# Simulation of 2D Construction Details with DELPHIN

This tutorial demonstrates how 2D construction details can be simulated with DELPHIN. The general approach on setting up the model and defining all boundary conditions and outputs is shown. A simple thermal bridge problem is modeled and calculated.

2D Thermal Bridge Problem

The detail shown below illustrates the problem of an intersecting wall in a monolitic exterior wall. This model should be simulated with DELPHIN. The exterior wall is made of autoclaved aerated concrete and the intersecting wall is a brick wall.
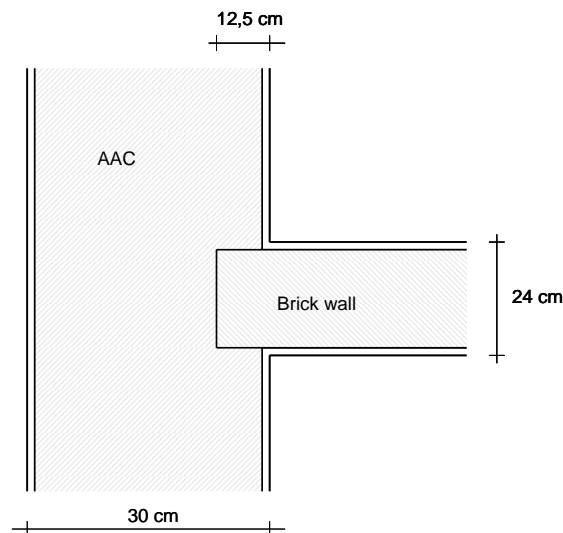
## Simplification of the Constructional Detail

Before a construction is entered into DELPHIN it should be attempted to simplify the model. In this just thermal example, we can omit the plaster layers. Also we can make use of the symmetric geometry.
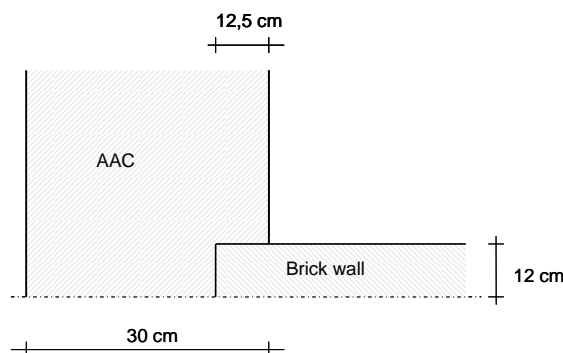
Define Construction Grid

Within DELPHN constructions are defined inside a rectangular construction grid. For this you need to define construction line first. The resulting rectangles are then filled with the corresponding materials, or in the case of whole left empty.

For our construction we can now define the following grid.
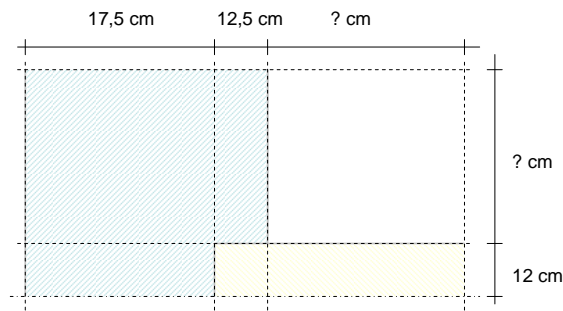
For the sections labeled ? we still need to define a length/width. We have to consider that the length of the intersecting wall and exterior wall are picked large enough, so that the influence of the thermal bridge is no longer important at the clip points. Usually, you can take a length of twice the wall thickness to get an estimate. In our case, we select 60 cm for the length of the exterior wall and 50 cm (rounded from 24 cm * 2) for the intersecting wall. If we should later see in the results that the influence of the thermal bridge extends to these clip points, we can just enlarge the sections a bit.

