Wall heating with the pipe model with DELPHIN 6.1

Heiko Fechner

Table of contents

1. Overview

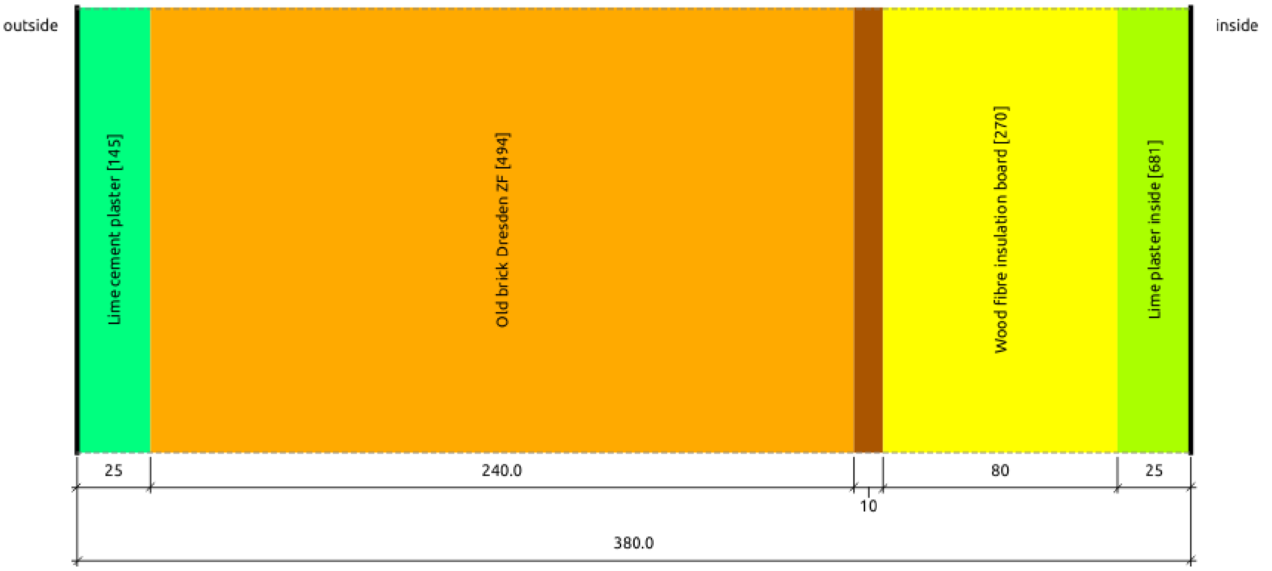
This tutorial is intended to show the basic use of the pipe model in DELPHIN using the example of a wall heating system. This tutorial applies to DELPHIN version 6.1.7. The basic construction consists of an internally insulated brick wall. A wall heating system is now to be integrated into the interior plaster. The construction is done step by step:

- Creation of the basic construction in 1D with the project wizard
- Extension to a 2D construction with a heating pipe
  - Constant mass flow
  - Time-dependent mass flow
- Extension to two parallel heating pipes

1.1. Basic construction

A simple brick wall with internal insulation and 2.5 cm thick internal plaster is to be created. The project wizard can be used for this, as described in Tutorial 1. The result should look like the picture below.

Construction sketch



Interfaces (boundary conditions)

Side	Position	Name	Type
left	outside	Outside	engineering outdoor
right	inside	Inside	engineering indoor

Location settings

Country: DE: Deutschland |EN: Germany

Town: DE: Potsdam |EN: Potsdam

Remarks: Nordostdeutsches Tiefland Mecklenburg-Vorpommern (ohne Küstenbereich); Altmark (mit Wendland); Bundesländer Brandenburg (m Fläming und Niederlausitz) und Berlin; mittlere Elbniederungen bis Dres einschl. Dresdner Elbtal; Magdeburger und Obersächsische Börden; nordöstliches und östliches Harzvorland; Goldene Aue; Sächsische Tieflandsbucht

Figure 1. Representation of the finished base construction (from input report)

Materials

	Name	λ	ρ	μ	Aw	w80	wsat
		W/mK	kg/m^3	-	kg/m^{2}s^{05}	kg/m^3	kg/m^3
	Lime cement plaster [145]	0.5500	1270.0	12.0	0.00930	59.6	450.0
	Old brick Dresden ZF [494]	1.0123	1975.7	41.1	0.01678	7.4	169.4
	Glue mortar [682]	0.4083	1252.2	14.9	0.00831	37.1	342.9
	Wood fibre insulation board [270]	0.0420	150.0	3.0	0.07000	17.6	600.0
	Lime plaster inside [681]	0.3773	1245.3	8.4	0.04544	54.2	283.4

Figure 2. Materials of the finished base construction (from input report)

Now a wall heating system is to be integrated into this construction. There are several possibilities in DELPHIN:

1. Boundary condition with specified temperature
2. Simple energy source
3. Pipe model

For cases 1 and 3, the simulation must be carried out in 2D so that a pipe can be mapped correctly.

In the case of a boundary condition with a specified temperature, this can be specified as time-dependent, but there is no control here and there is no way to simply switch off the heating. Switching off the heating would be tantamount to an adiabatic boundary condition. Since the transition coefficient is constant for a temperature boundary condition, there is always a transition.

A simple energy source allows more possibilities. It can be controlled with respect to a sensor (output) to a setpoint and it can also be switched off. However, the source power must be pre-calculated. Such a source can also be used in a 1D calculation. In this case, however, the heat is distributed evenly over a layer.

1.2. Pipe model

DELPHIN also has a special boundary condition which represents a pipe in cross-section with fluid flowing through it in the z-direction. A more detailed description can be found here: doi.org/10.1051/e3sconf/202017212012. The pipe model requires the specification of the following parameters (constant):

- Pipe length
- Pipe wall thickness

- Outer pipe diameter
- Thermal conductivity of the pipe wall
- Heat capacity or type of fluid
- Fraction of the pipe to be used (for pipes in symmetry planes)

Climate conditions are also required (time-dependent):

- Supply temperature or heat load
- Mass flow
- Internal heat transfer coefficient (not for adaptive model)

There are two usage types in DELPHIN 6.1.7:

- specified flow temperature
- specified heat load

In the following, the model with the flow temperature is always used. The model then calculates the return temperature and heat losses for each time step. These can also be defined as output. There are two subtypes of this model:

- Transition coefficient and heat capacity of the fluid specified
- Specified fluid type

In the first variant, the heat transfer coefficient from the fluid to the inner pipe wall must be specified as a climate. The specific heat capacity of the fluid is also required as a parameter. In the second variant, the type of fluid is specified. The other necessary parameters are then calculated from this in DELPHIN. For the fluid these are:

- Density
- Heat capacity
- Dynamic and kinematic viscosity
- Thermal conductivity

All these data are dependent on the temperature. These data are then used to determine whether the flow of the fluid for the given pipe is laminar or turbulent at the current flow velocity. Based on this, the heat transfer coefficient is then determined. This is why the model is called an 'adaptive model'. The following fluids are currently available:

- Pure water
- Water with ethylene glycol
- Water with propylene glycol

If water with added glycol is selected, the mixing ratio can also be specified. The following image shows the dialog with the adaptive model selected for pure water.

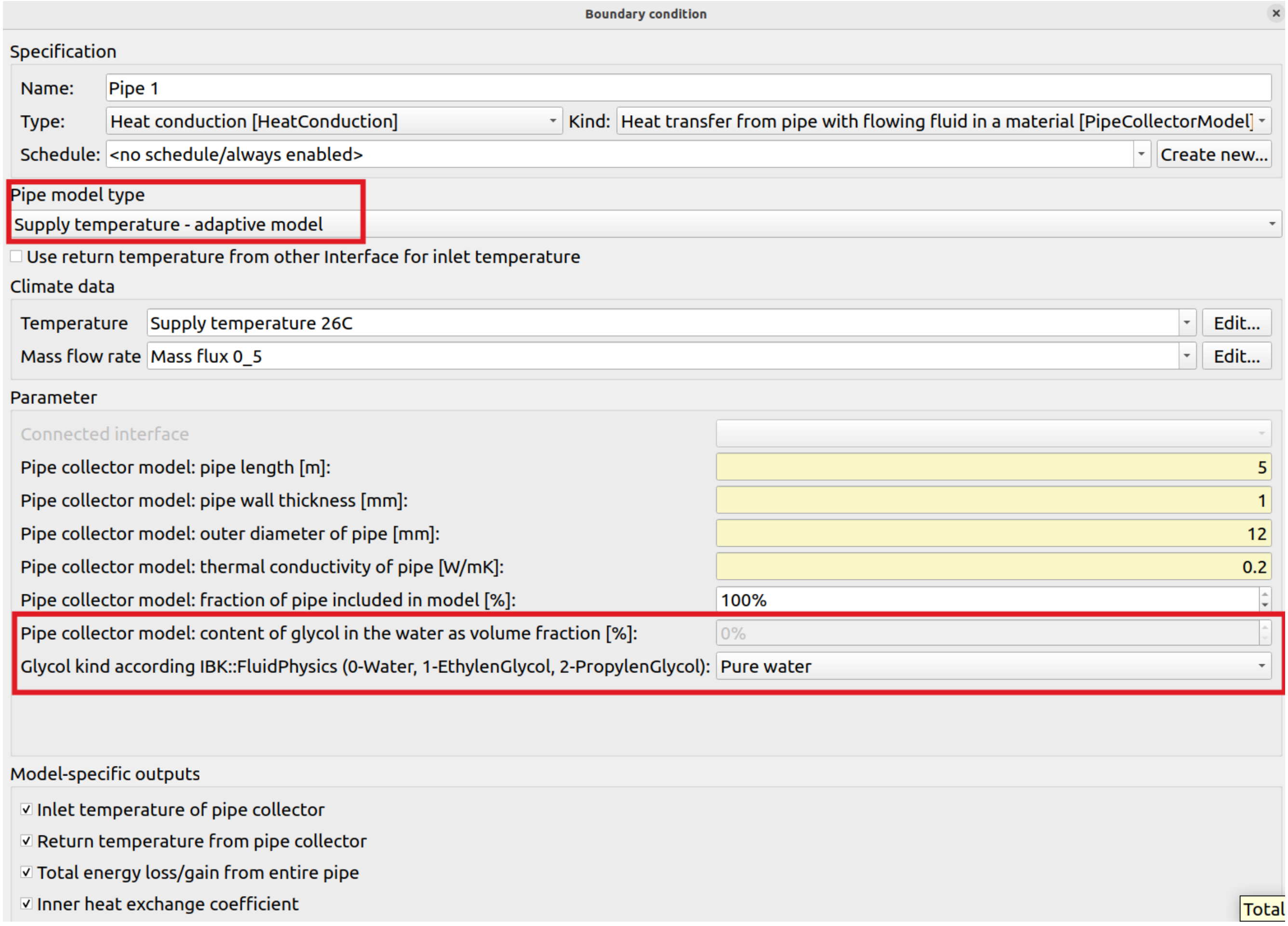


Figure 3. Boundary condition for the pipe model with adaptive model

This boundary condition must then be assigned to a surface. This surface is then assigned to the geometry.

Important A separate interface with its own boundary condition is required for each pipe. The pipe boundary conditions must not be used more than once!

2. Create a 2D structure

2.1. Creating or adapting the geometry

First, a pipe is to be integrated into the interior plaster. As the pipe model is a boundary condition, the edges must be created first. The following image shows the positioning of the pipe.

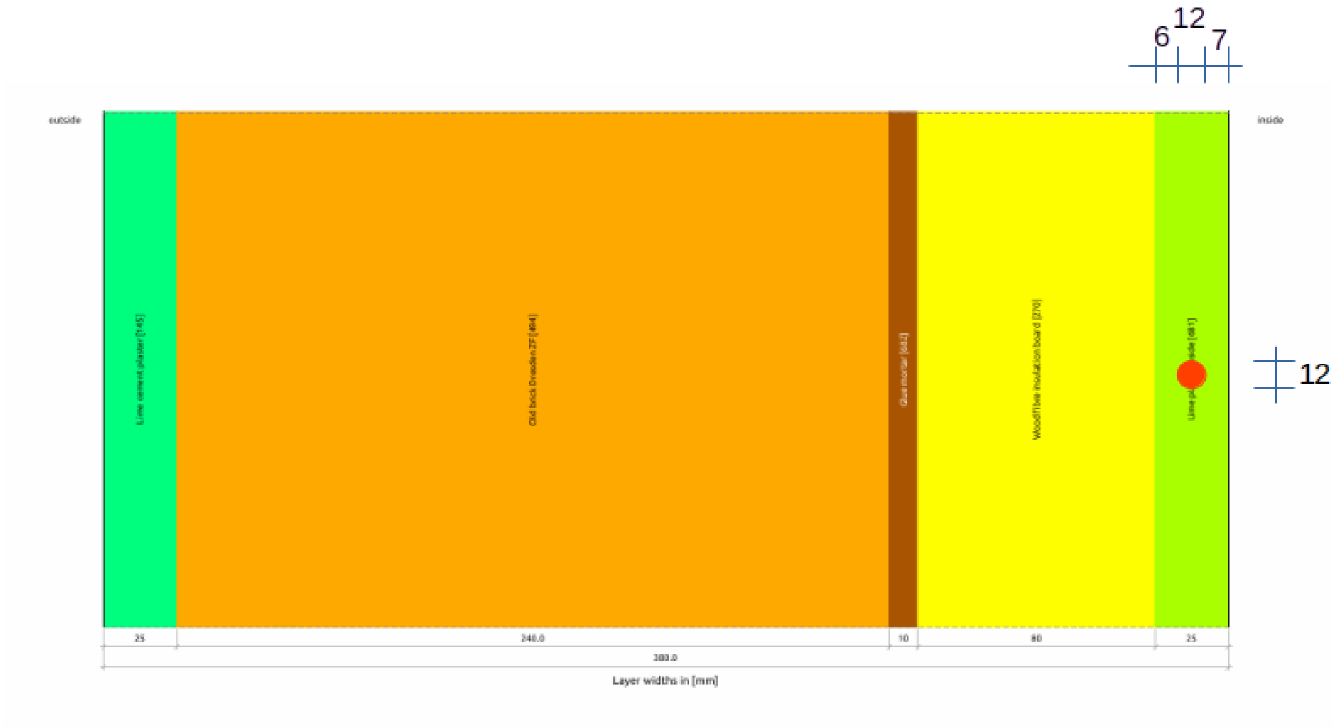


Figure 4. Construction with position of the pipe

The cut-out for the pipe should be approximately the same size as the pipe itself. However, an exact match of circumference or area is not necessary. The heat transfer is always calculated for the round pipe and then divided into the volume elements. The following figure shows the division.

