Model created in COMSOL Multiphysics 6.4

Borehole Heat Exchanger with Pipe Flow

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Renewable energies are a growing industry and the geothermal energy branch is a hot topic of active research. Over the past few decades, different techniques were established to extract geothermal heat from shallow to deep subsurface levels. The closed-loop borehole heat exchanger (BHE) is a standard approach for lower- and mid-depth applications.

This model shows how to compute an array of borehole heat exchangers (BHEs) for shallow geothermal energy production. The BHEs are simplified as lines using the Nonisothermal Pipe Flow interface. The array is embedded into a layered subsurface model with groundwater flow in one of the layers.

Model Definition

The following model solves for the heat transport around a shallow geothermal installation embedded in a geological domain. The domain is separated into various parts, representing different geological layers with their particular properties. The thermal influence of seasonal temperature changes at the surface is taken into account using the COMSOL Multiphysics internal ambient weather database (in this case meteorological weather data from Berlin Tempelhof, Germany).

The geometry contains a three-by-three borehole heat exchanger (BHE) array that is 150 m deep and located in layered bedrock. Between 10 m and 50 m below the surface is an aquifer where groundwater flow occurs, causing horizontal convective heat transport. The BHEs are approximated by lines. Water of 10°C is pumped into the pipes at a volumetric flow rate of 6 l/min, corresponding to 10^{-4} m^3/s. Figure 1 shows the geometry as implemented in the model.

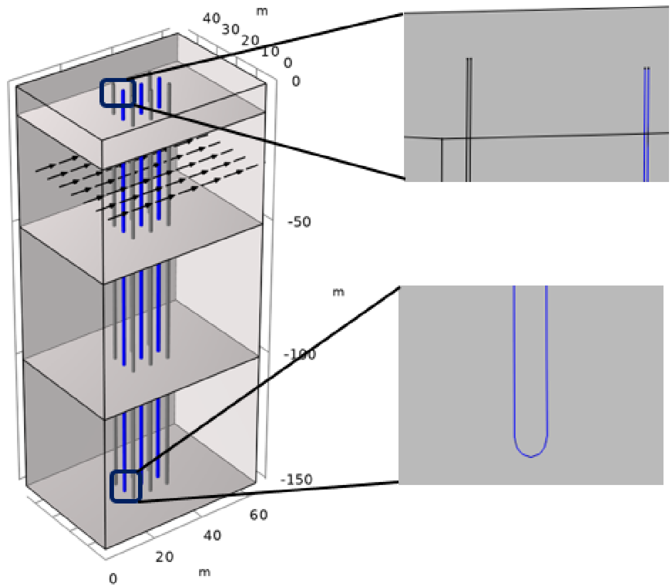


Figure 1: Geometry of the BHE model. The boreholes whose temperatures have been further investigated and displayed in the Results section are marked in blue. The black arrows illustrate the aquifer flow field.

Although the boreholes look just like lines, each borehole is modeled as a bent pipe with inlet and outlet at the top.

The heat transfer in the whole model domain is computed using the Heat Transfer in Porous Media interface. Within the aquifer layer, the aquifer flow velocity is implemented as a convective velocity field $\mathbf{u}$ in the heat equation

$$(\rho C_\mathrm{p})_\mathrm{eff}\frac{\partial T}{\partial t}+\rho_\mathrm{f}C_\mathrm{p,f}\mathbf{u}\cdot\nabla T-\nabla(k_\mathrm{eff}\nabla T) = Q \qquad (1)$$

The heat transfer between the BHEs and the surrounding bedrock is modeled using the Pipe Wall Heat Transfer multiphysics coupling node.

