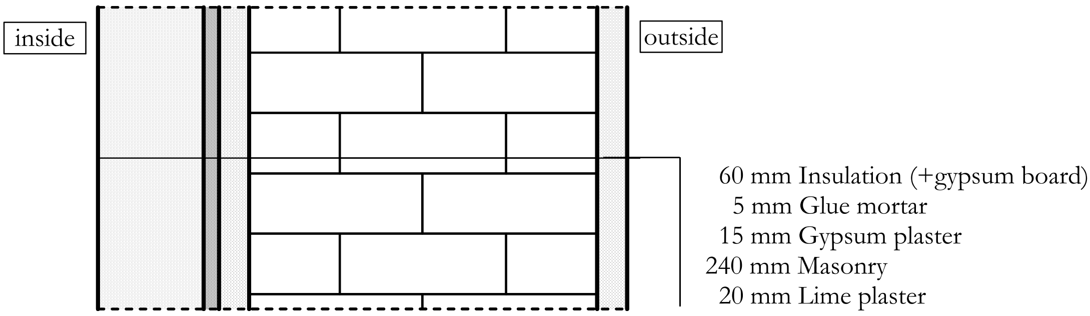
DELPHIN 6.1 – Tutorial 1

Outside Wall with Interior Insulation

In this tutorial a typical old masonry will be used analyzed that is modeled and simulated before and after mounting an inside insulation.



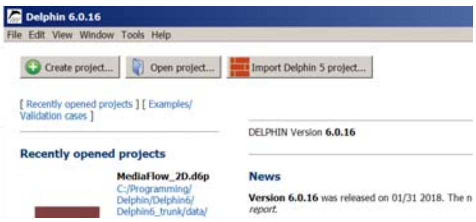
First, the previous construction (without inside insulation system) shall be evaluated with respect to its hygric and thermal performance. Afterwards a variation study is done to find a suitable inside insulation system.

1 *Part 1: Simulation of Previous Construction.*

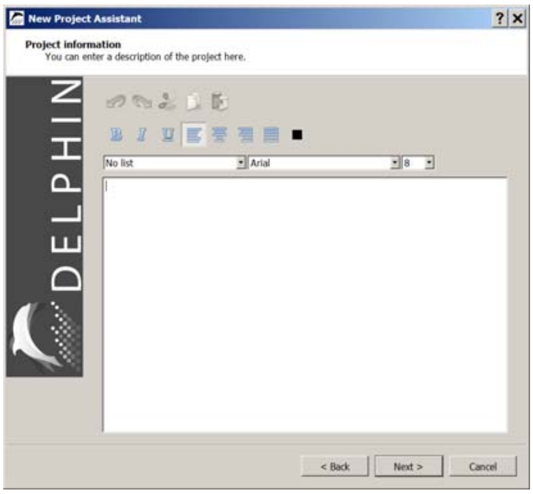
This part of the tutorial covers the principle steps in creating DELPHIN simulation projects.

1 Project Setup and Modeling of the Construction

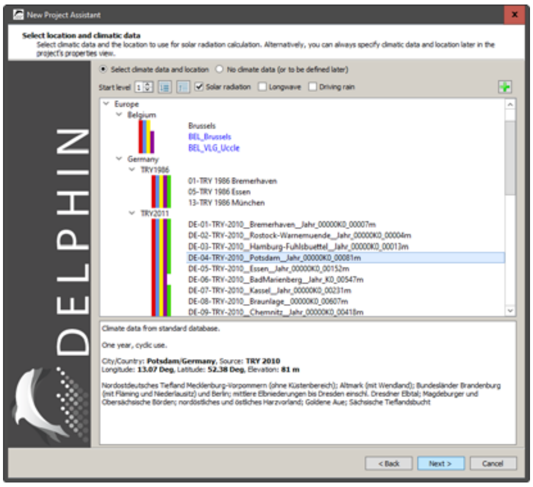
After starting DELPHIN one can see different buttons for the project setup and control in the left top area of the main window. The first step is generating a new project. The *Create project*-button (or File → New...) opens an assistant for creating a new project:



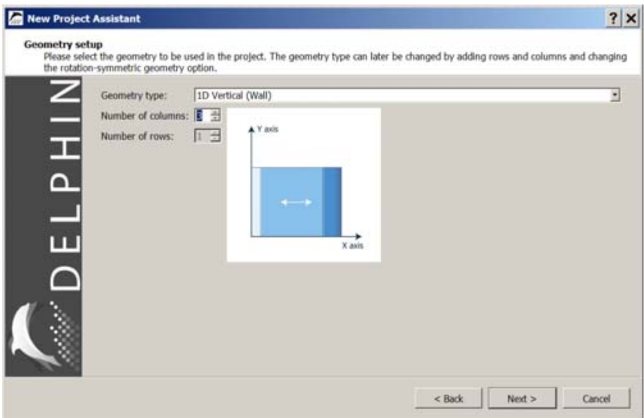
This assistant guides you through the first steps. After confirmation with click on *Next* a dialog opens where one can add some project information (see next picture). Here the current project can be described. This description will be shown in the project selection list next time you start DELPHIN.



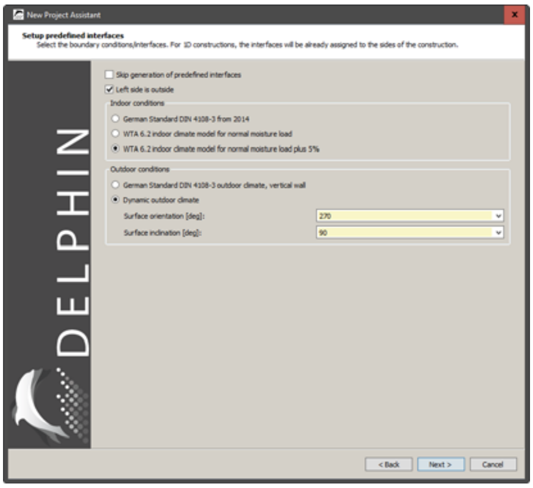
The next dialog allows you to select a location. One can select a location from a list with hourly weather data from the DELPHIN climate database. For this tutorial project the location TRY 2010 Potsdam, Germany should be selected.



After a click on *Next* the geometry setup will be shown.



Here one can select the basic geometry type and the main construction grid. We want to calculate a one-dimensional wall structure therefore 1D Vertical (Wall) should be selected. The type of construction can be changed at any point in time later. The existing historic construction has 3 layer (columns). The thicknesses of these layers can be edited later. In the next dialog the boundary conditions can be set.



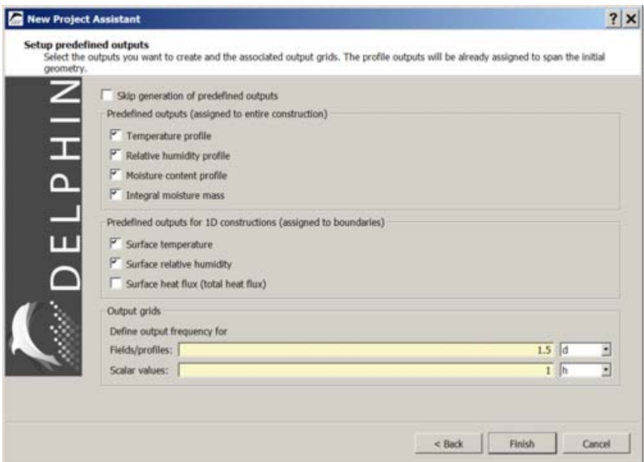
Checking the topmost selection skips this process. The boundary conditions must then be defined manually. In the second selection box you can define where the outside should be. If this box is checked, outside is on the left side, otherwise it is on the right. In this tutorial the left should be inside, so the hook must be removed. Afterwards the indoor climate and the outdoor climate are set. Here the following is selected:

- For the indoor climate, a climate calculated according to WTA leaflet 6.2 with a normal humidity load +5% is selected. This climate is also recommended according to DIN 4108-3 as a design climate for residential spaces.

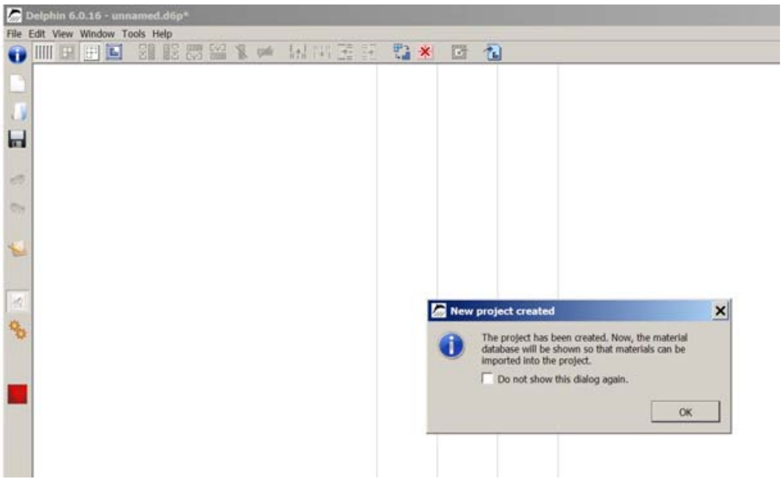
- For the outdoor climate, the data of the previously set climate location is used. The inclination is 90°, i.e. a vertical wall. This wall is oriented to the west (270°). In Germany this orientation is usually associated with the highest rain loads.

