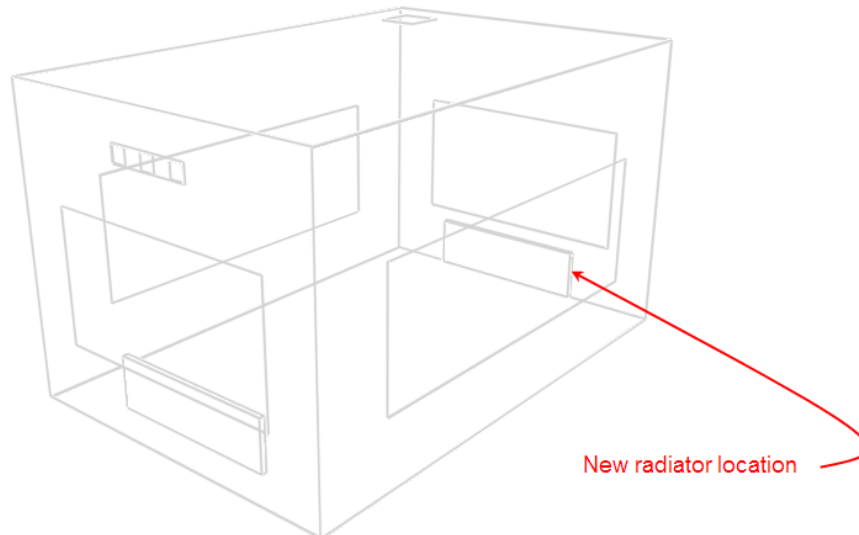
- PMV
- PPD
- Comfort Indices
- Mean Radiant Temperature
- Operative Temperature



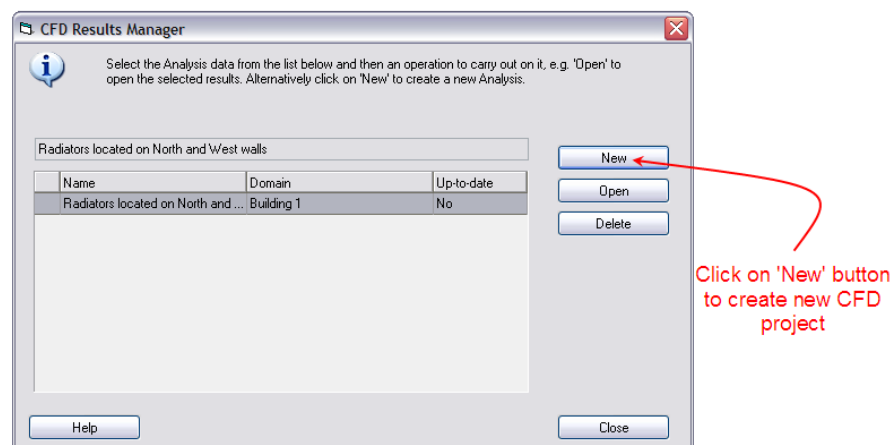
CFD Results Manager



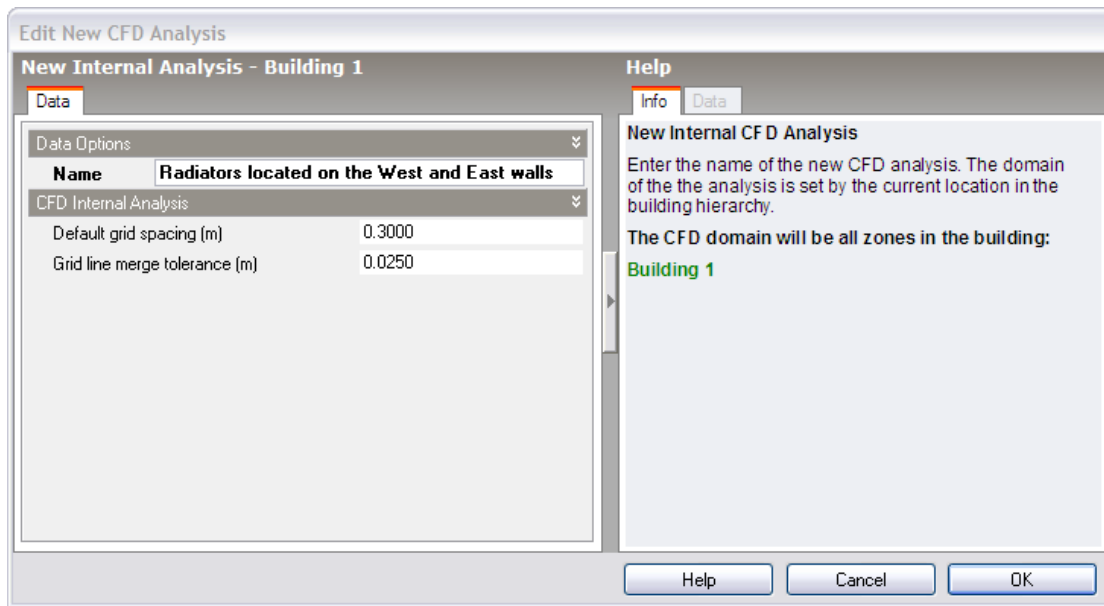
The **CFD results manager** tool allows you to create new CFD projects, open existing projects or remove existing projects. For example, considering the internal analysis example illustrated in the Displaying Results section, you may want to conduct another analysis after moving the position of a radiator 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:

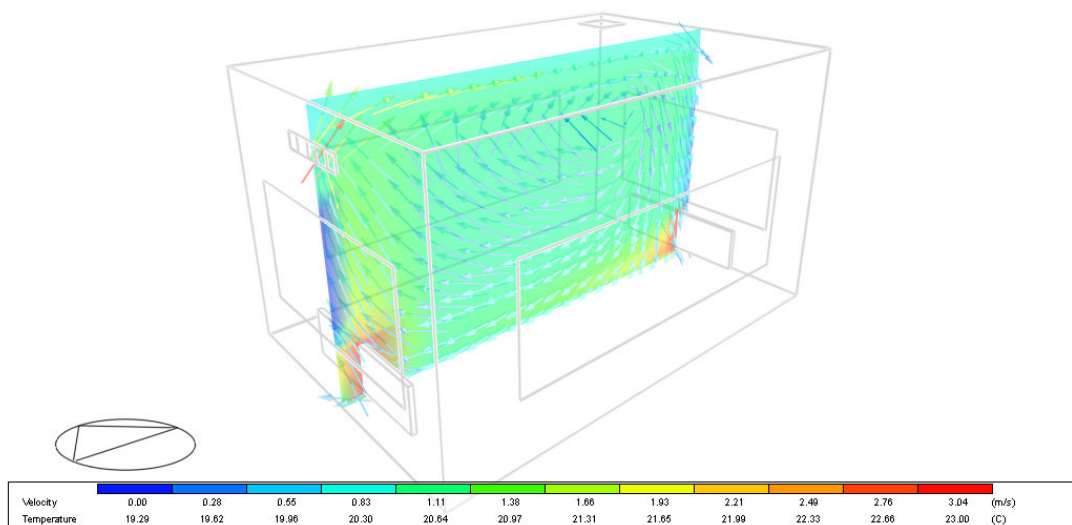


The CFD Results Manager dialog incorporates a list of all available projects together with buttons that allow you to create a new project, open a project or delete a project. For each project entry in the list, the first column indicates the project name, the second column the model domain object used 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:

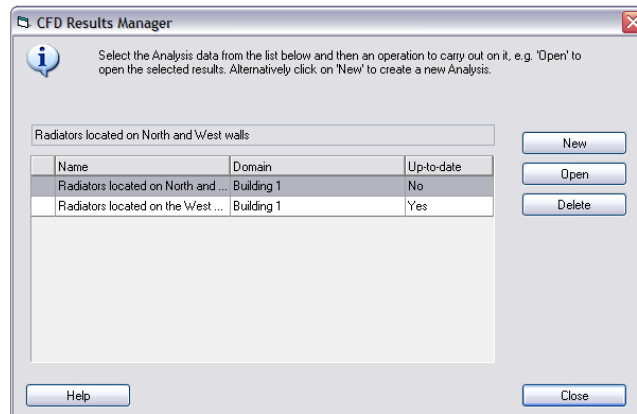


See the ‘Setting Up a New Internal CFD Analysis’ section for details of creating a new internal analysis.

After creating the new analysis, you can then conduct the calculations:



You can then use the CFD Results Manager to open the previous project in order to compare results:



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.

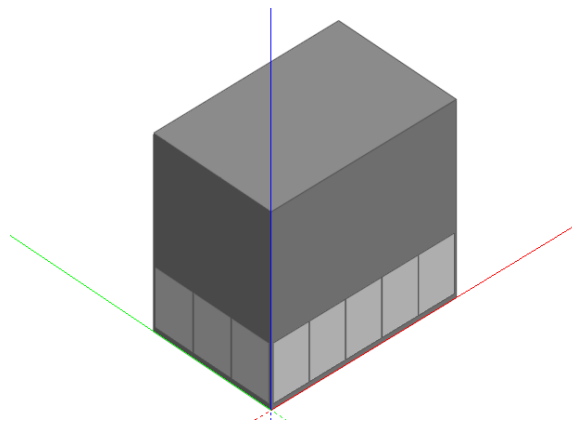
External analyses are simpler than internal analyses in that you do not need to define any internal boundary conditions. Also they are isothermal and take account principally of only velocity and pressure.

Refer to internal analysis for details of grid editing and monitor cell selection.

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.



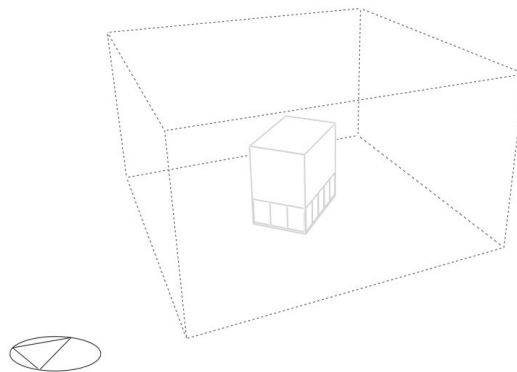
Creating a new CFD external analysis

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.