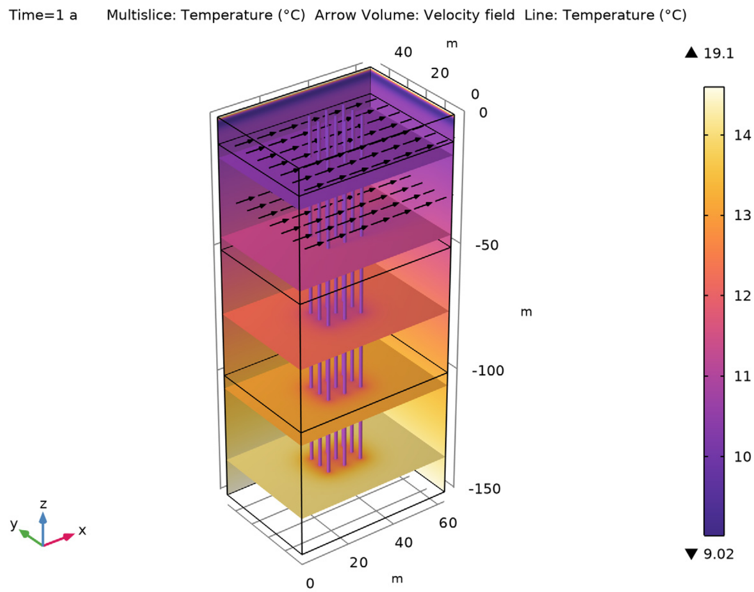


Figure 2: A multislice plot of the temperature after 1 year of simulated time. The borehole temperatures are displayed in a line plot and the arrow field shows the aquifer flow velocity.

Figure 2 and Figure 3 are examples of predefined ways to display the 3D temperature distribution in COMSOL Multiphysics. The top surface temperature changes seasonally between about 0°C and 22°C according to the meteorological weather data from Berlin Tempelhof, Germany, which leads to a layered temperature structure near the surface as shown in Figure 2. Both figures are dominated by the initial temperature profile, which shows a temperature increase with increasing depth. However, the cooling effect of the BHEs and the elongated heat pattern in the aquifer layer are clearly visible.

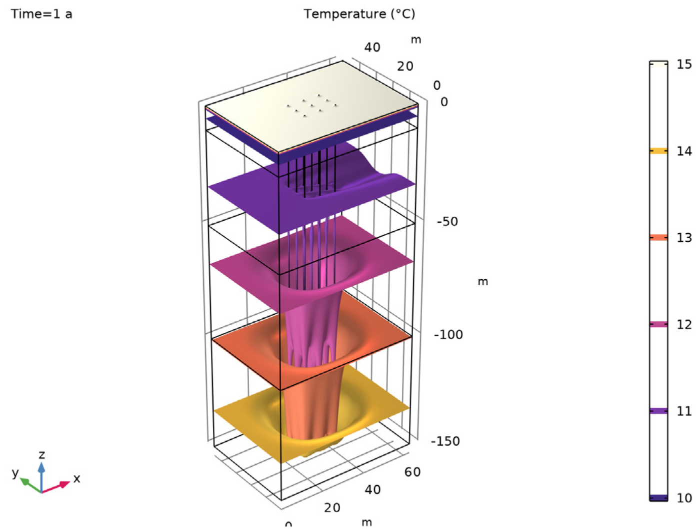


Figure 3: Isothermal layers around the BHEs after 1 year of simulated time.

