Outside Wall with Interior Insulation

In this tutorial a small example of a typical old masonry wall will be used that is modeled and simulated before and after fitting of an inside insulation.

Wall structure (from inside to outside):
- 15 mm Gypsum plaster
- 240 mm Brickwork
- 20 mm Lime cement plaster

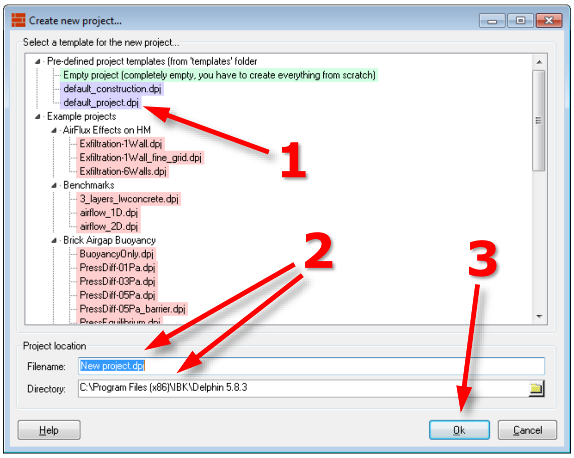
First, the previous construction (without inside insulation) shall be evaluated with respect to hygric and thermal performance. Afterwards a variation study is done to find a suitable inside insulation system.

Part 1: Simulation of Previous Construction.

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

Project Setup and Modeling of the Construction

At first, after starting the Delphin program, only the small main menu is visible at the top of the screen, where the different buttons for the project setup and control can be found. The first step is the creation of the new project. The New-button (or File → New...) opens the dialog for creating a new project:



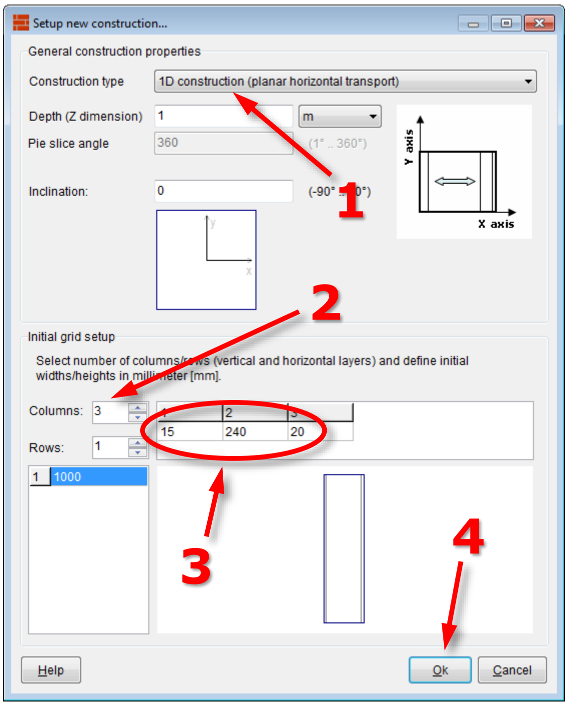
Important in this dialog is the selection of the project template. This can be either an empty default project, or one of the example projects. Select here **default_project.dpj**, enter file name and path and confirm the dialog.

Please make sure to select the project template **default_project.dpj**, because some of the steps below depend on this.