Input of the model into DELPHIN

We use the same procedure as shown in tutorial 1. When setting up a new project using the default template, we select 2D construction in the construction definition window. We also input the initial grid dimensions. Later, we can always change the row heights and column widths or add new layers.
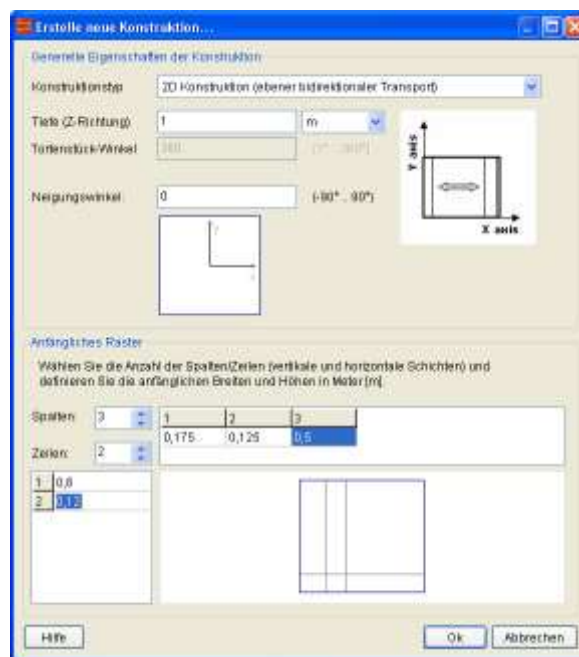
When entering the basic grid geometry you should note that the **grid widths and heights are in m**.

Now we import the materials and assign them to the respective sections in the construction. We use the **Autoclaved Aerated Concrete** and **Brick Joens** as materials.

In the following, we talk about the assignments. These are rectangular ranges of the construction which can be selected with the mouse or keyboard in the construction grid. In the footer of the construction window you can see the current selection, for example showing 1,1....3,2:

For 2D constructions you can choose among different possibilities to assign materials to the respective sections.

1. Subsequent selection of ranges with elements of a specific material and assignment until all elements have been assigned (can be a bit cumbersome with complex 2D details)
2. Making use of selections that overlay each other and overwrite previously made selections. In our example we could first select the range 1,1...2,2 and assign the AAC material. The we can select 2,2...3,2 and assign the brick material. With this operation we overwrite the AAC with brick in the element 2,2.

The second option allows often a quicker setup of the construction model. There is also a special case of a material assignments, particularly suitable for hollow sections or corners. In the tool bar you can use the symbol ▢ to assign an empty range. With this you can override previously made assignments (if you want to remove a material assignment, it is better to just remove the respective assignment in the assignments list window).
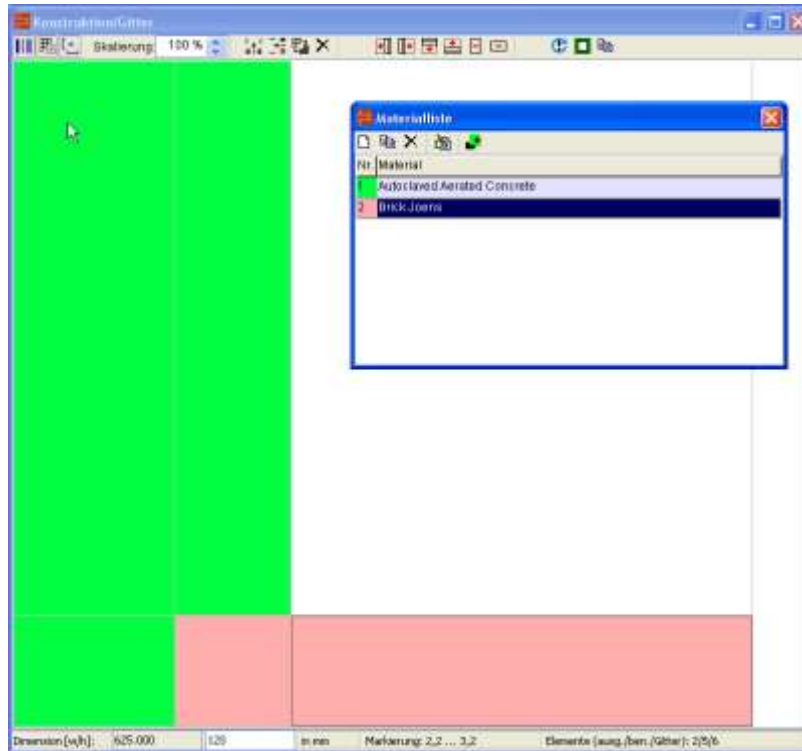