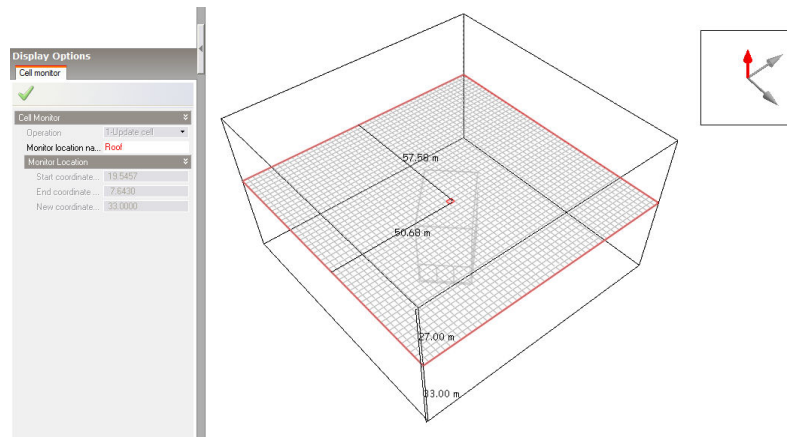


Notice the rotation of the domain with respect to the building to account for the wind direction.

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.

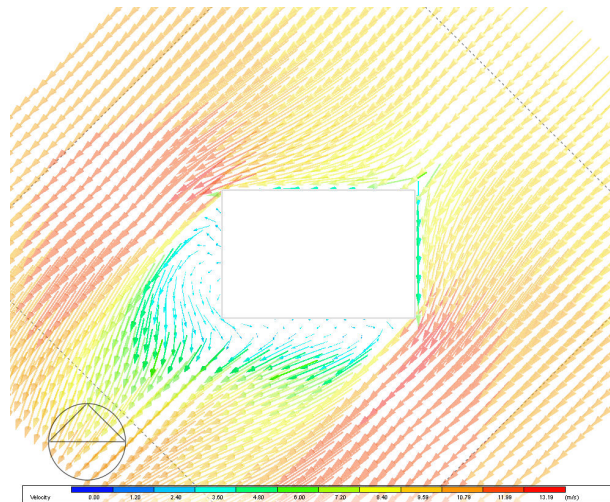
Keep the default options except for **Iterations** which should be set to **5000** and under the Residual Display header, switch on the **X, Y and Z-axis velocity residuals**.

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.

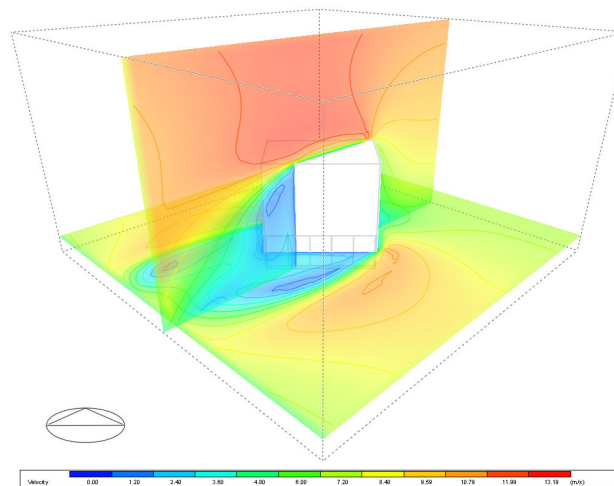
Press the **Cancel** button to close the calculation dialog.

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.



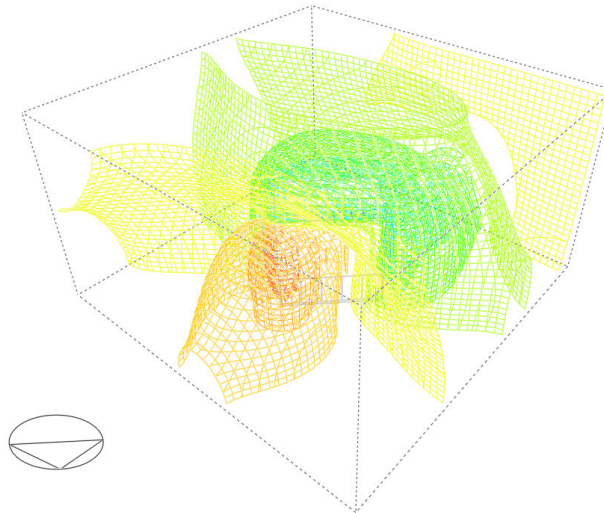
1. Clear the current slice from the display using the **Clear all slices** tool.
2. Switch off **Velocity vectors** under the Slice Settings header on the Display Options panel and switch on the **Velocity contours** and **Filled Velocity contours**.
3. Click on the green tick **Apply changes** button.
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