By clicking on Next, the selected surfaces are generated and assigned to the construction and a new dialog will be shown.

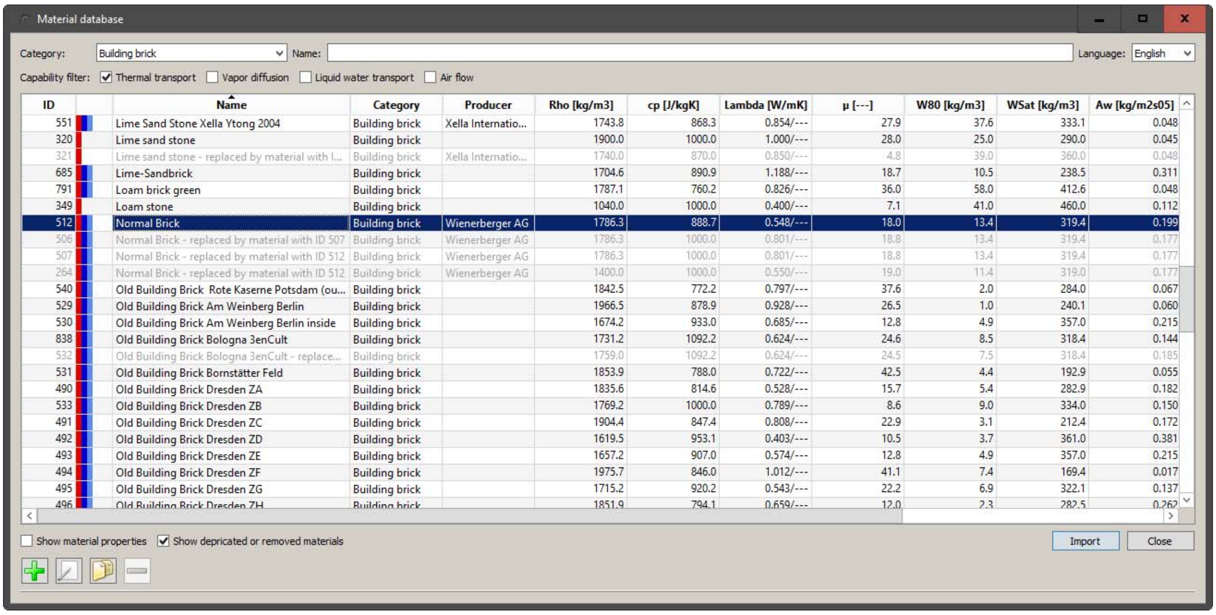
The next dialog allows to select a small amount of predefined outputs and the basic output schedule.



With the chosen settings above all field and profile outputs will be created each 1.5 days. For all other outputs an hourly schedule will be used.
This is the last dialog of the new project assistant. After clicking on *Finish* the main window will be shown and one can proceed with the selection of the materials.



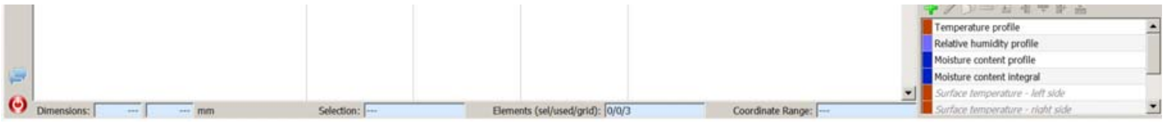
The main window shows the basic construction and a message box which let you start the material database selection. The displayed question can be skipped for further projects.



For the original construction we need three materials:

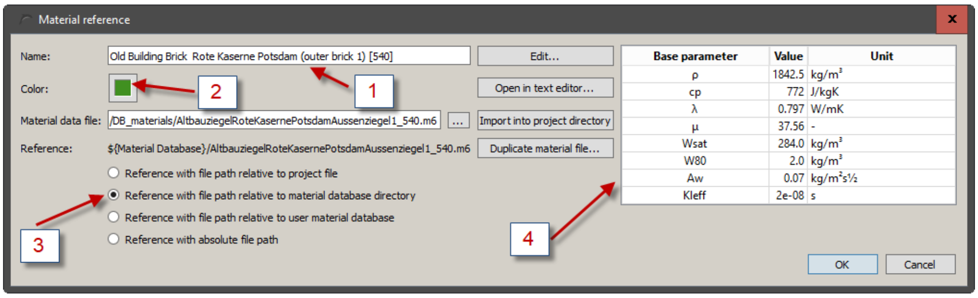
- Masonry: Normal Brick with ID 512
- Inside plaster: Lime plaster (historical) with ID 148
- Outside plaster: Lime cement plaster with ID 145

The material database view shows material names, producers (if known) and some basic parameters. Also important is the material ID (see first column). This is a unique ID that facilitates the identification of a material. Furthermore there is a filter for the material category and a name filter. As example for selecting the brick masonry one can use the category 'Building brick'. A click on *Import* adds the material to the project. Proceed in the same way with the two plasters. The picture below shows the main window after this process. One can see in the right top edge the three materials gray and italic. This means that the material are imported into the project but are not assigned to the construction yet. In the next steps the materials should be assigned and the thickness of the layers must be changed.



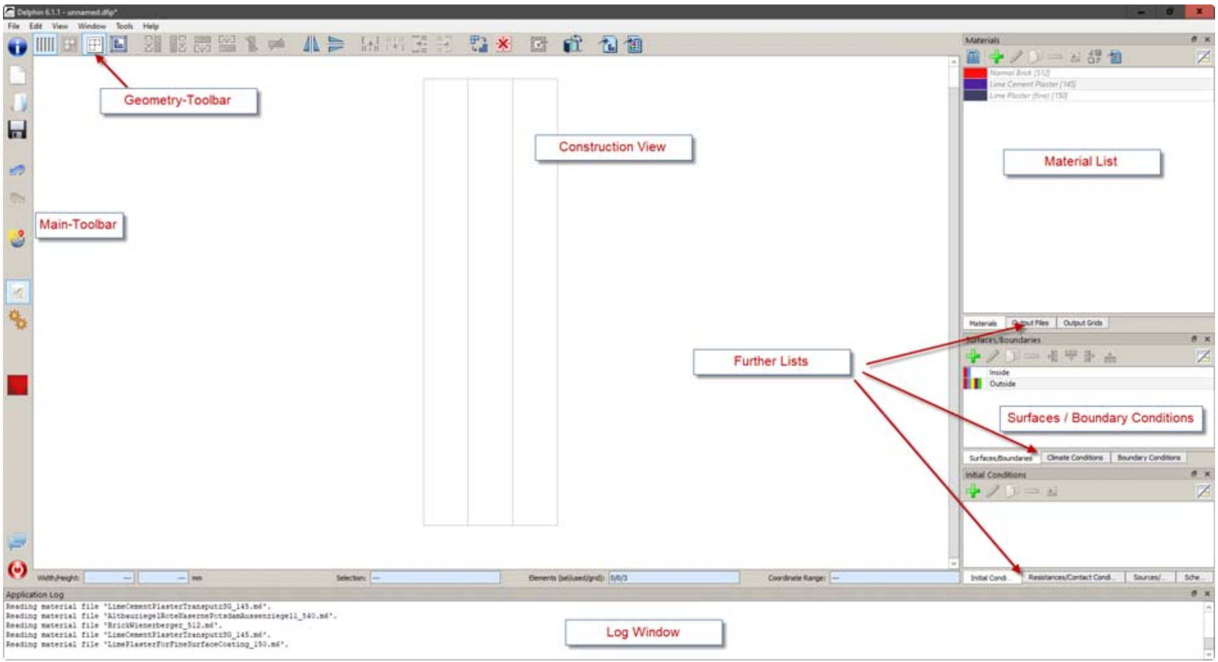
Once you have selected a material, either **double-click** the material, or click on the "**Import**" button (3). Then the import dialog opens allowing you to select some properties of the material reference.

Note, that <u>importing a material</u> actually means <u>creating a reference to a material file</u>, which contains all parameters of a material used in the project. The dialog gives some options on how to reference materials:



The name (1) is just an identification for this material and can be changed to any descriptive text. In addition, the color (2) can be adjusted so that materials are easily distinguished. Lastly, you can choose how the file path should be stored within the project (3). This is important when project files are copied around or moved across the network, because of which relative paths should be preferred normally. On the right side of dialogue (4) a selection of material parameters is shown. In this tutorial, you can simply keep the defaults in this dialog.