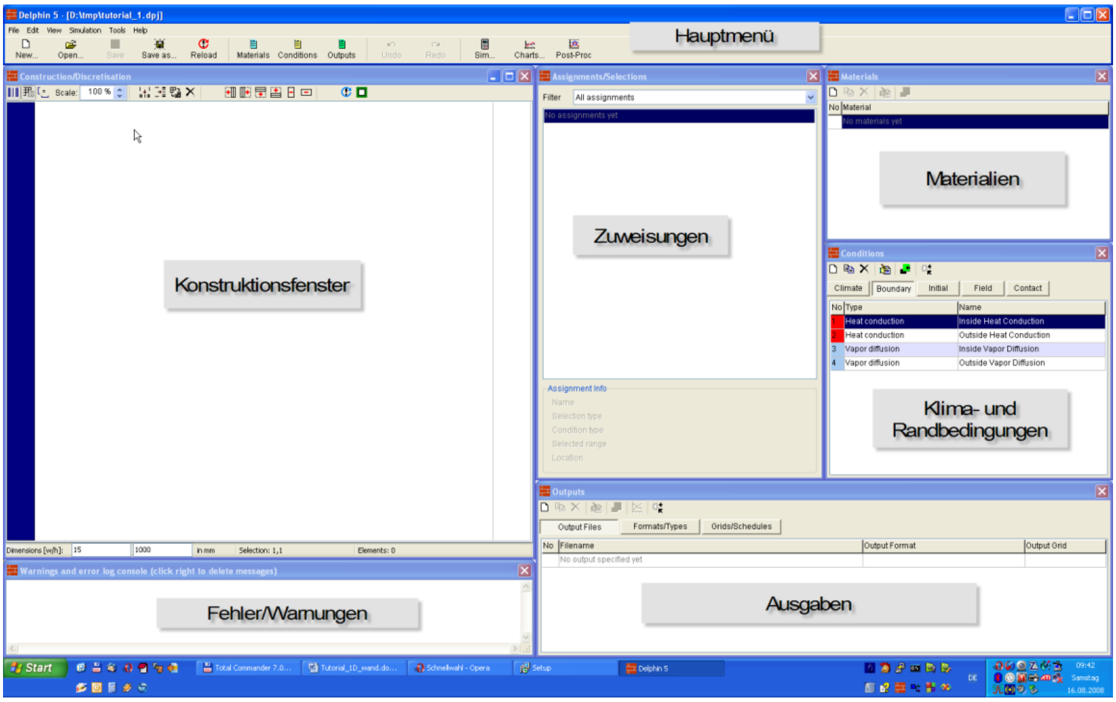
If later similar projects need to be created, you can copy your own project into the **templates** directory within the Delphin installation directory.

After confirmation of the dialog the construction dialog opens (see screenshot on next page). Here, the principle construction type is selected and the initial dimensions are entered.

Please select **1D construction (planar horizontal transport)**, and use 3 as number of initial columns of the construction. Then you can enter the column widths 15, 240 and 20 (in millimeter). For one-dimensional constructions it is important to ensure that the height (or width, in case of vertical constructions like roofs) is set to 1 m. This simplifies the analysis and interpretation of the results later.

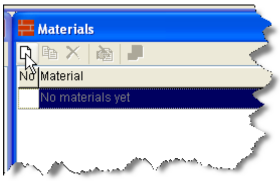


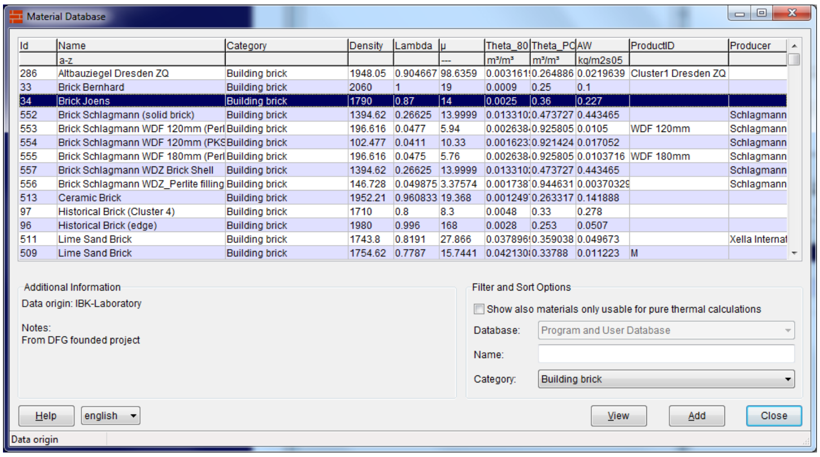
After confirming the dialog the program views appear showing a 3-layered construction, yet without materials.



The layout of the windows can be adjusted at will. The View-menu contains the commands for the layout of the views. The layout shown above is suitable for higher screen resolutions.

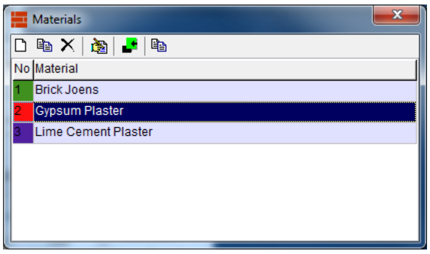
The shown construction is still empty and in the next step the materials need to be imported. First click on the New-button in the material list view (see left screenshot below) or choose after right-click in the Materials window >> Insert from data base. Opening the material data base takes a small while.





You can now select materials in the material import dialog (screenshot to the right). To import specific materials, switch the categorization (screenshot to the right top) to "Alphabetically Sorted List".

Now select one after another the materials **Gypsum Plaster** (ID 71), **Brick Joens** (ID 34) and **Lime Cement Plaster** (ID 145). You can select multiple materials with Ctrl+click and Shift+click. After the import the materials appear in the material list view.

