Created in COMSOL Multiphysics 5.6

Flow in a Fractured Reservoir

Introduction

This model shows how to set up a simulation of flow through a fractured reservoir. The reservoir includes a discrete fracture network (DFN), where the fractures have a randomized distribution of position, size, orientation and aperture. The model uses the Discrete Fracture Network add-in to create randomized fractures in an existing geometry.

Model Definition

The model geometry is depicted in Figure 1.

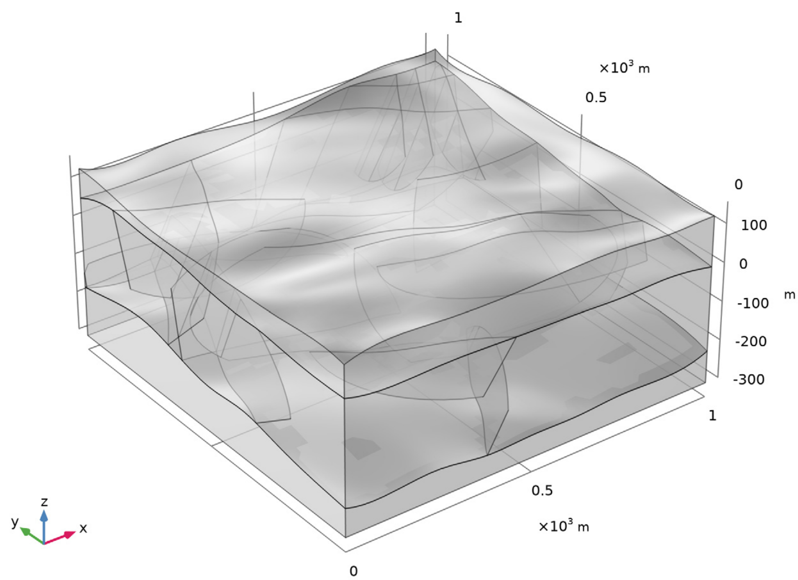


Figure 1: Model geometry.

The fracture network in the middle layer is created using the Discrete Fracture Network — 3D add-in. It serves as an example of how to use the add-in. The fracture positions follow a uniform random distribution function, their sizes follow a power-law distribution function, and their orientations follow a Fisher distribution. Details about these distribution functions can be found in the documentation of the add-in. The model is designed in such a way that the results are always identical provided that you follow the instructions. The DFN is reproducible because the same random seed is used for all distributions. The individual steps can nevertheless be applied to basically any DFN. Since the add-in can be used to create DFNs of any complexity, which places high demands on the processing of the geometry, it is recommended to use the CAD kernel, which is why this is also the case here.

Beside the geometry, the add-in also generates variables for the aperture d_f (m) of the fractures and uses them in a fracture flow feature according to the cubic law. It defines the fracture permeabilty as

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

with f_f being the fracture's friction factor. The flow is driven by the gradient of the pressure p (Pa) only, which is described by Darcy's law

$$\mathbf{u} = -\frac{\kappa}{\mu}(\nabla p + \rho \mathbf{g})$$

together with mass conservation

$$\nabla \cdot (\rho \mathbf{u}) = 0$$

with κ (m^2) being the porous matrix permeability, and μ (Pa·s) and ρ (kg/m^3) the dynamic viscosity and density of water, respectively. In the fractures, the tangential form of Darcy's Law is the governing equation.

From above, the reservoir is fed by a precipitation rate and in the lower layer there is a slow flow through a slight gradient in the hydraulic head.

Results and Discussion

The resulting velocity distribution is shown in Figure 2. Comparing the velocity inside the porous matrix with the velocity field inside the fractures shows that the flow mainly occurs through the fractures.

