Created in COMSOL Multiphysics 6.0

Rock Fracture Flow

Introduction

Understanding how fracture flow develops has been an active research area since the 19th century (Ref. 1). The most common model for flow in a single fracture is the so-called "cubic law", in which the fracture is represented by two parallel plates separated by a small aperture. For the cubic-law assumption, the fluid is considered viscous and incompressible, and the hydraulic conductivity of the fracture depends quadratically on the fracture's aperture:

$$K_s = \frac{\rho g}{12\mu}a^2$$

which involves the following variables:

- Fluid density, ρ
- The acceleration of gravity, g
- The fluid's dynamic viscosity, μ
- The fracture's aperture or width, a

The transmissivity of the fracture is then proportional to both the hydraulic conductivity and its aperture, giving rise to what is known as the cubic law:

$$T_s = aK_s = \frac{\rho g}{12\mu}a^3$$

The flux in the fracture is then given by the velocity

$$\mathbf{q} = -K_s\nabla H$$

which depends on the hydraulic conductivity K_s, the hydraulic head, $H = H(x, y)$, and the hydraulic gradient ∇H. The discharge per unit of fracture width can also be computed from the hydraulic gradient and transmissivity

$$Q = a|\mathbf{q}| = T_s|\nabla H|$$

These expressions are valid as long as the flow is laminar. In cases of deviation from ideal conditions (for instance, rough surfaces) the cubic law can be adjusted by a roughness factor f, so the transmissivity is

$$T_s = \frac{\rho g}{12\mu f}a^3$$

A potential flow model describing fluid movement in a rock fracture uses the Reynolds equation

$$\nabla \cdot (-T_s \nabla H) = 0$$

This example uses interpolation data calculated from a fracture with an aperture that changes with position, $a(x, y)$. The data is defined in a text file.

Model Definition

The model consists of two components: the first solves the 2D problem, while the second solves a 3D problem. For the 3D problem, the hydraulic gradient ∇H is given with tangential derivatives.

The computational domain is rectangular. Set a hydraulic head of 20 mm at the upper boundary and 0 mm at the lower boundary. This creates a hydraulic head difference of 20 mm that drives the fluid flow. Both the left and right boundaries are impermeable.

The COMSOL installation includes a text file `aperture_data.txt`, which contains the sample aperture data in the form of a 100-by-100 matrix. This synthetically generated dataset corresponds to an aperture with a fractal dimension of 2.6. You import the aperture data into the COMSOL Multiphysics physics interface by defining an interpolation function, which you then use as the aperture $a(x, y)$ in the cubic law equation.

Results

The plot in Figure 1 shows the velocity magnitude using colored surface data and the aperture data as the z-coordinate (the height).

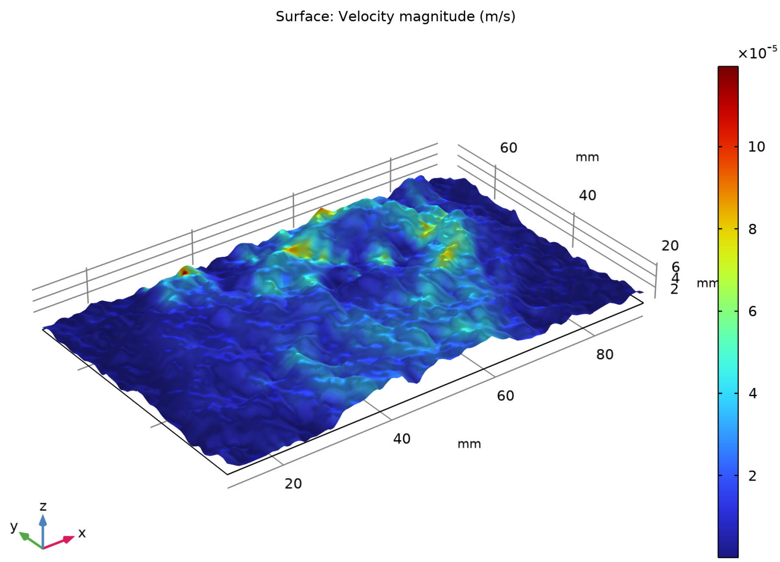


Figure 1: The velocity magnitude and aperture distribution.