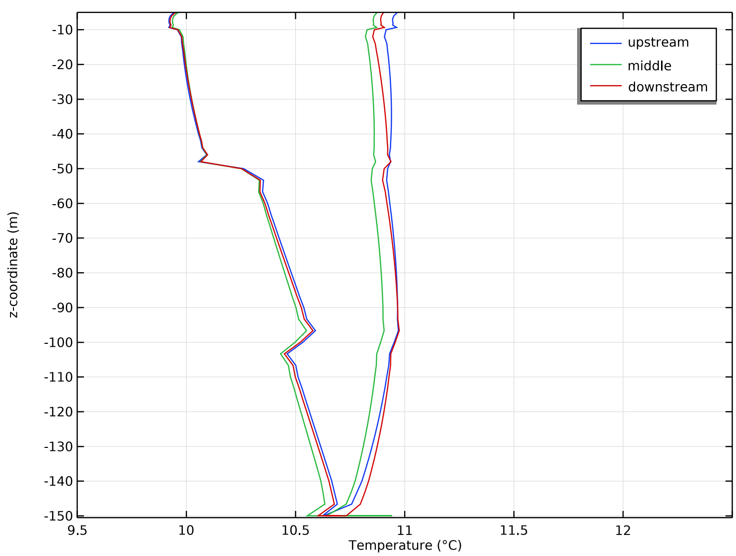


Figure 4: Temperature profiles at the three different borehole positions relative to the aquifer flow field.

Shown in Figure 4 are the pipe-surrounding temperatures of the three BHEs in the middle of the array (as marked in Figure 1). The temperatures in the parts with downward flowing water are lower than in the parts with upward flowing water, as expected due to the heat exchange with the surrounding bedrock. The temperature gradient is much smaller than the initial one because the cooling effect of the BHEs is larger for larger temperature differences and varies slightly with different thermal properties of the different porous material layers.

Due to the thermal interaction between the heat sinks, the temperature of the middle BHE relative to the aquifer flow field (green line) is lower than that of the other two in the pipe sections with upward flow. An effect of the aquifer can hardly be seen in the temperature profiles.

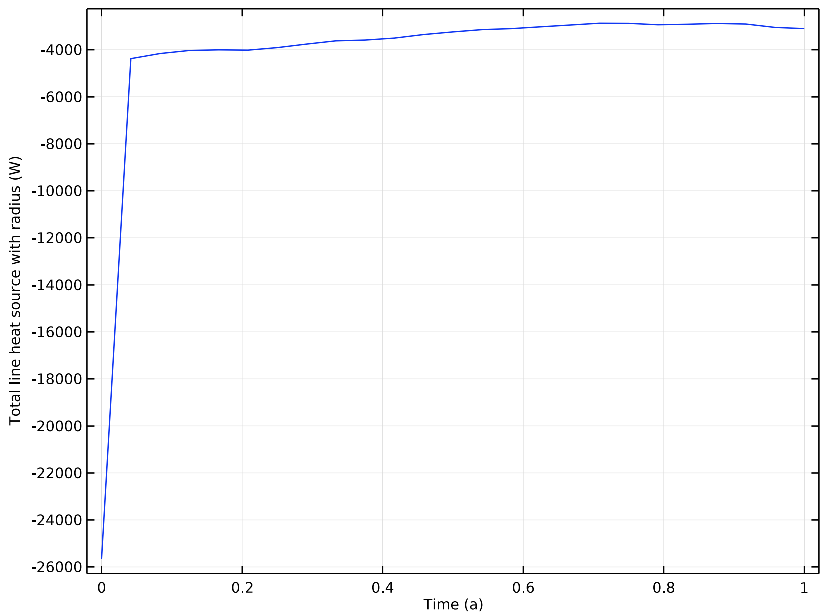


Figure 5: Total line heat source integrated over all BHEs as a function of time.

Figure 5 shows the total line heat source with time, integrated over all BHEs. It decreases fast in the beginning of the simulation and more slowly when the heat transfer between pipes and porous media reaches quasi-equilibrium. Finally it seems to level out at about –3000 W.

Notes About the COMSOL Implementation

The boreholes are modeled as lines using the Nonisothermal Pipe Flow interface. The heat transfer between the pipes and the surrounding soil is modeled using the **Pipe Wall Heat Transfer** multiphysics coupling node. This node can only be activated on boundaries where a **Wall Heat Transfer** node is active. The **Wall Heat Transfer** node allows to define multiple wall layers — each defined in a **Wall Layer** subnode — with different material properties, which is used here to represent both the pipe walls and the grouting between the pipes and the borehole walls.

