



# Flow in a Fractured Reservoir

## Introduction

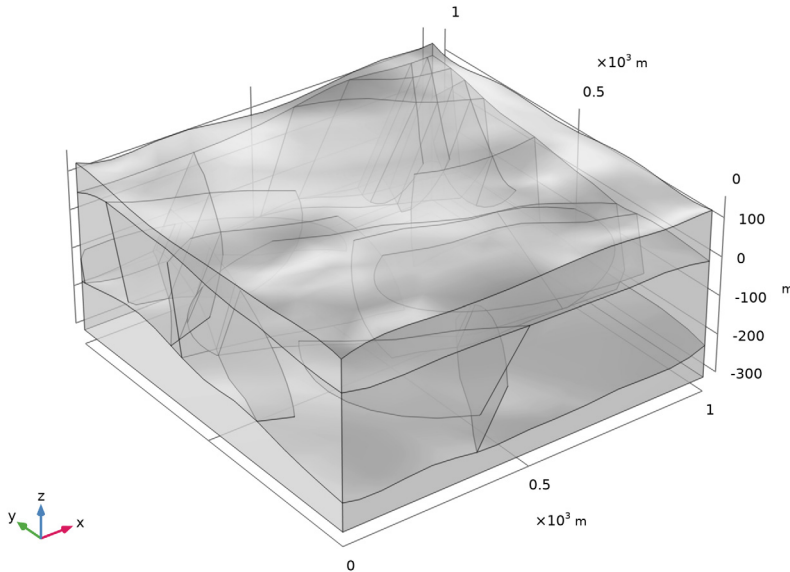
---

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 fracture networks of any complexity, which places high demands on the processing of the geometry, it is recommended to use the CAD kernel.

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

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

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  $\kappa$  ( $m^2$ ) being the porous matrix permeability, and  $\mu$  (Pa·s) and  $\rho$  ( $kg/m^3$ ) 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.

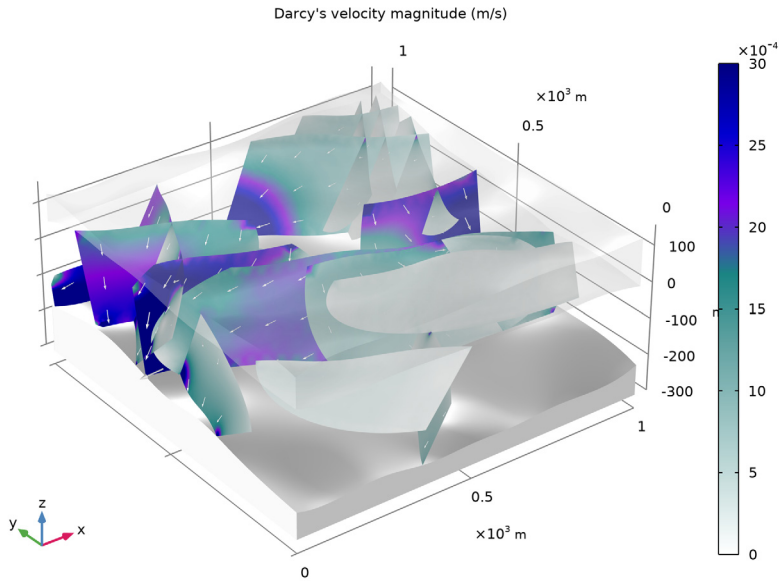


Figure 2: Velocity distribution in the reservoir.

---

**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 I



Using the CAD kernel is recommended for modeling 3D discrete fracture networks, especially with increasing complexity of the network. A note appears at the bottom of the add-in's settings window if it is not activated.

- 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 `fractured_reservoir_flow_geom_sequence.mph`.
- 3 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 **Model Builder** window, under **Component 1 (comp1)** click **Geometry I**.
- 2 In the **Add-in Libraries** window, in the tree, select the check box for the node **Subsurface Flow Module>discrete\_fracture\_network\_3d** (if it is not already selected).
- 3 Click  **Done**.
- 4 In the **Developer** toolbar, click  **Add-ins** and choose **Discrete Fracture Network — 3D>Discrete Fracture Network — 3D**.

## GLOBAL DEFINITIONS

*Discrete Fracture Network — 3D I*

- 1 In the **Model Builder** window, under **Global Definitions** click **Discrete Fracture Network — 3D I**.

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 determines the bounding box of this geometry and also detects if domain selections are available.

4 From the **Selection** menu choose **Middle** which is a selection already defined in the geometry sequence.

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

6 Enter the following settings:

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

7 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.

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

9 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.

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

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

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

13 Click **Add**.

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

Add another fracture set with different values.