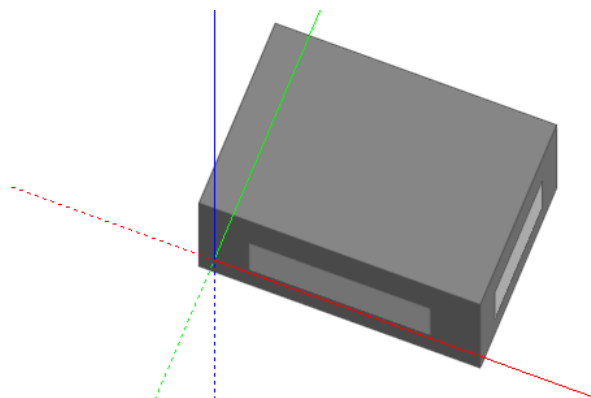
This internal CFD example will guide you through the basic minimum steps required for an internal CFD analysis:

- 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.

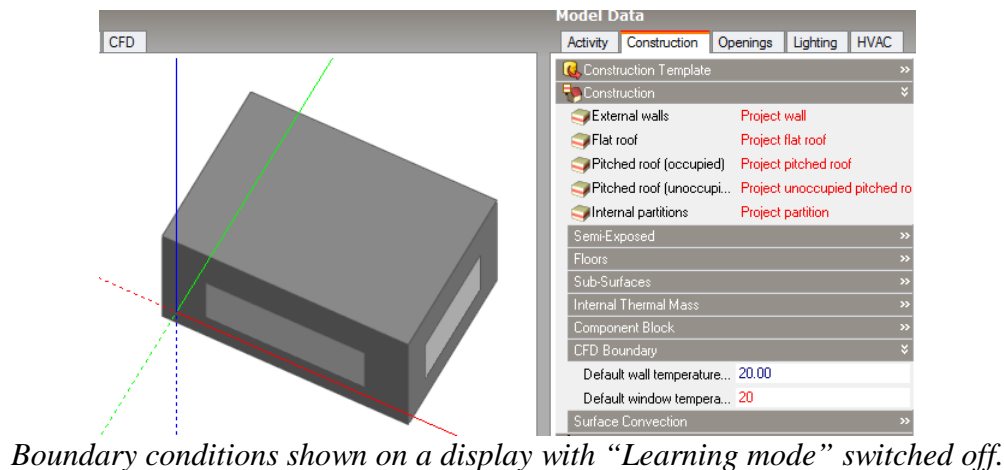


Default wall and window boundary temperatures

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 under the Construction header on the Construction tab of the Model data panel.

At the building level, change the default window temperature to 20°C.

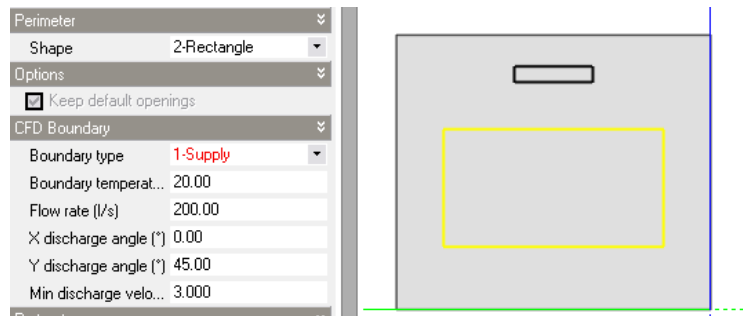


Go down to the zone surface facing North and change the default wall temperature to 18°C and the default window temperature to 10°C. Go through the same procedure for the west facing zone surface.

Adding surface supply diffuser and extract grille

Supply grille

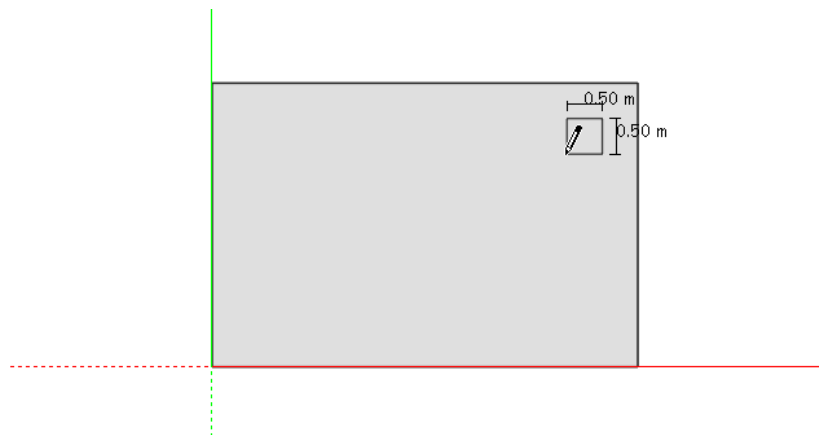
1. Click on the zone in the navigator and select the North-facing surface in the Edit screen. Double-click to drop down to the surface level.
2. Click on the **Normal** view button to obtain a normal view of the surface.
3. Click on the **Add CFD boundary** tool. The **CFD boundary** data panel is displayed 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.

