

IDA Indoor Climate and Energy 4.0

EQUA Simulation AB, September 2009

Manual version: 4.0

Copyright EQUA Simulation AB 2009

Contents

<u>1. About the Manual.....</u>	<u>6</u>
<u>2. Basic Principles of IDA and IDA Indoor Climate and Energy.....</u>	<u>7</u>
<u>2.1. Introduction.....</u>	<u>7</u>
<u>2.2. The three levels of user interface.....</u>	<u>7</u>
<u>2.3. Forms and dialogs</u>	<u>9</u>
<u>3. Model description</u>	<u>10</u>
<u>3.1. The Primary system.....</u>	<u>10</u>
<u>3.1.1. The Boiler</u>	<u>11</u>
<u>3.1.2. The Chiller</u>	<u>11</u>
<u>3.2. The air handling system.....</u>	<u>11</u>
<u>3.3. The zone models.....</u>	<u>14</u>
<u>3.4. Solar radiation modeling</u>	<u>15</u>
<u>3.5. Airflows</u>	<u>16</u>
<u>3.5.1. Air flow input forms</u>	<u>17</u>
<u>3.6. General information regarding zone cooling and heating room units.....</u>	<u>19</u>
<u>3.7. Ideal heaters and coolers.....</u>	<u>19</u>
<u>3.8. Hydronic heating devices.....</u>	<u>20</u>
<u>3.9. Cooling units</u>	<u>21</u>
<u>3.9.1. Active beams</u>	<u>22</u>
<u>3.9.2. Heating/Cooling floor (Expert edition)</u>	<u>23</u>
<u>4. CAD and image import.....</u>	<u>25</u>
<u>4.1. Supported file formats.....</u>	<u>25</u>
<u>4.1.1. BIM</u>	<u>25</u>
<u>4.1.2. CAD and vector graphic files</u>	<u>25</u>
<u>4.1.3. Image files</u>	<u>25</u>
<u>4.2. Importing CAD objects and images.....</u>	<u>26</u>
<u>4.3. Moving and scaling CAD objects and images.....</u>	<u>26</u>
<u>4.4. Shading by imported 3D objects.....</u>	<u>26</u>
<u>4.5. Storage of CAD objects.....</u>	<u>26</u>
<u>4.6. Importing IFC files.....</u>	<u>27</u>
<u>4.7. Mapping data from IFC.....</u>	<u>27</u>
<u>4.8. Create zones.....</u>	<u>28</u>
<u>5. Getting started with the advanced level.....</u>	<u>30</u>
<u>5.1. Example 1: Presenting more data in an existing diagram.....</u>	<u>32</u>
<u>5.2. Example 2: Shade control by zone temperature (Expert edition required).....</u>	<u>32</u>

6. Tips and tricks.....	35
6.1. Speeding up computation.....	35
6.2. Numerical instabilities.....	36
7. ICE Reference Manual.....	37
7.1. General.....	37
7.1.1. IDA Indoor Climate and Energy	37
7.1.2. The Geometry in IDA Indoor Climate and Energy	37
7.1.3. Objects in IDA Indoor Climate and Energy (in alphabetical order)	39
7.1.4. Objects in IDA Indoor Climate and Energy (in hierarchical order)	40
7.2. Building simulation.....	42
7.2.1. General tab	42
7.2.2. Floor plan tab	58
7.2.3. 3D Plan tab	67
7.2.4. Simulation tab	72
7.2.5. Outline tab	76
7.2.6. Results tab	76
7.3. HVAC Systems.....	77
7.3.1. The Primary system	77
7.3.2. Air handling unit	78
7.3.3. Form for Heating Coil	79
7.3.4. Form for cooling coil	80
7.3.5. Form for Heat Exchanger	81
7.3.6. Edit temperature depending input	82
7.4. Zone.....	84
7.4.1. Zone form	84
7.4.2. Calculation of thermal bridge coefficients	85
7.4.3. Dialog for controller setpoints	87
7.4.4. Operative temperatures	88
7.4.5. Form for occupant load	89
7.4.6. Form for Lights	92
7.4.7. Form for equipment load	93
7.4.8. Convective internal mass	93
7.4.9. Thermal mass	94
7.4.10. Ideal cooler	95
7.4.11. Ideal heater	96
7.4.12. Local AHU	96
7.4.13. Zone advanced form	96
7.5. Walls, Floor and Ceiling.....	97
7.5.1. Form for wall, floor and ceiling	97
7.5.2. Editing objects on a surface	99
7.5.3. Dialog for wall definition	100
7.5.4. Dialog for surface	102
7.5.5. Wall part	102
7.5.6. Leak	103
7.5.7. Windows and openings	104
7.5.8. Heating, cooling, ventilation	118

7.6. IDA Resources and Database.....	125
7.6.1. Database objects in IDA Indoor Climate and Energy	125
7.6.2. Dialog for Schedule	126
7.6.3. Editing a Profile	129
7.7. Mathematical Model.....	131
7.7.1. Schema for zone (advanced level)	131
7.8. Results.....	132
7.8.1. Result – AHU (Air Handling Unit) temperatures	132
7.8.2. Result – Airflow through the Air Handling Unit	133
7.8.3. Result – Primary system’s temperatures	134
7.8.4. Result – Total heating and cooling	135
7.8.5. Result – Main temperatures	136
7.8.6. Result – Heat balance	137
7.8.7. Result – Air temperature at floor and ceiling	139
7.8.8. Result – Fanger’s comfort indices	140
7.8.9. Result – Air quality	141
7.8.10. Result – Daylight level	142
7.8.11. Directed operative temperatures	143
7.8.12. Air flow in zone	144
7.8.13. Airborne heat flow into zone	145
7.8.14. Surface temperatures	146
7.8.15. Surface heat fluxes	147
7.8.16. Result – Supplied Energy	147
7.8.17. Result – Systems energy	147
7.8.18. Result – Energy report for an air handling unit	149
7.8.19. Result - Lost work	150

1. About the Manual

The manual consists of two parts. The first six chapters can be read through. Remaining chapters are a reference manual. There is an index to help the reader locate the information required in the reference part.

Before reading the manual, it is advisable to follow the Getting Started Guide, which is a separate document. This will bring you through a worked example and fill in supporting information in the process.

Chapters 2 and 3 provide a general overview of program management and the actual simulation models. Chapter 4 treats import of CAD data and chapter 5 is an introduction to working at the advanced level. In Chapter 6 some tips for getting optimal performance are given.

2. Basic Principles of IDA and IDA Indoor Climate and Energy

2.1. Introduction

IDA Indoor Climate and Energy (ICE) is a program for study of the indoor climate of individual zones within a building, as well as energy consumption for the entire building. IDA Indoor Climate and Energy is an extension of the general IDA Simulation Environment. This means that the advanced user can, in principle, simulate any system whatsoever with the aid of the general functionality in the IDA environment.

Normally, the system to be simulated consists of a building with one or more zones, a primary system and one or more air handling systems. Surrounding buildings might shade the building. The air inside the building contains both humidity and carbon dioxide. Weather data is supplied by weather data files, or is artificially created by a model for a given 24-hour period. Consideration of wind and temperature driven airflow can be taken by a bulk air flow model. Predefined building components and other parameter objects can be loaded from a database.

2.2. The three levels of user interface

The user interface is divided into three different levels, with different support and scope for the user. At the simplest level, called wizard, the user enters input into one or more forms, which are displayed in sequence. The user is then given the opportunity of carrying out a simulation directly, or transferring the data entered to the next level, called the standard level. Usage of the IDA Room wizard, which is the only wizard that is included in the standard distribution, is treated in online help texts. Press the ?-button in the interface (Figure 2.1).

The screenshot shows the 'roomwiz1.idm - Room Wizard' window with several tabs: 'Simple data', 'General', 'Geometry and horizon', 'Walls and floor', 'Loads and ventilation', and 'Results'. The 'Simple data' tab is active, displaying the following fields:

- Location and case:** Sizing case (Summer, checked 'with cooling', Winter), Simulation date (15 Jul 2002, 15 Jan 2002), Location (Stockholm/Bromma), Max temp. (26.1 °C), Min temp. (17.3 °C).
- Zone and materials:** Envelope (Medium), Window area incl. frame (1.2 m²), Glazing (2 pane glazing, clear, 4-12-4), Internal shading (No internal shading), Orientation (South). A 3D diagram shows a room with dimensions 4 m, 2.5 m, and 2.6 m.
- Thermal loads:** Hum of occup (1 items, 0 hours), Light (50 W, 8 hours), Other loads (150 W, 8 hours).
- Operation:** Supply air flow (20 l/s, 72 m³/h), Fan operation time (24 hours), Supply air temp. (16 °C), Thermostat setpoint (22 °C).

Buttons at the bottom include 'Start simulation' and 'Create building model'.

Figure 2.1. One of the tabs of IDA room

At the standard level, the user is given greater freedom to design a building model. This level defines geometry, materials, controller settings, loads, etc., in a manner that should be easy to

handle for a majority of engineers. The basic steps of using the standard level are covered by the Getting Started Guide, which is distributed as a separate document.

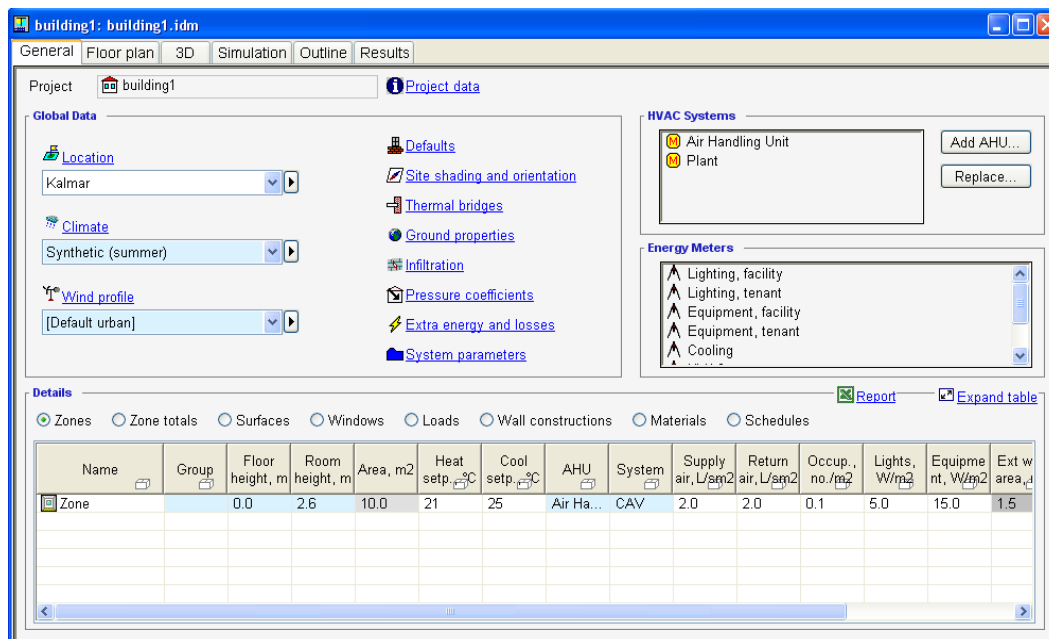


Figure 2.2. Main form for the building at standard level

At the advanced level, the simulation model is no longer defined in physical terms, but in the form of connected component models, whose meaning is defined by equations. At this level, the individual time evolution of variables can be studied. All equations, parameters and variables can be examined at this level.

A user of the Expert Edition of the program may also edit the connection structure at the advanced level. Some of these operations are easy to carry out, e.g. changing a proportional controller with a thermostat. Others are more complicated and require a deeper knowledge of the design of the models.

Use of the advanced level is introduced in Chapter 5. There is also a great deal of information about this in on-line help texts.

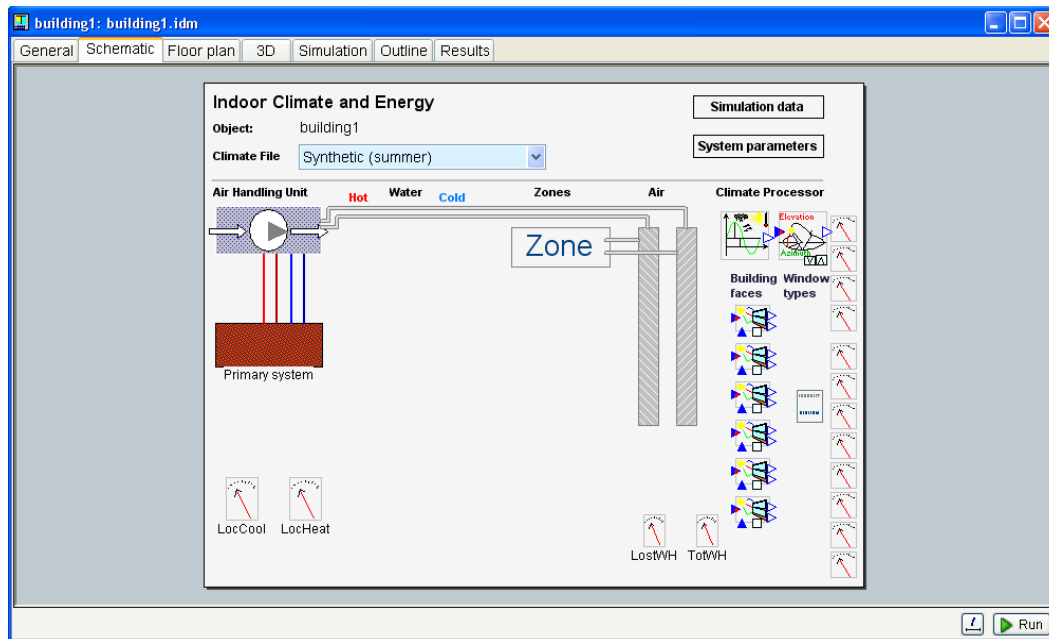


Figure 2.3. Main Form for the building at the advanced level

2.3. Forms and dialogs

The Windows part of the program (everything but IDA Room, which runs in a web browser) is built up around forms and dialogs. The forms contain no Cancel button, i.e. there is no access to earlier versions of a form; only the current version and its parameter values exist. Forms do not lock each other, and so several windows with forms can be open simultaneously (but only the one where work is in progress is active). A form can be printed. Simulations and many other operations can be carried out without having to first close open forms. Dialogs, i.e. input windows with OK and Cancel buttons, work in IDA as in most other Windows programs. They lock everything else but the current dialog window.

3. Model description

This chapter treats the mathematical models of IDA Indoor Climate and Energy together with some of their input forms. The reader is assumed to have mastered the basics of the program by first following the getting started guides. More information can be found on the User's page: Help menu, IDA on the Web, ICE User Support.

A building model consists of a single or several thermal zones, a single or several air handling units and a single primary system (a boiler and a chiller). When a new model is initiated at the standard level, default air handling units and primary systems are inserted automatically. The default systems have unlimited capacity for providing the zones with air and water at given temperatures. By default, the supply air temperature is kept constant at 17 °C; the chilled water temperature is 15 °C and the heated water temperature is a function of the outdoor air temperature. For many studies nothing needs to be altered in the HVAC systems.

It is not possible to remove the HVAC systems altogether from a building, but if mechanical air flows are set to zero in all zones, no connections to the air handling unit will be created, and all flows in the unit will be (close to) zero. Similarly, if no water based room units are inserted into zones (only ideal heaters and coolers are used) all flows in the plant will also be negligible.

This description deals firstly with the primary system, then follows the supply chain to the AHU (Air Handling Unit), and finally to the zones.

3.1. The Primary system

In the default configuration, the primary system consists of seven components (designated 1-6 respectively in Figure 3.1): chiller (1) and a schedule (2) for its operation as well as a boiler (3), a controller (4) for hot water supply temperature and a schedule (5) for night setback operation. Also connected to the boiler is a schedule for its operation (6). The six energy meters in the lower right corner monitor energy consumption of various categories in the primary system.

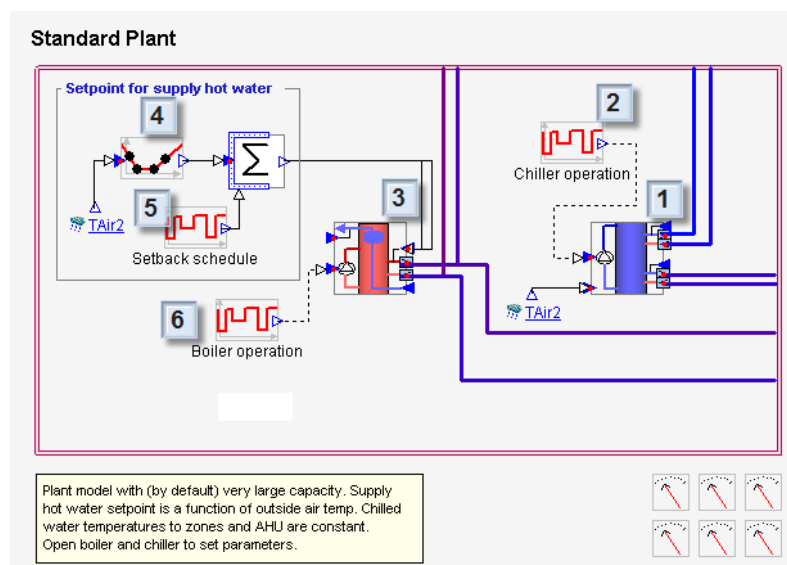


Figure 3.1. The primary system in the default configuration

3.1.1. The Boiler

The boiler consumes energy, e.g. gas and electricity, and produces warm water with given temperature and pressure for circulation through water based heat exchangers in the building. It also consumes energy for production of domestic hot water and pumping. Boiler efficiency is by default constant, as specified in the Defaults form. Open the component to view key parameters.

It is also possible to model water heating efficiency as a function of boiler temperature and part load. The model and parameter definitions used are the same as those of EnergyPlus. In ICE 4.0, there is no database support for this performance data.

Domestic hot water is always produced at a constant efficiency. The needed consumption data is most conveniently specified in the Defaults form.

Pumping power consumption for hot water circulation can be specified in two ways: (1) as a function of water flow through the boiler (default) or (2) as a proportion of distributed heat.

The water flow option assumes an ideal pressure controlled pump with constant efficiency. Alternatively, by setting the efficiency to a large number and specifying the k1 parameter, pumping effort can be given in proportion to distributed heat. PSetMax should still have a somewhat realistic value.

Pumping power for domestic hot water circulation can also be specified via k2, which specifies pumping power as a fraction of domestic hot water heating power.

The setpoint for the hot water supply temperature comes from a special controller component connected to the boiler. The controller provides a graph showing the setpoint as a function of the outdoor air temperature. (Press F1 in the dialog for detailed instructions.)

3.1.2. The Chiller

The chiller and its circulation circuit operate in a similar way, but differ in some aspects from the boiler. The chiller uses electrical power to produce chilled water at two different constant temperatures (but with the same pressure). The colder water, normally 5 °C, supplies the AHU. The somewhat warmer temperature, normally 15 °C, goes to the zones.

Similarly as for the boiler, EnergyPlus correlations for modeling temperature and part load behavior may be applied.

3.2. *The air handling system*

In the default configuration, the air handling system consists of the following components (Figure 3.2): supply air temperature setpoint controller (1), exhaust fan (2), heat exchanger (3), heating coil (4), cooling coil (5), supply fan (6), schedule (7) for operation of both fans and a schedule for the operation of the heat exchanger (8). The unit provides temperature-controlled air at a given pressure. Some key parameters of individual components are presented in the form; open them to edit.

The supply air temperature setpoint is connected to the heat exchanger and to both coils. All three components have separate ideal control circuits, which independently strive to maintain the setpoint. After the coils, the supply fan raises the supply air temperature further by either a fixed number of degrees (default) or by depositing motor and drive losses to the air stream.

In the setpoint controller, three methods are provided for setpoint selection. In the default AHU, the setpoint is set to constant, 16 °C. The second alternative is to let the setpoint vary with time according to a schedule. Thirdly, an option is available where the setpoint is calculated as a (user-defined) function of outdoor air temperature. Note that if for example the chiller (that supplies the cooling coil from the plant) has been turned off or has insufficient capacity, the supply air will not be cooled to the setpoint.

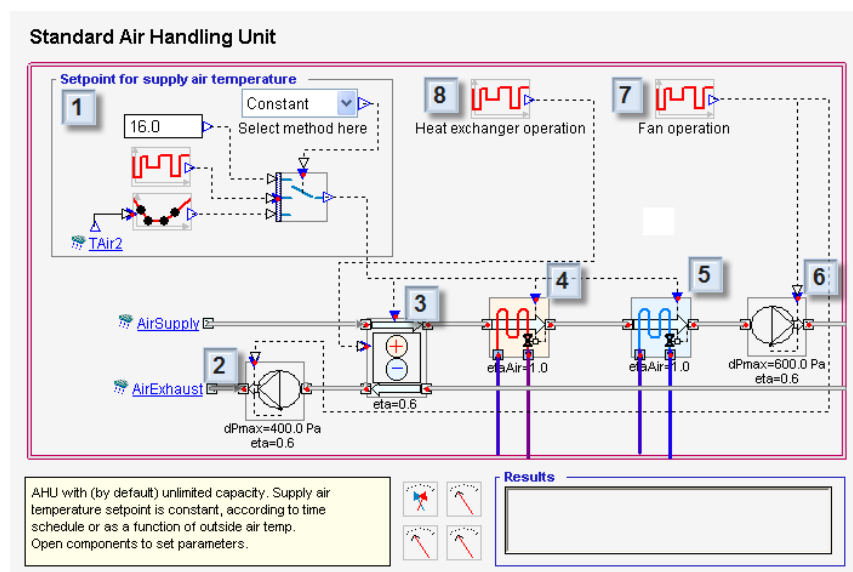


Figure 3.2. Default air handling unit

The heating coil has two important parameters: the air side temperature effectiveness and the desired water side temperature reduction. Capacity control is achieved by adapting the actual effectiveness up to the given maximum level. The necessary water flow is calculated and the water temperature is reduced, if possible, by the desired number of degrees. There is no bypass on the liquid side; control is achieved by simply restricting the water flow.

In the default configuration, the temperature effectiveness is set at 1.0. There are basically two situations when it may be desirable to change this to a more realistic value: when sizing the actual coil by means of simulation experiments, or when making energy calculations in cases where the boiler efficiency is dependent on temperature conditions. In addition, the simplest and quickest way of removing the entire coil is to set the effectiveness to zero.

The cooling coil works in the same way as the heating coil, but is mathematically more complicated because air dehumidification is calculated. For wet operation the given effectiveness is defined as $(1 - \text{bypass factor})$, according to ASHRAE's nomenclature. Physically, this means that the state of the cooled air in the psychrometric chart lies somewhere on a straight line between the state for the incoming air and the apparatus dew point temperature on the saturation curve. In the model, the average temperature of the liquid side defines the apparatus dew point. On this line, the given efficiency indicates the status: 0 is no cooling whatsoever, and 1 indicates maximum cooling, which also means that the air at

most is chilled to (the arithmetic) mean value of the liquid incoming and outgoing temperatures.

Similarly, the heat exchanger is controlled by adapting the actual effectiveness selected by the model, up to the maximum limit set by the user, so that the setpoint for the supply air temperature is reached (if enough heat is available). The temperature of the (often chilled) exhaust air, which may not fall below a certain level (TEXHOUTMIN parameter), sets another limit. This is to avoid freezing. Note that for rotating heat exchangers, it is usually possible and desirable to cool the exhaust air below freezing.

The heat exchanger takes into consideration condensation on both the supply and exhaust sides. During wet processes, the given efficiency is interpreted as (1 - "bypass factor"), in the same way as for the cooling coil, but the apparatus dew point for the heat exchanger is defined as the incoming temperature for the opposite medium.

The fans have ideal pressure control with given setpoints and by default constant efficiencies, i.e. they supply a fixed pressure head. In most cases, both these parameters only have significance for calculating fan electricity consumption. By default, the user gives directly the increase in air temperature by the fan (and the system). As an option, the temperature rise may be computed automatically with a given percentage of motor and drive losses being deposited in the air.

For modeling of CAV systems, the fan pressure rise and efficiency at the intended operating point should be entered (pressure can also be given in terms of specific fan power, SFP). For VAV systems, on the other hand, the performance should be diminished for flows below the design point. ASHRAE Standard 90 prescribes a way to do this that has been implemented as a default option in the fan model. This part load efficiency reduction is activated by entering the rated design flow.

The fans operational schedule is connected to both the supply and exhaust fans. When the control signal is zero, the fans supply a very low pressure head (for numerical reasons, greater than zero). The fan schedule is also connected to all the air terminals in the zones. When the fans switch off, all terminals are closed. This is to avoid a spontaneous flow through the system caused by chimney effect.

The fan schedule, like other schedules, gives normally values of 0 (off) and 1 (on) as output signals. However, it is meaningful for the fan schedule to sometimes give other values, thereby forcing ventilation flow. For example, if a value of 1.2 is given, all terminals will supply a 20% higher flow than that selected locally in the zone. This applies to both CAV and VAV systems. In the same way, a value of 0.5 results in half the flow.

The local terminal connection to the central schedule has no counterpart in actual systems, but has been introduced partly to avoid unintentional spontaneous flow in the system, and partly to permit forcing the flow in case of CAV.

The heat exchanger is similarly controlled by a schedule. To turn it off, the schedule should give the value 0. A value 1 turns on the heat exchanger. No other values are meaningful.

3.3. The zone models

For the HVAC systems, the presentations for the standard and advanced user interface levels are the same (for the distinction between standard and advanced, see Section 5). However, in the case of the zone models, the presentations are completely different. This section explains the principal features of the zone models, and also provides a brief overview of the advanced level in this context. Actually, most users will never have to deal with the advanced level, but it is useful to know it exists, and the physics is more easily explained from the point of view of the advanced level.

IDA Indoor Climate and Energy provides two different zone models. One of these, the climate model, is very detailed - it calculates for example a vertical temperature gradient. The second model, the energy model, has a more conventional level of precision, and is based on a mean radiant temperature. Both zone models are based on the same description of the building, given in the standard level. All models of components in and around the zone, such as windows, radiators, controllers, leaks, terminals etc., are common to both the energy and climate models. The climate model is currently available only for zones with a rectangular geometry. From version 4, the energy model is default for new zones, but this can easily be changed in the Defaults form. More details about the mathematical models can be found in *Models for Building Indoor Climate and Energy Simulation*¹ or by studying the NMF code in the Code tab of the component window at the advanced level.

Figure 3.3 shows a schematic view (advanced level) of a zone with the climate model. To access this window, press Build model in the Simulation tab, after entering all the information required in the standard level. In this case this includes selection of climate model, an ideal heater, a cooling panel and external shading. After the advanced level model has been generated, the user is able to select between General (standard level) and Schematic (advanced level) tabs for zones as well as for the building. The various component groups are numbered in the figure as follows:

1. Supply and exhaust air terminals
2. Ceiling/Floor²
3. Air leak to ambient
4. Solar irradiation and external film coefficient, external wall
5. One external and three internal walls
6. Window and shading calculation components
7. Proportional controller for occupant automatic clothing adaption
8. PI controller for ideal heater
9. Cooling panel with controller and ceiling section behind
10. The actual zone model in which radiation, convection and loads etc. are modeled
11. Post processing components for results capture.

¹ which can be downloaded from the User's page: Help menu, IDA on the Web, ICE User Support

² This object is both ceiling and floor, since the model was generated to represent zones above and below with identical conditions as the current zone.

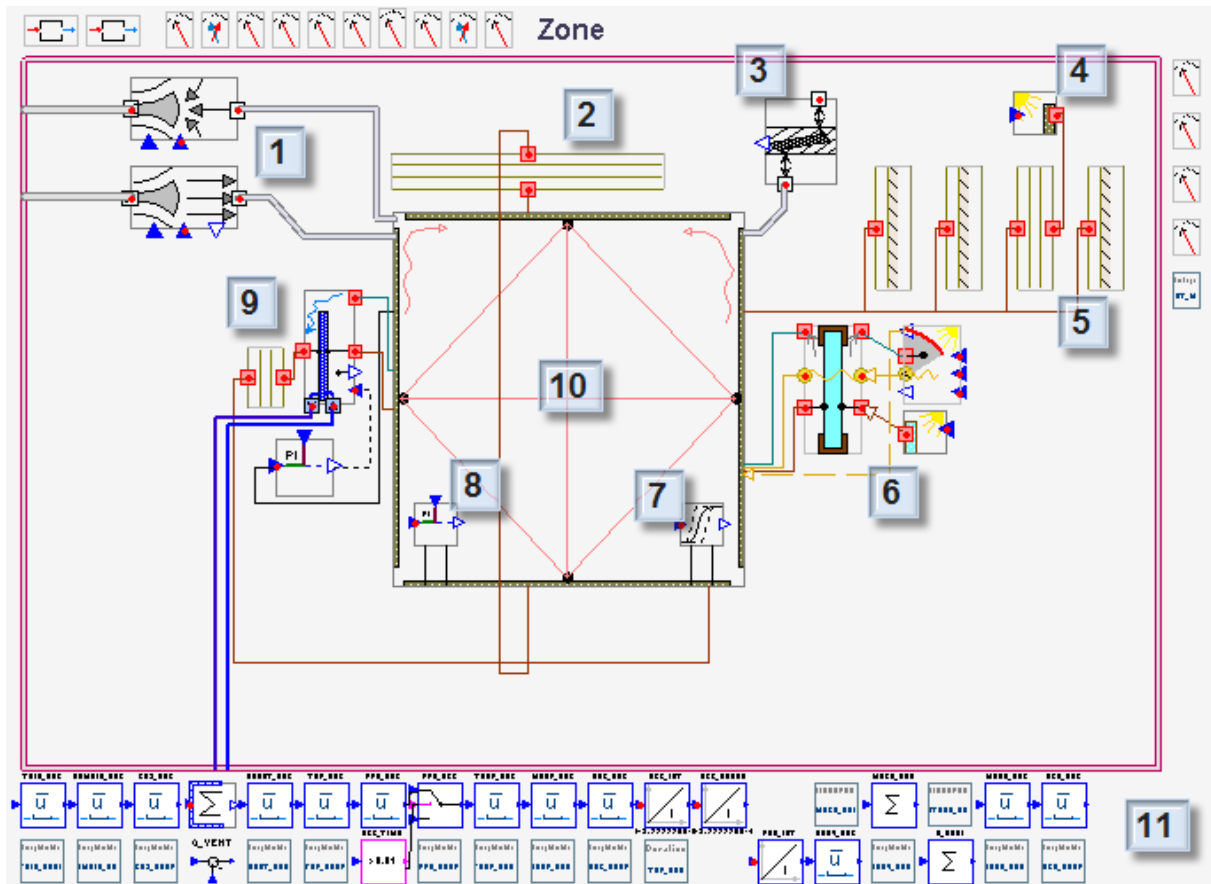


Figure 3.3. Schematic view (advanced level) of Climate zone model.

3.4. Solar radiation modeling

A key issue in building simulation is the treatment of direct and diffuse solar radiation. Let us follow the main steps in the treatment of sunlight entering the building. If synthetic weather is used, ICE first computes direct and diffuse sunlight intensities based on the clearness number given in the Location object. At the advanced level, the components used for this computation are found in the Schematic tab of the building form under the heading Climate processor.

In the next model in the chain, the solar position in the sky is computed and all the signals computed so far are sent to a set of Face models, where the climatic conditions outside each main building surface (Facade) are computed. In this model, the distribution of diffuse radiation in the sky is also computed, by default using the Perez model.

In the next step, solar radiation on an individual object such as a window is computed. Connected to each window model (See Figure 3.3) is a shading calculation model (Shade), that computes the shading of both direct and diffuse light on the receiving surface. This model puts all shading surfaces in one 'bin', including building self shading, shading by neighboring buildings and shading by (possibly movable) objects directly outside of the window (External shading). Diffuse light from the sky is also shaded, but no reflections other than ground reflection are accounted for. Ground reflection for each face can be set from the Property page if the face name is selected in the Floor plan tab. Diffuse radiation from the ground is not shaded by external objects. All external shades are considered to be opaque.

The shade model is very difficult to interact with directly at the advanced level, since each surface has been subjected to several coordinate transformations. The actual shading factors are precomputed for all (plausible) solar locations and are stored as parameters in the shade model for the simulation.

Once in the window, diffuse and direct light are reflected and transmitted depending on the window model used. The standard window model uses a fixed curve for the angle dependence. Integrated window shading (internal or external shades in the plane of the window) will reduce radiation by multiplying the basic window parameters.

The detailed window model (Expert Edition), makes a layer by layer computation of multiple reflections and each layer temperature is computed.

Once inside the zone, diffuse light is spread diffusely, while the exact target location of the direct light beam is computed. However, the whole surface of the window is considered as the light source, not just the portion of the glass which is actually not shaded by external objects. After the first reflection on a zone surface, the direct beam is spread diffusely in the room. And also here, the whole surface that is hit is regarded to reflect with equal intensity, not just the lit portion of this surface.

Internal windows and open doors transmit light in a similar way as an external window, i.e. the whole opening is regarded as being lit, even if only a small part of the door receives direct sunlight. The light intensity is of course adjusted accordingly. Similarly, light that enters through one window and exits through another external window, for example in a corner room, is treated in a physically reasonable way.

3.5. Airflows

This section covers the models and input which determine the airflows through the building. IDA Indoor Climate and Energy enables the user to take account of natural ventilation, i.e. flows driven by wind pressure and the stack (chimney) effect.

In the simplest case, each zone has three paths for airflows: through the supply and exhaust terminals and via leakage through the envelope. When two zones are placed adjacent to each other and there is an opening in the common wall between them or when windows are open or when additional leaks have been added, additional flow paths are created. However, let us start with the simplest case.

The mechanical ventilation terminals are always of VAV (Variable Air Volume) type. This means that as long as there is sufficient pressure head from the fans, a given flow is maintained as requested by the zone itself. In CAV (Constant Air Volume), the control signal is kept constant and a constant flow will be maintained, irrespective of pressure. In a VAV system, a controller regulates the flow with respect to temperature, carbon dioxide, humidity or pressure levels in the zone.

Since, in the simplest case, flows through two of the three paths are given, the size of the third flow, through the leak, is important only for the pressure in the zone, and not for net flows. The pressure in the zone, in turn, is important only if there is a connection with other zones, through other leaks or open doors. In these situations, net flows are determined by zone pressures and relative opening sizes.

With version 4, it is also possible to define additional given in/exfiltration flows, i.e. balanced flows into and out of a zone. Since these incoming and outgoing (mass)flows are always equal, they will not affect the pressure of a zone. The given in/exfiltration will only act to exchange heat, moisture and CO₂ with the ambient.

3.5.1. Air flow input forms

Figure 3.4 shows the zone form. The input fields in the Ventilation section govern air flows. Also involved are Controller setpoints, clicking on the link field opens the dialog (see Figure 3.5).

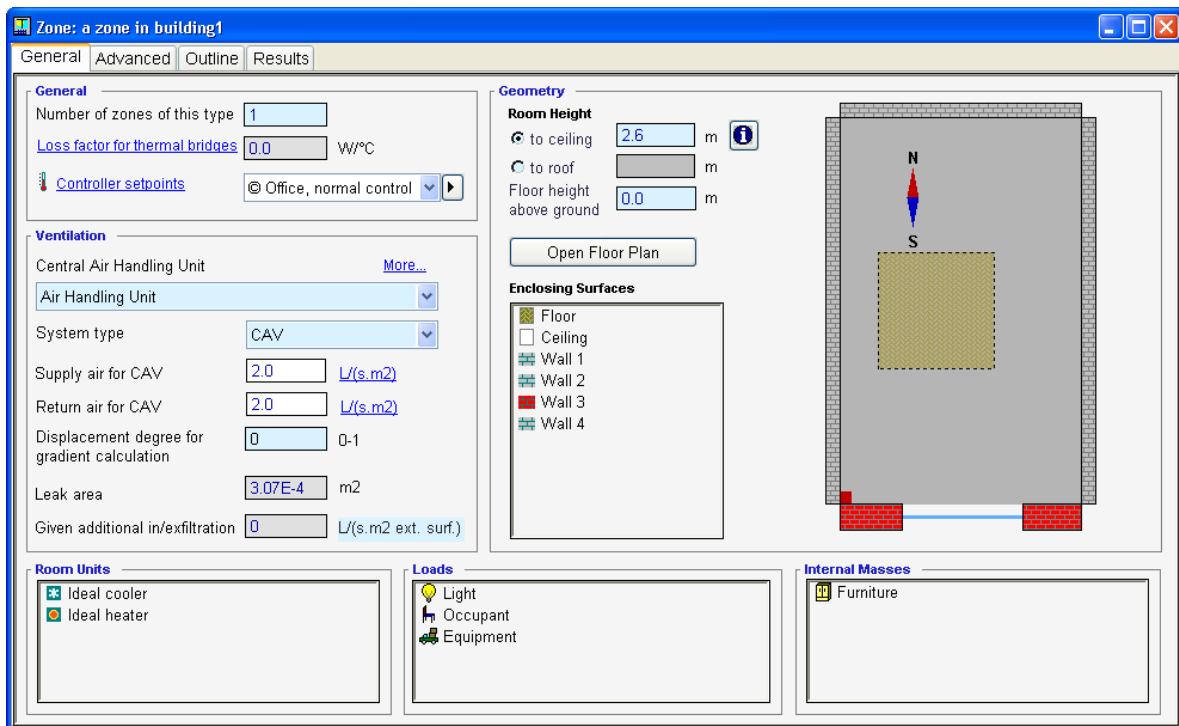


Figure 3.4. The zone form.

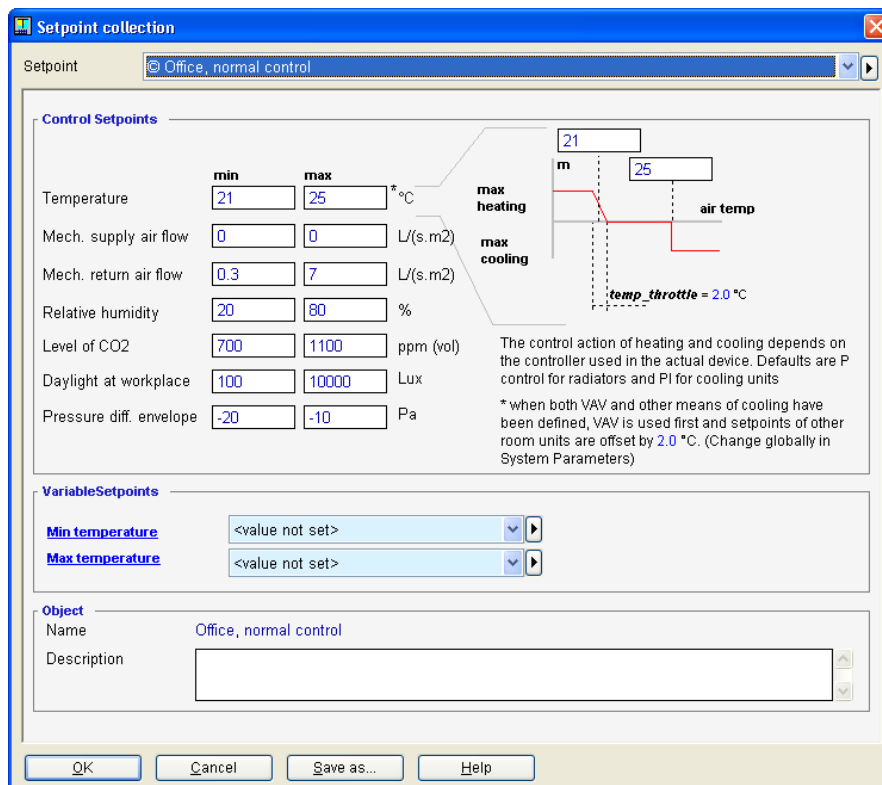


Figure 3.5. Control setpoints.

In the zone form, the user can select System type (CAV, VAV with temperature control, VAV with CO₂ control or VAV with humidity control, VAV with both temperature and CO₂ control, and VAV with pressure control). For CAV, the required airflows are given directly in the zone form³. Leak area gives the size of the combined envelope leak⁴. Leaks are inserted in all external walls, 1 m above floor level. (The height is only relevant for natural ventilation situations.) Both Leak area and Given additional in/exfiltration are by default computed automatically based on information for the whole building that is given in the Infiltration form which is reached from the Building form.

In the combo box Controller setpoints (Figure 3.4), the user can select an object with a collection of relevant zone-level control setpoints. The Control setpoints dialog (Figure 3.5) provides all input data for zone climate quality requirements. Here, only those parameters, which are of importance for airflows, are discussed. For a VAV system, the given minimum value of Mechanical exhaust airflow provides the lowest allowable airflow, with the maximum value providing the highest one. (In the example above, an exhaust only VAV system has been specified.) In the case of CAV, as was already pointed out, the desired flow is given directly in the zone form. If the chosen CAV flow falls out of the quality range provided in Control setpoints, a warning is issued when the simulation is started.

The other values in Figure 3.5 impact only upon the corresponding VAV control scheme. If the user selects CO₂ control, (see Figure 3.4), the airflow is varied in proportion to the CO₂ content of the air in the zone. A CO₂ value, which equals or exceeds the given max-value (in Figure 3.5), results in the maximum flow through the exhaust terminal, with a minimum CO₂

³ The link "l/s m²" next to the input field enables input in various units.

⁴ Equivalent Leakage Area (ELA) at 4 Pa and Cd=1. ASHRAE Fundamentals provides a number of values regarding leaks according to this definition, as well as conversion methods between various representations of leak sizes.

value producing the given minimum airflow. The humidity control function is entirely analogous with respect to relative humidity (assuming supply air will dry the zone).

Pure temperature controlled VAV systems function somewhat differently. Here, the maximum comfort temperature value is used. Forcing of the VAV flow begins somewhat below (normally 1 °C) the indicated maximum value (P control). Full exhaust flow (see Figure 3.5) is reached at somewhat above (normally 1 °C) the maximum temperature value. The throttling range is normally 2 °C, but can be selected at the building level under System parameters.

The pure temperature controlled VAV scheme will only use VAV action for cooling zone air, i.e. if there is a need for heat and the supply air is warmer than the zone, this will not be recognized by the controller.

Combined temperature and CO₂ control, on the other hand, will be “smart” enough to both heat and cool with the supply air. It relies on PI controllers instead of P ditto, and will therefore not have any offset error. In addition, it will also force air flow if needed to maintain CO₂ level below the maximum limit.

Pressure controlled VAV is normally used for return air flow control where some other VAV method is used to supply air into adjacent zones. It will attempt to maintain zone pressure, within the given range, with respect to ambient pressure using a proportional controller (measured as pressure drop in the local ambient leak). In the example in Figure 3.5, the zone is maintained between 10 and 20 Pa below ambient pressure.

3.6. General information regarding zone cooling and heating room units

Local heating or cooling is supplied to the zone by room units. All room units are listed in the zone form. Some units such as ideal heaters and coolers, do not have a given location in the room. These can be introduced directly into the list in the zone form. Most hydronic units must, on the other hand, be located on a specific zone surface and they are instead inserted by dragging them onto a surface.

The temperature setpoint for cooling devices is normally taken directly from the Control setpoints, max value for Temperature⁵ (see Figure 3.5). The corresponding value for heaters is the Temperature min value.

3.7. Ideal heaters and coolers

The ideal room units should be used to condition the zone when no detailed information about an actual room unit, such as a fan coil or active chilled beam, is available or this amount of detail is unmotivated. They have no given physical location on any room surface and are not connected to the plant of the building.

⁵ An exception is made when the air system is pure temperature controlled VAV. Waterborne cooling units in this case have their setpoints displaced by usually 2 °C. (Can be given centrally in System parameters). This means that the airflow is first forced to max value, after which the waterborne cooling unit sets to work.

3.8. Hydronic heating devices

Heat emission from hydronic heating devices is calculated using

$$P = K \cdot l \cdot dT^N,$$

where l is device length and dT is the temperature difference between the water and the zone air. K and N are constants characterizing a device of a certain height (or width for ceiling devices).

Figure 3.6 has a radiator inserted on a wall surface. Its main form has been opened and a dialog for alternative input has been opened from the form. Often the values of K , N and $Height$ come from the database (gray colored text in the form). The user needs only designate surface area (given graphically) and the design water flow.

The warm radiator surface that is exposed to the zone is defined by the box that is drawn when the unit is inserted on a wall surface. The size of this box has meaning for the heat transfer. The total emitted heat is always given by the expression for P . However, the division between radiation and convection is calculated based on the surface temperature of the device, which is connected to the water temperature, and the given exposed surface. The zone “sees” a warm surface and calculates the radiation and convection from this surface. The main part of the remaining heat (needed to complete P) is emitted convectively directly to the air (corresponding to convection behind a radiator). A small part of the heat goes to increase the temperature on the portion of wall behind the heating device. The heat transfer coefficient between the device surface and the wall behind is considered in the basic case to be completely dominated by radiation and is calculated by the model.

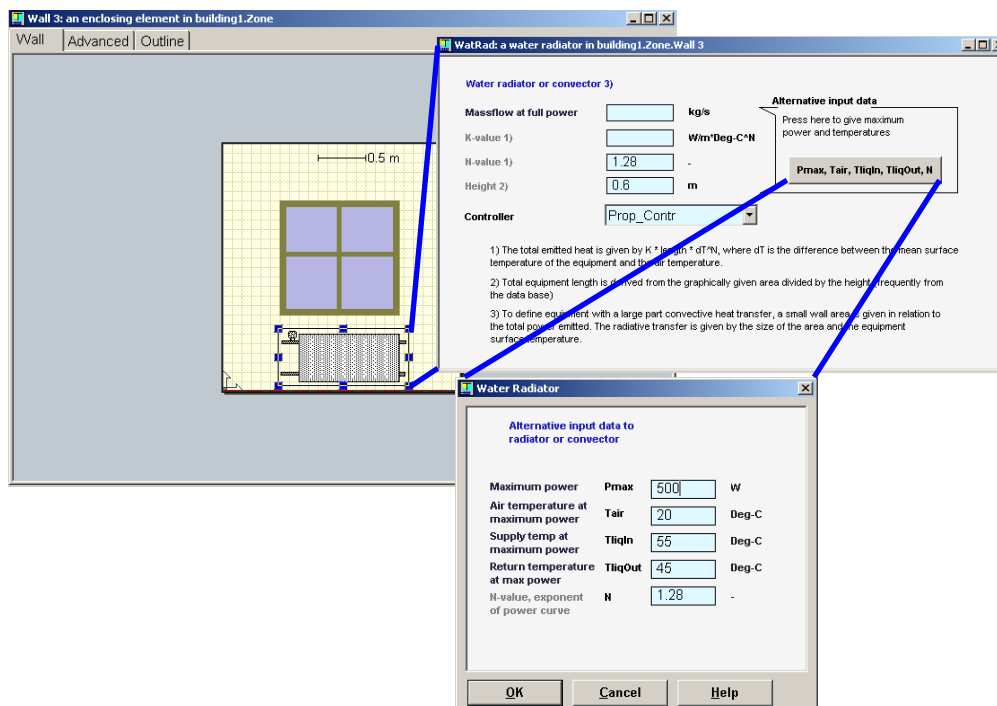
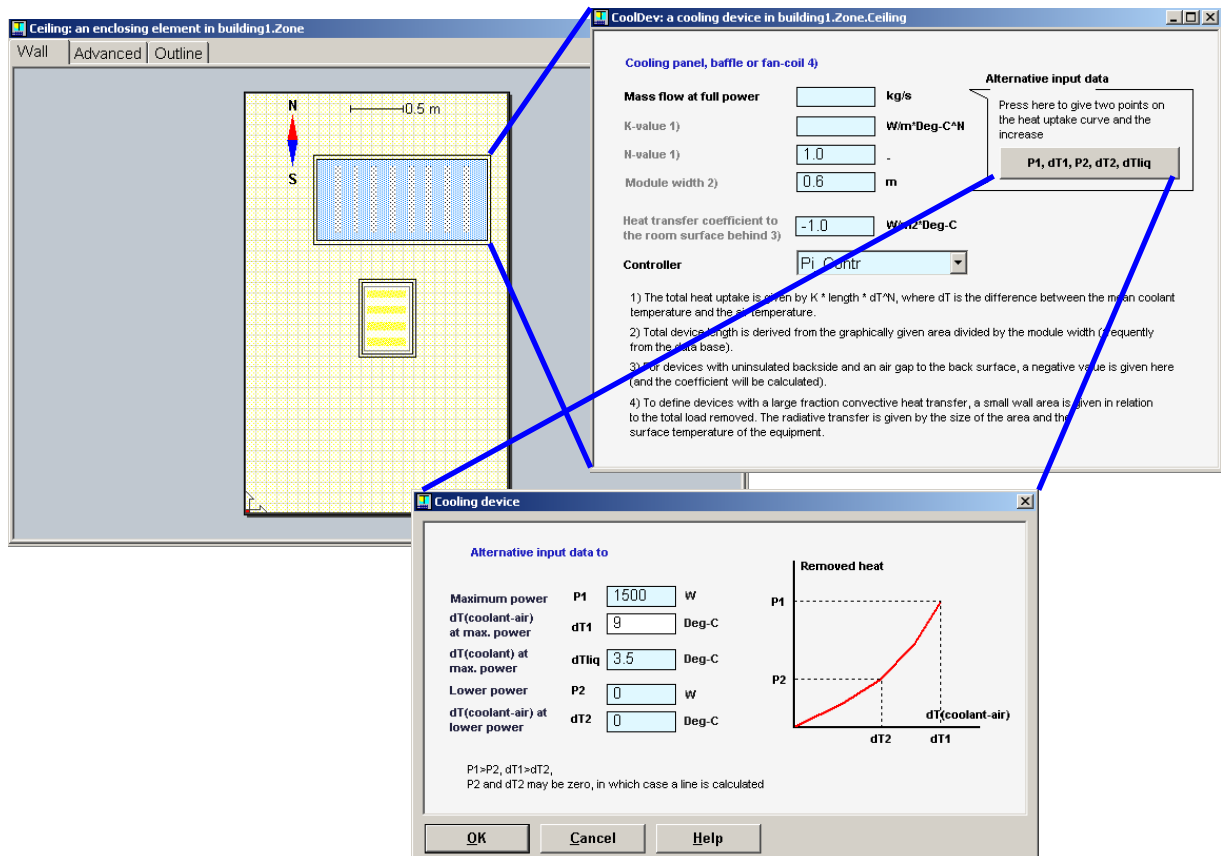


Figure 3.6. A radiator on an external wall, its standard form, and a dialog for alternative input.