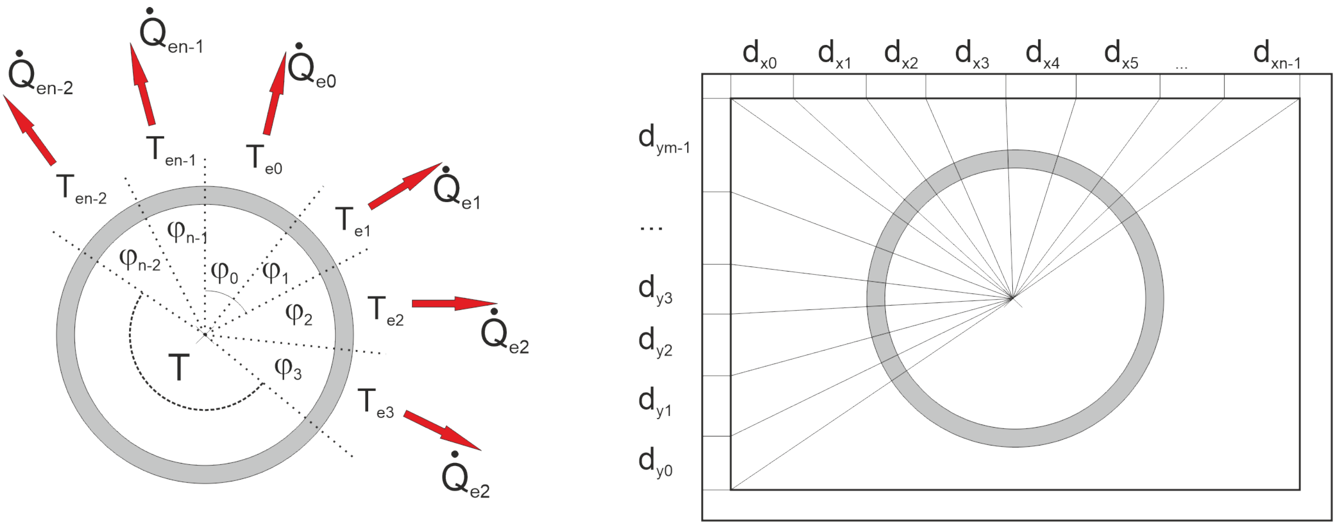


Figure 5. Distribution of the heat transfer from the pipe to the structure

The pipe should have an outer diameter of 12mm and the installation distance should be 6cm. The following changes must therefore be made to the geometry:

- Add two rows
- Change the row heights to 30mm, 12mm and 30mm

- Division of the inner plaster layer into 3 columns with thicknesses of 6mm, 12mm and 7mm

First save the 1D project under a new name. Then you should remove the existing discretization.

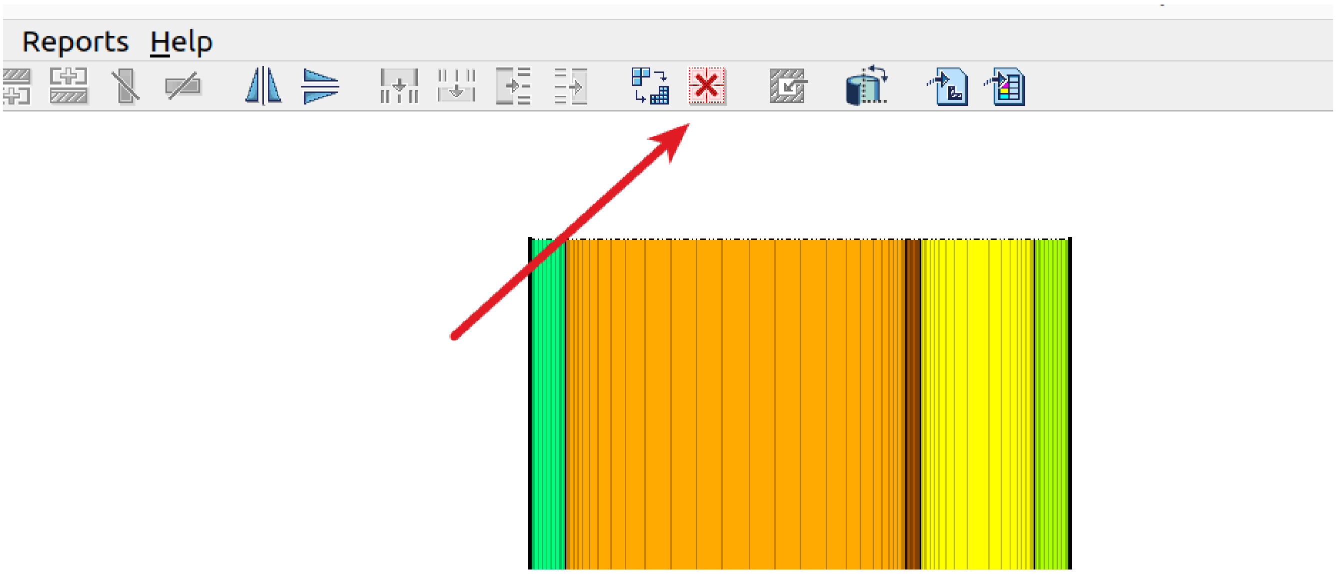


Figure 6. Remove the existing discretisation

It may be that some layers are split into two layers after removal. This is not a problem and can remain so. DELPHIN tries to keep the positioning of edge outputs and therefore creates additional columns.

Rows are added as follows:

- Select any cell in the 1D construction (1)
- Click on the button to add rows - dialog opens
- Enter the new row height (3) and close the dialog (2)
- Repeat the process for the next row

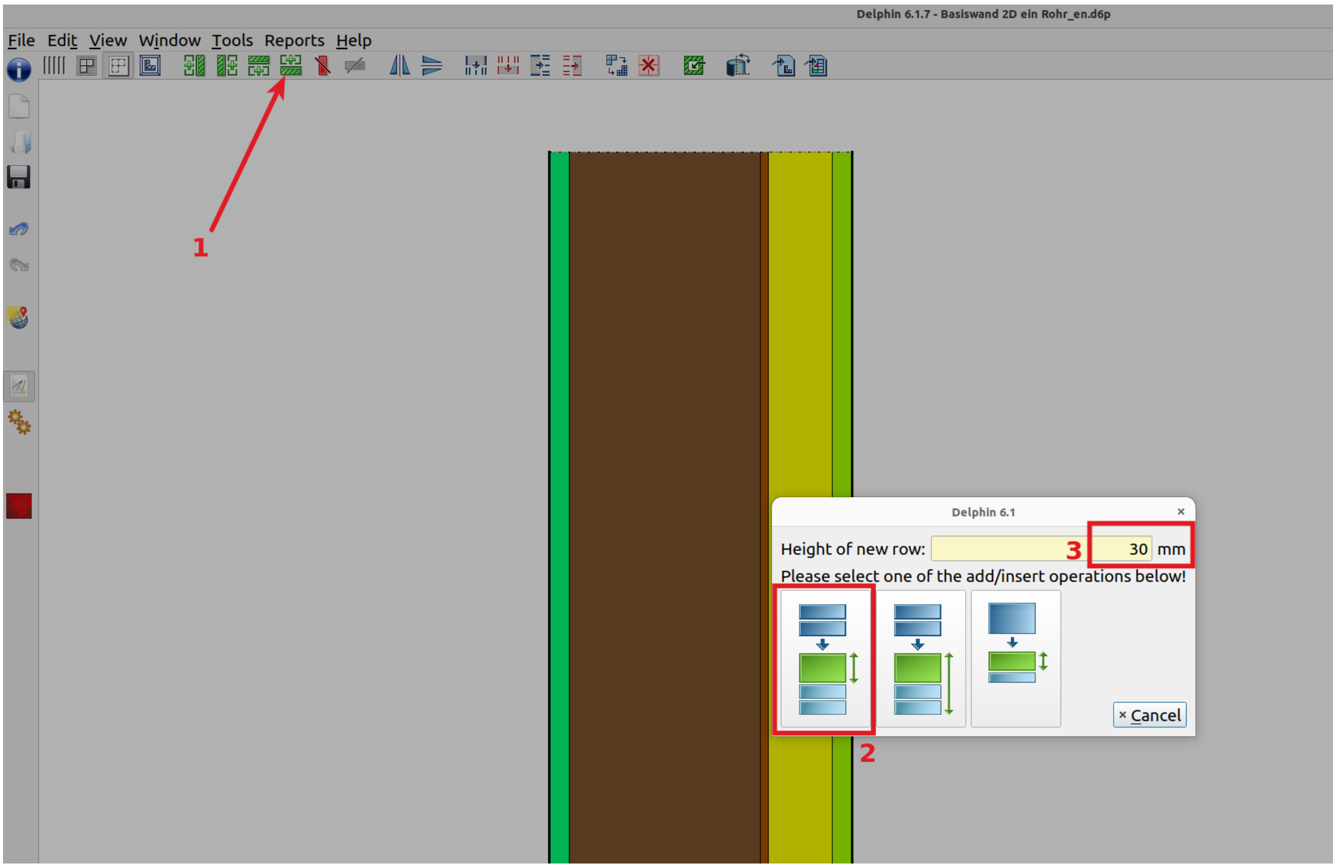


Figure 7. Add a line to the construction

The row height can be adjusted by selecting the respective row (one element in the row is sufficient) and then adjusting the height at the bottom left.

Figure 8. Adjusting the height of a row

Now that the construction has 3 rows of the appropriate height and the inner plaster layer has been divided, you need to create space for the pipe. To do this, mark the element where the pipe is to be positioned and click on the button to remove the material. The following two images show the process.

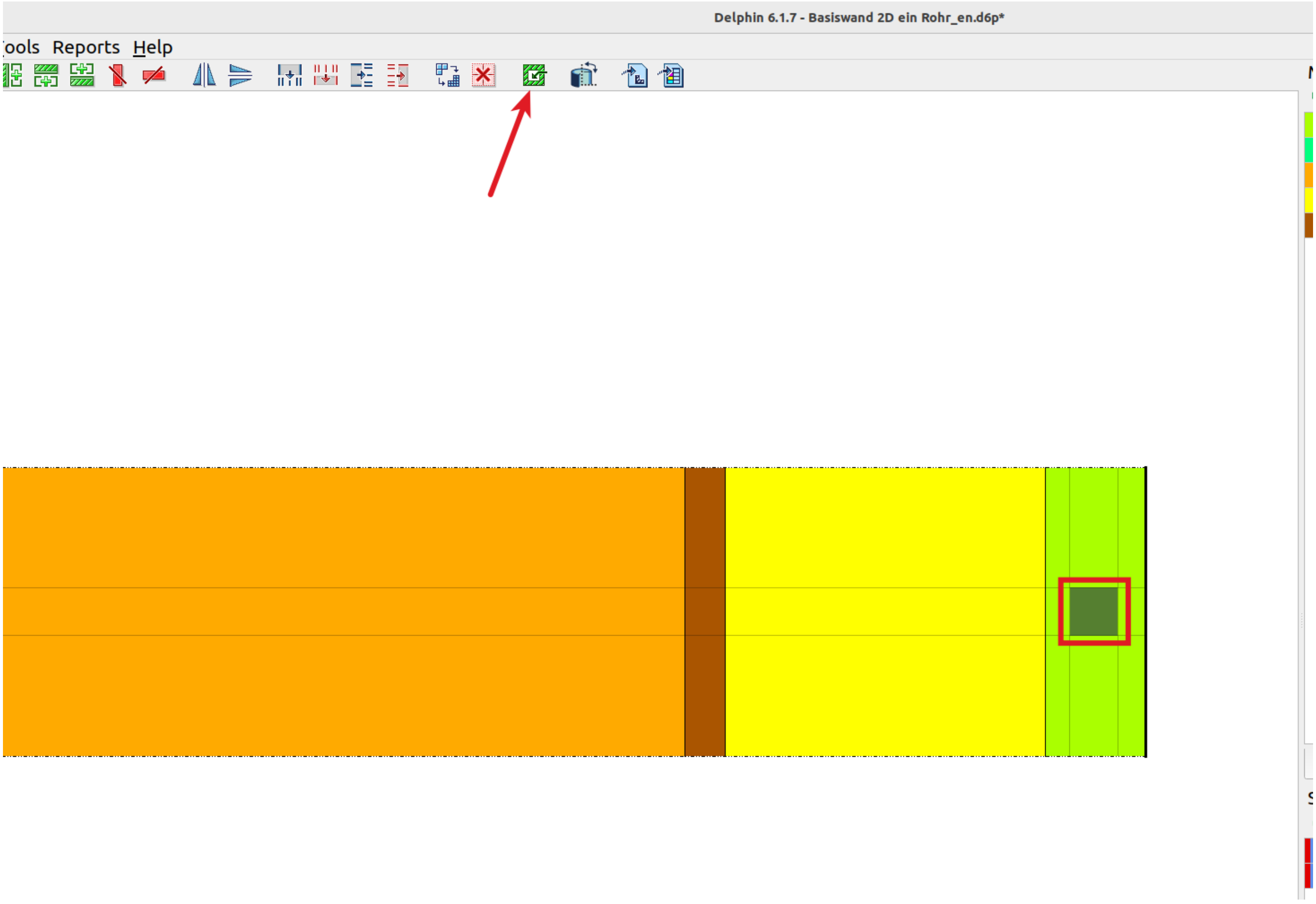


Figure 9. Removing the material assignment from an element

Figure 10. Construction with removed material assignment

2.2. Creating and assigning the pipe model as a boundary condition

This element now has edges and therefore boundary conditions can be assigned there. First, a boundary condition and a surface must be created for the pipe model. We start by creating the boundary condition by clicking on the button with the green plus in the *Boundary conditions* area. In the dialog that then appears, we select *Heat pipe* as the type and *Heat exchange to a pipe with flow through* as the type. For 'Type of pipe model', select 'Inlet temperature - defined parameters'. The following values should be used for the climate conditions and parameters.