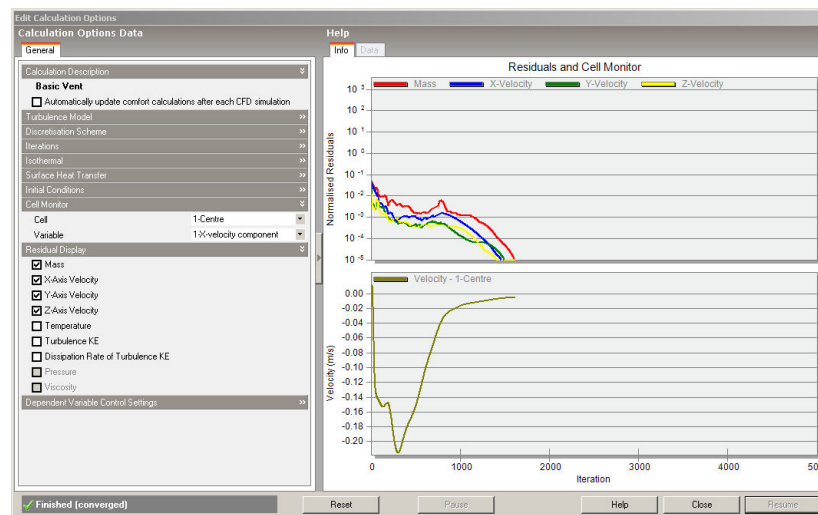
You will see the model in wire-frame mode by default and at this stage the CFD analysis has been created including the default grid and the model is ready for

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).

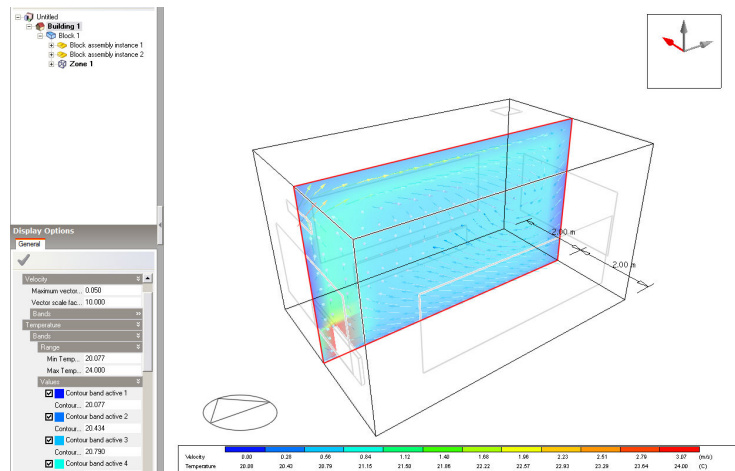


Press the **Cancel** button to close the calculation dialog.

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:

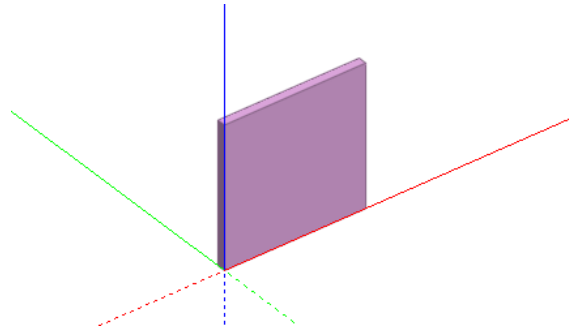
- Create a component representing a constant temperature radiator, convert the component to an assembly, assign CFD boundary conditions to the assembly and add the assembly to the model assembly library.
- Place the previously created radiator assembly within the zone.
- Edit the CFD grid.
- Add cell 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.

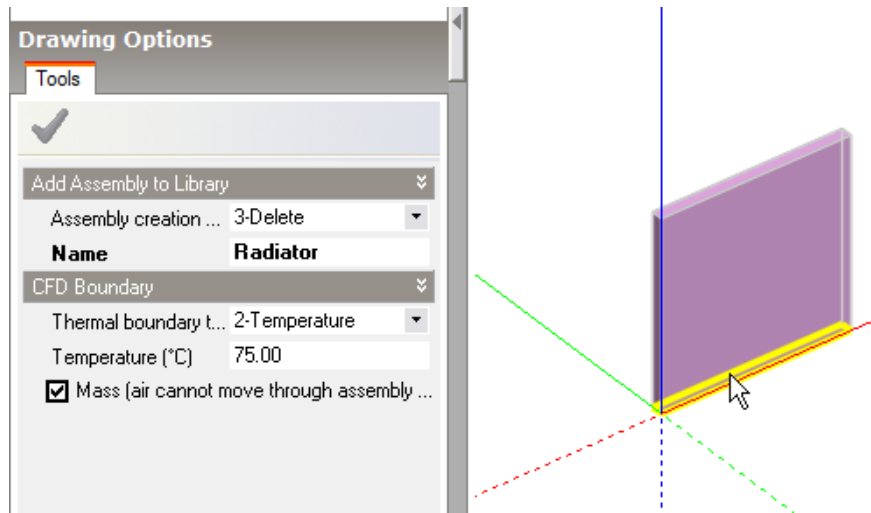
In this exercise a radiator assembly is to be created from a single component block, which is to be drawn using a rectangular extrusion.