Once done with all three materials, you can close the material database view. You will now see the central modeling window:

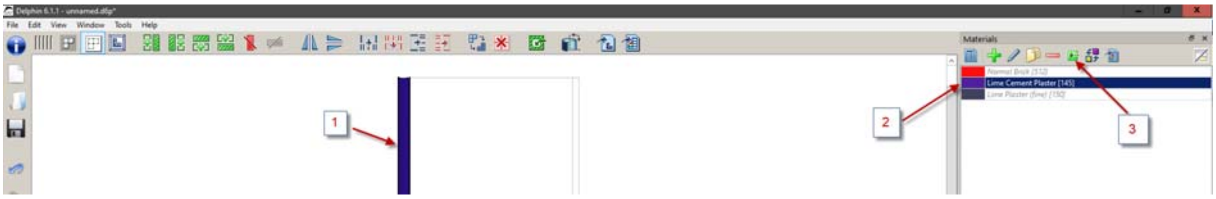


The layout of the windows can be adjusted at will. The various definition lists can be shown/hidden via the Window-menu on the upper left. Also, they can be dragged around and put together in tabs. For this and the following tutorials, we will use the standard layout.

2 Assigning materials and sizing layers

For changing the thickness of the left layer click on the left column and change the size in the edit field left of 'Dimensions' to 15 mm.

After changing the thicknesses of the other columns assign the materials to the construction.



To do this, you need to select a layer (or several layers) with the mouse (1). Then select the desired material in the material list (2) and finally press the green assignment button (3). The order of the actions can be reversed as well.

After assigning all materials the construction view should show the layers of the construction with the corresponding material colors. Furthermore the material names in the list will be shown bold. If you now select a material in the material list, the corresponding layer or layers will be highlighted. This *highlighting* is independent of the actual *selection* in the construction view.

In general all assignments can be checked in detail in assignment lists. The material assignment list is opened by pulling the lower margin of the material view to the top.

Next you have to create the interfaces (boundary conditions) for inner (left) and outer (right) side.

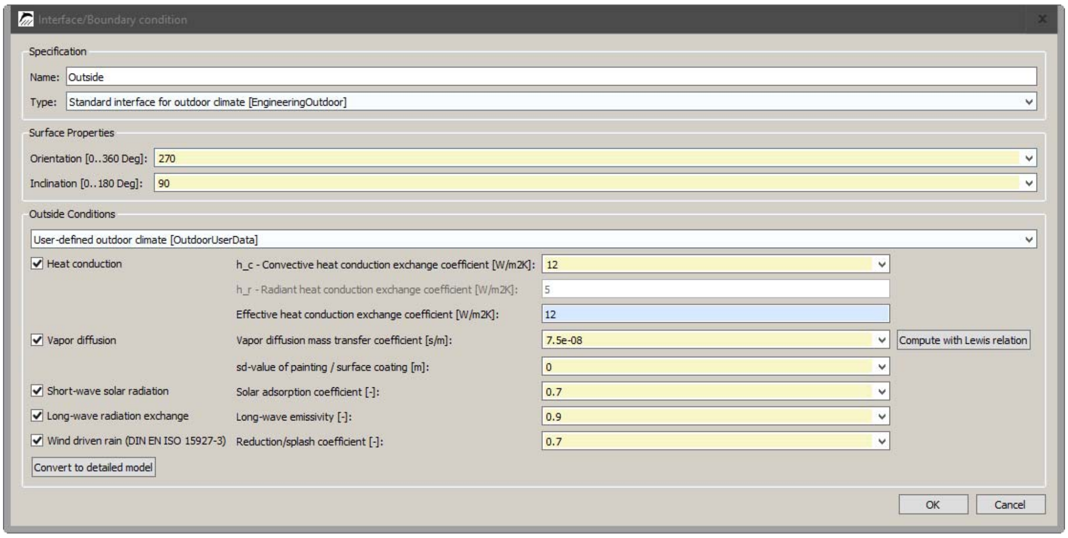
3 Interfaces – boundary conditions

The boundary conditions are already created and assigned by the new project assistant.

On the bottom right in DELPHIN (1), the list of existing surfaces is shown. A surface is a summary of different boundary conditions and the corresponding climate data. Here, there are surfaces for inside and outside. The coloured markings show which boundary conditions are used:

- Red - heat transfer
- Blue - Vapour diffusion
- Yellow - short wave solar radiation
- Brown - long wave radiation balance
- Green - driving rain
- Dark blue - water contact
- Light blue - Air flow

If a surface is marked in the list (click on it) then the assignment to the construction is shown as a thick dotted line in the view (2). A double click on a surface opens the settings dialogue.



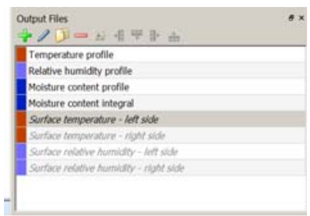
For more information regarding boundary conditions and climate see appendix A.

4 Outputs

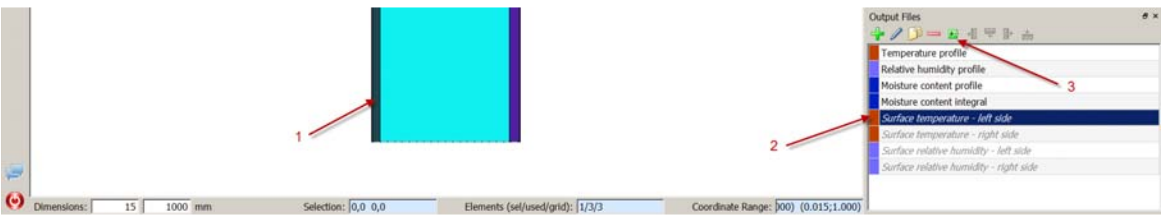
Now outputs can be set. For this tutorial the following outputs will be used:

- Temperature profile - temperatures in the whole construction over time
- Relative humidity profile - relative humidities in the whole construction over time
- Moisture content profile - volumetric moisture content in the whole construction over time
- Moisture content integral - moisture mass in the total construction
- Surface temperature left side - temperature on the inner surface
- Surface relative humidity left side - relative humidity on the inner surface

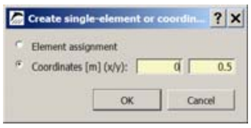
All outputs are defined from the 'create project' assistant but only the first four are already assigned to the (whole) construction. This is shown by the 'Output files' dialog. All assigned outputs are written with bold characters not assigned constructions italic.



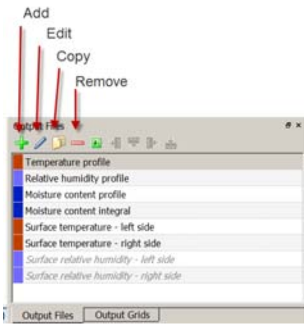
Now the two surface related outputs shall be assigned. Click on the most left column with the surface (1), click on the output file in the list (2) and then on the assignment button (3), or in reversed order.



After clicking on the assignment button a dialog will occur. Here one can decide if the output should be related to a volume element or to a position. For a surface output a position related assignment should be used. The coordinate system for the geometry runs from left to right. That means the left most side has the x coordinate 0. Please fill out the dialog as the picture below shows.

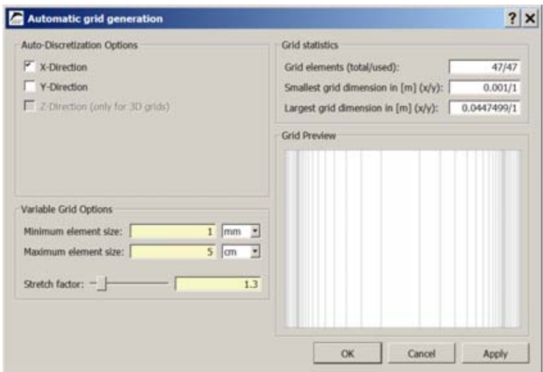


After clicking on *Ok* the assignment will be performed and a small red dot will mark the position in the construction view. As next step assign the surface relative humidity in the same way. The 'Output files' dialog will show all assigned outputs bold. One can add, remove or change the outputs by using the buttons on the top of the dialog. The other tab shows the 'Output grids' that means the time schedules. If this (or another) tab is not shown check it on the upper left in the >> Main menu >> Window item



5 Discretization

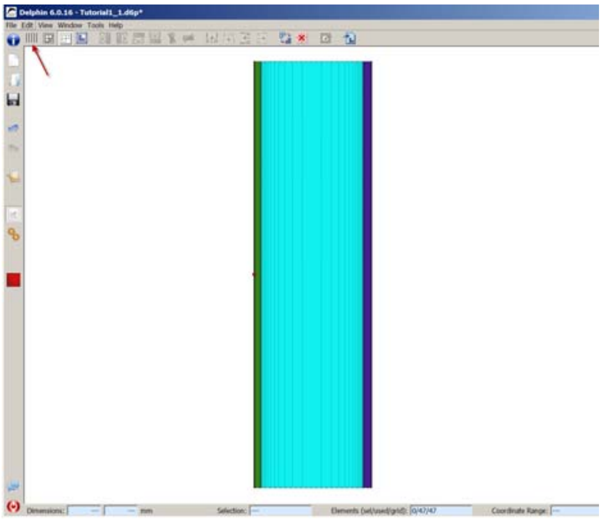
Next you can discretize the construction, i. e. divide in many small elements. For this you can use the dialog for Automatic Discretization, accessible from the menu buttons in the construction view (see left screenshot below).