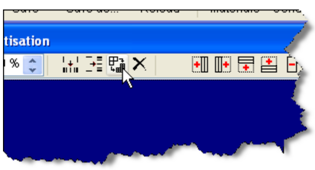


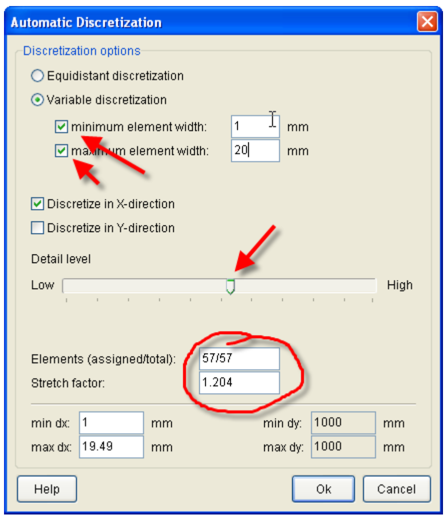
Now the materials are assigned to the appropriate layers of the construction.

To do this, you need to select a layer (or several layers) with the mouse. Then select the desired material in the material list and finally press the green assignment button .

After assigning all materials the construction view should show the layers of the construction with the corresponding material colors. 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.

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).

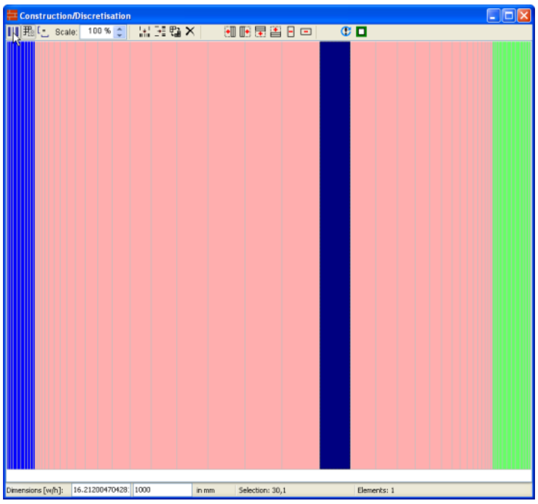


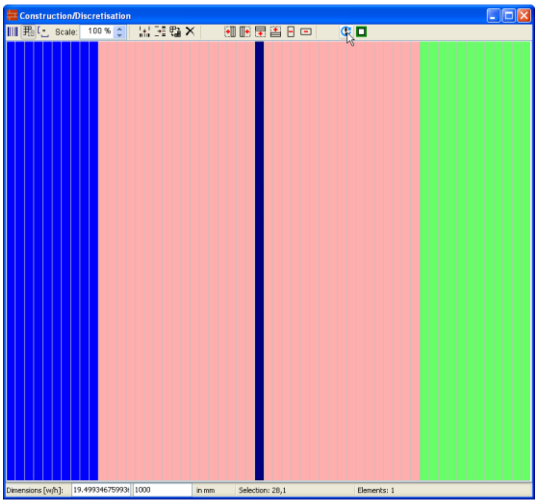


In the Automatic Discretization dialog we will only discretize the construction in X direction (since we have wall with only horizontal transport direction). We use variable discretization, that generates smaller elements near the boundary of the construction and at material interfaces. Inside of the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

Important parameters in this dialog are: Minimal and maximal element widths (1mm and 20mm are good default values), and the detail level. 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 is suitable for 1D-constructions. For 2D-constructions a higher stretch factor of approx. 1.5 or higher is advisable due to long simulation duration. While adding more layers in a construction be sure that every material layer has to consist of at least three discretised elements. As well between adjacent discretised elements of different materials element thicknesses should not differ by more than 1.5.

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 boundary elements.





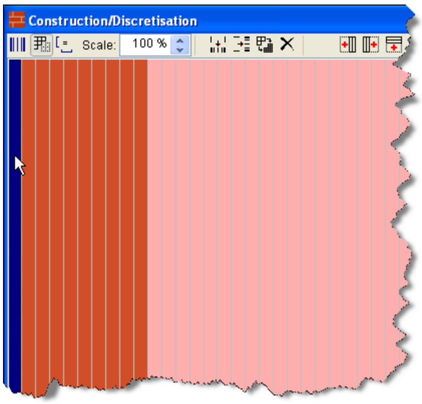
The left screenshot shows the normal proportional view whereas the right screenshot shows the equidistant view.

