

DELPHIN 6 – Tutorial 2

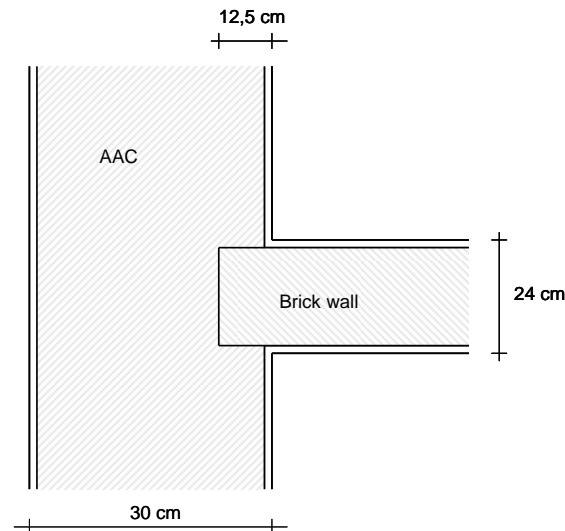
1.	Simulation of 2D Construction Details with DELPHIN	2
1.1.	2D Thermal Bridge Problem.....	2
1.2.	Simplification of the Constructional Detail.....	2
2.	Define Construction Grid	2
3.	Input of the model into DELPHIN	3
4.	Assigning interfaces (boundary conditions)	5
5.	Grid generation	7
6.	Outputs	8
7.	Post-processing	10
8.	Conclusion	11
9.	Additional tasks.....	11

1. Simulation of 2D Details of Constructions with DELPHIN

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

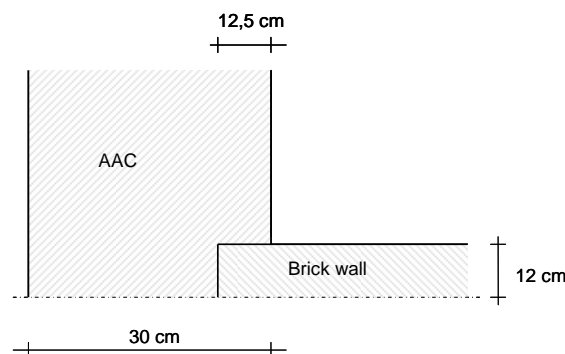
1.1. Two-dimensional Thermal Bridge Problem

The detail shown below illustrates the problem of an intersecting wall in a monolithic exterior wall. This model shall be simulated with DELPHIN. The exterior wall is made of autoclaved aerated concrete and the intersecting wall is a brick wall.



1.2. Simplification of the Constructional Detail

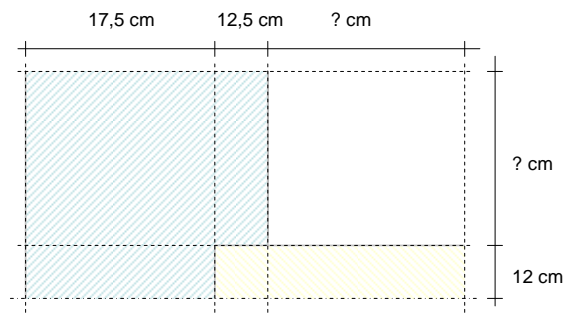
Before a construction is entered into DELPHIN it should be attempted to simplify the model. In this example the plaster layers are omitted. Another possibility to simplify the construction is making use of the symmetric geometry and set the cut along the axis of the inside wall. This will reduce the number of discretized elements.



2. Define Construction Grid

Within DELPHIN constructions are defined inside a rectangular construction grid. Firstly construction lines are defined, then the resulting rectangles are filled with the corresponding materials or, in the case of a hole, left empty.

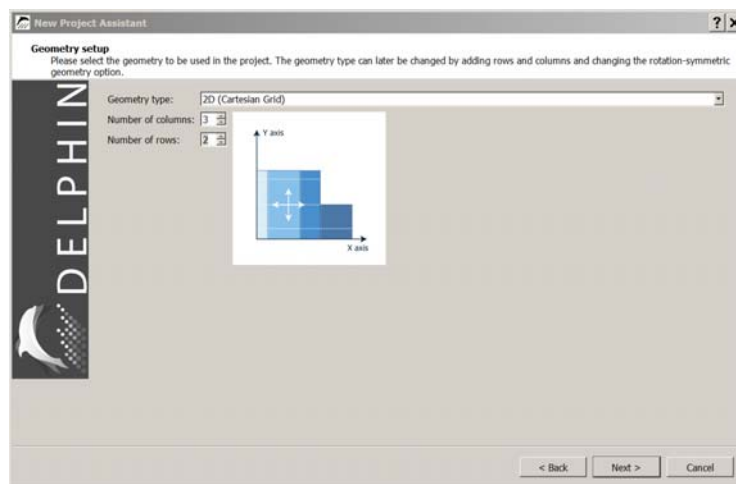
The following grids defines the construction:



For the sections labeled '? cm' a length/width is needed still. It should be taken into account 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. Usually, at least twice or better thrice the thickness of the wall is a good choice for the length. In this case, 60 cm are selected for the length of the exterior wall and 50 cm for the intersecting wall. In case it will become visible from the results that the thermal bridge influences the one-dimensional wall construction, the sections can be enlarged a bit.

3. Input of the model into DELPHIN

The same procedure as shown in tutorial 1 will be followed. When setting up a new project, the project information will be left empty and Potsdam will be set as location for weather data. Then select '2D construction' in the construction definition window. Row heights or column widths can be changed at any time easily. Equally additional layers can be added or deleted.



The next dialog sets basic outputs. For now, the settings are kept as they are. Later additional outputs will be added.

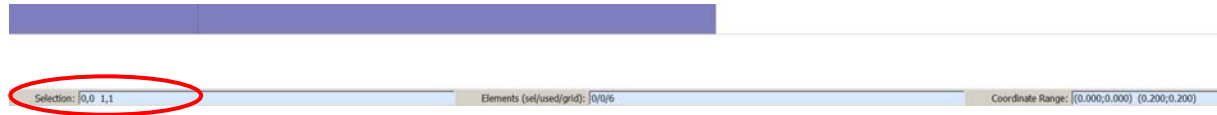
Now materials shall be imported from the data base into the project and assigned to the respective sections in the construction: **Autoclaved Aerated Concrete** (ID 1) and **Brick Joens** (ID 34).

The next step is to change row and column sizes. Starting with columns first select the left most column. Then change the width in the edit field in the footer to 175 mm.



Proceed in the same manner with all other columns and rows.


In the following, assignments are described more precisely. All discretized elements are rectangular. Assignments can be done to single elements or ranges of the construction. The elements 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 0,0....1,1 (Delphin starts to count at 0):



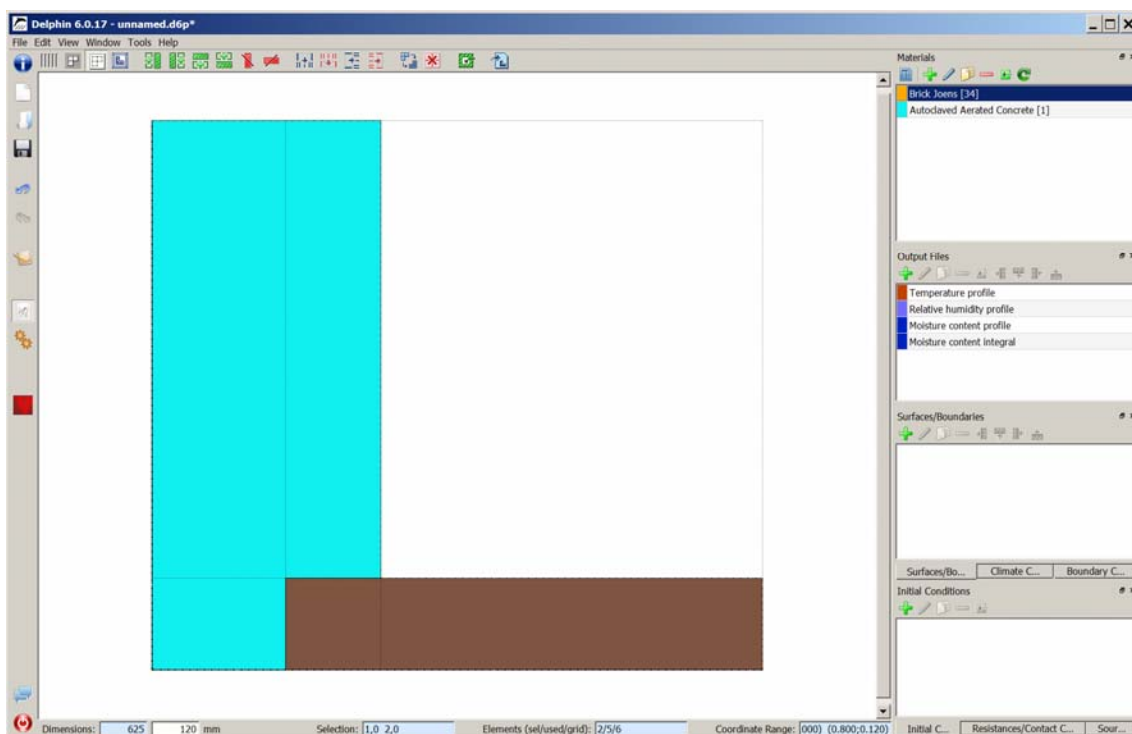
That means from coordinates $x=0$ and $y=0$ (lower left corner) to $x=1$ and $y=1$ (4 elements). The counting starts in the left bottom corner like a normal cartesian coordinate system.