Grid generation

After we have created the construction grid and assigned the materials, the simulation grid needs to be created. This is done most easily with the dialog for automatic discretization. In this dialog you should make sure that both directions are checked (this is the default for 2D constructions), alternatively you can also specify the minimum element widths. In our example we use the standard settings and create the following computational grid.
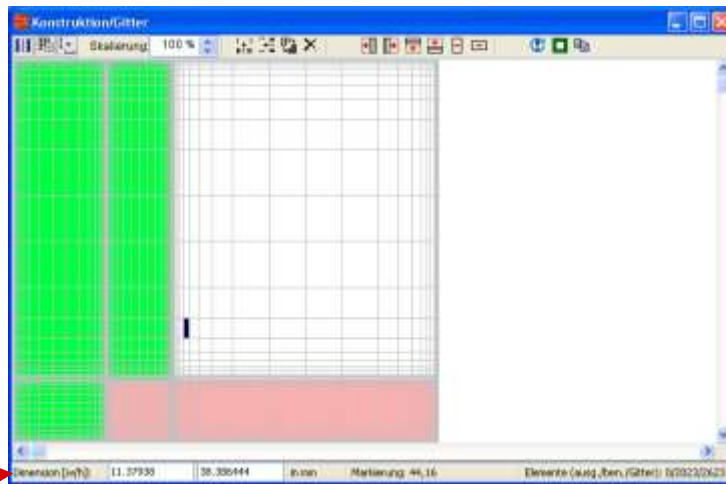
Status line

At this point we may take a moment to explain some functions of the construction grid. In the status bar (footer) the current selection is shown, and also two input fields for the width and height of the currently selected column and row. If you have selected multiple rows or columns the respective input field just shows the sum of the widths/heights, but the input field itself is disabled. You can only change the row height or column width of a single row or column at a time. The status bar also shows the grid size in three numbers. First number is the number of currently selected elements, second number is the number of used grid cells (elements with materials assigned) and the last number is the total grid size. A value of < 6000 is generally to be recommended for quick calculations. For purely thermal calculations you may use values up to 30000 and still get reasonable performance.

The automatic discretization algorithm has generated a grid that is refined at all boundaries. However, we have a symmetry plane at the bottom and top side of the construction. Here we don't need the refinement because we don't have temperature gradients across a symmetry line. Therefore we could save grid elements by manually enlarging the elements using the manual discretization dialog, accessible via the toolbar buttons ⊞ ⊡. Saving grid cells can significantly increase the calculation speed in hygrothermal calculations of complex details. In our simple example, however, this is not necessary.

The first top tool buttons ▌▌ ⊞ ⊡ in the construction window serve to alter the display of the simulation model in the construction grid: proportional versus equidistant drawing, show/hide grid line, draw with the same aspect ratio or stretch both axes to fill the window.

Assigning boundary conditions

We need to assign boundary conditions (BC) to each boundary of the construction. The following sketch illustrates the necessary boundary conditions and the related climatic conditions.
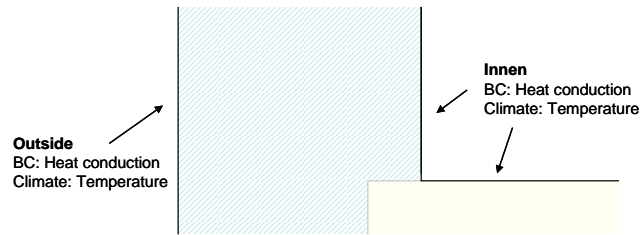
At symmetry lines and at borders of the calculation domain (where 1D conditions are expected) you do not need to assign boundary conditions. A boundary without assigned conditions is perfectly tight and adiabatic. In our example we just need heat conduction boundary conditions with the corresponding temperatures.