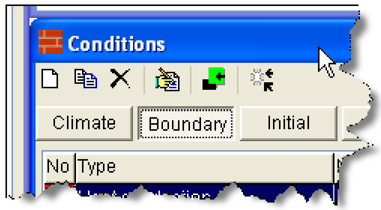
Boundary, Initial, and Simulation Conditions

As the next step all boundary conditions need to be specified and assigned. The project template already contains boundary conditions for the inside and the outside. The specification of own boundary conditions from scratch is covered in the next part of the tutorial.

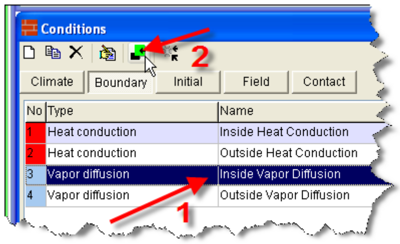
The assignment of boundary conditions is the next step, so that the construction 'knows' to which surface the boundary condition is applied. The usual order is: select a range of elements in the construction view, select a condition, and execute the “assignment” action (the green assignment button).

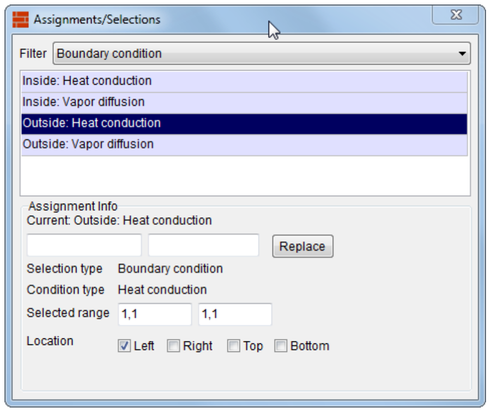
In the case of the boundary conditions, select at first one of the outer layers of the construction. In this tutorial the left side is supposed to be the inside, so the left-most layer should be selected first (see left screenshot below).





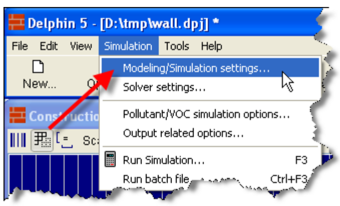
Then, switch in the conditions view to the tab with the boundary conditions "Boundary" (right screenshot above). Finally, select one or more boundary conditions (see left screenshot below) and use the green assignment button (see left screenshot below) to assign the conditions to the selected layers.



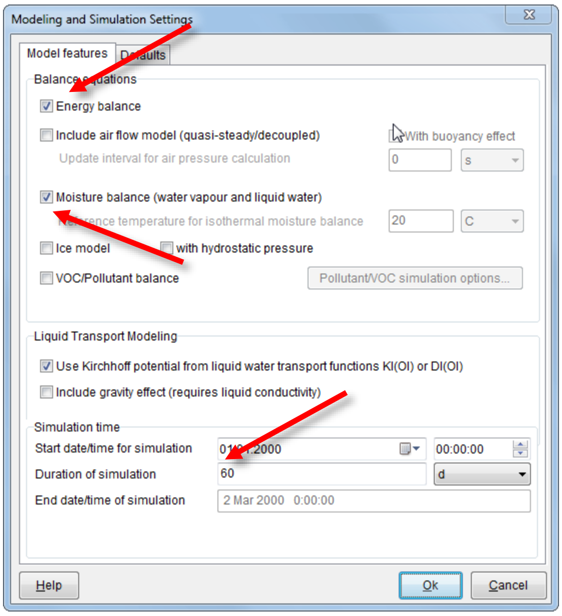


The assignment list shows all assigned boundary conditions and also provides information about the location and the side (surface) of the assignment (right screenshot). All boundary conditions in the template project need to be assigned to the appropriate construction side (2 left and 2 right, thermal conductivity and vapor diffusion respectively), as shown in the screenshot below.

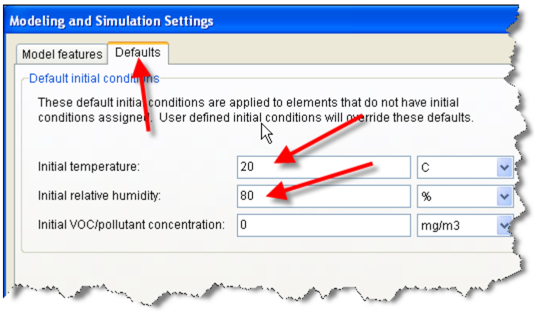
After specifying the boundary conditions it is time to select the simulation and modeling options. Open the modeling dialog:



In this dialog you can select the basic properties of the physical model. For this example you need to enable the balance equations for heat and moisture transport:



Other important settings in this dialog are the simulation duration and the start time point of the simulation. 60 days should be used as duration of the simulation. The start date is not important for a design simulation. However, the initial conditions need to be specified. These can be globally set in the "Defaults" tab:

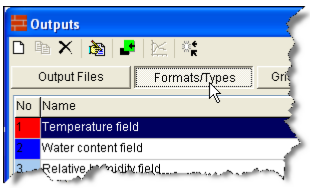


Typical settings are 20°C and 80% relative humidity, which should also be used in this example.