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

1. Consecutive selection of ranges with elements of a specific material and assignment until all elements have been assigned (can be laboriously with complex 2D details).
2. Making use of selections that overlay each other and overwrite previously made selections. In the presented example first the range 0,0...1,1 (the first two columns) shall be selected and assigned with AAC. Then the elements 1,0...2,0 are marked and brick material is assigned. With this operation the AAC element 1,0 is overwritten with brick.

The second option often allows 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 one can use the symbol  to assign an empty range. In so doing previously made assignments can be overwritten. Such an area is empty and also is not regarded as air. Therefore it is possible to add boundary conditions there. An air cavity in a construction can be carried out with the aid of an air “material” instead.

If a user merely wants to remove a material assignment it is better just to remove the respective assignment in the assignments list.



4. Assigning interfaces (boundary conditions)

In this chapter interfaces will be assigned to certain boundaries of the modeled 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) boundary conditions are not needed. A boundary without assigned boundary conditions is perfectly tight and adiabatic. In this example only heat conduction boundary conditions with the corresponding temperatures are needed.

Two interfaces have to be defined: one for outside and one for the inside climate. The project assistant allows to select a location for the outside climate. The outside climate settings should be set like the picture below.

The screenshot shows the 'Interface/Boundary condition' dialog box. The 'Specification' section has 'Name: [Outside]' and 'Type: [Standard interface for outdoor climate [EngineeringOutdoor]]'. The 'Surface Properties' section has 'Orientation [0..360 Deg]: [180]' and 'Inclination [0..180 Deg]: [90]'. The 'Outside Conditions' section has a dropdown menu set to 'User-defined outdoor climate [OutdoorUserData]'. It contains several checkboxes and input fields:

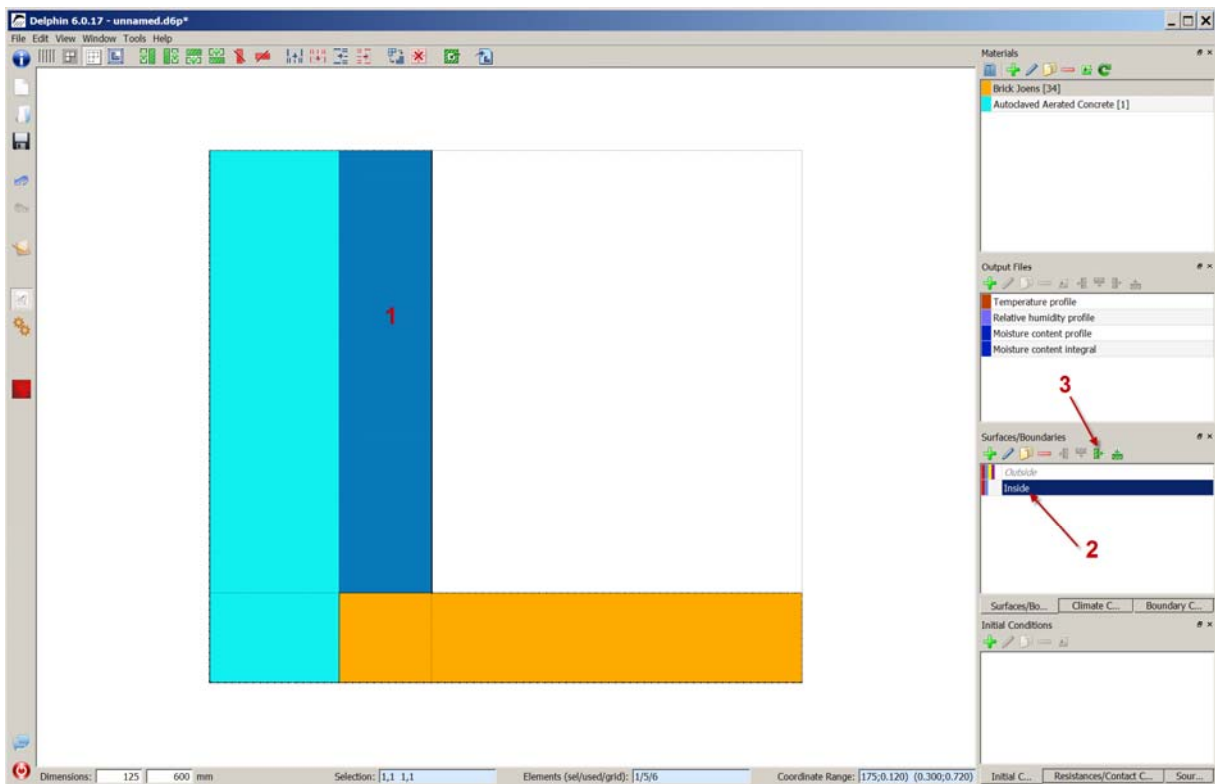
- Heat conduction: h_c - Convective heat conduction exchange coefficient [W/m2K]: [12]; h_r - Radiant heat conduction exchange coefficient [W/m2K]: [5]; Effective heat conduction exchange coefficient [W/m2K]: [12]
- Vapor diffusion: Vapor diffusion mass transfer coefficient [s/m]: [7.5e-08]; sd-value of painting / surface coating [m]: [0]; Compute with Lewis relation: [button]
- Short-wave solar radiation: Solar adsorption coefficient [-]: [0.7]
- Long-wave radiation exchange: Long-wave emissivity [-]: [0.9]
- Wind driven rain (DIN EN ISO 15927-3): Reduction/splash coefficient [-]: [0.7]