The default project template already contains climatic and boundary conditions, which we will just use here. The assignment of boundary conditions is done as described in detail in Tutorial 1:

1. Select range of elements where boundary conditions need to be assigned
2. Select boundary conditions
3. Assign boundary condition (by clicking on the green arrow button in the condition view)

The following screenshot shows how the boundary condition is assigned to the inside wall surface. It is recommended to use the equidistant drawing option of the construction grid to simplify selection of the element range.
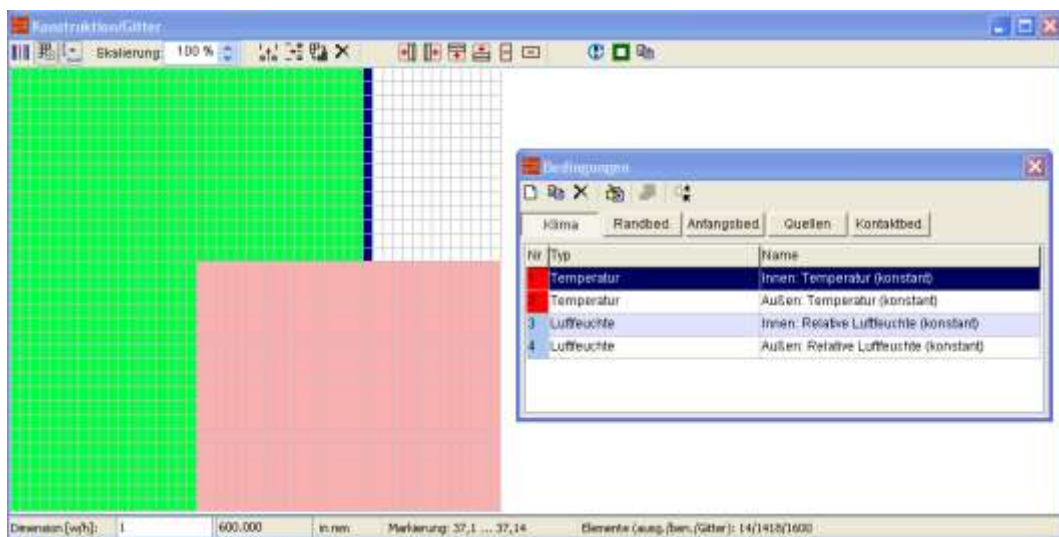

After clicking the assignment button a dialog pops up with the side selection options.
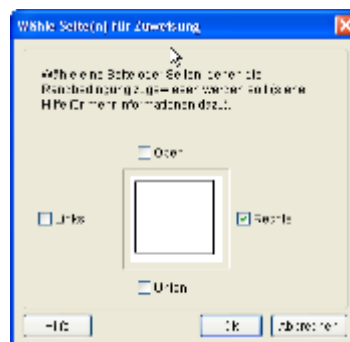
Here, you need to select the respective side of an element for the assignment of the BC. You have to remember that **boundary conditions inside the construction grid (i.e. between materials) are invalid**.