After confirmation of the dialog all settings required for the simulation are given in the Delphin project. However, so far no outputs have been requested. The next steps are selection of desired outputs.

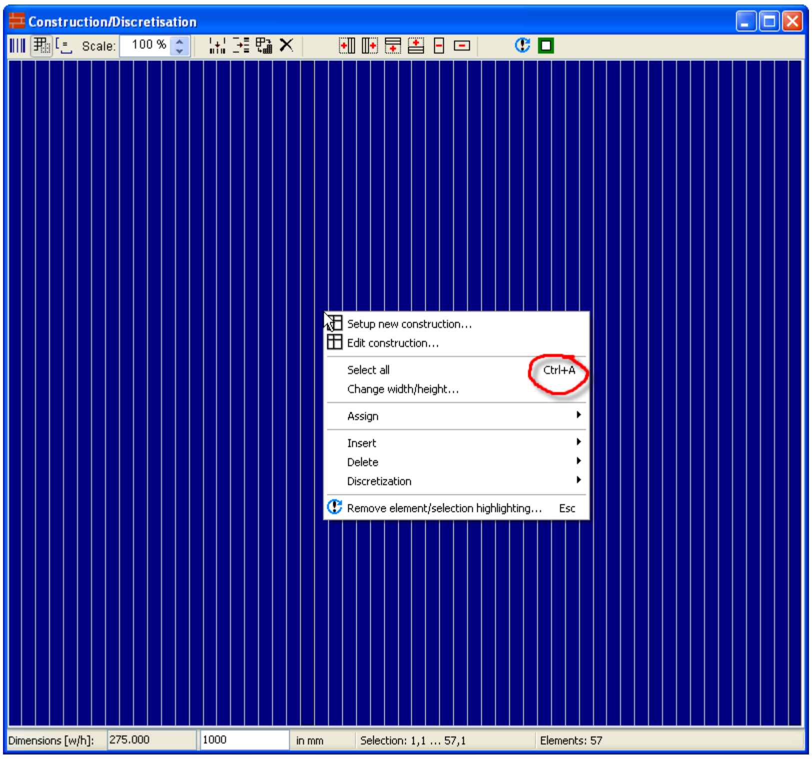
Outputs

Some outputs are already pre-defined in the default project template. These are shown in the output format list, visible in the "Formats/Types" tab of the outputs list view:



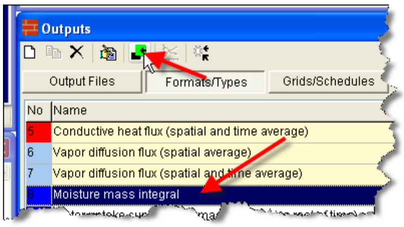
The formats define which quantities (temperature, relative humidity, mass of condensate, etc.) will be monitored. Also, the formats specify whether spatial or temporal averages or integrals should be calculated.

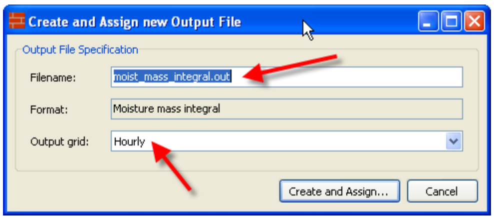
The formats can now be assigned to ranges of the construction, which are of interest. As usual, at first a range of elements needs to be selected in the construction view. In this tutorial, all outputs should be made for the entire construction, so you can simply select all layers.



As shown in the screenshot, the context menu of the construction view contains the option "Select all". Alternatively, you can use the usual shortcut Ctrl+A to select all layers.

After selecting the range of elements of the construction, you can select an output format and assign it with the green assignment button (see left screenshot below).