- Temperature - constant 26°C
- Mass flow - constant 0.039kg/s (corresponds to v=0.5m/s for this pipe)
- Heat transfer coefficient - constant 200W/m^{2}K (laminar flow)
- Pipe length - 5m
- Pipe wall thickness - 1mm
- Outside pipe diameter - 12mm
- Thermal conductivity of the pipe wall - 0.2W/mk (PE)
- Heat capacity of the fluid - 4180J/kgK (water)
- Fraction - 100% (pipe is completely in the material)

Under *Pipe model type* the entry *Supply temperature - defined parameters* should be selected. The climate data can be entered via new climate conditions to be created. The outputs at the bottom left of the dialog can all be switched on. The dialog should then look something like this (any name for climate).

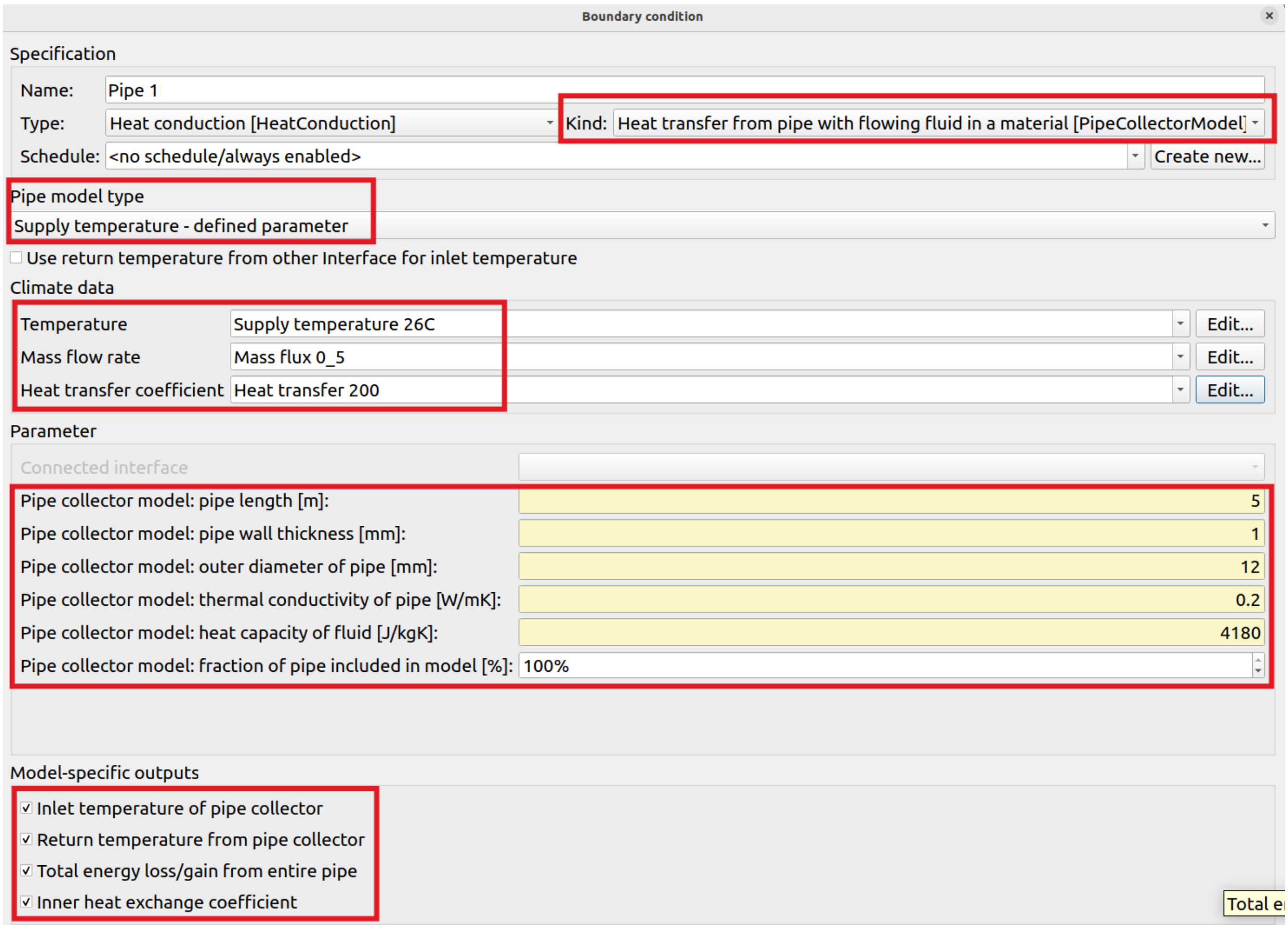


Figure 11. Boundary condition for pipe model

The following image shows an example of the mass flow as a climate condition.

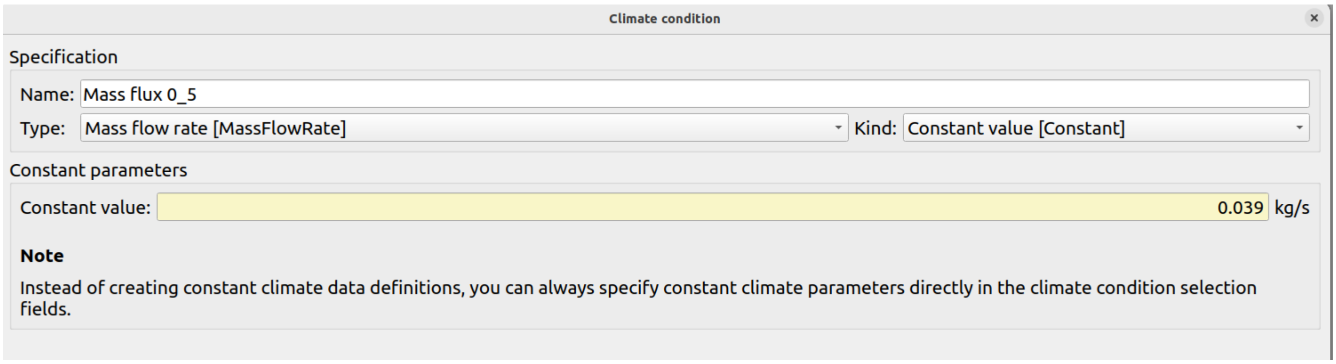


Figure 12. Climate condition for mass flow

Next, the interface for the pipe model must be generated. This is necessary because only surfaces can be assigned to the construction and not the boundary conditions directly. A surface serves as a container for boundary conditions, so to speak. To do this, we switch to the *Surfaces/Boundaries tab and click on the green plus. In the dialog that then appears, select _Detailed/scientific interface…* as the type. Alignment and inclination do not play a role here. All you have to do is enter a name and select the condition you have just created for the pipe in the boundary condition list.

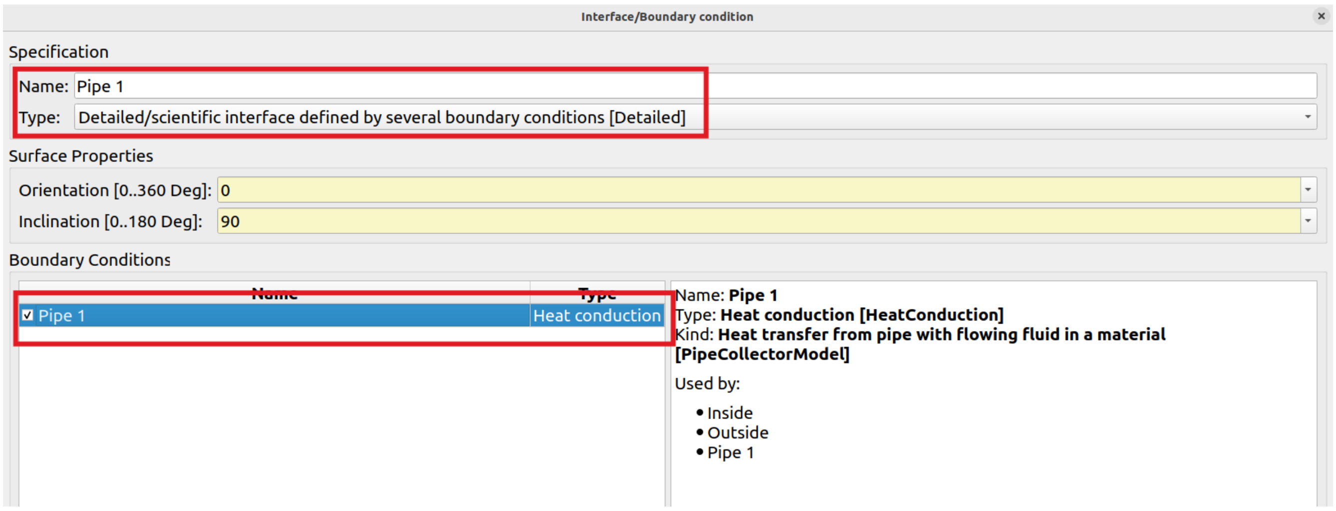


Figure 13. Dialog for interface for the pipe boundary condition

This surface must now be assigned to all 4 sides of the previously cut out area. To do this, first select the surface, then an element next to the cut-out (here on the left) and then click on the correct assignment button. After all, the side must always be assigned to the cut-out. For example, if you have selected the element to the left of it, use the green button with the arrow from the right (see image).

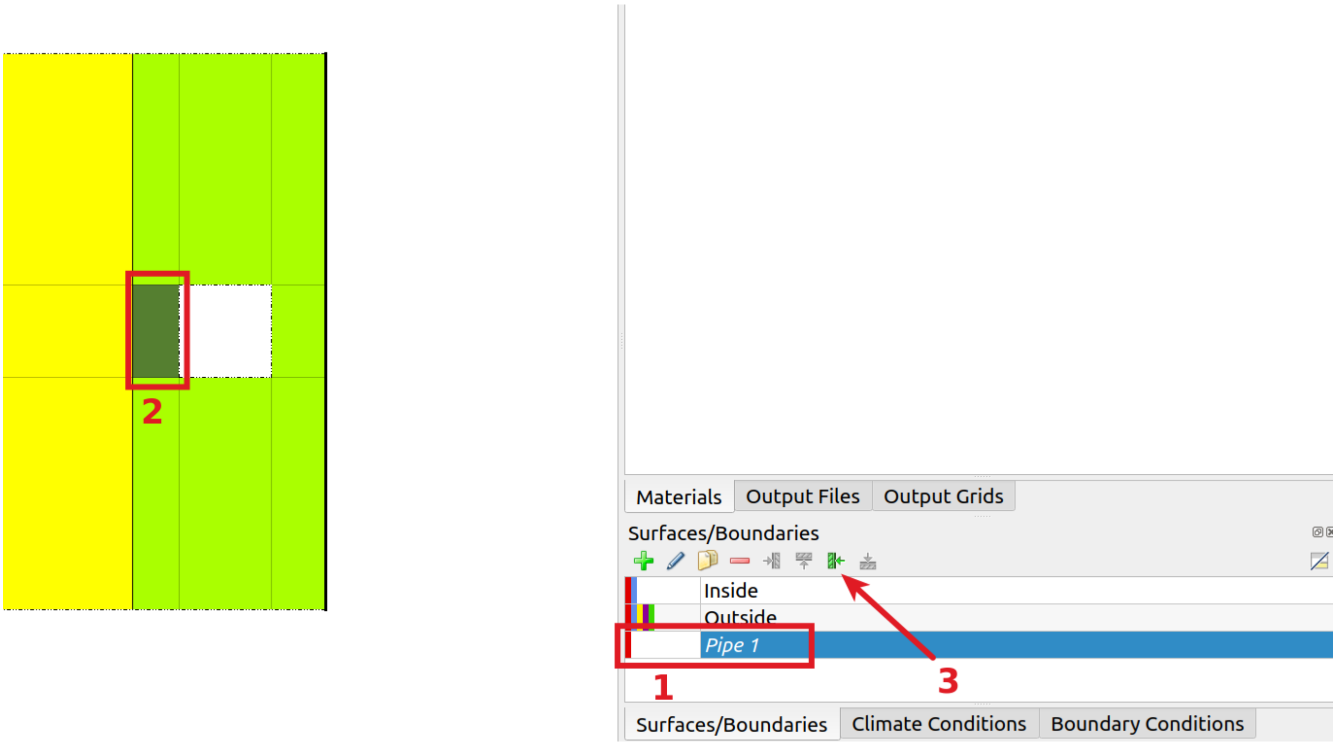


Figure 14. Assignment of the left side

You then do the same with all the other sides. When you are finished, you can click on the interface entry in the interface list to check. All 4 edges should then be marked. Furthermore, in the normal view, all 4 sides should be shown with thick solid lines.

Finally, the construction must be discretized again and you can calculate.

