Created in COMSOL Multiphysics 5.6

Geothermal Doublet

Introduction

The use of geothermal energy has become an important topic worldwide, not only sparked by the debate on climate change. Particularly in regions with lower geothermal energy, the requirements for geothermal plants are high in order to ensure a continuous and long-term energy supply.

This example of a hydrothermal doublet studies the coupled porous media flow and heat transfer problem using the Darcy's Law and Heat Transfer in Porous Media interfaces. It shows how to set up a system of a layered subsurface with different geological and thermal properties including fractures and the injection and production side of a doublet.

Model Definition

The model geometry (Figure 1) consists of three geological layers ranged by a fault zone. The layers, their elevation, and the fault zone are interpolation functions from an artificial dataset. Different hydraulic and thermal properties are defined for the layers. The evolution of the flow and temperature field over 10 years is studied.

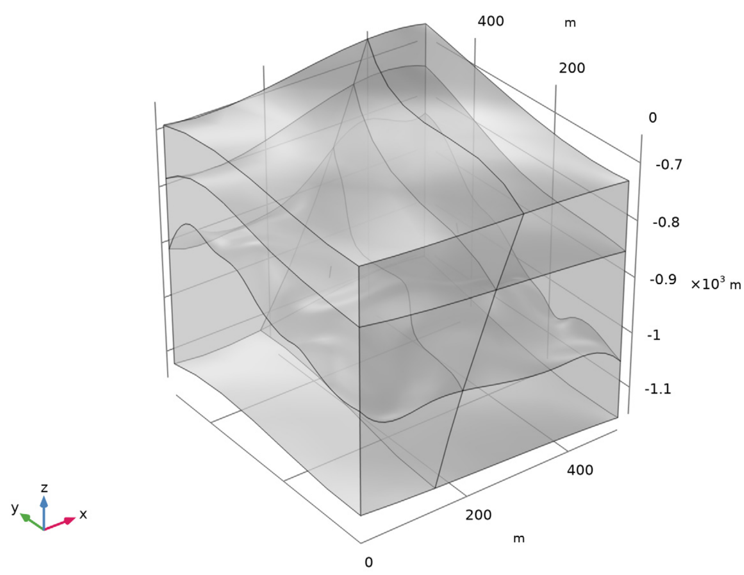


Figure 1: Geometry of the model including three geologic layers, fracture, and the doublet's production and injection side (edges).

FLUID FLOW

The flow in the fracture is in general much faster than in the surrounding porous matrix. The cubic law is a common correlation for modeling fracture flow. It defines the permeability κ_f (m^2) in the fracture according to

$$\kappa_f = \frac{d_f^2}{12 f_f}$$

where d_f (m) is the fracture's aperture and f_f the roughness factor. With this definition, the name cubic law results from the definition of the fracture's transmissivity

$$T_r = d_f k_f = \frac{d_f^3}{12 f_f}.$$

A hydraulic gradient in x direction is applied as boundary condition on the vertical boundaries. Top and bottom boundary are defined as impermeable. Two edges represent the geothermal doublet injection and production side. Injection and pumping is modeled using the Well feature. Water injection is defined using the mass flow rate option with $M_0 = \rho_W \cdot 120$ l/s.

HEAT TRANSFER

An initial geothermal gradient of 0.03 K/m is applied. The vertical boundaries are defined as open boundaries. This means that a temperature condition using the same geothermal gradient is active if $\mathbf{n} \cdot \mathbf{u} < 0$ (inflow) or otherwise $-\mathbf{n} \cdot \mathbf{q} = 0$ (outflow). Analogously to the Fracture Flow boundary condition in the Darcy's Law interface, the heat transfer in the fracture is modeled using the fracture boundary condition of the heat transfer interface.

To model the heat source term caused by the injection well, a line heat source feature is applied according to

$$Q = C_p \frac{M_0}{l} (T_{inj} - T)$$

where C_p (J/(kg·K)) is the water heat capacity, l the well length, $T_{inj} = 278$ K the injection temperature, and T the current temperature.

Results and Discussion

The pressure field is shown in Figure 2.