To facilitate adding new heating devices without direct knowledge of K and N , a possibility for alternative input is given (see Figure 3.6, lower box). The values for design mass flow and K are calculated from the power given by the user at the specified temperature conditions. The user also provides a value for N in this case. Note that K is calculated from the information given in the alternative input. This is thereafter the value for K . If the size of the graphical box is later changed, the device is likely to have an unintended maximum power. Note also that the actual maximum heating capacity of the device will vary with actual room and supply water temperatures. The temperature conditions given in Alternative data are only used to calculate K and have no impact on the simulated supply and return water temperatures⁶.

3.9. Cooling units

The Cooling device is used for radiative and convective units. Cooling units operate completely analogous to waterborne radiators, with the exception of a few differences in input, which are discussed here.

Figure 3.7. A cooling panel on the ceiling surface, its standard form, and a dialog for alternative input.

Here, the height of a radiator corresponds instead to a *Module width*, to which K and N refer. The total length is calculated as the given box area divided with *Module width*⁷. There is a difference in that the heat transfer coefficient between the back of the device, and the surface

⁶ The actual water supply temperature is given by the boiler; the return temperature will vary with the actual massflow through the unit, the current room temperature and the size (K value) of the heating unit. However, if actual temperature conditions are made to coincide with those given in the dialog, exactly the given power will be emitted.

⁷ The same is done for the radiator.

behind (often the ceiling) is given directly in the main form. If an (arbitrary) negative figure is entered, the heat transfer coefficient is calculated in the same way as for the heating device, i.e. as if all heat transfer was done by radiation. This is a good approximation for a device that has no insulation at all.

The dialog for alternative input has somewhat different parameters for cooling units. Absorbed power and temperature differences between air and water may be given for two points on the power curve. For max power, the temperature drop of the water is also given.

3.9.1. Active beams

Active beams serve both as supply air terminals and as cooling devices with significant convection. Their performance depends on the amount of supply air that is passed through but they normally retain a heat transfer contact with room air also in the case of zero supply air flow. The radiative coupling with the room is neglected in the present model. Beams are mostly used for cooling but may also heat the room air.

Two input data options are available: Simplified and Manufacturer's. The latter means that the performance parameters K and N are given as functions of air flow. This alternative is mostly used when data is automatically imported from an on-line manufacturer's database or from IDA Room. The Simplified option is based on two user supplied performance points, at design conditions and at zero flow (Figure 3.8). N is for this case set to 1.5.

The screenshot shows a software dialog box titled "Beam: a device in building1.Zone.Ceiling". It has two tabs: "General" and "Outline". The "General" tab is active, showing "Simplified input data to chilled/heated beam".

There are two radio buttons: "Use manufacturer's data" (unselected) and "Simplified model:" (selected). To the right of these buttons is explanatory text: "When using the detailed model, the power at the given condition is calculated from the manufacturer's data and shown in this form. To edit manufacturer's data, switch to the outline tab. In the simplified model, the power parameters are user-supplied. No manufacturer-supplied data are used."

Under "Simplified model:", there are two columns: "Cooling" and "Heating".

- "Power at design air flow": Cooling (0.0 W), Heating (0.0 W)
- "Power at zero air flow": Cooling (0.0 W), Heating (0.0 W)
- "Design air flow": 0.0 L/s
- "Design conditions":
 - "dT(coolant - zone air) at max power": Cooling (8.5 Deg-C), Heating (20 Deg-C)
 - "dT(coolant) at max power": Cooling (3 Deg-C), Heating (10 Deg-C)
- "Controller": PI (dropdown menu)
- "Target setpoint": Air temperature (dropdown menu)

Figure 3.8 Active beam form

In the constant flow (CAV) case, the given Design air flow is regarded to pass through the beam when fans are running. When fans are forced, the beam flow is correspondingly increased. For VAV, the Design air flow will pass through the beam at full VAV flow. At lower flows, the flow through the beam will be proportionately decreased.

The total design air flow through all beams must not exceed the total for the zone. If beam air flow is less than the requested total for the zone, the remaining part is regarded as being supplied through conventional terminals.

3.9.2. Heating/Cooling floor (Expert edition)

If a floor heating/cooling object is inserted on the floor of a zone, the floor construction for this area is divided into two parts, above and below the heated layer. Between the two, a heat

exchanger model is inserted corresponding to the piping layer. Quite often a floor heating circuit will heat the room below almost as much as the room it belongs to.

The floor coil model assumes that the active layer can be treated as an infinitely conductive plane in the floor slab, i.e. all 2D effects are disregarded. Heat transfer is calculated with a logarithmic temperature difference between the fluid and this plane of constant temperature. The user supplied total heat transfer coefficient between the fluid and the plane, includes

1. Convection between medium and tube wall
2. Heat conduction through the tube walls
3. “Fin efficiency” corresponding to the distance between immersed tubes or actual fins.

The modeling approach will in steady state correspond to the Resistance method of the standard EN 15377-1.

The floor coil circuit can have its own three-way valve and pump circuit keeping the massflow constant (default). Emitted power is then controlled by varying the supply water temperature. PI, P or on-off control can be selected (PI is default). A further alternative Always on is also available, which will keep both the boiler mass flow and the coil circulation massflow permanently at their design values. Control must then be maintained by the boiler temperature controller.

In the case where no separate coil pump circuit is used, the four control options will instead act by limiting the massflow through the coil, or keeping it constant at design conditions in the Always on case.

hc-floor: object in building1.Zone.Floor

Floor heating and cooling (temperature control)

	Cooling	Heating	Unit
Design power	40.0	40.0	W/m ²
Design Temperature difference	5.0	5.0	Deg-C
Controller	PI		
Target setpoint	Air temperature		

The design power and temperature differences are used to calculate supply massflows. The actual emitted power may become smaller if the heat resistance in the floor construction is too large.

Coil massflow

Given by supply massflows (no separate circulation pump)
 Constant flow, giving at most a coil temp. difference Deg-C

In the constant coil flow case, a circulation pump provides a constant coil massflow for both cooling and heating. (Pump electricity is not accounted for).

Location in floor slab

Depth under floor surface m

The floor heat coil becomes a tempered layer at the given depth in the floor construction, which is defined elsewhere for the whole floor.

Heat transfer coefficient

H-water-pipe-fin* W/(m² K)

Normally set to 6 for aluminium fins in a wooden construction and to 30 for pipes immersed in concrete. However, the total heat transfer is normally largely determined by the resistance in the floor construction above and below the coil, in which case this parameter becomes less important.

* In the nomenclature of the EN 15377-1 standard:
 $1/H\text{-water-pipe-fin} = R_w + R_r + R_x = R_t - R_z$

Figure 3.9 Floor heating form

4. CAD and image import

In IDA Indoor Climate and Energy it is possible to import CAD objects and image files. These can be used as a base when creating the simulation model, or as shading elements that cast shadows onto the simulation model. There are three categories of CAD objects and image files; building information models (BIM), CAD and vector graphic files, and image files.

BIM files contain 3D geometry as well as properties for walls, windows and materials etc. Building sections, zones and windows etc. can be automatically created from the geometrical information and the properties of objects in the BIM file can be mapped to the corresponding objects in the simulation model. The 3D geometry of a BIM file can also be selected to shade the simulation model.

CAD and vector graphic files contain 3D or 2D geometry. A section of this geometry is shown as lines in the floor plan tab and these lines can be used to snap building sections and zones etc. in the floor plan. 3D CAD objects can be selected to shade the simulation model.

Image files contain raster images. These are shown in the floor plan tab when the section is close to the location of the image. The images can be used as a background when drawing building sections and zones etc. in the floor plan.

4.1. Supported file formats

4.1.1. BIM

Industry Foundation Classes (*.ifc)

4.1.2. CAD and vector graphic files

AutoCAD (*.dwg⁸, *.dxf, *.dwf)

3D Studio (*.3ds)

Computer Graphics Metafile (*.cgm)

Corel Presentation Exchange (*.cmx)

MicroStation DGN (*.dgn)

Micrografx DRW (*.drw)

Gerber File Format (*.gbr)

Scalable Vector Graphics (*.svg)

Printer Command Language (*.pcl, *.prn, *.prt)

Macintosh PICT (*.pct)

HP-GL/HP-GL2 (*.plt)

WordPerfect Graphics (*.wpg, *.vwpg)

4.1.3. Image files

Bitmap (*.bmp)

JPEG Interchange Format (*.jpeg, *.jpg)

Portable Networks Graphics (*.png)

ZSoft PC Paint (*.pcx)

⁸ DWG files are assumed to be two-dimensional, i.e. any 3D geometry is flattened to 2D at import.

Tagged Image File Format (*.tiff,*.tif)

Adobe Photoshop (*.psd)

Truevision (*.tga)

Windows Meta File (*.emf,*.wmf)

4.2. Importing CAD objects and images

CAD objects and image files are either imported with respect to the building coordinate system, and are then moved with the building if the building is repositioned or rotated, or they are imported with respect to the site coordinate system, and remain fixed if the building is repositioned or rotated.

Import a CAD object/image file with respect to the building coordinate system by clicking the Import CAD button on the floor plan tab. Alternatively, select Import CAD on the Insert menu while the 3D tab is shown.

Import a CAD object/image file with respect to the site coordinate system by clicking the Import site CAD button on the Site object dialog opened by clicking Site shading and orientation on the General tab. Alternatively, select Import CAD to site on the Insert menu while the 3D tab is shown.

To place a CAD object/image at the current mouse pointer in the 3D view, use Right mouse button menu > Import CAD or Right mouse button menu > Import site CAD.

4.3. Moving and scaling CAD objects and images

A CAD object is automatically scaled and positioned so that it corresponds to the simulation model. The scale and position of a CAD object can be seen and edited by double-clicking on the object. A section of the CAD object or image is shown in the floor plan tab if the floor plan level is within the bounds of the object. Select this section by clicking on it⁹. Move and change size of the CAD object/image by dragging/resizing the section.

4.4. Shading by imported 3D objects

3D CAD objects can be selected to shade the simulation model. Check the Calculate shadows checkbox in the dialog shown when the object is double-clicked. Every non-transparent surface of the 3D object is included in the shadow calculation and visualization.

4.5. Storage of CAD objects

By default the geometrical information of CAD objects is saved in the system file (*.idm). The original CAD file is then not needed after the import. However, if a CAD file is big, very large system files can be created and the performance of IDA ICE can be slowed down. Thus, if a CAD file is big, the option of not saving it in the system file is given in the Preferences dialog, which is opened at import. In that case, only a shortcut to the original CAD file is saved in the system file and the original CAD file needs to be saved in the location specified by the shortcut. If the CAD file is placed in the ICE system folder (the folder with the same name as the system), the shortcut is relative and the CAD file is automatically copied with the system to a new location. Otherwise the shortcut is absolute and the CAD file is not copied with the system. The definition of a big CAD file can be changed in the Preferences dialog

⁹ To select a CAD object, click on the geometry lines. To select an object that is behind another object, press the ctrl-key and click on the object until it is selected.

from the default of 10000 vertices. The Preferences dialog can also be opened from the Options menu.

4.6. Importing IFC files

In ICE it is possible to import 3D building information models (BIM) via IFC files. Most 3D CAD applications can export architectural data in IFC format.

The most important information that is transferred is geometrical data, i.e. the shape and position of zones, windows, doors, building faces etc. Zones in ICE are automatically created from so called space objects in the IFC model. It is not sufficient that the CAD model only contains wall objects, spaces that fill the voids between walls must also have been created, a semi-automatic process in most CAD tools.

ICE can also utilize other types of information in the CAD model, such as wall constructions, should they be present. One can find more detailed information about the ICE IFC implementation on the user's page: Help menu, IDA on the Web, ICE User support. Select Topic notes on the page.

4.7. Mapping data from IFC

Start with a building without zones and select the Floorplan tab. Press IFC... to select an IFC file for loading. There are some sample IFC files in the installation, normally located in C:\Program Files\IDA\samples\ICE\IFC.

The first task is to map named data objects in the IFC model (if any are present) to corresponding IDA resources. Press IFC Mapping in the Floorplan tab to open the Mapping dialog (Figure 4.1). If wall constructions have *not* been described in detail in the CAD model, select directly Wall Constructions in the Category combo box. This will present a list of all wall types that have been found in the IFC model. Since ICE needs more detailed information about a wall, IFC wall types need to be manually associated with ICE wall constructions. To bind a certain IFC wall type to an ICE construction, select both the IFC wall type and the corresponding ICE resource and press Map to selected. A right pointing arrow in the IFC Data list indicates that the item has been bound. Usually one first has to load relevant ICE resources from the database by pressing Load from Db. To inspect the selected IDA resource, press View.

Repeat the procedure for window types as well. Here, one usually has to first create relevant windows in the ICE database, including internal shadings etc.

If wall constructions have been described in the IFC model with layer thicknesses and material names, one can automatically create corresponding ICE constructions. In this case, one starts instead with binding IFC material names with ICE material resources. Once the materials have been mapped, ICE wall construction resources are created by pressing Import from IFC when the relevant IFC wall type has been selected.

Any object in the IFC model which is not explicitly mapped to an IDA resource will be set to its default value, which is given on the Defaults tab in the building form.

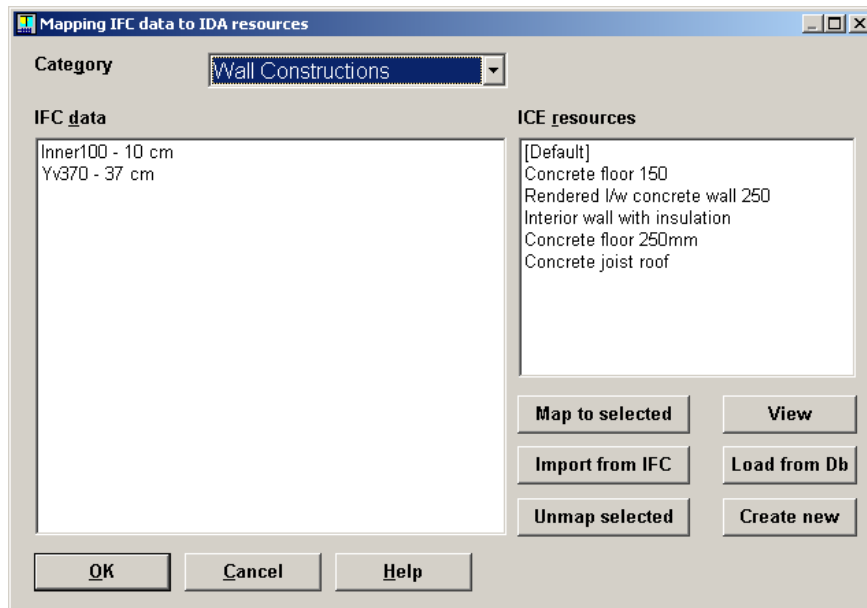


Figure 4.1 IFC Mapping dialog

4.8. Create zones

An IFC model may contain more than a single floor. A horizontal section (slice) of the building at a certain level is shown in the Floorplan tab (Figure 4.2). To select a different level, press the button Level: *xx* m, where *xx* is the floor height from ground of the current level. In the Level dialog, the building height from ground (Building top) and height coordinate of the floor slab with respect to ground (Building bottom) are also shown, as interpreted from the IFC file. These numbers are not always correct for the user's purpose.

To define which spaces in the IFC model that should constitute a (thermal) zone in the simulation model, click on (select) the neighboring spaces that should be included. (Click again to unselect a space.) Think about zone economy, i.e. do not create more zones than you think is physically motivated for the current study. To create an ICE zone from the selected IFC spaces, press New zone. Try to give as many reasonable defaults as possible for new zones; after creation these values must be edited separately for each created zone.

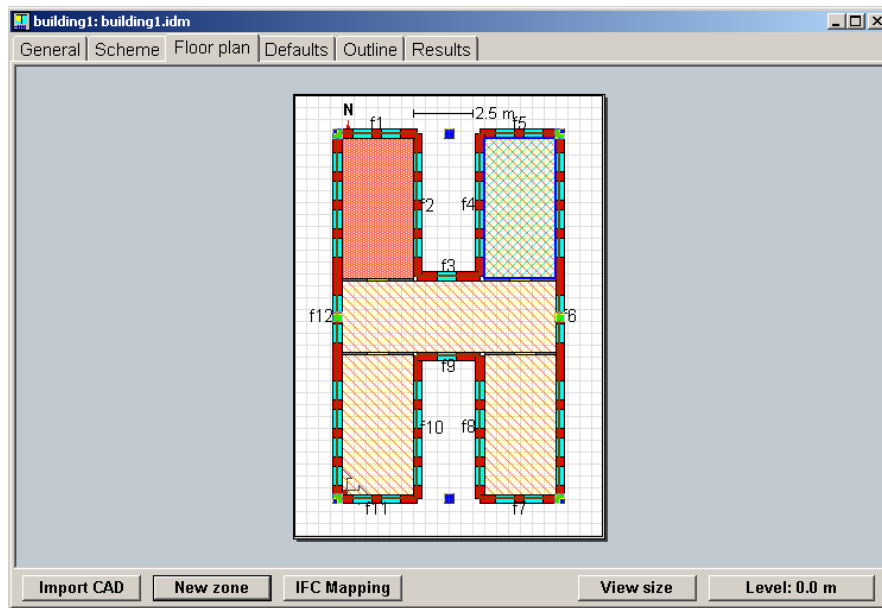


Figure 4.2 IFC model with an ICE zone, a selected IFC space and an unselected IFC space.

The created ICE zone get the geometry of the corresponding IFC space(es). If for example a space is taller than the floor to floor distance of the building, the corresponding ICE zone will also reach over more than a single floor. One can change the horizontal section level during the zone creation process but it is currently not possible to combine several spaces vertically into a single zone. ICE is limited to zones with flat floor and ceiling. To create more complex geometries one has to define several zones, possibly connected by large openings. Note that the Floorplan view displays two models simultaneously, the zones of the created ICE model and the spaces of the IFC model. Both categories of “rooms” can be individually selected and ICE zones can also be opened.

If the IFC model is revised during the ICE modeling project, there is some support for retaining previous work. New IFC models can be loaded while modeling, either replacing the existing model or adding to it, e.g. loading several floors that are in separate IFC files. When a new file is loaded, the user is given the option to replace or add to the current IFC model and to replace or keep mapping information and existing ICE zones.

One should note that IFC is still a standard under very active development and the quality of implementations will vary as will the content of “typical” IFC files. Today, few tools rely on having access to correct 3D information so quite often the 3D content of project drawings will not be quality controlled. However, the IFC standard has beyond doubt reached a stage of usefulness and it is likely that the number of quality implementations of IFC tools will increase rapidly in the years to come.

5. Getting started with the advanced level

The standard level interface covers the most common simulation tasks, but sometimes it is useful to examine other variables than those available here or to replace a component model. To accomplish this one turns to the advanced level interface, where the system is described in a mathematical sense with components containing equations, variables and parameters.

Access to the advanced level is provided for both the Standard and Expert editions of IDA ICE. However, Expert Edition users can manually edit (reconnect) component diagrams at the advanced level, while Standard Edition users can only inspect, log variables and change parameters. Some of the examples below require the Expert Edition, but this will then be mentioned in the introduction.

Work at the advanced level is best presented in terms of demonstration, so the written account here is quite brief. Look also at the User's web page for more information on work at the advanced level, demonstration movies etc.

In some cases the system structures at the standard and advanced levels match each other quite well. The air handling unit, for instance, has different components: fans, coils etc. The same description can then be used for both the standard and advanced levels. The same is true for the primary system. For the actual building description, however, entirely different descriptions are used for the standard and advanced levels.

Most components at the advanced level are described with equations. Components are interconnected by creating equalities between variables that appear on interfaces. An example is the interconnection between the cooling and heating coils in the air handling unit. Both the outflow interface of the heating coil and the inflow interface (terminal) of the cooling coil contain variables for: pressure, temperature, massflow, moisture and carbon dioxide concentrations.

To work on the advanced level, select Build model Simulation tab. This creates the schematic view of the current building. If the model has been built previously, Schematic is already available as a view of the system. Figure 5.1 shows the appearance of the air handling unit in the schematic view (which is also the standard view for the air handling system).

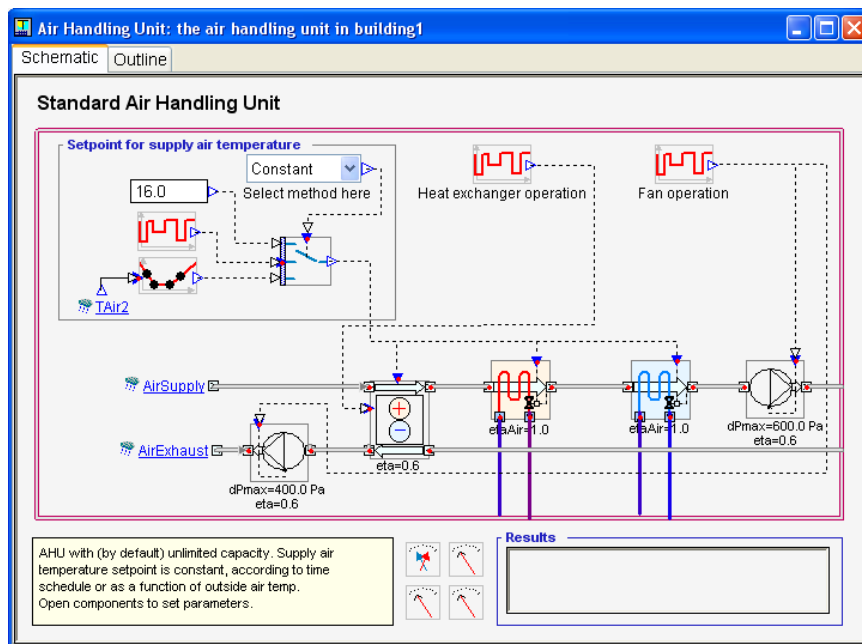


Figure 5.1 The air handling unit.

The behavior of a component is described by equations, variables and parameters. The difference between parameters and variables is that the former will never change their value during a simulation, whereas a variable might. A window area is an example of a "natural" parameter and the room air temperature is always a variable.

The description of most components is done using a special language called NMF (Neutral Model Format). Click on the Code tab of any of the AHU components, to see the NMF code. It contains the following main sections:

Abstract	A brief textual description of the model
Equations	The actual mathematical description (formalized according to a strict syntax but quite readable also for humans)
Links/Interfaces	A description of the ports or terminals of the component. A fan would typically have at least two links for incoming and outgoing air flow and could also have links for power supply and control signals.
Variables	Variables to be calculated by the model.
Parameters	Quantities that characterize the component, e.g. a vector of numbers describing the fan curve.
Parameter processing	Computer code which converts user supplied parameters into those that actually appear in the equations

From version 4, some models in the ICE library are instead described by the Modelica language (www.modelica.org)¹⁰.

We will now go through a few examples of useful operations at the advanced level of ICE.

¹⁰ The IDA Modelica development environment is not yet publicly released and focus here will be on NMF. The NMF development environment is shipped with the Expert Edition of IDA ICE 4.0.

5.1. Example 1: Presenting more data in an existing diagram

Suppose we are interested in the air temperature after the heat exchanger in the air handling unit and would like to see the graph together with the other temperatures of the AHU. Open the AHU window (Figure 5.1) and double click on the connection between the heat exchanger and the heating coil. A small window is opened showing the connection between the two interfaces; SUPOUT (SUPPLY OUT) of the heat exchanger (hx) is connected to the AIRFLOWIN interface of the heating coil (hc). To see the actual variables of the connection, double click on the box hx.SUPOUT. TSUPOUT is the variable we are looking for, double click on it to get the form at the lower right of Figure 5.2.

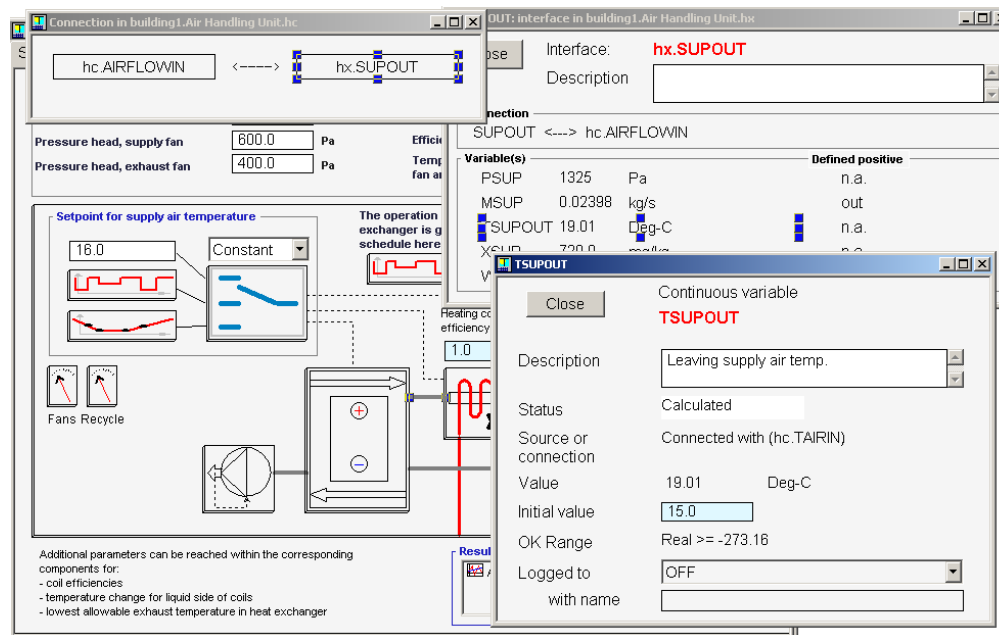


Figure 5.2

All relevant information about the variable is displayed in this window. At the bottom of the window, there is a combo box for logging the variable to a diagram. Select “Outdoor and HVAC temperatures” in the combo box. Make a simulation, and inspect the added graph in the diagram.

5.2. Example 2: Shade control by zone temperature (Expert edition required)

Normally, window integrated shading is controlled by the amount of radiation that penetrates the glazing. However, in some applications it can be useful to let the zone air temperature determine whether shades should be drawn instead. From version 4, it is possible to define customized controls for shading (and window opening) at the standard level.

We look at a simple case consisting of a single zone with a window in one wall and intend to have an external blind shading the window depending on the zone air temperature and governed by a thermostat. First insert an external blind. Double click on the external wall in the zone form (Figure 3.4), then double click on the window to open the window form. Search in database under “Integrated Window Shading” and select “External blind” as Device as shown in Figure 5.3.

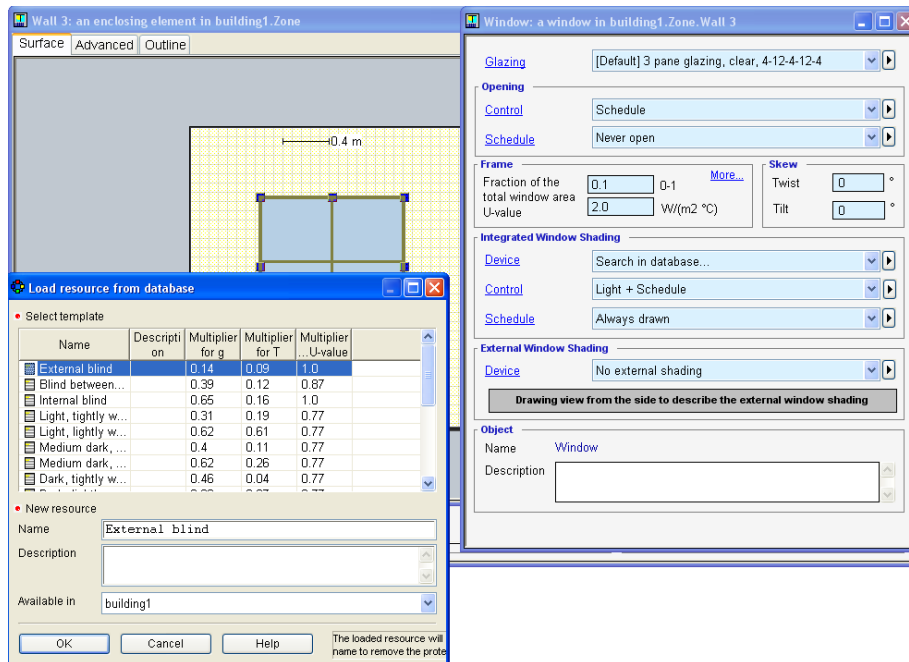


Figure 5.3

Next we define the control system to use. Right below the previous choice in the window form select “New” instead of “Light + Schedule” as Control and give an appropriate name. An empty macro form appears. Drag a thermostat from the Blocks palette and connect the measure link on the thermostat to the zone air temperature as shown in Figure 5.4. Then connect the out signal link to macro interface shading.

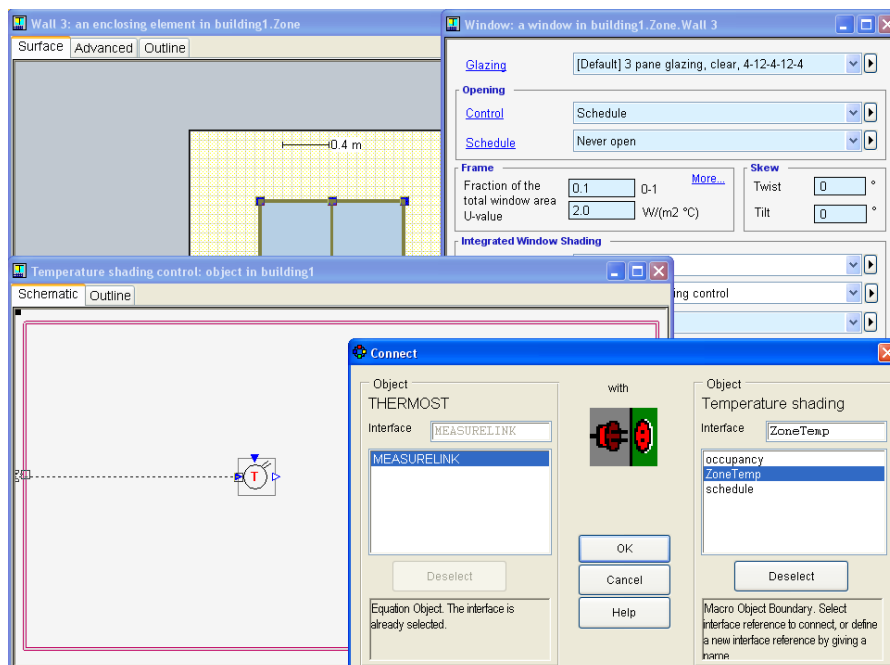


Figure 5.4

To achieve the right performance of the thermostat we have to give a dead band with sign; in this case a negative sign to get out signal 1 for high measure signals and 0 for low. Finally we insert a Constant field from the Inout palette and connect it to the set point link of the

thermostat. With a given set point value (22°C in Figure 5.5) the control system is defined. After the simulation the effect of the shade control can be checked in the diagram “Heat balance of building”.

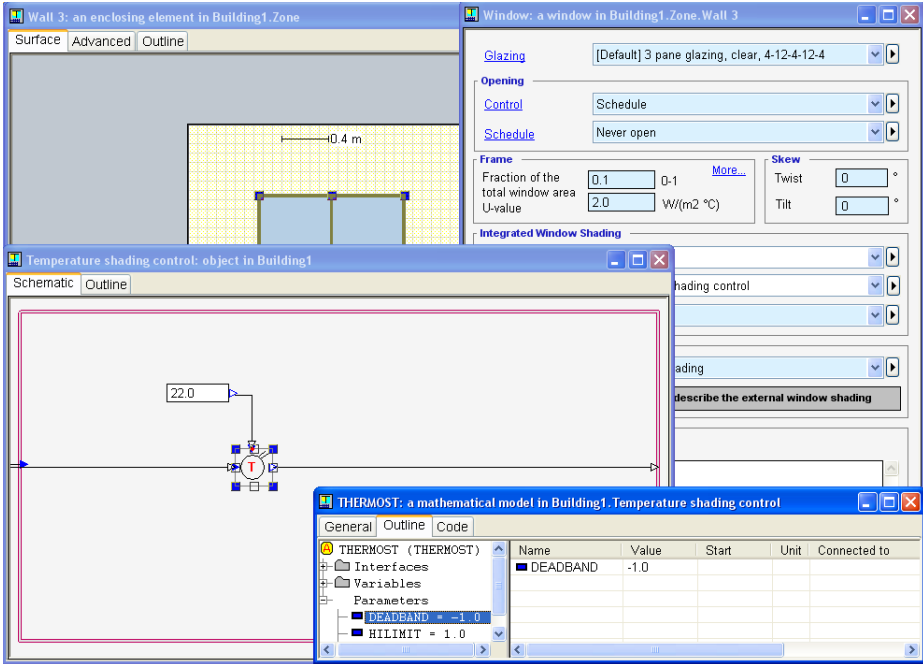


Figure 5.5

6. Tips and tricks

6.1. *Speeding up computation*

IDA Indoor Climate and Energy builds a single large simultaneous system of equations for all processes in the building. This system of equations contains several ordinary differential equations and has therefore different “built in” time constants. The room air will for example react quickly on a convective heat load, while the ground layer below the building can have a time constant several orders of magnitude larger. This system of equations is solved with numerical methods that adapt the timestep to the frequency content of the solution. Short time constants in the model in combination with high frequency content in driving functions (many starts and stops) can lead to long execution times.

Internally generated starts and stops (events) will also lead to short timesteps. Try for example to use a thermostat for radiator control.

The sheer number of equations in the system is naturally significant for the execution time, which increases roughly linearly with the problem size¹¹. Therefore, it is vital not to model any unnecessary detail, and not for example model a large number of identical objects such as windows or cooling beams, when they could be replaced by a single larger one. Each added zone will contribute several hundred variables to the system of equations.

The single most important factor for speeding up calculations is to have a reasonable amount of detail in time schedules (driving functions). Describing many sharp transitions will usually only result in increased computation time and not have a significant impact on results.

For a model with reasonably simple time schedules, it can often prove effective to loosen the numerical tolerance to make the solver take longer and thereby fewer timesteps. Two important solver parameters are Tolerance and Maximum timestep. These are accessed from the Advanced tab of the Simulation data dialog. Often the tolerance can be relaxed to, say, 0.1-0.3 from the standard value 0.02.

Looser tolerance will normally lead to acceptable loss of accuracy for accumulated quantities such as monthly energy consumption. For computing design (extreme values) of quantities such as heating or cooling load, one should be more careful with using loose tolerances and a large max timestep. Too loose a tolerance will on the other hand lead to decreased robustness, forcing the solver to often have to back up and retry with a smaller timestep, ultimately leading again to even longer execution time (if indeed the simulation is successful).

To learn about the statistics of a simulation in terms of number of timesteps, variables, restarts etc. view the file screen.txt in the IDA temporary directory idamod40¹². At the end of this file some statistics are given.

¹¹ For more primitive numerical methods, the execution time will typically grow as the cube of the problem size.

¹² This directory will be created in the standard temporary directory of the machine, which may occur in different locations depending on Windows version used. Sometimes the quickest way is to search for ”idamod40”, also showing hidden files.

6.2. Numerical instabilities

With a tool like IDA Indoor Climate and Energy it is easy to build large non-linear systems of equations and solve them for thousands of time points. However, it is impossible to, even theoretically, guarantee the success of the solution procedure. Any non-linear system of equations may have more than a single solution, or none at all. Numerical computer programs are in this respect different from most other types of software, where it may be, at least theoretically, possible to create a bug-free code. IDA Indoor Climate and Energy also has a more difficult task than most other building simulation software, where less freedom is given to create mathematically complex models.

A major part of the IDA development work is devoted to improving the solver performance on difficult cases. However, this work is altogether dependent on close interaction with users. It is vital that users, which have built reasonable and meaningful models that are difficult to solve, take the trouble of sending the model to the support office. This is most easily done by using the Mail support function on the Help menu.

Some physical processes more often lead to problematic models. Generally speaking, models with significant natural ventilation flows through openings or leaks are the most difficult to solve, especially when the effect of wind pressure is included. If, in addition, vertical temperature gradients are to be simultaneously computed, one obtains a severely stiff and non-linear system of equations.

A common type of error is when the user has graphically defined a piecewise linear controller and unintentionally entered several points near each other, giving the curve (in micro scale) several sharp corners and jumps. The solver will invariably have problems to negotiate the sharp turns of such a graph. Check the table view of such a curve for unintentional points.

A model that in spite of numerous revisions continues to give some trouble is the radiator and cooling panel model. If the user requests a large (or very small) maximum power in relation to the physical size of the device, the model may lead to failed simulations. The solution is to alter the physical dimensions somewhat.

A general way of dealing with difficult cases is to decrease the tolerance parameter that was discussed in the previous section. This forces the solver to be more careful and take smaller steps, which in most cases improves robustness. A tolerance of 0.001 or even smaller can sometimes be used.

A frequent situation is that any change in input data makes a previously failed run go through. This is not as strange as it sounds, since each change will lead to a different sequence of timesteps and in this way the exact combination of values that led to the failed timestep is avoided.

Another often effective trick is to replace sharp steps in schedules with steep ramps, especially for the fan control schedule. This will enable the solver to gradually over a short time period approach the new solution and thereby reach it more securely. If, in addition, the start and endpoints of the ramp are marked as input events, the solver will be even more cautious. Input events are marked by repeating the same time point twice in the profile, something best done in the Data tab view.

7. ICE Reference Manual

7.1. General

7.1.1. IDA Indoor Climate and Energy

IDA Indoor Climate and Energy (ICE) is a simulation application for accurate study of indoor climate of individual thermal zones in a building as well as energy consumption for the entire building. The user interface has been designed to make it easy to build up and simulate simple cases, but also to offer the advanced user the full flexibility of IDA, to facilitate the simulation of complex or unusual cases.

The system to be simulated consists of a building with one or more zones (rooms) and a primary system (the subsystem containing primarily hydronic components such as chillers and boilers) and one or more air handling units. The default plant and AHU have unlimited capacity for providing zones with air and water at given temperatures. For many studies nothing needs to be altered in these central HVAC systems.

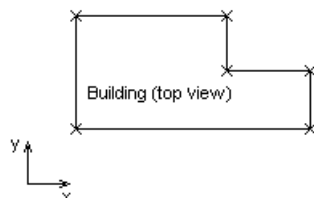
With version 4, two additional concepts are added: ideal room units (cooler and heater) and local air handling units. Local AHUs are the same thing as central AHUs, but they serve only a single zone and they are reached only from this zone (under the More... button). Ideal room units provide zones with heating and cooling, but they are not physically connected to the central plant. They can be thought of as self contained boilers and chillers, consuming electricity or fuel and serving the zone with heating or cooling.

Surrounding buildings or other objects might shade the building. The air inside the building contains both humidity and carbon dioxide. Weather data is supplied by weather data files containing information on actual or synthetic weather. The effects of wind on the building may be taken into consideration. Predefined building components can be loaded from a database. This can also be used to store personally defined building components.

The program is built up around forms and dialogs. The Status bar can be found at the bottom of the IDA window. In many cases it explains in brief the significance of the different areas (list boxes, parameter boxes, etc.) that can be found under the cursor in the different forms and dialogs. It is recommended that the user get into the habit of reading the text in the Status bar if anything is unclear.

7.1.2. The Geometry in IDA Indoor Climate and Energy

A building in IDA Indoor Climate and Energy can contain one or several zones (rooms). A zone is either a prism with any number of vertical walls, a flat horizontal floor and one ceiling or a part of such prism limited by the building's roof. In addition to windows and openings (doors), different types of heating and cooling units can exist within these restricted surfaces.

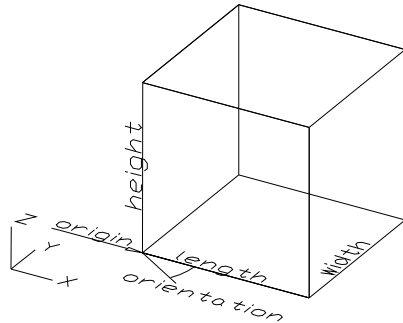


The geometry of the building is defined by the corners of one or more building sections (see above figure). The corners' x and y co-ordinates are indicated in a local coordinate system for the building (called the building system below) that moves and rotates with the building.

Before editing, the origin of coordinates for the building system coincides with the lower left

corner of the building. The building system is then also at the origin of coordinates for the global coordinate system (which is marked by the origin marker).

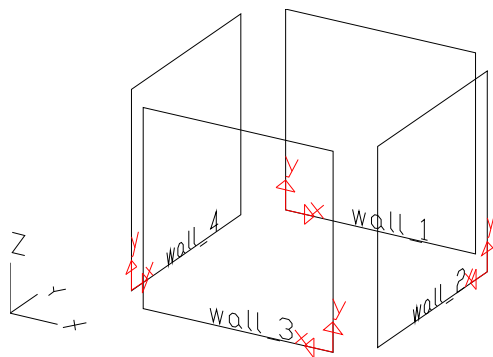
The shape of building section is limited to prisms with vertical walls or a part of such prism limited by the roof. The roof may consist of one or more flat polygons



The zone geometry is described relative to a point in the building system, where the z-coordinate gives the height of the floor above ground level.

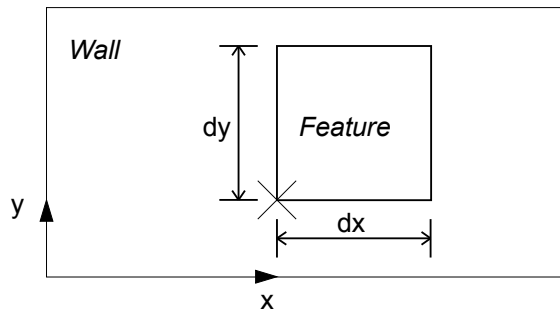
The zone is defined by its height plus the corners of the floor. The corners' x and y coordinates are indicated in a local coordinate system for the zone (called the zone system below) that moves and rotates with the zone. The zone system is defined by the origin's coordinates (in the building system) and the rotation angle around the z-axis.

The shape of the zones are limited to prisms with vertical walls. If the prism is intersecting with the roof, only the part under the roof is included in the zone.



A local coordinate system (called the surface system) is defined on every enclosing surface. For vertical walls, the origin of coordinates is located in the lower left corner of the surface (from inside the zone). For floors and horizontal ceilings, the surface system coincides with the zone system (with z-coordinate omitted). For slanted ceilings (or fragments of ceiling), the x-coordinate is always horizontal passes thru the lowest corner of the ceiling and the y-coordinates is located in the ceiling's plane, is perpendicular to the x-axis and passes thru the leftmost corner of the ceiling.

The surface system is used to indicate the position for features (such as openings) on the surface:



The geometry for objects like windows and heating devices is defined by a rectangle. The insertion point of the rectangle (lower left corner) is given in the surface system. If the surface (wall, floor, or ceiling) is not rectangular, the position of an object is the intersection of the rectangle with surface.

7.1.3. Objects in IDA Indoor Climate and Energy (in alphabetical order)

IDA Indoor Climate and Energy contains a number of objects of interest to the user. See also Objects in hierarchical order.

- Active beam
- Air handling unit
- Balcony, screen and marquee
- Building
- Building section
- Chimney
- Choice of output
- Climate definition
- Climate file
- Control of integrated shading
- Controller setpoints
- Convective interior mass
- Cooling coil
- Cooling device
- Electric radiator
- Equipment load
- External window shading
- Extra energy and losses
- Fan coil
- Face
- Floor heating
- Ground conditions
- Heat exchanger
- Heating coil
- Heating device
- Heated/cooled floor
- Heated/cooled panel
- Ideal cooling device
- Ideal heating device
- Infiltration
- Integrated shading

Leak
Lights
Location
Material
Occupant load
Opening
Opening control
Piecewise proportional controller
Pressure coefficients
Primary system
Profile
Project data
Results
Roof
Schedule
Simulation data
Site object
Skylight
Surfaces_
System parameters
Thermal bridges (building level)
Thermal bridges (zone level)
Thermal mass
Wall definition
Wall, Floor and Ceiling
Wall part
Water radiator
Wind profile
Window
Window (detailed)
Zone
Zone defaults

7.1.4. Objects in IDA Indoor Climate and Energy (in hierarchical order)

The hierarchical structure for the objects in IDA Indoor Climate and Energy are shown in the formation below. Objects are edited either in forms or dialogs.

See also Objects in alphabetical order.

Building Location

-

- Wind profile
- Climate data
- Zone defaults
- Site object
- Simulation data
- System parameters
- Choice of output
- Project data
- Climate file

- Ground conditions
- Thermal bridges
- Infiltration
- Extra energy and losses
- Results objects
- Air handling unit
- Primary system
- Building section
 - Faces
 - Roof
 - Faces
- Zone
 - Controller setpoints
 - Thermal bridges
 - Results objects
 - Surfaces - Construction
 - Inner surface
 - Outer surface
 - Wall part
 - Window -
 - Glass construction
 - Integrated shading
 - Schedule for integrated shading
 - Control of integrated shading
 - External shading
 - Balcony, screen and marquee
 - Opening schedule
 - Opening control
 - Window (detailed) -
 - Window construction
 - Schedule for integrated shading
 - Control of integrated shading
 - External shading
 - Balcony, screen and marquee
 - Opening schedule
 - Opening control
 - Opening -
 - Opening schedule
- Leak
- Chimney
- Skylight
- Active beam
- Heating device
 - Electric radiator
 - Cooling device
 - Floor heating
 - Heated/cooled floor
 - Heated/cooled panel
- Fan coil
- Local AHU
- Ideal cooling device
- Ideal heating device

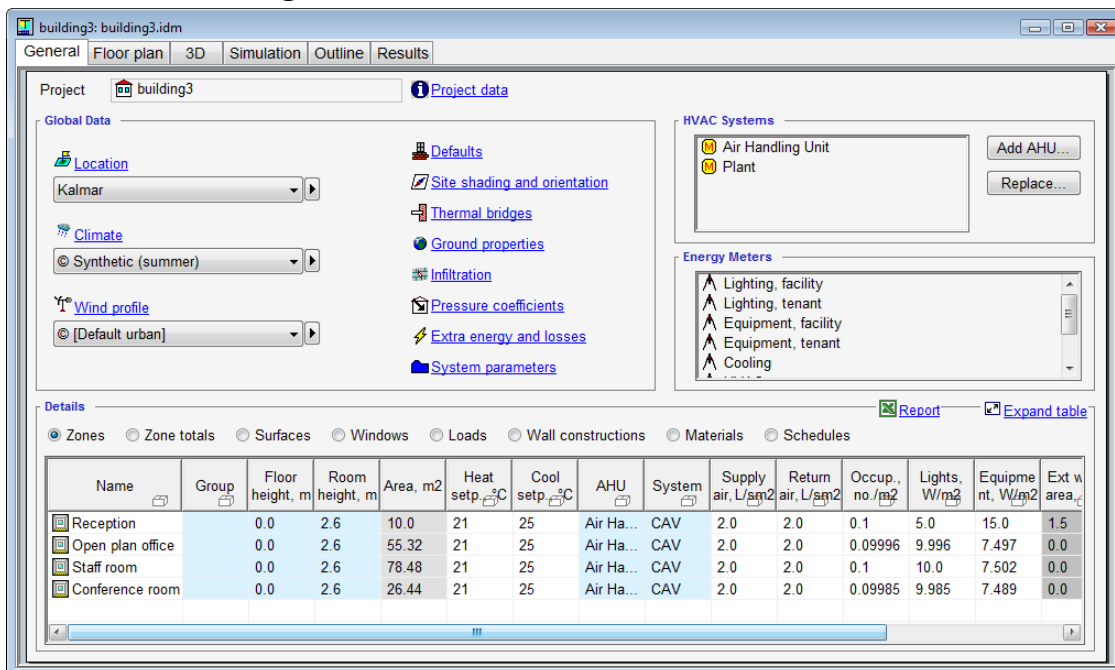
Lights - Schedule
 Occupant load - Schedule
 Equipment load - Schedule
 -
 Convective internal mass
 Thermal mass

Energy meter - Energy cost

7.2. Building simulation

7.2.1. General tab

7.2.1.1. The building's form



An object of the Building type is presented in a form on the screen. This contains all the objects connected to the building. The building's form is automatically displayed each time the user selects to open an already existing, or a new, document (please see Working with IDA systems). The form above shows a sample appearance of the building form. It contains standard versions of the air handling unit, the primary system and energy meters in the HVAC-system and Energy meter list boxes.