When all boundary conditions have been assigned, it is possible to verify the placement of the assignments by clicking on individual boundary conditions. For example, if you select the heat conduction boundary condition for the inside, the construction view will highlight all elements that have this assignment.
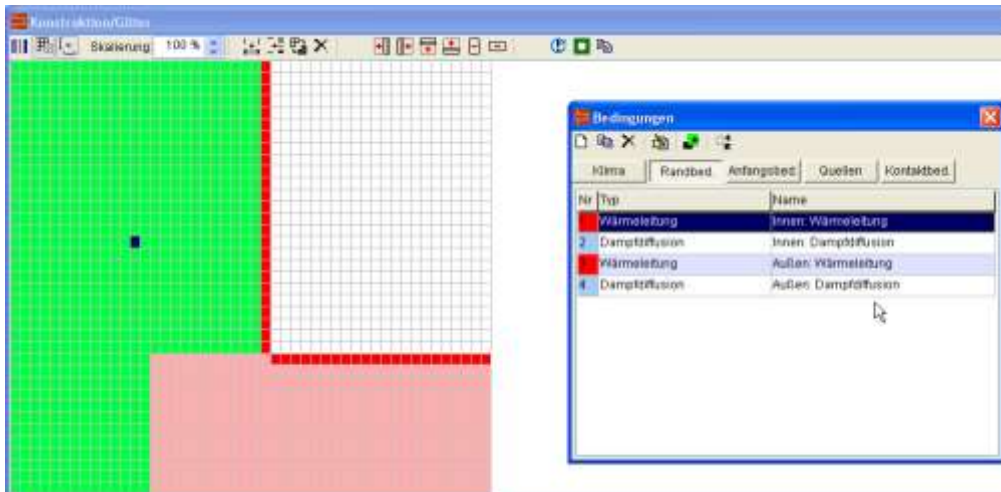

In the assignment list window you can also check all assignments individually. There should be 3 boundary condition assignments in total: one for the heat conduction at the outside, and two for the heat conduction at the inside, just as it is shown in the screenshot above.

Tasks

In a simple thermal bridge calculation the temperature field is interesting purely for visual analysis, e.g. to find the point with lowest temperatures in complex geometries. Critical for design decisions and code compliance are, however, the temperatures in critical sections, most importantly the inner wall surfaces. Also of interest are the actual heat fluxes through the inside wall surface to quantify the heat loss through the thermal bridge.

The post-processing allows extracting temperatures at specific points as well as horizontal and vertical cuts. Therefore we will first assign a temperature output to the whole construction.

The default project template already contains some pre-defined output formats. In the output window you may, however, also define your own output formats. The following screenshot shows a possible definition of a format for a temperature field output.
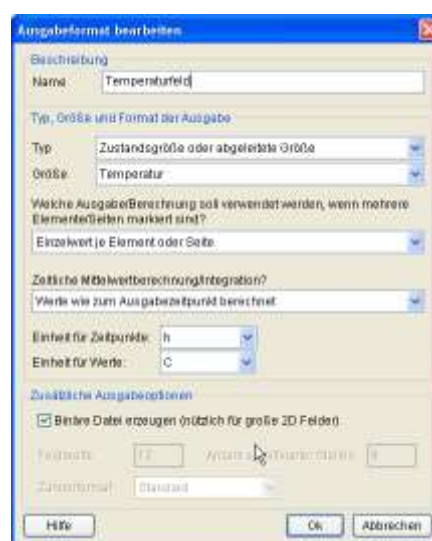
The assignment of a temperature field output is done as usual in the three steps:

1. Select range of elements for which the output is to be obtained

2.   Select output format (or output file, in case you want to combine several selections into a single output file)
3.   Assign the output format or output file definition

If an output format is assigned, a dialog is shown where you can specify additional information for the output file to be created (output format, output time grid, and file name of the output file).

In our example, we want to create an output for the whole construction. The keyboard shortcut Ctrl+A can be used to select all elements, alternatively you can also select the whole construction with the mouse. When specifying the output time grid, it is here meaningful to use hourly values (even though we are just interested in the final steady-state result).