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.



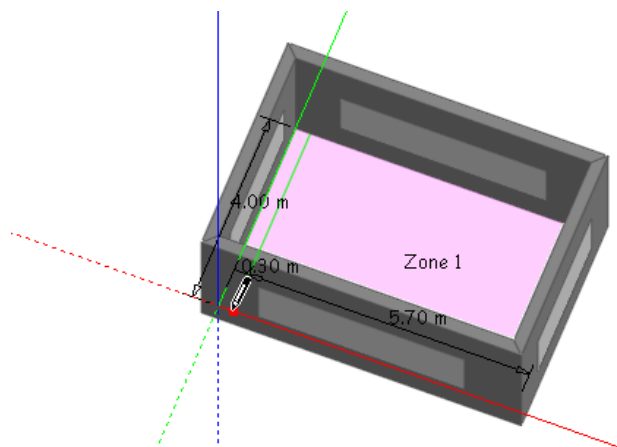
Positioning radiators within the block

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.

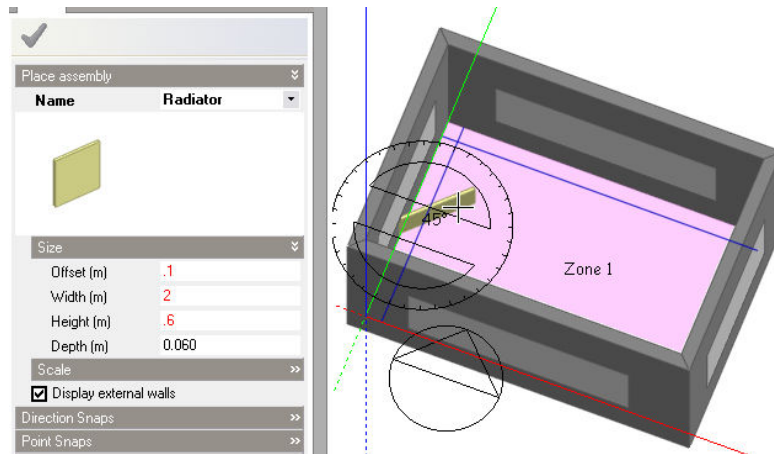
To do this, orbit the model to get a clear view of the west-facing end wall from the inside of the block and then click on the **Place construction line** tool.

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.

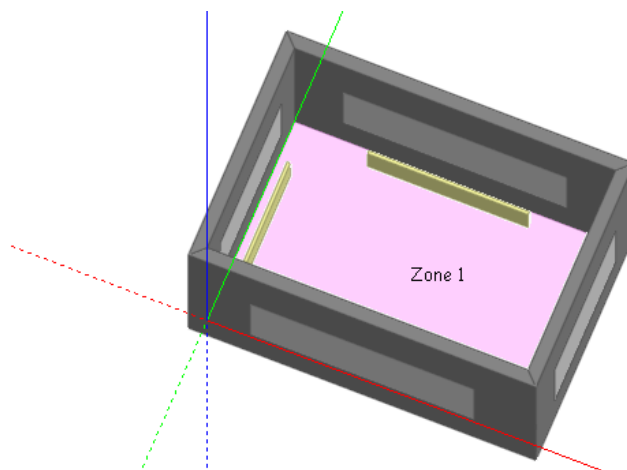


Click on the **Place assembly** tool and as you move the cursor back across the Edit screen, you will see that a copy of the previously created radiator assembly is attached to the cursor and the **Place assembly** data panel is displayed at the bottom left of the

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.

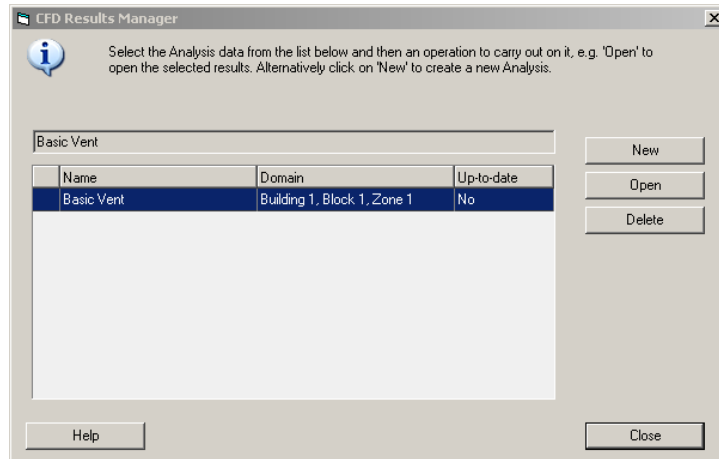


Repeat this procedure to place a 3.0m radiator adjacent to the North-facing wall.



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.