In this tutorial driving rain is not considered. The wall orientation is 180 Deg (South). The inside interface should use constant conditions according the german standard 4108-3 from 2014.

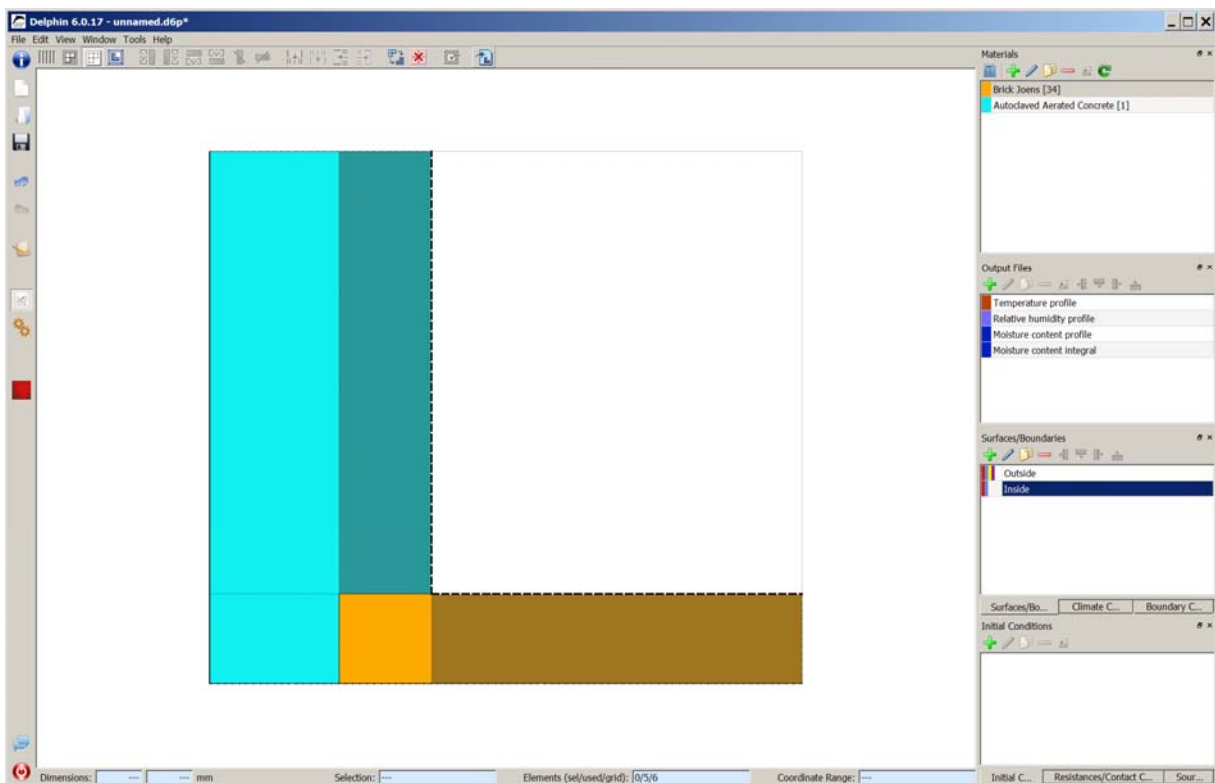
The assignment of boundary conditions is done as described in detail in Tutorial 1:

1. Select range of elements where boundary conditions need to be assigned
2. Select interface
3. Assign interface (by clicking on assignment button with correct side)

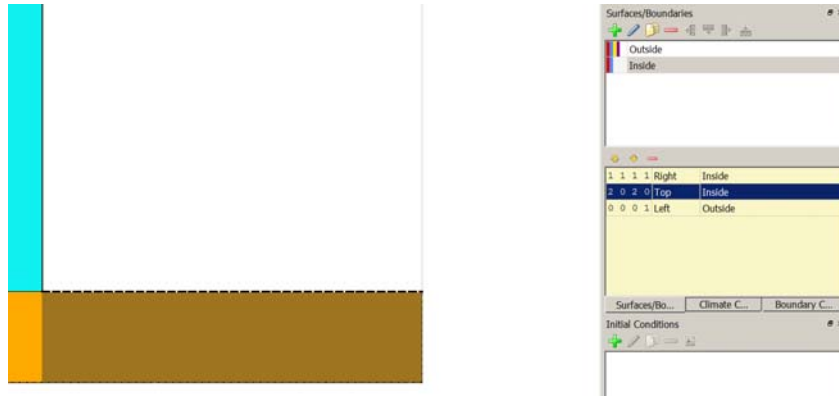
The following screenshot shows how the boundary condition is assigned to the inside wall surface.




When all boundary conditions have been assigned, it is possible to verify the assignments by clicking on individual boundary conditions. For example, if you select the 'Inside' interface, the construction view will highlight all sides with this assignment by a dotted line.

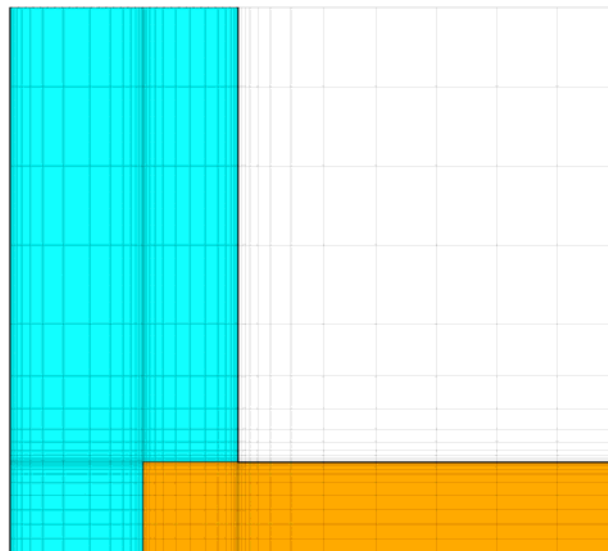


In the assignment list window you can also check all assignments individually. In this example, there should be 3 interface assignments in total: one for outside and two for inside climate, just as it is shown in the screenshot below.

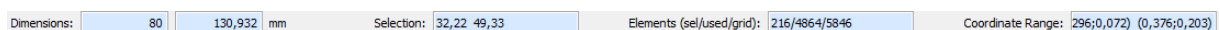


5. Grid generation

After the construction grid has been determined and the materials have been assigned, 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 as this is a 2D construction (default for 2D constructions), alternatively you can also specify the minimum element widths. In this case use a maximum element size of 20 mm and a stretch factor of 1.6. The next picture shows the resulting grid.



Some remarks to explain different functions of the status bar in the construction grid.



The status bar (footer) gives several information when one or more elements are selected:

- At 'Dimensions' the width and height of the currently selected column and row are given. If multiple rows or columns were selected these fields just show the sum of all widths/heights and the input function of the field is disabled. If only one row or height has been selected the column width or row height can be changed.
- At 'Elements' the status bar shows three numbers: the first gives the number of currently selected elements, the second number gives the number of used grid cells (elements with materials assigned) and the last

number gives the total grid size. A value of < 5000 is generally to be recommended for quick calculations. For purely thermal calculations you may use values up to 20000 and still get reasonable performance.