Reference

1. www.comsol.com/blogs/modeling-geothermal-processes-comsol-software

---

Application Library path: `Pipe_Flow_Module/Heat_Transfer/borehole_heat_exchanger_pipe_flow`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Nonisothermal Flow** > **Nonisothermal Pipe Flow (nipfl)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer** > **Porous Media** > **Heat Transfer in Porous Media (ht)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies** > **Time Dependent**.

8 Click **Done**.

GEOMETRY 1

The geometry sequence can be imported from an external file. As the pipe curvature at the bottom of the boreholes is very small compared to the borehole depth and the array size, the **automatic detection of small details** checkbox is cleared in the **cleanup** section of the **Geometry** node.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `borehole_heat_exchanger_pipe_flow_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.
4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.
5 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
6 In the **Settings** window for **Geometry**, locate the **Cleanup** section.
7 Clear the **Automatic detection of small details** checkbox.

The geometry sequence is parametric. Add a few more parameters used setting up the physics.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
pth	3[mm]	0.003 m	Pipe wall thickness
u_aquifer	40[m/a]	1.2675E-6 m/s	Groundwater flow velocity
vfr	6[dm^3/min]	1E-4 m³/s	Volumetric flow rate at pipe inlet

MATERIALS

Define the different geological layers with their particular properties and use the selections provided by the **Block** feature in the geometry sequence.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in** > **Water, liquid**.

3 Click the **Add to Component** button in the window toolbar.

4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Water, liquid (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Edge**.

3 From the **Selection** list, choose **Work Plane: Boreholes**.

Porous Material: Holocene Sediments

1 In the **Model Builder** window, right-click **Materials** and choose **More Materials** > **Porous Material**.

2 In the **Settings** window for **Porous Material**, type `Porous Material: Holocene Sediments` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Top (Block 1)**.

4 Locate the **Phase-Specific Properties** section. Click **Add Required Phase Nodes**.

Fluid 1 (pmat1.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat1.fluid1)**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Material** list, choose **Water, liquid (mat1)**.

Solid 1 (pmat1.solid1)

1 In the **Model Builder** window, click **Solid 1 (pmat1.solid1)**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `0.7`.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1600	kg/m³	Basic
Heat capacity at constant pressure	Cp	2000	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1.2	W/(m·K)	Basic

Porous Material: Pleistocene Sands

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials** > **Porous Material**.

2 In the **Settings** window for **Porous Material**, type `Porous Material: Pleistocene Sands` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Aquifer (Block 1)**.

4 Locate the **Phase-Specific Properties** section. Click **Add Required Phase Nodes**.

Fluid 1 (pmat2.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat2.fluid1)**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Material** list, choose **Water, liquid (mat1)**.

Solid 1 (pmat2.solid1)

1 In the **Model Builder** window, click **Solid 1 (pmat2.solid1)**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `0.75`.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1700	kg/m³	Basic
Heat capacity at constant pressure	Cp	800	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1.3	W/(m·K)	Basic

Porous Material: Pleistocene Glacial Till

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials** > **Porous Material**.

2 In the **Settings** window for **Porous Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Glacial Till (Block 1)**.

4 In the **Label** text field, type `Porous Material: Pleistocene Glacial Till`.

5 Locate the **Phase-Specific Properties** section. Click **Add Required Phase Nodes**.

Fluid 1 (pmat3.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat3.fluid1)**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Material** list, choose **Water, liquid (mat1)**.

Solid 1 (pmat3.solid1)

1 In the **Model Builder** window, click **Solid 1 (pmat3.solid1)**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `0.85`.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	2100	kg/m³	Basic
Heat capacity at constant pressure	Cp	900	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1.8	W/(m·K)	Basic

Porous Material: Tertiary Sands

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials** > **Porous Material**.

2 In the **Settings** window for **Porous Material**, type `Porous Material: Tertiary Sands` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Core (Block 1)**.

4 Locate the **Phase-Specific Properties** section. Click **Add Required Phase Nodes**.

Fluid 1 (pmat4.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat4.fluid1)**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Material** list, choose **Water, liquid (mat1)**.