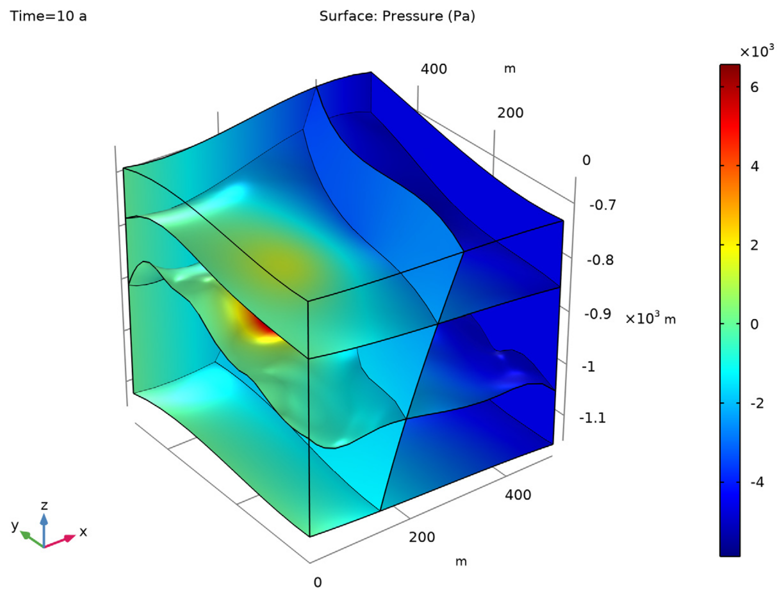


Figure 2: Pressure distribution after 10 years.

Figure 3 shows the temperature distribution in the fracture together with streamlines that visualize the flow field, showing the flow from the injection to the production side of the doublet.

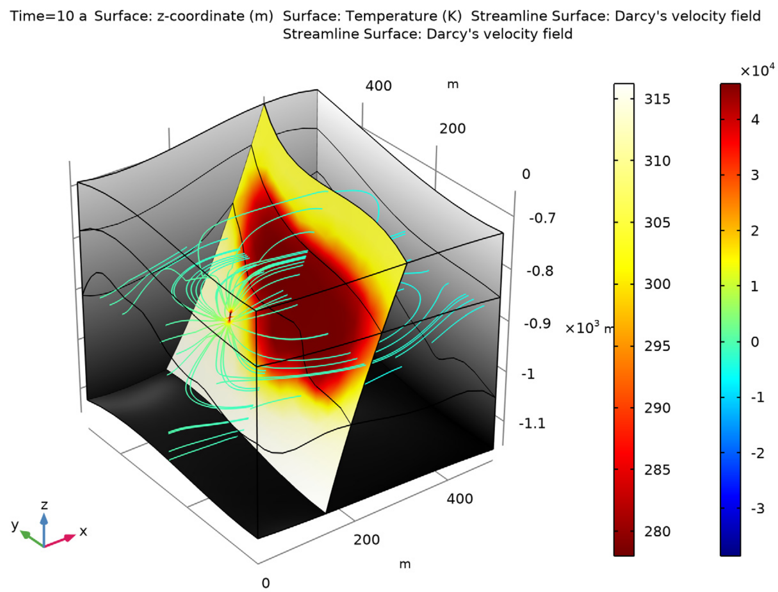


Figure 3: Temperature distribution in the fracture and streamlines for the Darcy velocity field after 10 years.

It is interesting to evaluate the production temperature over time. As shown in Figure 4, the production temperature decreases about 20 K over 10 years. This indicates that the doublet at the operating conditions will not provide a stable long term energy supply and that a different configurations should be tested. This can be done by varying one or more of the parameters in the model to see which setup is more appropriate.