- At 'Selection' and 'Coordination Range' the fields show the number resp. the coordinates of the currently selected elements, starting with the first element in horizontal direction, then the first element in vertical direction, starting from the lower end, and the last element in horizontal resp. vertical direction. DELPHIN counts the elements starting with '0', not '1'.

The automatic discretization algorithm generates a grid that is refined only towards boundaries where an (climate) interface is assigned. This is the reason why the interface assignments should be done before discretization.



The tool buttons on the upper left in the construction window serve to alter the display of the simulation model in the construction grid: proportional versus equidistant drawing, show/hide grid lines, draw with the same aspect ratio and stretch both axes to fill the window.

6. Outputs

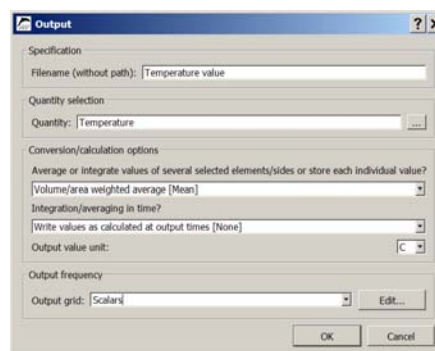
In a simple thermal bridge calculation the temperature field is interesting purely for visual analysis, e.g. to find the point with the lowest temperature in a complex geometries. Critical for design decisions and code compliance are, however, the temperatures and relative humidities in critical sections,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

The post-processing allows extracting temperatures (humidities, ...) at specific points as well as horizontal and vertical cuts. At first the temperature field of the whole construction will be generated.

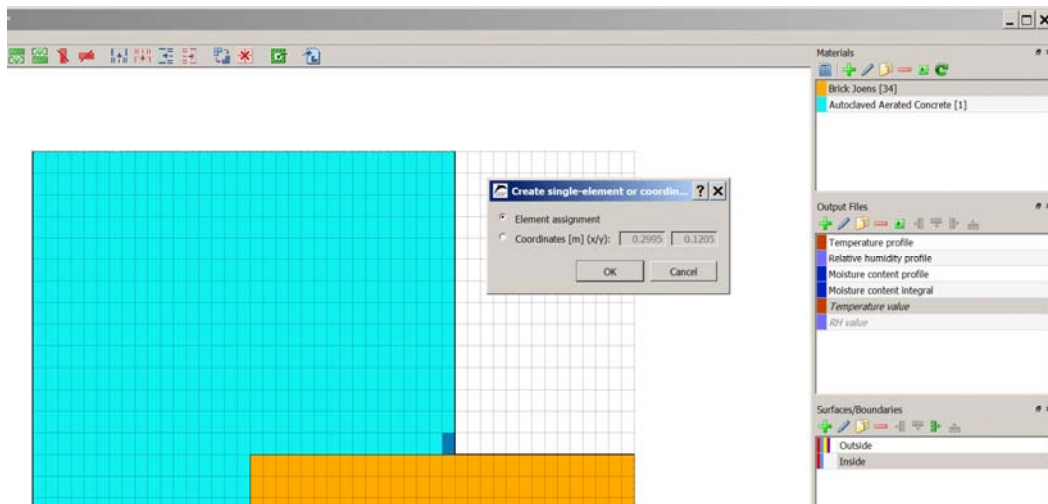
The Project Assistant contains some pre-defined outputs. In the current case these are:

- Temperature, relative humidity and moisture profile
- Moisture mass integral

In the window of 'Output files' a lot of different own outputs can be defined. The following screenshot shows a possible definition of a format for a temperature value output at one point or area. It would be useful to specify in the name where exactly the output will be assigned to.



In the same way an output for relative humidity can be generated. Then both should be assigned to the critical point of the construction. In this example this is corner AAC touches the inside wall. As elements can become quite small and the selection of a few small elements can be challenging, most of the times it is convenient to switch to the equidistant view. Then the elements can easily be selected and assign with the temperature and relative humidity.

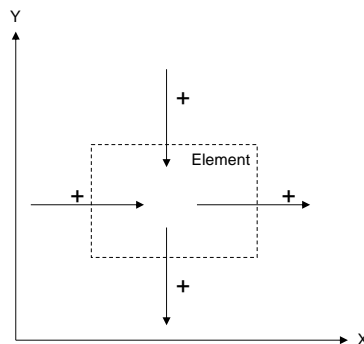


DELPHIN offers to assign outputs to element/-s and to coordinates. Here the assignment of a single element assignment instead of coordinates is chosen.