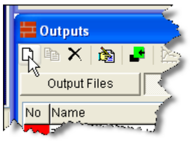


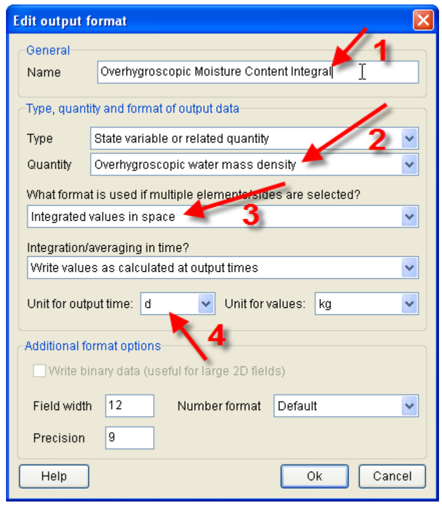


In this tutorial the Moisture Mass Integral shall be assigned first.

After pressing the assignment button the dialog for creating/defining an output file is shown (see right screenshot). Here you need to type a unique file name (without path !). Furthermore, you need to select an output grid. Output grids define when and how often outputs should be made. For integral values, corresponding to a single value per output, you can use hourly values. For fields and profiles you should use larger intervals (e.g. daily output intervals).

Sometimes the desired output formats are not included in the list. In this case you can create your own format, using the "New" command in the output format tab (see screenshots on next page).

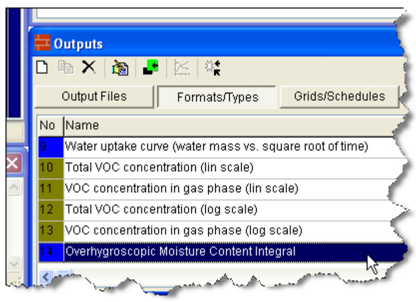


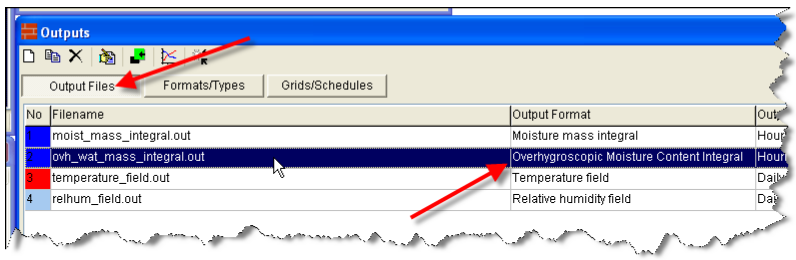


For this example we need to monitor the over-hygroscopic moisture content (= condensate). Im Output Format edit dialog (right screenshot), the following inputs are required: Unique identification name, "Overhygroscopic water mass density" as Quantity, "Integrated values in space" (spatial integration, since we are interested in the total mass of condesate) and as unit for output time points "d" (we calculate 60 days, so we better plot the outputs also in days).

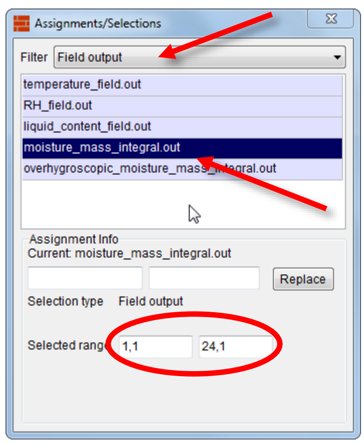
Once the dialog has been confirmed, the new format appears in the list of output formats/types (see left screenshot below). We can now assign the format to the whole construction and create a new file for this output.

For all defined outputs a separate output file is created. All output files are shown in the output file list in the "Output Files" tab. The right screenshot below shows the newly created and assigned output format for the overhygroscopic moisture content.





As already described, the connection between output files and the construction is created via assignments. These are also shown in the assignment list, where you can switch the assignment filter to "Field output":

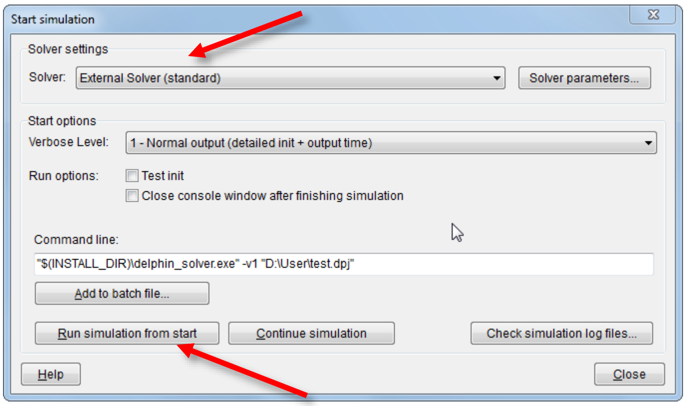


For each assignment you can see the range of elements, the type, and the location of the assignment. ELEMENT indicates here an element specific assignment. For boundary conditions the selected side of an element is shown instead. Note: depending on the discretization used in your project, the range may differ from the one shown in the screenshot.