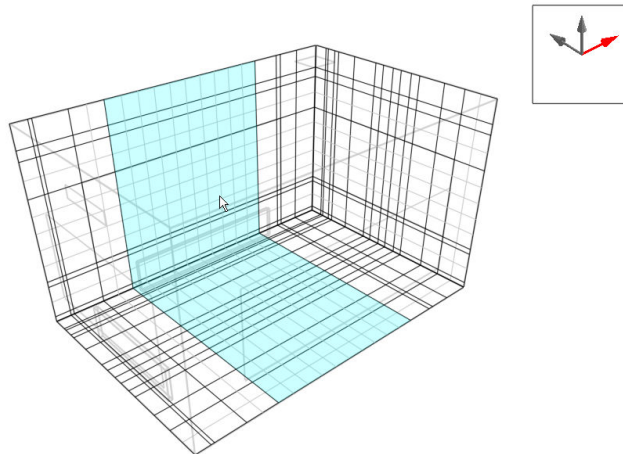
Click on the **New** button to create a new analysis using the model in its current state. The **New CFD analysis** dialog is displayed. Enter **Radiators and Vent** in the Name field and leave the default grid spacing and grid line merge tolerance at their default settings. The default grid spacing 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.

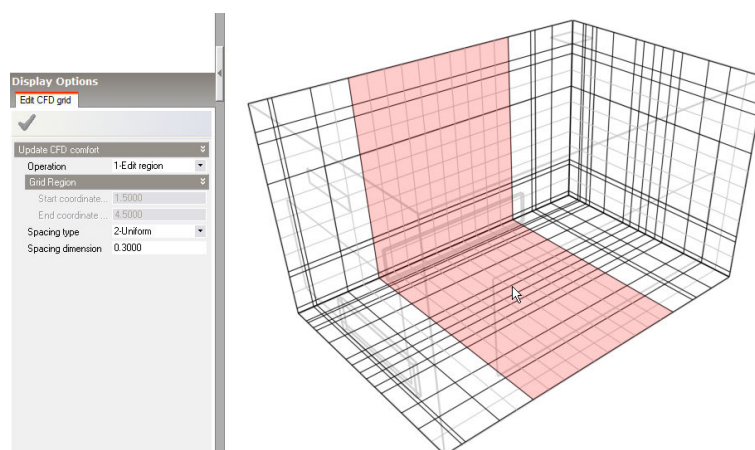
You will see the model in wire-frame mode by default and at this stage the CFD analysis has been created including the default grid and the model is ready for calculations.

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.

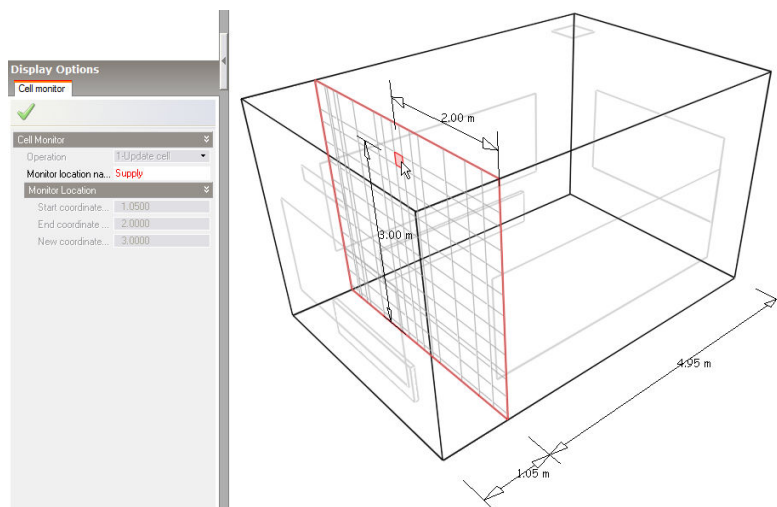
After editing the default grid, you can reset it to the default by clicking on the **Reset CFD grid** tool. Also, to obtain information about the grid, you can click on the **Show CFD grid statistics** tool which brings up a dialog indicating the number of cells, required memory, available memory and maximum cell aspect ratio.

For the purpose of this tutorial, the default grid will be used.

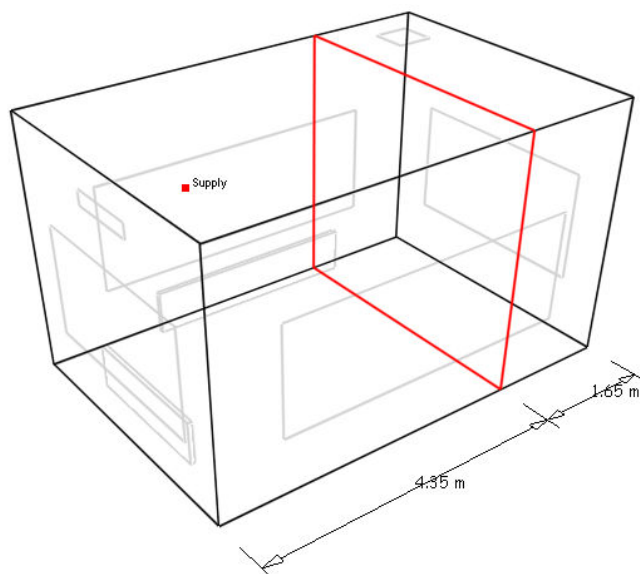
Setting up monitor cells

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.