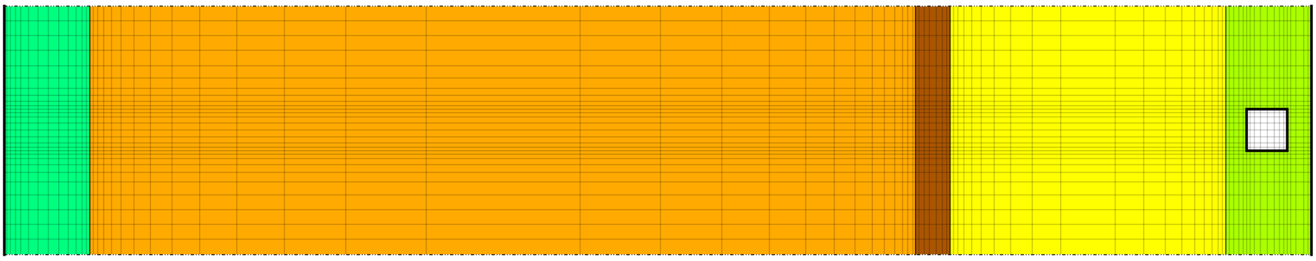


Figure 15. Finished discretized construction with pipe

2.3. Evaluation

The evaluation now only serves to check whether the wall heating works. The evaluation is carried out with the PostProc 2 software. First you can look at a temperature field on any winter day.

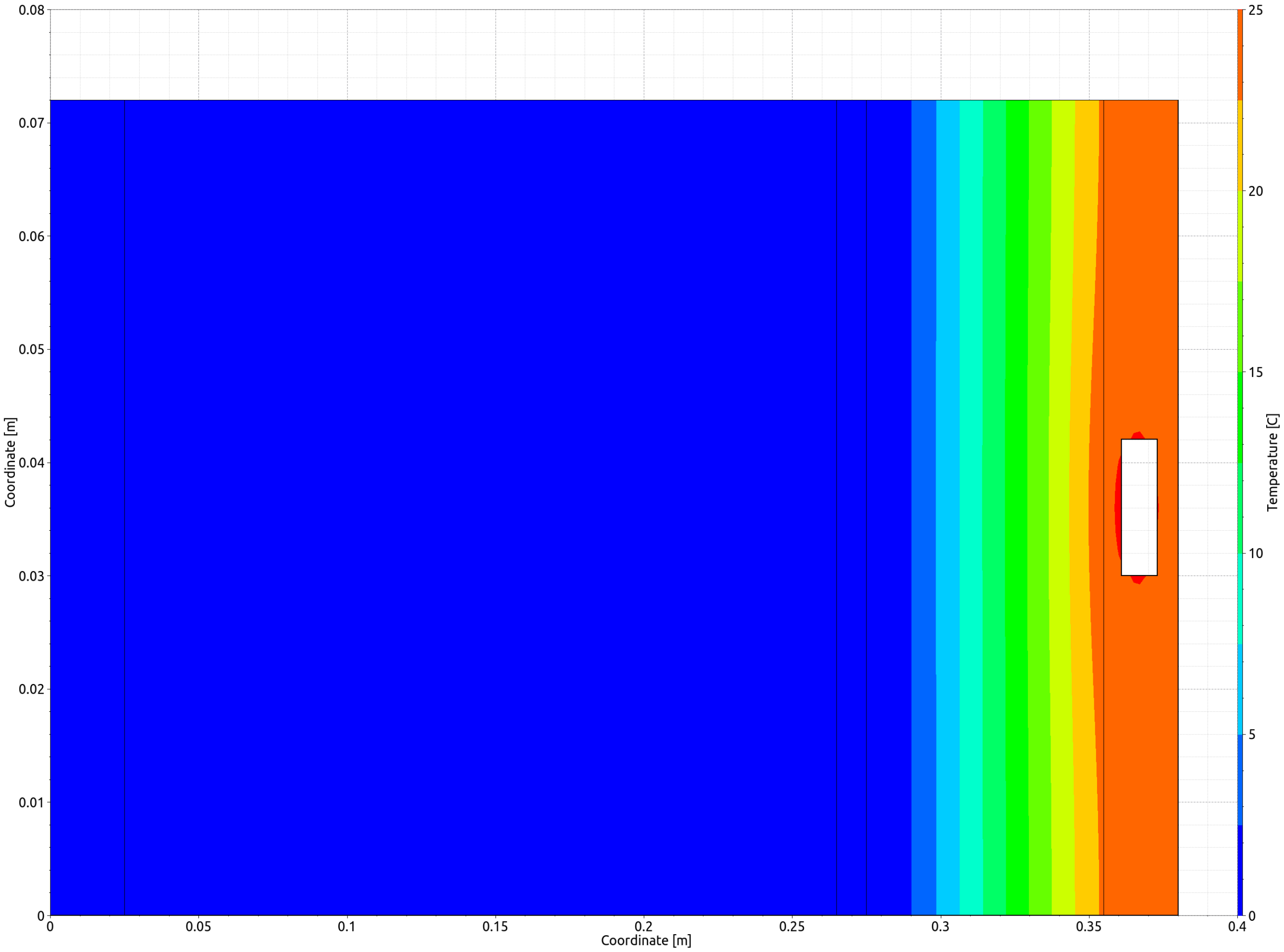


Figure 16. Temperature field on the first of December

The temperature field clearly shows the effect of the pipe heating. The next diagram shows the average indoor surface temperature for one year.

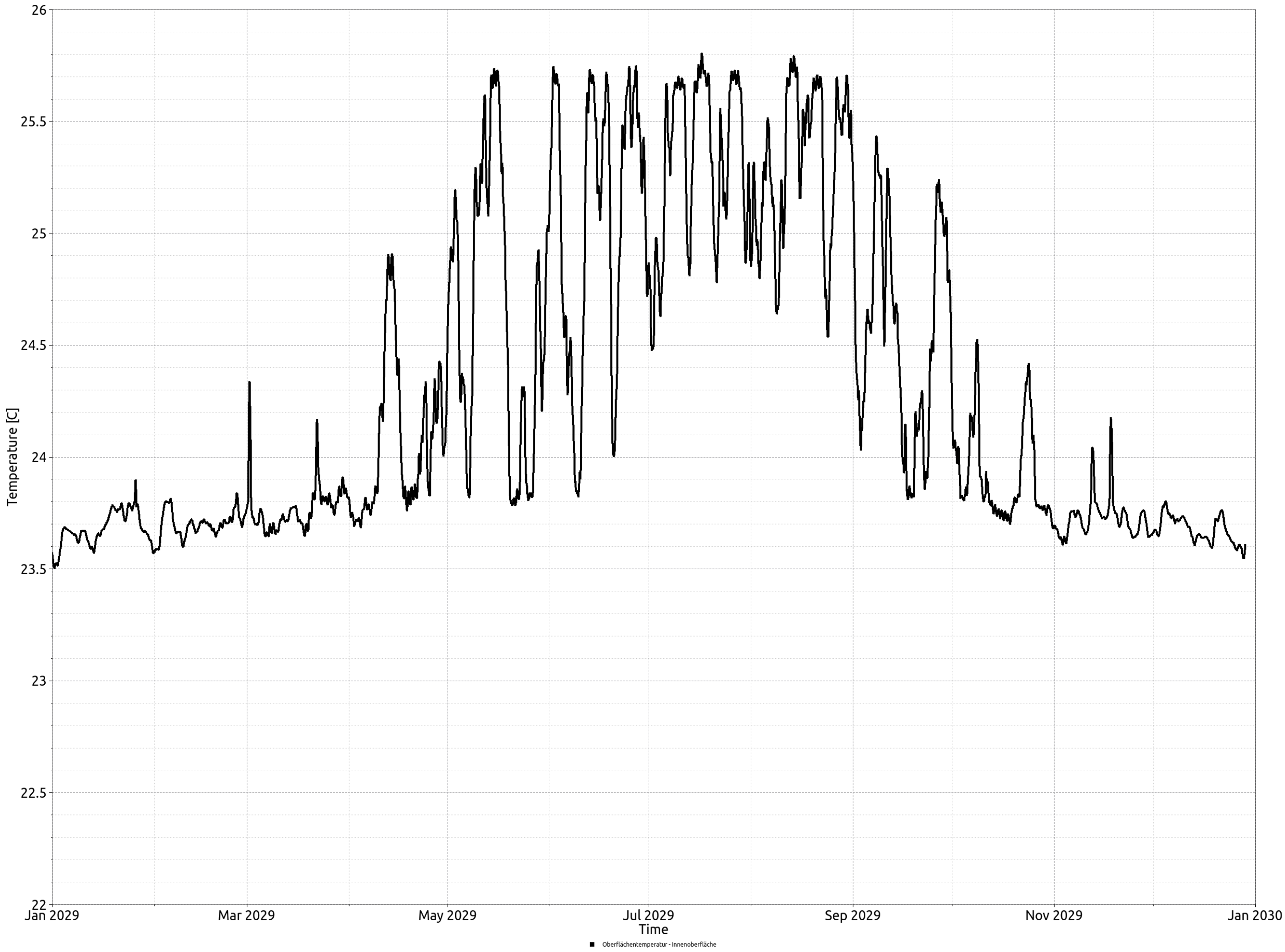


Figure 17. Average temperature on the inside surface in the last year

From spring to fall, the surface temperature is often close to 26°C, which corresponds to the supply temperature. This is now due to the fact that heating is always on, even in summer. To improve the behavior, a time-dependent mass flow can be used. A simple variant would be to set the mass flow to 0 outside the heating period. A suitable climate file must be created for this.

2.4. Creating a variant with time-dependent mass flow

First, the existing project must be saved under a new name using 'Save as…'. Then you can make all the necessary changes. The first step is to create a new climate file.

There are various formats for climate data. Descriptions of these can be found at Tutorial 6. In this case, the ccd format will be used. The structure is as follows:

```
# Comments
# Comments
Keyword Unit
dd hh:mm:ss Value
```

The keyword for the mass flow is *MassFlowRate* with the unit kg/s. As mentioned above, the default value is 0.039 kg/s. With the selected pipe cross-section and a water density of 1000kg/m^3, this value corresponds to a flow rate of 0.5m/s. For demonstration purposes, the mass flow from April 1 to October 1 should now be set to 0. The ccd format is a pure text format. A good text editor should be used to create and/or edit text files. WINDOWS in particular does not include anything usable as standard. One possibility would be the free editor Notpad++ (notepad-plus-plus.org/). It is also possible to use a spreadsheet program such as Excel. There are then extended calculation options, but no good export to text. Here you usually have to use a text editor. The following procedure:

- Create a text file
- Open in the text editor
- Enter keyword and unit
- Enter any number of time/value pairs
    - Always use the . as decimal separator
    - Times should always go from day 0 1:00:00 (January 1) to day 365 0:00:00 (December 31 24:00)

The file could look like this:

```
# Mass flow for heating
# Heating only January to the end of March and from October 1

MassFlowRate kg/s

0 01:00:00 0.039
89 23:00:00 0.039
90 00:00:00 0.0
272 23:00:00 0.0
273 00:00:00 0.039
365 00:00:00 0.039
```

Day no. 90 corresponds to April 1 and day no. 273 to October 1. One hour is assumed in each case for the increase and decrease of the mass flow. This text file now has the extension ccd and is saved in a folder with the name *climate* in the same folder as the project file. Then go back to DELPHIN and create a new climate condition. To do this, first click on the button with the green plus in the Climate conditions tab. In the dialog that appears, you still have to set the type. In this case, this would be *Mass flow* and *Data points*. Then select the climate file by clicking on the small button with the 3 dots. If the climate file is correct, the data will be displayed in the diagram; if not, an error message will appear. Finally, give the new climate condition a meaningful name and close the dialog.

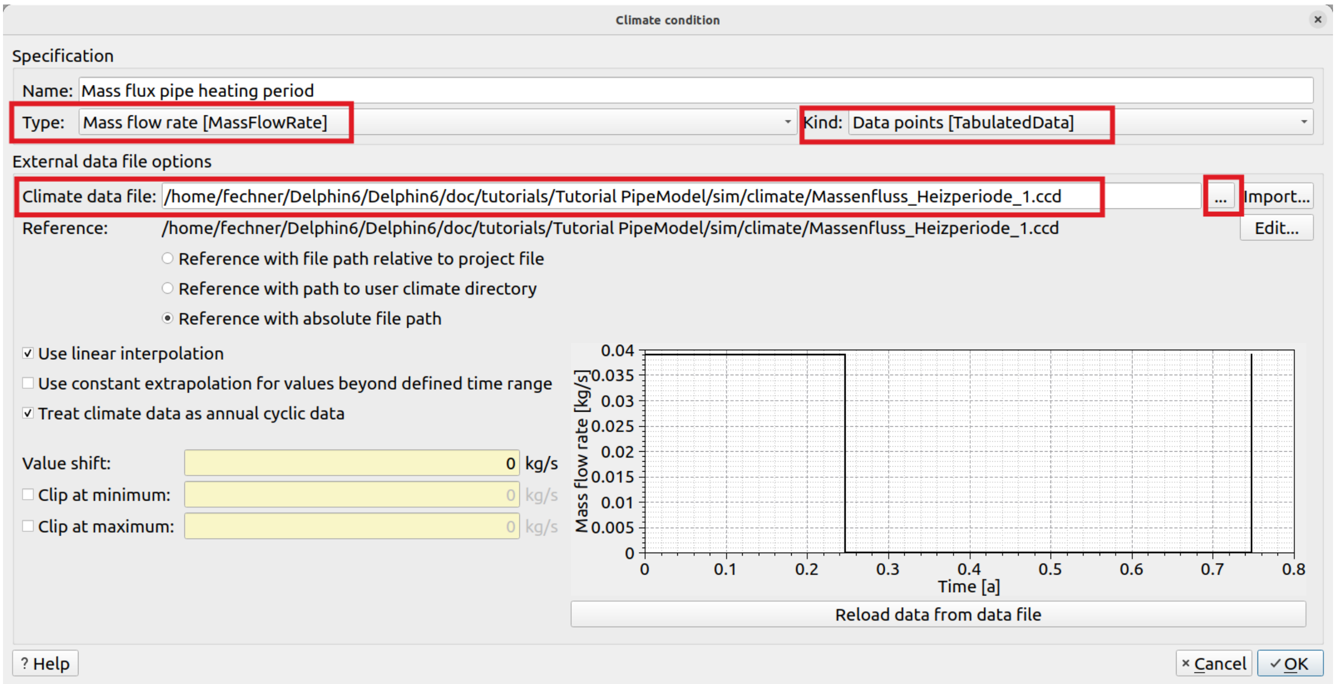


Figure 18. Climate condition with selected file

Now only this climate condition has to be used in the boundary condition for the pipe model. You can adapt the existing boundary condition or create a new one and make the change there. The second option should be selected here so that all adjustments in the project and variants remain clearly visible. To do this, copy the existing boundary condition by first selecting the entry to be copied and then clicking on the *Copy* button.