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

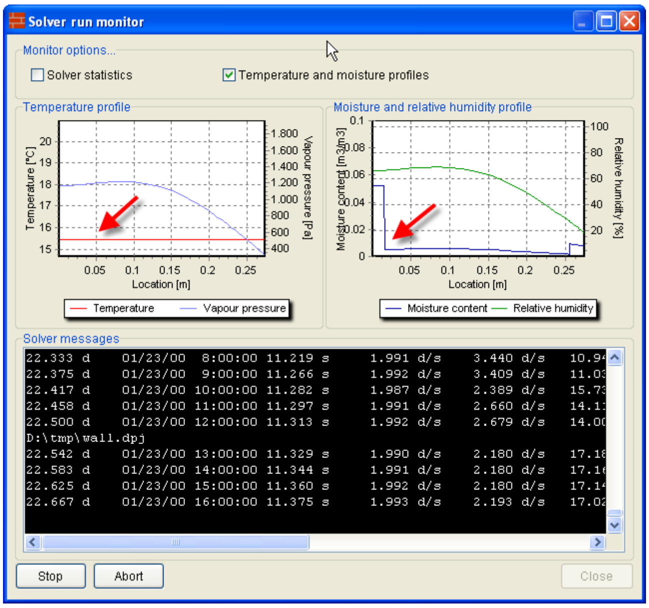
Run Simulation

The simulation is run in the simulation window, which is found in the main menu: Simulations >> Run simulation... or via the command button Sim...:



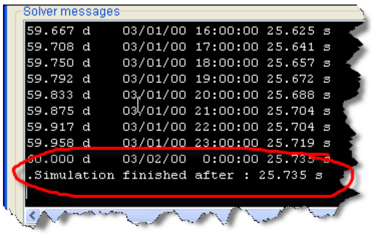
In this dialog you can choose between internal solver (with graphical output of the calculation process) and two external solvers. The latters are much faster and particularly for 2D simulations the recommended choice. In this example the internal solver is shown.

The simulation is started with “Run simulation from start“ and the simulation window opens:

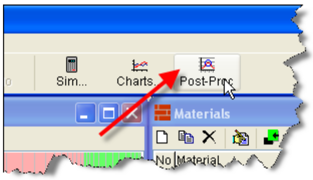


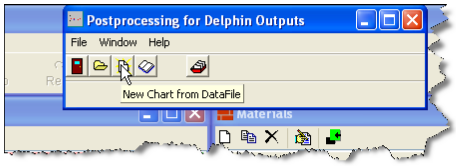
The simulation window shows the current temperature and moisture profiles that can be used to quickly check the results. Also, the current solver performance statistics can be shown.

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:





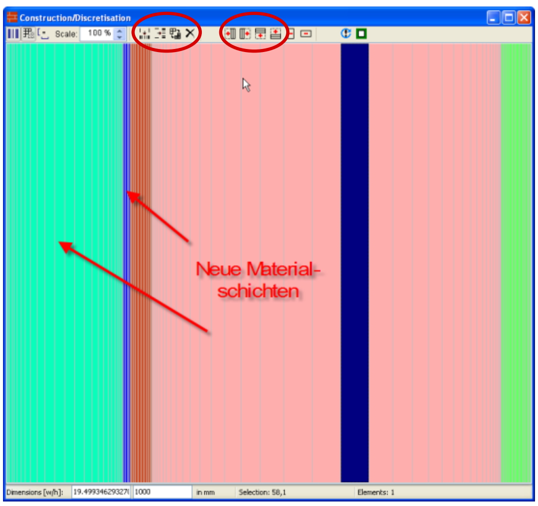
In the post-processing window you can create new charts using the "New" button (see right screenshot above). Further tutorials give an introduction into post-processing with Delphin.

Part 2 : Adding a Capillary Active Inside Insulation

Since the thermal insulation does not meet modern requirements of buildings, an additional insulation is added. In this tutorial we assume that adding an insulation at the outside is not possible (perhaps a historical facade, etc.) Therefore, a capillary active insulation system (using in this example calcium silicate boards) shall be used. The CaSi insulation is fitted using a glue mortar, internal plaster is resigned.

At first, the materials „ CaSi adhesive mortar (light)" (ID 123) and „Calcium Silicate Board" (ID 571) 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, right ellipse). After specifying the thicknesses of the new layers (5mm coving plaster/glue mortar, 80mm insulation) using the input fields at the bottom of the construction view, you can assign the materials to the respective layers (see construction sketch at the very begin of this document).