Field descriptions, etc:

Location

Location of the building. The chosen location does not have to correspond exactly to the location of the climate file.

Climate

Choice of climate object. The object contains a reference to a climate file and information about this file *or* indicates that synthetic climate is to be used (design day data for summer or winter in the Location object).

Wind profile

Choice of wind profile object. Wind profile is only important for studies where pressure coefficients are given.

Project data

Object for documentation of the simulation object and the current choice of parameters. Project data are written on reports etc.

Defaults

Default settings for all zones and HVAC systems.

Site shading and orientation

Building orientation (compass arrow) and definition of separate shading objects, e.g. trees, hills and neighboring buildings..

Thermal bridges

Coefficients for calculation of loss factors for thermal bridges in zones.

Ground properties

Model and parameters for temperature conditions below the building.

Infiltration

Method and parameters for building air leakage

Pressure coefficients

Coefficients for calculation of wind pressure on external surfaces of the building.

Extra energy and losses

Losses from HVAC distribution systems, hot water and other energy use items

HVAC System

List of central hydronic and air handling units..

New AHU ...

Add an extra AHU (If the existing one is not needed: Do 'Replace' instead)

Replace AHU ...

Replace the selected AHU with a new one.

Energy meters

List of energy account items, with info on cost, CO₂, and primary energy factor

Details

This table shows the main parameters of zones and other objects. The row of radio buttons allows selecting the category of objects to be shown in the table. In the table, you may:

- edit parameters (except in grey fields)
- select cells and copy/paste their contents to and from other applications
- open objects (double click the object in the left column)
- sort the contents by clicking a column header
- show a parameter on the 3D plan (click "box" image in the column header)
- export all tables to Excel

7.2.1.2. Dialog for location

Position	
Country	Sweden
City	Kalmar
Latitude	56.68 N °
Longitude	16.3 E °
Elevation	0 m
Time zone	1 E h

	Winter	Summer	
Dry-bulb min	-10	17	°C
Dry-bulb max	-5	25	°C
Wet-bulb max	-5	18	°C
Wind direction	180	90	°
Wind speed	4	4	m/s
Clearness number	1.0	1.0	0-1

Object	
Name	Kalmar
Description	

This dialog describes an object of the Location type. It contains the geographical position and design weather data for the place where the calculation object is assumed to lie.

The given weather data for winter and summer design conditions are used when synthetic weather has been selected in Heating<<link>> or Cooling<<link>> load calculations. This data is also used for Custom simulations<<link>> when the Synthetic (winter or summer) option has been selected in the Climate field in the building form

Field descriptions, etc:

Location: Choice of location. The rest of the dialog shows the details of the selected location.

Position: Country country or geographical area for calculation

Position: City place for calculation

Position: Latitude calculation object's latitude (Deg). To avoid sign confusion, the directions from equator are denoted by N (north) and S (south)

Position: Longitude calculations object's longitude (Deg). To avoid sign confusion, the directions from Greenwich are denoted by E (east) and W (west)

Position: Elevation calculation object's height over sea level (m)

Position: Time zone calculation object's time zone (h). To avoid sign confusion, the directions from Greenwich are denoted by E (east) and W (west), e.g. 1 E for Central Europe.

Design days: Dry-bulb min Lowest dry bulb temperature during the day (°C)

Design days: Dry-bulb max Highest dry bulb temperature during the day is considered to occur at 15:00. (°C)

Design days: Wet-bulb max Highest wet bulb temperature during the day is considered to occur at 15:00. (°C)

Design days: Wind direction The wind is only relevant to the airflow through the building if pressure coefficients are given and more than one leak or opening has been defined (Deg)

Design days: Wind speed Wind direction and speed is kept constant during the day (m/s)

Design days: Clearness number Reduction factor for direct and diffuse sunlight. 0 = dark, 1 = clear, dry and cloudless atmosphere, 1.15 = extremely clear conditions. (-)

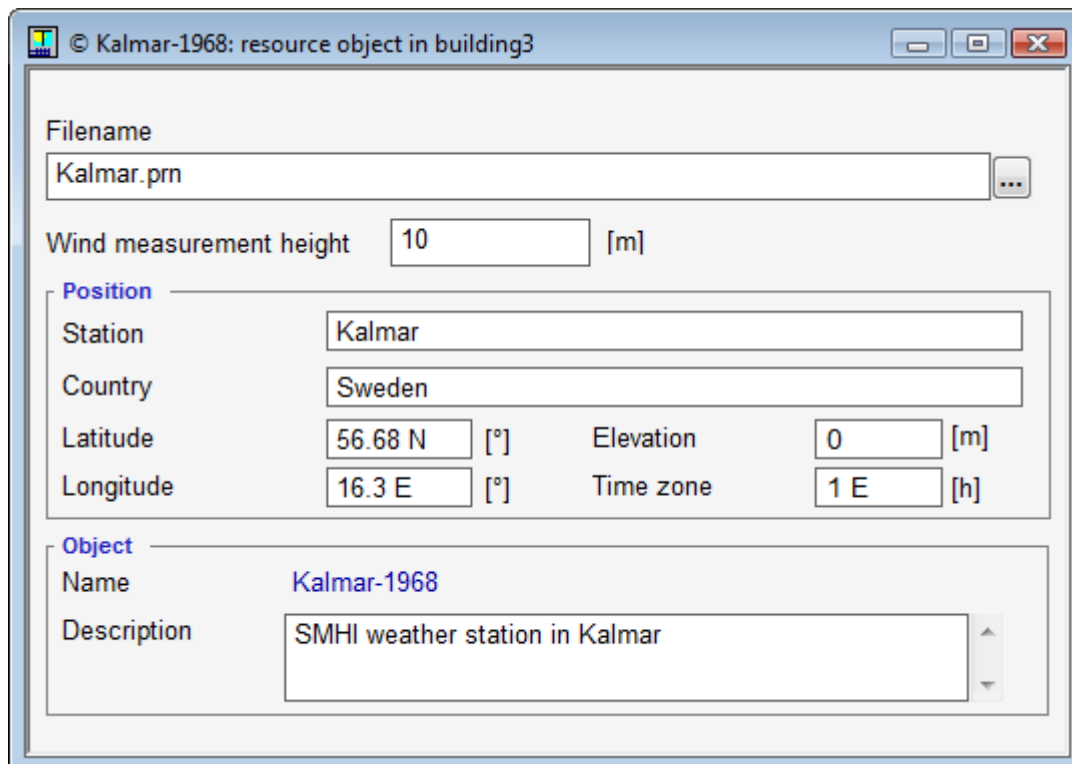
Object: name and description

The Dialog for a location is opened when the Right button is pressed with the cursor over an object of the Location type (e.g. Malmö) in the Location field in the building form or in the form for IDA-resources.

For synthetic climate, the outdoor air temperature varies sinusoidally between the given min and max values. The warmest time of day is assumed to be at 15:00, for dry as well as for wet bulb temperature.

A clearness number indicates the presence of clouds or unclear atmospheric conditions. For a clear, dry and cloudless sky, 1 is given. A normal daily average for a clear day summer day can be 0.8. Under extremely clear conditions, particularly in northern countries, 1.15 may occur. Further information about clearness number and additional references can be found in ASHRAE Fundamentals.

7.2.1.3. Form for climate definition



© Kalmar-1968: resource object in building3

Filename: Kalmar.prn

Wind measurement height: 10 [m]

Position

Station: Kalmar

Country: Sweden

Latitude: 56.68 N [°] Elevation: 0 [m]

Longitude: 16.3 E [°] Time zone: 1 E [h]

Object

Name: Kalmar-1968

Description: SMHI weather station in Kalmar

This form describes a Climate object, which provides information about an actual climate data file on disk. Indicated here is the path to the file (.prn) which contains the data, geographical position of the station, height of the wind measurement, etc.

The description gives qualitative information about the data, its selection, extreme weather periods etc.

Field descriptions, etc:

Filename the name and place for the data file, which must have a special IDA-format, is given here.

Wind measurement height wind speed data in the file corresponds to measurements at this height over the ground (m)

Position: Station name of measuring station

Position: Country country or geographical area of measuring station

Position: Latitude measuring station's latitude (Deg) To avoid sign confusion, the directions from equator are denoted by N (north) and S (south)

Position: Longitude measuring station's longitude (Deg). To avoid sign confusion, the directions from Greenwich are denoted by E (east) and W (west)

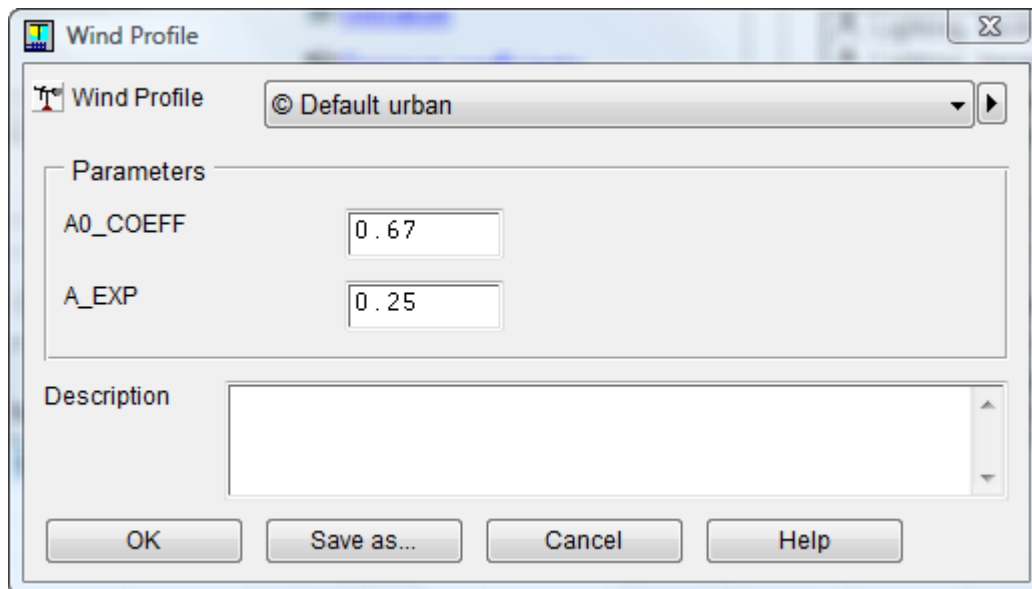
Position: Elevation measuring station's height over the sea level (m)

Position: Time zone measuring station's time zone (h). To avoid sign confusion, the directions from Greenwich are denoted by E (east) and W (west), e.g., 1 E for Central Europe.

Object: Name and description

The form for Climate definition is accessed when the Right button is pressed with the cursor over an object of the Climate data type (e.g. Helsinki) in the Climate field in the building form or in the form for IDA-resources.

7.2.1.4. Dialog for wind profile



This dialog describes an object of the Wind profile type. A description of the wind's profile can be found uppermost in the box.

Field descriptions, etc:

Name Choice of wind profile. The rest of the dialog shows the details of the selected profile.

Description

A0_COEFF coefficient in power law expression for wind speed

A_EXP exponent in power law expression for wind speed

The wind is only relevant to the airflow through the building if pressure coefficients are given for the building facades.

The wind speed is considered to vary vertically. It is zero on the ground and reaches a speed corresponding to that given in the weather data at the Href height. This height is normally 10 meters and this value applies if synthetic weather is used. When a weather file with measured data is used, the height is taken from the wind measurement height field in the form for Climate definition.

Wind speed at different heights is given by the following power law expression:

$$U = U_{\text{measured}} * A0_COEFF * (H/H_{\text{ref}})^{A_EXP},$$

where U is the wind speed in (m/s) at height H in m. Umeasured is wind speed for the actual time in the weather file.

The wind profile's form is accessed when the Right button is pressed with the cursor over an object of the Wind profile type (e.g. Default urban) in the Wind profile field in the building form or in the form for IDA-resources.

7.2.1.5. Default settings

Building defaults

Elements of Construction

- External walls: © Rendered l/w concrete wall 250
- Internal walls: © Interior wall with insulation
- Internal floors: Concrete floor 150
- Roof: © Concrete joist roof
- External floor: © Concrete floor 250mm
- Glazing: © 3 pane glazing, clear, 4-12-4-12-4
- Door construction: [use wall construction]
- Integrated window shading: © No integrated shading

Generator Efficiencies

	Electric	Fuel	District
Heating COP	1	0.9	1
Cooling COP	3	2	1
Domestic hot water COP	1	0.9	1

Other

- Zone model Fidelity: Energy
- Room unit cold surface temperature: 15 °C

[IDA Resources](#)
[Database](#)

Input given here will be used *unless* other data has been given in the zone or subsystem.

Elements of construction

External walls

Construction for external walls lacking descriptions in the zone.

Internal walls

Construction for internal walls lacking descriptions in the zone.

Internal floors

Construction for internal floors lacking descriptions in the zone.

Roof

Construction for roofs lacking descriptions in the zone.

External floor

Construction for floor slabs lacking descriptions in the zone. A ground insulation layer is normally described as part of the ground structure.

Glazing

Glazing for windows lacking descriptions in the zone (only for simplified window model).

Door construction

The default construction for doors. The default setting for this parameter is "[use wall construction]" that means where be no special treatment of wall material (this setting is compatible with ICE 3.0 version)

Integrated window shadings

The default shading device for simplified window model.

Generator efficiencies

Default efficiencies for plant components and ideal room units. When Meter type of an Energy meter is changed, the default efficiencies of its "clients" will be changed automatically.

Other

Zone model fidelity

Selects the degree of accuracy for the mathematical models of the zones. If the Climate model is selected, a very detailed physical model of the building and its components with, for example, the possibility for a vertical temperature gradient will be simulated. If instead the Energy model is selected, a simpler physical model is used. It has a more conventional degree of accuracy based on a mean radiant temperature. Consequently, the calculation time will be somewhat shorter, all else being equal.

The energy model does not calculate directed operative temperature. The climate model is currently restricted to zones with rectangular geometry.

Both zone models use the same description of the building. All the component models in and around the zone such as windows, radiators, controllers, leaks, terminals, etc. are common for the simplified and detailed zone models.

Choice of model can also be made for every zone locally in the zone's form in the Model fidelity combo box on the Advanced tab. Choice for the whole building in the dialog above does not apply to those zones in which local choices have been made.

Ideal cooler coil temperature

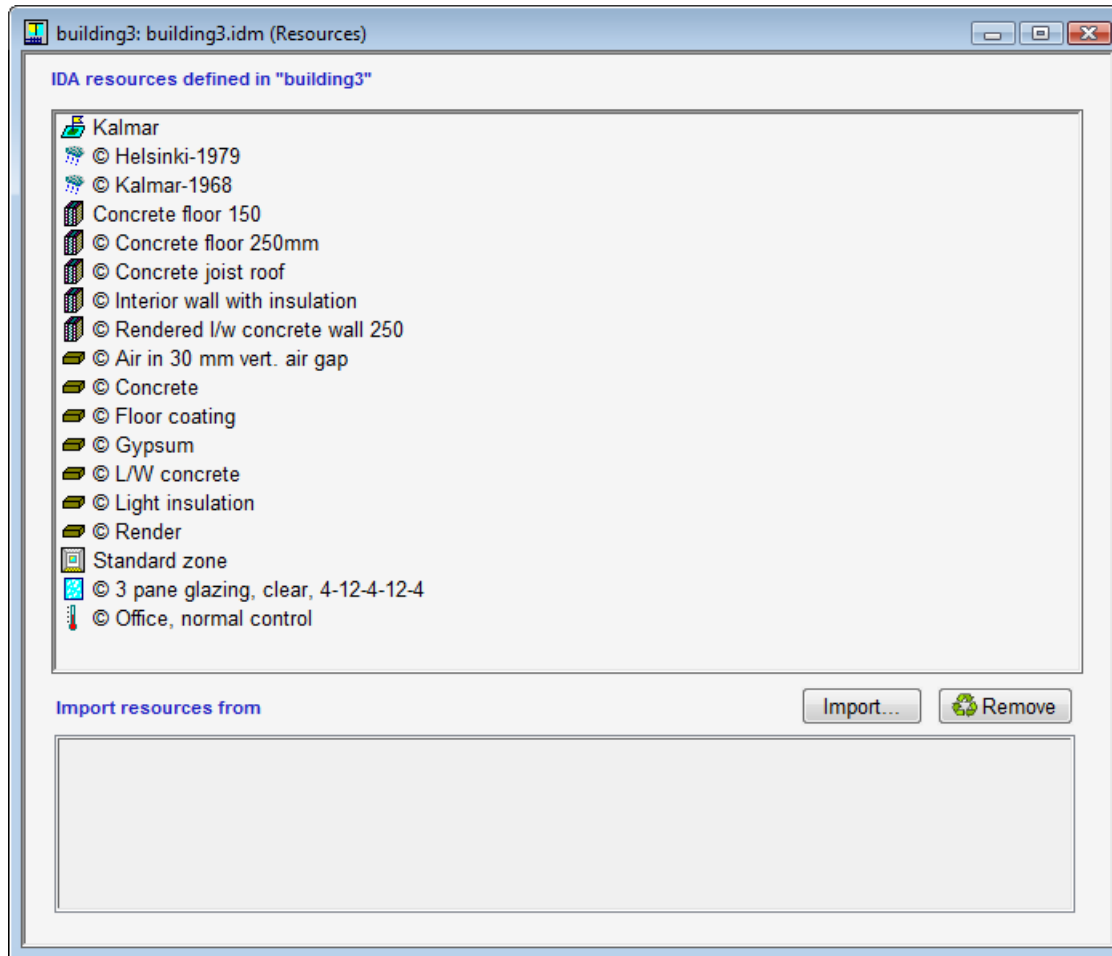
Ideal coolers in zones need a temperature to account for possible condensation.

Links

IDA resources, i.e. used input data objects.

Database

7.2.1.6. Forms for IDA-resources



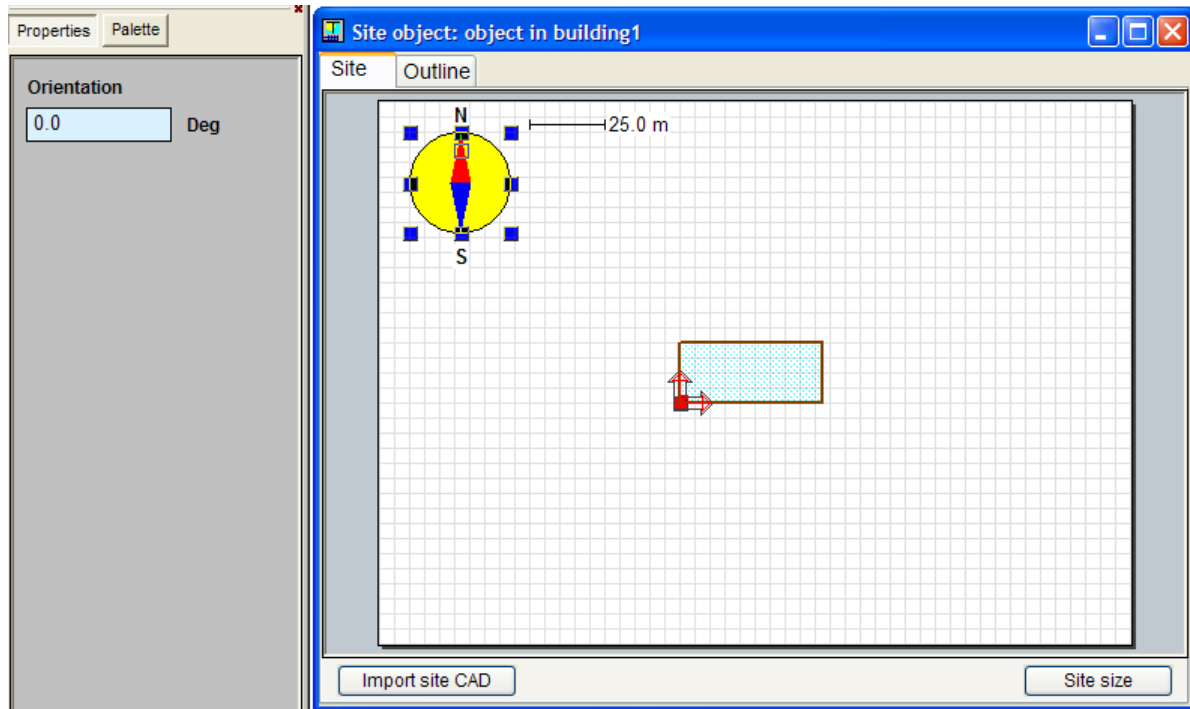
This form contains the IDA-resources currently or previously used in a project. If, for example, an Internal wall with insulation is chosen somewhere in the building, the IDA resources Internal wall with insulation, Gypsum, Air in 30 mm vertical gap and Lightweight insulation will appear in the list. (The Internal wall with insulation contains references to a set of material resources).

A change made in an IDA resource at the building level applies to all the instances in the building where it has been used. If, for example, the density is changed for the IDA resource concrete from 2300 kg/m^3 to 2400 kg/m^3 at the building level, this change will influence all walls and floors in the building that contain concrete.

Resources that have no references to them (aren't used) can be deleted. Right click to check where a resource is used.

The IDA-resources handled in IDA Indoor Climate and Energy can also be stored in the database. A list of all available types can be found in the Database objects in IDA Indoor Climate and Energy.

7.2.1.7. Edit the position, orientation and surrounding of the building



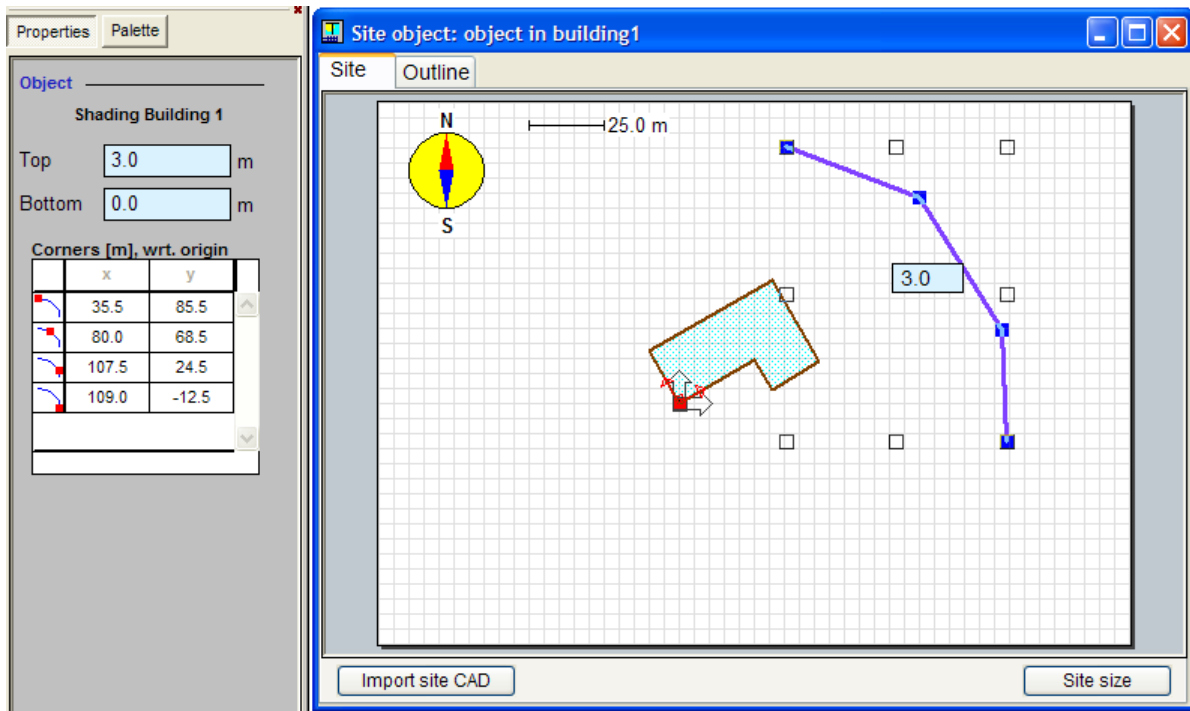
In the form Site shading and orientation, the building orientation is specified together with possible shading objects that are not attached to the building.

The easiest way to change the building orientation is by turning the compass needle in the upper left-hand corner of the window. Select the compass needle and drag the small square close to the compass's North point (N) to the required position. The desired compass direction can also be given numerically in the Property page when the compass has been selected.

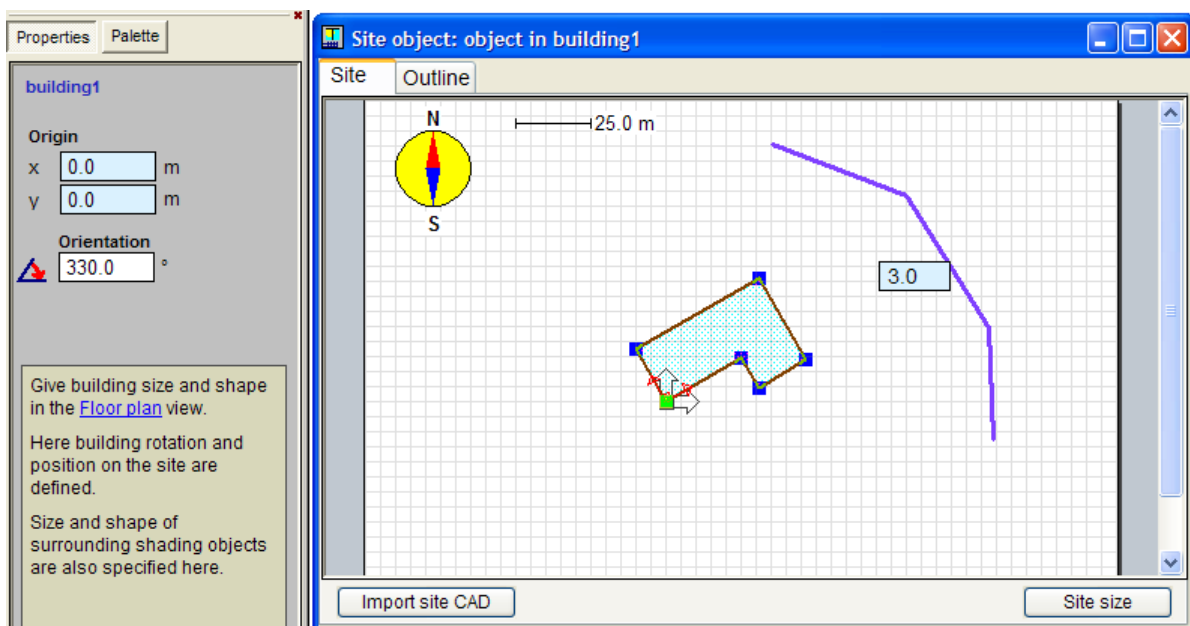
Position and height of shading objects can be defined. This is done with the aid of a polyline along the object's defining surface. To create a shading object such as a building, drag Shading building from the Palette. Now create a polyline by clicking at its starting point, break points, and end point within the form. Click once with the right button and select OK to end. The desired height is introduced into the blue box in connection to the shading object.

The figure below illustrates a polyline with four points and a height of 4.5 meters. The shading objects can be moved and changed. An object can be selected with the left button, making the surrounding rectangle visible and movable. Use the right button menu to select Edit and edit the polyline, as previously described.

Similarly, horizontal shading surfaces can be defined as polygons.



The building can also be moved and rotated in the same way as a card on a table. Do this by holding down the ALT-key and drag at a point within the building's boundary. The building is then drawn along and follows the same movement pattern as a playing card being pulled by a finger on a table. Friction between the card and the table controls the card's movement. It is advisable to avoid editing a rotated building, due to limitations in screen resolution. The position of the building can be edited on Properties page. The coordinates of the origin are given in the site plan coordinate system. Orientation indicates how the building has been rotated in relation to its original position.

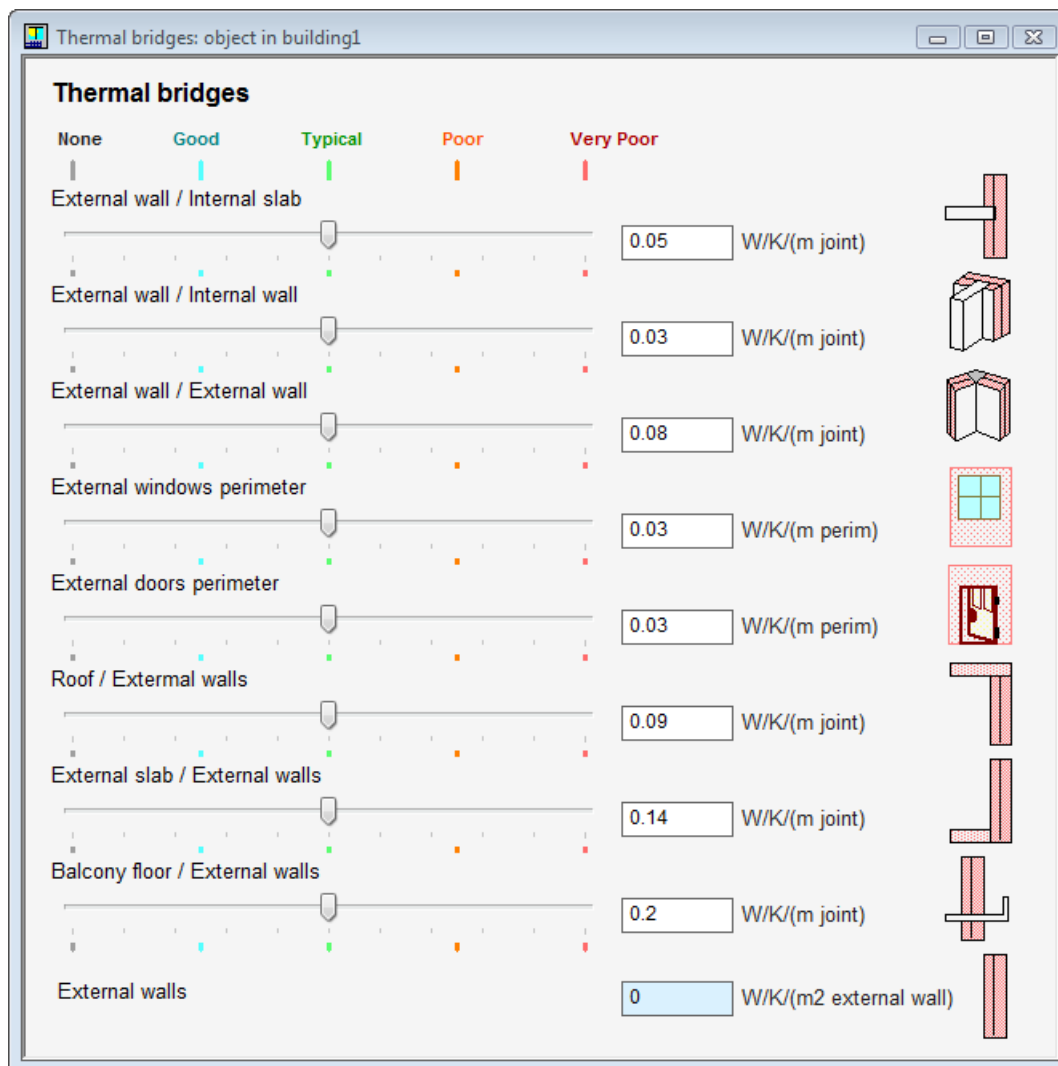


7.2.1.8. Thermal bridges

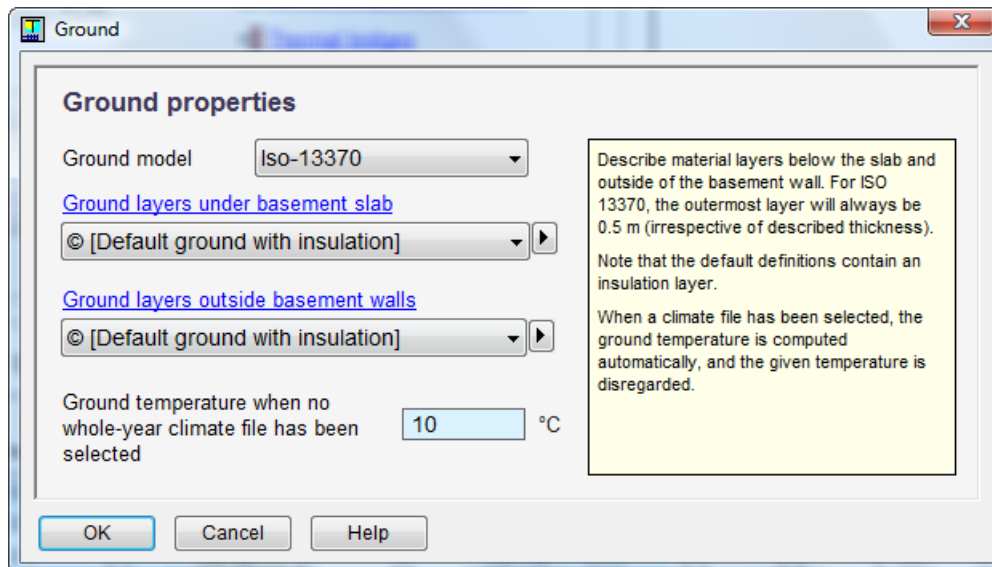
These coefficients are used to calculate the loss factors in thermal bridges in zones. The total loss factor for a zone is calculated as sum of loss factors in bridges created by different construction elements.

The coefficients are given per unit of element size (in most cases per meter). The sizes of elements are by default calculated from zone geometry, but may be also given by the user. The user may also specify extra loss in thermal bridges. Local changes in geometrical or bridge coefficients are done in the zone form.

See also
Calculation of thermal bridge coefficient



7.2.1.9. Ground properties



The dialog for an object of the type Ground conditions, is displayed by clicking on the button with the same name in the building form. Here the calculation model, ground layers and temperature conditions under and around the building can be described.

Field description:

Ground model

Use ISO 13370 (default) or method from IDA ICE 3

Ground layers under basement slab

Ground layers under slab down to virtual (ISO 13370) or constant (ICE 3) temperature.

Ground layer outside basement walls

Ground layers outside wall to virtual (ISO 13370) or ambient (ICE 3) temperature.

Ground temperature when no whole-year climate file has been selected

Temperature used in situations when no climate file is known, e.g. when synthetic design day is selected.

ISO 13370 model

When multiple ground layers have been specified in the Ground properties dialog, only the outermost layer is regarded as "ground" in the ISO sense. Only the material properties of this layer are used, not the specified thickness. Other described layers are regarded as part of the building (in the ISO sense) and are added to layers that have been described elsewhere for building walls and floors.

The ISO model has been implemented for slab on ground and for heated basement (slab below ground) situations. ISO methods for suspended floors (ventilated crawl spaces) and edge insulation are not supported.

The virtual temperature that is defined by the standard 0.5 m below the building, is currently not time-shifted, since this is not required by the standard.

ICE 3 model

The ground layer under the basement floor is connected to a constant temperature, which is computed as the mean air temperature of the selected climate file. The layers around the basement walls are coupled to a facade object for crawl space, which is kept at ground surface temperature. Note that no 2D or 3D effects are modelled for either of these couplings.

7.2.1.10. Infiltration

Data in the infiltration form is used to specify unintentional air flows over the building envelope without having to visit each zone. The Leak area and Given additional in/exfiltration parameters of each zone are by default linked with the global Infiltration data.

Two basic methods of infiltration modeling are supported by IDA ICE: given (equal) in- and exfiltration flows and, alternatively, flows based on leak sizes, fan pressure, wind pressure and thermal buoyancy effects. The latter is default.

When wind driven flow (the latter) has been selected, leaks are automatically introduced in each external wall of each zone (not in floors or roofs). The combined sizes of all these leaks is given here as a single flow/pressure point. This data is then distributed to each zone according the Zone Distribution method (by default according to total wetted external area.) However, note that the unit for leakage is different in the zone form, where leakage is instead specified in terms of Equivalent Leakage Area at 4 Pa.

It is possible to break the link to the global infiltration data from a zone and give another leakage area locally (just type in the input field). It is also possible to set the Leak area of the zone to a very small number and instead introduce "manual" leaks in specific places on the walls of the zone. Note, however, that it is not permitted to set leakage to zero. Completely air tight zones will yield a singular system matrix and the problem cannot be solved.

When instead fixed flows are specified, constant envelope air flows to and from each zone are introduced. However, small envelope "natural ventilation" leaks are also introduced to absorb pressure differences because of for example imbalances in mechanical supply and return flows.

A common problem when fixed flows are used in combination with unbalanced mechanical ventilation is that pressure difference over the envelope becomes unrealistically large. A warning for this is then issued during the simulation and the remedy is to specify larger leak areas in zones with unbalanced mechanical ventilation.

7.2.1.11. Pressure Coefficients

face \ angle	0	45	90	135	180	225	270	315	Face-Azi...
Building body									
f1	0.5	0.25	-0.5	-0.8	-0.7	-0.8	-0.5	0.25	0.0
f2	0.5	0.25	-0.5	-0.8	-0.7	-0.8	-0.5	0.25	90.0
f3	0.5	0.25	-0.5	-0.8	-0.7	-0.8	-0.5	0.25	180.0
f4	0.5	0.25	-0.5	-0.8	-0.7	-0.8	-0.5	0.25	270.0
Crawl space	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Roof	-0.1	-0.1	-0.1	-0.1	-0.1	-0.1	-0.1	-0.1	0.0

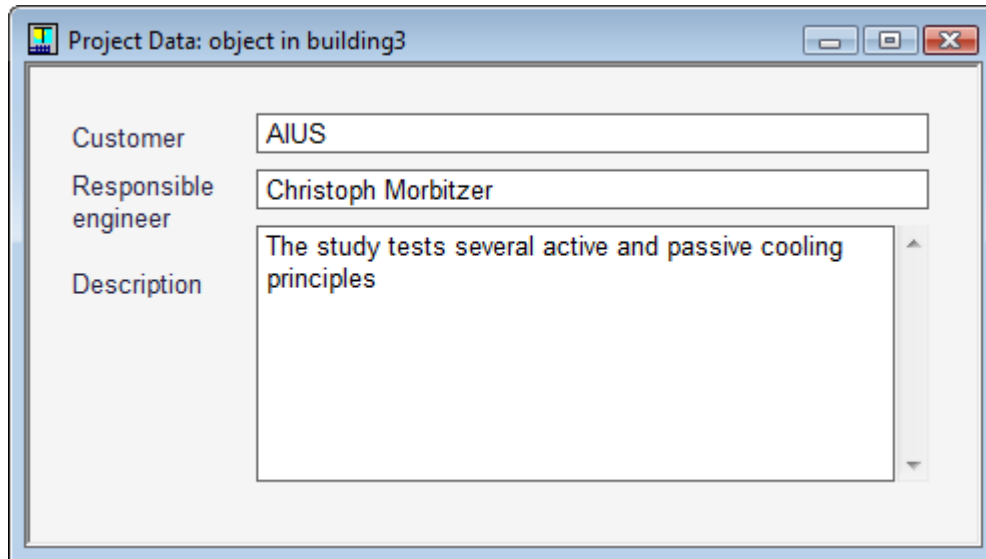
Pressure Coefficients are used to calculate wind pressure on the different external surfaces of a building in relation to the speed of wind at roof height. See ASHRAE Fundamentals. The wind only influences airflow through the building if non-zero pressure coefficients are given. Pressure coefficients are dependent on the building's shape as well as on surrounding aerodynamic conditions. CFD calculations or wind tunnel measurements are required for good precision. In simple cases, handbook data can give acceptable results (at least much better than using fixed infiltration flows). A common handbook data set (from the Air Infiltration and Ventilation Centre) can be accessed by selecting a face or faces and pressing Auto fill.

The wind contribution to pressure is assumed to be constant over the whole surface and is given by the following expression:

$$P_OUT := (PRESSURE-COEFF * RHO * ON-SITE-WIND**2)/2,$$

where the applicable PRESSURE-COEFF depends on facade and wind direction (linear interpolation between given directions), RHO is the air density and ON-SITE-WIND is the speed of wind at the roof height of the building.

7.2.1.12. Dialog for project data



The screenshot shows a dialog box titled "Project Data: object in building3". It contains three input fields:

- Customer:** AIUS
- Responsible engineer:** Christoph Morbitzer
- Description:** The study tests several active and passive cooling principles

The dialog for an object of the Project data type is displayed by clicking in the box with the same name in the building form. A project can be documented here by entering information regarding customer and responsible engineer, etc.

Field description:

Customer text

Responsible engineer text

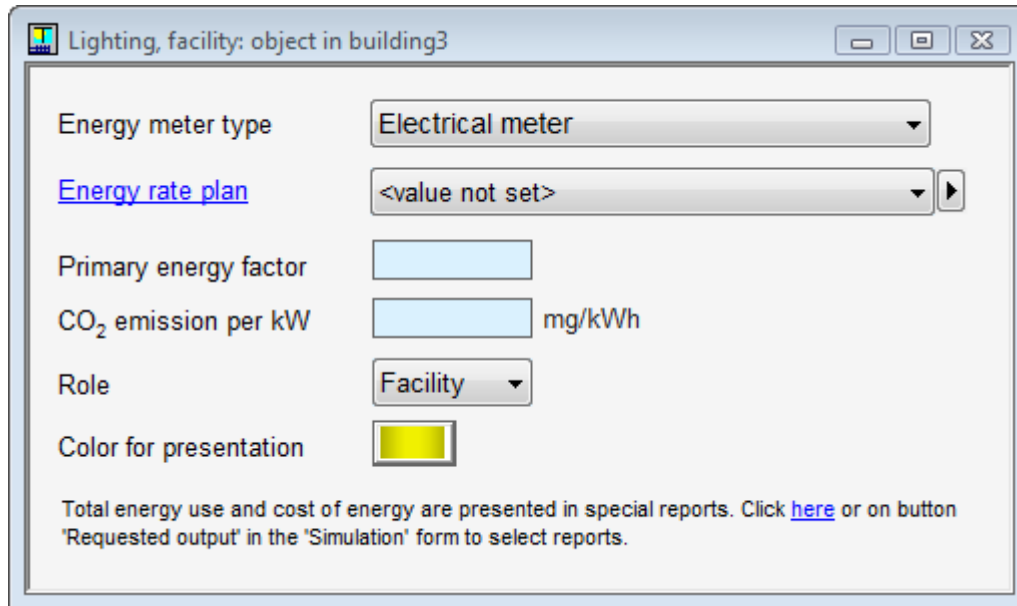
Description This gives a short description of the simulated system.

The text is printed on generated reports.

7.2.1.13. System parameters

<not written yet>

7.2.1.14. Form for energy meter



Lighting, facility: object in building3

Energy meter type: Electrical meter

Energy rate plan: <value not set>

Primary energy factor: []

CO₂ emission per kW: [] mg/kWh

Role: Facility

Color for presentation: [Yellow]

Total energy use and cost of energy are presented in special reports. Click [here](#) or on button 'Requested output' in the 'Simulation' form to select reports.

Energy meters represent items in energy reports. Sometimes they are also referred to as energy accounts. Equipment that use energy will always report to a specific meter and this meter can normally be selected from the client. Energy meters can be renamed by right clicking, but then you have to manually change equipment that report to them. New energy meters can also be dragged from the Palette.

Field descriptions, etc:

Energy meter type Select energy type. This choice will influence default COPs that are used for generating equipment. Fuel is measured in terms of its heating (calorific) value.

Energy rate plan Select object with time dependent energy price information.

Primary energy factor Used energy is multiplied by this factor to compute primary energy use.

CO₂ emission per kWh Used to compute CO₂ emissions from used energy.

Role This choice influences how the used energy is reported.

Color for presentation Used for graphical result presentation of energy use.

Total energy consumption and costs are presented in special reports. Click on Requested output in the simulation form.

The form for energy meter is opened by double clicking on an object of the Energy meter type (e.g. Electric meter) in the Energy meters box in the building form.

7.2.1.15. Dialog for energy price

Energy rate plan

Energy rate plan

Energy rate plan

Currency €

Fixed cost, €/year 400

Energy rates, €/kWh

1: 0.16 2: 0.048 3: 0.0 4: 0.0

Schedule for rate © Schedule for electricity tariff

Object

Name Energy rate plan

Description Valid from 01.07.2009

OK Cancel Save as... Help

Up to four rates are given here and a schedule which selects a rate at every point in time

In this dialog a description of the objects of the Energy price type can be given.

Field descriptions, etc:

Currency Currency name.

Fixed cost, Kr/year Fixed yearly cost.

Energy rate, Kr/kWh

Schedule for rate Choose a schedule that selects rate at different times. Output signal should be 1,2,3 or 4 at all times.

Object: Name and description

The dialog for an energy price is opened from the Energy meter form by opening the object in the Energy rate plan field.

7.2.2. Floor plan tab

7.2.2.1. Floor Plan

The **Floor Plan** view shows the building sections, zones, and imported CAD data at the given height.

All objects are shown in building coordinates. A compass shows the orientation of the building. To change the orientation, go to Site view.

Supported operations on Floor Plan:

Add, move, resize, edit, rename, and remove building sections and zones.

CAD import

See also

Edit the zone position and size in the floor plan
The building geometry

7.2.2.2. Building shape

The building shape is given as one or more **Building Sections**.

The shape of a section a prism with vertical walls or a part of such a prism limited by the roof. The roof may consist of one or more flat polygons that cover the whole roof.

The sections may be arranged both vertically and horizontally. Sections should not intersect. Zones may traverse several building sections.

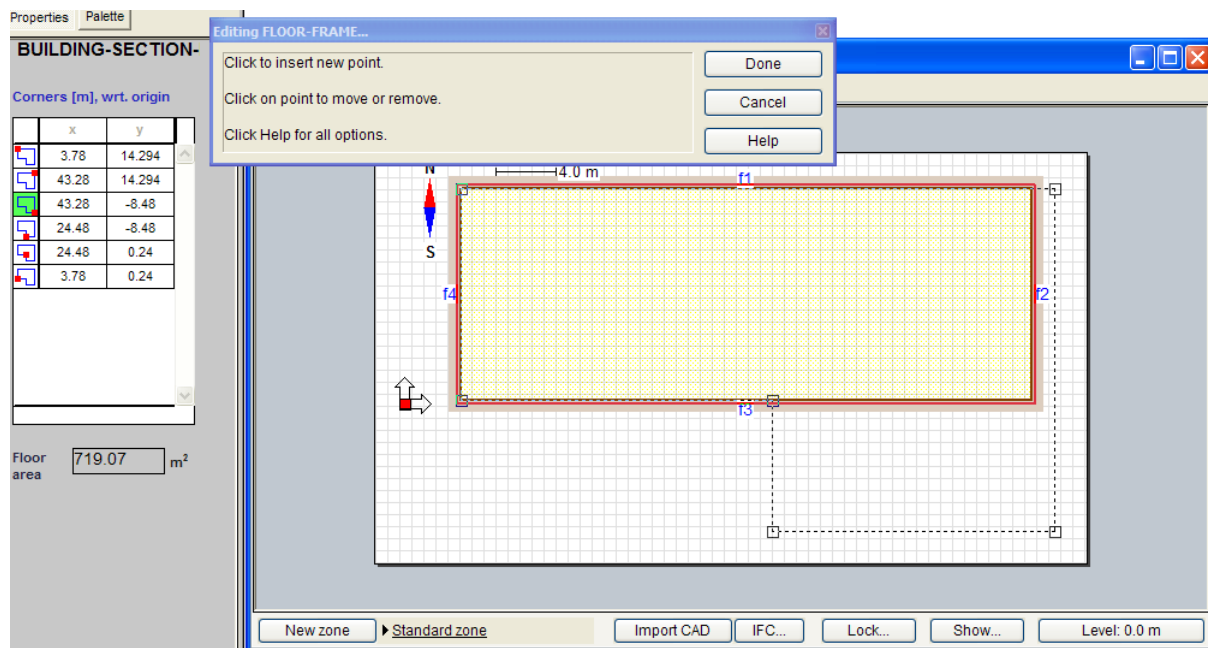
To edit a section

To replace the rectangular shape of a building section by an arbitrary polygon, right click somewhere within the building and select Edit. The perimeter of the building is changed to a so-called polyline. A polyline consists of line segments and break points, the latter marked by small rectangles. The polyline can be edited in five ways:

- Its breakpoints can be dragged to the desired positions for the building's corners.
- Its line segments can be dragged to the desired positions for the building's walls.
- A new breakpoint can be introduced by clicking on or close to the line.
- An existing breakpoint can be deleted by clicking on it.
- Breakpoints for non right-angled corners can be introduced by holding the Control key down and simultaneously clicking on a segment of the line.
- If you hold a point or a segment while dragging it near to some other figure shown on the drawing, the point (or the segment) is snapped to that figure.