Solid 1 (pmat4.solid1)

1 In the **Model Builder** window, click **Solid 1 (pmat4.solid1)**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `0.8`.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	1900	kg/m³	Basic
Heat capacity at constant pressure	Cp	850	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1.4	W/(m·K)	Basic

The **Material** node indicates that there are still missing material properties, which is the permeability required for Darcy's Law. The permeability in the lower two layers is about two orders of magnitude smaller than in the upper two. Therefore add the permeability only for the upper layers.

NONISOTHERMAL PIPE FLOW (NIPFL)

Now start to set up the physics. Start with the Nonisothermal Pipe Flow interface which should only be active on the pipe edges.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Nonisothermal Pipe Flow (nipfl)**.

2 In the **Settings** window for **Nonisothermal Pipe Flow**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Work Plane: Boreholes**.

Pipe Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Nonisothermal Pipe Flow (nipfl)** click **Pipe Properties 1**.

2 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

3 From the list, choose **Circular**.

4 In the d_i text field, type `2*(r_pipe-pth)`.

Temperature 1

1 In the **Model Builder** window, click **Temperature 1**.
2 In the **Settings** window for **Temperature**, locate the **Temperature** section.
3 In the T_{in} text field, type `283.15[K]`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3 In the T text field, type `283.15[K]`.
4 In the p text field, type `1[atm]`.

Pressure 1

1 In the **Model Builder** window, click **Pressure 1**.
2 In the **Settings** window for **Pressure**, locate the **Boundary Pressure** section.
3 In the p_0 text field, type `1[atm]`.

Inlet 1

1 In the **Physics** toolbar, click **Points** and choose **Inlet**.
2 In the **Settings** window for **Inlet**, locate the **Point Selection** section.
3 From the **Selection** list, choose **Pipe Inlet (Work Plane: Boreholes)**.
4 Locate the **Inlet Specification** section. From the **Specification** list, choose **Volumetric flow rate**.
5 In the $q_{v,0}$ text field, type `vfr`.

Heat Outflow 1

1 In the **Physics** toolbar, click **Points** and choose **Heat Outflow**.
2 In the **Settings** window for **Heat Outflow**, locate the **Point Selection** section.
3 From the **Selection** list, choose **Pipe Outlet (Work Plane: Boreholes)**.

Wall Heat Transfer 1

1 In the **Physics** toolbar, click **Edges** and choose **Wall Heat Transfer**.
2 In the **Settings** window for **Wall Heat Transfer**, locate the **Edge Selection** section.
3 From the **Selection** list, choose **Work Plane: Boreholes**.

As the **Wall Heat Transfer** settings are influenced by the multiphysics coupling between heat transfer in pipes and heat transfer in the surrounding porous medium, add the corresponding multiphysics node before continuing.

MULTIPHYSICS

Pipe Wall Heat Transfer 1 (pwhtc1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Edge** > **Pipe Wall Heat Transfer**.

NONISOTHERMAL PIPE FLOW (NIPFL)

Wall Heat Transfer 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Nonisothermal Pipe Flow (nipfl)** click **Wall Heat Transfer 1**.

Wall Layer 1

1 In the **Physics** toolbar, click **Attributes** and choose **Wall Layer**.
2 In the **Settings** window for **Wall Layer**, locate the **Specification** section.
3 From the k list, choose **User defined**.
4 In the text field, type `0.38[W/m/K]`.
5 From the Δw list, choose **User defined**.
6 In the text field, type `pth`. Now you have added the thermal conductivity of the pipe material and the pipe wall thickness. In a second layer consider the grouting material which fills the space between pipes and borehole walls.

Wall Heat Transfer 1

In the **Model Builder** window, click **Wall Heat Transfer 1**.

Wall Layer 2

1 In the **Physics** toolbar, click **Attributes** and choose **Wall Layer**.
2 In the **Settings** window for **Wall Layer**, locate the **Specification** section.
3 From the k list, choose **User defined**.
4 In the text field, type `2.4[W/m/K]`.
5 From the Δw list, choose **User defined**.
6 In the text field, type `r_bore-r_pipe`.