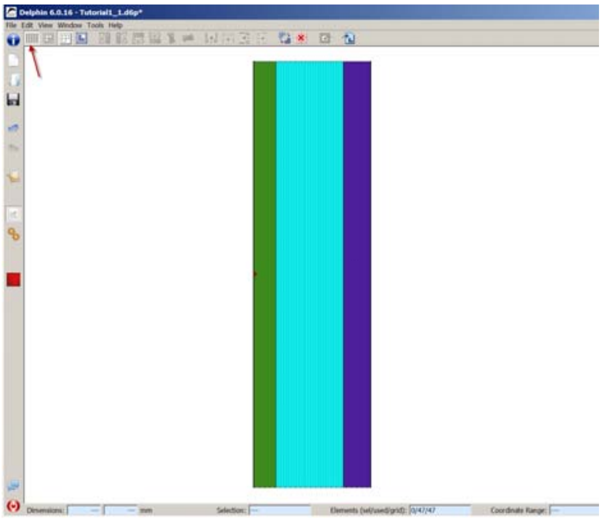


In the Automatic Discretization dialog we will only discretize the construction in X-direction - as we have to deal with a wall with horizontal transport direction only. We use variable discretization that generates smaller elements near the boundary of the construction and at material interfaces. Inside of the construction the element widths are gradually enlarged.

Important parameters in this dialog are: Minimal and maximal element widths (1 mm and 50 mm are good default values), and the stretch factor. The discretization detail can be adjusted with the slider which adjusts the stretch factor and consequently the number of elements. A stretch factor of 1.2 to 1.4 is suitable for 1D-constructions. For 2D-constructions a higher stretch factor of approx. 1.5 is advisable to avoid long simulation durations.

As soon as this dialog was confirmed, the construction is shown as collection of many elements. Because of the small size of the boundary elements, it is useful to switch to equidistant display of the construction (button to the very left of the button bar in the construction view). In this view mode all elements are shown with the same dimensions, regardless of their actual size. This simplifies the selection of thin or boundary elements.

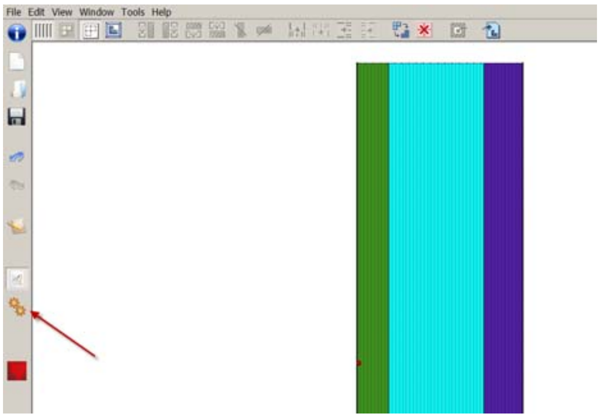




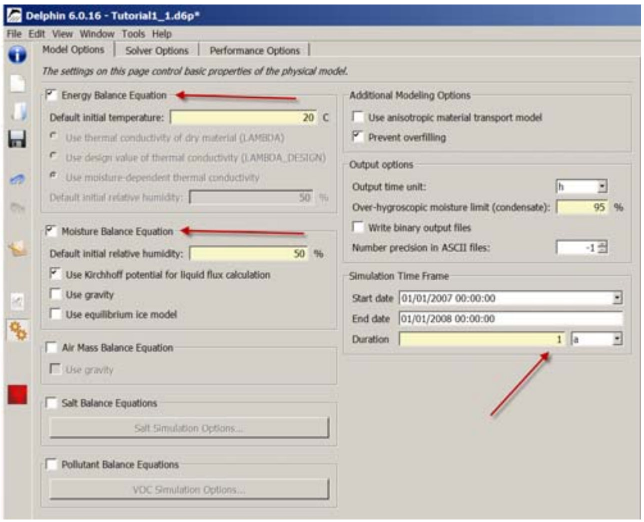
The left screenshot shows the normal proportional view whereas the right screenshot shows the equidistant view.

6 Initial and Simulation Conditions

After specifying the boundary conditions it is time to select the simulation and modeling options. Open the modeling dialog by click on the selected button in the tool bar (or >> Main menu >> View, or F7):

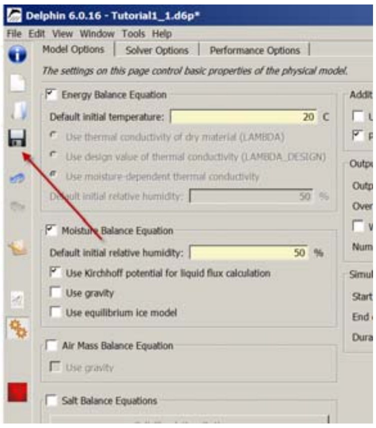


This dialog contains three different tabs to change options. The first tab allows to select the basic properties of the physical model. For this example you need to enable the balance equations for heat and moisture transport only:



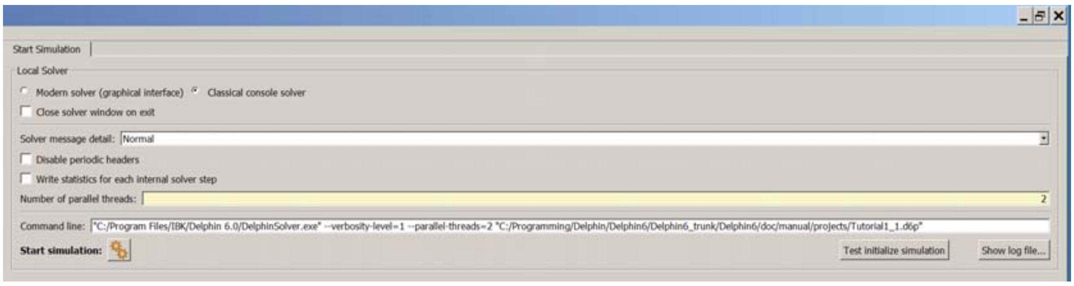
Other important settings in this dialog are the duration and the starting time of simulations. Here, we use only one year as duration of the simulation. The starting time is not important for a design simulation. However, the initial conditions need to be specified. They can be set globally in the "Defaults" section of the balance equation

Now all project settings are complete and it is advisable to save the project (Ctrl+S).



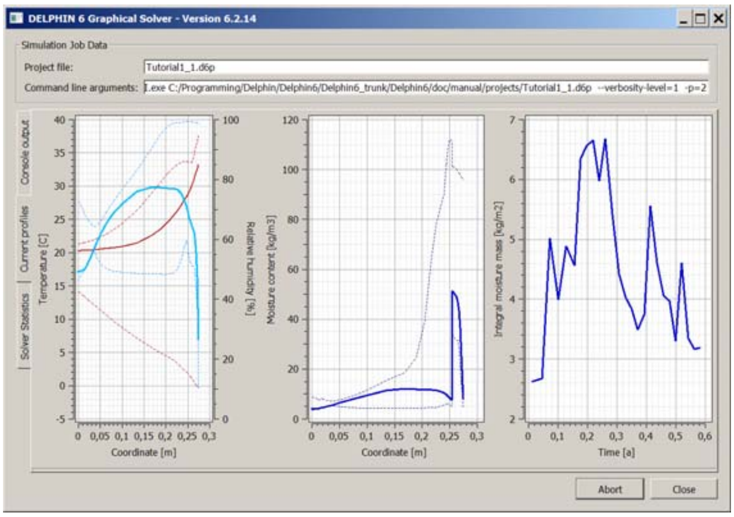
7 Run Simulation

In the same dialog as before one can also start the simulation.