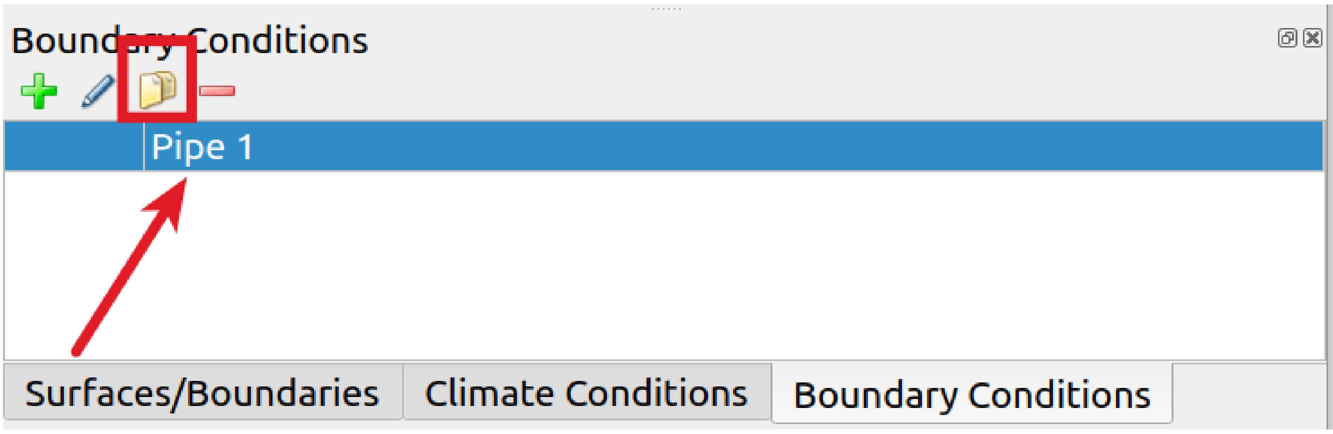


Figure 19. Copying a boundary condition

After the copying process, the dialog for the new boundary condition opens automatically. Among other things, the selected climate conditions can be seen there. To use the new climate for the mass flow, click on the list selection (small triangle on the right) and select the appropriate climate condition.

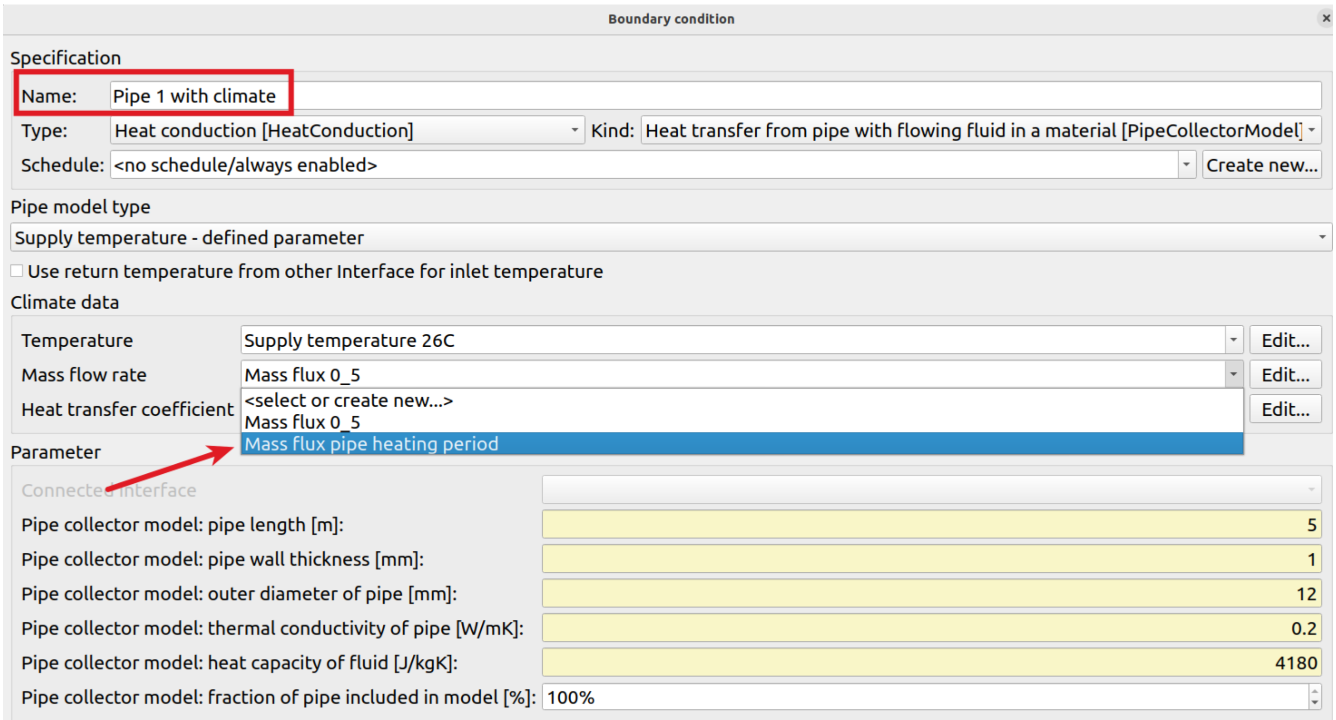


Figure 20. Selection of a new climate condition

Finally, adjust the name.

To ensure that the new boundary condition is also used, it must be selected in the assigned surface. Here too, you can create a new surface or adapt the old one. In this case, it is easier to adapt the existing surface because then the assignments do not have to be changed.

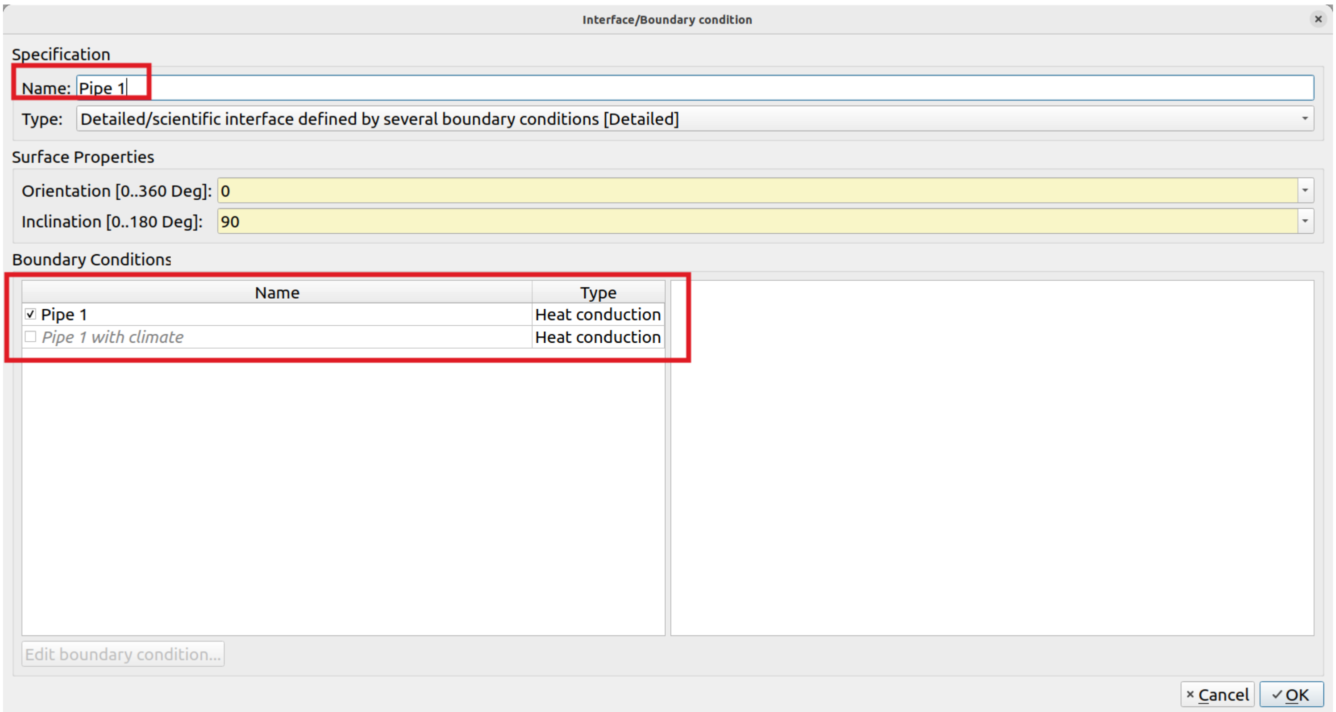


Figure 21. Customizing the interface for the pipe

The image above shows the existing interface for the pipe. The following must now be done here:

- Deactivate boundary condition 'Pipe 1'
- Activate boundary condition 'Pipe 1 with climate'
- Rename the surface

The result must then look like this:

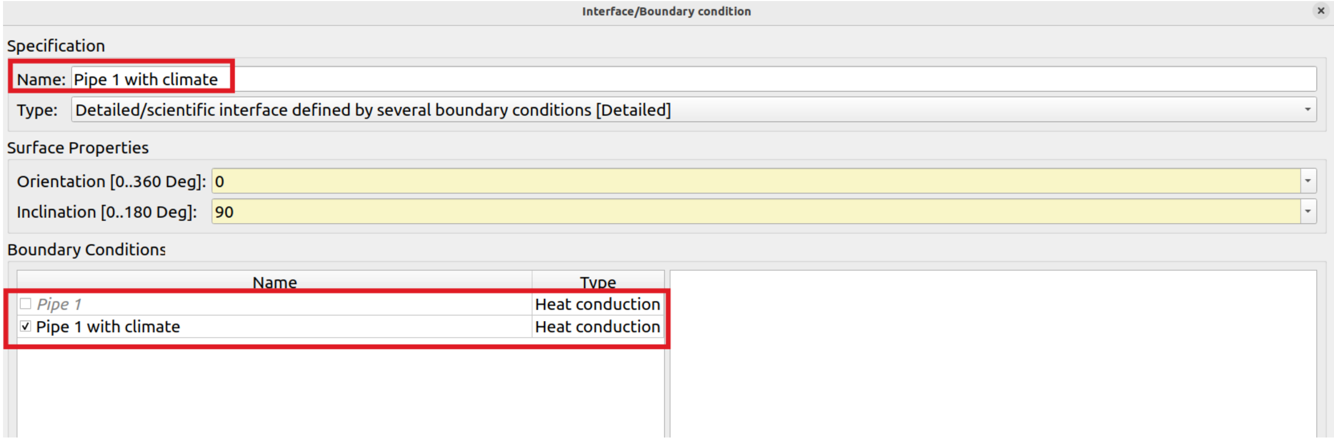


Figure 22. Interface with new boundary condition

That’s all and the new variant can be calculated. The following diagram shows a comparison of the surface temperatures in one year.

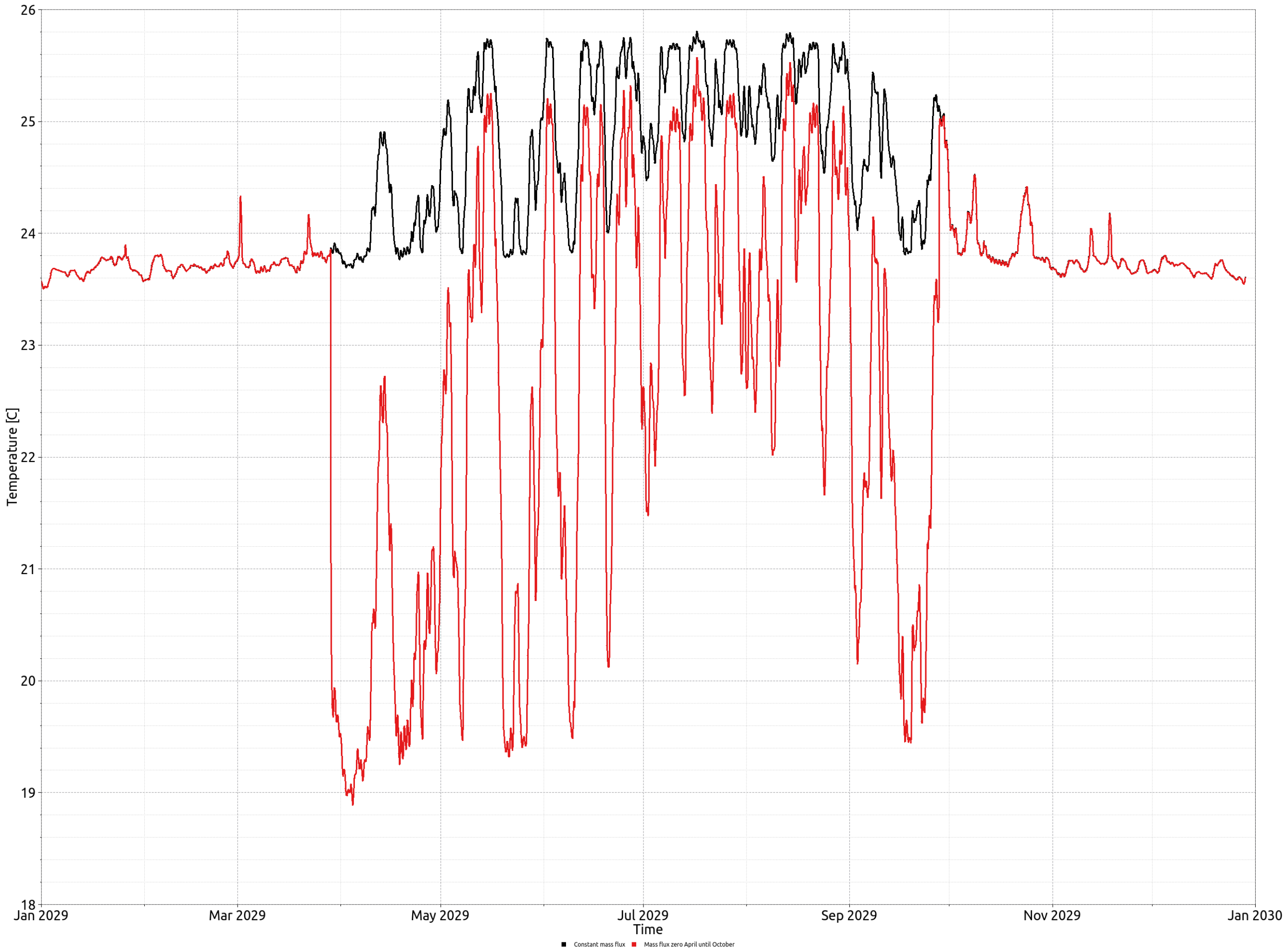


Figure 23. Surface temperature inside with and without mass flow adjustment

2.5. Variant with two pipes

In the first variant, only one pipe was used and the construction was cut at the top and bottom in the middle between the pipes. This forms a wall in which any number of pipes lie at the same distance and all have the same flow temperature (collector).