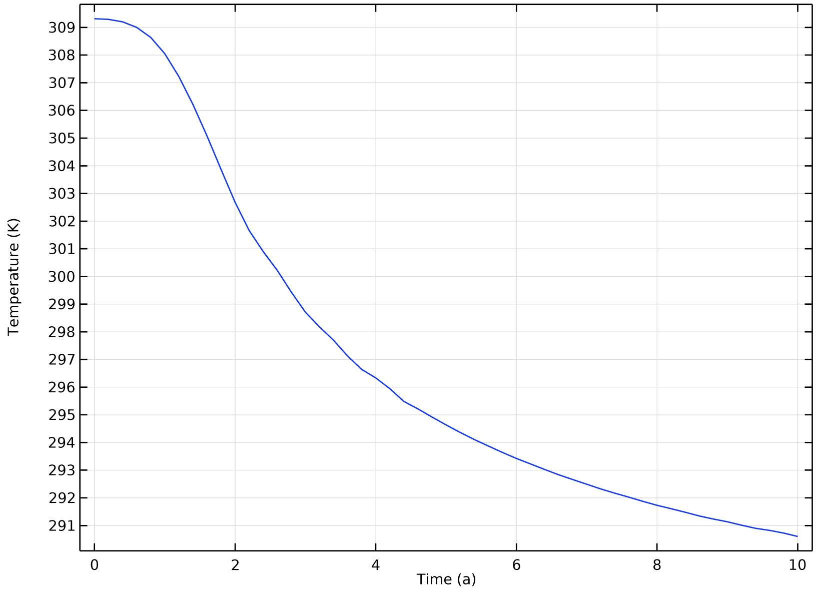


Figure 4: Production temperature over 10 years.

Application Library path: `Subsurface_Flow_Module/Heat_Transfer/geothermal_doublet`

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>Heat Transfer in Porous Media (ht)**.

5 Click **Add**.

6 Click **Done**.

ROOT

Start with loading the parameterized geometry sequence into the model.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `geothermal_doublet_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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

The geometry parameters that are used were added to the **Parameters** list automatically. Add a few more parameters that are used to set up the physics interfaces.

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
r_bore	1[m]	1 m	Borehole skinzone radius
pump	120[l/s]	0.12 m³/s	Pumping rate
deltaH	1[mm/m]	0.001	Hydraulic head gradient
T_top	283[K]	283 K	Surface temperature
T_inj	278[K]	278 K	Injection temperature
delta_Tz	0.03[K/m]	0.03 K/m	Geothermal gradient
d_f	0.2[cm]	0.002 m	Fracture thickness
f_f	1.6	1.6	Fracture roughness factor

Selections help to improve the whole modeling process. Define them now and use them later where needed.

DEFINITIONS

Injection well

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Selections>Explicit**.

3 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

6 Select Edge 19 only.

7 In the **Label** text field, type `Injection well`.

Production well

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Edge 37 only.

5 In the **Label** text field, type `Production well`.

Top layer

1 In the **Definitions** toolbar, click **Explicit**.

2 Select Domains 3 and 6 only.

3 In the **Settings** window for **Explicit**, type `Top layer` in the **Label** text field.

Middle layer

1 In the **Definitions** toolbar, click **Explicit**.

2 Select Domains 2 and 5 only.

3 In the **Settings** window for **Explicit**, type `Middle layer` in the **Label** text field.

Bottom layer

1 In the **Definitions** toolbar, click **Explicit**.

2 Select Domains 1 and 4 only.

3 In the **Settings** window for **Explicit**, type `Bottom layer` in the **Label** text field.

Fracture

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 21 only.

5 Select the **Group by continuous tangent** check box.

6 In the **Label** text field, type `Fracture`.

Outer boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 7 only.

5 Select the **Group by continuous tangent** check box.

6 Select Boundaries 1, 2, 4, 5, 7, 8, 11–13, 16, 17, 19, 22, and 25–29 only.

7 In the **Label** text field, type `Outer boundaries`.

Add materials to the materials node, but do not define their properties at this point. After setting up the physics interface COMSOL Multiphysics automatically detects which material properties are needed.

MATERIALS

Top Layer

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

2 In the **Settings** window for **Material**, type `Top Layer` in the **Label** text field.

Middle layer

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Middle layer` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Middle layer**.

Bottom layer

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Bottom layer` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Bottom layer**.

Fracture

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Fracture` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Fracture**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

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

4 Click **Add to Component** in the window toolbar.

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

DARCY'S LAW (DL)

Now, set up the physics interfaces.

Fluid and Matrix Properties 1

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

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

3 From the **Fluid material** list, choose **Water, liquid (mat5)**.

Fracture Flow 1

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

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

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

Fluid and Fracture Properties 1

1 In the **Model Builder** window, expand the **Fracture Flow 1** node, then click **Fluid and Fracture Properties 1**.

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

3 From the **Fluid material** list, choose **Water, liquid (mat5)**.

4 Locate the **Fracture Properties** section. From the **Permeability model** list, choose **Cubic law**.

5 In the f_f text field, type `f_f`.

Aperture 1

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

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

3 In the d_f text field, type `d_f`.

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 **Outer boundaries**.