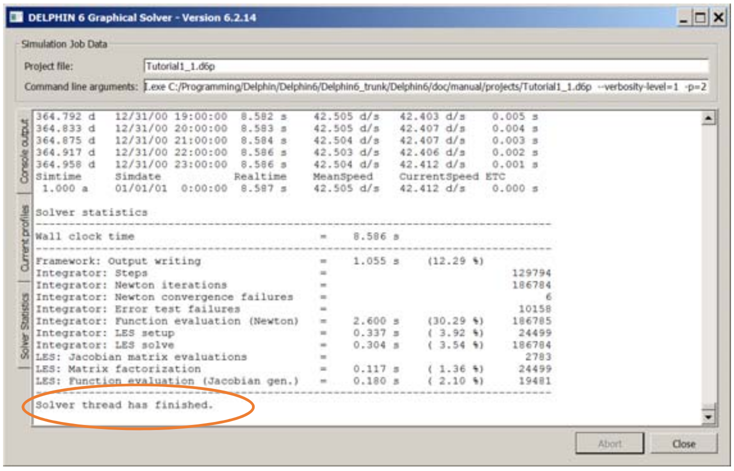


In this dialog you can choose between a solver with graphical output during the calculation process and an external solver. The latter is faster and particularly for 2D simulations the recommended choice. In this example, and generally for 1D simulations, the internal solver is advisable. It is also possible to set the *Number of parallel threads*. Especially for 2D simulations this can speed up the simulation many times over. The simulation is started with "Start simulation" and the simulation window opens:

The simulation window shows the current temperature, relative humidity and moisture profiles as well as the total moisture content over time which can be used to quickly check the results. Also, the current solver performance statistics can be shown (left vertical tab 'Solver statistics').


DELPHIN 6 Graphical Solver - Version 6.2.14
Simulation Job Data
Project file:
Tutorial1_1.d6p
Command line arguments:
Coordinate [m]
Time [a]
Temperature [C]
Relative humidity [%]
Moisture content [kg/m3]
Integral moisture mass [kg/m2]
Console output
Current profiles
Solver Statistics
Abort
Close

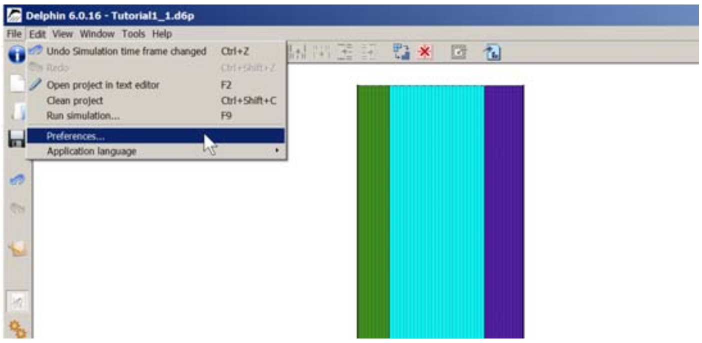
Once the simulation is complete



the calculation results can be visualized and analyzed with the post-processing. The command button "Post-Proc" starts the post-processing tool:

There are currently two different post processors: the original one from DELPHIN 5 and the newly developed version (PostProc2). The new postprocessing contains the most important features of the old version, so it can be used in most cases. It also has new features that facilitate the evaluation and especially the comparison of data.

PostProc 2 is used for this tutorial. PostProc 2 has to be installed before because it is not included in the DELPHIN 6 setup. You can find the installer for it on the download page (http://www.bauklimatik-dresden.de/downloads.php) under PostProc. The default settings of the post processor can be changed in >> Edit >> Settings >> External programs.



Choose the tab 'External tools' and select which kind of post-processing shall be used.

Click on the corresponding button of the post-processor version to select it. After opening the PostProc 2, the data of the current project should normally already be loaded. If not, these data must be loaded from the project directory.

For further information regarding the postprocessor: see tutorial 3 (for PostProc 2, PDF, english) on the website http://bauklimatik-dresden.de/delphin/documentation.php. There is also an online help which can be accessed via the following link: http://www.bauklimatik-dresden.de/postproc/help/en/index.html

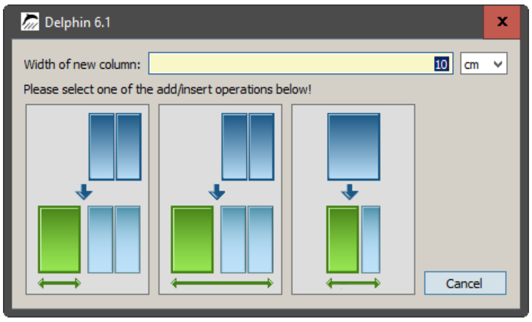
2 *Part 2: Adding a Capillary Active Inside Insulation*

Since the thermal resistance of the existing wall does not meet modern requirements of buildings, an additional insulation is added. In this tutorial we assume that adding an insulation on the outside is not possible (perhaps due to a historical facade, etc.), therefore a capillary active insulation system with calcium silicate will be used. The CaSi insulation is fixed with a glue mortar.

In order not to delete the project with the historic, uninsulated wall, save the project with different file name first. That way the previous results remain for comparison.

At first, the materials „Calsitherm KP Glue Mortar“ (ID 705) and „Calsitherm Climate Board F“ (ID 706) need to be imported into the project from the material database.

Then you can add two new layers in the construction view to the left of the innermost layer (see command buttons in menu bar of construction view, left ellipse) with the according thickness (5mm glue mortar and 80mm insulation). To do this, select the innermost element and then click on the button for Add layer. Then the following dialogue opens:



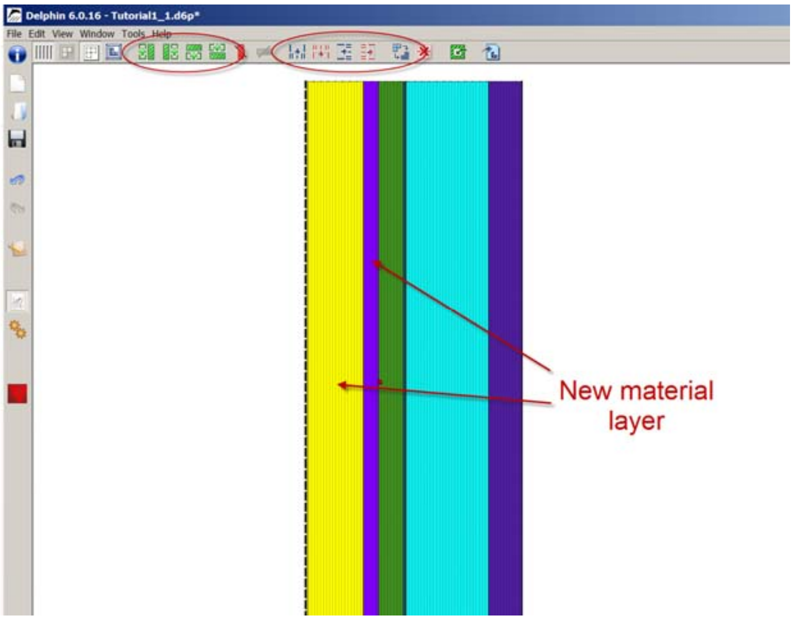
In the input line above you can specify the thickness of the new layer. Below it there are three graphic buttons that show how to use this thickness:

- Left - thickness corresponds to the whole new layer
- Centre - thickness corresponds to the construction width plus the new layer
- Right - thickness corresponds to the marked column plus the new layer

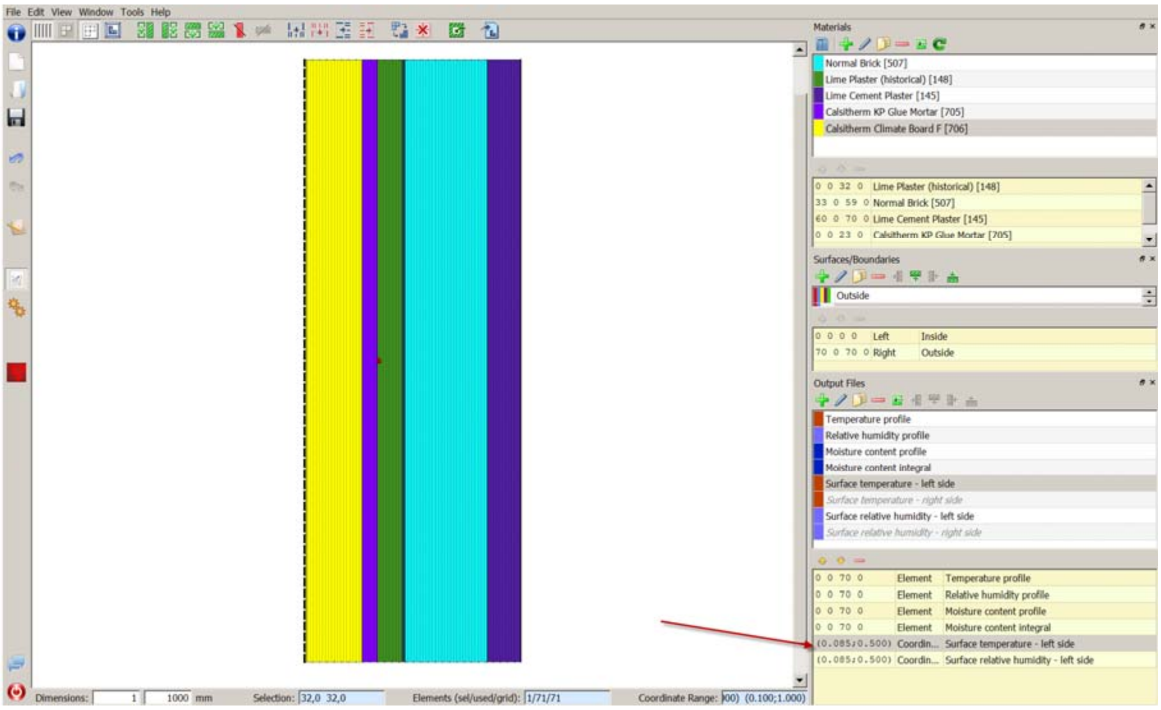
If you just want to add a new layer of the appropriate thickness, click on the left button.