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


16 In the **Number of Fractures** text field type 6.

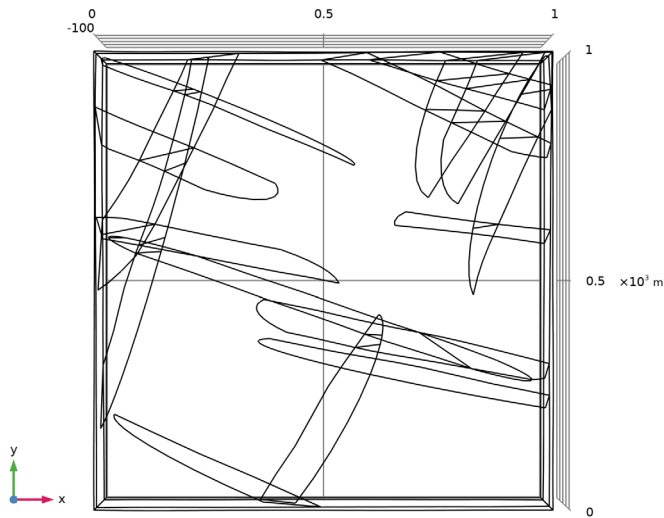
- 17 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Size** section.
- 18 Enter 19820308 for the random seed to create a new size distribution.
- 19 In the **Settings** window for **Discrete Fracture Network - 3D**, locate the **Orientation** section.
- 20 Enter the following settings:


Parameter	Value
Strike	125
Dip	70

- 21 Enter 19820308 for the random seed.
- 22 Click **Add** to add a second fracture set.

**GEOMETRY I**



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



- 2 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.

## 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 Global Materials** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

## DEFINITIONS

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

### *Ambient Properties 1 (ampr1)*

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.
- 2 Right-click **Definitions** and choose **Shared Properties>Ambient Properties**.
- 3 In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.
- 4 In the  $P_{0,amb}$  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  $\mathbf{r}_{ref}$  vector as

0	x
0	y
-300	z



### *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 (mat1)**.

### *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 (mat1)**.

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*

- 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 *High permeable* in the **Label** text field.
- 3 Locate the **Porosity** section. In the  $\epsilon_p$  text field, type 0.25.
- 4 Locate the **Homogenized Properties** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1000[mD]	m <sup>2</sup>	Basic


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

### *Low permeable*

- 1 Right-click **Materials** and choose **More Materials>Porous Material**.
- 2 Select **Domain 2** only.

- 3 In the **Settings** window for **Porous Material**, locate the **Geometric Entity Selection** section.
- 4 From the **Selection** list, choose **Middle**.
- 5 In the **Label** text field, type `Low permeable`.
- 6 Locate the **Porosity** section. In the  $\epsilon_p$  text field, type `0.1`.
- 7 Locate the **Homogenized Properties** section. In the table, enter the following settings:


Property	Variable	Value	Unit	Property group
Permeability	<code>kappa_iso ; kappaii = kappa_iso, kappaij = 0</code>	<code>20 [mD]</code>	<code>m<sup>2</sup></code>	Basic

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


### DARCY'S LAW (DL)

Next, define the boundary conditions.

#### *Precipitation I*

- 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 (ampri)**.
- 5 Select the **Slope correction** check box.

#### *Hydraulic Head I*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Hydraulic Head**.
- 2 Select Boundaries 1, 2, 25, and 80 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 I

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

- 3 From the **Element size** list, choose **Finer**.
- 4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

#### *Free Tetrahedral 1*

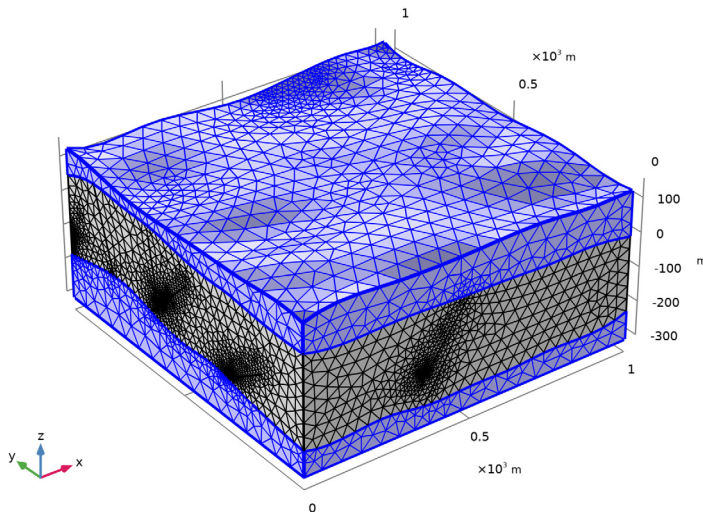
- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
- 2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose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


#### *Free Tetrahedral 2*

- 1 In the **Mesh** toolbar, click  **Free Tetrahedral**.
- 2 In the **Settings** window for **Free Tetrahedral**, click  **Build All**.



The fractures create fine details in the geometry that are difficult to handle for the mesh. Where these details are located is indicated in an information node below the feature. With a mesh fine enough, you can resolve these details well, but for the overall model, such a fine mesh is not necessary.



## 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**.

## DEFINITIONS (COMPI)

Create a new selection including all fractures.

### 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
d1.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
d1.U	m/s	Darcy's velocity magnitude

- 5 Click  next to  **Evaluate**, then choose **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.

### RESULTS

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

### 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 **d1.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 `d1.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**.