Click once with the Right button and select OK to end editing, or alternatively press Enter (to accept) or Esc (to cancel). As an example, the figure below has had a new breakpoint introduced directly to the right of the middle on the building's south wall. The wall to the right of this breakpoint has then been dragged down and an angle-shaped building has been created.

See Edit lines and polygons for more details.



Note also that the program automatically allocates names to all the facades created by the introduction of new breakpoints. This allocation has not yet taken place in the figure above, but can be seen in the last figure in this part.

The building can also be assigned geometry in the form of coordinates. This is done on the Properties page for building section. This page is shown on the Side bar when the section is selected.

To add a new section

Add object of type "Building section". It is also possible to copy a section from another (or the same) building.

To move a section

Drag the section with mouse.

To rotate the section, keep the Alt button pressed.

To align (or stick) a side of the section to another object (zone, another section, a line in CAD drawing), place this side near to that object and keep it for a while with left mouse button pressed.

It is also possible to move a section by editing the coordinates on the Properties view of the Side Bar. This is the way to move the section in z-direction.

Notes. 1. To avoid confusions, do not move building sections far away from the origin of the coordinate system.

2. If you want to move the whole building (relative to the surrounding building) or rotate it, it is better to do it on the site view. In that case the building will be moved together with the building's coordinate system.

To resize a section

When a section is selected, the selection mark contains 8 active points at the corner and the center of the sides of the section (of the bounding rectangle if the section is not rectangular).

To resize the section, drag an active point with mouse.

To align (or stick) a side of the section to another object (zone, another section, a line in CAD drawing), drag this side near to that object and keep it for a while with left mouse button pressed.

It is possible to resize several objects (sections and zones) together.

It is also possible to resize the section by editing the sizes on the Properties page of the Side Bar. This is the way to change the height of section.

To rename a section

See Object name.

To remove a section

See Removing components. Note that the building should have at least one section.

To edit the roof

Click the section with right mouse button and select Edit roof from shortcut menu. See Editing roof for details.

There is also a link on the Properties view of the Side Bar.

7.2.2.3. Editing Roof

The **Roof Editor** view shows the roof of a building section. The numerical details are shown on the Properties page of the Side Bar.

Initially the roof is flat and horizontal.

The user can make the roof slanted (by editing the heights of the roof's corner) and/or divide the roof in parts. Each part should be a flat polygon and the parts should completely cover the roof.

When the user leaves the Roof Editor view, the program calculates the shape of the section's sides and of the walls and the ceiling of the zones that are [partially] inside the section. The user may also press "Apply" button to do the same calculations before leaving the view.

To divide the roof

The roof is divided in two steps

1. Define the vertices (corners) of the roof parts. To do this, press "Edit vertices" button and then click with mouse the positions of the vertices. A new vertex is inserted at every click. Press "Done" button to finish this step.

It is also possible to move and delete vertices in "Edit vertices" mode. See Editing lines for details.

The coordinates of the vertices may be also edited on the Properties page of the Side Bar.

2. Define the roof parts. To define a part, press "Add part" button, then mark all vertices of this part with mouse and press "Make part" button. Repeat this operation until the roof is completely covered by the parts.

To edit the height of the roof

Until the user has changed the heights of roof vertices, the height of a [simple horizontal] roof is specified on the building section's Properties page on the [Side Bar](#) (the field labeled "z max").

To define non-horizontal roof (or parts of the roof), the user may edit the z-coordinates of the roof corners on the roof's Properties page. But often this is not convenient because some of heights might be calculated from other ones (the roof parts are flat, so it is enough to specify the height of 3 points on each part). For this reason IDA ICE provides a special tool to set the roof height.

To set the heights of [some of] roof corners, do the following:

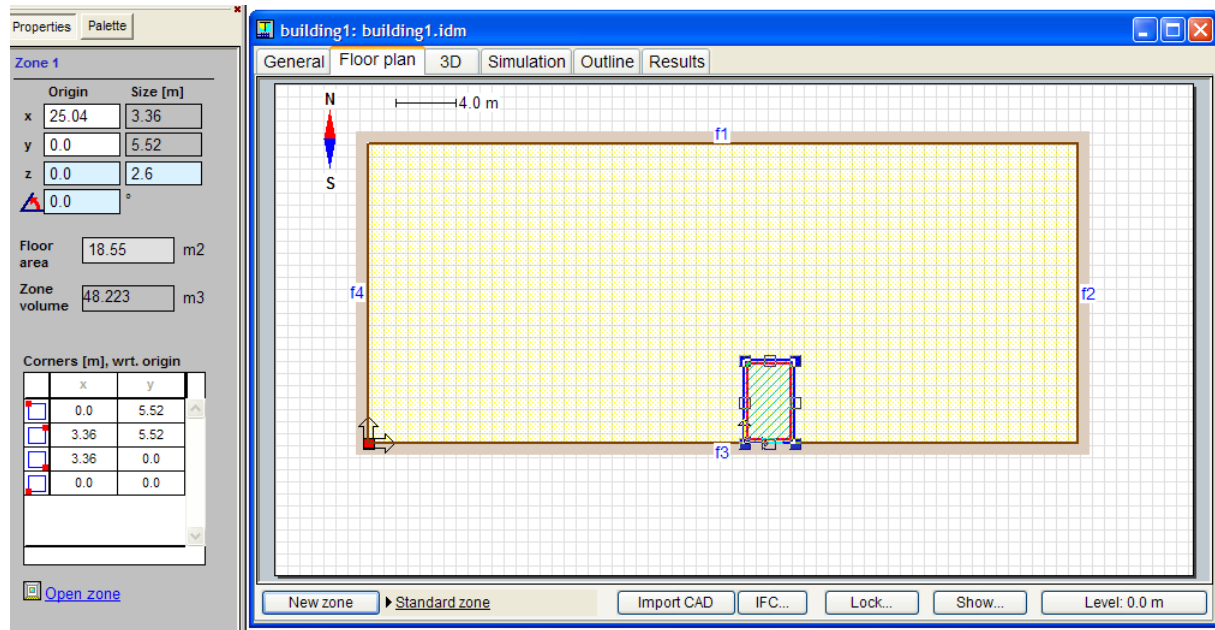
1. Press "Set height" button (on the bottom of the Editing Roof view). IDA ICE will display a Mode dialog that indicates that you are now in "Setting roof height" mode.
2. Select (by clicking with mouse) the vertices those height you would like to change or to use for calculating other heights. Every time you click a vertex, its height becomes available for editing (just above the vertex in the drawing) and you can change it if desired.

At every step, all heights that a selected or may be calculated from the selected ones are shown on the drawing. The changed (either by the user or by the program) are written in red, the unchanged are written in blue. The selected vertices are marked with green circles around them, the calculated ones are marked with red circle. The roof parts are filled in yellow (if flat) or cyan (if non-flat).

3. If you select a vertex that is already selected, you will be prompted to choose either to unselect it or to edit its height.
4. To accept the changes, click "Ok" button in the Mode dialog. You can also withdraw all changes by pressing "Cancel" button.

Note: If you change the section height (field "z max" on the building section's Properties page) for a section where the heights of the roof corners is explicitly defined, the whole roof (i.e., all vertices) will be moved up or down by the value of the change of "z max".

7.2.2.4. Edit the zone position and size in the floor plan



Clicking on the Floor plan button in the zone's form opens a form for the floor plan. (Can also be reached from the building level, by the Floor plan tab.) The position and size of a zone can be changed here. The active zone is indicated by a red border. Any other existing zones in the building, both on the same floor and on other floors, (paler gray lines), are also visible. All zones in the same floor, plus the building shape, can be edited.

To change the position of a zone select it, put the cursor somewhere within the zone's boundary lines and drag it with the mouse to the desired position in the building. The coordinates for the cursor position can be found in the lower right corner of the IDA-window. Hold the zone for a few seconds in a position close to an exterior or interior wall (even on another level), and it will automatically snap into position. This can include a smaller rotation if necessary.

The size of the zone can be changed by first selecting it and then dragging one of the small rectangles either in the corners or in the middle of the sides.

It is also possible to give numerical values for the zone's corners in the form for the **Properties page**.

The property page shows also the floor area and the zone volume.

The floor area is used for presentation of different parameters "per 1 m² of floor area". By default, the floor area is calculated by the program as the area inside walls. The user may change the default value (for example, to exclude unused parts of zone). In this case it will be displayed with yellow background. To restore the default, remove the value.

The zone volume is calculated by the program as the volume inside the walls, floor, and ceiling.

See Geometry in IDA Indoor Climate and Energy for a description of how the geometry and coordinate systems in IDA Indoor Climate and Energy are defined.

The shape of the zone can be edited after right clicking in the zone and selecting Edit. The editing is handled in the same way as for the building (see Building shape).

The zone can also be rotated by holding down the Alt-key, while dragging a point within the zone's boundary lines. The zone is dragged along and follows the same movement pattern as

a playing card being pulled by a finger on a table. Friction between the card and the table controls the card's movement.

Note that if a zone is moved so that, e.g., an external wall becomes an internal wall, the wall construction corresponding to the new position will be selected automatically. This also holds for parts of walls.

Walls are automatically coupled to zones and exterior walls that they adjoin. The couplings should not be created by the user.

Along the bottom of the form, there are the following buttons:

New zone

Add one or more zones, either by drawing on the floor plan, or by selecting spaces imported from IFC model.

Options for new zone

Click the menu button to change the options for creating zones.

Template for new zone

Click the template name to choose or/and edit the template.

Import CAD

Import of an IFC model, or CAD drawing, or Windows graphics as background.

IFC...

Opens IFC menu:

Import...

Import of an IFC model, or CAD drawing, or Windows graphics as background.

Mapping...

The transfer of data objects from the IFC model to IDA is described here.

Remove

Remove 'the imported IFC model

IFC spaces to single zone

When adding zone, create a single zone from all marked IFC spaces.

Every IFC space to zone

When adding zone, create a zone from each marked IFC spaces

Lock

Disable moving and resizing of objects of given types.

Show

Menu with different options that control the appearance of the floor plan.

Visual filter / Remove filter

Shadow or hide objects of some types on both floor plan and 3D view.

Plan size

Presents the relation of the drawing pad to the coordinate system. The view size may be also changed by dragging the page's edges.

Level: xxx m

Press here to view the floor plan at some other height in the building.

7.2.2.5. Zone defaults

Settings for new zones

Use template: Standard zone

General | Loads | Advanced

Controller setpoints

© Office, normal control

Room units

Cooling

Heating

Furniture

Covered part of the floor: 0.2 (0-1)

Weight / area with furniture: 25 (kg/m²)

Room height: 2.6 m

Air

Select AHU: Air Handling Unit

System type: CAV

Supply air for CAV: 2.0 L/(s.m²)

Return air for CAV: 2.0 L/(s.m²)

Displacement degree for gradient calculation: 0 (0-1)

Ok Save as... Cancel Help

In this dialog, templates for new zones can be created and modified. A new template is created by modifying parameters and then save under a new name (Save as ...). A library of typical zones can be created in this way.

Field descriptions, etc:

Zone name

The name of the new zone template

Use template

List of alternative settings for new zones

Save as

Create a new template

Ok

Accept changes

General page

Controller setpoints

Choice of control values for temperature, air quality, and light.

Room units

Check to add ideal heating or cooling units to zones

Furniture

Covered part of the floor

Fraction of floor area which is covered by furniture. (0-1)

Weight / area with furniture

Total weight of furniture divided by the covered area. (kg/m²)

Room height

The distance between floor and ceiling (m).

Air

Select central air handling unit

Choice of air handling unit from the types available at the building level.

System type

Type of ventilation system. Constant or variable flow, with different controls.

Supply air for CAV

Mechanical supply airflow for CAV systems (l/s m² floor area). VAV flows are given in Controller setpoints.

Return air for CAV

Mechanical return airflow for CAV systems (l/s m² floor area). VAV flows are given in Controller setpoints.

Displacement degree for gradient calculation

From 0 (well-mixed) to 1 (displacement ventilation), or a negative value for temperature gradient given by the user

Loads page

For automatic addition of occupant and equipment loads. These will be added to a new zone in proportion to the initial zone floor area. If the zone size is later on changed, the loads will not be changed accordingly.

Select type and schedule

Occupants

Equipment

Lights

Advanced page

Element of construction

External walls

Construction for external walls not described in the zone.

Internal walls

Construction for internal walls not described in the zone.

Internal floors

Construction for internal floors not described in the zone.

Roof

Construction for roofs not described in the zone.

External floor

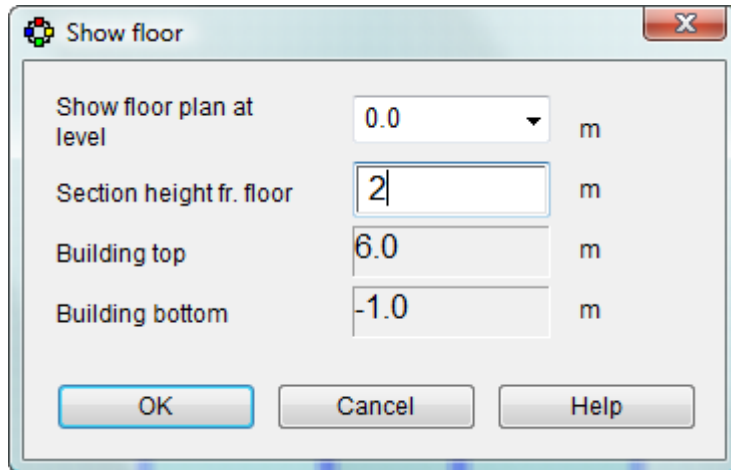
Construction for ground floor slabs not described in the zone. A ground insulation layer is normally described as part of the ground structure.

Room unit power

Specify the power of ideal heater and cooler per 1 m² of initial size of floor area.

See Geometry in IDA Indoor Climate and Energy for a description of how the geometry and coordinate systems in IDA Indoor Climate and Energy are defined. The form for building geometry is reached by selecting Properties in the Options menu. See Edit the shape, orientation and surrounding of the building.

7.2.2.6. Floor level dialog



Show floor plan at level

Select from the list of floor levels or type a new one

Section height fr. floor

Show all zones that overlap with layer from "level" to "level"+"section height"

Building top

The height of the building top above ground

Building bottom

Height of ground floor slab above ground (negative if below ground)

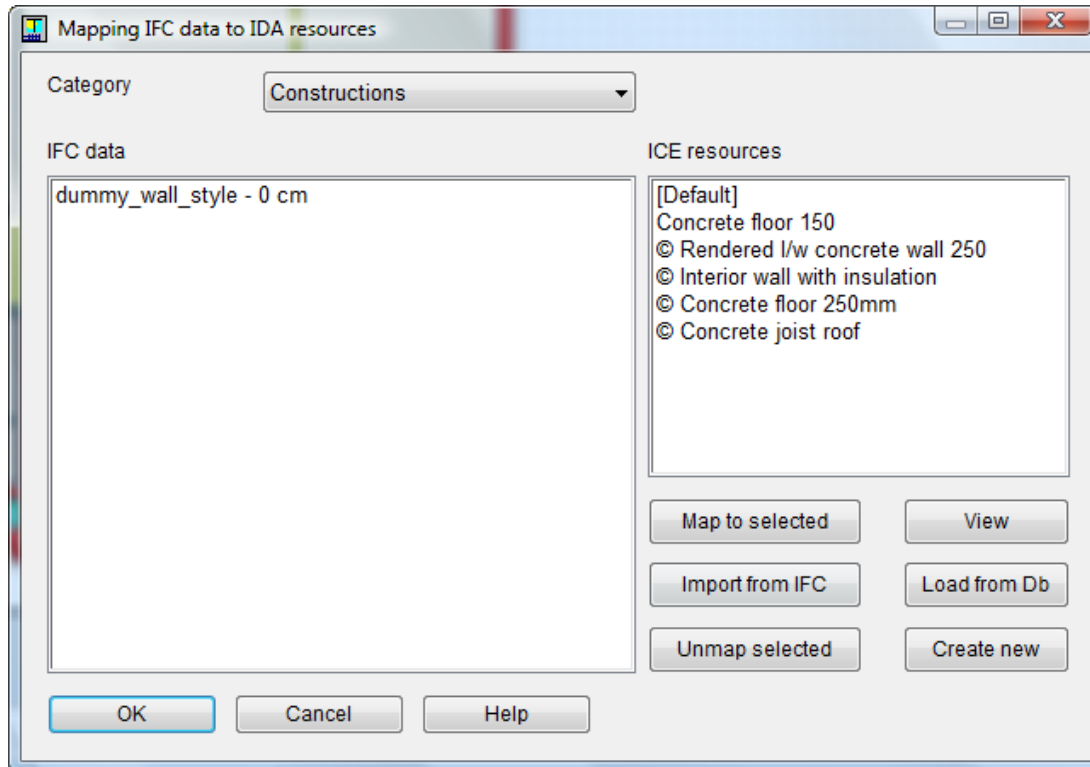
7.2.2.7. IFC and CAD import

ICE can import CAD IFC files of IFC release 2.0, 2x, 2x2 and 2x3 generated by, e.g., ArchiCAD, Revit, Architectural Desktop, MagiCAD Room etc. ICE imports information about wall, window and door positions. ICE relies on the existence of ifcSpaces for creation of simulated zones. ICE imports also styles for walls, windows and materials that can be used to provide an appropriate property set to a group of imported objects in ICE.

See the manual and special ICE topic document IFC Import.pdf for details.

IDA can also import drawings, either in popular CAD and vector formats (.dwg, .dxf, .dwt and other) or converted to the common Windows' graphical formats (.bmp, .jpeg, .tiff, .wmf, .emf). Such drawings may be displayed in the background of the floor plan view or of the site view. They are also displayed in 3D view.

7.2.2.8. IFC mapping



Category

Select an IFC category for mapping (begin with materials)

IFC data

Available IFC data of the selected category

ICE resources

Available IDA resources of the selected category

Map to selected

Map the selected IFC objects to the selected IDA resource

First select one or more IFC objects and one IDA resource

Unmap selected

Disconnect selected IFC objects from their IDA resources

Import from IFC

Make a new IDA resource from IFC object. Only import of walls is implemented. Import not possible if the materials used in the wall are not already mapped to ICE

View

Show the selected IDA resource

Load from Db

Load resource from ICE database

Create new

Create a new IDA resource

See also Chapter 7 of the manual for further instructions.

7.2.3. 3D Plan tab

7.2.3.1. 3D Plan

Navigation

Rotate

Left mouse button: Press down and move mouse

Pan

Middle mouse button (or both the left and the right mouse buttons on a two button mouse): **Press down and move the mouse**

Zoom

Right mouse button: Press down and move the mouse Move the mouse upwards to zoom in and downwards to zoom out.

Set center of rotation

Press F Sets the point around which the model is rotated and towards which it is zoomed. Alternatively use **Right mouse button menu -> Set focus**

Tip: Use this function frequently to make navigation as easy as possible.

Fit the model in the 3D view

Press R Zooms and pans the model so that the entire model is visible. Alternatively use **Right mouse button menu -> Zoom extents** or **Show button -> Zoom extents**

Object interaction

Select object

Left mouse button: Click Selected object is shown in red. The properties of the object are shown in the sidebar.

Open object

Left mouse button: Double-click Opens dialog for selected object. Alternatively use **Right mouse button menu -> Open**

Controlling visibility

Create section of model

x+, x-, y+, y-, z+, z- buttons Creates a section through the building. When the section is activated, **press Ctrl-key, click and move the mouse pointer within the red frame**, to move the section.

Set visibilities of object categories

Right mouse button menu -> Visual filter... Opens the Show objects of type dialog where the visibilities of object categories are set. Use **Right mouse button menu -> Remove filter** to make all objects visible. Alternatively use **Show button -> Visual filter...** and **Show button -> Remove filter**

Visualizing input data and simulation results

Visualizing input parameter values

Details table at General tab: Click box icon of one parameter Shows a graphical overview of the parameter in the 3D tab.

- Numerical values of parameters are shown with colors on the surfaces of the objects. A scalar bar is displayed to correlate a specific color to a parameter value.
- Some parameter values, e.g. the amount of supply air for each zone, are shown with colored arrows. A scalar bar is displayed to correlate a specific color to a parameter value.
- Parameters with a finite number of options, e.g. the occupant schedule for each zone, are shown with colors on the surfaces of the objects. Color coded legends are displayed to correlate a specific color to a parameter value.
- Names of objects are shown as text.

Tip: If a section is activated with one of the x+, x-, y+, y-, z+, z- buttons, only names of the objects in the section are shown.

- Close the parameter visualization **Press H** or **Right mouse button menu -> Hide parameter**.




Visualizing simulation results summary

Summary table at Results tab: Click box icon of one variable Shows a graphical overview of the variable in the 3D tab.

- The values of the variable are shown with colors on the surfaces of the objects. A scalar bar is displayed to correlate a specific color to a variable value.
- Close the results visualization **Press H** or **Right mouse button menu -> Hide parameter**

Visualizing simulation results over time




Click Animation button Opens the Show animated results dialog. In the Show animated results dialog, **select variable to animate** and **click Show**.

- An animation starts and the values of the variable are shown with colors on the surfaces of the objects. A scalar bar is displayed to correlate a specific color to a variable value.
- Pause the animation **Click the Play/Pause button** 
- Start the animation **Click the Play/Pause button**  a second time.
- Speed up/slow down the animation **Change the Tplay value** (the time it takes to animate 24 hours of simulation).
- Jump forward/backward in the animation **Use the scroll bar**, or **change the simulation time value**.
- Close the animation **Use the Close button** . Alternatively **Press H** or **Right mouse button menu -> Hide animation**.

Tip: To highlight difference between close variable values, narrow the visualized range by entering min and max range of values in the Show animated results dialog. Check Auto to reset to full variable range. Check Absolute value to visualize the absolute value of the variable.

Visualizing sun path and shading over time

Click Animation button Opens the Show animated results dialog. In the Show animated results dialog, **check Show shadows** and **click Show**.

- An animation starts with shadows, following the sun path of the day, displayed on the 3D model.
- Pause the animation **Click the Play/Pause button** 
- Start the animation **Click the Play/Pause button**  a second time.
- Speed up/slow down the animation **Change the Tplay value** (the time it takes to animate 24 hours of simulation).
- Jump forward/backward in the animation **Use the scroll bar**, or **change the simulation time value**.
- Close the animation **Use the Close button** . Alternatively **Press H** or **Right mouse button menu -> Hide animation**

Incorporating CAD objects and images

Inserting CAD objects and images

Insert -> Import CAD Opens the Import CAD dialog. In the Import CAD dialog, **select file** and **click Open**. Alternatively **click Import CAD button** or **IFC... button -> Import...** on the **Floor plan tab**.

- The CAD object/image is shown in the 3D view and a section of it is shown on the floor plan tab.
- The CAD object/image is inserted with respect to the building coordinate system, i.e. it will move with the building when the building is repositioned.
- Move and change size of the CAD object/image by **dragging it/resizing it in floor plan** or **editing the parameters in the dialog shown when object is double-clicked**.
- Include the object (not image files) in the shadow calculation and visualization **Check the Calculate shadows checkbox in the dialog shown when object is double-clicked**.
- By default CAD objects are save in the system file (*.idm) so that the original CAD file does not need to be saved. If the CAD file is big, the option of not saving it in the system file will be given. This will speed up the performance of IDA ICE, but the original CAD file needs to be saved.

Tip: To place a CAD object/image at the current mouse pointer, use **Right mouse button menu -> Import CAD**

Inserting CAD objects and images to site

Insert -> Import CAD to site Opens the Import CAD dialog. In the Import CAD dialog, **select file** and **click Open**. Alternatively **click Import site CAD button** on the **Site object dialog** open by clicking Site shading and orientation on the General tab.

- The CAD object/image is shown in the 3D view and in the Site object dialog.
- The CAD object/image is inserted with respect to the site coordinate system, i.e. it will not move with the building when the building is repositioned.
- Move and change size of the CAD object/image by **dragging it/resizing it in the Site object dialog** or **editing the parameters in the dialog shown when object is double-clicked**.
- Include the object (not image files) in the shadow calculation and visualization **Check the Calculate shadows checkbox in the dialog shown when object is double-clicked**.
- By default CAD objects are save in the system file (*.idm) so that the original CAD file does not need to be saved. If the CAD file is big, the option of not saving it in the system file will be given. This will speed up the performance of IDA ICE, but the original CAD file needs to be saved.

Tip: To place a CAD object/image at the current mouse pointer, use **Right mouse button menu -> Import site CAD**

Supported file formats

7.2.3.1.1.1.1.1. *BIM*

IFC (*.ifc)

7.2.3.1.1.1.1.2. *CAD and vector graphic files*

AutoCAD (*.dwg, *.dxf, *.dwt)

3D Studio (*.3ds)

Computer Graphics Metafile (*.cgm)

Corel Presentation Exchange (*.cmx)

MicroStation DGN (*.dgn)

Micrografx DRW (*.drw)

Gerber File Format (*.gbr)

Scalable Vector Graphics (*.svg)
Printer Command Language (*.pcl, *.prn, *.prt)
Macintosh PICT (*.pct)
HP-GL/HP-GL2 (*.plt)
WordPerfect Graphics (*.wpg, *.vwpg)

7.2.3.1.1.1.3. *Image files*

Bitmap (*.bmp)
JPEG Interchange Format (*.jpeg, *.jpg)
Portable Networks Graphics (*.png)
ZSoft PC Paint (*.pcx)
Tagged Image File Format (*.tiff, *.tif)
Adobe Photoshop (*.psd)
Truevision (*.tga)
Windows Meta File (*.emf, *.wmf)

Printing high resolution images

Print

File -> Print... Prints the 3D view as a high resolution image. Alternatively use **ctrl-P**

Tip: To change the resolution of the printed image, hold down the shift key while selecting print. Enter the magnification in the dialog that is opened.

Export

File -> Export image... Captures the 3D view to an image file of format *.jpeg, *.jpg, *.png, *.tiff, or *.bmp.

Tip: To change the resolution of the exported image, hold down the shift key while selecting export image. Enter the magnification in the dialog that is opened.

Export high resolution image

Right mouse button menu -> Export high res. image... Captures the 3D view to an image file of format *.jpeg, *.jpg, *.png, *.tiff, or *.bmp. First, **set the magnification in the dialog that is opened.**

Changing appearance

Change background color

Right mouse button menu -> Background color -> Custom... Opens the Color dialog where the background color is set. Use **Right mouse button menu -> Background color -> Default** to set the background color back to the default. Alternatively use **Show button -> Background color -> Custom...** and **Show button -> Background color -> Default**

Wireframe

Right mouse button menu -> Wire frame Shows the 3D model with lines and without any surfaces. Alternatively use **Show button -> Wire frame**

Stereo image

Show button -> Stereo image Shows the 3D view in stereo for viewing with red/cyan glasses.

7.2.4. Simulation tab

7.2.4.1. Simulation tab

Requested output

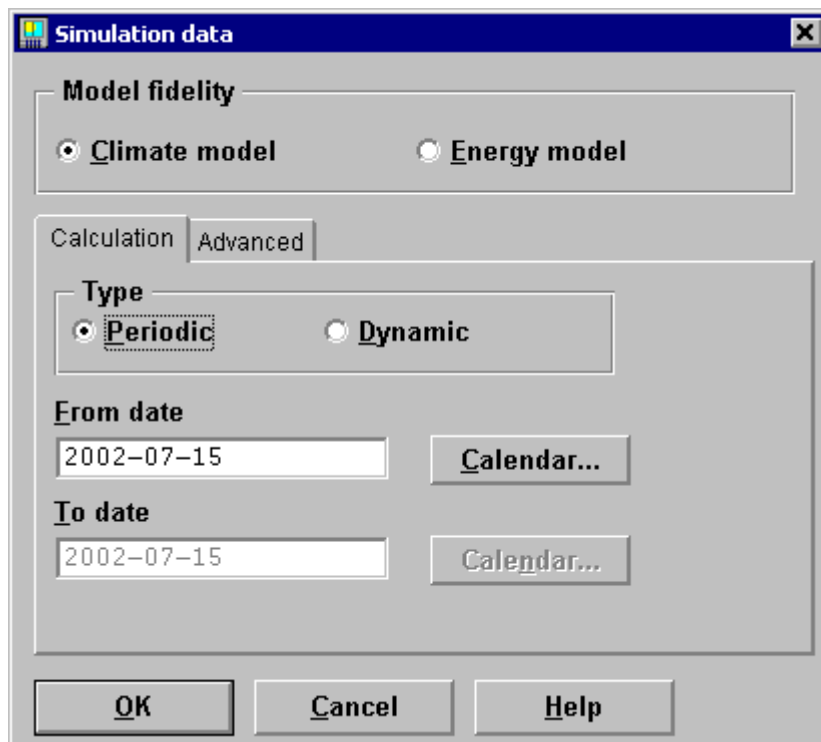
Specifies what diagrams and reports that will be created during the simulation.

Simulation data

Open Simulation data dialog to select simulation times and tolerances, etc.

After a simulation, the result diagrams and reports chosen can be found under the Results tab (see Dialog for choice of output).

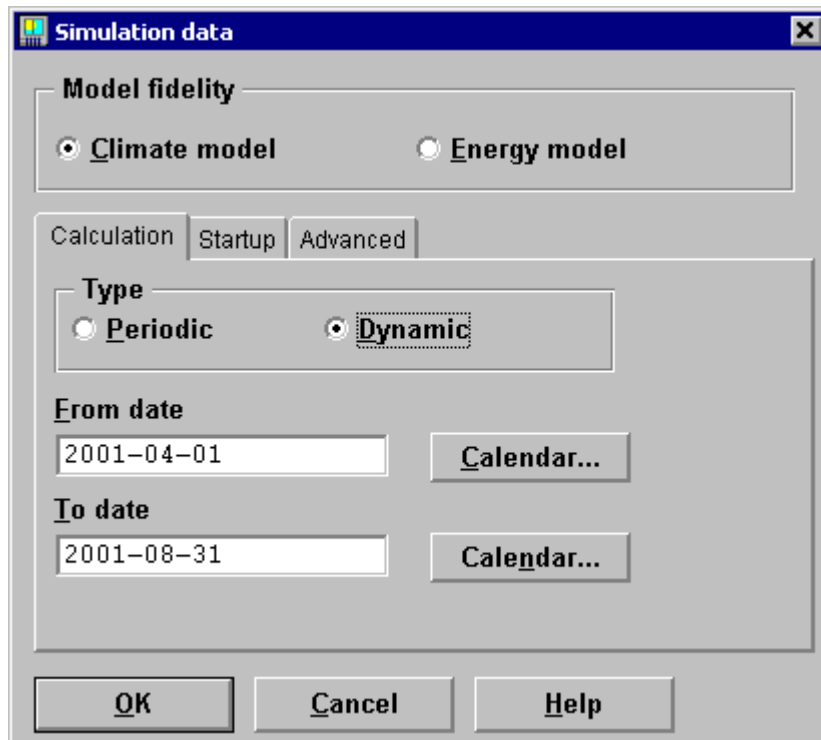
7.2.4.2. Choice of Simulation data



The simulation data object is used to define Custom simulations and to change solver parameters for all types of simulations. Custom simulations should be defined when any of the pre defined standard simulations Heating, Cooling or Energy are inappropriate, for example when only a month should be simulated.

In the Calculation tab, select between a periodic and a dynamic simulation. A periodic simulation means a certain period is simulated a number of times until the system has stabilized and no longer changes from simulation to simulation (a periodic state).

A dynamic simulation means that the simulation starts at a particular time and ends at another time. Both these times are indicated in the same way as the date for periodic simulation. How the simulation is to be initialized must also be indicated for a dynamic simulation. For this reason, another tab, named Startup, is added to the Simulation data dialog.



The startup phase can also be periodic or dynamic. Make the selection in the same way as above. A periodic startup phase means that the selected period is simulated a number of times until the system has stabilized. A dynamic startup phase means that a selected number of days are simulated before the proper simulation starts.

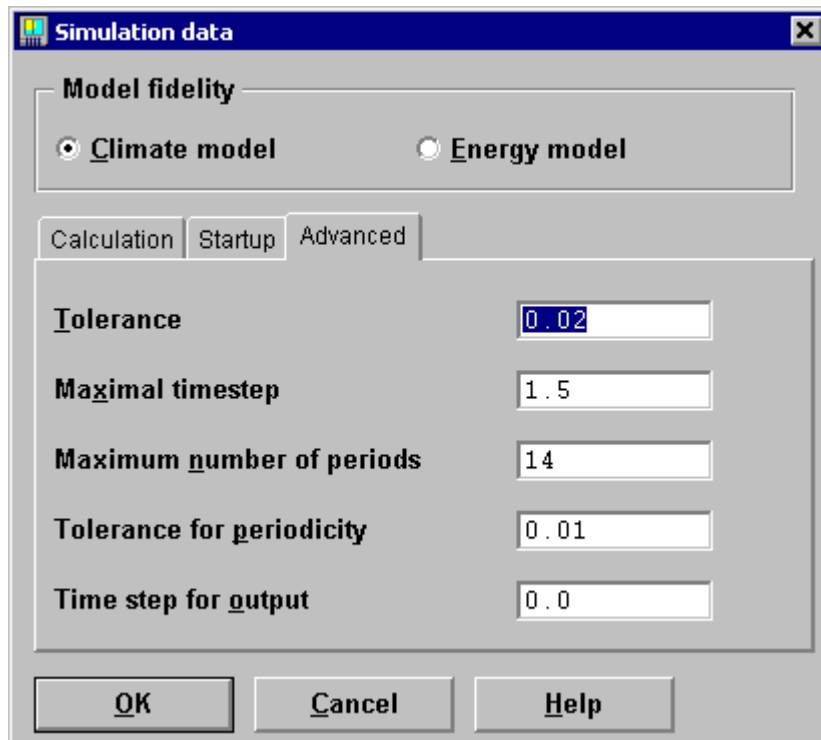
How long the startup phase should be depends on how heavy the building is and on its contact with the ground. The heavier the building, the longer startup phase is required. Approximately two weeks should however always be enough for most buildings. In both cases, no intermediate results are saved from the startup phase.

Output step If greater than zero, recorded times in output files will have this time step (h).

If a Time step for output has been given, result files will be interpolated to have a fixed time step, enabling for example Excel comparison of results between different runs. Simulation time step is still variable.

If a Time step for output is zero (or left blank), the time step of output will be same as solver timesteps, i.e. not equidistant in time.

Note that result files will contain instantaneous values of measured variables, i.e. not integrated values over the given step. In order to obtain average, e.g. hourly, results, use Table tab of result diagrams instead.



Integration parameters can be selected under the Advanced Tab in the dialog. See also the manual for description of these parameters.

Field descriptions, etc:

Tolerance This number determines how accurately equations are to be solved (-)

Maximal timestep The largest timestep to be taken (h)

Maximum number of periods Maximum number of periods in a periodic simulation.

Tolerance for periodicity Tolerance for periodicity in a periodic simulation (-)

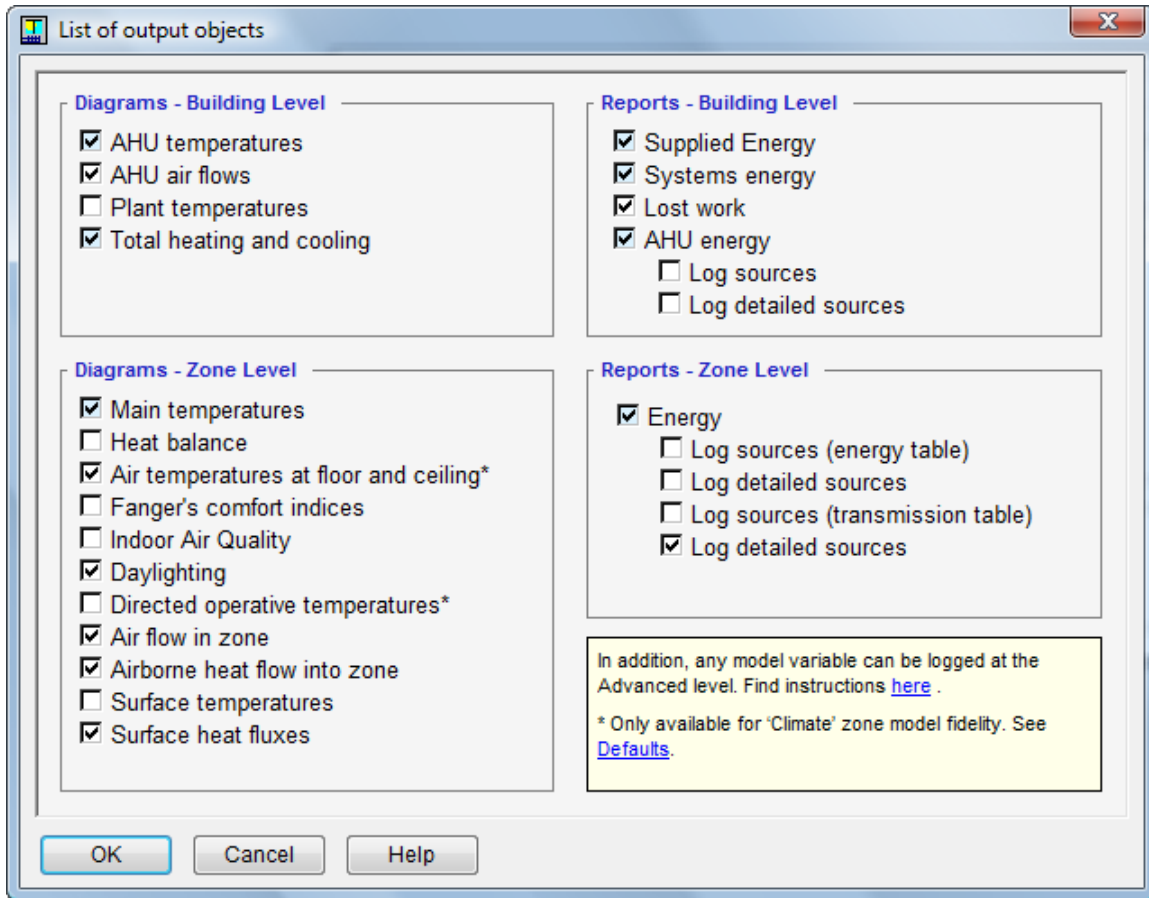
The tolerance given is the degree of accuracy reached in the calculated variables. It is absolute for small quantities (normally < 1) and relative for larger quantities (normally >1). The precise definition of Tolerance is given in the documentation for IDA Solver, which can be obtained from EQUA. Relaxing the tolerance should normally result in a faster simulation. However, if the tolerance is relaxed too much, the solver will have greater trouble to find a solution at all and this in turn costs execution time and causes poorer stability.

For a dynamic simulation, the solver will select the time step up to the Maximal timestep specified. Allowing very long timesteps may create stability problems for the solver that in turn cost time to resolve.

In the case of a periodic simulation, the number of periods can be limited and a tolerance chosen for the relative changes required from day to day, for the simulation to be considered periodic.

See also Topic note on timesteps at the ICE User Support web page (Help menu).

7.2.4.3. Dialog for choice of output



Output from a simulation is presented as diagrams and reports. There are several different diagrams to choose from. The selection is made from the Choice of output dialog that is accessed by clicking on the Requested output button in the building form or if Choose output is selected in the Tools menu. The following diagram and reports exist (those marked with # are pre-selected but can be deleted):

Diagrams

Building level:

- AHU temperatures(#) Air temperatures in central AHU
- AHU airflows Air flows through central AHU
- Plant temperatures Plant temperatures: Boiler & chiller, in & out
- Total heating and cooling (#) Heat and cooling supplied by plant and ideal room units

Zone level:

- Main temperatures(#) Air and operative temperatures
- Heat balance Detailed heat balance for zone
- Air temperatures at floor and ceiling Air temperatures at floor and ceiling in case of displacement ventilation.
- Fanger's comfort indices Fanger's comfort indices, PPD, PMV
- Indoor Air Quality Indoor air quality measures: Ac/h, CO2 level, humidity
- Daylighting Daylight level at desktop (1st person)
- Directed operative temperatures
- Air flow in zone Air flows in zone, in/out through internal/external walls or mechanical ventilation
- Airborne heat flow into zone Airborne net heat flows into zone, through internal walls, envelope and mechanical ventilation

Surface temperatures Temperatures of zone surfaces
 Surface heat fluxes Convective and long wave radiative heat flux of zone surfaces

Reports

Building level:

- Supplied energy Totals of energy, purchased or locally generated, including cost, CO2 and primary energy
- Time series of underlying measurements can be logged by logging sources
- Systems energy Overview of energy transferred by HVAC systems
- Lost work Account of work hours lost due to over or under heating
- AHU energy Energy transferred by individual Air Handling Units
- Time series of underlying measurements can be logged by logging sources

Zone level:

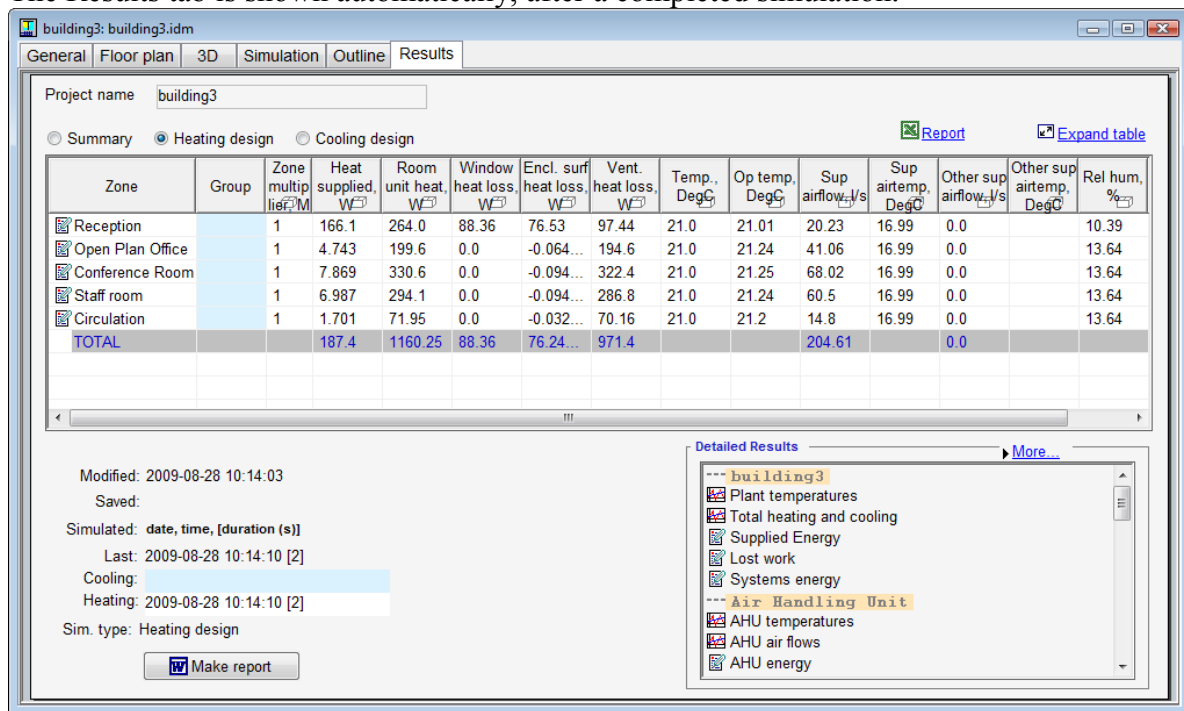
- Energy Zone sensible energy balance
- Time series of underlying measurements can be logged by logging sources

7.2.5. Outline tab

7.2.6. Results tab

7.2.6.1. Simulation Results

The Results tab is shown automatically, after a completed simulation.



Field description:

Detailed results

A list of all result objects requested before the simulation.

Modified

The time when the model was last changed. Only shown if the model was changed after the latest simulation.

Simulated

The time when the latest simulation was run.

Saved

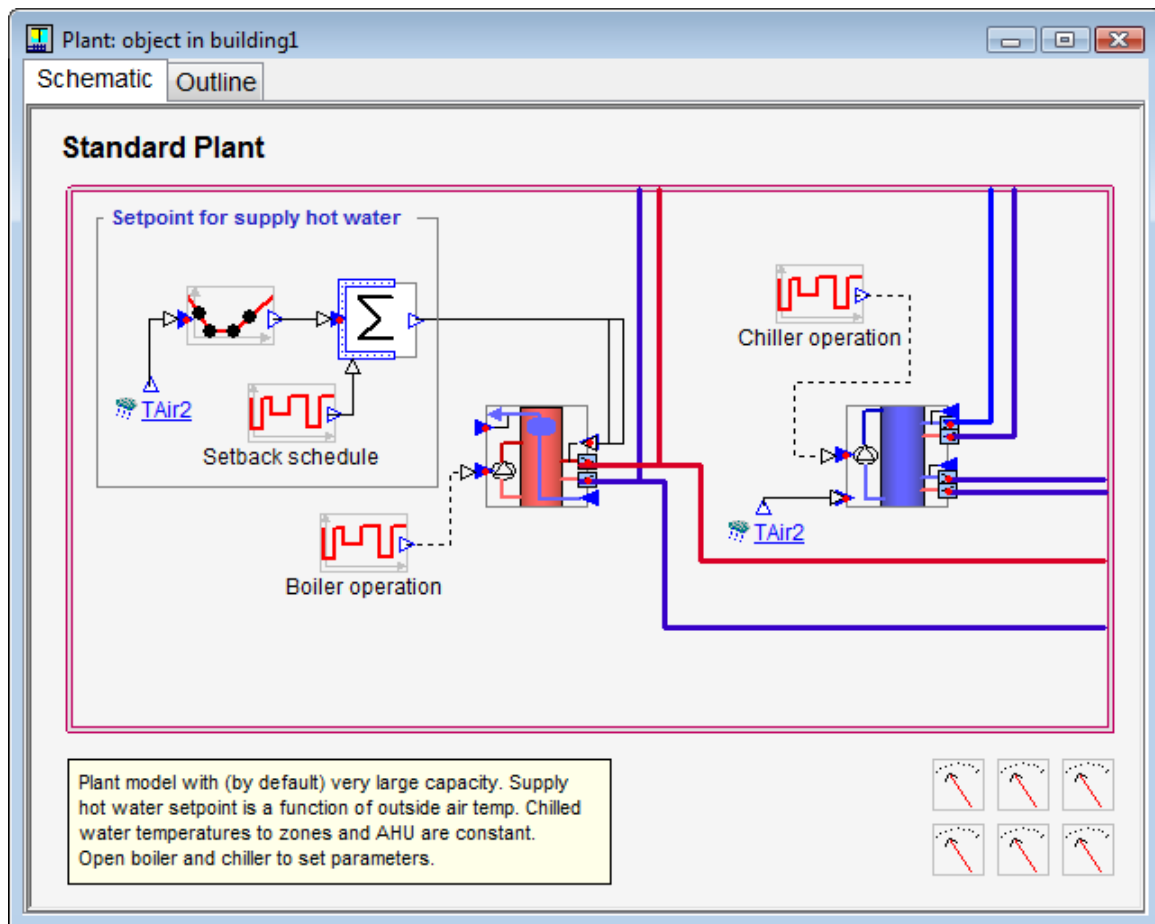
The time when the case was last saved on disk.

Make report

When pressed, the results are combined into a single Word document. Implemented only for Word 2000 or later.