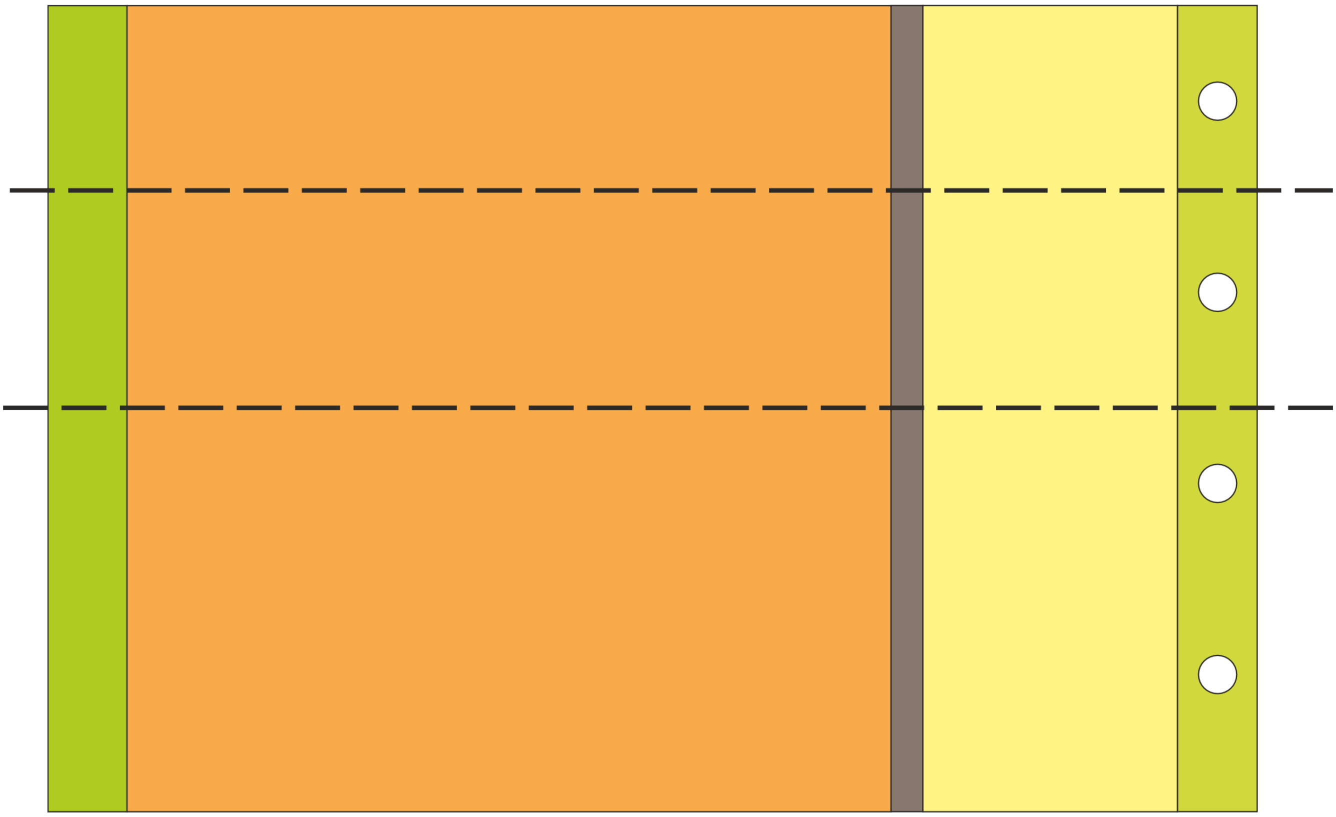


Figure 24. Wall construction with wall heating and cutting planes for a pipe

Now a system is to be calculated in which the pipes lie as a loop in the wall. This means that only one pipe is operated directly with the supply temperature and the next with the return temperature of the first pipe. The system will be demonstrated using the example of a pipe loop with two pipes. In order to be able to map two pipes, the sectional planes are now placed so that they pass through the pipes in the middle. The following illustration shows the principle.

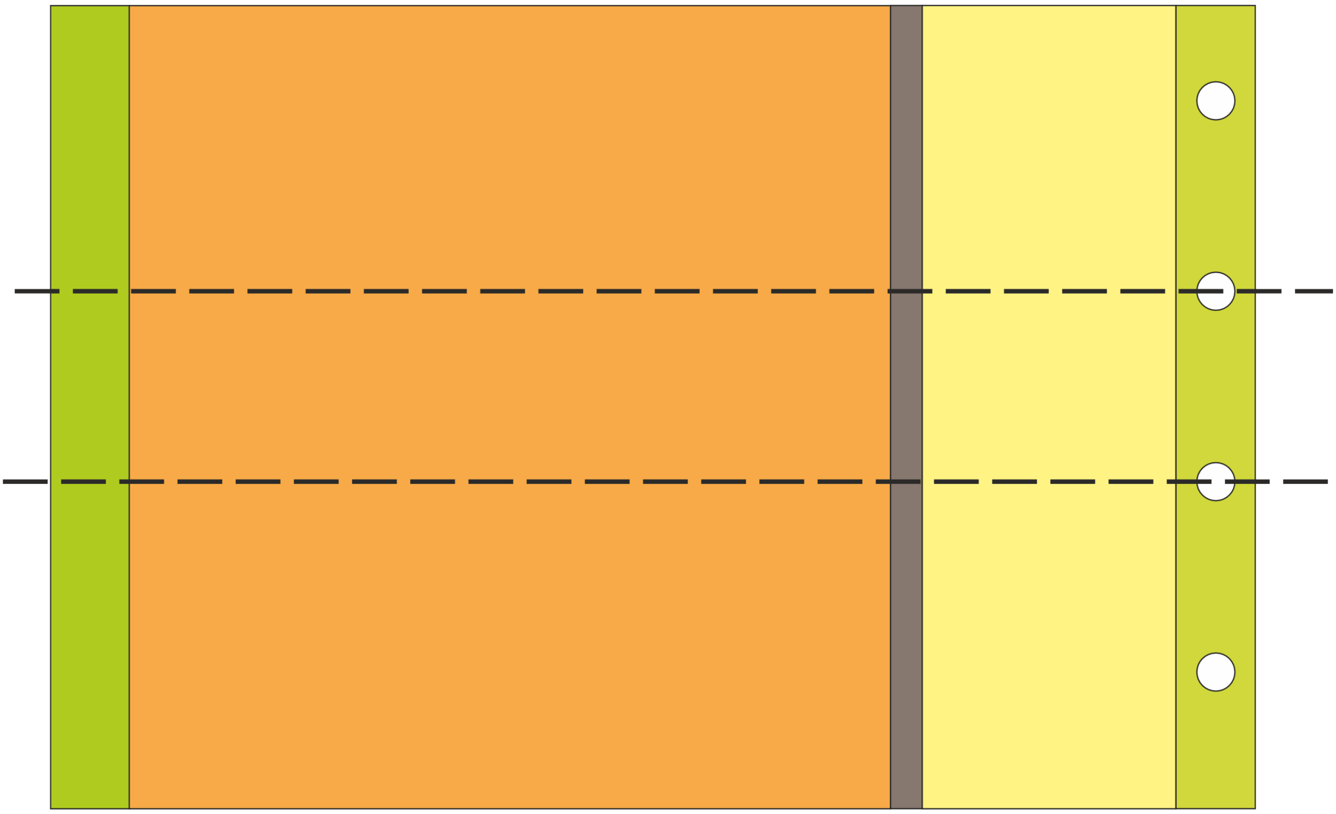


Figure 25. Wall construction with wall heating and cutting planes for two pipes

The conversion process is as follows:

- Save the project under a new name
- Remove the discretization
- Conversion of the geometry
  - Delete the top line
  - Change the height of the top row to half the pipe diameter (6mm)
  - Add a new line with a height of 6mm at the bottom
  - Remove the material for the bottom pipe

The following image shows the geometry after the conversion

Figure 26. Basic geometry after conversion to two tubes

The upper pipe still has the assignment of the surfaces. The height of this line has been halved compared to the original construction because the section plane runs through the pipe. This must now be noted in the boundary condition. To do this, open the boundary condition and change the value for the pipe fraction from 100 to 50%.

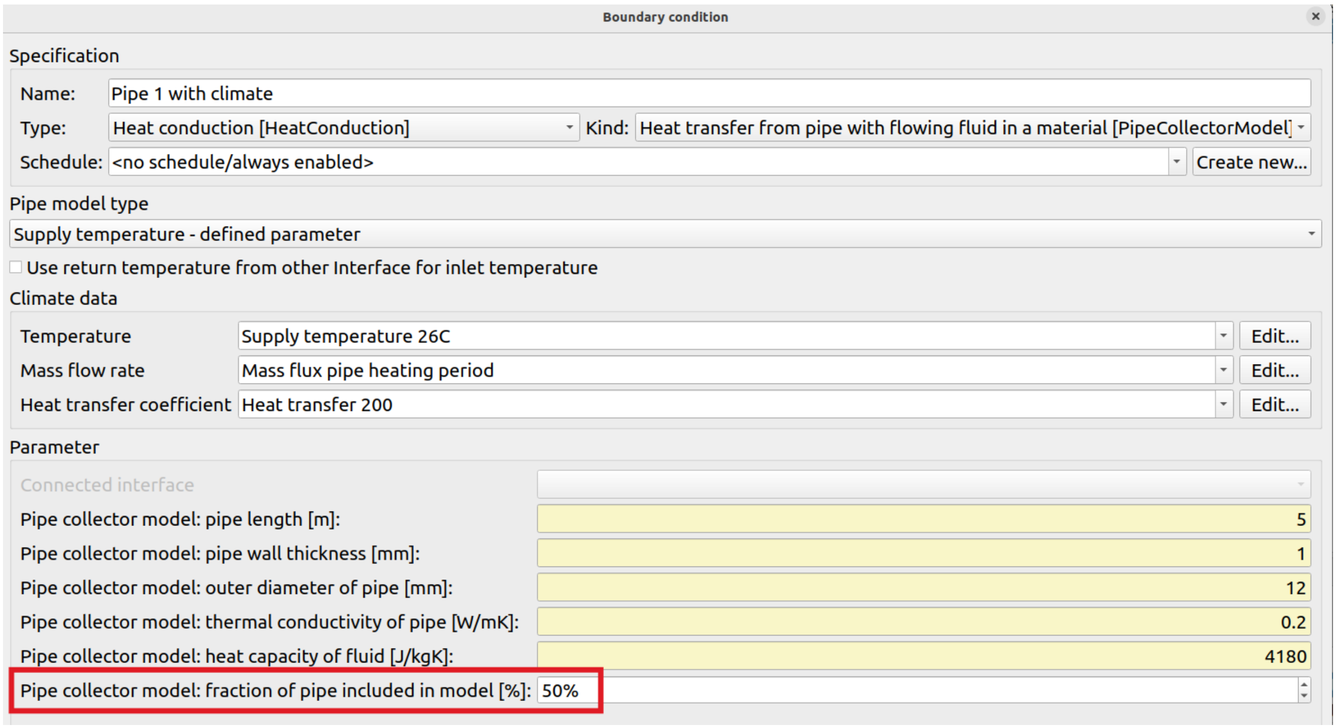


Figure 27. Dialog for boundary condition pipe 1 with 50% share

All other parameters can remain as they are. A new boundary condition and a new surface must now be created for the second pipe. The easiest way to do this is to copy and adapt the existing entries. Only the name of the boundary condition should be changed first. The setting for the pipe coupling is made later. A second surface must first exist for this. Here too, the existing surface is copied and the name adapted. The newly created boundary condition is then selected. The result is shown in the next screen.