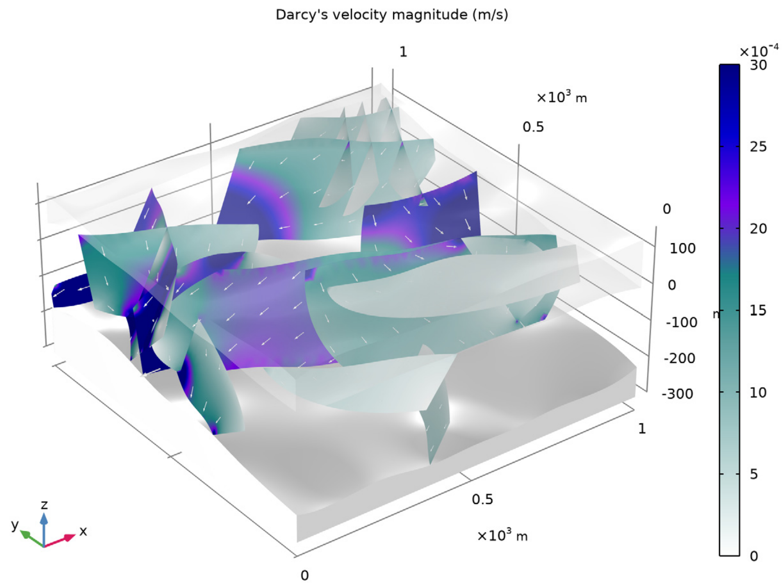


Figure 2: Velocity distribution in the reservoir.

Application Library path: `Subsurface_Flow_Module/Fluid_Flow/fractured_reservoir_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>Porous Media and Subsurface Flow>Darcy's Law (dl)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
3 From the **Geometry representation** list, choose **CAD kernel**.

   Using the CAD kernel is recommended for modeling 3D discrete fracture networks.
4 In the **Geometry** toolbar, click **Insert Sequence**.
5 Browse to the model's Application Libraries folder and double-click the file `fractured_reservoir_flow_geom_sequence.mph`.
6 In the **Geometry** toolbar, click **Build All**.

For generating a fracture network consisting of fractures that use randomized distribution functions for size, position and orientation, import the Discrete Fracture Network - 3D add-in.

In the **Home** toolbar, click **Windows** and choose **Add-in Libraries**.

ADD-IN LIBRARIES

1 In the **Add-in Libraries** window, click **Done**.
2 In the **Developer** toolbar, click **Add-ins** and choose **Discrete Fracture Network - 3D>Discrete Fracture Network - 3D**.

GLOBAL DEFINITIONS

Discrete Fracture Network - 3D 1

1 In the **Model Builder** window, under **Global Definitions** click **Discrete Fracture Network - 3D 1**.
2 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **General** section.

3 In the **Number of Fractures** text field type 10.

Because a geometry is present the add-in detects its bounding box and uses it per default.

4 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Size** section.

5 Enter the following settings:

Parameter	Value
Minimum axis length	200
Maximum axis length	500
Power law exponent	2.2

6 Click the **Use random seed** check box to get a reproducible size distribution. Whenever the add-in uses these size parameters and this random seed the size distribution and position is identical.

7 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Orientation** section.

8 Enter the following settings:

Parameter	Value
Strike	30
Dip	85
Dispersion coefficient	50

The dispersion coefficient determines the variance around the strike and dip angle. The larger the value, the smaller the deviation from the given angles.

9 Click the **Use random seed** check box to get a reproducible orientation distribution.

10 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Properties** section.

11 In the **Proportionality factor** text field enter 1e-5.

GEOMETRY 1

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

2 Click **Add**.

3 Click the **Wireframe Rendering** button in the **Graphics** toolbar to get a better view.

The fractures should only cross the middle layer. Modify the geometry sequence manually to satisfy this requirement.

Partition Objects 1 (dfnpar1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Partition Objects 1 (dfnpar1)**.
2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
3 Find the **Objects to partition** subsection. Select the **Activate Selection** toggle button.
4 In the list, choose **spl1(3)** and **spl1(4)**.
5 Click **Remove from Selection**.
6 Select the object **spl1(2)** only.
7 Click **Build Selected**.

Add another fracture set with different values.