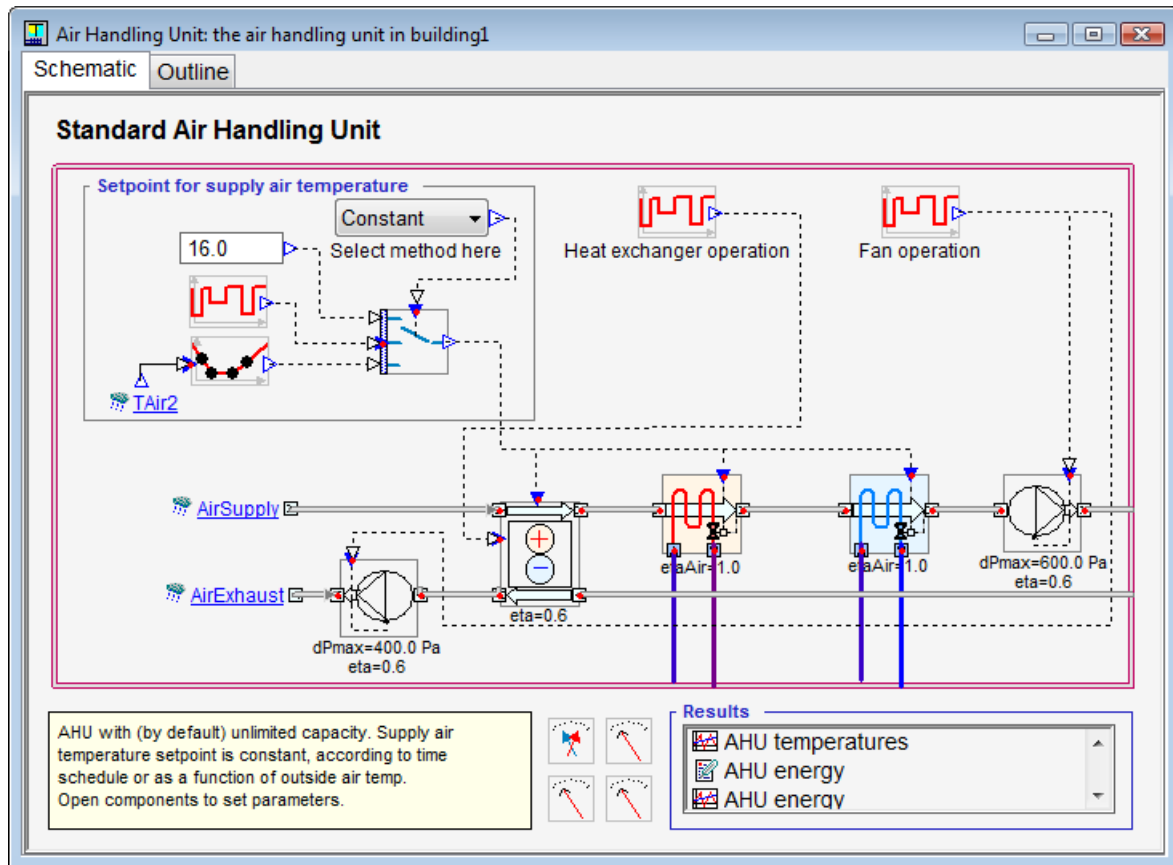
7.3. HVAC Systems

7.3.1. The Primary system



For documentation, see the section on Primary system in the manual.

7.3.2. Air handling unit



Read about the Air handling unit in the manual.

7.3.3. Form for Heating Coil

hc: a mathematical model in demo02.Air Handling Unit

General | Outline | Code

close Object name **hc** Object type **HCSIMCTR**

Simplified heating coil model with air temperature control

Variables

TAIRIN	20.0	Deg-C	Temp. of entering air
TAIROUT	25.0	Deg-C	Temp of leaving air
TLIQIN	30.0	Deg-C	Temp of entering liquid
TLIQOUT	29.0	Deg-C	Temp of leaving liquid

Parameters

ETAIR	1.0	dimless	Air side effectiveness at capacity
DTLIQ	20.0	Deg-C	Liq side temperature drop

Interfaces

AIRFLOWIN <---> hx.SUPOUT
AIRFLOWOUT <---> cc.AIRFLOWIN
LIQFLOWIN <---> AHU_Sup_Hot
LIQFLOWOUT <---> AHU_Rtn_Hot
TEMPSETPOINT <---> SupSwitch.OUTSIGNALLINK[2]

The figure illustrates the form used for an object of the Heating Coil type. A heating coil is one of the parts in the air handling unit. The form is most easily opened by double clicking on the symbol for heating coil (hc) in the air handling unit's Schema.

Field description, etc:

ETAIR Air side effectiveness at capacity (-)

DTLIQ Liquid side temperature drop (°C)

The heating coil has two important parameters, ETAIR, the air side temperature efficiency and DTLIQ, the desired waterside temperature reduction. Capacity control is achieved by adapting the actual efficiency up to the given maximum efficiency level. The necessary water flow is calculated and the water temperature is reduced, if possible, by the desired number of degrees. There is no bypass on the liquid side; control is achieved by simply restricting the water flow.

In the default configuration, the temperature efficiency is set at 1.0. There are two situations when it may be desirable to change this to a more realistic value: for coil sizing by means of simulation experiments, and when making energy calculations in cases where the boiler efficiency is dependent on temperature conditions. In addition, the simplest and quickest way of removing the entire coil is to set the efficiency to zero.

7.3.4. Form for cooling coil

cc: a mathematical model in demo03.Air Handling Unit

General | Outline | Code

close Object name **cc** Object type **CCSIMCTR**

Simplified cooling coil model with air temperature control

Variables

TAIROUT	18.0	Deg-C	Temp of leaving air
TAIRIN	18.0	Deg-C	Temp of entering air
TLIQOUT	5.0	Deg-C	Temp of leaving liquid
TLIQIN	5.0	Deg-C	Temp of entering liquid

Parameters

ETAIR	<input type="text" value="1.0"/>	dimless	Air side effectiveness at capacity
DTLIQ	<input type="text" value="5.0"/>	Deg-C	Liq side temp rise

Interfaces

AIRFLOWIN <---> hc.AIRFLOWOUT
 AIRFLOWOUT <---> sf.FANIN
 LIQFLOWIN <---> AHU_Sup_Cold
 LIQFLOWOUT <---> AHU_Rtn_Cold
 TEMPSETPOINT <---> SupSwitch.OUTSIGNALLINK[3]

The figure illustrates the form used for an object of the Cooling coil type. A cooling coil is one of the parts of the air handling unit, which is included in every building created. The form is most easily opened by double clicking on the symbol for the cooling coil (cc) in the air handling unit's Schema.

Field description, etc:

ETA Air side effectiveness at capacity (-)

DTLIQ Liquid side temperature rise (°C)

The cooling coil works in the same way as the heating coil but is mathematically more complicated because air dehumidification is calculated. The given efficiency is defined as (1 - "bypass factor") according to ASHRAE's nomenclature. Physically, this means that the state of the cooled air in the psychrometric chart is on a straight line between the state for the incoming air and the apparatus dew point temperature on the saturation curve.

In the model, the average temperature of the liquid side defines the apparatus dew point. On this line, the given efficiency indicates the status: 0 is no cooling whatsoever, and 1 indicates maximum cooling, which also means that the air temperature may be chilled to (the arithmetical) mean value of the coolant's incoming and outgoing temperatures

7.3.5. Form for Heat Exchanger

hx: a mathematical model in demo02.Air Handling Unit

General | Outline | Code

close Object name **hx** Object type **HXSIMCTR**

Air to air heat exchanger with bypass control and antifreeze protection

Variables

TSUPIN	10.0	Deg-C	Entering supply air temp.
TSUPOUT	15.0	Deg-C	Leaving supply air temp.
TEXHIN	25.0	Deg-C	Entering exhaust air temp.
TEXHOUT	20.0	Deg-C	Leaving exhaust air temp.

Parameters

ETA	<input type="text" value="0.6"/>	dimless	Air side effectiveness at capacity
TEXHOUTMIN	<input type="text" value="1.0"/>	Deg-C	Minimum achievable leaving dry-bulb

Interfaces

```

SUPIN <--- demo02.CLIMATE.AIRSUPPLY
SUPOUT <---> hc.AIRFLOWIN
EXHIN <---> Air_Rtn
EXHOUT <---> rf.FANIN
TEMPSETPOINT <---> SupSwitch.OUTSIGNALLINK[1]

```

The figure illustrates the form used for an object of the Heat Exchanger type. A heat exchanger is one of the parts in the air handling unit. The form is most easily opened by double clicking on the symbol for heat exchanger in the air handling unit's Schema.

Field description, etc:

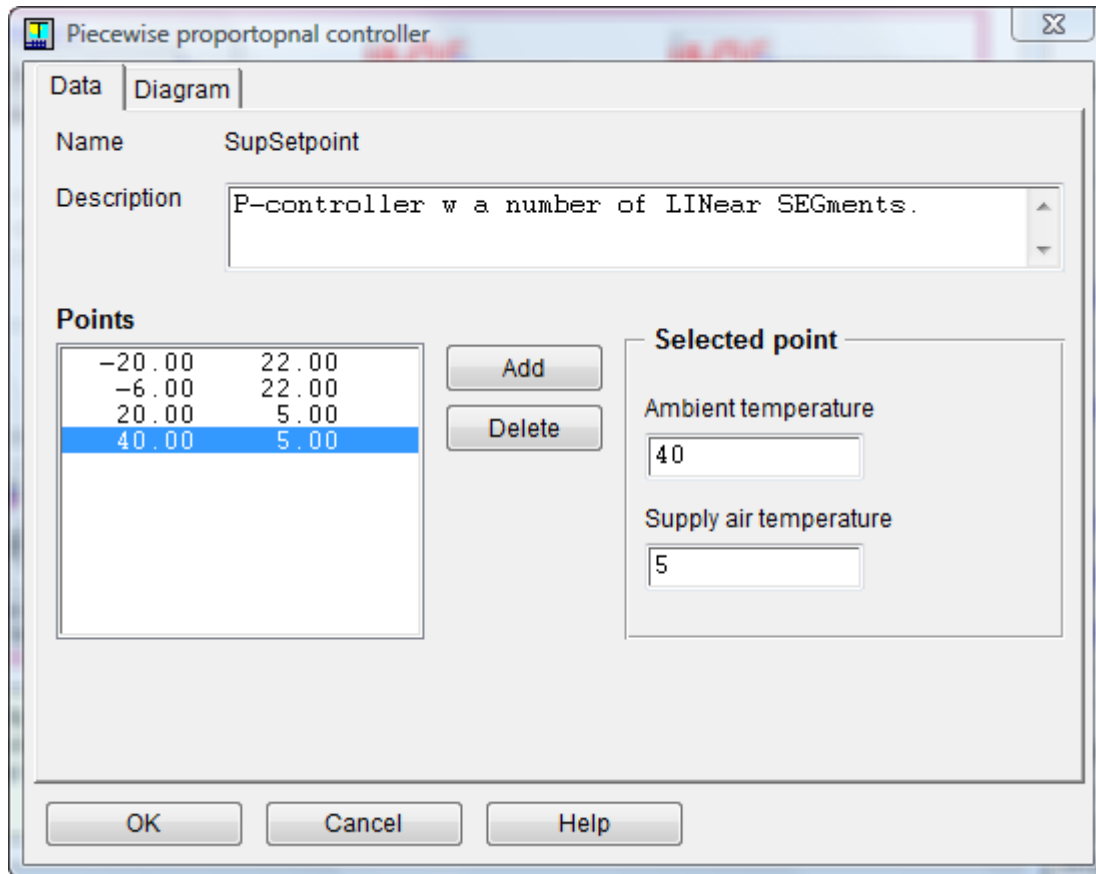
ETA Effectiveness at full capacity (-)

TEXHOUTMIN Minimum allowable temperature of exhaust air stream (°C)

The heat exchanger is controlled by the actual efficiency selected by the model (up to the maximum limit set by the user), so that the setpoint for the supply air temperature is reached, if enough heat is available. The temperature of the (often chilled) exhaust air, which may not fall below a certain level (TEXHOUTMIN parameter in the form), sets another limit. This is to avoid freezing for non-rotating heat exchangers.

The heat exchanger takes into consideration condensation on both the supply and exhaust sides. During wet processes, the given efficiency is interpreted as (1 - "bypass factor") in the same way as for the cooling coil, but the apparatus dew point for the heat exchanger is defined as the incoming temperature for the opposite medium.

7.3.6. Edit temperature depending input



Certain input in IDA Indoor Climate and Energy depends on outdoor air temperature. Under the Data Tab in the piecwise proportional controller dialog, a number of points can be given to define such dependency. As an example, the Dialog box is opened in the form for primary system and air handling unit by double clicking on the symbol with the following appearance:



Field descriptions, etc:

Description Object description.

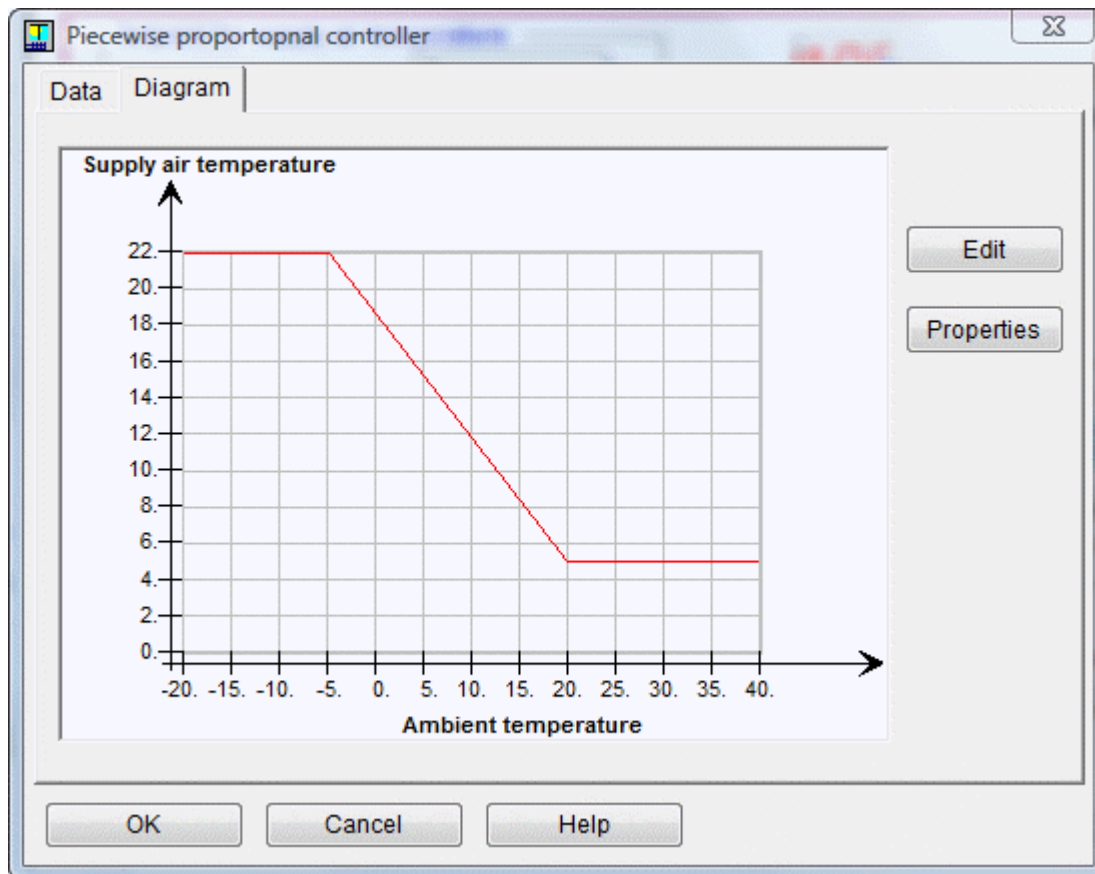
Points

Add

Delete

Selected point: Ambient temperature

Selected point: Supply heating water temp



Under the Diagram Tab, the temperature dependency is presented in the form of a curve. This can be edited, which is done as a polyline. When the curve is edited, the points under the Data Tab are then corrected.

Field descriptions, etc:

Edit

Properties

A polyline consists of line segments and break points, the latter marked by small rectangles. In this case, it can be edited in three ways after double clicking on it or clicking on the Edit button:

1. Its break points can be dragged to the desired position.
2. A new break point can be inserted by clicking on or close to the line.
3. An existing break point can be deleted by clicking on it.

To end editing, click once with the Right button and select OK. The curve can then be examined before leaving the dialog. Alternatively, the curve can be edited by defining points under the Data Tab.

7.4. Zone

7.4.1. Zone form

This form presents an object of the Zone type. It is most easily opened by double clicking on the name of a zone in the Zones list box in the building form or by double clicking on a zone in the Floor plan.

Field descriptions, etc:

Number of zones of this type Number of rooms with these conditions. Central consumption of air, water and electricity is increased by this factor.

Loss factor for thermal bridges Direct heat transfer between room air and outside air, Watts per degree (C) temperature difference. (W/°C). This parameter is normally given by values entered under Thermal bridges in the building form.

Controller setpoints Choice of control values for temperature, air quality and light .

Air

Select air handling unit Choice of air handling unit for this zone. First define systems at the building level.

More... Click here to add multiple central or local air-handling units that serve this zone. If the zone already contains multiple AHU's, this link is marked with red asterisk. The fields described below are marked in the same way if the field's value is different in additional AHU's.

System type Type of ventilation system. Constant or variable flow, with different controls. To create an exhaust only or supply only system - or no mechanical system at all – select CAV and set appropriate flows below to zero.

Supply air for CAV Mechanical supply air flow for CAV systems (l/s m² floor area). VAV flow is given under Controller setpoints. Clicking on the link "l/s m²", opens a dialog where the flow can be specified in alternative ways (l/s or m³/h for entire zone, m³/h/m², or air changes per hour).

Return air for CAV Mechanical return air flow for CAV systems (l/s m² floor area). VAV flow is given under Controller setpoints. Clicking on the link "l/s m²", opens a dialog where the flow can be specified in alternative ways (l/s or m²/h for entire zone, m³/h/m², or air changes per hour).

Displacement degree for gradient calculation From 0 (well-mixed) to 1 (full displacement ventilation), or a negative value for a fixed temperature gradient given by the user.

Leak area ELA Equivalent leakage area in the envelope, defined at 4 Pa, Cd=1 and, 1 m above floor. Of importance only when other leaks, e. g., internal openings, also exist. (m²). This parameter is normally given by values entered under Infiltration in the building form.

Given additional in/exfiltration Fixed flow, independent of pressure. Leak area should be small (but nonzero) when this option is used. This parameter is normally given by values entered under Infiltration in the building form. Read more about infiltration modeling in that form.

Geometry

Room height: to ceiling Distance between the floor surface and the ceiling (m). The ceiling or some parts of the ceiling may be below this level if the building's roof is too low to fit requested zone height. If "to roof" is selected, the field "to ceiling" shows the height level where the lights and active beams are located.

Room height: to roof Select this to extend the room's ceiling to the building's roof.

Floor height above ground Distance between the floor surface and the ground (m).

Floor plan Open to see this zone from the Floor plan view.

Enclosing surfaces By default the zone is rectangular with six surrounding surfaces, called main surfaces. When an alternative geometry is defined, the main surface names will be extended with a letter suffix (e.g. wall 1 can be split into wall 1a and wall 1b). Windows, doors, and radiators, etc are introduced as objects on the corresponding zone surfaces.

Room units Shows the list of cooling and heating units in zone. The ideal cooling and heating devices may be added here; other devices may be added to zone surfaces.

Loads Full list of zone loads. Add equipment, light, and occupant loads here.

Internal masses Extra thermal mass, i.e. mass in addition to that of zone partitions and floors.

Read more about the zone models and these parameters in Chapter ??? of the manual <<can we link to pdf??>>.

7.4.2. Calculation of thermal bridge coefficients

The total loss factor in a zone's thermal bridges is calculated as sum of loss factors in bridges created by different construction elements listed in the form.

The coefficients per unit of element size (in most cases per meter) are by default given at building level.

The sizes of elements are by default calculated from zone geometry, but may be also given by the user.

The values specified explicitly by the user are marked with yellow background. To restore the default setting, click the yellow field with right mouse button and select **Mapping | Restore link** from the context menu.

The user may also specify an extra loss for this zone, independent of given geometry.

See also

Thermal bridge coefficient



Loss factor for thermal bridges: object in building1.Zone



Calculation of thermal bridge coefficient [W/K]

External walls	<input type="text" value="0"/>	*	<input type="text" value="6.5"/>	m ²
External wall / internal slab	<input type="text" value="0.05"/>	*	<input type="text" value="2.5"/>	m
External wall / internal wall	<input type="text" value="0.03"/>	*	<input type="text" value="5.2"/>	m
External wall / external wall	<input type="text" value="0.08"/>	*	<input type="text" value="0"/>	m
External windows perimeter	<input type="text" value="0.03"/>	*	<input type="text" value="4.9"/>	m
External doors perimeter	<input type="text" value="0.03"/>	*	<input type="text" value="0.0"/>	m
Roof / external walls	<input type="text" value="0.09"/>	*	<input type="text" value="0.0"/>	m
External floor / external walls	<input type="text" value="0.14"/>	*	<input type="text" value="0.0"/>	m
Balcony floor / external walls	<input type="text" value="0.2"/>	*	<input type="text" value="0"/>	m
Extra loss	<input type="text" value="0"/>			W/K

7.4.3. Dialog for controller setpoints

Setpoint collection

Setpoint: Office, normal control - CM

Control Setpoints

	min	max	Unit
Temperature	21	25	°C
Mech. supply air flow			L/(s.m ²)
Mech. return air flow	0.3	7	L/(s.m ²)
Relative humidity	20	80	%
Level of CO2	700	1100	ppm (vol)
Daylight at workplace	100	10000	Lux
Pressure diff. envelope	10	20	Pa

max heating
max cooling
air temp
temp_throttle = 2.0 °C

The control action of heating and cooling depends on the controller used in the actual device. Defaults are P control for radiators and PI for cooling units

* when both VAV and other means of cooling have been defined, VAV is used first and setpoints of other room units are offset by 2.0 °C. (Change globally in System Parameters)

Variable Setpoints

Min temperature: <value not set>

Max temperature: <value not set>

Object

Name: Office, normal control - CM

Description:

OK Cancel Save as... Help

The dialog for objects of the Controller setpoints type defines all the quality requirements for the zone's climate.

Field descriptions, etc:

Temperature

Min Setpoint for heating controller. (°C)

Max Setpoint for cooling controller. (°C)

Mech. supply air flow

Min Minimum flow at VAV. Recommended minimum airflow. (l/s/m²). When left blank, balanced ventilation is assumed (value is taken from field below)

Max Maximum flow at VAV. Recommended maximum airflow. (l/s/m²) When left blank, balanced ventilation is assumed (value is taken from field below)

Mech. return air flow

Min Minimum flow at VAV. Recommended minimum airflow. (l/s/m²)

Max Maximum flow at VAV. Recommended maximum airflow. (l/s/m²)

Relative humidity

Min Level of humidity below which minimum VAV flow is kept. Only for VAV with humidity control. (%)

Max Level of humidity above which maximum VAV flow is kept. Only for VAV with humidity control. (%)

Level of carbon dioxide

Min Level of carbondioxide below which maximum VAV flow is kept. Only for VAV with CO2 controlling. (ppm (vol))

Max Level of carbondioxide above which maximum VAV flow is kept. Only for VAV with CO2 controlling.. (ppm (vol))

Light at workplace

Min Light intensity at the working surface below which a maximum of electric light is turned on. (Lux)

Max Light intensity at the working surface above which the electric light is fully off. (Lux)

Pressure diff. envelope

Used for VAV with pressure control (normally return air only). Used to extract air in VAV systems when air is supplied in other zones by VAV with other type of control.

Min Pressure difference over zone envelope, when minimum amount of air should be extracted. (Pa)

Max Pressure difference over zone envelope, when maximum amount of air should be extracted. (Pa)

Object: name and description

The dialog for controller setpoints is opened by the Right button menu with the cursor over an object of the Controller setpoints type, (e.g. Office, normal standard) in the Controller setpoints field in the zone form or in the form for IDA-resources.

The setpoints specified in the dialog are used by controllers of the zone climate.

See also the IDA ICE manual

7.4.4. Operative temperatures

The operative temperature is the average of the air temperature and a radiation temperature at a certain point. This is the temperature which is felt by a person in a room. It is calculated for each occupant load taking account of its position in the room. When an occupant load is first inserted it is given a default position in the middle of the room, with a center of gravity (sitting) at 0.6 m above the floor. This position can be changed by opening the surface editor for the floor.

To learn about the operative temperature at a point without introducing an additional thermal load, the number 0 is given in the field for Number of people in the form for the Occupant load.

Directed operative temperatures, in six directions, are available in a zone using the Climate model fidelity.

7.4.5. Form for occupant load

Occupant: a group of occupants in building3.Reception

Number of people in group

Schedule Always present

Activity level MET

Clothing

Constant ± * CLO

Schedule

[*clothing is automatically adapted between limits to obtain comfort]

Object

Name **Occupant**

Description

Every object of the Occupant load type (zero or more people) inserted in a zone (see Insert object) is edited in this form.

Field description, etc:

Number of people Number of people in the group.

Schedule Schedule for the presence of the group. Smoothing applied by default. The output signal must be within [0, 1].

Activity level Activity level according to Fanger (met)

Clothing Degree of clothing according to Fanger. (clo)

Object: Name and description

See below for a more detailed description of met and clo:

The form for an occupant load is opened e.g. by double clicking on an object of the Occupant load type (e.g., Occupant load 1) in the Loads box in the zone form.

The occupant load's position on the floor in the zone can be changed in the Surface editor (for the floor). An occupant load is indicated by a chair icon. Several occupant loads can occur in the same zone. When the Climate zone model has been selected, operative temperature will depend on the position.

Measuring points for operative temperature coincide with occupant loads. To follow the operative temperature at a certain point without loading the zone, select 0 for the number of people or the "always off" schedule.

Selecting activity level (met)

The activity levels and the amount of clothing, defines how much heat (sensible and latent) and carbon dioxide a person emits. 1 met corresponds to 58.2 W per m² body surface, which is the amount one sitting, inactive person is assumed to emit. In IDA Indoor Climate and Energy, body surface has been selected to be 1.8 m², corresponding to an average adult. The examples below give the emitted total power and activity levels during different activities, which are assumed to be continuous. (Source: ASHRAE Fundamentals).

<i>Activity</i>	<i>met</i>	<i>Power</i> <i>[W]</i>
Sleep	0.7	72
Reclining rest	0.8	81
Seated quiet resting	1.0	108
Standing, relaxed, rest	1.2	126
Walking (3.2 km/h)	2.0	207
Walking (4.3 km/h)	2.6	270
Walking (6.4 km/h)	3.8	396
Reading, seated	1.0	108
Typing	1.1	117
Office walking	1.7	180
Office lifting/packing	2.1	216
Cooking	1.6-2.0	171-207
Housecleaning	2.0-3.4	207-360
Light machine work	1.8-2.4	189-252
Heavy machine work	4.0	423
Pick and shovel work	4.0-4.8	423-504
Dancing, social	2.4-4.4	252-504
Aerobics/work out	3.0-4.0	315-423
Tennis	3.6-4.0	378-486
Basketball	5.0-7.6	522-792
Competitive wrestling	7.0-8.7	738-909

How to select clothing (clo)

The amount of clothing has a large influence on the comfort experienced, which is measured with PPD (percentage of people dissatisfied) and PMV (predicted mean vote). It also has some influence on the power emitted by a person. It 1 clo equals a heating resistance of 0.155 m²K/W. The examples below give the clo-values for different types of clothing (source: ASHRAE Fundamentals).

Comfort measures such as PMV and PPD are only meaningful if people are appropriately dressed, otherwise they will nearly always feel too hot or cold (just like in real life). In practice, it is quite time consuming and difficult to construct schedules that provide a clothing for a whole-year simulation. Therefore, IDA ICE has from version 4 been equipped with a simple method for adapting clothing to sensed PMV.

Upper and lower bounds on clo are given by the user, by providing a clo tolerance in the occupant load form. The lower bound should represent the socially acceptable lower clothing limit in the given environment (perhaps in a casual office, 0.6) and the upper, the culturally accepted winter attire, say 1.1. The algorithm simply treats the occupant as a proportional

controller, when PMV is at the lower bound (by default -1), the person will wear maximum clothing and similarly on the hot side, minimum clothing will be worn at the upper PMV bound (default +1). PMV limits can be selected in System Parameters<<link>> on the building form.

In reality, studies show that people in an office environment will decide how to dress in the morning, mostly based on outdoor conditions, and will not continuously adapt their clothing during the day. It is therefore not realistic to set the PMV bounds too close to zero (approaching ideal control).

<i>Clothing</i>	<i>Clo</i>
Shorts, short-sleeve shirt	0.36
Trousers, short-sleeve shirt	0.57
Trousers, long-sleeve shirt	0.61
Same as above, plus suit jacket	0.96
Same as above, plus vest and T-shirt	1.14
Trousers, long-sleeve shirt, long-sleeve sweater, T-shirt	1.01
Same as above, plus suit jacket and long underwear bottoms	1.30
Sweat pants, sweat shirt	0.74
Knee-length skirt, short-sleeve shirt, panty hose, sandals	0.54
Knee-length skirt, long-sleeve shirt, full slip, panty hose	0.67
Knee-length skirt, long-sleeve shirt, half slip, panty hose, long-sleeve sweater	1.10
Same as above, replace sweater with suit jacket	1.04
Ankle-length skirt, long-sleeve shirt, suit jacket, panty hose	1.10
Long-sleeve coveralls, T-shirt	0.72
Overalls, long-sleeve shirt, T-shirt	0.89
Insulated coveralls, long-sleeve thermal underwear, long underwear bottoms	1.37

7.4.6. Form for Lights

Light: the lights in building3.Reception

Number of units: 1

Control strategy: Schedule

Schedule: © Always on

Rated input per unit: 50.0 W

Luminous efficacy: 12 lm/W

Convective fraction: 0.3 0-1

Energy meter: [Default] Lighting, facility

Object

Name: Light

Description:

Field description, etc:

Number of units Number of lighting units. Total power emitted is the rated input times this number.

Control strategy Operation method for lighting. Only 'Schedule' at present.

Schedule Operation schedule for lights. Smoothing applied by default. The output signal must be in the interval [0, 1]

Rated input per unit Consumption of electrical power when lights are on. (W)

Luminous eff. Number of lumen emitted per watt supplied electrical power. (lm/W)

Convective fraction Fraction of rated input emitted as convective heat. (0-1)

Energy meter

Object: Name and description

The form for a light is opened e.g. by double clicking on an object of the Lights type in the Loads box in the zone form.

The light's extension and position on the zone's ceiling can be changed in the Surface editor (for the ceiling), where a light is marked with a specific symbol. At the moment, only one light source is allowed in a zone. Typically, the size and location of the light has but a small influence on the room climate, except when a large output is emitted from a small surface. To avoid this, the light can be extended over a major part of the ceiling.

If the Control strategy is set to Setpoints+Schedule, the light output is varied depending on available daylight, during the periods when the selected schedule is on (> 0). If the daylight falls below a min level, set in the Controller setpoints dialog, the light output is set at nominal power. If the daylight exceeds the max level, the lights are off. Between these limits, the control is proportional.

7.4.7. Form for equipment load

Equipment: a set of equipment units in building3.Reception

Number of units: 1

Schedule: © Always on

Emitted heat per unit: 150.0 W Only this consumes electrical energy

Long wave radiation fraction: 0.0 0-1 Emitted as water vapor, i.e. the evaporation heat is not removed from the air

Moisture emission per unit: 0.0 kg/s

CO2 per unit: 0.0 mg/s

Energy meter: [Default] Equipment, facility

Object

Name: Equipment

Description:

Field description, etc:

Number of units Number of equipment units with these data.

Control strategy Control method.

Schedule Operation schedule. Smoothing applied by default. The output signal must be in the interval [0, 1].

Emitted heat per unit Total emitted sensible heat. (W)

Long wave radiation fraction Share of sensible heat that is emitted as long wave radiation. Radiation is distributed according to wall surface areas. (0-1)

Moisture emission per unit Water vapor emitted by the device. (kg/s) Vaporization heat is not accounted for.

The vapor is regarded to have the same temperature as the zone air.

CO2 per unit Carbondioxide emitted by the device. Note the unit! (mg/s)

Energy meter

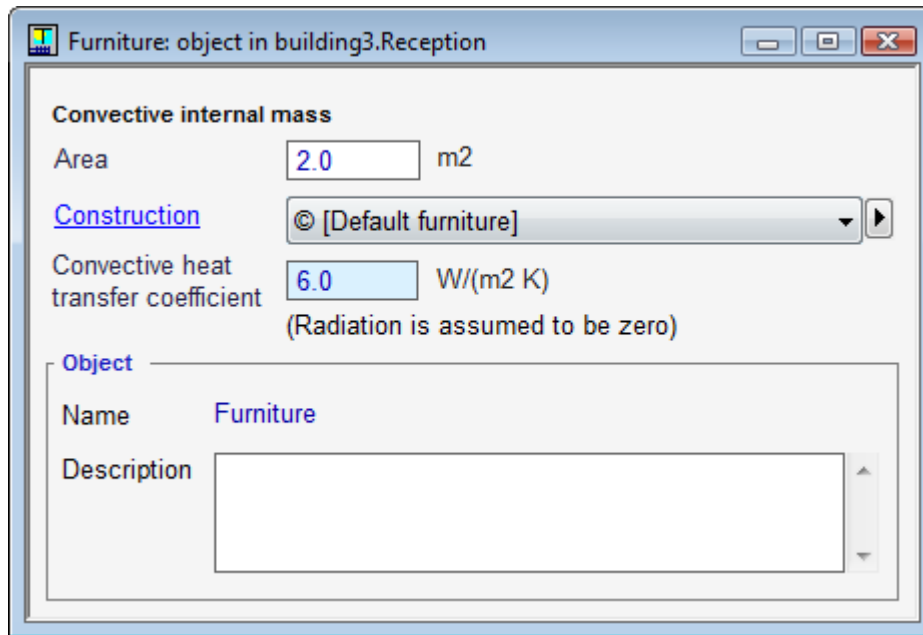
Object: Name and description

The form for an equipment load is opened e.g. by double clicking on an object of the Equipment load type in the Loads box in the zone form.

Several objects of the Equipment load type can be in one zone. Their placing in the zone is not considered to influence the results.

7.4.8. Convective internal mass

Objects of this type, inserted in a zone (see Insert object), are always edited in this form.



Field description, etc:

Area

Area per side (object is described as two-sided wall) [m2].

Construction

Approximate the object by wall-type construction.

Heat transfer coefficient

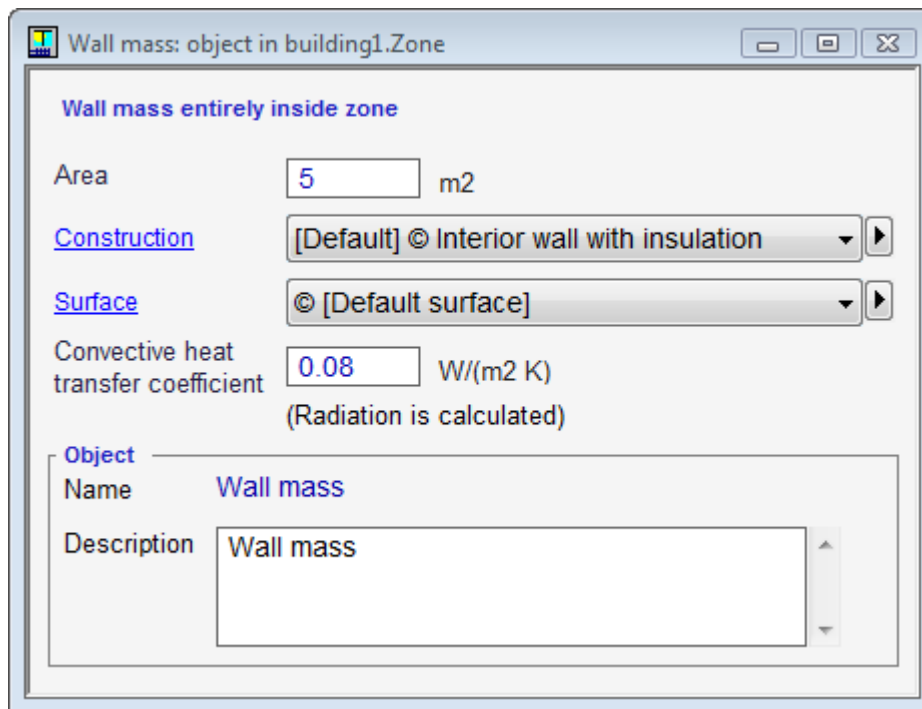
Convective heat transfer coefficient (no long-wave exchange is calculated) [W/(m2 K)].

This type of object is used to model internal masses that primarily interact with the zone air, and that are "seen" by the walls only to a minor extent. Examples could be store room shelves, furniture, plants, etc. These masses can have a relatively large influence on fast temperature variations but can normally be neglected in energy calculations.

The area should give an estimate of the total area exposed to the zone air. Note that all objects are treated as double sided, e.g. for a desk the top side area should be given. The construction specified should correspond to a section through the object and several material layers can be specified in the same way as for a wall.

7.4.9. Thermal mass

Objects of the type Thermal mass, inserted in a zone (see Insert object), are always edited in this form.



Field description, etc:

Area

Area per side of two-sided wall [m2].

Construction

Construction (order between sides is immaterial)

Heat transfer coefficient

Convective heat transfer coefficient (long-wave heat transfer is calculated by ICE) [W/(m2 K)].

This type of object is used to model masses like internal walls and floor slabs that are contained inside the zone, are "seen" by the zone walls, and thus partake in the radiation balance.

The given area should provide a figure for the area exposed towards the zone surfaces. Note that all objects are treated as double sided; the area of one side is given. The construction specified should correspond to a section through the object.

7.4.10. Ideal cooler

The ideal cooler is a room unit that cools the zone when no detailed information about an actual room unit, such as a fan coil or active chilled beam, is available or this amount of detail is unmotivated. It has no given physical location on any room surface and is not connected to the plant of the building. Physically, think of it as a standalone air conditioner with fixed performance parameters.

An ideal cooler is inserted by default when a new zone is created (unless it has been removed from the zone template). The default capacity of the ideal cooler is given per m² floor area in the zone template and should normally be selected to be large enough to safely be able to cool the zone under all conditions. A PI controller will then be used to keep the room air (or operative) temperature at the cooling setpoint (as specified in the Setpoint collection.)

In order to study the performance of a specific type of room unit, or a room unit which is hydronically connected to the plant, remove the ideal room unit from the zone form, and

insert instead a physical room unit to a zone surface or, alternatively, drag in a fan coil or similar device from the palette to the zone form.

7.4.11. Ideal heater

The ideal heater is a room unit that heats the zone when no detailed information about an actual room unit, such as a radiator or convector, is available or this amount of detail is unmotivated. It has no given physical location on any room surface and is not connected to the plant of the building. Physically, think of it as a standalone fuel or electric heater with fixed performance parameters and no flue gas emissions.

An ideal heater is inserted by default when a new zone is created (unless it has been removed from the zone template). The default capacity of the ideal heater is given per m² floor area in the zone template and should normally be selected to be large enough to safely be able to heat the zone under all conditions. A PI controller will then be used to keep the room air (or operative) temperature at the heating setpoint (as specified in the Setpoint collection.)

In order to study the performance of a specific type of room unit, or a room unit which is hydronically connected to the plant, remove the ideal room unit from the zone form, and insert instead a physical room unit to a zone surface or, alternatively, drag in a fan coil or similar device from the palette to the zone form.

7.4.12. Local AHU

A local air handling unit works in just the same way as a central air handling unit, except it serves only a single zone and can only be reached from that zone. The heat and air from the unit in the zone is recorded as coming from mechanical ventilation.

A local air handling unit is introduced in a zone by following the More... link in the Ventilation section of the zone form. Then press Add AHU, and select among available local air handling units.

Local AHUs may only operate in CAV, with flows given in the table.

7.4.13. Zone advanced form

Model fidelity 'Default' = choice is done in the Default form at the building level. 'Climate' = better result, gradients included. 'Energy' = faster, no gradients. Selecting 'Climate' or 'Energy' here has priority.

Zone group The group name used for creating group based reports

Air velocity in the occupied zone Air velocity for the calculation of comfort index (m/s).

Default electric meters for equipment and lights

7.5. Walls, Floor and Ceiling

7.5.1. Form for wall, floor and ceiling

Wall 4: an enclosing element in building1.Zone

Surface Advanced Outline

Thermal Connection

The wall is thermally connected with adjacent zone or building face. If there are multiple adjacent objects, the wall is divided in parts.

Ignore adjacency to faces

The parts having no adjacent zone or face are connected as below:

Ignore net heat transmission

Constant temp on other side °C

Similar + offset °C

Connect to face

Connect to ground

Note: If "Ignore net heat transmission" is selected for both ceiling and floor, and neither of them is adjacent to another zone, they are treated as adjacent to each other.

Construction

[For external constructions](#) [Default] © Rendered l/w concrete wall 250 ▾ ▶

[For internal constructions](#) [Default] © Interior wall with insulation ▾ ▶

[Inner surface](#) © Default surface ▾ ▶

[Outer surface](#) © Default surface ▾ ▶

Open the form for walls, floor and ceiling by double clicking on the corresponding object of type Wall, Floor or Ceiling in the zone form, and selecting the Advanced tab.

Field descriptions, etc:

Thermal connection

The wall is thermally connected with adjacent zone or building face. If there are multiple adjacent objects, the wall is divided in parts

Ignore adjacency to faces

If the building shape is too complicated to be fully described in ICE, it may be desirable to explicitly describe the thermal connections between zones and building external surfaces (faces). By checking this box, the automatic detection of adjacent face is suppressed (but the adjacency between zones is still detected).

The parts having no adjacent zone (or face, if not suppressed) are connected as below described in following fields:

Ignore net heat transmission

Normally select this alternative for internal walls. An adiabatic surface is placed in the geometric middle of the wall.

Constant surface temp on other side

Select this only when no adjacent zone is simulated, but heat transmission is significant. A surface temperature is specified on the opposite side of the wall (note: not air temperature). **NB Use with caution! Could introduce significant energy transports and will ruin the building energy balance.**

Similar + offset

When an offset of zero is given, this alternative is roughly equivalent to the Ignore net... option. However, here, adjacency to a similar zone with a constantly different temperature can be specified. **NB Use with caution! Could introduce significant energy transports and will ruin the building energy balance.**

Connect to face:

Manual connection to given face (façade); specify face in adjacent field. The facade itself does not have to adjoin the wall. This allows modeling of buildings with complex shape.

Connect to ground

Use for 'slab on grade' or basement floor; under-slab-construction and ground temperature are given in the Ground properties dialog in available from the main building form

Construction Choice of material and thickness of layers.

For external walls Choice of materials and thickness of layers for the part of the wall that is external.

For internal walls Choice of materials and thickness of layers for the part of the wall that is internal.

Inner surface Description of the optical properties of the inner surface. These are of minor importance in normal cases.

Outer surface Description of the optical properties of the outer surface. Only for building elements facing ambient.

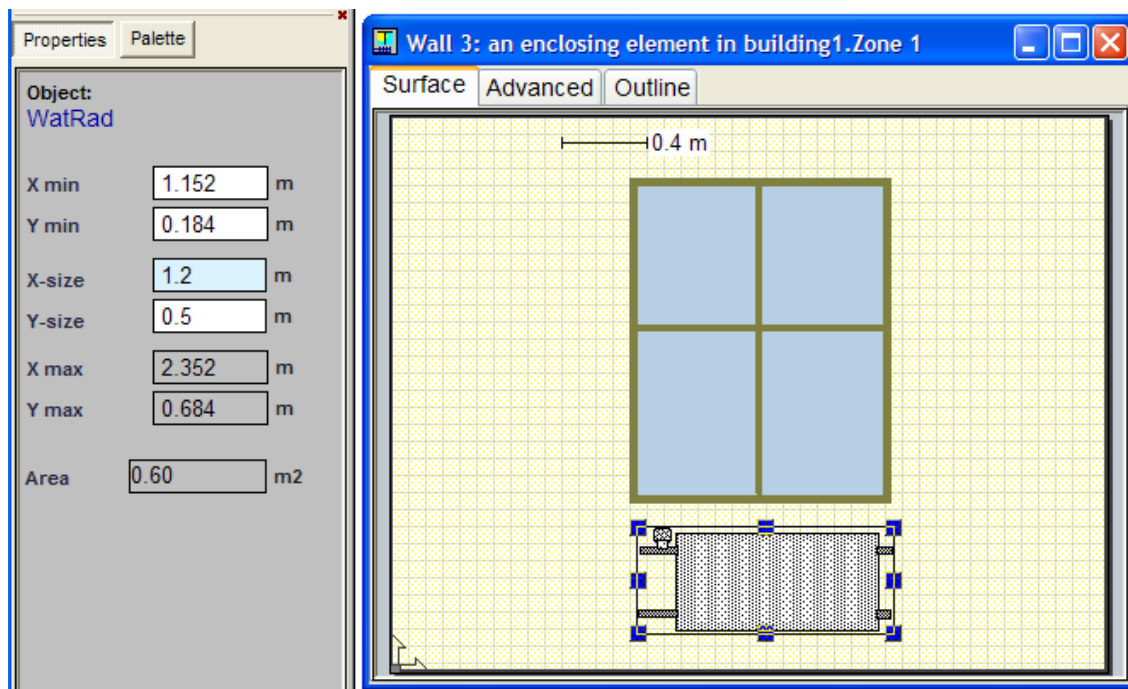
If [Default] is left in the construction fields, the corresponding construction is fetched from the Defaults tab in the building form.

Walls are automatically coupled to geometrically adjoining zones or facades. The user should not create these couplings manually.

If the wall is not adjoining a facade or another zone, one has a free choice of thermal boundary conditions. The choice **Ignore net heat transmission** will give a wall with thermal mass exposed but no heat transmission over extended periods. Other choices are explained above.

Note that when a zone is moved in a way that, e.g. changes an external wall to an internal one, the choice of wall construction will be automatically adapted to the new role. This also applies to parts of walls.

7.5.2. Editing objects on a surface



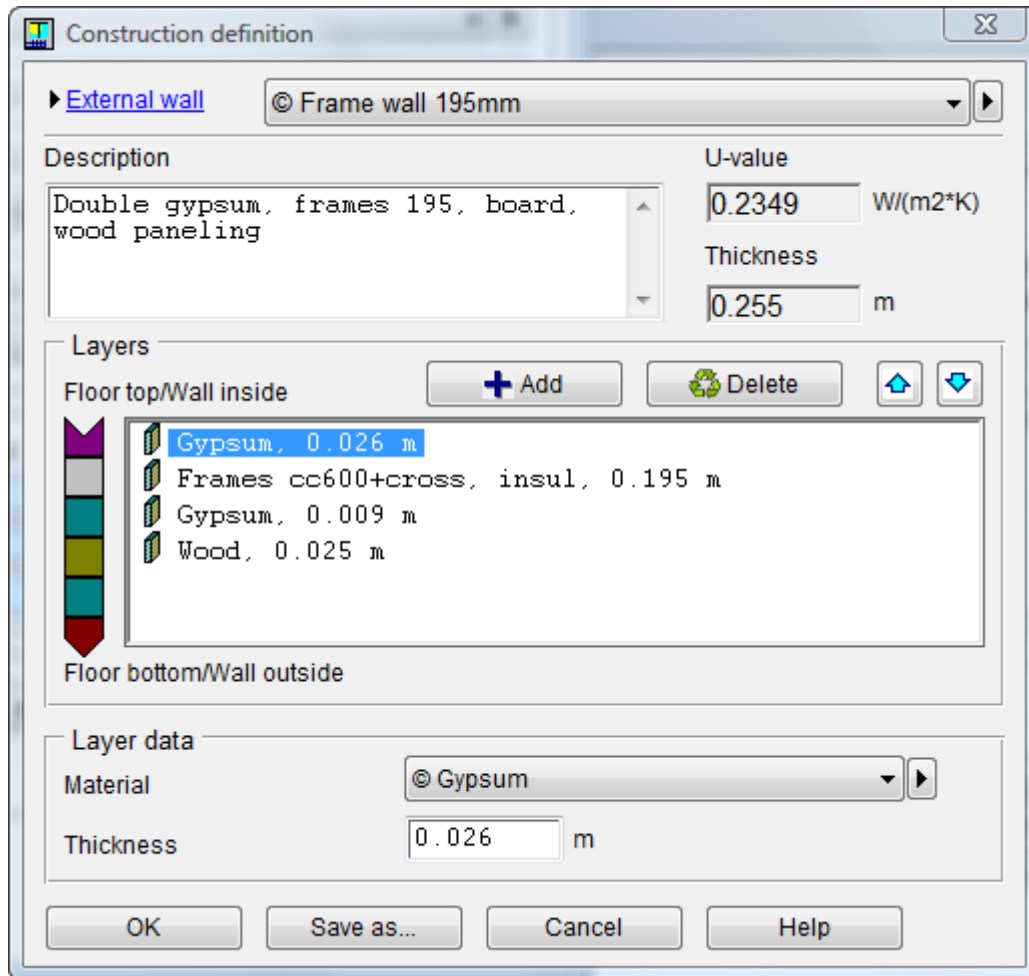
Double clicking in the box Drawing describing objects on the surface, in the form for objects of type Floor, Wall and Ceiling, opens a so-called surface editor which shows a drawing of the respective surface.

To insert an object on the surface, drag it from the palette page on the Side bar. Alternatively, you can double click an object on palette and then place the cursor somewhere within the drawing, hold down the left button and drag the object to the desired size. Now release the button.

The size and position of the object can be changed. Change the size by first selecting the object and dragging one of the small rectangles in the object's corners or in the middle of its sides. Change the position by placing the cursor within the object's boundary lines, holding down the left button, and dragging the object to the desired position.