Sometimes you may just want to obtain an output of a part of the construction, for example, the total moisture mass in a certain material. In our case, we could just want to obtain the temperature field of the aerated concrete (even though this doesn't really make much sense). However, we have to assign an output now to an element range that is not rectangular. So we have to make two assignments. For such cases, the following procedure is recommended:

1.   In the output window select the register card "Output Files".
2.   Create a new output file definition (press "New" tool button) and select here the temperature field as output format, a time grid and a file name, for example "temperature_field_aac.out" and confirm the dialog.
3.   Now select the element ranges covers by the AAC material in two steps and press the assignment button in the output window. When done correctly, a click on the new output file will highlight just the AAC material.

This procedure is quite useful if integral outputs are necessary, such as integral moisture content of a insulation with non-rectangular shape.

For the output of an average temperature in a specific area of the construction, such as inner corner, one more output format has to be defined which differs from the indications in the previous page: instead of "Single values for each element or side" choose the format "Averaged values of elements….". This format can be assigned to one or several elements and delivers every output point in time one value per assignment.

When obtaining outputs of heat flux of other flux densities, one has to keep in mind the sign definition for flux quantities in DELPHIN. If you combine flux outputs not correctly, sometimes flux densities with the same direction (for example heat flux from the inside to the outside) may cancel each other out.

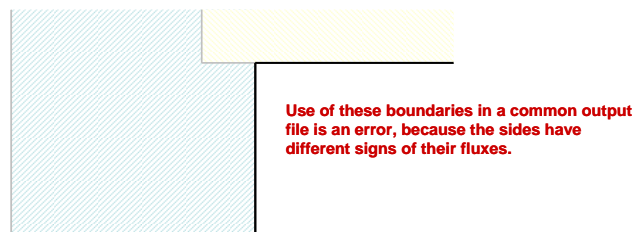The following diagram illustrates the sign definition used in DELPHIN.
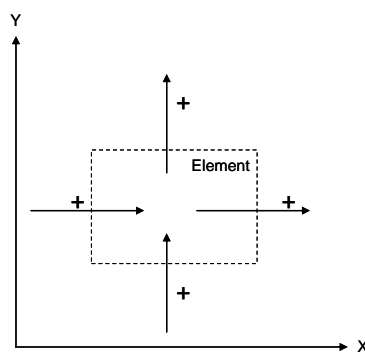
This sign definition is used independently whether a side is between two elements or a boundary side. Flux densities in direction of the coordinate axes are always positive. That has the advantage that one can define cut planes through

the construction and obtain the heat flux through this cut plane, as long as the sides used in the cut plane are of the same sign.

An example may help to illustrate this. If we want to obtain the heat flux from the inside to the outside in our example thermal bridge, we need to output the integral (spatial integral) of the heat flux densities across the inner surface. If we define the construction as we did before, we could simply create an output file for integral heat flux densities and assign this output file to the two interfaces of the construction to the inside, as shown in the picture below.

Use of both boundaries in common output file is ok, because both sides are "positive Flux" sides.
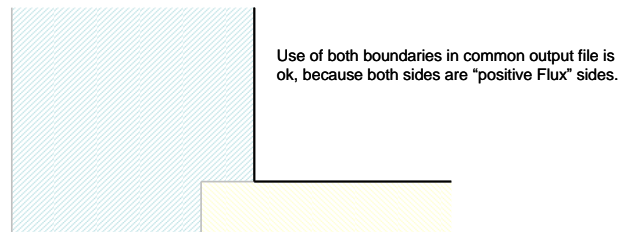
Both boundaries have the same sign definition which allows us to combine the two outputs into a single file for the integral heat flux density.

Use of these boundaries in a common output file is an error, because the sides have different signs of their fluxes.

If the construction were defined as above, the assignment would be wrong. A heat flux from the inside to the outside would yield a positive flux for the top interface (flux in positive y-direction), yet the flux through the vertical interface would yield a negative flux (against the x-direction). When you add these up in an integral heat flux output the flux would cancel each other, even though both flux indicate a positive energy flux from the inside to the outside.

This strict sign definition has, however, the advantage, that the heat loss through the construction can be obtained much more easily. First we use the output format for the averaged heat flux density.