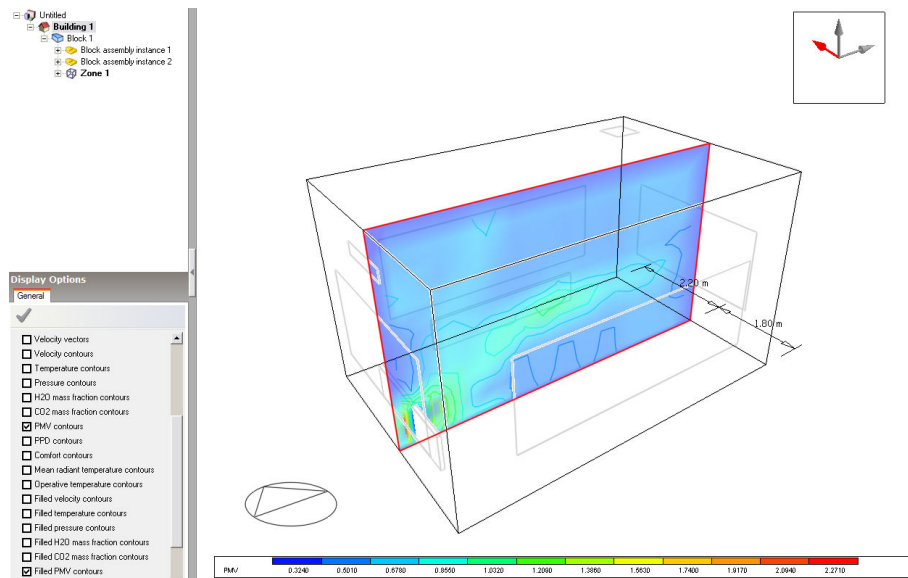
You can revert to the absolute minimum and maximum values in the model by using the Tools > **Reset CFD display variables** menu command.

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 Result 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 I - Calculated Natural Ventilation

This section is provided for background information.

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:

$$q = C \cdot \delta P^n$$

Where:

q is the volumetric flow through the opening/crack.

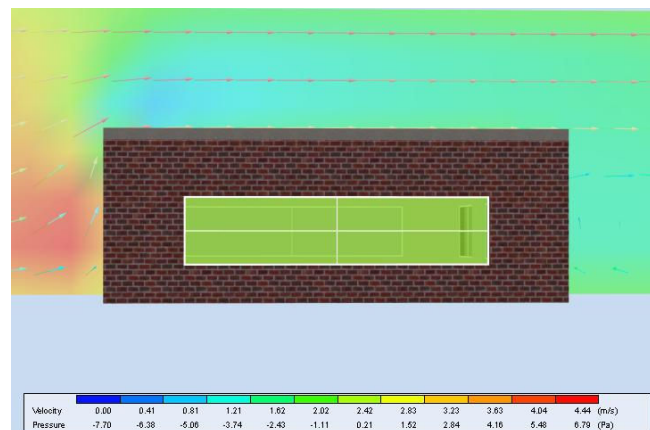
δP is the pressure difference across the opening/crack.

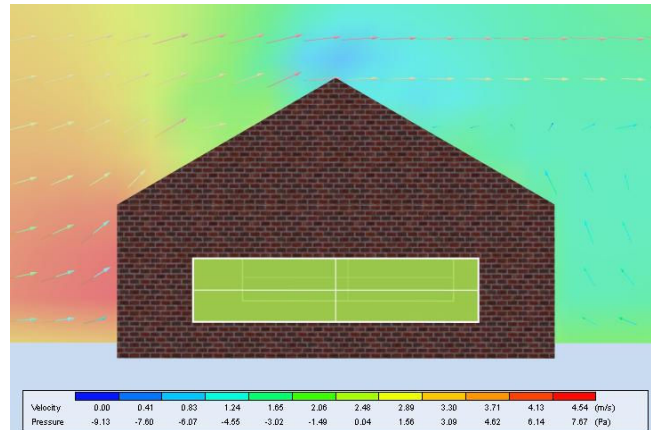
n is the flow exponent varying between 0.5 for fully turbulent flow and 1.0 for fully laminar flow.

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 DesignBuilder CFD screenshots above show examples of this for 2 typical building configurations. Positive wind pressure is shown in red and negative pressure in blue.

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 Wind Pressure Coefficients (WPC) templates and provide default pressure coefficients suitable for use in basic design calculations for buildings having no more than three stories. The WPC data is entered at the surface level.

For more detailed analysis, or for buildings having 4 or more stories you should ideally obtain specific pressure coefficient data from:

1. Wind tunnel measurements on a physical model or
2. CFD analysis of from wind tunnel measurements

Enter the data for each surface under the Pressure Coefficients header on the Openings tab. It will be possible to directly calculate wind pressure coefficients using DesignBuilder CFD external analysis in future versions.