Then you can assign the materials to the respective layers (see construction sketch at the very beginning of this document).

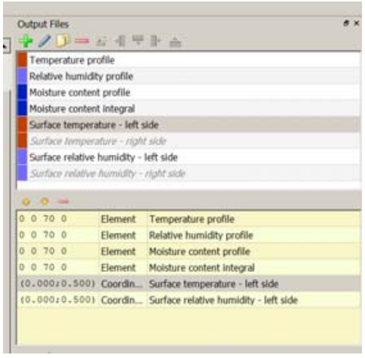
Finally, you can use the Automatic Discretization dialog again to create a grid for the new material layers, or manually discretize each layer with the Discretization dialog (see respective buttons in menu bar, right ellipse).



Whenever the construction has changed it is advisable to check that all boundary conditions and outputs are still assigned to the correct layers. Interfaces of 1D constructions are automatically set to the right boundary. In this case the original surface outputs keeps in the old inside plaster (see small red dot in picture above). In order to set the outputs again at the inner surface one have to change the assignment. Assignment lists normally are hidden. In order to show all lists click in the >> Main menu >> Window: Toggle visibility of all assignment windows.



The red arrow marks the position of the assignment parameter for the (former) surface outputs. As you can see the x-coordinate now is set 0.085 m. This has to be changed to 0.0 m: double-clicking on the assignment allows you to edit values.



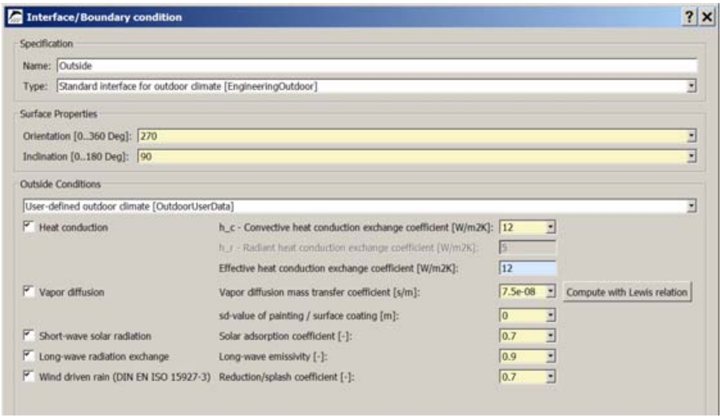
Now the simulation can be repeated.

… End of 1st Tutorial …

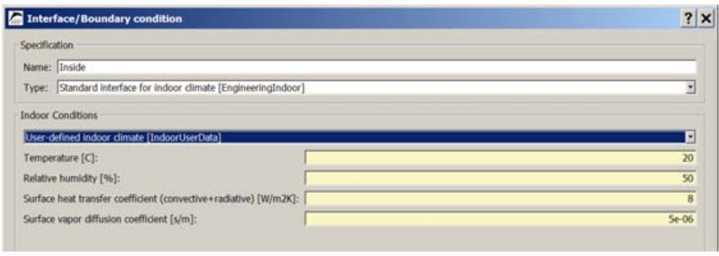
3 *Appendix A – climate and boundary conditions*

1 Cretae Interfaces – boundary conditions

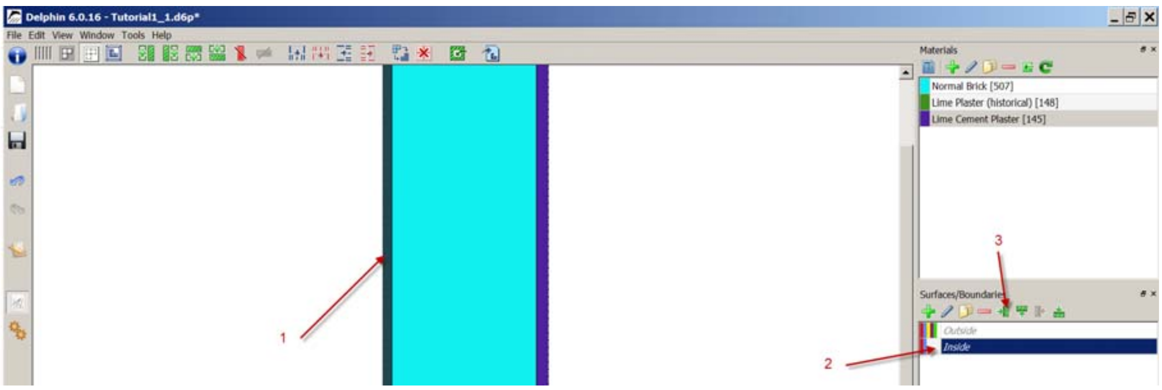
First one has to create the interfaces for inside and outside. Click on the green plus button in the 'Surface/Boundaries' dialog. For this tutorial standard interfaces will be used.



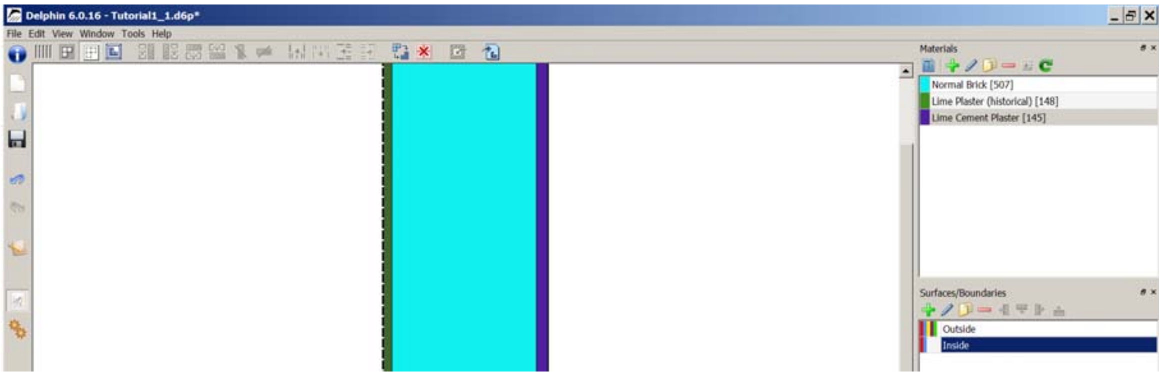
The right picture shows the dialog with the settings for an outside interface. The orientation is set to 270 Deg (West) and the inclination to 90 Deg (vertical wall). The selected type 'Standard interface for outdoor climate' will use the climate data from the selected location (see create project assistant). The outside conditions should be set to 'User-defined'. Here one can set which boundary condition is used and other basic settings. Next an interface for the inside boundary will be created. Use the settings shown in the next picture.



After creating the interfaces they need to be assigned to the boundaries. This works similar to the material assignment.



The inside climate will be located to the left. Select a column (1), select the interface to assign (2) and last click on the button for assignment to a left side (3). Proceed with the right side for the outside interface. One can check if the interface is assigned correctly by clicking on an interface in the 'Surface/Boundaries' list. The interface will be marked in the construction view with a dashed line (see picture below).

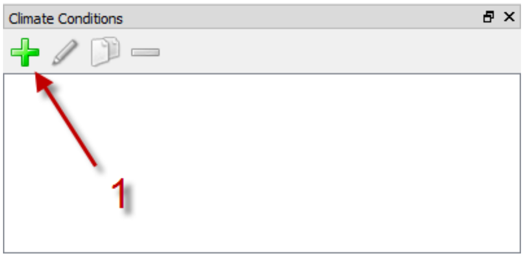


2 Creating Climate Conditions

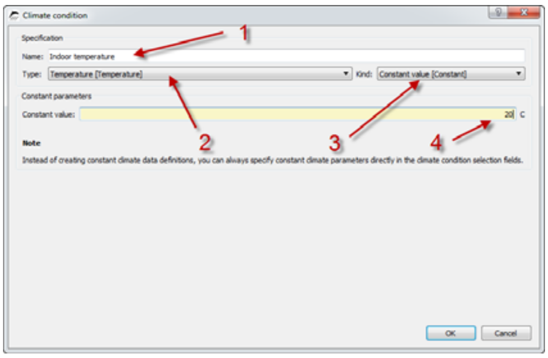
As mentioned before, the indoor/outdoor conditions or sides of the construction are not fixed. Instead, they are specified by a selection climate data and boundary conditions. This chapter shows how to create definitions manually. This allows more options.