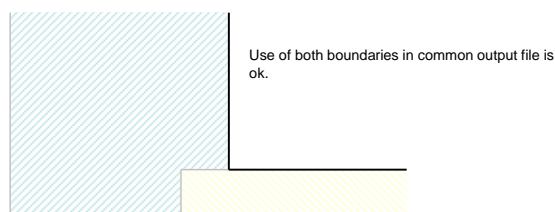
The next output will be an output for heat flow densities. This is useful to obtain the additional energy losses of the thermal bridge. In this case two outputs for each inner side are necessary.

For fluxes DELPHIN 6 has two different settings:

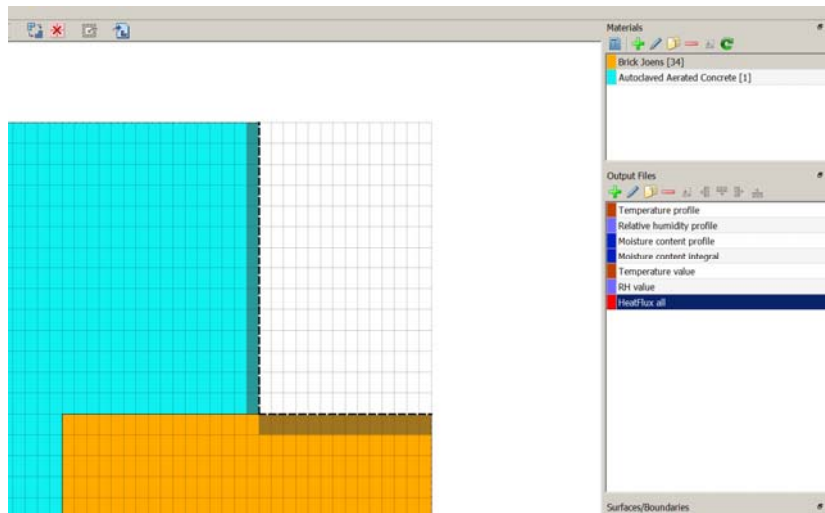
- Boundary fluxes are positive for inflow and negative for outflow
- Field fluxes depend on direction definition (see picture below)



If a selection contains boundary elements as well as field elements the setting for field conditions will be used. It is possible to assign a flux output on different element sides for one output file. In this case the assignment takes place consecutively for both sides. In case of an averaging output was defined the result will be the area/volume averaged values for both sides. This can be done e. g. for all boundaries.



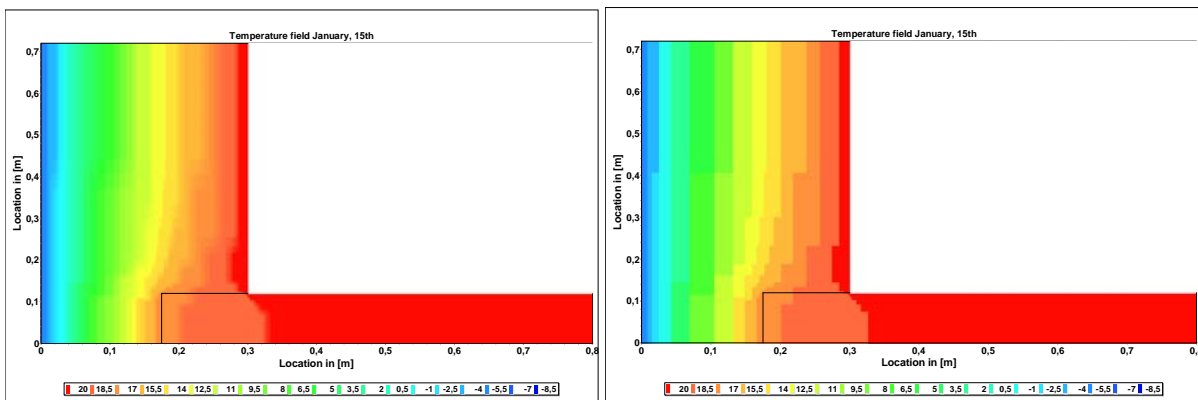
Here such a boundary output is created for heat flux.