When defining the output it is important to set the output type to a "Flux type" and select the weighted average as calculation method.

Once the output format has been defined, the output can be made by placing a vertical cut through the construction, as shown in the following screenshot.
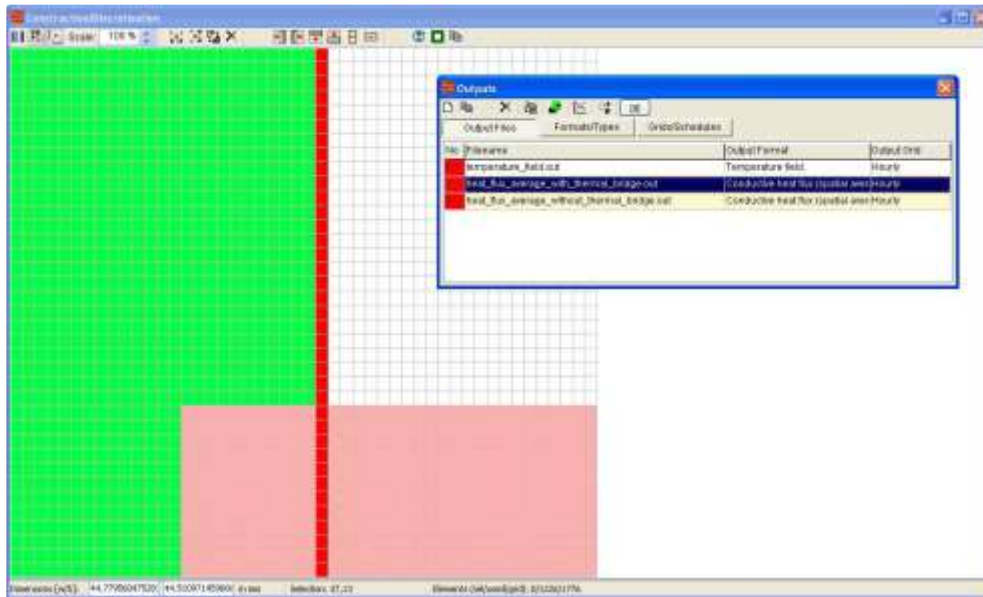
For comparison you can create another output just for the inner boundary element of the first row. Here we are far away from the thermal bridge and should have (approximately) one dimensional conditions. A heat flux obtained here corresponds to the heat flux of the wall without the thermal bridge. We can use this as reference value for the rest of the wall.

## Simulation Settings and Starting the Simulation

Once the construction is modeled and boundary conditions and outputs have been assigned, we only need to set the simulation options. In the simulation settings dialog we deactivate the moisture mass balance, and reduce the simulation time to 2 days. Within this time we should reach a steady state.

Now the simulation can be run and should be finished within a few seconds.

Post-Processing

The analysis of the outputs is done with the DELPHIN post-processing tool, or alternatively with other external tools such as TecPlot. The details of how to use the DELPHIN post-processing is done in a separate tutorial, here we will just analyze the results.