For hygrothermal analysis, we want to use winter climate. That means indoors a temperature of **20 °C**, and relative humidity of **50 %**, and outside **-5 °C** and **80 %** R.H. throughout a period of **90 days**.

The climate is represented by a total of four climate conditions, which we will create first. To create a climate condition, go to the **Climate Conditions** list window. As with all definition windows, pressing the plus button (1) creates a new definition.



Doing this for Climate Conditions open the dialog for climate data. There you can specify the climate data, basically a single climatic data component: identification name (1), **Temperature** as type of quantity (2), variability in (3) where we use **Constant** and finally the value in (4).

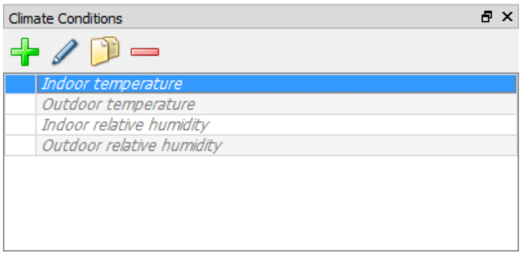


The following table lists names, types and values for the four conditions to be created:

Descriptive name	Condition type	Value (without unit)
Indoor temperature	Temperature	20
Outdoor temperature	Temperature	-5
Indoor relative humidity	Relative Humidity	50

Outdoor relative humidity	Relative Humidity	80

Once all of these are created, the list of conditions should look as follows:

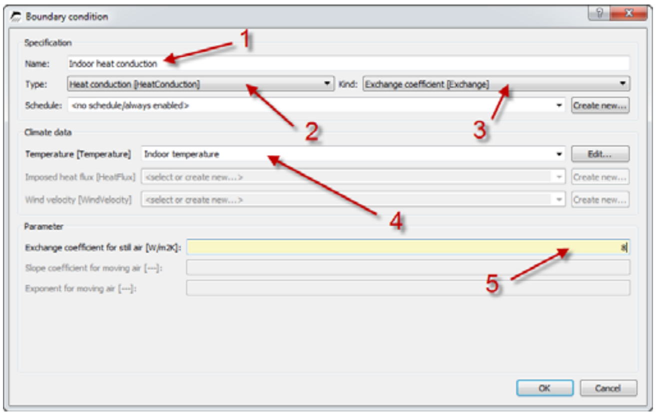


Note, all definitions are shown in gray italic, which means they are not being used in the model. In the next step, we'll change that.

3 Creating Boundary Conditions

The climate data is needed and referenced in boundary condition definitions. Boundary conditions effectively describe how the construction surface interacts with the climate, resulting in heat and moisture fluxes across the construction surface. Different physical effects such as heat conduction, vapor diffusion, air infiltration, water contact etc. are all described in different boundary conditions.

Similarly as for climate conditions, select the Boundary Conditions window and create in total four boundary conditions. Start with the first boundary condition by clicking on the plus-button in the **Boundary Conditions** list window.



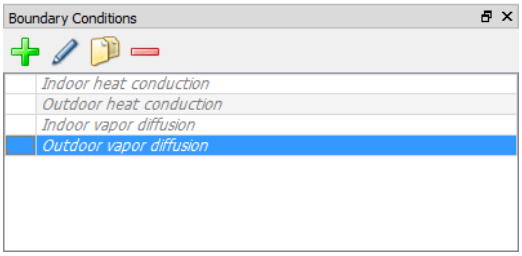
Within the dialog enter a descriptive name (1), select a boundary condition type (2), select a calculation model (3), specify requires climate conditions (4) and finally set the model parameters. For heat conduction, the relevant parameter is the heat transfer coefficient. In this tutorial we do not model long wave radiation exchange separately, so the heat transfer coefficient is an effective heat transfer coefficient including convection and mean long wave radiation exchange (8 = 3 + 5 W/m^2K). Also, the heat conduction boundary condition needs the temperature adjacent to the wall surface, which is in our case the “Indoor temperature”, created in the last step.

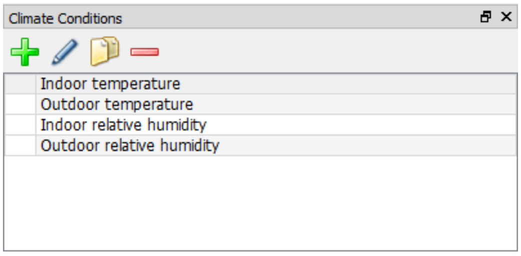
The following table lists the names, types, parameters and climate condition dependencies of all four boundary conditions to be created:

Descriptive name	**Type**	**Kind (model)**	**Parameter**	**Dependent Climate Conditions**
Indoor heat condition	Heat Conduction	Exchange Coefficient	8 W/m^2K	Indoor temperature
Outdoor heat condition	Heat Conduction	Exchange Coefficient	17 W/m^2K	Outdoor temperature
Indoor vapor diffusion	Vapor Diffusion	Exchange Coefficient	25e-9 s/m	Indoor temperature Indoor relative humidity

Outdoor vapor diffusion	Vapor Diffusion	Exchange Coefficient	75e-9 s/m	Outdoor temperature Outdoor relative humidity

Note how the descriptive name of each climate condition is used to reference it within other definitions. The Boundary Condition list window should now show all four boundary conditions as unused definitions. The climate conditions, however, are now shown as used (black).

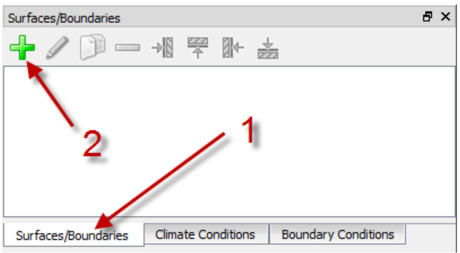




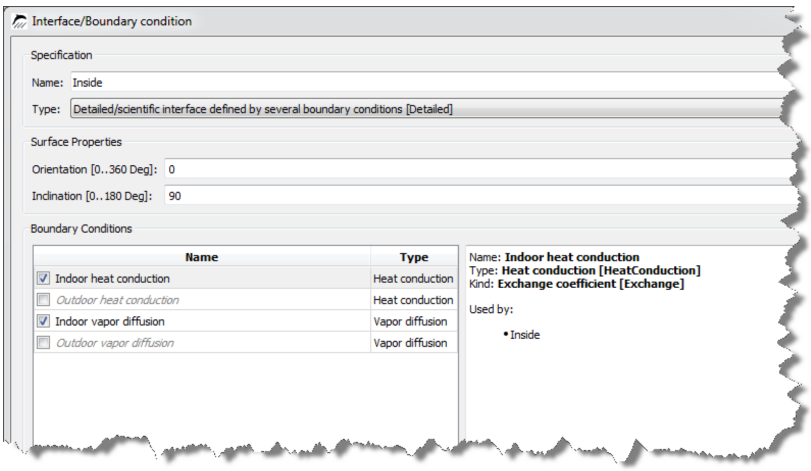
4 Creating and Assigning Interface Definitions

Boundary conditions are grouped together in Surface/Boundary definitions. These represent a surface to some space, for example a room or the outside.

In the next step, we create two surfaces/boundaries, one for the inside and one for the outside. Go to the Surfaces/Boundaries definition list window (1) and press the + button (2):



As with all definitions in DELPHIN, give the new interface a unique description name, for example "Inside". Then, select the type of surface/boundary. In this tutorial, we have created boundary conditions manually, so we use **Detailed/scientific interface**. Afterwards check the boundary conditions that shall be associated with this interface/surface. For the inside surface, check "Indoor heat conduction" and "Indoor vapor diffusion".



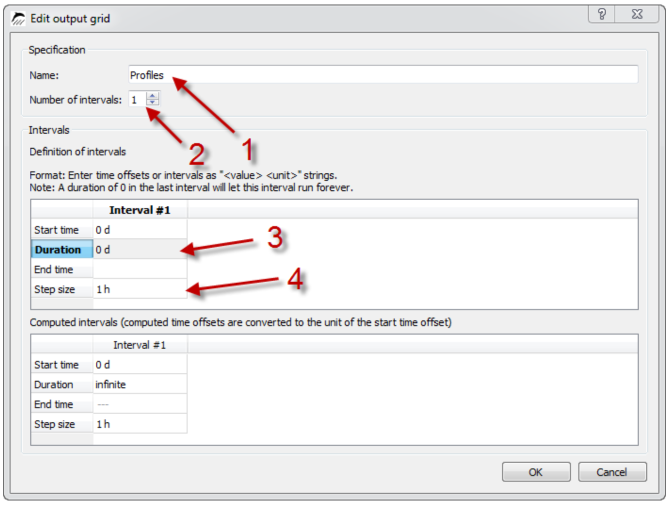
Similarly, create an interface "Outside" which uses the outdoor boundary conditions.