GLOBAL DEFINITIONS

Discrete Fracture Network - 3D 1

1 In the **Model Builder** window, under **Global Definitions** click **Discrete Fracture Network - 3D 1**.
2 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **General** section.
3 In the **Number of Fractures** text field type 6.
4 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Size** section.
5 Enter 19820308 for the random seed to create a new size distribution.
6 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Orientation** section.
7 Enter the following settings:

Parameter	Value
Strike	125
Dip	70

8 Enter 19820308 for the random seed.
9 Click **Add** to add a second fracture set.

GEOMETRY 1

Again, modify the geometry sequence.

Partition Objects 2 (dfnpar2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Partition Objects 2 (dfnpar2)**.

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

3 Find the **Objects to partition** subsection. Select the **Activate Selection** toggle button.

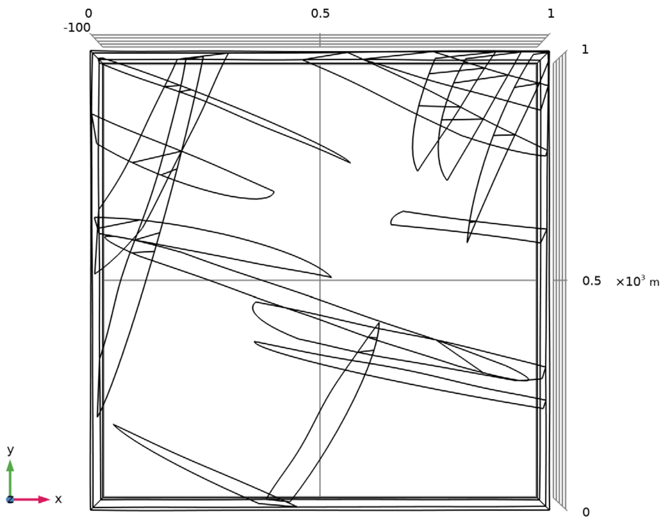
4 In the list, choose **spl1(3)** and **spl1(4)**.

5 Click **Remove from Selection**.

6 Select the object **dfnpar1** only.

7 Click **Build Selected**.

8 Click the **Go to XY View** button in the **Graphics** toolbar and compare with the image below.



9 Click the **Go to Default View** button in the **Graphics** toolbar to return to the default view.

Next, add the materials. The top and bottom layers consist of a permeable material, whereas the center layer has a comparatively low permeability. The values are specified later after the physics is set up.

MATERIALS

High permeable

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 `High permeable` in the **Label** text field.

Low permeable

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

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

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

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

DEFINITIONS

The surface is exposed to the environment. Create ambient conditions and define a precipitation rate.

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 Conditions** section.

3 In the $P_{0,\mathrm{amb}}$ text field, type `550[mm/a]`.

DARCY'S LAW (DL)

Next, set up the physics. Start with activating gravity. Define the reference position as the lowest point of the reservoir.

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 **Gravity Effects** section.

3 Select the **Include gravity** check box.

Gravity 1

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

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

3 Select the **Specify reference position** check box.

4 Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as

0	x

0	y
-300	z

Fluid and Matrix Properties 1

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

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 (mat3)**.

Fluid and Fracture Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Darcy's Law (dl)>Fracture Flow 2** 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 (mat3)**.

The add-in automatically created aperture variables under the **Definitions** node, which are used by the cubic law.

MATERIALS

It remains to set up the missing material properties.

High permeable (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **High permeable (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	`.25`	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	`1000[mD]`	m²	Basic

Low permeable (mat2)

1 In the **Model Builder** window, click **Low permeable (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	`0.1`	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	`20[mD]`	m²	Basic

DARCY'S LAW (DL)

Next, define the boundary conditions.

Precipitation 1

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

2 Select Boundary 10 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 the **Slope correction** check box.

Hydraulic Head 1

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

2 Select Boundaries 1, 2, 24, and 67 only.

3 In the **Settings** window for **Hydraulic Head**, locate the **Hydraulic Head** section.

4 In the H_0 text field, type `0.5[m/km]*x`.

MESH 1

Next, set up the mesh. For discrete fracture networks the mesh may require manual modifications due to the complexity of the geometry. Start by meshing the domain where the fractures are located.

