Model created in COMSOL Multiphysics 6.4

Borehole Heat Exchanger

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 line heat sinks with a uniform extraction rate. The array is embedded into a layered subsurface model with groundwater flow in one of the layers.

Model Definition

This model has been set up for comparison with the model using the Pipe Flow Module found in the application library under *Pipe_Flow_Module/Heat_Transfer/borehole_heat_exchanger_pipe_flow* and is also presented in a COMSOL Blog post (Ref. 1). While that model uses the Nonisothermal Pipe Flow interface, the current model shows an alternative approach using line heat sources.

The model-geometry and boundary-condition setups are similar to those of the model version with pipe flow (Figure 1). The main difference is that here the boreholes rather than the pipes are represented, and therefore only straight lines instead of u-shaped curves at the bottom are implemented.

The borehole heat exchangers (BHEs) are arranged in a three-by-three array that is 150 m deep and located in layered bedrock. Between 10 m and 50 m is an aquifer where groundwater flow occurs, causing horizontal convective heat transport.

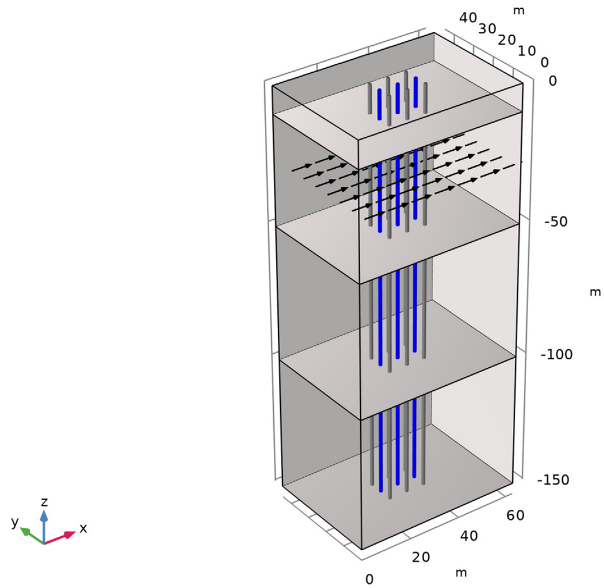

Figure 1: Geometry of the BHE model. The boreholes whose temperatures have been further investigated and displayed in the Results section are marked in blue. The black arrows illustrate the aquifer flow field.

The heat transfer in the whole model domain is computed using the Heat Transfer in Porous Media interface. In addition, the fluid flow in porous media is calculated within the top layer and the aquifer layer using the Darcy's Law interface.

The borehole heat exchangers are modeled as line heat sources which extract energy from the model. The heat rate is set to −3000 W, which corresponds to the value that is calculated in the model using the Pipe Flow Module.

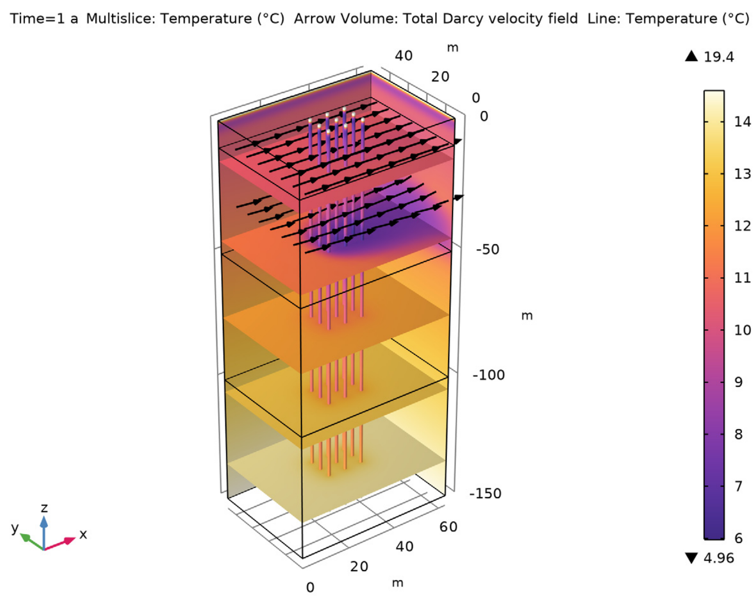

Figure 2: Temperature distribution after 1 year of simulated time within and around the BHEs.

Figure 2 and Figure 3 are examples of predefined ways to display the 3D temperature distribution in COMSOL Multiphysics. Both figures are dominated by the initial temperature profile, which shows a temperature increase with increasing depth. The top surface temperature changes seasonally between about 0°C and 22°C according to the meteorological weather data from Berlin, Germany, which leads to a layered temperature structure near the surface as shown in Figure 2. Figure 3 shows how the heat pattern elongates in the direction of flow due to convection in the region of the aquifer.