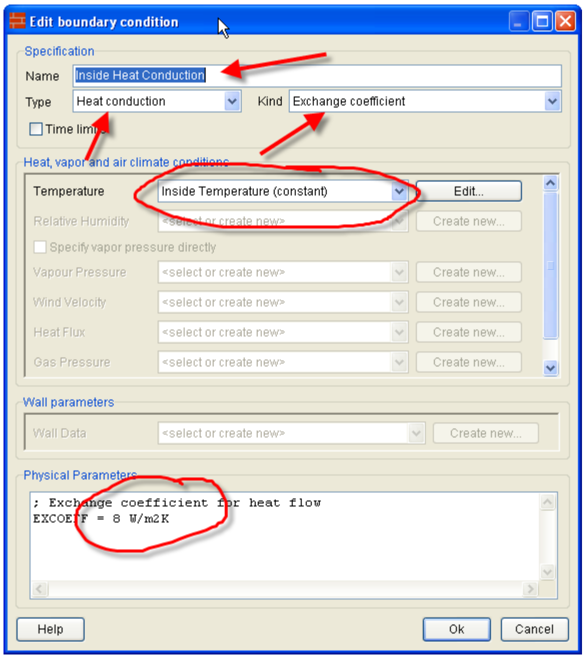
Finally, you can use the Automatic Discretization dialog again to create a grid for the new material layers, or manually discretize each layer with the Discretization dialog (see respective buttons in menu bar, left ellipse).



Whenever the construction has changed it is advisable to check that all boundary conditions and outputs are still assigned to the correct layers.

Now the simulation can be repeated (save the project with different file name first, so that the previous results remain for comparison).

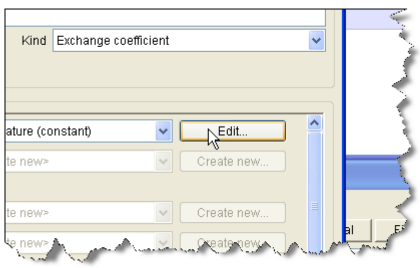
However, the selected boundary conditions in the project template do not quite match the EN/DIN requirements. Therefore, as the next step in this tutorial we will adjust the boundary conditions. The project template contains boundary conditions for heat conduction and vapor diffusion. Open, for instance, the heat transfer boundary condition with name "Inside Heat Condutction" (select the RB in the conditions view, "Boundary" tab, and click on the "Edit" tool button). Now you see the boundary condition dialog:

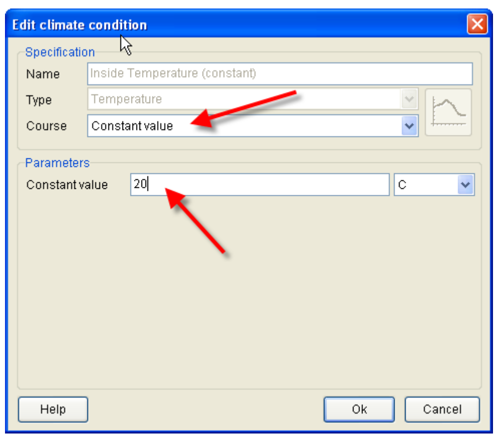


The highlighted inputs are necessary, to sufficiently define the boundary condition. Important properties for boundary conditions are always

- the unique identification name,
- the type (for instance, *Heat Conduction* or *Vapor Diffusion*),
- the kind of boundary condition (corresponding to a certain physical model),
- the required climatic conditions, and finally
- the parameters for the boundary condition model.

In this tutorial the default parameters for the boundary condition are correct, except for the climatic conditions. Depending on the boundary condition type and kind one ore more climatic conditions may be required. These can be selected and modified using the drop-down lists and the edit buttons besides the lists.





The dialog for the climatic condition allows changing and viewing of the climate data. Again, a unique identification name is required, as well as the type of the climate component (for instance temperature or relative humidity), the course (can be constant, following a sinusoidal wave, or double-harmonic function, or can be read from a climate data file). The course can be defined as constant, sinus curve, double sinus curve (daily and yearly course) or as arbitrary e.g. measured data course from a file.

For the definition of the climate on the inside and outside wall surfaces, the corresponding temperatures and relative humidities need to be adjusted. The outside climate should be adjusted to -10°C and 80% RH, whereas the inside climate should be set to 20°C and 50% RH.

If the simulation is repeated with the changed conditions, the amount of interstitial condensate should increase. If the inside insulation system works as expected, the total overhygroscopic mass obtained at the end of the condensation period should still be within the acceptable limits.

… End of 1st Tutorial …