4 Locate the **Hydraulic Head** section. In the H_0 text field, type `-deltaH*x`.

Use the **Well** feature to define the injection and production side of the doublet.

Well 1

1 In the **Physics** toolbar, click **Edges** and choose **Well**.

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

3 From the **Selection** list, choose **Injection well**.

4 Locate the **Well** section. In the d_w text field, type `2*r_bore`.

5 From the **Specify** list, choose **Mass flow**.

6 Locate the **Mass Flow** section. In the M_0 text field, type `pump*dl.rho`.

The expression `dl.rho` refers to the water density that is defined by Darcy's Law interface.

Well 2

1 In the **Physics** toolbar, click **Edges** and choose **Well**.

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

3 From the **Selection** list, choose **Production well**.

4 Locate the **Well** section. In the d_w text field, type `2*r_bore`.

5 From the **Well type** list, choose **Production**.

6 From the **Specify** list, choose **Mass flow**.

7 Locate the **Mass Flow** section. In the M_0 text field, type `pump*dl.rho`.

HEAT TRANSFER IN POROUS MEDIA (HT)

Fluid 1

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

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

3 From the **u** list, choose **Darcy's velocity field (dl)**.

4 Locate the **Heat Conduction, Fluid** section. From the k_{f} list, choose **User defined**. In the associated text field, type `mat5.def.k(T)`.

5 Locate the **Thermodynamics, Fluid** section. From the ρ_{f} list, choose **User defined**. In the associated text field, type `mat5.def.rho(T)`.

6 From the $C_{p,\mathrm{f}}$ list, choose **User defined**. In the associated text field, type `mat5.def.Cp(T)`.

7 From the γ list, choose **User defined**. In the associated text field, type `mat5.def.gamma_w(T)`.

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**.

Initial Values 1

Define the geothermal gradient as initial values.

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 `T_top-0.03[K/m]*z`.

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 **Outer boundaries**.

4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T_top-delta_Tz*z`.

Fracture 1

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

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

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

4 Locate the **Shell Properties** section. From the **Shell type** list, choose **Nonlayered shell**. In the L_{th} text field, type `d_f`.

5 Locate the **Fluid Material** section. From the list, choose **Water, liquid (mat5)**.

6 Locate the **Heat Convection** section. From the $\mathbf{u}$ list, choose **Darcy's velocity field (dl)**.

7 Locate the **Porous Material** section. From the θ_p list, choose **Volume fraction (dl/ff1/dlm1)**.

8 Locate the **Heat Conduction, Porous Matrix** section. From the k_p list, choose **User defined**. In the associated text field, type `3`.

9 Locate the **Thermodynamics, Porous Matrix** section. From the ρ_p list, choose **User defined**. In the associated text field, type `1.2e3`.

10 From the $C_{p,p}$ list, choose **User defined**. In the associated text field, type `800`.

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 **Edge Selection** section.

3 From the **Selection** list, choose **Injection well**.

4 Locate the **Line Heat Source** section. In the Q_l text field, type `mat5.def.Cp*dl.well1.Ml*(T_inj-T)`.

5 Locate the **Heat Source Radius** section. Select the **Specify heat source radius** check box.

6 In the R text field, type `r_bore`.

MATERIALS

Now you can define the material properties.

Top Layer (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Top Layer (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	0.1	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1e-10[cm^2]	m²	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	2	W/(m·K)	Basic

Property	Variable	Value	Unit	Property group
Density	rho	1300	kg/m³	Basic
Heat capacity at constant pressure	Cp	900	J/(kg·K)	Basic

Middle layer (mat2)

1 In the **Model Builder** window, click **Middle layer (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	0.4	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1e-6[cm^2]	m²	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	3	W/(m·K)	Basic
Density	rho	1900	kg/m³	Basic
Heat capacity at constant pressure	Cp	850	J/(kg·K)	Basic

Bottom layer (mat3)

1 In the **Model Builder** window, click **Bottom layer (mat3)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	0.3	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1e-7[cm^2]	m²	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	3.5	W/(m·K)	Basic

Property	Variable	Value	Unit	Property group
Density	rho	2300	kg/m³	Basic
Heat capacity at constant pressure	Cp	850	J/(kg·K)	Basic

Fracture (mat4)

1 In the **Model Builder** window, click **Fracture (mat4)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	0.6	1	Basic

MESH 1

Adjust the default mesh settings slightly to make sure, that the injection and production well are resolved properly.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Locate the **Mesh Settings** section. From the **Sequence type** list, choose **User-controlled mesh**.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 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 **Edge**.