Numerical values for the object's size and position can also be given on the **Properties page** (situated on the Side bar), while the object is selected.

7.5.3. Dialog for wall definition



Objects of the Wall definition type are used to define the construction of walls and floors. A wall definition has a name, a description and a number of layers.

Field descriptions, etc:

Category Select the category of construction (internal floor, external wall etc). When editing the construction of some object, only the compatible categories are available. Select "Generic" to allow using the construction in any case.

Wall definition Choice of Wall definition object. The rest of the dialog shows the details of the selected object.

Description Object description.

U-value Overall U-value, including film coefficients ($W/(m^2 * K)$)

Layers: Floor top/Wall inside A list of layers in the wall. NB! Also Roof constructions are described from top to bottom.

Add Button to add additional layer.

Delete Button to delete an existing layer.

Promote Button to move a layer up one step.

Demote Button to move a layer down one step.

Layer data

Material The button with the right arrow gives a menu of possible operations:

Open Open the Material dialog.

New resource Create a new material.

Load from database... Load object of the Material type from database.

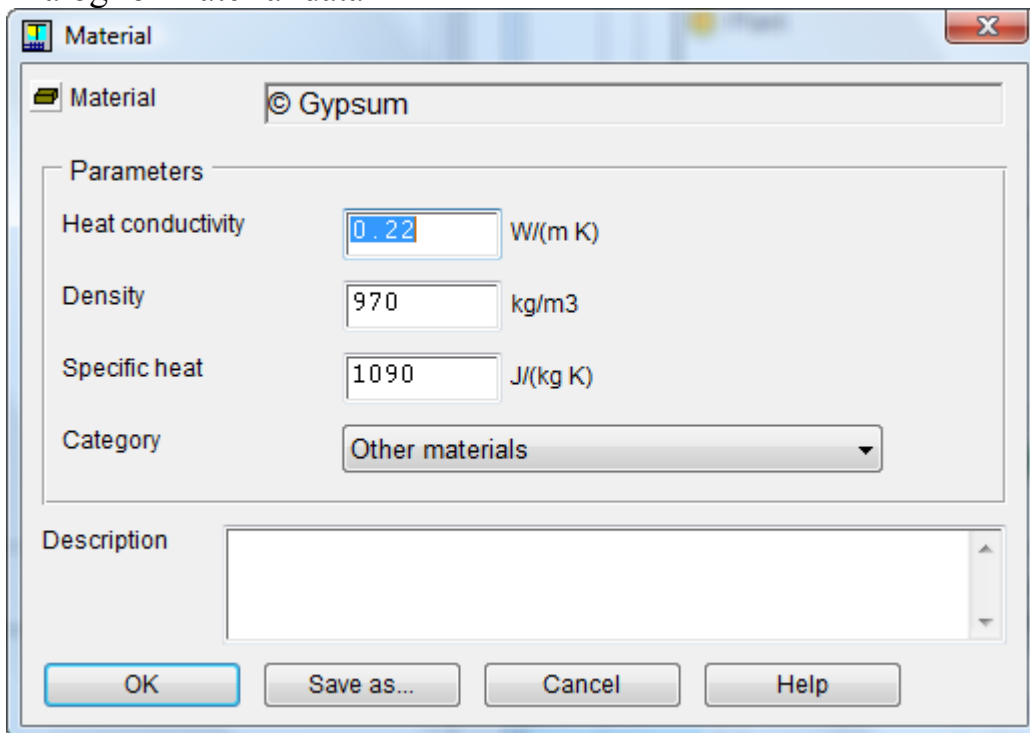
Write to database Save the user-defined material to the database.

Thickness Layer thickness. (m)

The dialog for a wall definition is opened by the menu option Open, after pressing the Right button with the cursor over an object of the wall definition type (e.g Internal wall with insulation) in the Construction field in the form for a wall or in the form for IDA-resources. The layers the wall consists of are displayed in the list box in the Layers box. Floor layers are given in order of floor surface to floor bottom (usually ceiling below). Wall layers are given in order of inside to outside. The Add button is used to add new layers in a wall definition. The layers selected in the list box can be deleted with the Delete button and moved up or down respectively with the Promote and Demote buttons.

Layer data for the selected layer is displayed in the Layer data box to the right in the dialog. Every layer has a name, a description and a thickness given in meters, and consists of a material that can be selected under the Material heading. If the desired material isn't on the list, additional material definitions can be loaded from the database by clicking right arrow symbol and selecting Load from database. Use the Open choice in the same menu to open the selected material definition. Similarly, a new material can be created by selecting New Resource and the selected material can be stored in the database by selecting Write to database.

Dialog for material data



The screenshot shows a dialog box titled "Material" with a close button (X) in the top right corner. The "Material" field contains "© Gypsum". Below this is a "Parameters" section with four rows: "Heat conductivity" with a value of "0.22" and units "W/(m K)"; "Density" with a value of "970" and units "kg/m3"; "Specific heat" with a value of "1090" and units "J/(kg K)"; and "Category" with a dropdown menu set to "Other materials". At the bottom is a "Description" text area. The dialog has four buttons: "OK", "Save as...", "Cancel", and "Help".

Field descriptions, etc:

Name Choice of Material object. The rest of the dialog shows the details of the selected object.

Heat conductivity Heat conductivity. (W/m*K)

Density Density. (kg/m³)

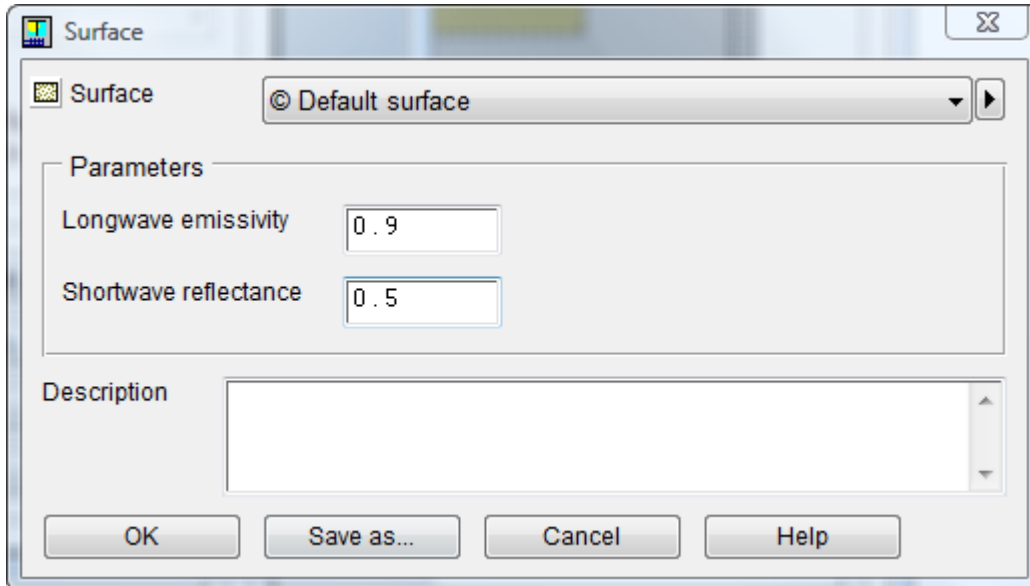
Specific heat Specific heat. (J/kg*K)

Category The type of material (used to systematize the database)

Description Material description.

The Dialog for a material is opened from Wall definition dialog by clicking on the Open button. Access a resource from the Right button menu with the cursor over an object of the Material type (e.g., Concrete), in the form for IDA-resources.

7.5.4. Dialog for surface



This dialog is used to edit objects of the Surface type. A surface object has a name (Default surface in the figure), a description and two parameters.

Field descriptions, etc:

Name Choice of Surface object. The rest of the dialog shows the details of the selected object.

Description

Longwave emissivity (0–1)

Shortwave reflectance (0–1)

The dialog for a surface is opened by the menu option Open, after pressing the Right button with the cursor over an object of the Surface type, (e.g. Light surface) in one of the Interior surface or Exterior surface fields in the form for wall, ceiling and floor or in the form for IDA-resources.

7.5.5. Wall part

Object: building → zone → enclosing surface → wall part

Available from: Wall editor

Description: Describes a part of wall, floor or ceiling with different construction or adjacency conditions, or with different surface temperature.

Views: General (identical to enclosing element's advanced view)

A wall part is an object that can be inserted into another wall and which can be given other properties than the rest of the wall. All parameters that can be specified for a wall, under the Advanced tab, can be specified independently for a wall part.

When a wall adjoins more than one zone or building face, each separate contact will automatically be modeled as an implicit wall part. The user should *not* specify these partitions explicitly.

Each wall part (both explicit and implicit) is modeled with its own (time dependent) surface temperature. Wall partitions can be used to model non-homogeneous temperature distributions over a wall.

Wall features (windows, radiators, etc) can be placed inside wall parts, but must not intersect the part's borders (on either side of the wall).

See also:

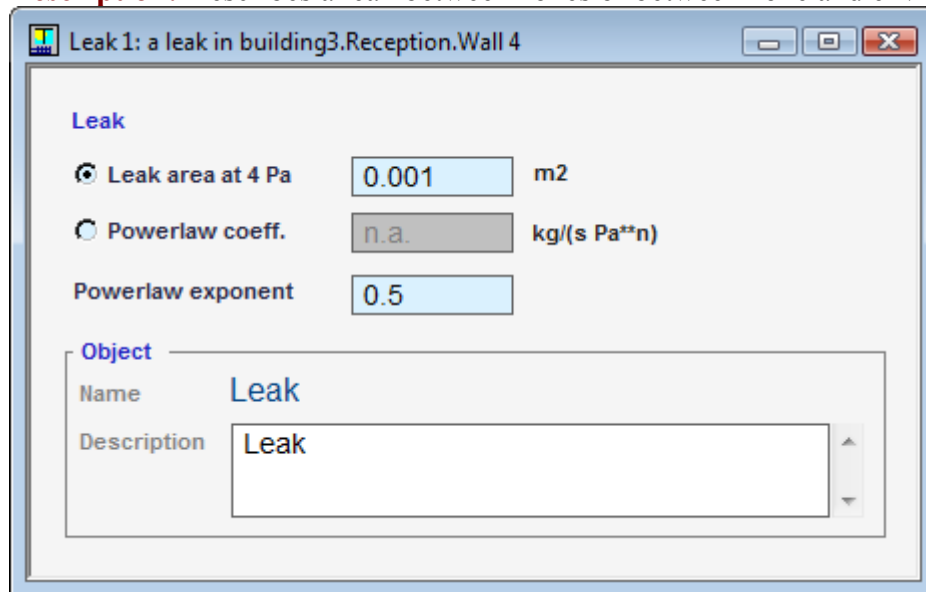
Enclosing element, advanced view.

7.5.6. Leak

Object: building → zone → enclosing surface → leak

Available from: Wall editor

Description: Describes a leak between zones or between zone and environment.



Field description, etc:

x, y The position of the leak on the wall. Only the y coordinate will affect the result. (m)

Leak area ELA Equivalent leakage area in the envelope, defined at 4 Pa, Cd=1. [m²].

Powerlaw coeff. [kg/(s*Paⁿ)]

A leak between zone and environment is always present in the ICE Building model. The size of this (default) leak is given in the Zone form. Additional leaks to the environment and leaks between zones may be inserted in walls, ceiling, and floor, see Wall editor form.

A leak is positioned 1 m above the floor level in each external wall, or in the ceiling if no external wall exists, or in the floor towards crawl space if no external surfaces exist. Thus also completely internal zones have some connection with the ambient.

7.5.7. Windows and openings

7.5.7.1. Form for window

Window: a window in building1.Zone.Wall 3

Glazing [Default] © 3 pane glazing, clear, 4-12-4-12-4

Opening

Control Schedule

Schedule © Never open

Frame

Fraction of the total window area 0.1 0-1 [More...](#)

U-value 2.0 W/(m2 °C)

Skew

Twist 0 °

Tilt 0 °

Integrated Window Shading

Device [Default] © No integrated shading

Control Light + Schedule

Schedule © Always drawn

External Window Shading

Device Awning, 3x1 m, 10 cm window recession - cm

Drawing view from the side to describe the external window shading

Object

Name Window

Description

A form for editing the properties of a window is opened (e.g., by double clicking on the window) in Drawing describing objects on the surface. (The surface editor is displayed by double clicking in the drawing box in the form for wall, floor and ceiling where the window is).

Field descriptions, etc:

Glazing Choice of glass configuration, includes SHGC-, T- and U-value for the glass.

Opening (Expert edition only)

Control Selection of control strategy for window opening. Supported strategies

Schedule

The opening is controlled by time schedule (or not controlled at all)

On-off control + schedule, PI-control + schedule

The opening is controlled by air temperatures (both internal and external) in the range from 0 (fully closed) to the value given by the schedule

New

Define a new opening control strategy

User-defined control strategies

The strategies that had been defined by selecting "New"

Schedule Schedule for degree of window opening. 0 = fully closed, 1 = fully open.

Schedule smoothing applied by default

Frame properties

Fraction of the total window area The unglazed area of the window divided by the whole window area, defined by the outer frame measures. (0-1)

U-value Heat transfer coefficient for the unglazed part of the window, including interior and exterior film coefficients. (W/°C)

More... Opens a dialog with detailed frame construction.

Skew [Deg]

Specifies the orientation of the glass surface relative to the wall surface in horizontal (**Twist**) and vertical (**Tilt**) direction. The values of Twist and Tilt are added to the azimuth and slope of the wall's external surface in order to get the azimuth and the slope of the external glass surface.

Integrated window shading

Device Choice of curtains or blinds.

Control Selection of control strategy for integrated shading device.

Schedule Schedule for integrated window shading. Schedule smoothing applied by default

Selecting 'Light intensity + Schedule' draws the shading when the schedule is 'on' and the incident light exceeds 100 W/m² on the inside of the glass. This level can be changed at the building level in outline view, System parameters, internal_shading_control_level.

Selecting 'Schedule', makes only the schedule's values apply, 1=completely drawn, 0=completely open

Select "New" to define a new shading control strategy.

The list of control strategies contains also all user-defined shading control strategies.

External window shading

Device Choice of external window shading. Object for near-window shading, eg., awnings, side fins, recess depth etc.

Drawing view from the side to describe the external window shading External window shading. Quick way to open shading editor.

Object: Name and description

The form contains a box named Drawing view from the side, to describe the external shading. Double clicking on this opens a form, which is used for editing any shading objects outside the window. This form has two tabs; under the Control tab, the type of control can be specified (no control, wind and/or sun control).

The form for a window can also be opened by clicking the right button over an object of the window type in the surface editor.

7.5.7.2. Detailed window (Expert edition only)

DetWin: a window in building1.Zone 1.Wall 3

Glazing © Double Clear Air

Opening

Control Schedule

Schedule © 08-17 weekdays

Frame

Fraction of the total window area: 0.1 (0-1) More...

U-value: 2.0 W/(m² °C)

Skew

Twist: 0 °

Tilt: 0 °

Integrated window shading

Draw Control Light + Schedule

Draw schedule © Always drawn

Slat angle: 0.0 °

External window shading

Device © No external shading

Drawing view from the side to describe the external window shading

Object

Name: DetWin

Description: Window (detailed)

[* Schedule smoothing applied. Change in [System parameters](#)]

A form for editing the properties of a detailed window is opened (e.g., by double clicking on the window) in Drawing describing objects on the surface. (The surface editor is displayed by double clicking in the drawing box in the form for wall, floor and ceiling where the window is).

Field descriptions, etc:

Glazing Choice of window configuration (glass and shading panes, gaps).

Opening

Control Selection of control strategy for window opening. Supported strategies

Schedule

The opening is controlled by time schedule

On-off control + schedule, PI-control + schedule

The opening is controlled by air temperatures (both internal and external) in the range from 0 (fully closed) to the value given by the schedule

New

Define a new opening control strategy.

User-defined control strategies

The strategies that had been defined by selecting "New"

Schedule Schedule for degree of window opening. 0 = fully closed, 1 = fully open.
Schedule smoothing applied by default.

Frame properties

Fraction of the total window area The unglazed area of the window divided by the whole window area, defined by the outer frame measures. (0-1)

U-value Heat transfer coefficient for the unglazed part of the window, including interior and exterior film coefficients. (W/°C)

More... Opens a dialog with detailed frame construction.

Skew [Deg]

Specifies the orientation of the glass surface relative to the wall surface in horizontal (**Twist**) and vertical (**Tilt**) direction. The values of Twist and Tilt are added to the azimuth and slope of the wall's external surface in order to get the azimuth and the slope of the external glass surface.

Integrated window shading

Draw control Selection of control strategy for integrated shading device.

Draw schedule Schedule for integrated window shading. Schedule smoothing applied by default.

Selecting 'Light intensity + Schedule' draws the shading when the schedule is 'on' and the incident light exceeds 100 W/m² on the inside of the glass. This level can be changed at the building level in outline view, System parameters, internal_shading_control_level.

Selecting 'Schedule', makes only the schedule's values apply, 1=completely drawn, 0=completely open

Select "New" to define a new shading control strategy.

The list of control strategies contains also all user-defined shading control strategies.

External window shading

Device Choice of external window shading. Object for near-window shading, eg., awnings, side fins, recess depth etc.

Drawing view from the side to describe the external window shading External window shading. Quick way to open shading editor

Object: Name and description

The form contains a box named Drawing view from the side, to describe the external shading. Double clicking on this opens a form, which is used for editing any shading objects outside the window. This form has two tabs; under the Control tab, the type of control can be specified (no control, wind and/or sun control).

The form for a window can also be opened by clicking the right button over an object of the window type in the surface editor.

7.5.7.3. Form for Skylight

The screenshot shows the 'SkyLight: a skylight in building1.Zone.Ceiling' window. It is divided into three main sections: Geometry, Construction, and Object.

Geometry Section: Contains input fields for X (0.83 m), Y (2.57 m), DX (0.8 m), DY (0.8 m), DZ (0.8 m), L1 (0.2 m), L2 (0.2 m), L3 (0.2 m), and L4 (0.2 m). To the right, there are two diagrams: a top-down view of the skylight's rectangular base with dimensions L1-L4 and DX-DY, and a 3D perspective view of the truncated pyramid with height DZ and base dimensions DX-DY.

Construction Section: Contains a table for area calculations and dropdown menus for material selection.

	Wall 1	Wall 2	Wall 3	Wall 4	Roof
Glass area [m2]	0.0	0.0	0.0	0.0	0.0
Total area [m2]	0.49	0.49	0.49	0.49	0.16

Below the table are dropdown menus for 'Walls' (set to '[Default] © Concrete joist roof') and 'Glazing' (set to '[Default] © 3 pane glazing, clear, 4-12-4-12-4').

Object Section: Contains a 'Name' field with 'SkyLight' and a 'Description' field with 'SkyLight'.

The skylight is a roof lantern. It can be used to study solar radiation and climate effects for different forms of glazed openings in roofs, e.g. glazed saddle roofs. The skylight object is shaped as a pyramid with rectangular base and with the top removed by a horizontal cut. Thus, it has five sides, which can be glazed to different extents.

A detailed ray tracing calculation is performed, and e.g. direct light entering through one surface and leaving through another is handled correctly. Light reaching the base of the pyramid is transmitted to the zone as diffuse light.

Each skylight object inserted in a zone (see Insert object) is edited in this form.

Field description, etc:

Geometry:

- X** Coordinate of the lower left corner, can be entered in Applet (m)
- Y** Coordinate of the lower left corner, can be entered in Applet (m)
- DX** Extension of base rectangle in x-direction (m)
- DY** Extension of base rectangle in y-direction (m)
- DZ** The height of the skylight above the roof (m)
- L1, ... L4** Distance in the roof plane between the borders of the base and top rectangles; four values for the four sides of the rectangle, numbered by the closest zone Walls 1-4 (m)

Construction:

Glass area: Wall 1, ... 4 Glazed area on side towards Wall 1 [m²]

Glass area: Roof Glazed area in top surface of pyramid [m²]

Walls Construction selected for unglazed parts of all five sides [Data base selection of bwall construction]

Glazing Selection of glazing construction, optical and thermal properties [Data base selection of glazing construction]

In the zone model, the skylight has the thermal properties of a wall subsurface. The temperature of this surface is set to a weighted average of the temperatures of the glazed and unglazed parts of the five surfaces of the skylight. This model should rather well represent the long-wave and convective properties of a heated skylight. Downdraughts from cold vertical surfaces in the skylight will not be as well reproduced.

7.5.7.4. Form for opening (door)

Door: an opening in building3.Reception.Wall 4

Opening

Construction [Default] [use wall construction]

Opening schedule © Never open

Inner surface © [Default surface]

Outer surface © [Default surface]

Leak area at 4 Pa 0.01 m2 (for closed internal doors)

Object

Name Door

Description Large vertical opening

NB! Doors can only be opened in the Expert edition of IDA ICE. However, also Standard edition users should add internal doors in relevant places, since they provide a leakage path between zones also when closed. They may also have a different construction.

Field description, etc:

Construction Choice of door construction.

Two special values:

[use wall construction] means that wall has the same construction as the surrounding wall.

Opening without door. In this case all other parameters are irrelevant

Opening schedule Schedule for degree of window opening. 0 = fully closed, 1 = fully open.

Leak area ELA Equivalent leakage area when the door is closed, defined at 4 Pa, Cd=1 and, at the door vertical midpoint.

Inner surface Optical properties of the inner surface

Outer surface Optical properties of the outer surface (relevant for inner doors only)

Object: Name and description

An opening between two zones (or in an external wall) gives rise to a (sometimes bi-directional) air flow, that will reduce the differences in temperature, humidity and carbon dioxide content between the air masses. The radiation through the opening is also considered.

The opening is defined by inserting an opening component in a wall. For internal walls, this is only done in one of the zones. The other zone then displays a gray surface on its corresponding section of wall.

The form for an opening is opened by double clicking on an object of the Opening (door) type in the Surface editor (The surface editor is reached by double clicking in the drawing box in the form for wall, floor and ceiling, ceiling or floor that contains the opening).

7.5.7.5. Dialog for Glass construction

The screenshot shows the 'Glass construction' dialog box. The title bar includes a close button (X). The main window has a dropdown menu showing '© 2 pane glazing, clear, 4-12-4'. Below the dropdown are three radio buttons for 'Shading coefficients': 'Absolute value' (selected), 'Single pane reference', and 'Double pane reference'. Under 'Absolute value', there are three input fields: 'g, Solar Heat Gain Coef (SHGC)' with value '0.76', 'T, Solar transmittance' with value '0.6764', and 'Tvis, Visible transmittance' which is empty. To the right, under 'Description', there are four input fields: 'Glazing U-value' with value '3.0' and unit 'W/(m2*K)', 'Internal emissivity' with value '0.9' and range '0-1', and 'External emissivity' with value '0.9' and range '0-1'. At the bottom are four buttons: 'OK', 'Save as...', 'Cancel', and 'Help'.

The Glass construction dialog is used to describe optical and thermal properties of the window glazing for the standard (simplified) window model.

Field descriptions, etc:

Name Choice of Glass construction object. The rest of the dialog shows the details of the selected object.

Shading coefficients: Absolute value Select to specify window data without reference glazing

Shading coefficients: Single pane reference Select to specify data with single glass reference

Shading coefficients: Double pane reference Select to specify data with double pane reference

Shading coefficients: g, Solar Heat Gain Coefficient (SHGC) Displayed when Absolute value has been selected above. Fraction of the radiation, incident on the window, that heats

the room. SHGC includes both the radiation that passes through the window directly and the radiation that is first absorbed in the panes and thereafter reaches the zone as convection and long-wave radiation.

SHGC is short for Solar Heat Gain Coefficient and is sometimes also called: TST, total transmission, SF, etc.

Shading coefficients: T, Solar transmittance Displayed when absolute value has been selected above. Fraction of incident radiation that passes the glazing as direct radiation, NB! It must always be smaller than g.

T is sometimes also called T_{sol}, ST, DET, etc.

See below for parameters for single and double pane reference

Glazing U-value Heat transfer coefficient for the glazing (without frame), including internal and external film coefficients (W/m² °C) The window models calculate actual internal and external (wind dependent) film coefficients. From the given U-value the program subtracts 0.17 (m² K)/W and the remaining U-value is considered to equal the heat transfer coefficient of the glass.

Internal emissivity Innermost glass emissivity (inwards) for longwave radiation.

External emissivity Outermost glass emissivity (outwards) for longwave radiation. Has, in normal cases, limited influence on the result.

The total entering heat in the case when the window is perpendicular to the entering radiation is given by the following expression:

$$P_{total} = g * A * I$$

where A [m²] is glass surface area and I [W/m²] is the total incoming radiation. This calculation takes into consideration both direct and diffuse radiation on the facade. The model will also compensate for different incidence angles using a predefined curve, see the NMF code at the advanced level for details.

The T and T_{vis} parameters, determine the degree at which solar radiation passes through the window without first being absorbed:

$$P_{shortwave} = T * A * I$$

$$P_{visual_shortwave} = T_{vis} * A * I <<check??>>$$

$P_{total} - P_{shortwave}$ determines the part of the radiation that reaches the zone via absorption in the window.

See further information about solar radiation computation in the manual.

For combinations of glazing and integrated shadings, e.g., curtains or blinds, data is given for the integrated shading object that modifies the given parameters for the glazing. See Dialog for integrated shading.

Parameters for single and double pane reference

Shading coefficients: F1, total shading coefficient Displayed when Double pane reference has been selected above. Fraction incoming radiation (in relation to a 2-pane window) that heats the zone. F1 includes both the radiation that passes through the window directly and the radiation that is first absorbed in the panes and thereafter reaches the zone as convection and long-wave radiation.

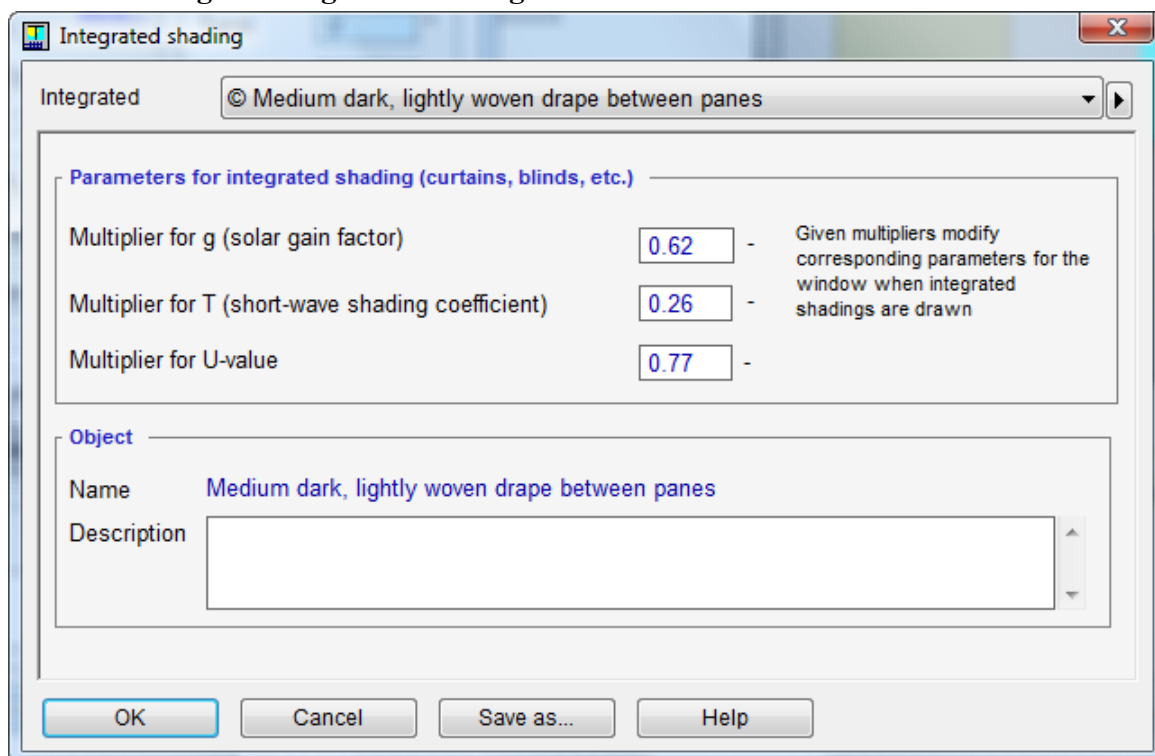
Shading coefficients: F2, shortwave shading coefficient Displayed when Double pane reference has been selected above. Fraction of incoming radiation (in relation to a 2-pane window) which passes through the pane. This is radiation that passes the window in the form of shortwave radiation.

Shading coefficients: Sc, total shading coefficient Displayed when single pane reference has been selected above.

Shading coefficients: Ssc, shortwave shading coefficient Displayed when single pane reference has been selected above.

The dialog for a glass constructions is opened from the Right button menu with the cursor over an object of the Glass construction type (e.g. 3-glass, clear, 4-12-4-12-4) in the Glazing field in the form for a window or in the form for IDA-resources.

7.5.7.6. Dialog for integrated shading



This dialog is used for giving parameters belonging to the objects of the integrated shading type, for example, a curtain or blind. The multipliers modify the corresponding parameters for the window when the shading is "on". This object is only used for the standard window model; integrated shading is handled differently in the detailed window model.

Field descriptions, etc:

Integrated shading Choice of integrated shading object. The rest of dialog displays the details of the shading object

Multiplier for g, Solar Gain Factor

Multiplier for T, Solar transmittance

Multiplier for U-value

Object: Name and description

Integrated shading (blinds, curtains, etc.) concerns all types of shading in the window's plane, even external blinds. (External shading concerns permanent shading objects, on the facade, near the window, e.g. side fins etc.).

Three parameters are given for integrated shading. These provide multipliers, which indicate the effects of the shading in combination with the glazing. See Dialog for glass construction for definitions of SHGC, T and U for the glazing.

When the inner shading is on (drawn) the effective parameters become

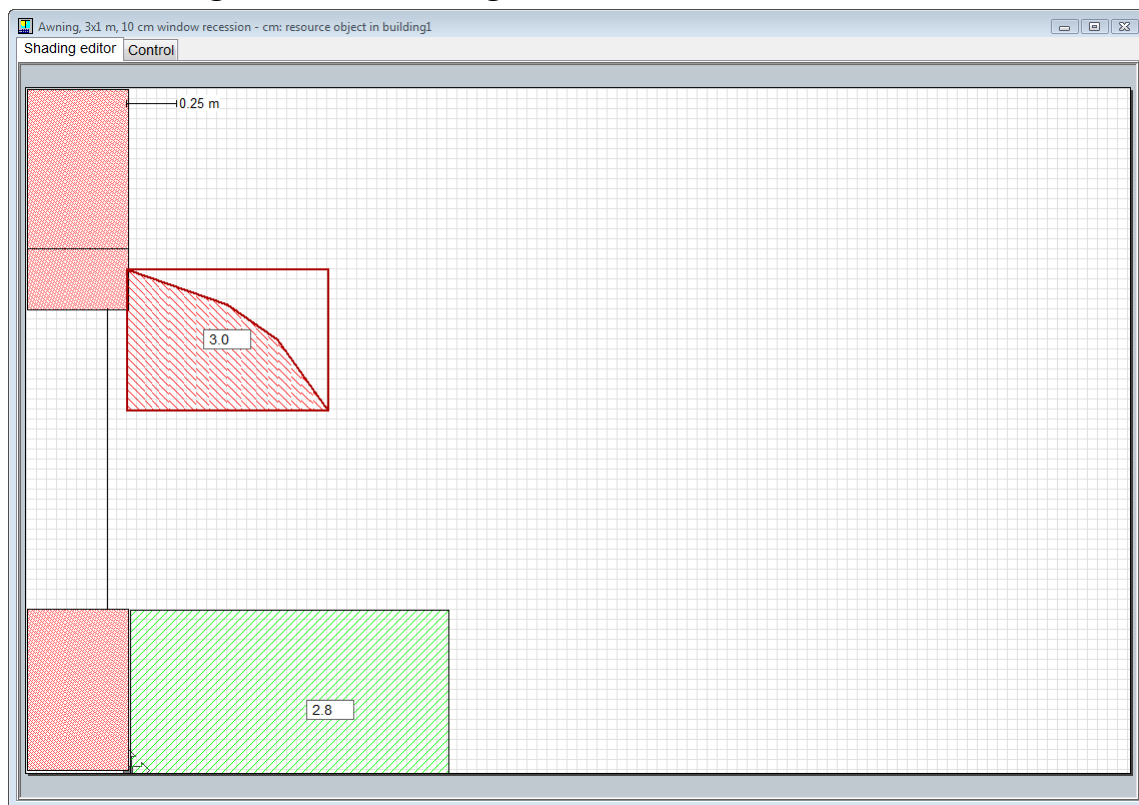
$g_{\text{effective}} = g * \text{multiplier for } g$

$T_{\text{effective}} = T * \text{multiplier for } T$

$U_{\text{effective}} = U * \text{multiplier for } U$

The dialog for an integrated shading is opened from the Right button menu with the cursor over an object of the integrated shading type (e.g. Light dense curtain between the panes) in the Integrated shading field in the form for one window or in the form for IDA-resources.

7.5.7.7. Editing an external shading



This form is used to edit an object of the External shading type. The window and the external wall are shown in a side view. An external shading is a combination of the shading elements Balcony with sides, Simple screen, Side fins and Marquee with sides. The window's external recess depth is also included in its external shading.

The Shading editor form is opened by double clicking on the Drawing view from the side to describe external shading button, in the window form. A new shading object is inserted by selecting New object... in the Insert menu with the Shading editor being the active window.

In the Insert object dialog then opened there are four alternatives:

Balcony with sides

Simple screen

Side fins

Marquee with sides

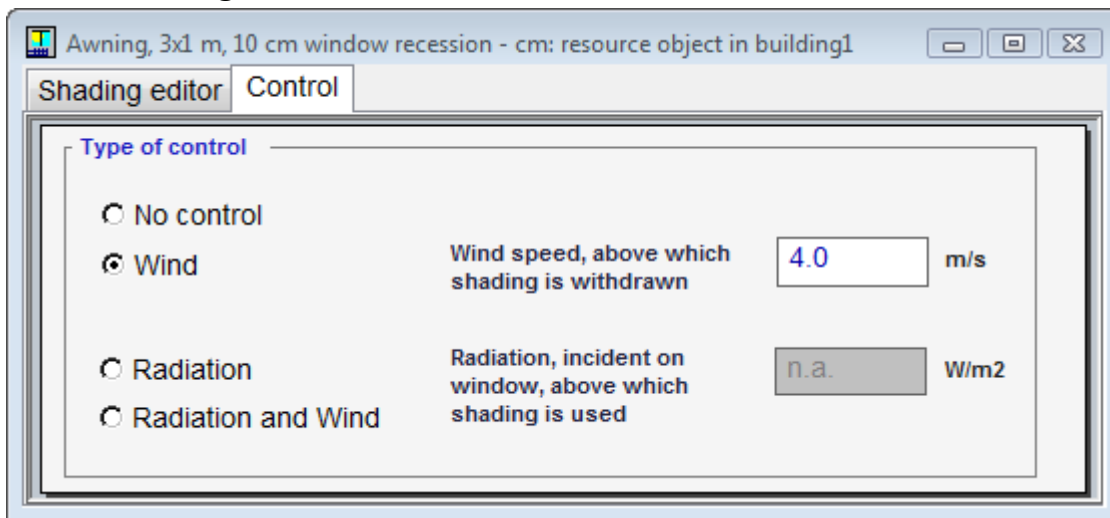
When the selection is made the editor is redisplayed. Place the cursor beside the wall to the right of the window, hold down the left button and drag the shading to the desired size. For the Simple screen alternative, draw a polyline by clicking a start point, break points, and end point in the form. End by clicking once with the Right button and selecting OK. The small number field is for the width of the shading (for side fins, the distance between them), i.e the extension perpendicular to the *editor* plane. Repeat this procedure to insert further shading objects.

The window's position in the wall, the so-called recess depth, is changed by dragging the window line to the desired position. The cursor changes appearance when it is placed over the window line.

The faint horizontal line in the wall over the window indicates ceiling height in the zone where the window is located.

In the Shading editor there is a tab Control. Via this, one can specify limits when the shading should be withdrawn due to strong wind or low solar radiation.

7.5.7.8. Shading control



No control

No automatic control

Wind

The shading is withdrawn at high local wind speed

Radiation

The shading is drawn (extended) at high solar radiation

Radiation and wind

Control by wind and sun; wind has priority

Wind speed, above which shading is withdrawn

If the wind speed for this facade exceeds this level, the shades are withdrawn.

Radiation, incident on window, above which shading is used

If the radiation in the plane of the window, with shades withdrawn, exceeds this level, the shades are extended.

7.5.7.9. Form for balcony, marquee and screens

The image shows three software forms for configuring shading devices:

- Screen:** Width: 9.0 m. Vertices table:

	x	y
	1.25	1.9
	2	1.9
	2	1.8
	1.25	1.8

 Shading type: Fixed, Variable.
- Awning:** Position [m]: Min (x: 0.0, y: 1.8), Size (x: 1.0, y: 0.7), Max (x: 1.0, y: 2.5). Width: 3.0 m. Shading type: Fixed, Variable.
- Balcony:** Position [m]: Min (x: 1.272, y: 2.144), Size (x: 2.0, y: 1.0), Max (x: 3.272, y: 3.144). Width: 3.0 m.

This form describes one of the objects Balcony with sides, Simple screen, Side fins and Marquee with sides.

Field description:

Width The extension of the shading device perpendicular to the editor plane (m)

Shading type Fixed or controllable device; select parameters for all controllable devices under the tab Control.

Vertices (m)

The form for a shading object is opened from the Right button menu with the cursor over an object of this type (e.g., Balcony with sides) in the Shading editor.

Shading objects cannot be changed to independent IDA-resources, but will be saved, as resources or data base objects, combined with the window's size and positioning in the wall (see Edit an external shading).

7.5.7.10. Glass Packet (Expert edition only)

The **Glass packet** dialog is used to describe optical and thermal properties of the window glazing for the advanced window model. The dialog shows the list of layers (panes, shading devices, and gaps), the parameters of the currently selected layer and the buttons used to add, remove, and reorder the layers.

A glass packet should consist of one or more glass pane, optional shading layer, and cavity layers between pane/shading layers.

Two types of shadings are currently supported: plain shading and Venetian blind.

Layer parameters:

Pane layer

Pane A reference to a pane description. The button with the right arrow gives a menu of possible operations: Open the Pane dialog; create a new pane description; load a pane from database; save the pane to database.

Cavity layer

Material A reference to a gas description. The button with the right arrow gives a menu of possible operations: Open the Gas dialog; create a new gas description; load a gas from database; save the gas to database.

Thickness The thickness of the cavity

Shade layer

Material A reference to a shade material description. The button with the right arrow gives a menu of possible operations: Open the Shade material dialog; create a new shade material description; load a shade material from database; save the shade material to database.

Venetian blind layer

Device A reference to a Venetian blind description. The button with the right arrow gives a menu of possible operations: Open the Venetian blind dialog; create a new Venetian blind description; load a Venetian blind type from database; save a Venetian blind to database.

7.5.7.11. Glass pane

The **Glass pane** dialog is used to describe the properties of a glass pane (for using in Glass packet for detailed window model).

Glass panes are resource objects; that means that the same pane description may be referenced multiple glass packets (or multiple layers of the same packet).

Pane parameters:

Glass pane Choice of Glass pane object. The rest of the dialog shows the details of the selected object.

The optical parameters are grouped in 3 columns: transmittance (that is the same for both directions), and 2 columns for reflectance, from front side and from back side.

Total shortwave

The coefficients of transmittance and reflectance mean for whole range of the solar shortwave radiation. The coefficients are applied for radiation in the direction normal to the pane. The coefficients for other incidence angles and for diffuse radiation are calculated from these ones.

Visible

The coefficients of transmittance and reflectance mean for the visible range of the solar radiation. The coefficients are applied for radiation in the direction normal to the pane.

The coefficients for other directions are calculated from these ones.

Diffusion

The coefficients of diffusion (dissipance), i.e., the fraction of the transmitted/reflected direct radiation that becomes diffuse after transmitting or reflection. These coefficients do not depend on the incident angle.

The diffusion coefficients are not stored in the database (because they mostly depend on the state of the glass surface and not on the glass type).

Thermal and other properties

Longwave

The table contains emissivity and transmittance. The transmittance is not stored in the database (almost always is zero)

Thickness [mm]

Thermal conductivity [W/(K.m)]

7.5.7.12. Venetian blind

The **Venetian blind** dialog is used to describe the properties of a Venetian blind (for using as shading layer in Glass packet for detailed window model).

The Venetian blinds are resource objects; that means that the same Venetian blind description may be referenced from multiple glass packets.

7.5.7.13. Shade material

The **Shade material** dialog is used to describe the properties of the material used in shading layers and in Venetian blinds in Glass packet for detailed window model.

The Shade materials are resource objects; that means that shading layers and Venetian blinds in multiple glass packets may reference the same Shade material description.

Shade material parameters:

Shade materials Choice of Shade materials object. The rest of the dialog shows the details of the selected object.

The optical parameters are grouped in 3 columns: transmittance (that is the same for both directions), and 2 columns for reflectance, from front side and from back side.

Total shortwave

The coefficients of transmittance and reflectance mean for whole range of the solar shortwave radiation. The coefficients are applied both to direct and diffuse radiation and do not depend on incidence angles. The transmitted/reflected radiation is assumed diffuse.

Visible

The coefficients of transmittance and reflectance mean for the visible range of the solar radiation. The coefficients are applied both to direct and diffuse radiation and do not depend on incidence angles. The transmitted/reflected radiation is assumed diffuse.

Thermal and other properties

Longwave

The table contains emissivity and transmittance.

Thickness [mm]

Thermal conductivity [W/(K.m)]

7.5.7.14. Gas properties

The **Gas properties** dialog is used to describe the properties of a cavity (for using in Glass packet for detailed window model).

Gases are resource objects; that means that the same gas description may be referenced from multiple glass packets (or from multiple cavities of the same packet).

7.5.7.15. Opening Control Macro

Opening Control macro is used to describe a custom opening control strategy for windows. To define a custom opening control strategy, select "New" in "Opening Control" field (in the window or detailed window form). This will open a new Opening Control macro.

To edit an existing Opening Control macro, select it in the "Opening Control" field and click the "Control" link. See Modeling about editing macro objects.

The control macro has the following predefined interfaces:

- schedule (input) connected with opening schedule defined in window form
- ZoneTemp (input) connected with air temperature in zone

- ZoneCO2 (input) connected with CO2 concentration in zone
- occupancy (input) connected to the schedule of the first occupant in the zone (or to 0 if no occupants in the zone)
- opening (output) gives the opening fraction (normally from 0 to 1)

If the control macro contains a Constant mapped to the maximum of the comfort temperature defined in zone's setpoint then, for zones where this setpoint is variable, the constant is automatically replaced with link to schedule.

The opening controls are shared between windows (i.e. a control defined in one window is available in all windows (both simplified and detailed). They are included in the list of IDA Resources. Use this list to manage the opening controls (e.g., copy to other system).

7.5.7.16. Integrated Shading Control Macro

Integrated Shading Control macro is used to describe a custom control strategy for integrated shading devices in windows.

To define a custom shading control strategy, select "New" in "Integrated Shading Control" field (in the window or detailed window form). This will open a new Integrated Shading Control macro.

To edit an existing Shading Control macro, select it in the "Integrated Shading Control" field and click the "Shading" link. See Modeling about editing macro objects.

The control macro has the following predefined interfaces and parameters:

- schedule (input) connected with opening schedule defined in window form
- ZoneTemp (input) connected with air temperature in zone
- occupancy (input) connected to the schedule of the first occupant in the zone (or to 0 if no occupants in the zone)
- shading (output) gives the shading fraction (normally from 0 to 1)
- SlatAngle (output) gives slat angle for detailed windows with venetian blinds
- If the value of the parameter Light Control is True, the control scheme is combined with standard light control, as described in documentation for the window and detailed window.

The integrated shading controls are shared between windows (i.e. a control defined in one window is available in all windows (both simplified and detailed). They are included in the list of IDA Resources. Use this list to manage the integrated shading controls (e.g., copy to other system).

7.5.8. Heating, cooling, ventilation

7.5.8.1. Room units for cooling and heating

Local heating or cooling is supplied in the zone by defining room units either directly in the zone (ideal heater and cooler, fan coil, local AHU) or on the walls, in the ceiling, or on the floor. There is general equipment for waterborne cooling (for both radiative and convective equipment), a water radiator (or convector), floor heating, and an electric radiator. The Expert edition supports also heated and cooled floor/ceiling.

See further the IDA ICE Manual.

7.5.8.2. Active Beam

Beam: a device in building3.Zone.Ceiling

General Outline

Simplified input data to chilled/heated beam

Use manufacturer's data
 Simplified model:

When using the detailed model, the power at the given condition is calculated from the manufacturer's data and shown in this form.
 To edit manufacturer's data, switch to the outline tab.
 In the simplified model, the power parameters are user-supplied. No manufacturer-supplied data are used.

	Cooling	Heating	
Power at design air flow	0.0	0.0	W
Power at zero air flow	0.0	0.0	W
Design air flow	0.0		L/s

Design conditions

	Cooling	Heating	
dT(coolant - zone air) at max power	8.5	20	Deg-C
dT(coolant) at max power	3	10	Deg-C

Controller: PI

Target setpoint: Air temperature

Use manufacturer's data Automatically selected when product data has been imported from manufacturer's database

Simplified model Select in order to give parameters manually

Power at design air flow, Cooling Waterborne cooling power at design air flow and given temperatures (°C)

Power at design air flow, Heating Waterborne heating power at design air flow and given temperatures (°C)

Power at zero air flow, Cooling Waterborne cooling power at zero air flow and given temperatures (°C)

Power at zero air flow, Heating Waterborne heating power at zero air flow and given temperatures (°C)

Design air flow Air flow through device at design conditions (l/s)

Design conditions

dT(coolant - zone air) at max power, Cooling Average temperature difference between coolant and room air at design power (°C)

dT(coolant - zone air) at max power, Heating Average temperature difference between hot water and room air at design conditions (°C)

dT(coolant) at max power, Cooling Temperature rise of coolant at design power (°C)

dT(coolant) at max power, Heating Temperature drop of hot water at design power (°C)

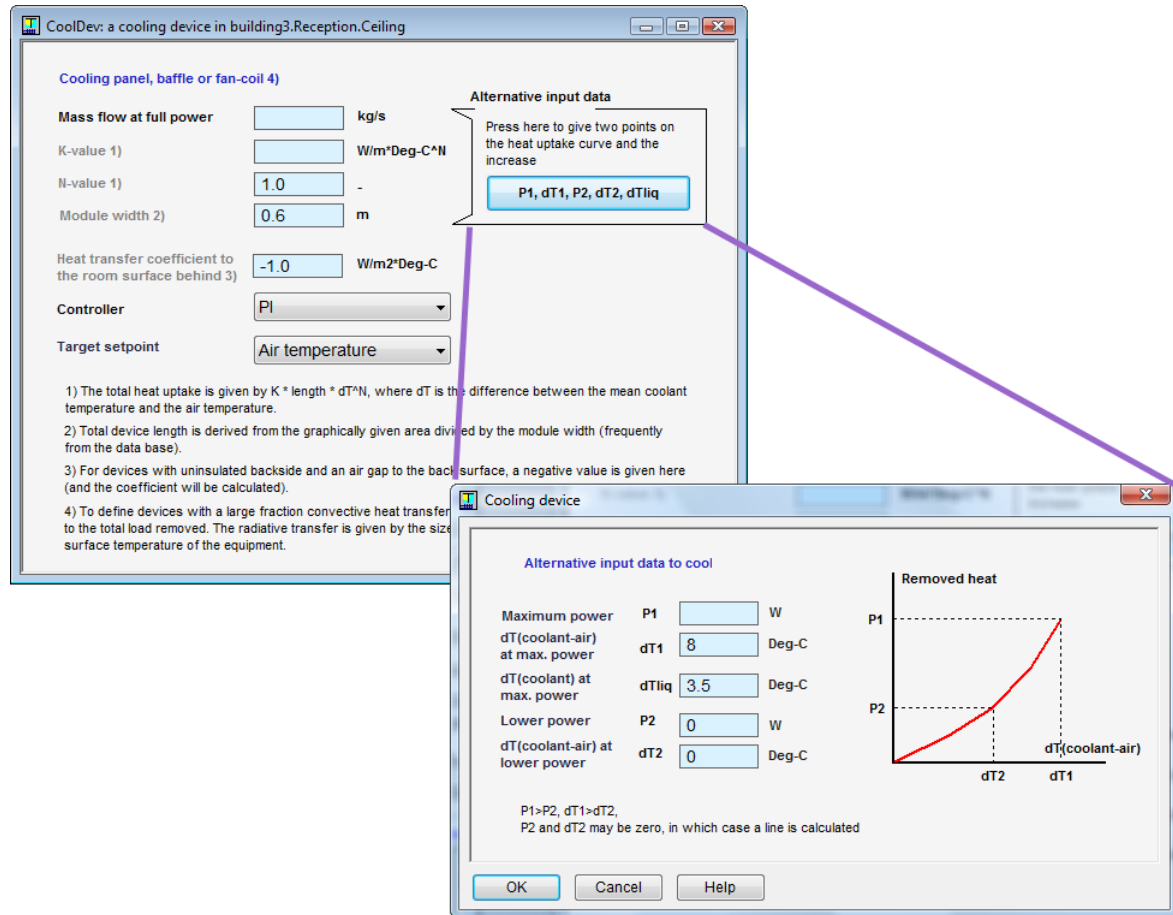
Controller Type of control

Active beams are air supply terminals combined with coils for cooling and sometimes also for heating. In addition to providing natural convection, which is active also without any mechanical supply air, the heat transfer is significantly improved by forced convection via the supply air stream.