The plot in Figure 2 is a visualization of the 3D model.

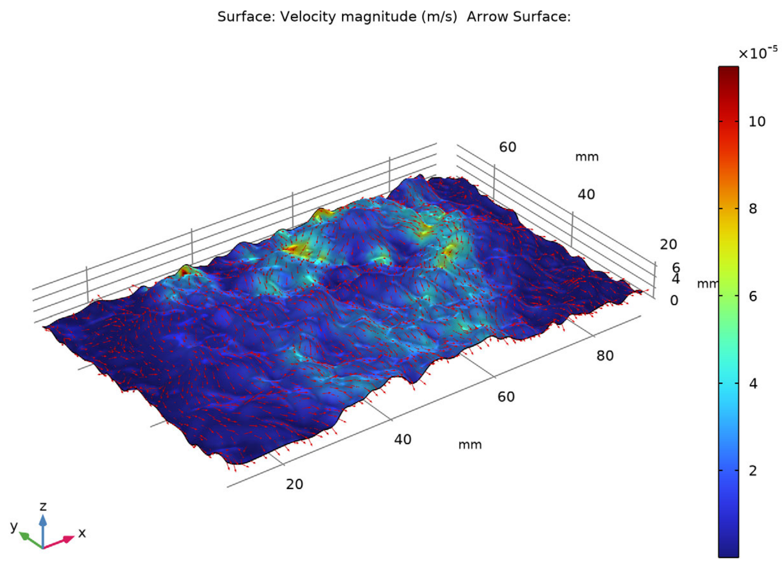


Figure 2: The velocity magnitude, direction and aperture data in the 3D model.

Notes About the COMSOL Implementation

The base package COMSOL Multiphysics does not include a physics interface that specifically solves for the cubic law, but there are two interfaces under the Mathematics branch you can use: the Convection and Diffusion interface for the 2D model, or the Coefficient Form Boundary PDE for the 3D case. Implement the Reynolds equation by setting the different coefficients for these PDEs. You can rename the dependent variable as `H` when adding the 2D physics interface, and `H2` when adding the 3D physics interface. With a license for the *Subsurface Flow Module* or *Porous Media Flow Module* a dedicated interface for modeling fracture flow including the cubic law is available.

Reference

1. P. Witherspoon and others, “Validity of cubic law for fluid flow in a deformable rock fracture,” *Lawrence Berkeley National Laboratory. LBNL Paper LBL-9557*, 1979.

Application Library path: `COMSOL_Multiphysics/Geophysics/rock_fracture_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 **2D**.
2 In the **Select Physics** tree, select **Mathematics>Classical PDEs>Convection-Diffusion Equation (cdeq)**.
3 Click **Add**.
4 In the **Dependent variable** text field, type `H`.
5 Click **Select Dependent Variable Quantity**.
6 In the **Physical Quantity** dialog box, type `head` in the text field.
7 Click **Filter**.
8 In the tree, select **Transport>Head (m)**.
9 Click **OK**.
10 In the **Model Wizard** window, click **Select Source Term Quantity**.
11 In the **Physical Quantity** dialog box, type `timechangeinpressurehead` in the text field.
12 Click **Filter**.
13 In the tree, select **Transport>Time change in pressure head (m/s)**.
14 Click **OK**.
15 In the **Model Wizard** window, click **Study**.
16 In the **Select Study** tree, select **General Studies>Stationary**.
17 Click **Done**.

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
rho	1000[kg/m^3]	1000 kg/m³	Fluid density
mu	0.001[Pa*s]	0.001 Pa·s	Fluid dynamic viscosity

Interpolation 1 (int1)

Define an interpolation function using the aperture data available in a file.