4 Select Edges 19 and 37 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type 4.

9 Click **Build All**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Heat Transfer in Porous Media (ht)**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** check box.

3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Darcy's Law (dl)>Well 1** and **Component 1 (comp1)>Darcy's Law (dl)>Well 2**.

4 Click **Disable**.

This first stationary step results in a groundwater flow field which serves as a starting point for the subsequent time dependent analysis of the performance of the geothermal doublet.

Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>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(0,0.2,10)`.

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 **Maximum step constraint** list, choose **Constant**.

Constraining the maximum time step ensures that the transient behavior of the whole system is resolved properly.

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

RESULTS

COMSOL Multiphysics automatically creates four default plots. Modify the pressure plot to create Figure 2.

Selection 1

1 In the **Model Builder** window, expand the **Results>Pressure (dl)** node.

2 Right-click **Surface** and choose **Selection**.

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

4 From the **Selection** list, choose **All boundaries**, and remove the top and front boundaries, which corresponds to:

5 Select Boundaries 3, 6, 9, 11–15, 17, 18, 20, 21, 23, and 25–29 only.

6 In the **Pressure (dl)** toolbar, click **Plot**.

Cut Plane 1

Create the plot in Figure 3 as follows:

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

4 In the **Z-coordinate** text field, type `-880`.

Cut Plane 2

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Y-coordinate** text field, type `250`.

Temperature and Flow Field

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature and Flow Field` in the **Label** text field.

Surface 1

1 Right-click **Temperature and Flow Field** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

5 Locate the **Expression** section. In the **Expression** text field, type z.

6 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

Selection 1

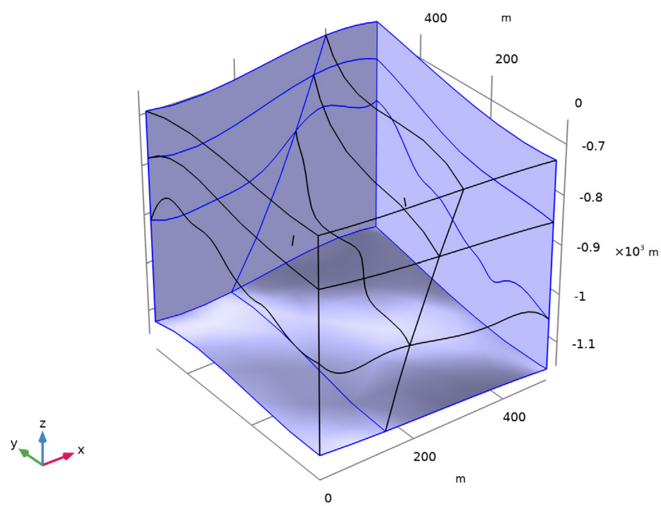
1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Outer boundaries**.

Modify this selection to obtain the figure below. This corresponds to the following step:

4 Select Boundaries 3, 11–13, 15, 17, and 25–29 only.



Surface 2

1 In the **Model Builder** window, right-click **Temperature and Flow Field** and choose **Surface**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

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

Temperature and Flow Field

In the **Model Builder** window, click **Temperature and Flow Field**.

Streamline Surface 1

1 In the **Temperature and Flow Field** toolbar, click **More Plots** and choose **Streamline Surface**.

2 In the **Settings** window for **Streamline Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 1**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Streamline Positioning** section. In the **Points** text field, type 30.

Color Expression 1

Right-click **Streamline Surface 1** and choose **Color Expression**.

Streamline Surface 2

1 In the **Model Builder** window, under **Results>Temperature and Flow Field** right-click **Streamline Surface 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 2**.

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

5 In the **Temperature and Flow Field** toolbar, click **Plot**.

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

Evaluate the production temperature and compare with Figure 4.

Line Average 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Average>Line Average**.

2 Select Edge 37 only.

3 In the **Settings** window for **Line Average**, click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Production Temperature

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Production Temperature` in the **Label** text field.