Wall Heat Transfer 1

In the **Model Builder** window, click **Wall Heat Transfer 1**.

Internal Film Resistance 1

In the **Physics** toolbar, click **Attributes** and choose **Internal Film Resistance**.

GLOBAL DEFINITIONS

Before setting up the heat transfer in porous media, add an analytic function to describe the initial temperature profile in the bedrock.

Analytic 1 (an1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global** > **Analytic**.

2 In the **Settings** window for **Analytic**, type `T0` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `283.15[K]+3[K]/100[m]*abs(z)`.

4 In the **Arguments** text field, type `z`.

5 Locate the **Units** section. In the **Function** text field, type `K`.

6 In the table, enter the following settings:

Argument	Unit
z	m

Furthermore, add the ambient properties to provide climate data for the upper boundary conditions.

DEFINITIONS (COMP1)

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Settings** section.

3 From the **Ambient data** list, choose **Meteorological data (ASHRAE 2021)**.

4 Locate the **Location** section. Click **Set Weather Station**.

5 In the **Weather Station** dialog, select **Europe** > **Germany** > **BERLIN TEMPELHOF (103840)** in the tree.

6 Click **OK**.

HEAT TRANSFER IN POROUS MEDIA (HT)

Now set up the physics of the Heat Transfer in Porous Media interface.

Porous Matrix 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Porous Media (ht)** > **Porous Medium 1** click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the **Define** list, choose **Solid phase properties**.

Porous Medium 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Porous Media (ht)** right-click **Porous Medium 1** and choose **Duplicate**.

2 In the **Settings** window for **Porous Medium**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 3 only.

Fluid 1

1 In the **Model Builder** window, expand the **Porous Medium 2** node, then click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Heat Convection** section.

3 Specify the $\mathbf{u}$ vector as

u_aquifer	x

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Porous Media (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the $T2$ text field, type `T0(z)`.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 13 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 From the T_0 list, choose **Ambient temperature (ampr1)**.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 In the **Settings** window for **Open Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior**.

4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T0(z)`.

Temperature 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T0(z)`.

MESH 1

Now mesh the geometry. The mesh parameters are chosen to resolve the distance between the upward- and downward part of each borehole pipe. A swept mesh in the vertical direction is more efficient than a triangular mesh.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.
2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 6 only.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Calibrate for** list, choose **Fluid dynamics**.
4 From the **Predefined** list, choose **Extremely fine**.
5 Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Minimum element size** checkbox. In the associated text field, type `bhd*0.25`.
8 Select the **Maximum element size** checkbox. In the associated text field, type `2`.
9 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.75`.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Build Selected**.

Free Tetrahedral 2

1 In the **Mesh** toolbar, click **Free Tetrahedral**.
2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 1 only.

Size 1

1 Right-click **Free Tetrahedral 2** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Calibrate for** list, choose **Fluid dynamics**.
4 From the **Predefined** list, choose **Coarse**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.5`.

Free Tetrahedral 2

In the **Model Builder** window, right-click **Free Tetrahedral 2** and choose **Build Selected**.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 4 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `15`.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 3 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `20`.

Distribution 3

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

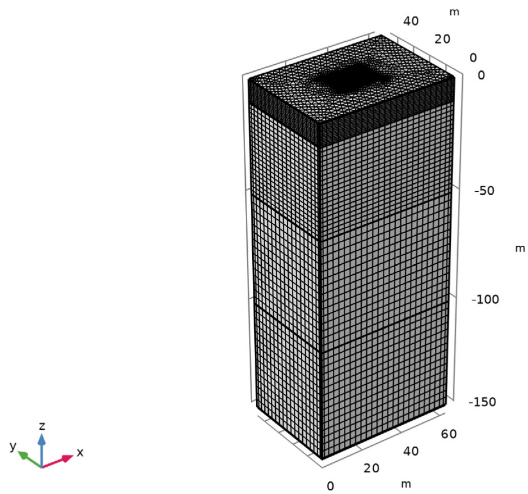
3 Click **Clear Selection**.