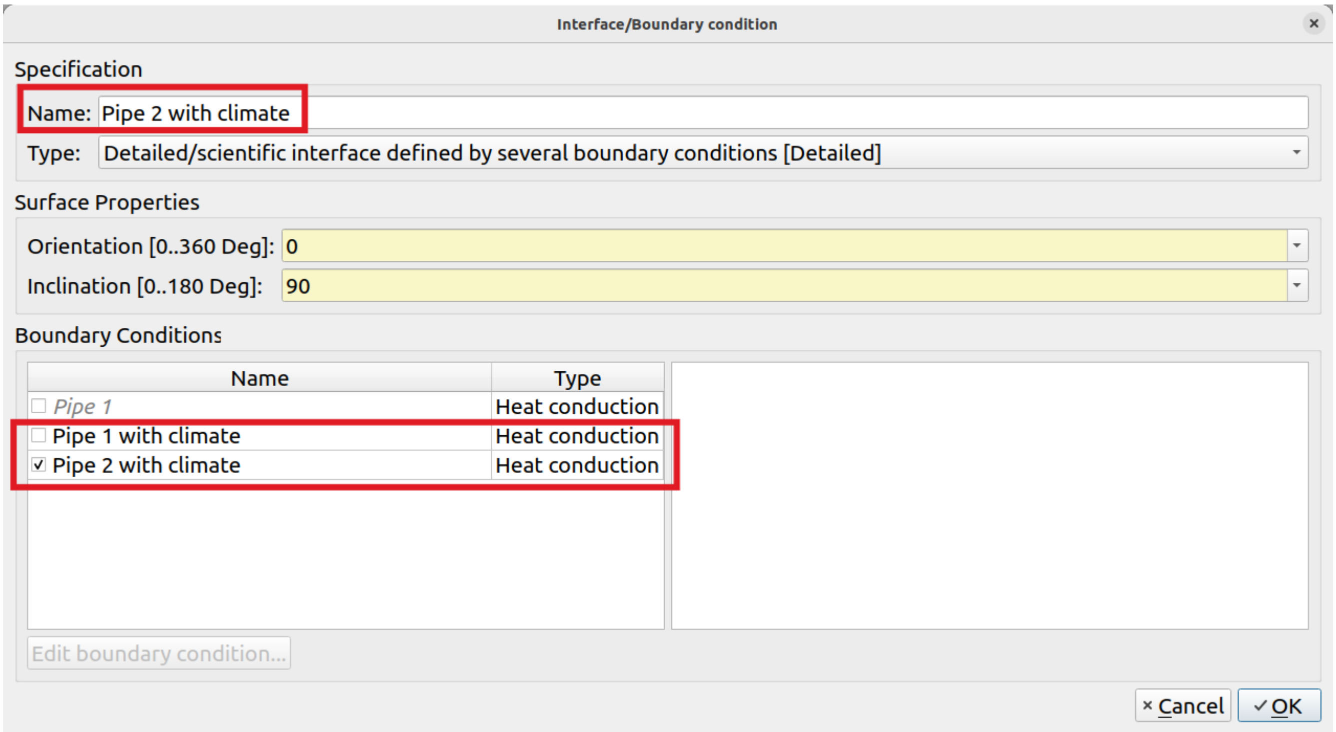


Figure 28. Interface for pipe 2

Now go back to the boundary conditions and open the entry for pipe 2. For the coupling, you must specify that a coupling exists and which surface the pipe is to be connected to.

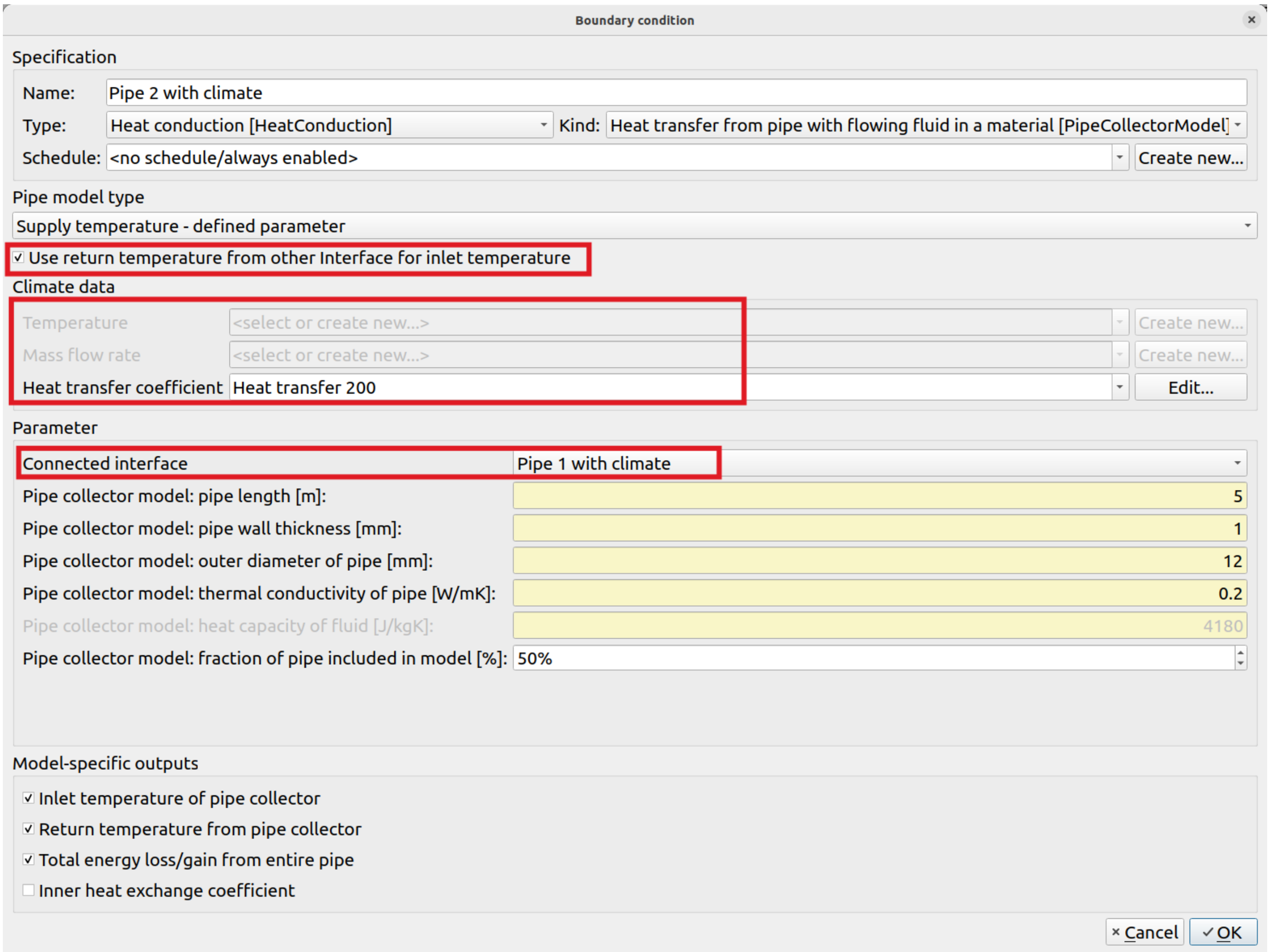


Figure 29. Boundary condition with coupling to the surface of pipe 1

As you can see in the image above, the following things have been changed:

- Checked the box 'Use the return temperature from other interface ...'
- The surface of Pipe 1 was selected in the 'Connected interface' selection box.

When selecting the interface to be connected, only those that can be connected are displayed. Your own surface and others for which a connection already exists are not offered for selection. Furthermore, the climate conditions for temperature and mass flow are deactivated because both are taken over by the connected boundary condition. The parameters should also not be changed. You can only link one pipe to another. Branching is not permitted.

Once the new boundary condition has been parameterized, you can assign the containing surface to the construction and discretize the construction again. Then, as always after a new discretization, the assignments must be checked. This usually only affects the outputs. In this construction, the outputs for temperature and relative humidity on the inner surface were not correct.

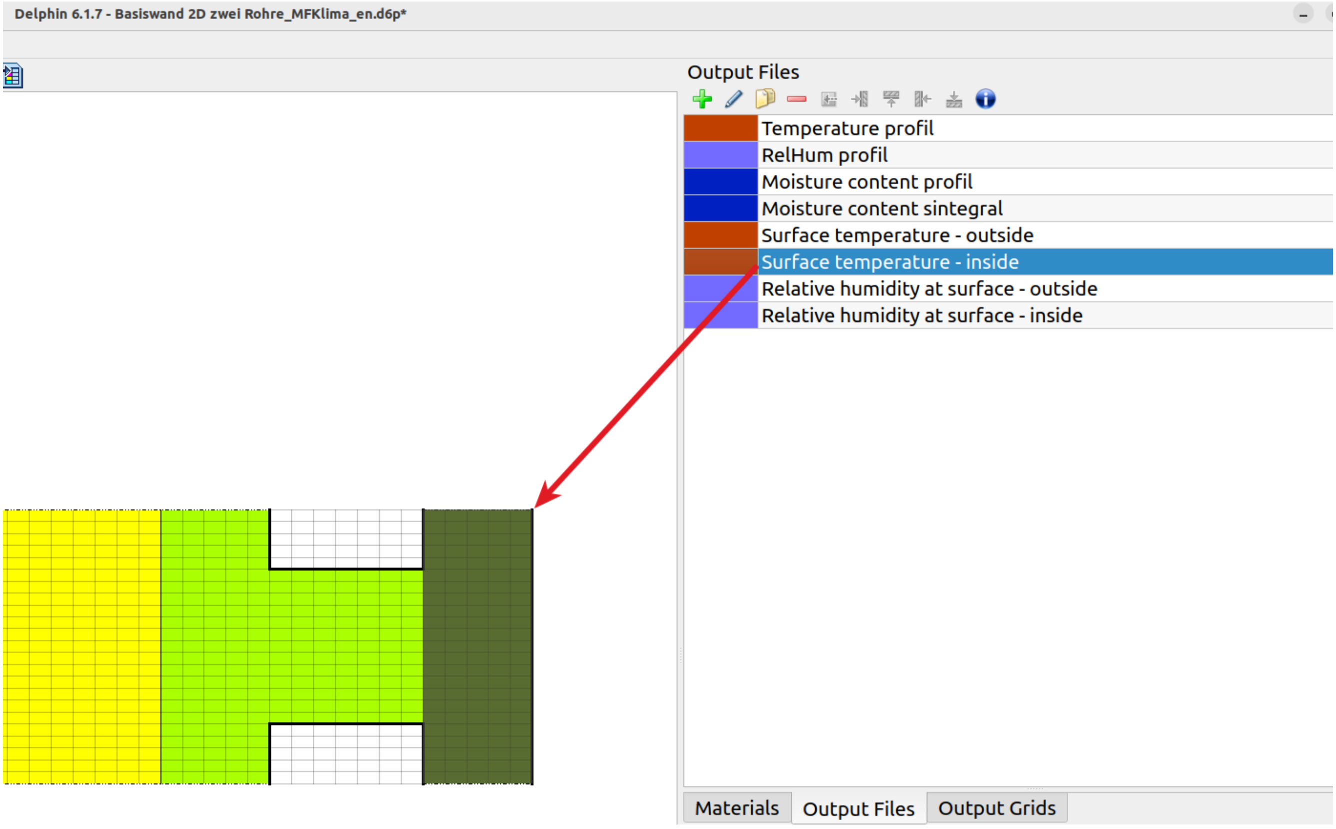


Figure 30. Incorrect assignment of outputs to the inner surface

As can be seen in the image above, the inner surface temperature is assigned to 5 elements width of the construction. This must now be corrected so that only the surface elements are assigned. To do this, open the assignment window for the outputs and search for the corresponding entry for the inner surface temperature.

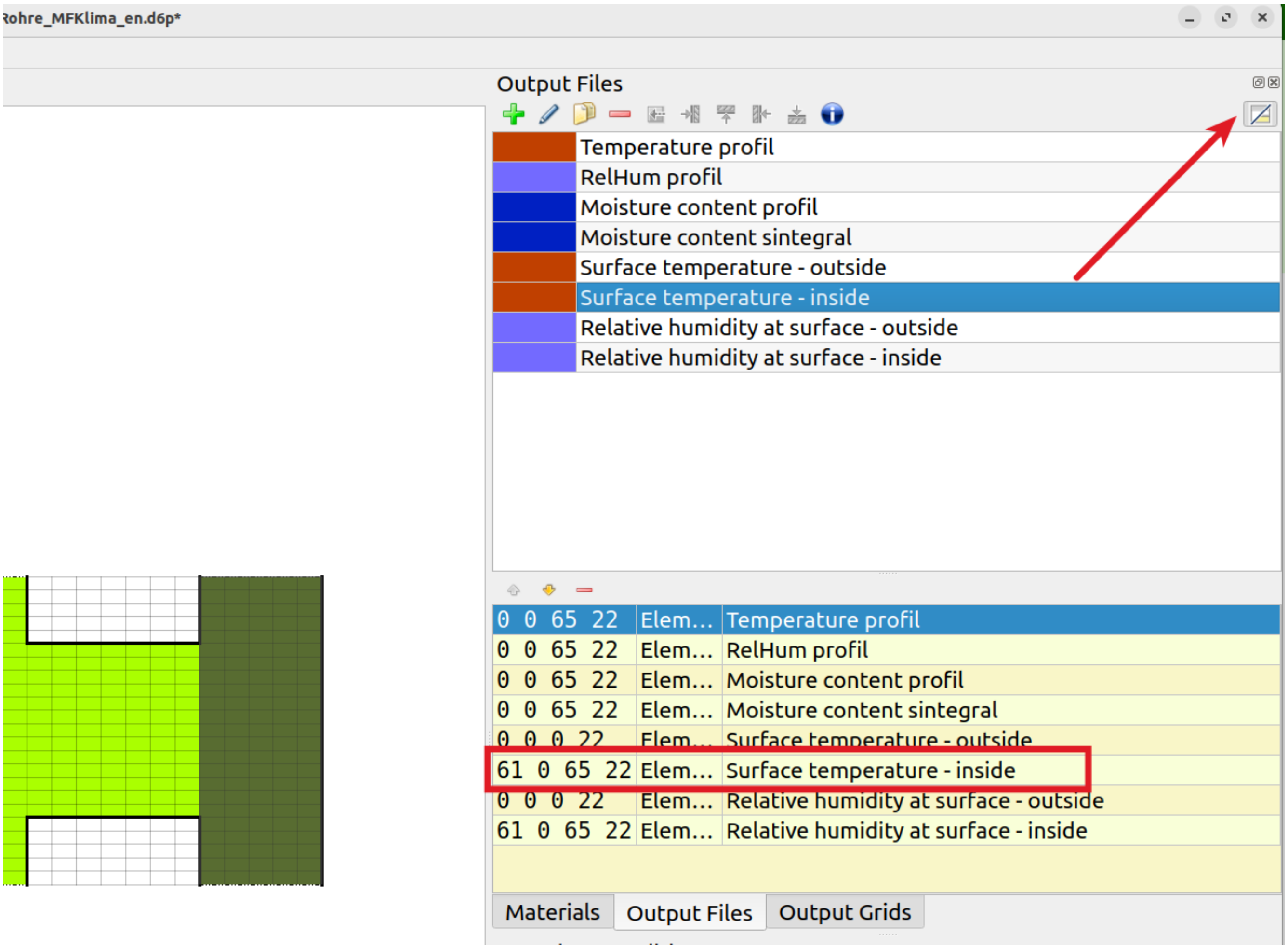


Figure 31. Assignment window with surface assignment

The assignment window shows the element numbers for the assignment area. The inner surface temperature is shown here: **61 0 65 22**. This means that an element range of 61 to 65 is selected for x and 0 to 22 for y. The y range is OK because it goes from the very bottom to the very top. However, the x-range should only have one element number, namely that of the element furthest to the right, and that would be 65. So all you have to do is change the 61 at the beginning to a 65 and the range is corrected. To do this, double-click on the entry in the assignment window, edit the entry and then click somewhere else to apply the change. The same change must be made for the humidity on the inner surface. The following image shows the result.