The calculated steady-state temperature field is shown below in the interpolated and discrete mode. In the latter mode (right) we can clearly see the underlying computational grid. In areas with large gradients the grid should be refined for more accuracy. Furthermore we see that at the top of the construction we just reach 1D conditions whereas the intersection wall is much too long (nothing happens further to the right). So we could shorten the wall at the right side by about 30 cm and save computation time (in our example this isn't relevant, but it may be in transient hygrothermal simulations).
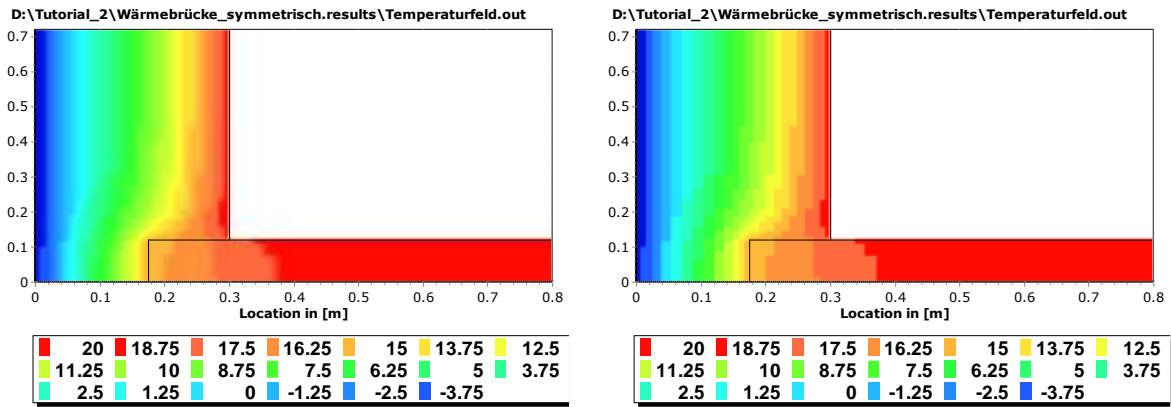
When we obtain the course of the temperature at the critical pointer (inside corner), we see that even after 2 days we haven't reached steady-state yet. For the screenshot below we have continued the simulation for another 2 days (first change the total simulation duration in >> Simulation >> Modeling/simulation settings dialog then use the "Continue" button in the simulation start dialog).
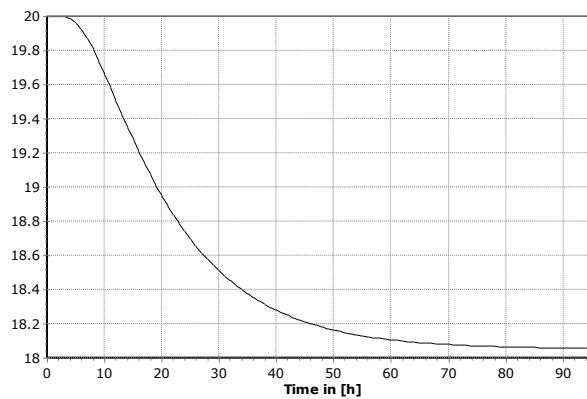
If you now consider the heat flux outputs below, we notice the negative sign of the flux, which means against x-axis and therefore from inside to outside, as expected.
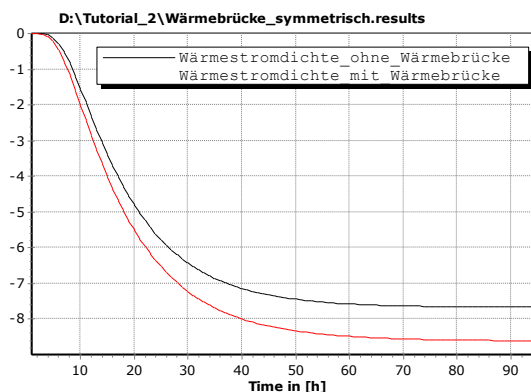
Also, we see that the thermal bridge does increase the heat flux through this part of the wall by about 12 % compared to a plain wall (red curve is the output with thermal bridge, black is the reference).

Summary

In this tutorial we saw the principle steps of setting up a simulation model of a 2D construction detail. Similarly, one can setup a simulation for hygro-thermal problems or much more complex geometries.

Tasks

1. Model the construction above without considering the symmetry and compare the results!
2. Refine the grid by adjusting the discretization options and compare the steady-state temperature in the wall corner (mind that after modifying the discretization you may need to update the assignments for flux outputs)!
3. Add an inside insulation and compare the influence of the thermal bridge on critical temperature and heat fluxes!
4. Run the simulation with and without inside insulation as hygro-thermal simulation. In addition to the heat conduction boundary condition, you need to specify also the vapor diffusion boundary conditions.

… End of 2nd tutorial …