4 Select Domains 2 and 5 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `15`.

Swept 1

Right-click **Swept 1** and choose **Build Selected**.



STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** checkbox.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **a**.

4 In the **Output times** text field, type `range(0,1/24,1)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Strict** to ensure that the changes of the ambient temperature are represented properly.

Step 1: Time Dependent

In the **Study** toolbar, click **Compute**.

RESULTS

To create Figure 2 and Figure 3 open the **Result Templates**.

RESULT TEMPLATES

1 In the **Home** toolbar, click **Windows** and choose **Result Templates**.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Porous Media** > **Temperature, Multislice (ht)** and **Study 1/Solution 1 (sol1)** > **Heat Transfer in Porous Media** > **Isothermal Contours (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Temperature, Multislice (ht)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **X-planes** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `lx`.

5 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `ly`.

7 Find the **Z-planes** subsection. In the **Planes** text field, type `5`.

8 Locate the **Expression** section. From the **Unit** list, choose **°C**.

9 Click to expand the **Range** section. Select the **Manual color range** checkbox.

10 In the **Maximum** text field, type `14.6`.

Temperature, Multislice (ht)

1 In the **Model Builder** window, click **Temperature, Multislice (ht)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** checkbox.

Arrow Volume 1

1 Right-click **Temperature, Multislice (ht)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Porous Media** > **Velocity and pressure** > **Porous medium, fluid** > **ht.porous.fluid.ux,...,ht.porous.fluid.uz - Velocity field**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Line 1

1 Right-click **Temperature, Multislice (ht)** and choose **Line** to add the pipe temperatures to the plot.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 From the **Unit** list, choose **°C**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Multislice 1**.

5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

6 In the **Temperature, Multislice (ht)** toolbar, click **Plot**.

7 Click the **Go to Default View** button in the **Graphics** toolbar.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

Isosurface 1

Here, specific levels are plotted to make it easier to compare the plot with other BHE models in our library.

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `10 11 12 13 14 15`.

5 Locate the **Expression** section. From the **Unit** list, choose **°C**.

6 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

Temperature Profile

Create Figure 4 by following the steps below.

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Time selection** list, choose **From list**.

4 In the **Times (a)** list box, select **1**.

5 In the **Label** text field, type `Temperature Profile`.

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Line Graph 1

1 Right-click **Temperature Profile** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Probe 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `z`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `T2`.

7 From the **Unit** list, choose **°C**.

8 Click to expand the **Legends** section. Select the **Show legends** checkbox.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`upstream`

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Probe 2**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`middle`

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Probe 3**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`downstream`

5 Click the **Go to Default View** button in the **Graphics** toolbar.

6 In the **Temperature Profile** toolbar, click **Plot**.

As the ambient temperature varies strongly with time and is much higher than in the subsurface layers, define axis limits to get a better view of the region of interest.

Temperature Profile

1 In the **Model Builder** window, click **Temperature Profile**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** checkbox.

4 In the **x minimum** text field, type `9.5`.

5 In the **x maximum** text field, type `12.5`.

6 In the **y minimum** text field, type `-150.5`.

7 In the **y maximum** text field, type `-5`.

8 In the **Temperature Profile** toolbar, click **Plot**.

Line Integration 1

In the next step you analyze the energy transferred from the soil to the BHEs.

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Line Integration**.

2 In the **Settings** window for **Line Integration**, locate the **Selection** section.

3 From the **Selection** list, choose **Work Plane: Boreholes**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Porous Media** > **Heat sources** > **ht.Qlrtot - Total line heat source with radius - W/m**.

5 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

2 Click the **Table Graph** button in the window toolbar.

RESULTS

Total Line Heat Source (Integrated over Pipes)

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Total Line Heat Source (Integrated over Pipes)` in the **Label** text field and compare with Figure 5.