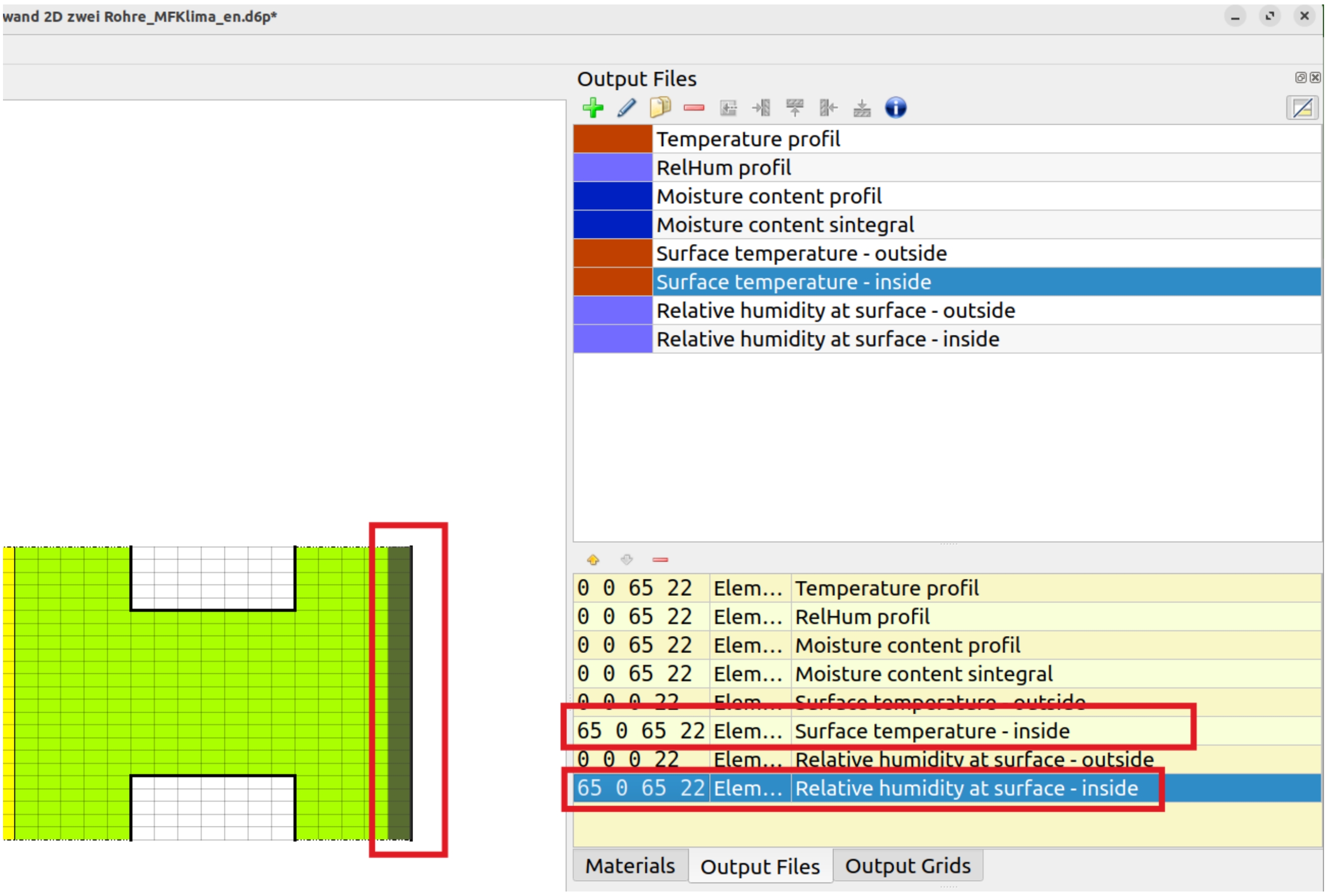


Figure 32. Assignment window with corrected entries

The construction can now be recalculated. The following two images show the temperature field on December 1st and a comparison of the mean surface temperatures for the models with one pipe and with two coupled pipes.

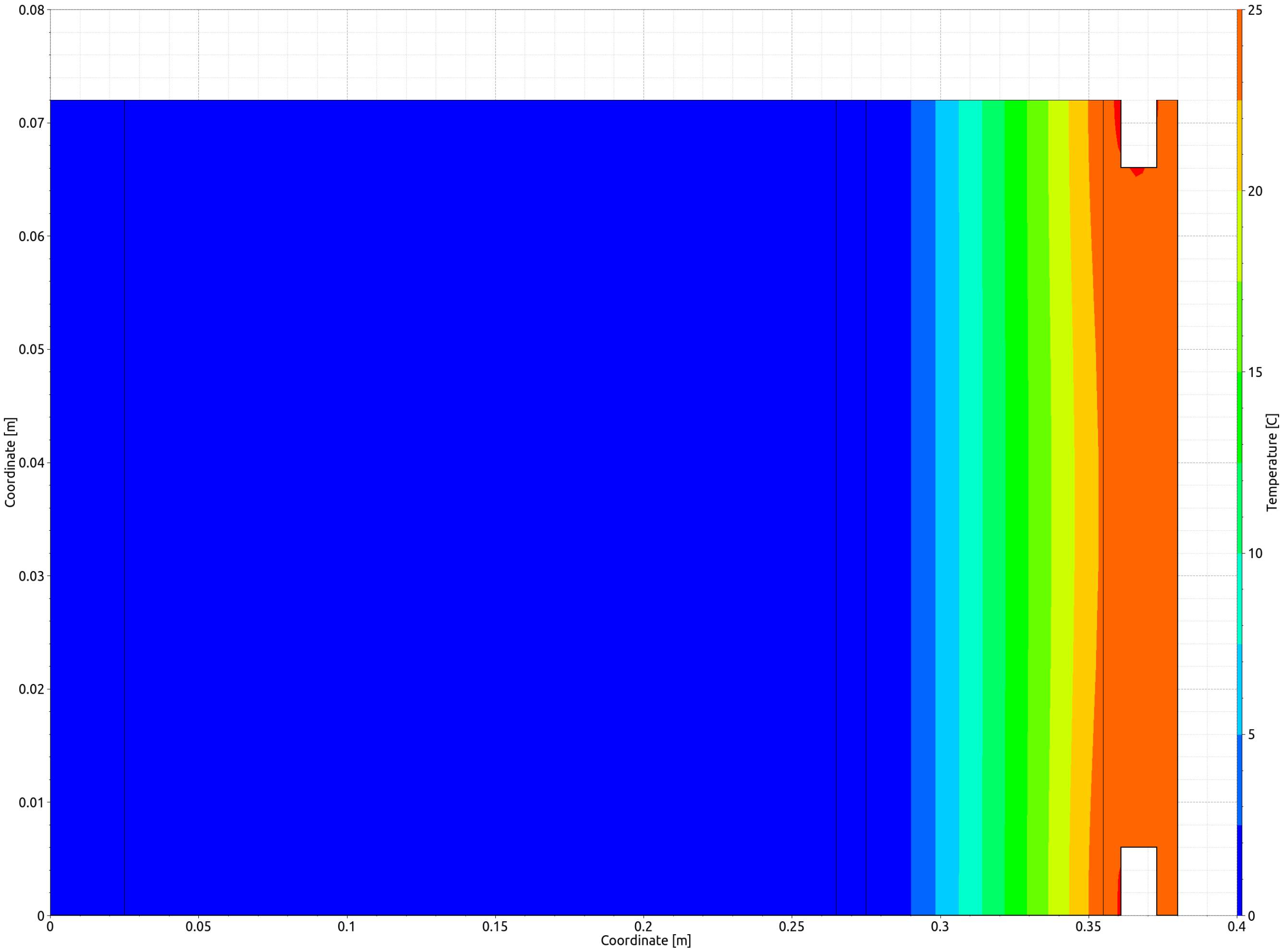


Figure 33. Temperature field on the first of December

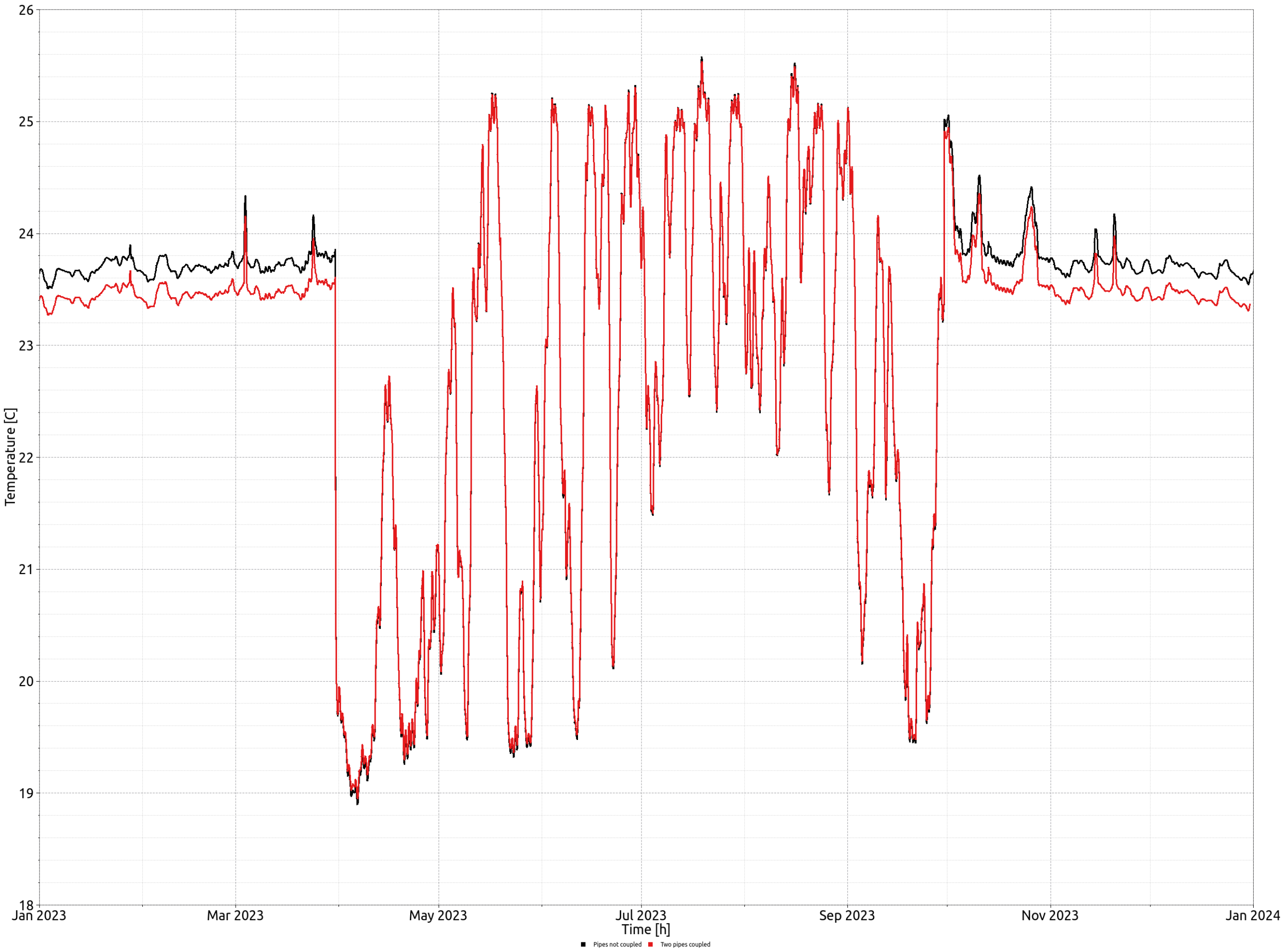


Figure 34. Surface temperature inside with and without raw coupling

3. Outlook

The pipe model allows a realistic representation of component-integrated heating and cooling systems. The model can also be used for all other situations in which flow-through pipes are used, such as underfloor heating or ground collectors. Especially in the latter case, the model allows an accurate and dynamic estimation of the extraction capacity of flat plate collectors and geothermal probes. The model is also often used in conjunction with building simulation. Just like DELPHIN, the SIM-VICUS software allows so-called FMUs (Functional Mocup Units) to be coupled out and simulated as a co-simulation using a master, such as our MASTERSIM. This coupling also allows external control systems or plant components to be included, which means that a large number of possible application scenarios can be covered.