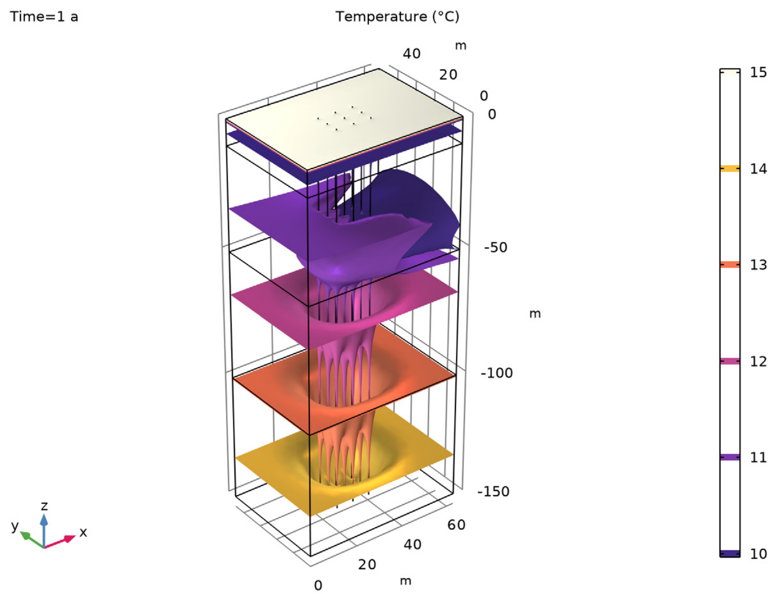

Figure 3: Isothermal layers around the BHEs after 1 year of simulated time.

When comparing Figure 2 and Figure 3 with the corresponding figures of the model using the Pipe Flow module, there is hardly any difference in the global temperature field (Figure 3). In Figure 2, the temperature of the pipe-surrounding bedrock is shown while for the version using pipe flow, the temperatures within the pipes are displayed. Therefore, the influence of the external temperature at the top is visible in this model version. However, the pipe flow model version shows a stronger cooling effect near the boreholes in the lower half of the model and a smaller cooling effect closer to the surface for the model version using pipe flow, as the heat extraction rate in the latter case can vary with height whereas here it is provided as a constant.

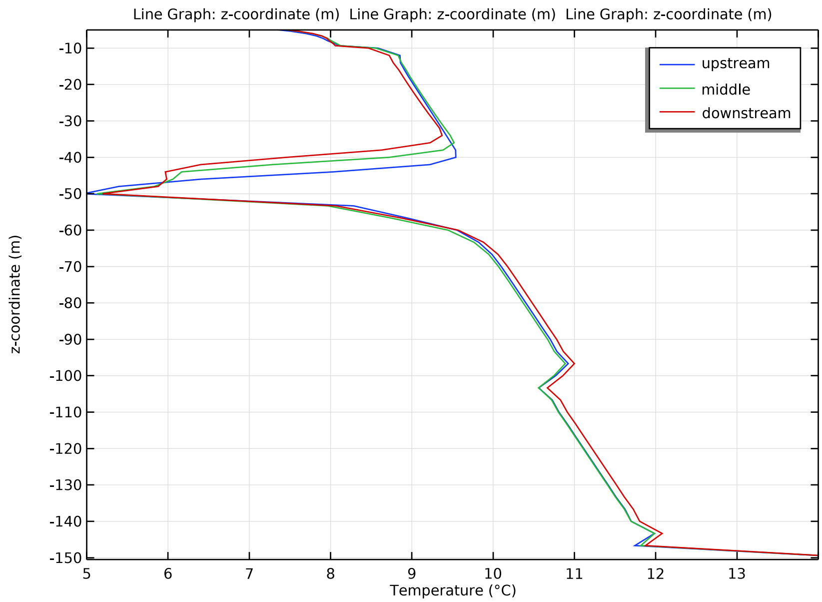

Figure 4: Temperature profiles at the three different borehole positions relative to the aquifer flow field.

Figure 4 displays the borehole wall temperatures of the three BHEs in the middle of the array (as marked in Figure 1). Due to the thermal interaction between the heat sinks, the temperature of middle BHE relative to the aquifer flow field (green line) is lower than the other two. Only in the region of the aquifer, the BHE further downstream than the other two (red line) is additionally affected by the heat exchange occurring upstream of it due to convection, resulting in a lower temperature.

Compared with the temperature profile of the model version using pipe flow, the temperature profiles here are dominated by the initial temperature profile. A vertically constant heat extraction only shifts the temperature profile to lower temperatures rather than influencing the temperature gradient. While in the model version using the Pipe Flow Module, the effect of the aquifer on the temperature profiles is exceeded by the effect of vertically varying energy extraction, it is clearly visible here as described above.

Notes About the COMSOL Implementation

The boreholes are modeled as line heat sources. To make it available to users that just have the Subsurface Flow Module or Porous Media Flow Module licensed, the line heat source option does not contain the radius of the line heat source. The option to enter a line heat source radius is available with the Heat Transfer Module and does actually affect the temperatures around the boreholes quantitatively by shifting the profiles about 0.5 K toward higher values. However, as described when interpreting Figure 4, for a vertically constant line heat source, the temperature profiles differ significantly from the ones modeled with the Pipe Flow Module in general, so the qualitative results of both the model and the model comparison stays the same.