4 *Appendix B – Outputs*

1 Output Grids and Frequency

Before defining the output quantities we would like to monitor, we need to define the frequency of outputs. This is done in output grid definitions. It is meaningful to limit outputs for profiles, since such output files tend to become large. A good procedure is to create two output grids, one for profiles, i.e. outputs that write values for all elements, and one for scalars, for example sensor values (temperatures, relative humidities at certain positions) or integral values (i.e. total moisture content).

In the **Output Grids** list window, create a new definition (plus-button) and give it an identification name (1). The output frequency need not be constant throughout the simulation, but can be different in separate intervals of the simulation. So next, you need to select the number of such intervals (2). For this tutorial, leave the interval number to 1.



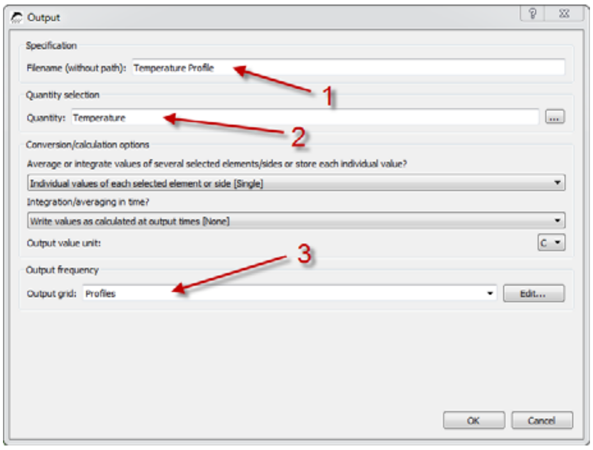
For each interval, you need to select the duration of the interval (3) and the output step size (4). Alternatively, you can specify start or end times of intervals, whereas time means relative time since start of the simulation.

The last interval can be given a duration of **0 d** (don't forget the time unit!). This means that the interval is running as long as the simulation runs.

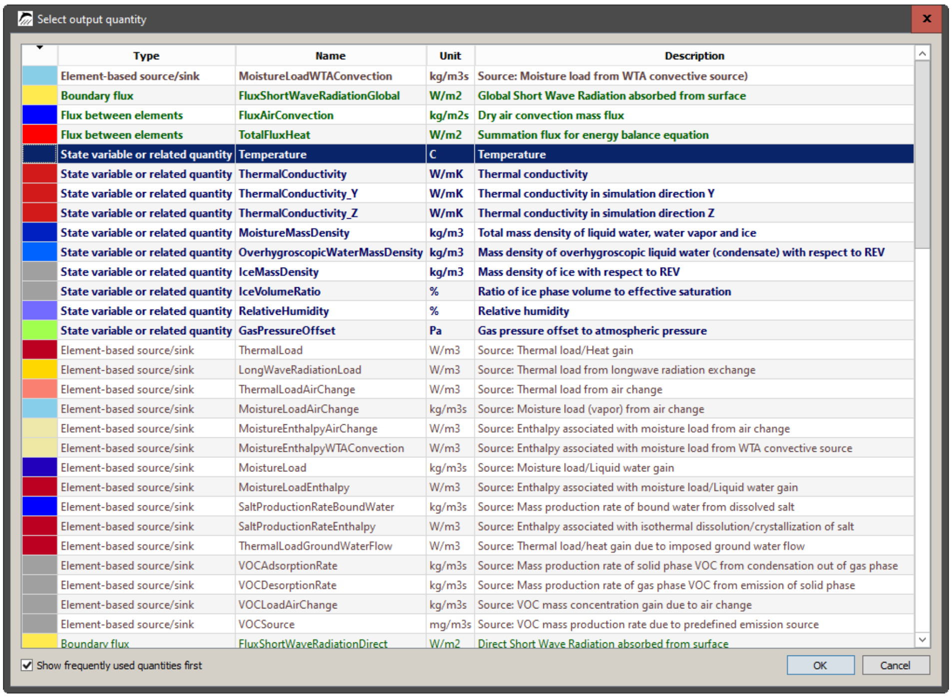
Create now output grids named "Profiles" and "Scalars" with one interval each, and output time steps of **1 h** for "Profiles" and **10 min** for "Scalars".

2 Output Files

Now outputs can be defined. This is done via defining output files. In the **Output Files** window, create a new output file definition (use the plus-button again). Start as usual with an identification name (1), but mind, that for output files the name is also used for the file name. That means you must not use characters that are not allowed for filenames (no slashes, colons etc.). In this tutorial, use "Temperature Profile" for the first output file definition.



Next you have to select the physical quantity for the output (2). In the quantity selection list that is opened with the associated button, you see all the quantities that can be calculated with DELPHIN.



In this list select the quantity "Temperature" and close the dialog.

To navigate quickly the table and quickly select the quantity you need, simply click on a table cell in the keyword or description column and start typing the name/keyword. The cursor will then jump to the cell. Also, you can sort quantities based on type and name. Furthermore, all frequently used quantities are displayed in bold at the top (can be switched off with a hook at the bottom left).

The selected quantity also defines the default output unit and whether it is an element or side-based quantity. Finally, we select the output grid (3). This defines how frequent outputs are written.

Now the **Output Files** list window should hold a single output "Temperature Profile". In the grid generation step the total geometry was discretized into many elements. Now we apply the created temperature output to all elements, which means we are obtaining the temperatures in all elements at each output time. This allows us to create temperature profiles across the geometry for each output time (see post-processing steps in part 2 of the tutorial).

To assign an output select all elements (either per mouse, or keyboard holding the Shift key, or by pressing Ctrl+A). Then select the output file definition and use the assignment button . Again, you should see the output file definition to change from gray italic to normal black, indicating its use in the project.

It is also possible to merge the results from several elements into a single number. For example, you can compute a weighted average value ("Mean" value), or an integral value ("Integral"), for example the integral moisture content. These spatial operations are termed **space type** of the output and can be selected in the output file definition dialog. When the values in each element shall be monitored (the default), the space type is "Single".

Simulation outputs are time dependent quantities. There are some time-related calculations (termed **time type**) built into DELPHIN. For example, you can compute an average value in time (keyword "Mean"). The averaging always starts at the begin of the simulation. More advanced averaging like gliding average calculation is done in the post-processing. Fluxes or sources can be integrated in time ("Integral"). If the value shall be written to file as it is calculated at the output time, the time type is "None". You can set this parameter also in the output file definition dialog.

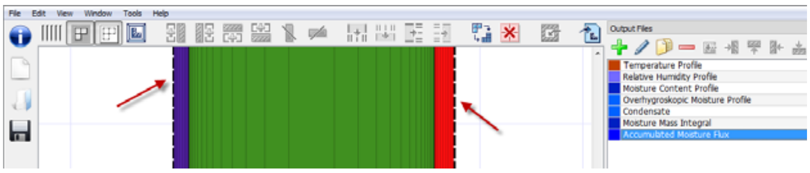
For analysis of the simulation it is meaningful to create the following outputs:

Output name	**Quantity**	**Schedule**	**Space type**	**Time type**
Temperature Profile	Temperature	Profiles	Single	None
Relative Humidity Profile	RelativeHumidity	Profiles	Single	None
Moisture Content Profile	MoistureMassDensity	Profiles	Single	None
Overhygroscopic Moisture Profile	OverhygroscopicWaterMassDensity	Profiles	Single	None
Moisture Mass Integral	MoistureMassDensity	Scalars	Integral	None
Condensate	OverhygroscopicWaterMassDensity	Scalars	Integral	None
Accumulated Moisture Flux	TotalFluxMoisture	Scalars	Integral	Integral

If you have to create many definitions with similar properties, use the "Copy definition" function via the button .

All output files except the last have to be assigned to the entire geometry. The flux output is used to obtain the accumulated moisture mass in the construction, which changes dynamically due to moisture fluxes into and out of the construction. If done correctly, the results obtained with this output should match that of the moisture mass integral (actually the difference of the moisture mass integral to the initial moisture mass integral).

Boundary flux outputs are generally assigned on the surfaces/boundaries of the geometry. In this tutorial, we will assign this output to the leftmost and rightmost element. First select (only) the leftmost element, then select the "Accumulated Moisture Flux" output and click on the assign-from-left button . Then do the same for the rightmost element and assign the flux from right. When you now select the flux output, the construction should highlight both boundary sides.



Due to the clustering of the grid near the boundary, it may be difficult to select the boundary elements (or you have to zoom in quite a bit). Alternatively, you can display the construction differently with all elements being shown with the same size. You can switch between these display modes with the button .

Now with the output definitions in place, we are ready to start the simulation.