The radiation component from active beams is normally quite small and it is neglected by the model, hence the actual position in the ceiling of the beam has no impact on calculation results.

Read more about active beams in the manual

7.5.8.3. Edit Cooling devices



A form for editing an object of the Cooling devices type is displayed by double clicking on a cooling device in the Drawing for describing an object on the surface, the so-called surface editor (Surface editor is displayed by double clicking in the drawing box in the form for the ceiling on which the cooling device is located). The figure also has a dialog for alternative input opened from the form.

Cooling devices operates completely analogous with heating device, with the exception of a few differences in input, which are described here.

Field descriptions, etc:

Massflow at full power Water flow through the equipment at design conditions. (kg/s)

K-value Removed heat per unit of equipment length and degree (raised to N). ($W/m^{\circ}C^N$)

N-value Exponent in the expression for removed heat. (-)

Module width Width of an equipment module for which given K and N are valid. (m)

Heat transfer coefficient to the room surface behind U-value for the insulation between the equipment and the wall/ceiling. ($W/m^2^{\circ}C$)

Maximum power, P1 Removed heat at full capacity. (W)

dT(coolant-air) at max. power, dT1 Mean temperature difference between the air and the coolant at full capacity, P1. ($^{\circ}C$)

dT(coolant) at max. power, dT_{liq} Temperature increase of water at full capacity. (°C)
Lower power, P₂ Removed heat at any partial load, e. g., half capacity. (W)
dT(coolant-air) at lower power, dT₂ Mean temperature difference between the air and the coolant at the partial load, P₂. (°C)

The form for a cooling device can also be opened from the Right button menu with the cursor over an object of the Cooling device type (e.g. Cooling device 1) in the form for surface editor or, for a resource(different form, includes alternative input, in the form for IDA-resources.

Here, the height of a radiator corresponds instead to a *Module width*, to which *K* and *N* refer. Just as for the radiator, the total length is calculated as the given box's area divided with *Module width*. There is a difference in that the heat transfer coefficient between the back of the equipment, and the surface behind (often the ceiling) is given directly in the main form. If an (arbitrary) negative figure is entered, the heat transfer coefficient is calculated in the same way as for the heating devices, i.e., as if all heat transfer was done by radiation. This is a good approximation for a device that has no insulation at all.

The dialog for alternative input has somewhat different parameters for cooling units. Absorbed power and temperature differences between air and water are given for two points on the power curve. For max power, the temperature drop of the water is also given. To accurately model zone thermal conditions, the surface areas of heating and cooling devices must be realistic in relation to emitted power.

See also Room units for Cooling and Heating.

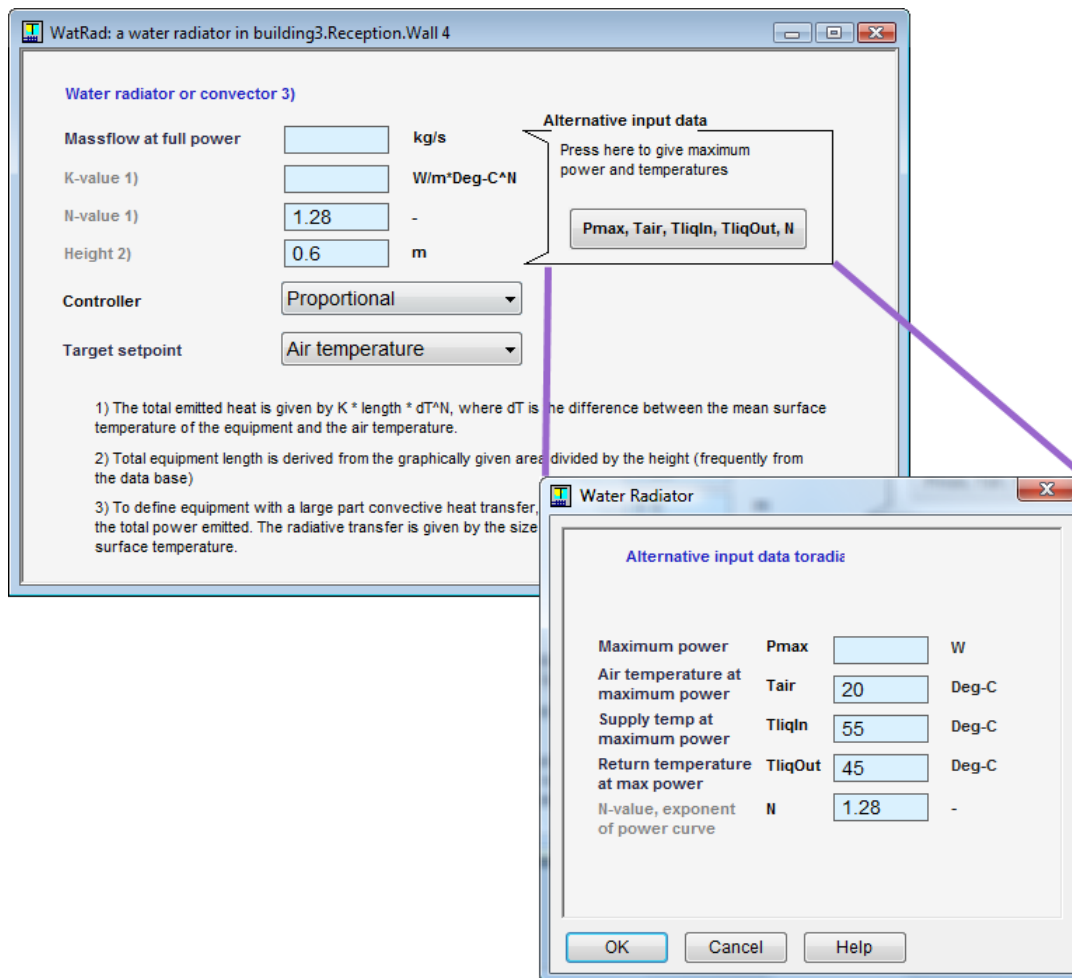
7.5.8.4. Edit waterborne heating devices

Heat emission from waterborne heating devices is calculated using

$$P = K * l * dT^N$$

where *l* is equipment length and *dT* is the temperature difference between the water and the zone air. *K* and *N* are constants characterizing a piece of equipment of a certain height. In the case of a radiator, *N* is often set at 1.28, which is why *K* gets the unit $W/(m * °C^{1.28})$. The logarithmic mean temperature difference is used to calculate *dT*.

The values of *K*, *N*, and the height of equipment of different makes are stored in the database. A database object should be defined for every principal configuration and occurring height, but the length of the equipment is first defined when it is inserted in the model.



A form to edit an object of the Heating device type is displayed by double clicking on a heating device in the Drawing for describing objects on the surface, the so-called surface editor. (Surface editor is displayed by double clicking in the drawing box in the form for wall, floor and ceiling on which the heating device is located) In the figure, a dialog for alternative input has also been opened from the form.

Field descriptions, etc:

Massflow at full power Water flow through equipment at design conditions. (kg/s)

K-value Emitted power per unit of equipment length and degree-C (raised to N). $W/(m * ^\circ C^{1.28})$.

N-value Exponent in the expression for emitted power. Frequently set to 1.28 for ordinary radiators. (-)

Height Equipment height at which given K and N are valid. (m)

Maximum power, Pmax Emitted power at full capacity. (W)

Air temperature at maximum power, Tair Room temperature at the measuring point at full capacity. ($^\circ C$)

Supply temp. at maximum power, TliqIn Incoming water temperature at full capacity. ($^\circ C$)

Return temperature at max. power, TliqOut Outgoing water temperature at full capacity. ($^\circ C$)

N-value, exponent of power curve, N See the base form for explanation of N-value. (-)

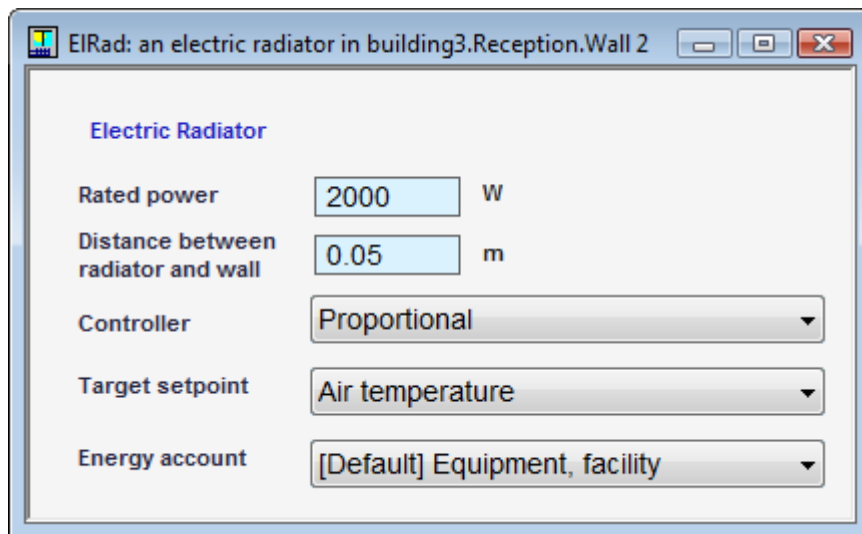
The form for heating device can also be opened from the Right button menu with the cursor over an object of the Heating device type (e.g. Water radiator 1) in the form for surface editor or, for a resource (different form, includes alternative input), in the form for IDA-resources.

Read about the radiator in Chapter 6 in the manual.

A detailed modelling of the room climate, especially operative temperatures, requires a reasonable agreement between heater or cooler surface size and specified power output.

See also Room units for cooling and heating.

7.5.8.5. Electric radiator



Electric Radiator	
Rated power	<input type="text" value="2000"/> W
Distance between radiator and wall	<input type="text" value="0.05"/> m
Controller	<input type="text" value="Proportional"/>
Target setpoint	<input type="text" value="Air temperature"/>
Energy account	<input type="text" value="[Default] Equipment, facility"/>

Rated power Maximum output (and input) power [W].

Distance between radiator and wall Gap behind radiator (m).

Controller Method of control of heater output. The temperature setpoint is fetched from Controller setpoints.

7.5.8.6. Floor Heating

FIHeat: a floor heating in building3.Reception.Floor

Floor heat (temperature control)

Design power output: 40.0 W/m²

Maximum temperature into coil: 27.0 Deg-C

Temperature drop across coil: 5.0 Deg-C

Controller: PI

Target setpoint: Air temperature

Location in floor slab

Depth under floor surface: 0.02 m

1) The floor heat coil becomes a heated layer at the given depth in the floor construction, which is defined elsewhere for the whole floor.

2) The design power and temperature drop are used to calculate a maximum massflow through the coil. The actual emitted power may become smaller, at the given maximum coil temperature, if the heat resistance in the floor construction is too large .

Heat transfer coefficient

H-water-pipe-fin: 10 W/(m² K)

Normally set to 6 for aluminium fins in a wooden construction and to 30 for pipes immersed in concrete. However, the total heat transfer is normally largely determined by the resistance in the floor construction above and below the coil, in which case this parameter becomes less important.

An object of the Floor heating type, inserted on a floor surface, can be opened for editing by double clicking on the object in the surface editor. (The surface editor is reached by double clicking on the floor in the zone form.)

Field descriptions, etc:

Floor heat (temperature control)

Design power output Power output at design conditions. (W/m²)

Maximum temperature into coil Highest permitted temp sent into floor. (°C)

Temperature drop across coil Temperature drop at design conditions. (°C)

Controller Type of control.

Location in floor slab

Depth under floor surface Depth of the water circuit plane below floor surface; check w floor construction

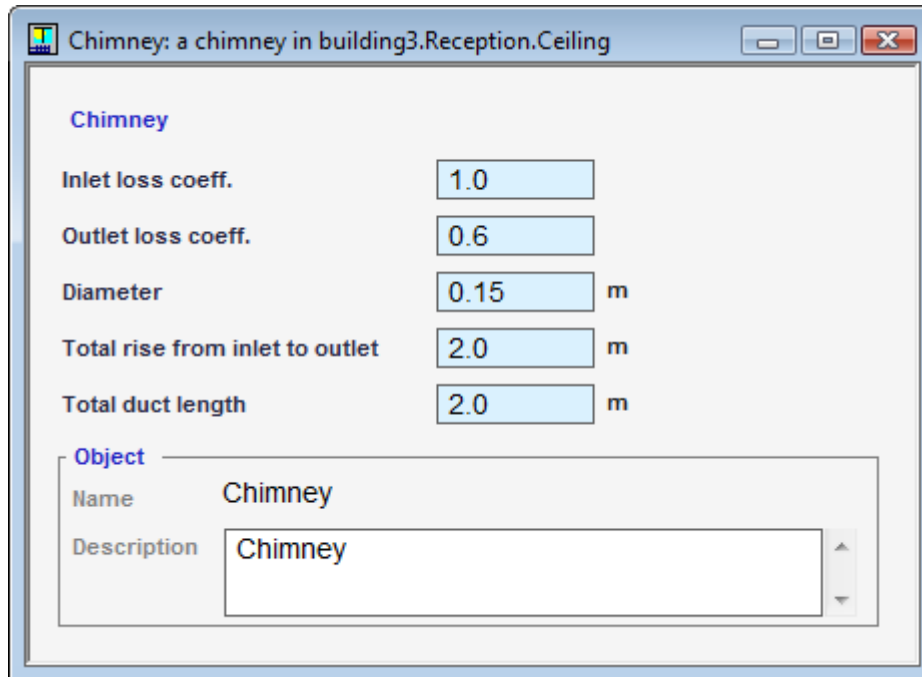
Heat transfer coefficient

H-water-pipe-fin For aluminium fins in a wood joist construction, 6 is a reasonable value; 30 for tubes in concrete. The total heat resistance is normally dominated by the resistance in the floor slab, and thus this parameter is not very critical.

Read about the model for floor heating in Chapter 6 in the manual.

See also Room units for cooling and heating.

7.5.8.7. Chimney



Parameter	Value	Unit
Inlet loss coeff.	1.0	
Outlet loss coeff.	0.6	
Diameter	0.15	m
Total rise from inlet to outlet	2.0	m
Total duct length	2.0	m

Object

Name	Chimney
Description	Chimney

Field description, etc:

Inlet loss coeff. Total pressure loss in the inlet air terminal [-]

Outlet loss coeff. Total pressure loss in the outlet air terminal [-]

Diameter Hydraulic diameter (m)

Total rise from inlet to outlet Height difference between inlet and outlet (governs the stack effect) (m)

Total duct length Total hydraulic length for the duct (provides flow resistance in addition to loss coeffs.) (m)

Object Name and description

The chimney can be inserted in ceiling or wall to describe a natural ventilation system. The model can calculate flow in both directions, i.e. if the zone pressure is low enough, air will enter the zone through the chimney. The rise of the duct and the vertical position of the input will determine the stack effects.

7.6. IDA Resources and Database

7.6.1. Database objects in IDA Indoor Climate and Energy

The following types of database objects, which can be found in the IDA database, are used in IDA Indoor Climate and Energy:

Wall definition

Material

Glass definition (detailed)

Glass pane

Shading material

Venetian blind

Gas properties

Glass definition (simplified)
Integrated shading
Location
Wind profile
Schedule
Profile
External shading
Climate data
Occupant load
Equipment load
Lights
Controller setpoints
Surface
Cooling device
Heating device
Window
Energy price
Settings for new zones

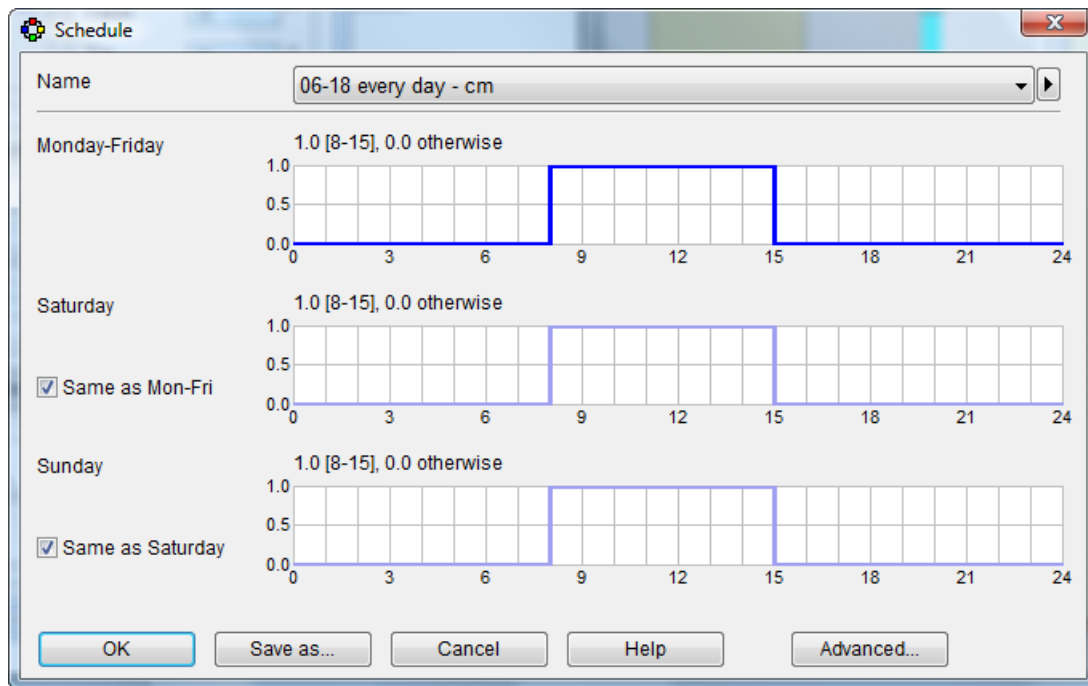
7.6.2. Dialog for Schedule

Schedules are used to define how something varies with time. Examples of this are the presence of occupants or operation time of fans.

The dialog for editing the schedule has two appearances, one simple and one advanced, which correspond to a simple and an advanced definition of a specific schedule. The program automatically controls which one of these is displayed, depending on the character of the schedule. A simple schedule can always be changed to an advanced schedule (by clicking on the Advanced key... in the simple schedule).

Simple schedule dialog

The simple schedule dialog graphically shows the profiles for workdays and weekends. They are edited by drawing horizontal segments. More complex schedules, for example taking account of holidays, are edited in the advanced dialog.



Field descriptions, etc:

Name Choice of Schedule object. The rest of the dialog shows the details of the selected schedule.

Monday-Friday The profile for workdays.

Saturday The profile for Saturday

Same as Mon-Fri Checked if no special schedule should be applied for Saturday

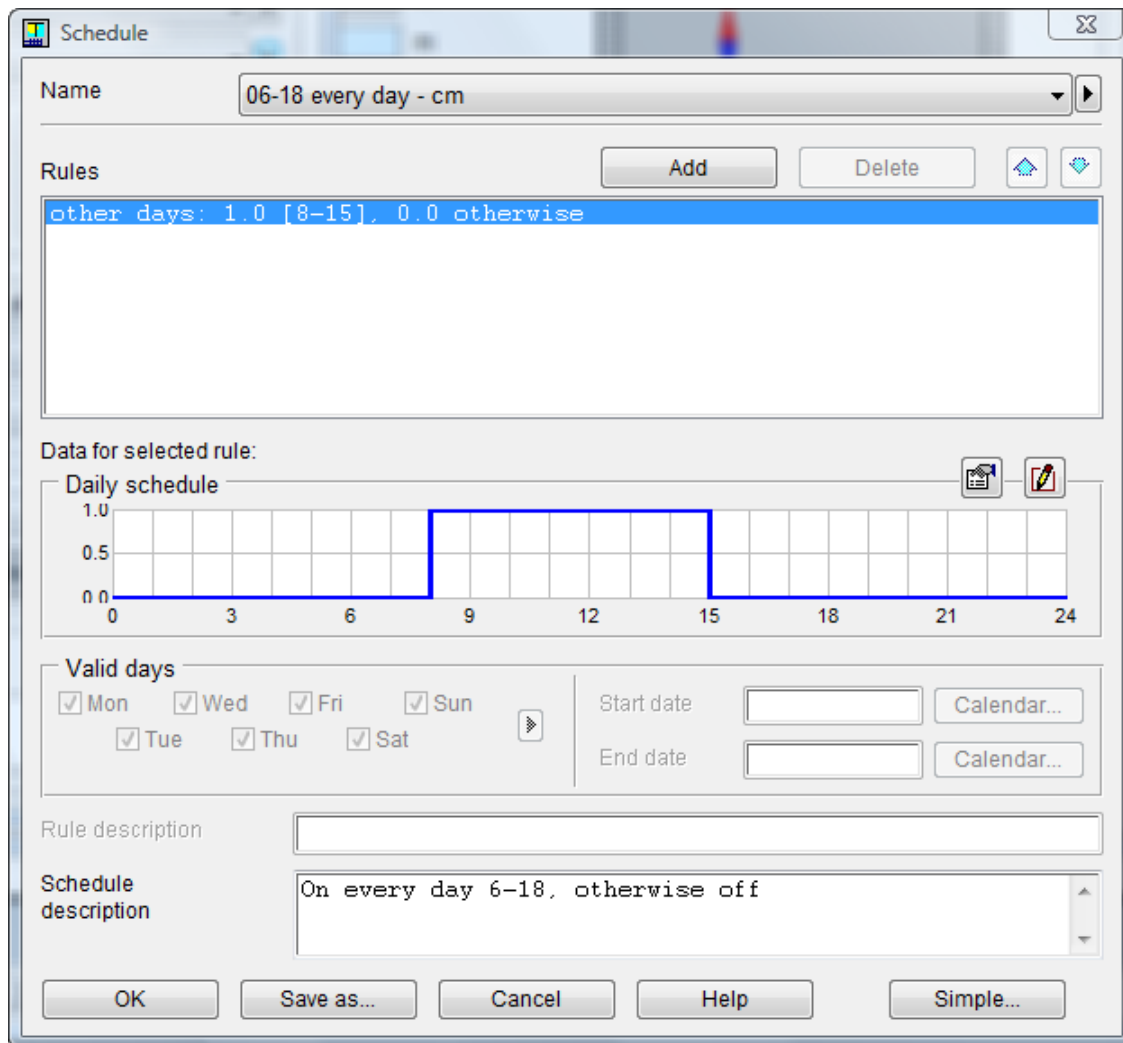
Sunday The profile for Sunday

Same as Saturday Checked if no special schedule should be applied for Sundays

Advanced Show the schedule in Advanced dialog

Advanced schedule dialog

If the simple definition is not enough to define the variation with time, an advanced definition can be given. A schedule has a name, a description and a number of rules. One of these is always in effect. The schedule value at any given point in time is the value of the rule in effect at that time.



Field descriptions, etc:

Name Choice of Schedule object. The rest of the dialog shows the details of the selected schedule.

Description Object description.

Rules The list of schedule rule.

Add Add a new rule to the Schedule.

Delete Delete the selected rule from the Schedule.

Promote Promote the selected rule in the Schedule.

Demote Demote the selected rule in the Schedule.

Daily schedule: Graphically show the daily profile of the selected rule. If the profile is simple enough, it may be edited directly by drawing its horizontal segments by the mouse cursor.

Daily schedule: Diagram settings Click this button to change the appearance of the profile diagram

Daily schedule: Edit profile Click this button for more editing options

Valid days: Select the weekday for selected rule

Valid days: Start date

Valid days: End date

Calendar: Click these buttons to select date from calendar

Rule description Additional annotation to the selected rule.

Schedule description Object description.

Simple: Switch to the Simple schedule dialog

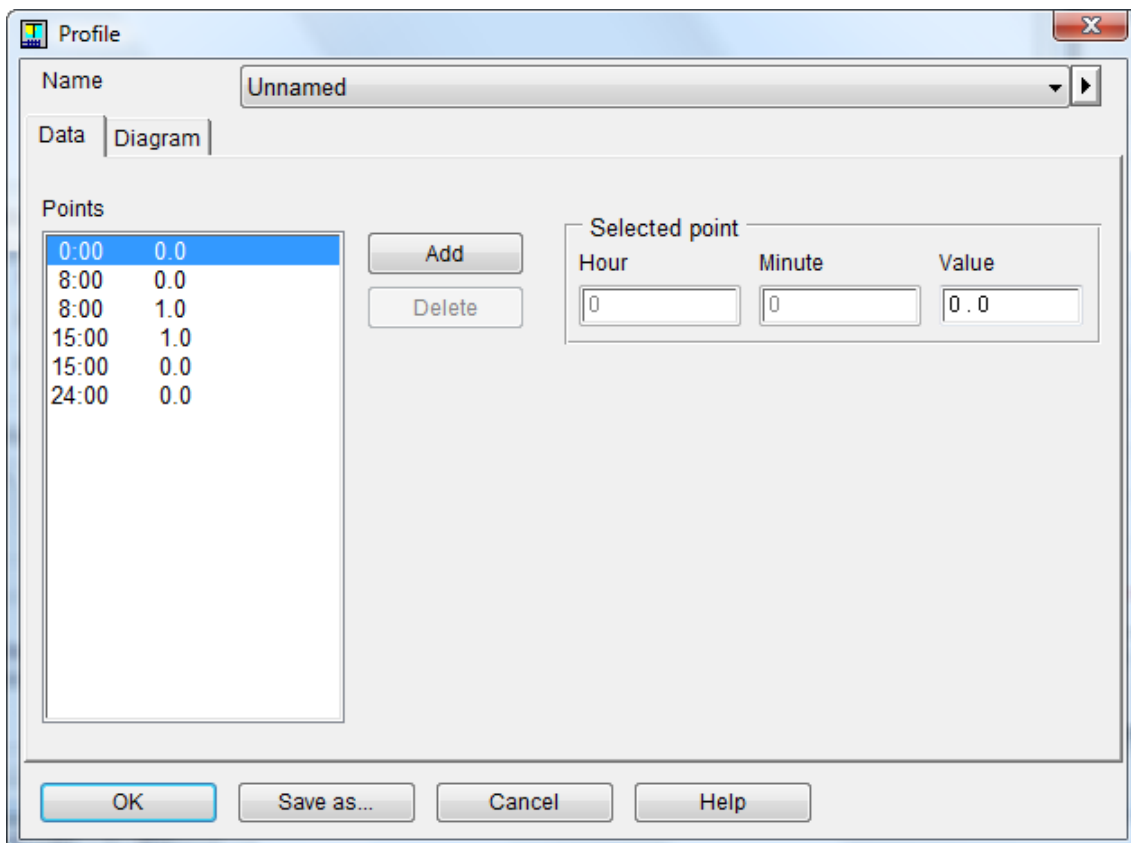
In the Rules list box, new rules can be added with the Add button, and existing rules deleted with the Delete button. The rule furthest up applies in the first hand, and the one furthest down applies last. The rule selected in the list box can be moved up and down with the Move up and Move down buttons, respectively.

The last rule in the list has a special meaning; it is the default rule. The rule output can be changed, but not its time of validity. The default rule is always valid (the Valid days box is inactive when this rule is selected), and the rule also always remains as the last rule. This guarantees that the schedule's value is always defined.

The Validity time for the selected rule is shown under the profile diagram. In the Start date and End date fields the validity time for the rule can be limited to a portion of the year. If no start date is indicated the rule applies from the beginning of the year. If no end date is indicated the rule applies until the end of the year. The Calendar button is used to open the Calendar dialog, which aids selection of an existing date. In the Valid days box, the rule can be limited to apply only to certain days of the week (by crossing those days the rule is applicable).

When a new schedule has been defined it is useful to test that it actually delivers the intended values. This can be done by right clicking the field where the schedule is selected and choosing Open with Diagram. This will "play" the schedule for the time period selected in Time slice on the Options menu.

7.6.3. Editing a Profile



Objects of the Profile type are used to define how a value varies during the day. Examples of these are the presence of people or operation time of fans. A profile has a name, a description and a number of diagram points with time and value. A profile can be edited by indicating

time and value points in text form (Data tab, figure above), or by drawing a diagram (Diagram tab, figure below).

By default the profiles are unnamed and stored as parts of Schedule objects. To create a named profile, select **Save as...** and give a name to a profile. To copy a named profile to an unnamed one, select **Save as...**, remove the profile name and click OK in "Save as..." dialog.
Field descriptions, etc:

Points

Add

Delete

Selected point: Hour Hour.

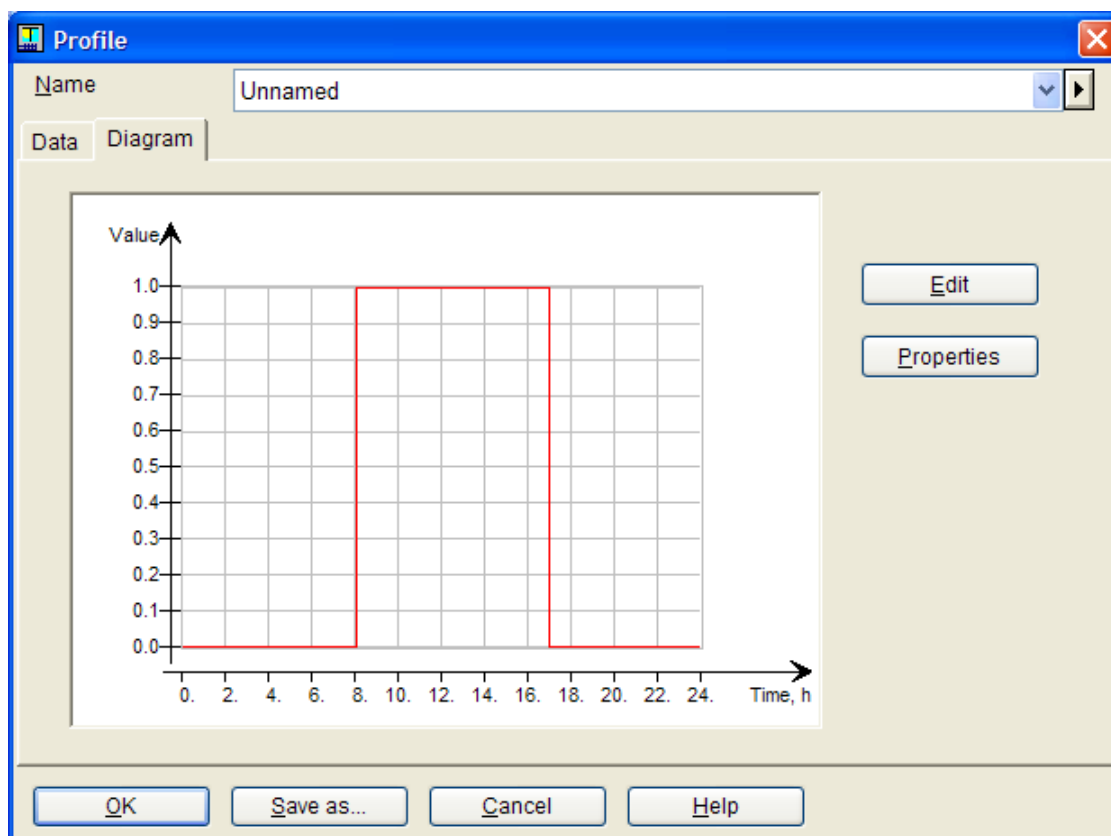
Selected point: Minute Minute.

Selected point: Value Value.

The Dialog box for a profile is opened from the advanced schedule dialog.

In the Points list box, time and value points can be added with the Add button. The Delete button is used to delete a selected point. The times 0:00 and 24:00 can never be deleted. The data values for the selected point can be changed in the Selected point box. Note that the point's value in the list box is not changed until the cursor is moved from the field that has been changed. (Hour, Minute or Value). When times are changed or added, the points are automatically rearranged into time order in the list box.

To edit the profile in diagram form instead, select the Diagram Tab.



Field descriptions, etc:

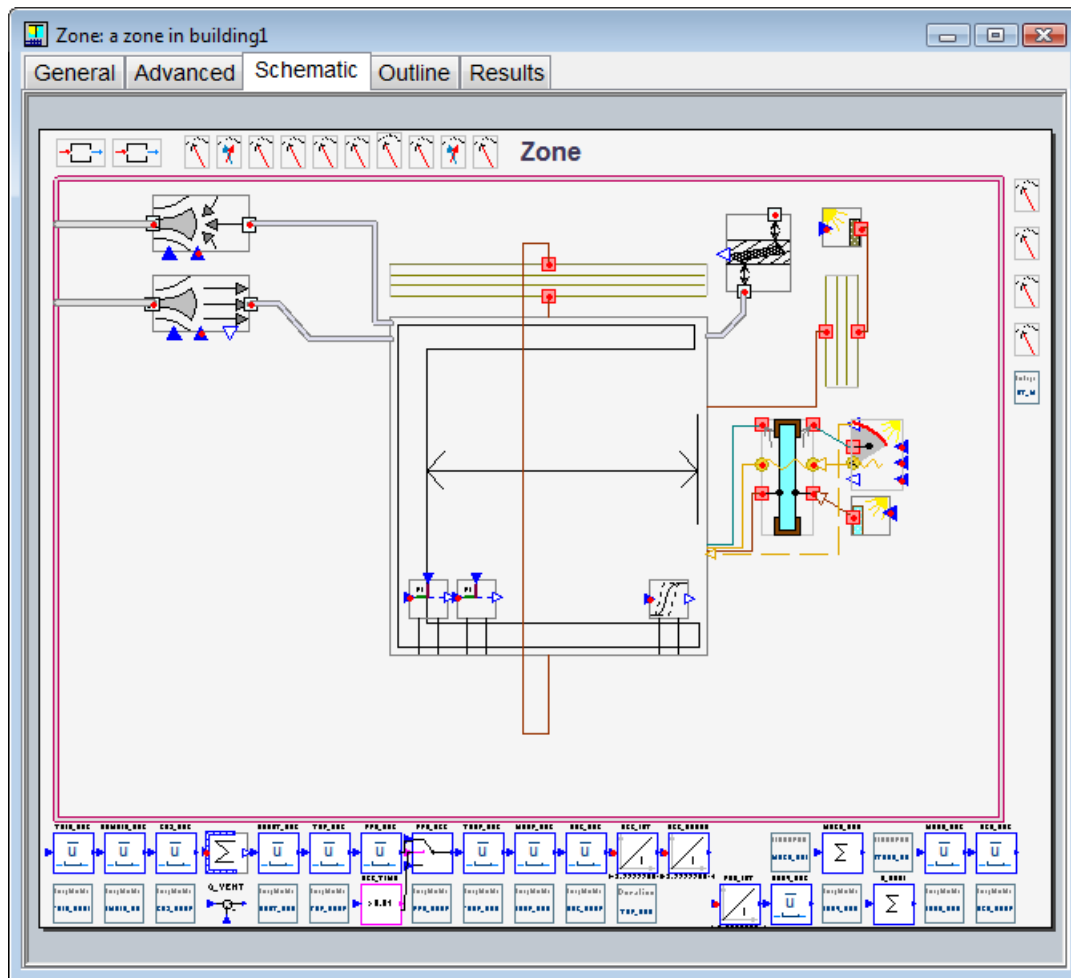
Edit

Properties

Click on the Edit button and the profile breakpoints are marked with small squares in the figure. These can be moved with the mouse. However, the breakpoints can never be moved in such a way that the profile no longer is a pure function of time. When editing is complete, click on the Done button. To cancel editing and return to the original state, click instead on the Abort button. The Properties button is used to change the properties of the diagram, the scale of the axes among other things.

7.7. Mathematical Model

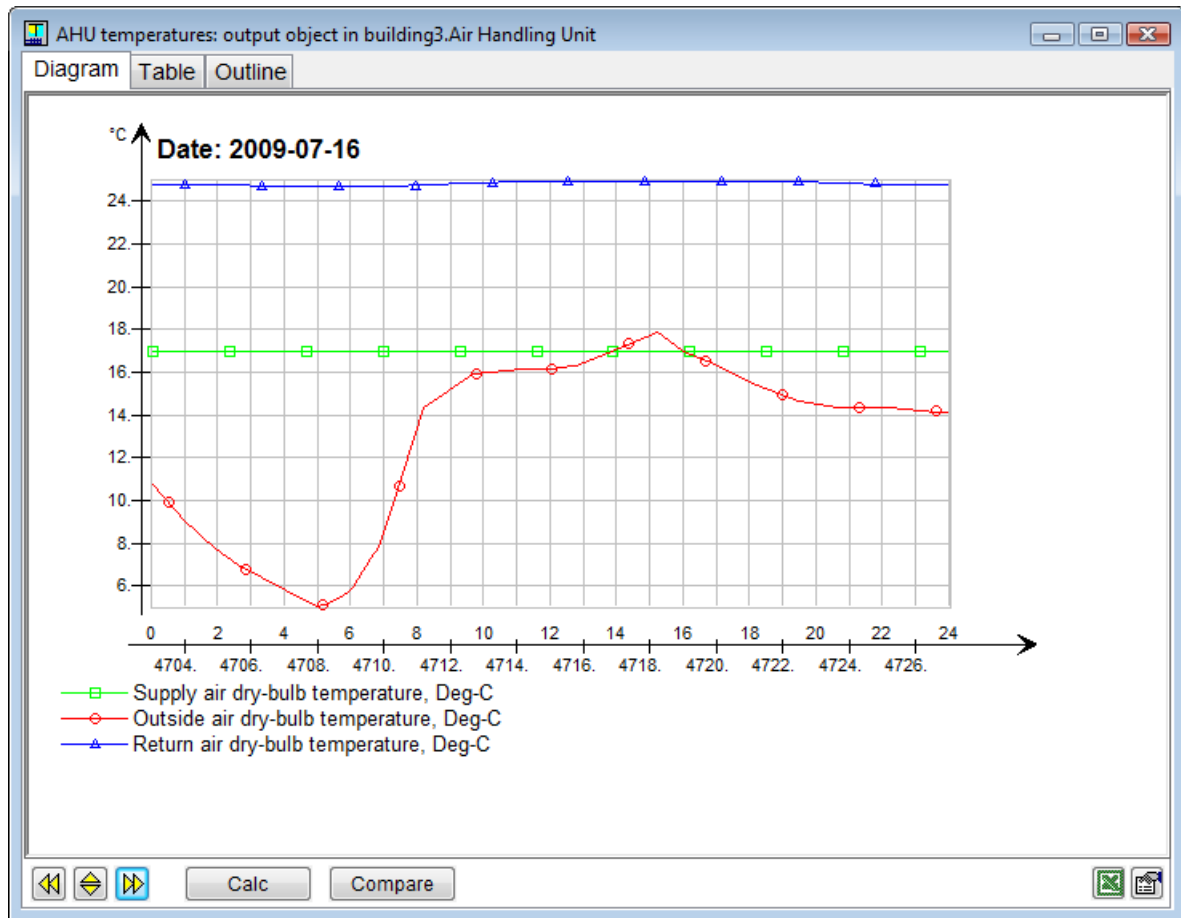
7.7.1. Schema for zone (advanced level)



See the IDA ICE manual.

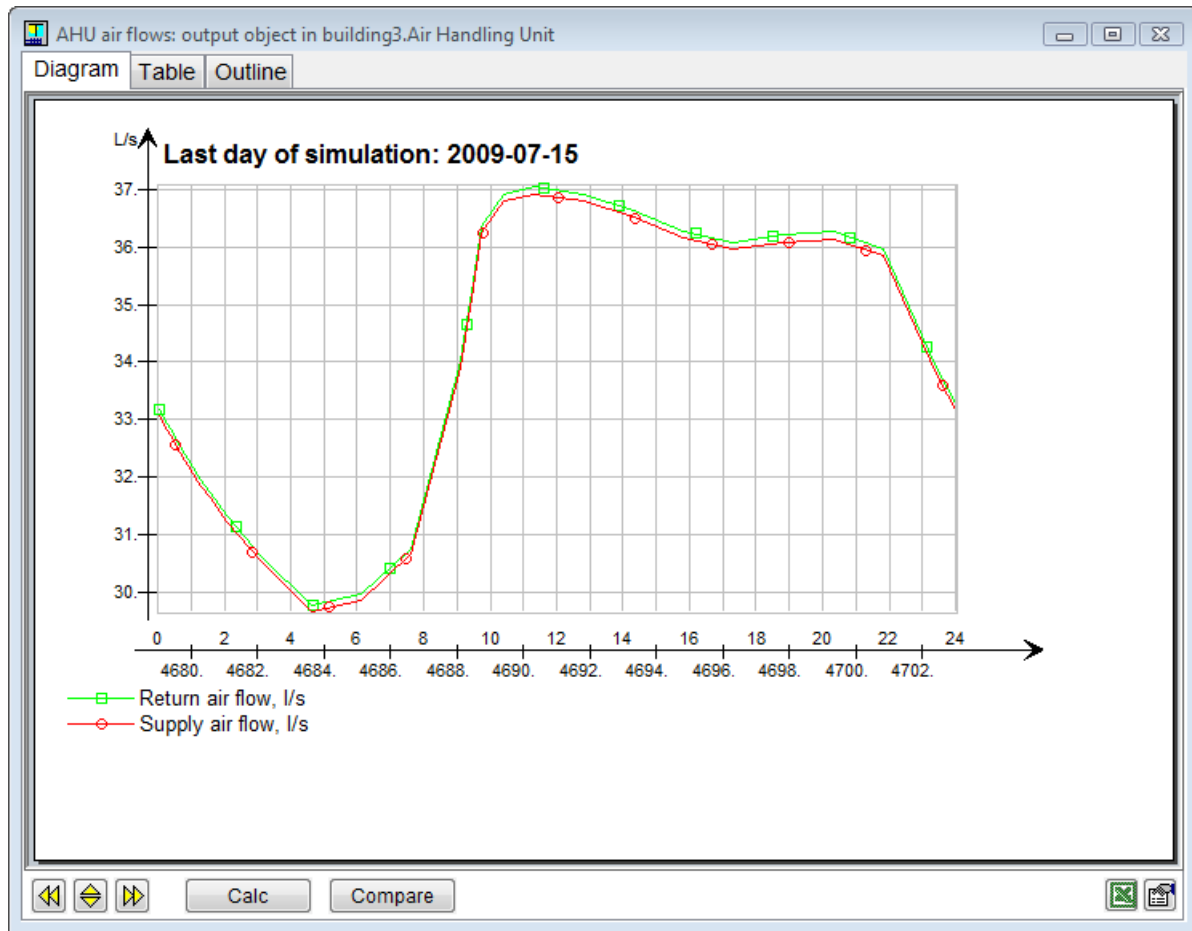
7.8. Results

7.8.1. Result – AHU (Air Handling Unit) temperatures



This diagram displays return air, supply air and outdoor air temperatures. The return air temperature is the mixed temperature where each zone's air is multiplied by weights (number of zones of this type) given in the zone's form. Air supply temperature gives the temperature, which reaches the zone's air terminal after any increase in the fans and duct system. Note that the setpoint for the supply air temperature applies to the air temperature before this increase.

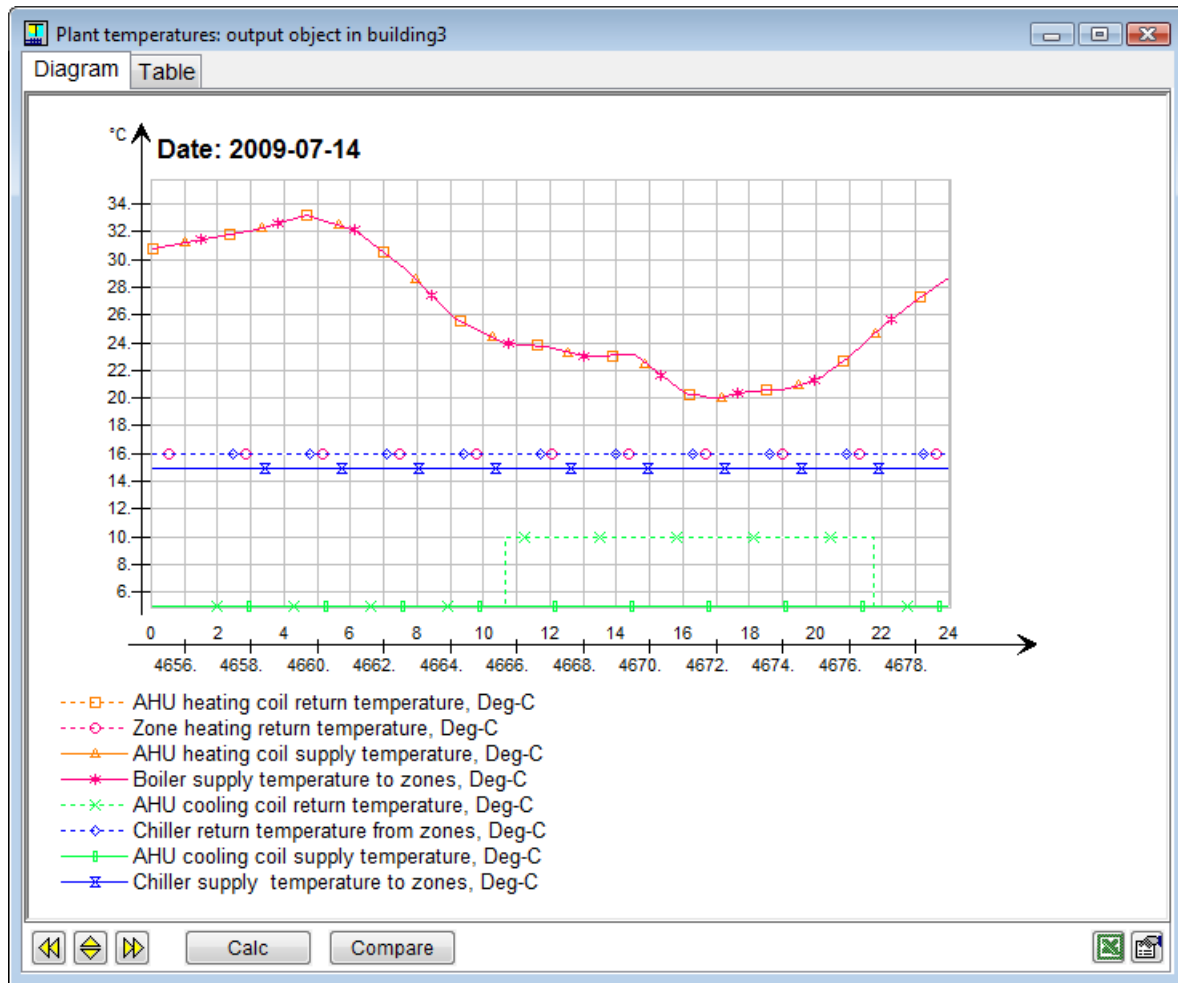
7.8.2. Result – Airflow through the Air Handling Unit



The flows accounted for are the combined flows for all zones served by the AHU. The flows in the zone are multiplied with weights (Number of zones of this type) which are indicated in each zone's form.