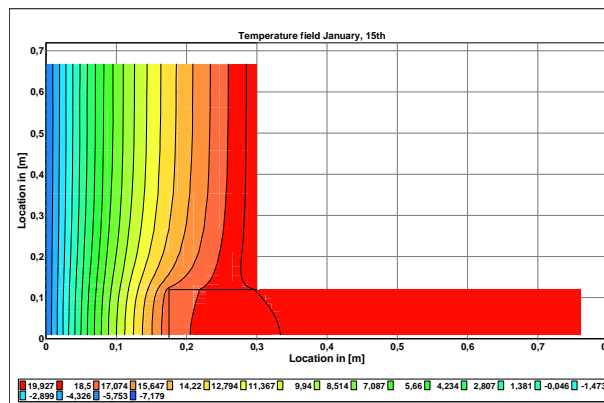
7. Post-processing

For detailed information about post-processing see the related tutorial 3. The following text gives a short overview with the DELPHIN 5 post-processor which is included in DELPHIN 6 (Edit >> Preferences >> External tools).

The next pictures show the temperature field on January 15th (after 360h) in interpolated (left) and non-interpolated mode (right). The non-interpolated clearly shows the discretization grid in behind.

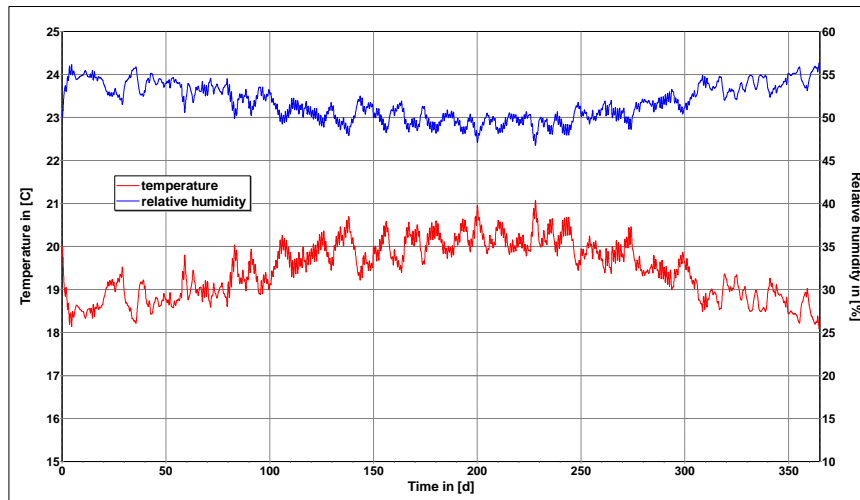


The following chart shows the same temperature field with isotherms.

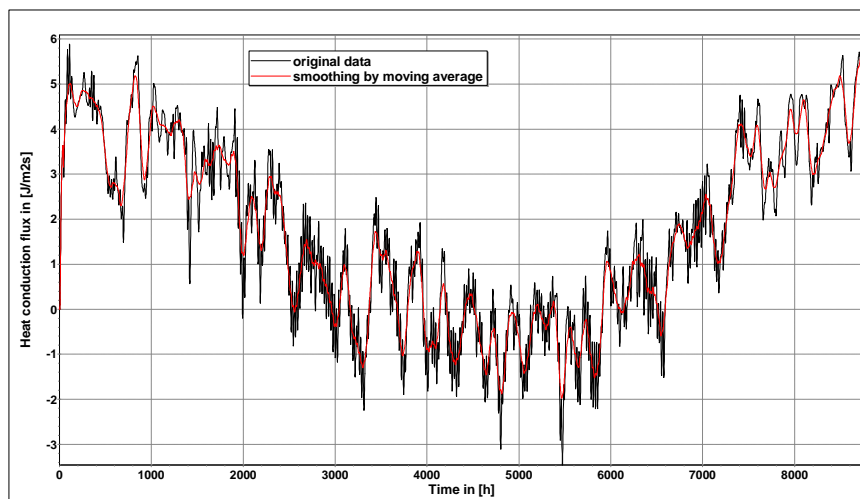


This chart kind allows to see well if the wall part is long enough in order to come into the undisturbed area. For the internal wall the undisturbed area is reached very soon whereas for the part of the outer wall should be extended.

The Delphin post-processor allows to use two y-axis with different units. The next picture shows the temperature and the relative humidity at the critical point in the corner grouped together.



In this construction no critical values are reached (80% RH). Therefore it is not necessary to use a mould model (included in post-processing, see tutorial). The last chart shows the averaged heat flux densities on the inner surfaces.



The black line represents the originally calculated values and the black line a smoothed version.

8. Conclusion

This step by step tutorial showed first steps for creating a two-dimensional construction. In the same way it is possible to create much more complex geometries with more physical effects (air flow, moisture sources etc.)

9. Additional tasks

1. Model the construction without considering symmetry and compare the results!
2. Refine the discretization and compare!
3. Add an inside insulation!

... End 2nd tutorial ...