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

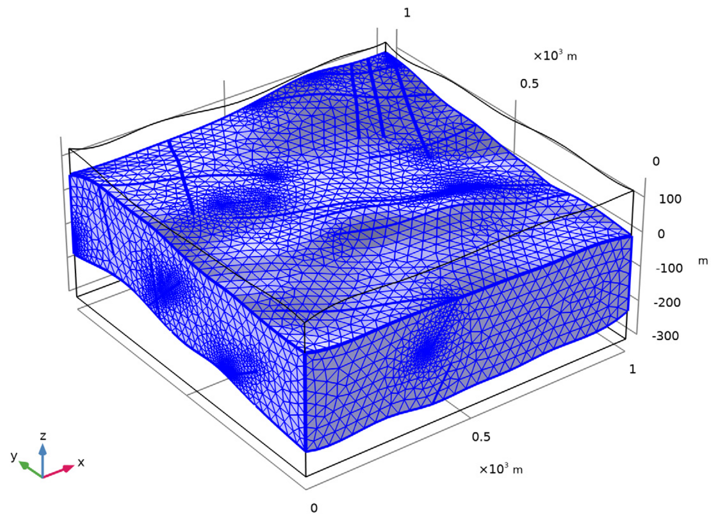
Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Middle**.

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 **Predefined** list, choose **Extra fine**.

4 Click Build Selected.



The mesh contains small elements around fine details resulting from the fractures.

Free Tetrahedral 2

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, 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** check box.

4 In the **Home** toolbar, click **Compute**.

Create a new selection first.

DEFINITIONS

Fractures

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

2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Fracture network (3D Fracture Network 1 1)** and **Fracture network (3D Fracture Network 2 1)**.

6 Click **OK**.

7 In the **Settings** window for **Union**, type `Fractures` in the **Label** text field.

RESULTS

Volume Average 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Derived Values** and choose **Average>Volume Average**.

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

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

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

Expression	Unit	Description
dl.U	m/s	Darcy's velocity magnitude

6 Click **Evaluate**.

The velocity is around 2e-9 m/s.

Surface Average 2

1 In the **Results** toolbar, click **More Derived Values** and choose **Average> Surface Average**.

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

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

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

Expression	Unit	Description
dl.U	m/s	Darcy's velocity magnitude

5 Click **Evaluate (Table 1 - Volume Average 1)**.

TABLE

1 Go to the **Table** window.

The velocity in the fractures is around 1.7mm/s and thus orders of magnitudes larger than in the matrix.

Create the plot of the Darcy velocity field (Figure 2).

RESULTS

Velocity

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

2 In the **Settings** window for **3D Plot Group**, type `Velocity` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Darcy's velocity magnitude (m/s)` .

Surface 1

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

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

3 In the **Expression** text field, type `dl.U`.

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

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `3e-3`.

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 **Fractures**.

Volume 1

1 In the **Model Builder** window, right-click **Velocity** and choose **Volume**.

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

3 In the **Expression** text field, type `dl.U`.

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

Selection 1

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

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

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

Volume 2

In the **Model Builder** window, under **Results>Velocity** right-click **Volume 1** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Volume 2** node, then click **Selection 1**.

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

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

Transparency 1

1 In the **Model Builder** window, right-click **Volume 2** and choose **Transparency**.

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

3 In the **Transparency** text field, type `0.8`.

Arrow Surface 1

In the **Model Builder** window, right-click **Velocity** and choose **Arrow Surface**.

Selection 1

1 In the **Model Builder** window, right-click **Arrow Surface 1** and choose **Selection**.

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

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

Arrow Surface 1

1 In the **Model Builder** window, click **Arrow Surface 1**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 From the **Arrow length** list, choose **Logarithmic**.

4 Select the **Scale factor** check box.

5 In the associated text field, type `3000`.

6 From the **Color** list, choose **White**.