Reference

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

---

Application Library path: `Subsurface_Flow_Module/Heat_Transfer/borehole_heat_exchanger`

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** > **Porous Media and Subsurface Flow** > **Darcy's Law (dl)**.
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

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_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.

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
Pext	3[kW]	3000 W	Heat extraction rate
u_aquifer	40[m/a]	1.2675E-6 m/s	Groundwater flow velocity

MATERIALS

Now set up the materials for each layer 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

Porous Material: Holocene Sediments

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: 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.

Porous Material: Holocene Sediments (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Porous Material: Holocene Sediments (pmat1)**.
2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.
3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1.3e-11	m^2	Basic

Porous Material: Pleistocene Sands (pmat2)

1 In the **Model Builder** window, click **Porous Material: Pleistocene Sands (pmat2)**.
2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.
3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1.3e-10	m^2	Basic

Continue with setting up the physics. Add the ambient properties to provide meteorological 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**.

DARCY'S LAW (DL)

Next, set up the physics for the Darcy's Law interface. It is sufficient to activate it only in the domains with significant flow velocities (that is, the aquifer layer and top layer).

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 In the **Settings** window for **Darcy's Law**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 3 and 4 only.

5 Locate the **Gravity Effects** section. Select the **Include gravity** checkbox.

Assume a constant groundwater flow velocity in the aquifer layer. This can be done by using the **Contributing Velocity** feature.

Porous Medium 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** click **Porous Medium 1**.

Contributing Velocity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Contributing Velocity**.

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

3 Click **Clear Selection**.

4 Select Domain 3 only.

5 Locate the **Contributing Velocity** section. Specify the $\mathbf{u}_{\mathrm{ctrb}}$ vector as

u_aquifer	x

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** click **Initial Values 1**.

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

3 Click the **Hydraulic head** button.

Hydraulic Head 1

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

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

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

Precipitation 1

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

2 Select Boundary 13 only.

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

4 From the P_0 list, choose **Ambient precipitation rate (ampr1)**.

5 Select Boundary 13 only.

DEFINITIONS (COMP1)

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 **Definitions** toolbar, click **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+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

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	z	-155	0	0	m

8 Click **Plot**.

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**.

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 T text field, type `T0(z)`.

Porous Medium 2

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domains 3 and 4 only.

Fluid 1

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

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

3 From the $\mathbf{u}$ list, choose **Total Darcy velocity field (dl/porous1)**.

Porous Matrix 1

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

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)**.

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)`.

Open Boundary 1

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

2 Select Boundaries 7, 8, 10, 11, 16, 17, 20, and 21 only.

3 In the **Settings** window for **Open Boundary**, locate the **Upstream Properties** section.

4 In the T_{ustr} text field, type `T0(z)`.

Line Heat Source 1

1 In the **Physics** toolbar, click **Edges** and choose **Line Heat Source**.

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

3 From the **Heat source** list, choose **Heat rate**.

4 In the P_{l} text field, type `-Pext`.

5 Locate the **Edge Selection** section. From the **Selection** list, choose **Array 1**.

MESH 1

The next step is to set up the mesh. Choose a fine mesh around the boreholes to guarantee that the heat exchange is correctly simulated. A swept mesh in the vertical direction is more efficient than a triangular mesh.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundary 3 only.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

3 From the **Geometric entity level** list, choose **Point**.

4 Select Points 11, 16, 21, 26, 31, 36, 41, 46, and 51 only.

5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.02`.

8 Click **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 Domains 1, 2, and 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.

6 Click **Build All**.

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 **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

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)** > **Darcy's Law** > **Velocity and pressure** > **dl.u,dl.v,dl.w - Total Darcy 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 temperatures around the boreholes to the plot.

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

3 In the **Expression** text field, type T2.

4 In the **Unit** field, type °C.

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

6 Locate the **Expression** section. In the **Expression** text field, type T.

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

Selection 1

1 Right-click **Line 1** and choose **Selection**.

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

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

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

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

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

Isothermal Contours (ht)

Here, specific levels are plotted to make it easier to compare the plot with the model version using the Pipe Flow module.

Isosurface 1

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

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

7 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`.

Line Graph 1

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

2 Click the **Go to XY View** button in the **Graphics** toolbar.

3 Click the **Select Box** button in the **Graphics** toolbar and span a rectangle around the point belonging to the most left borehole of those marked in Figure 1. This automatically marks all edges within and below the selection box. Now the edges 28-31 should be selected.

4 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

5 In the **Expression** text field, type `z`.

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

7 In the **Expression** text field, type `T`.

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

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

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

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

12 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 Click **Clear Selection**.

4 Select Edges 40–43 only. You could again use the **Go to XY View** and the **Select Box** buttons from the graphics toolbar to select the edges corresponding to the middle borehole.

5 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 Click **Clear Selection**.

4 Select Edges 52–55 only to select the edges corresponding to the right borehole of the middle row.

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

Legends
downstream

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

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

Because 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 `5`.

5 In the **x maximum** text field, type `14`.

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** and compare with Figure 4.