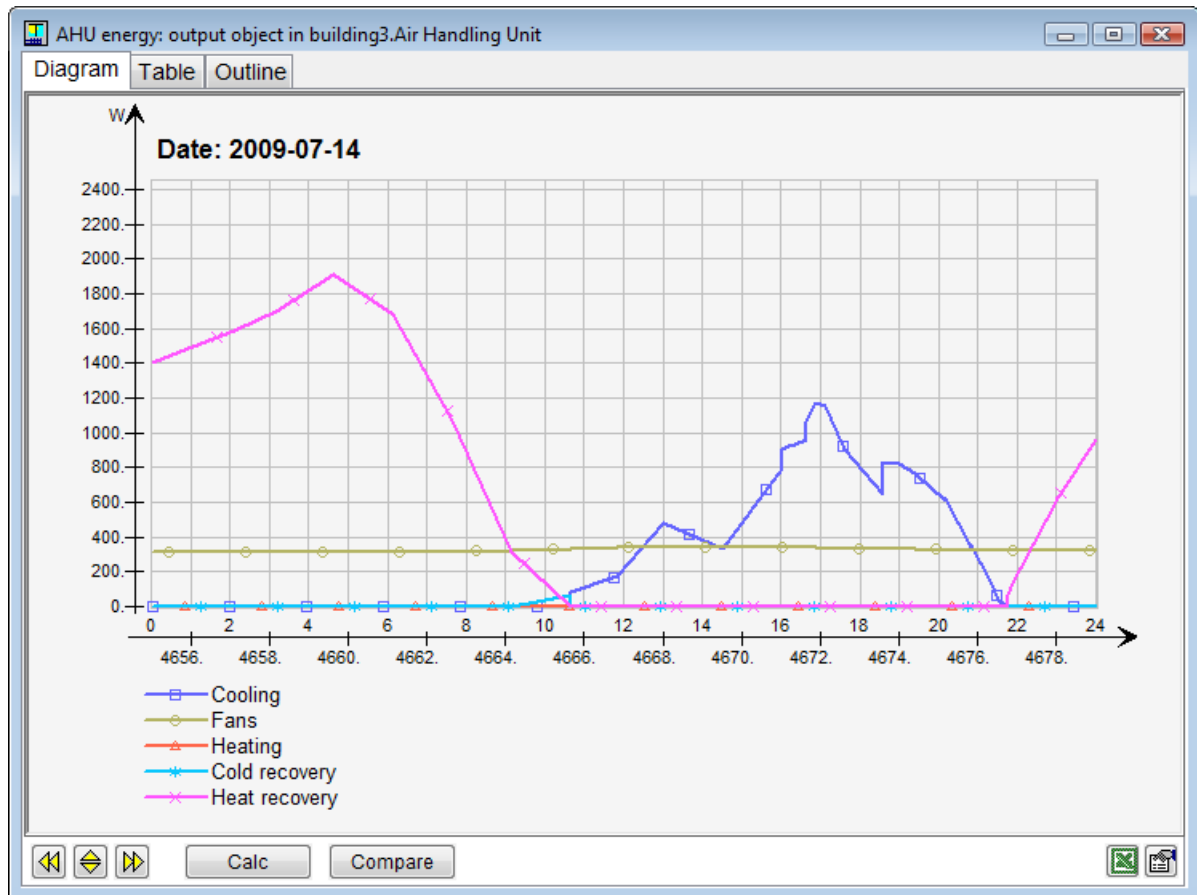
Flows are volumetric as measured at actual temperature conditions. They may therefore vary slightly with respect to zone setpoint flows, which are given as massflows, due to density variations.

7.8.3. Result – Primary system's temperatures



The supply and return temperatures for the primary system are displayed. Return temperatures from local cooling and heating devices are weighed together with weights (Number of zones of this type) which are indicated in the zone's form.

7.8.4. Result – Total heating and cooling

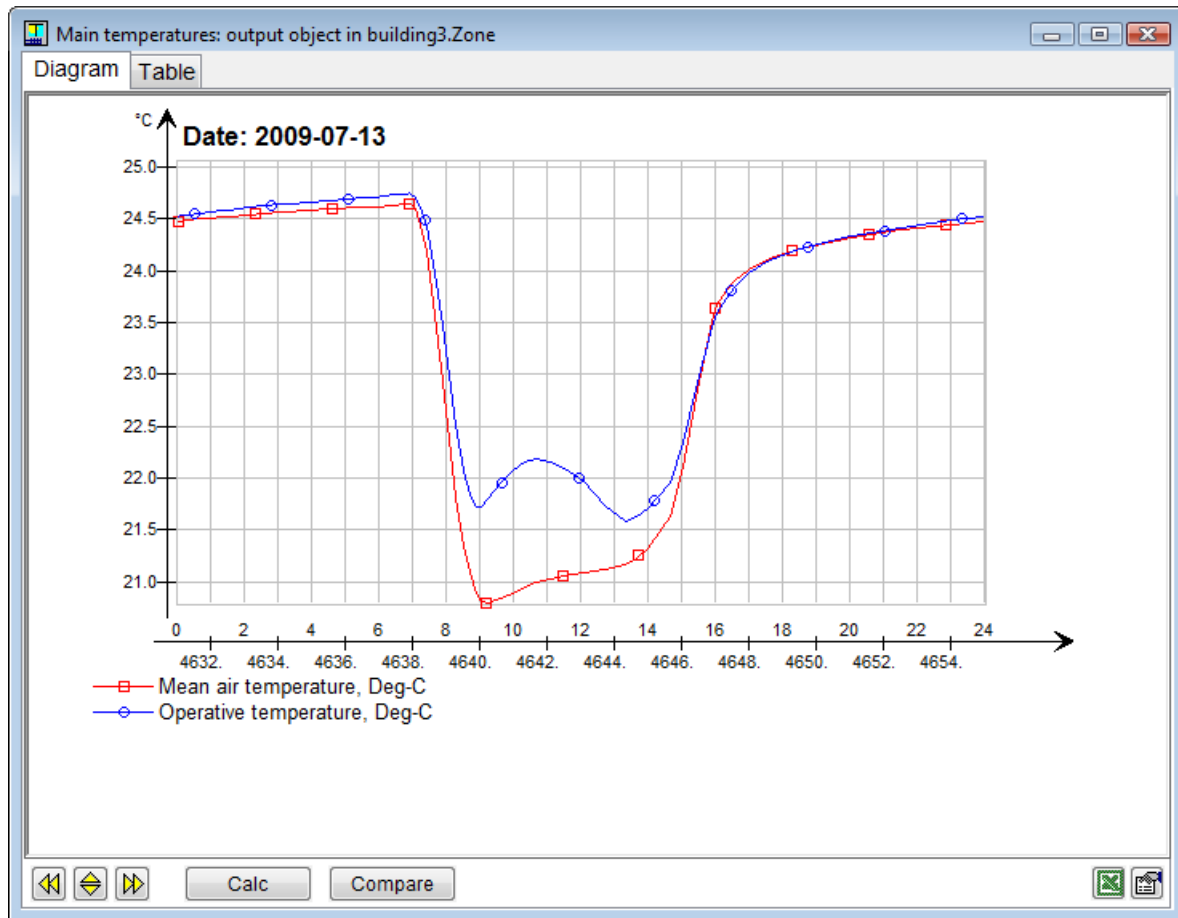


This diagram displays central and local heating and cooling power.

For water based circuits, the power is calculated based on the mass flow and temperature changes (supply – return) in respective plant circuit. Thus, this heat is computed after generation losses (COP) have been accounted for but before any distribution and emission losses. Cooling power is accounted for as a positive quantity.

The analogous results are presented for local ideal heaters and coolers, although these are not served by the plant.

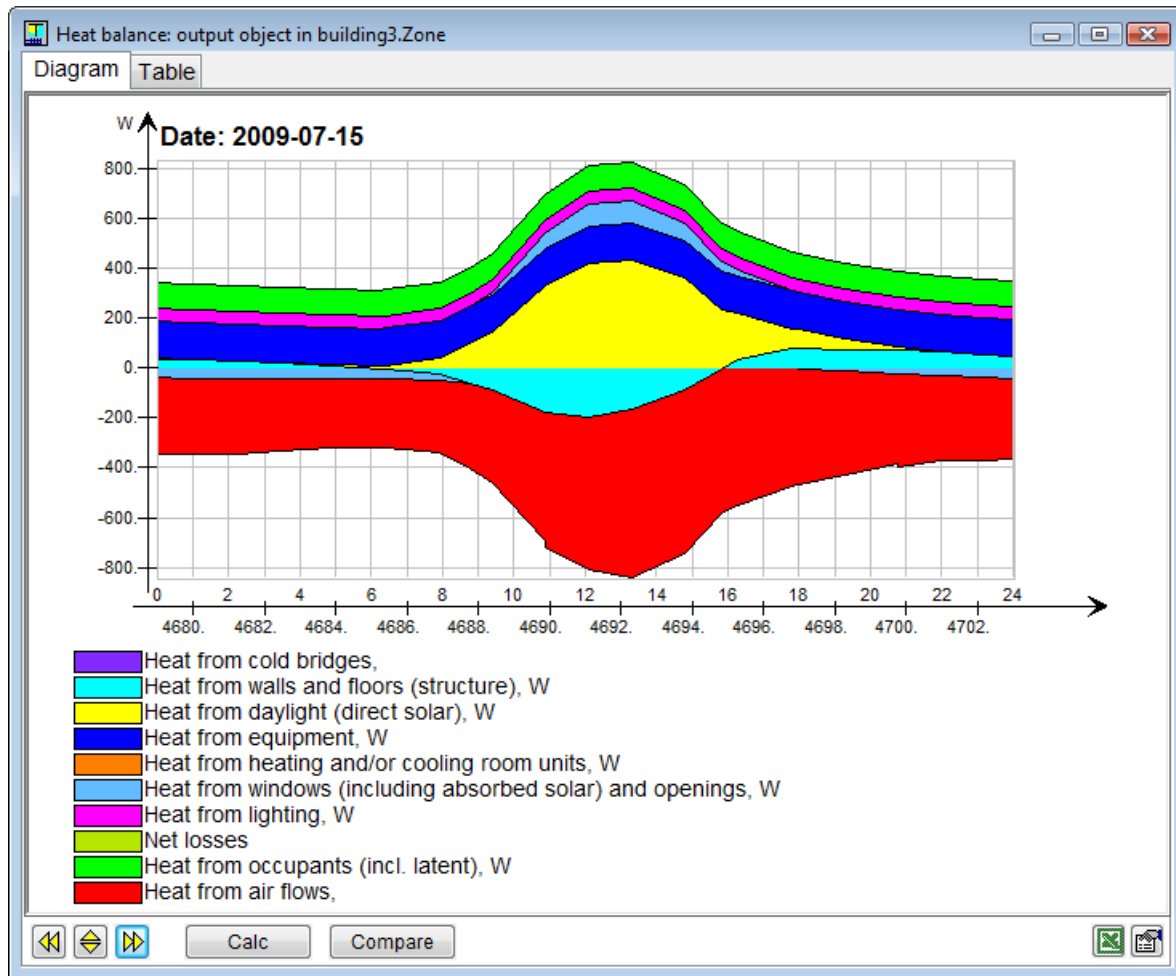
7.8.5. Result – Main temperatures



This diagram shows the mean temperature for the zone's air and operative temperatures for the locations of added occupant loads. When Climate model has been selected, the operative temperatures are shown for the position of every occupant load in the zone.

Occupant loads are inserted in the zone form and their position may be altered in the floor surface editor of the zone.

7.8.6. Result – Heat balance



This is the full latent (moist) and sensible (dry) heat balance of the zone. For a sensible only approximate heat balance, that may be easier to understand, log the details of the zone Energy report instead.

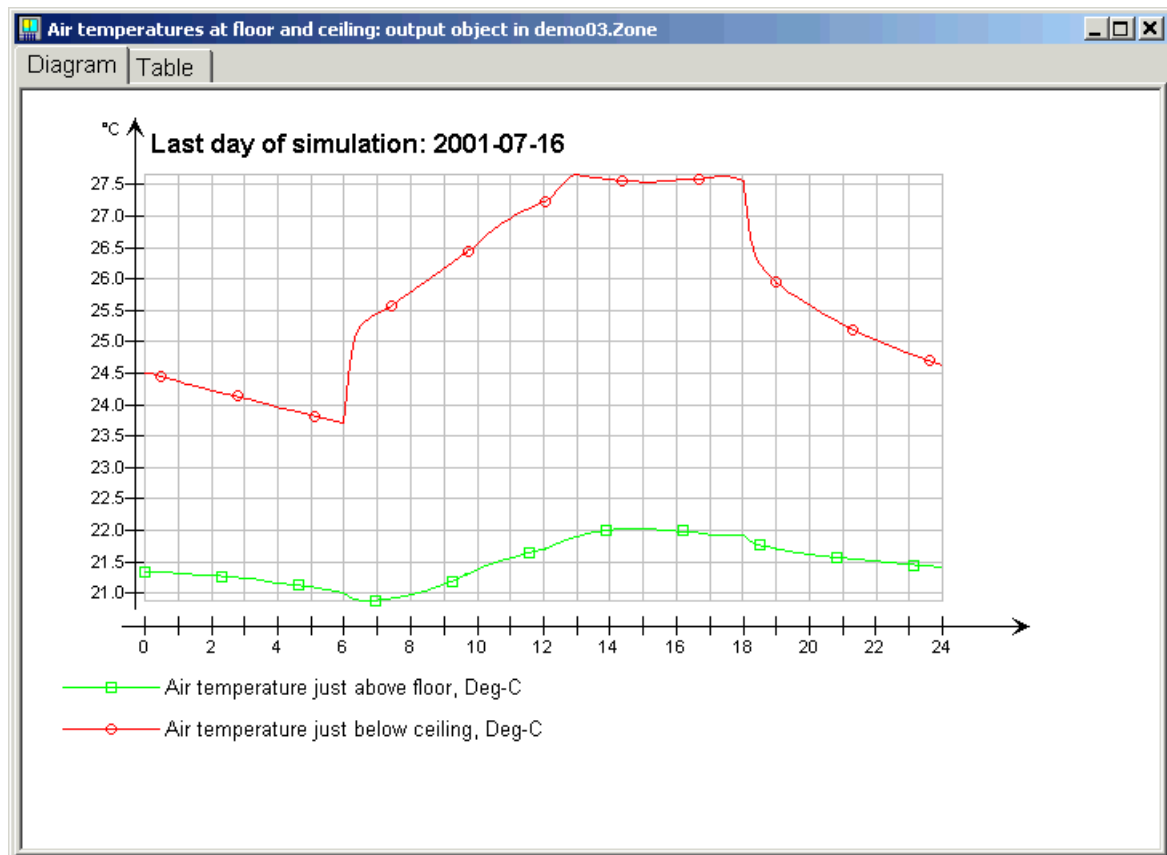
The control volume is the zone air wetted surface area, running on the zone side of any room units with an air gap behind. Contributions are divided into the following categories:

<i>category</i>	<i>comment</i>
Heat from thermal bridges	-
Heat from walls and floors (structure)	In the account, the control volume is just underneath each surface. Accordingly, the measure represents conductive heat through the structure including both storage, net transmission and any internal heat sources such as floor heating. Heat stored in internal masses, e.g. furniture, is also accounted for here.
Heat from daylight	Heat from sunlight entering through windows or open doors, minus corresponding exiting shortwave radiation. Absorbed and then retransmitted solar radiation is not included here (see below).
Heat from equipment	Heat from appliances such as computers, printers etc. Emitted as convection or radiation according to user input.

Heat from heating and/or cooling room units	Heat from controlled room units such as chilled beams, radiators and ideal units. For hydronic devices, the split between convection and radiation is automatically calculated (see manual.) NB! Floor heating is <i>not</i> included here but is regarded as coming from floor and ceiling.
Heat from windows (including absorbed solar) and openings	Heat from window surfaces, including both conducted heat and retransmitted absorbed solar radiation. Longwave radiation through openings (open doors) is included here. Incoming solar radiation affects room conditions in two ways: <ol style="list-style-type: none"> <li data-bbox="734 660 1382 772">1. as directly transmitted (short wave) radiation, which is repeatedly reflected at room surfaces but is ultimately absorbed by these <li data-bbox="734 772 1382 884">2. as heat which is first absorbed in the window pane/blind and then reaches the room through convection and radiation.
	The latter is accounted for here together with window transmission. The first is separately displayed (see below).
Heat from lighting	Electrical light power, emitted as convection or radiation according to user input.
Net losses	Net heat from distribution losses from pipes and ducts that are deposited in the zone. See Extra energy and losses
Heat from occupants (incl. latent)	Includes both sensible (dry) and latent (moist) heat emitted by the occupants. The moisture is ventilated away and will in this way be included in the air flow heat balance.
Heat from air flows	All air flows are accounted here, i.e. mechanical ventilation as well as infiltration and flows from other zones. For a detailed account of airborne heat flows, see also separate result object: Airborne heat flow into zone.

7.8.6.1.

7.8.7. Result – Air temperature at floor and ceiling

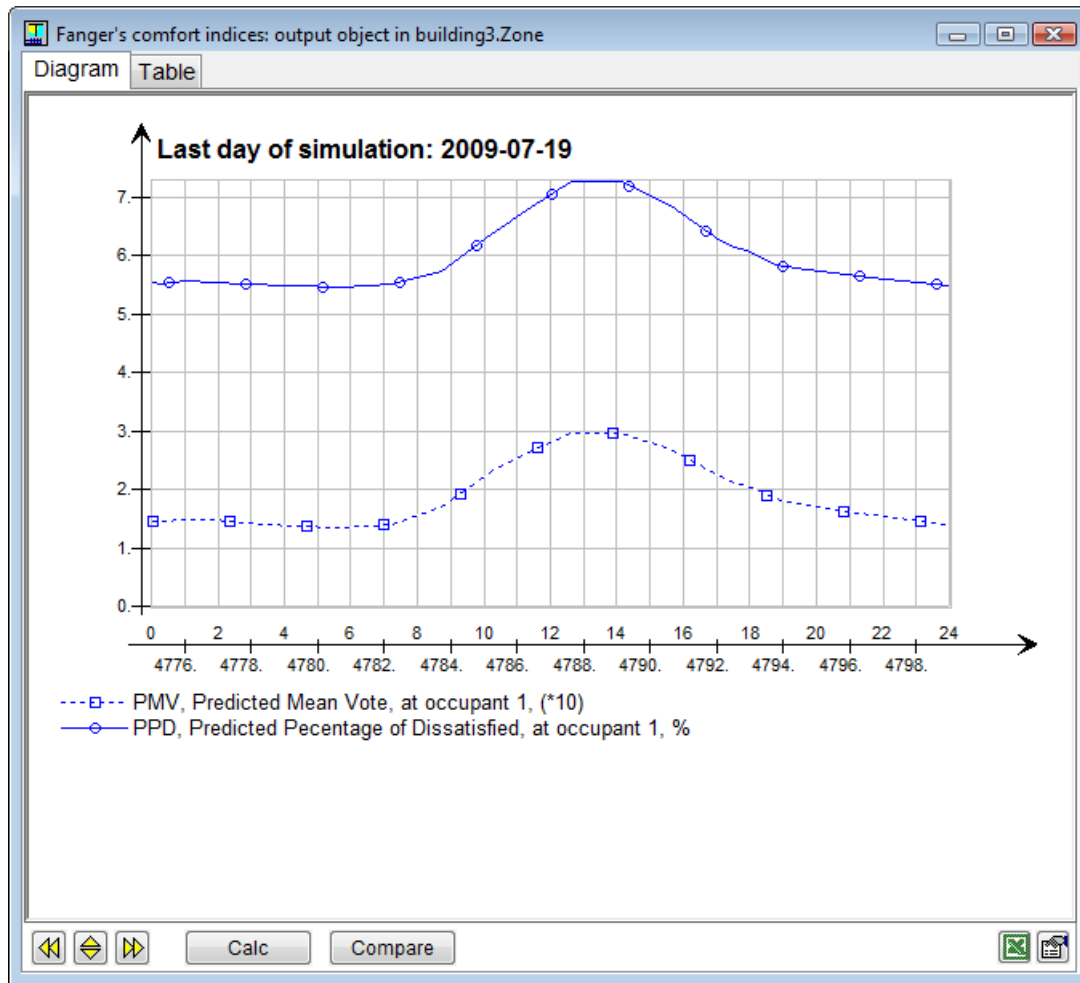


When a gradient calculation in the zone has been selected (non-well mixed zone), all temperatures become dependent on the height from the floor. Examples of influences of this are:

- - comfort around occupants
- - air removed via return air or leaks
- - convection at zone surfaces.

The result object is only available if Climate model has been selected in the zone's form.

7.8.8. Result – Fanger’s comfort indices



PPD =Predicted Percentage of Dissatisfied

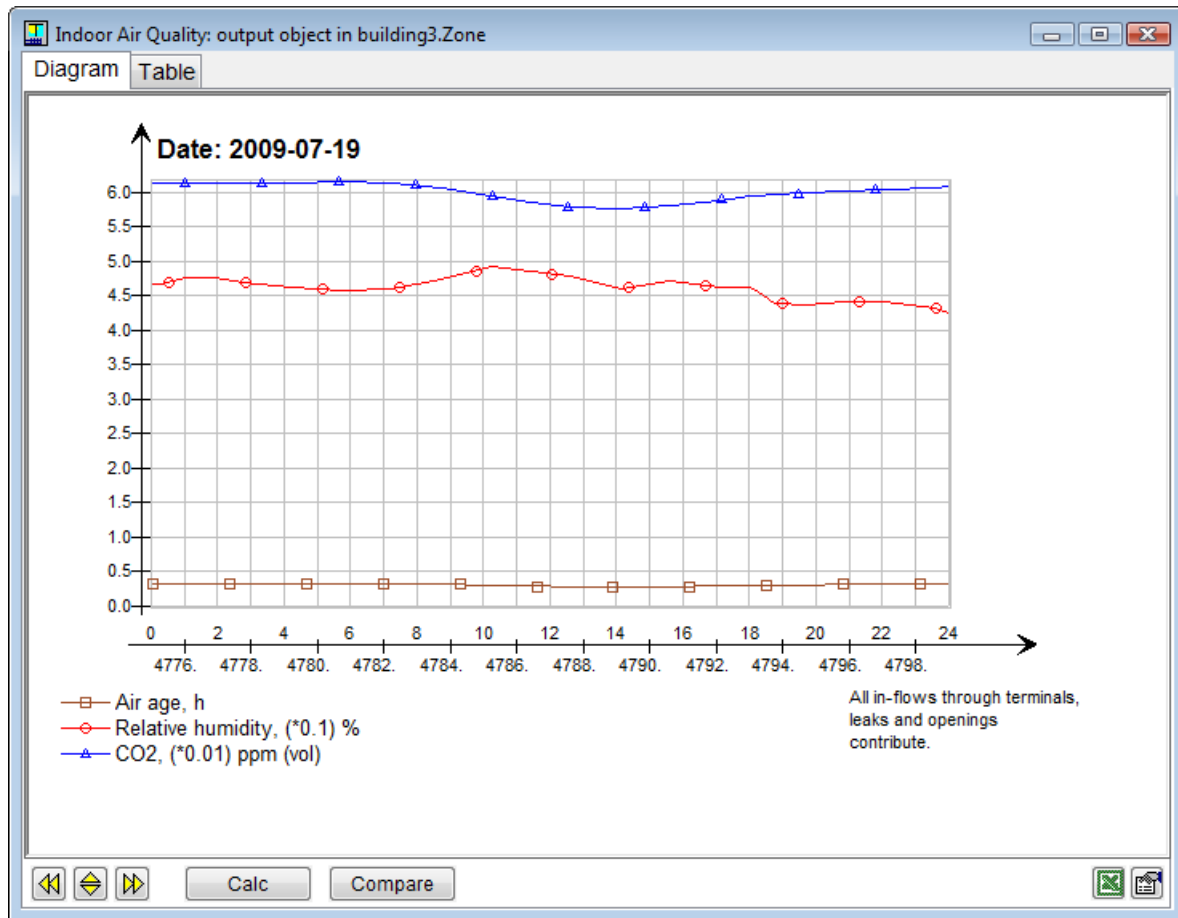
PMV = Predicted Mean Vote

These measures of comfort take into consideration temperature, radiation, moisture and draught as well as occupant clothing and level of activity. PPD varies between 0 and 100 and the ideal value is 5, i.e., at least five percent are always dissatisfied. If occupants in the zone think it is too warm or too cold, PMV varies between +3 (too warm) and -3 (too cold), and should preferably lie close to zero. NB! The measurement for PMV has been multiplied by 10 in the diagram.

The comfort measurement is calculated only for times when respective occupant loads are present.

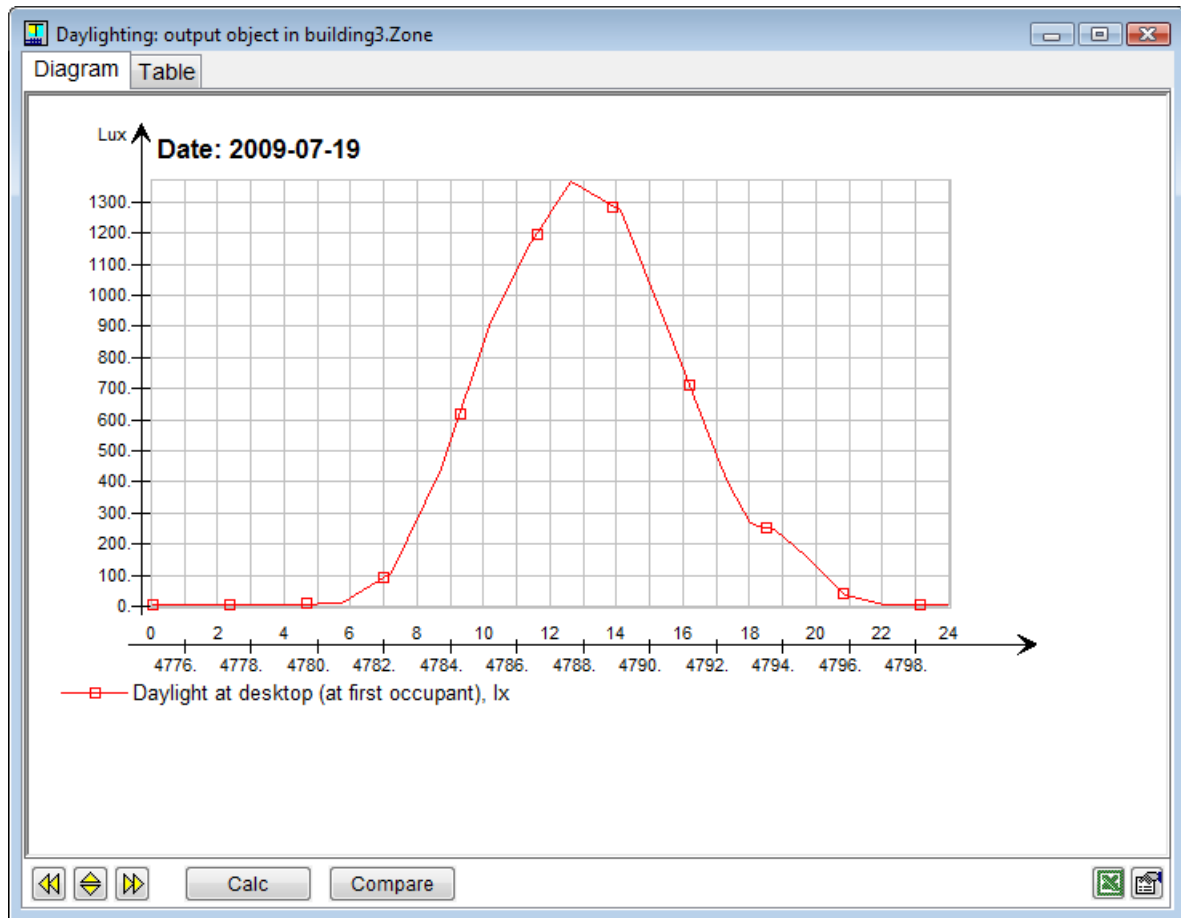
For further information about the index see ASHRAE Fundamentals.

7.8.9. Result – Air quality



Air age. A measure of how long an average air molecule has spent in the building. If a zone is ventilated by outside air only and is in steady state, this number is 1/'air changes per hour'. The age of air measure takes account also of air that has aged in neighboring zones. CO2 content given in ppm (with respect to volume), but is scaled here with a factor of 0.01 to make the diagram more understandable. Normal limits during demand controlled ventilation is 1000-1200 ppm, but considerably higher values can occur without health risks. Air humidity indicated in %, but has been scaled with a factor of 0.1 to make the diagram more understandable. Suitable limits vary greatly with the zone use.

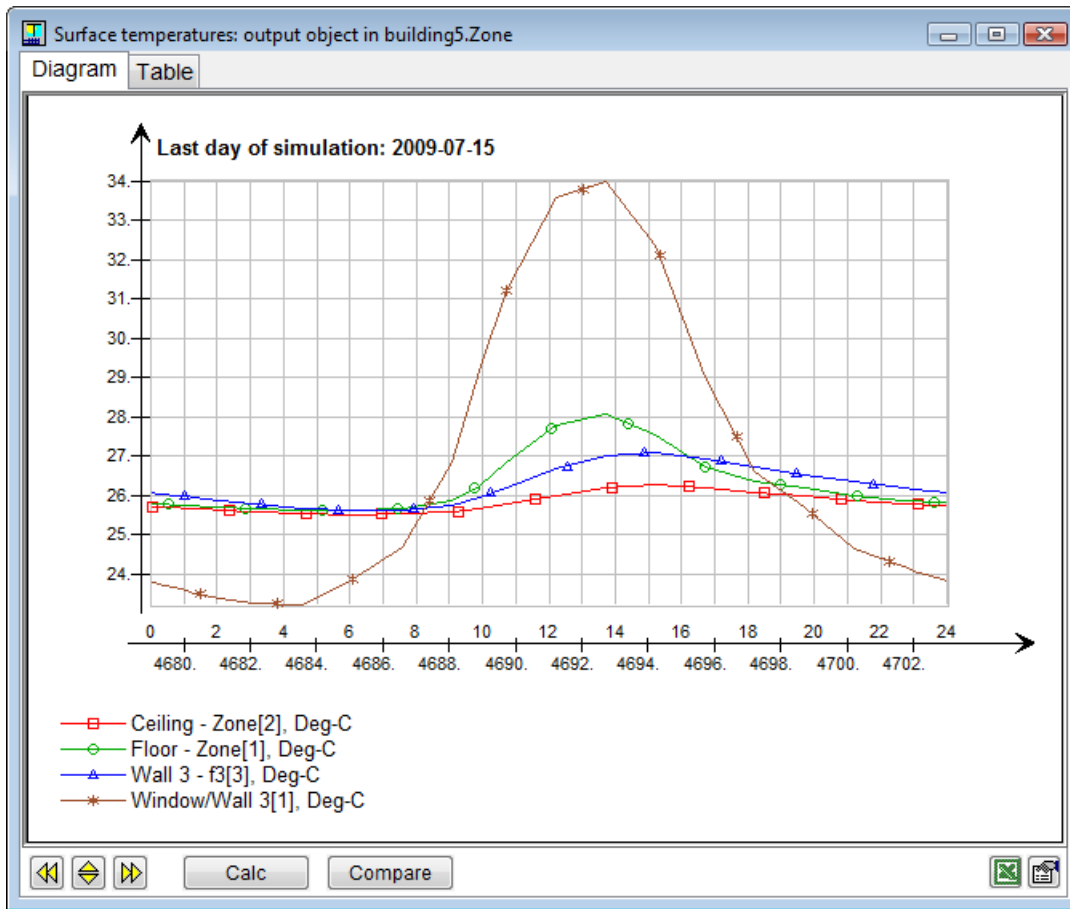
7.8.10. Result – Daylight level



Shows lighting by default on a horizontal surface at the first occupant load. Only daylight, not electric lighting is accounted for.

Average light levels in the zone can alternatively be measured if this is selected in System parameters.

7.8.11. Directed operative temperatures

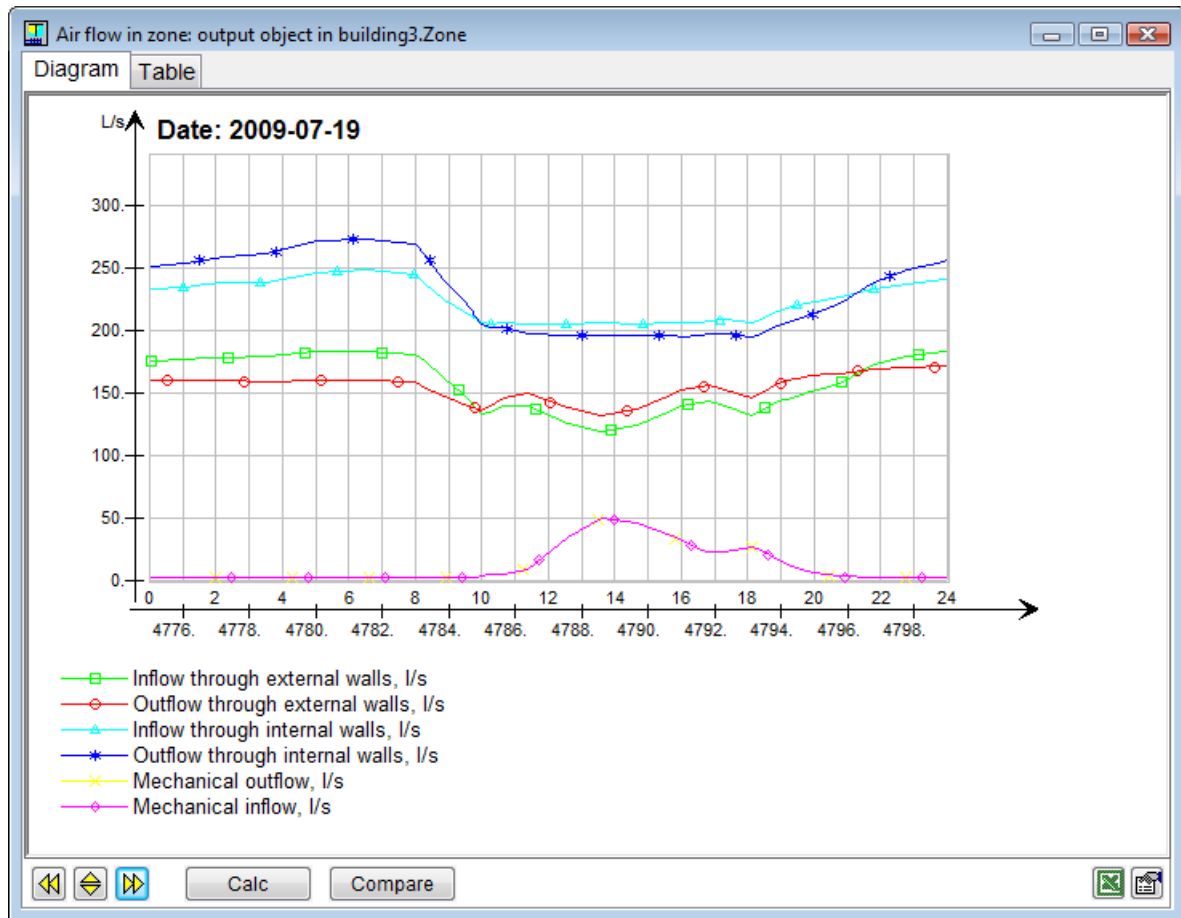


The directed operative temperatures are calculated for the position of the first occupant load in each of the six main directions. The positions of occupant loads are displayed as chairs in the surface editor of the floor. The height coordinate can be given adjacent to the symbol.

The directed operative temperature is calculated as the average of the local air temperature and the mean radiant temperature from surfaces that are visible in the current direction.

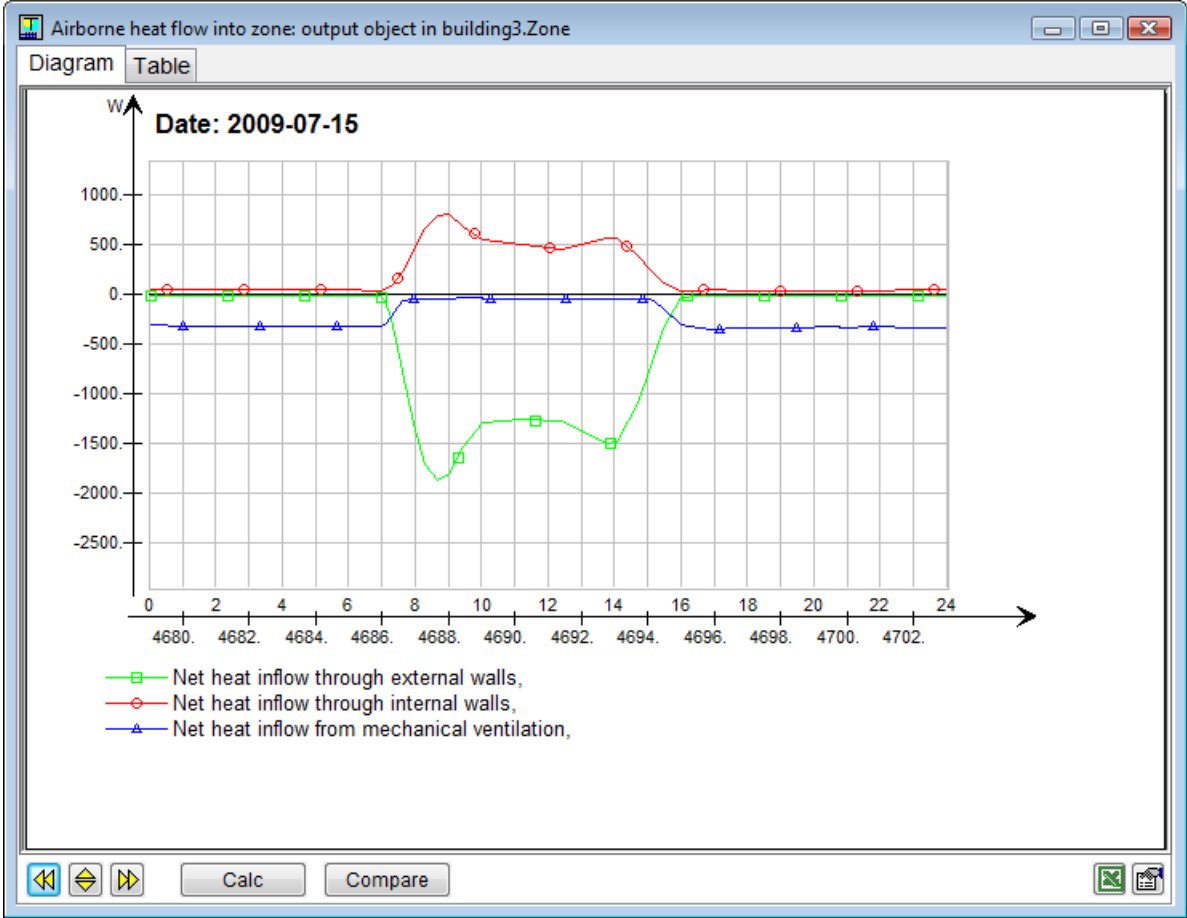
The result object is only available if Climate model has been selected in the zone's form.

7.8.12. Air flow in zone



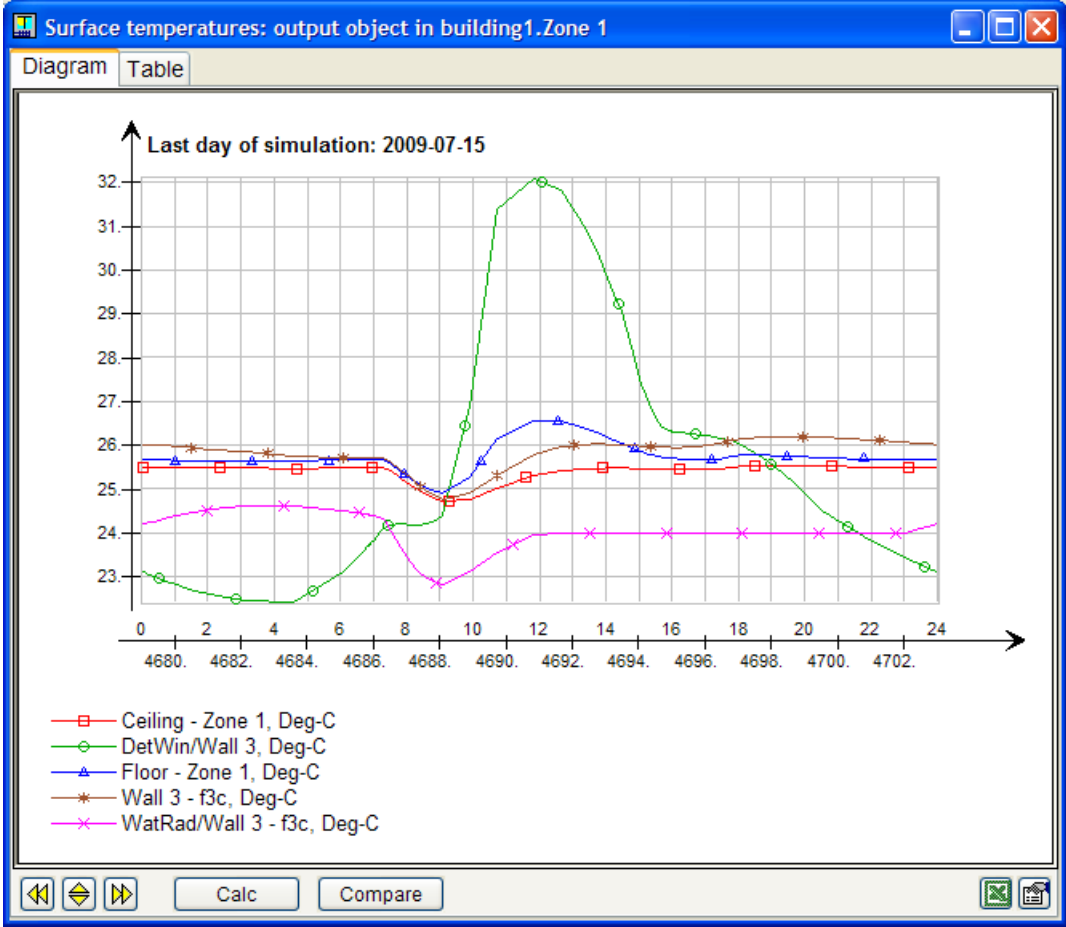
Displays volumetric air flows through openings, leaks and mechanical ventilation.

7.8.13. Airborne heat flow into zone



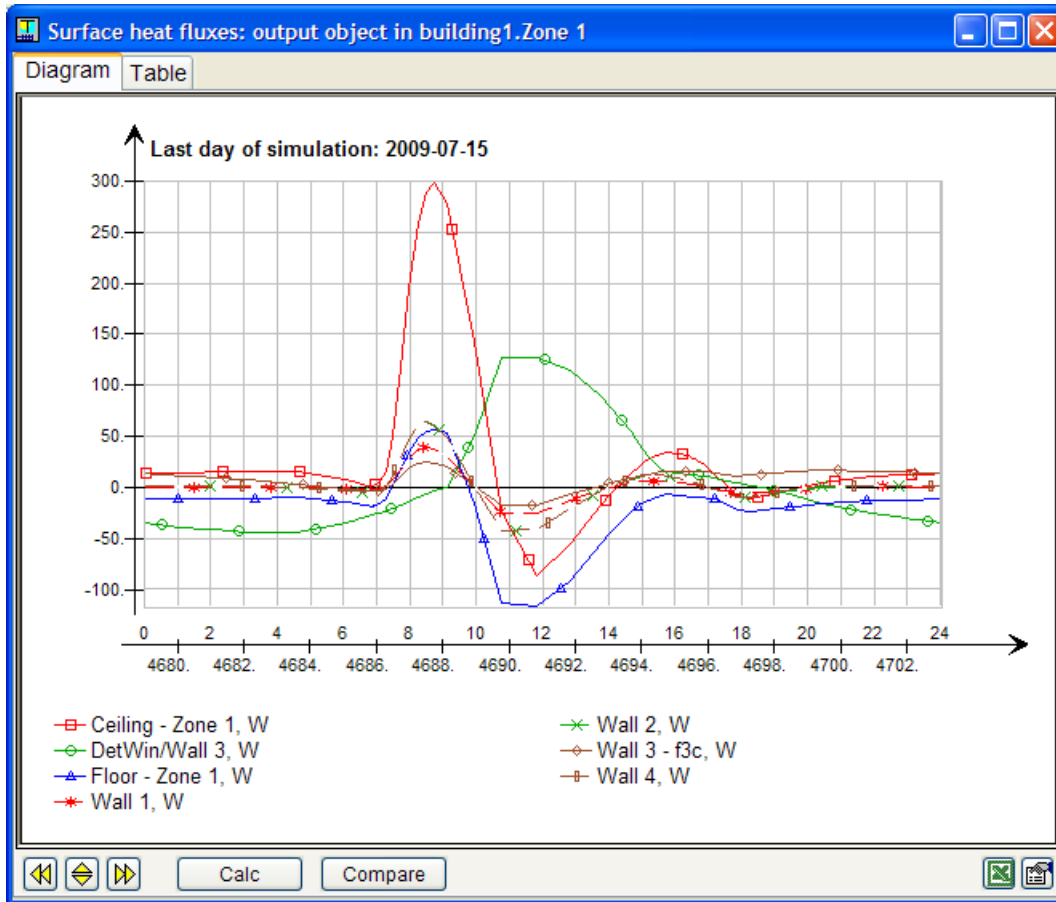
Displays heat flows (sensible and latent) via supply air streams through openings, leaks and mechanical ventilation.

7.8.14. Surface temperatures



Displays temperatures of individual zone surfaces.

7.8.15. Surface heat fluxes



Displays heat fluxes through individual zone surfaces. For transparent surfaces or openings, short wave radiation is not included.

7.8.16. Result – Supplied Energy

This report gives an overview of the total energy that has been purchased to (or generated within) the building. The items in the report correspond directly to defined Energy meters. In addition to, meter energy, cost, emitted CO₂ and used primary energy are also presented, both as absolute values and in relation to building floor area. The conversion factors between meter energy and the other measures are defined in Energy meters. If these factors are missing, no data is printed for the missing item in the Supplied Energy report.

Consumed amount of fuel is measured in terms of its heating (calorific) value in kWh.

The time evolution of the individual terms can be followed by selecting Log sources in the List of output objects.

7.8.17. Result – Systems energy

The report gives an overview of energy flows in the HVAC systems. Generally, results represent heat delivered by the plant and air handling units to the building distribution systems, i.e. flows will be reduced by distribution and emission losses before they reach their end use.

For cases with specified distribution losses, a separate table of distribution losses is presented.

Data for the systems energy overview is presented in the following categories:

<i>category</i>	<i>comment</i>
Zone heating	Heat from controlled room units such as, ideal heaters, radiators and fan coils. Floor heating <i>is</i> included. This heat will normally be reduced by distribution and emission losses before it reaches the occupied space.
Zone cooling	Cooling from controlled room units such as ideal coolers, chilled beams and panels. This cooling will normally be reduced by distribution and emission losses before it reaches the occupied space.
AHU heating	Heating energy supplied to all heating coils of air handling units. No distribution losses are applied to circuit between plant and AHU.
AHU Cooling	Energy removed by all cooling coils of air handling units. Note that a significant part of this energy may be latent due to condensation in coils. No distribution losses are applied to circuit between plant and AHU.
Heat recovery	Heating energy recovered in AHU air-to-air heat exchanger, mixing box or in heat recovery devices in the plant.
Cold recovery	D:o for cooling.
Humidification	Energy <i>used</i> by any central humidification equipment.
Fans	Energy <i>used</i> by fans. Fan electricity consumption is by default based on computed air flows and user supplied pressure heads and efficiencies.
Pumps	Energy <i>used</i> by pumps. Pump electricity consumption is by default based on computed water flows and user supplied pressure heads and efficiencies.
Domestic hot water	Energy delivered to domestic hot water circuit. The consumption rate is given in Extra energy and losses. Note that actual DHW end use may be smaller due to distribution losses.

Data for the distribution losses overview is governed by coefficients specified in Extra energy and losses. Results are presented in the following categories:

<i>category</i>	<i>comment</i>
Domestic hot water circuit	Heat lost from water in domestic hot water distribution systems, either due to permanent

Heating	circulation or to intermittent tapping. Heat lost from water in hot water distribution systems.
Cooling	Cooling energy lost from water in cold water distribution systems
Air ducts	Cooling energy lost from ductwork, both from transmission and air leakage.

A given percentage of each loss is emitted into zones according to floor area.

7.8.18. Result – Energy report for an air handling unit

The report gives an overview of energy flows in an individual AHU.

Data is presented in the following categories:

<i>category</i>	<i>comment</i>
Heating	Heating energy supplied to all heating coils.
Cooling	Energy removed by all cooling coils. Note that a significant part of this energy may be latent due to condensation.
Heat recovery	Heating energy recovered in AHU air-to-air heat exchanger or mixing box.
Cold recovery	Do for cooling.
Humidification	Energy <i>used</i> by humidification equipment.
Fans	Energy <i>used</i> by fans.

7.8.19. Result - Lost work

EQUA
SIMULATION TECHNOLOGY GROUP

Lost work, due to under or over heating

Project		Building	
Customer		Model floor area	10.0 m ²
Created by	kursdeltagare	Model volume	26.0 m ³
Location	Stockholm/Bromma	Model ground area	0.0 m ²
Climate file	Stockholm, Bromma-1977	Model external wall area	5.0 m ²
Case	building1	Window/Wall	30.0 %
Simulated	2009-08-28 11:42:25	Average U-value	1.018 W/(K·m ²)
		Envelope area per Volume	0.25 m ² /m ³

Month	Total Working Hours	Lost Working Hours
7	359.0	1.7
Total	359.0	1.7

IDA Indoor Climate and Energy
Version: 3.9027
License: ICE40:BETA07/URQ1

Too high or too low temperatures in a room result in production losses from workers. In ICE 3.0 some models for this according to Wyon has been included. For operative temperatures between 20 and 25°C no work is regarded as lost. Above and below these limits experiments indicate an average loss of 2 % in performance per degree.

Reference: Wyon, D. 2000. Individual control at each workplace: the means and the potential benefits In: Clements- Croome, D. (editor). Creating the productive workplace. E & FN Spon. London and New York.