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `rock_fracture_flow_aperture_data.txt`.

6 Click **Import**.

7 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
data	1

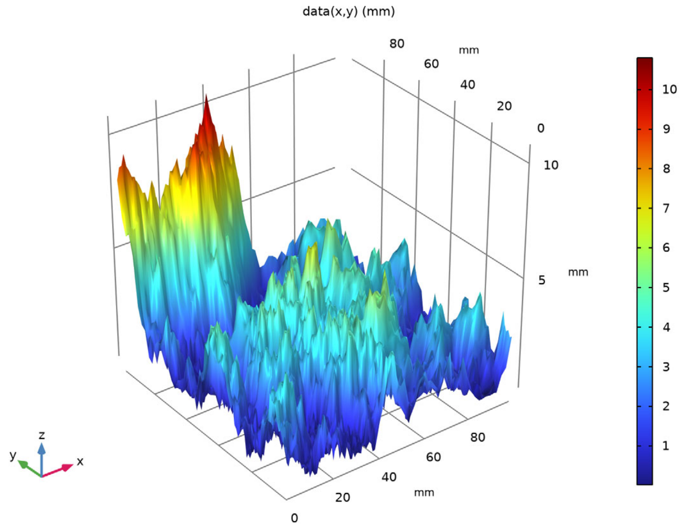
8 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
Argument 1	mm
Argument 2	mm

9 In the **Function** table, enter the following settings:

Function	Unit
data	mm

10 Click **Plot**. Compare with the image below.



GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

The model geometry is simply a rectangle.

Rectangle 1 (r1)

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type 80.
4. In the **Height** text field, type 50.
5. Locate the **Position** section. In the **x** text field, type 10.
6. In the **y** text field, type 20.
7. Click **Build Selected**.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
a	data(x,y)/1000	m	Aperture
Ks	a^2*rho*g_const/mu	m/s	Hydraulic conductivity
Ts	Ks*a	m²/s	Transmissivity
u	-Ks*Hx	m/s	Velocity, x component
v	-Ks*Hy	m/s	Velocity, y component
U	sqrt(u^2+v^2)	m/s	Velocity magnitude

CONVECTION-DIFFUSION EQUATION (CDEQ)

Convection-Diffusion Equation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Convection-Diffusion Equation (cdeq)** click **Convection-Diffusion Equation 1**.

2 In the **Settings** window for **Convection-Diffusion Equation**, locate the **Diffusion Coefficient** section.

3 In the c text field, type Ts.

4 Locate the **Source Term** section. In the f text field, type 0.

5 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type 0.

Next, define the boundary conditions.

Dirichlet Boundary Condition 1

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

2 Select Boundary 2 only.

Dirichlet Boundary Condition 2

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Dirichlet Boundary Condition** section.

4 In the *r* text field, type `20[mm]`.

MESH 1

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 **Extra fine**.

STUDY 1

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

RESULTS

Velocity (2D)

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

Surface 1

1 In the **Model Builder** window, expand the **Velocity (2D)** node, then click **Surface 1**.

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

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

Height Expression 1

1 Right-click **Surface 1** and choose **Height Expression**.

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

3 From the **Height data** list, choose **Expression**.

4 In the **Expression** text field, type `data(x,y)`.

5 Locate the **Axis** section. Select the **Scale factor** check box.

6 In the associated text field, type `1`.

7 In the **Velocity (2D)** toolbar, click **Plot**.

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

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

GEOMETRY 2

1 In the **Settings** window for **Geometry**, locate the **Units** section.

2 From the **Length unit** list, choose **mm**.

Parametric Surface 1 (ps1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Surface**.

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

3 Find the **First parameter** subsection. In the **Minimum** text field, type `10`.

4 In the **Maximum** text field, type `90`.

5 Find the **Second parameter** subsection. In the **Minimum** text field, type `20`.

6 In the **Maximum** text field, type `70`.

7 Locate the **Expressions** section. In the **x** text field, type `s1`.

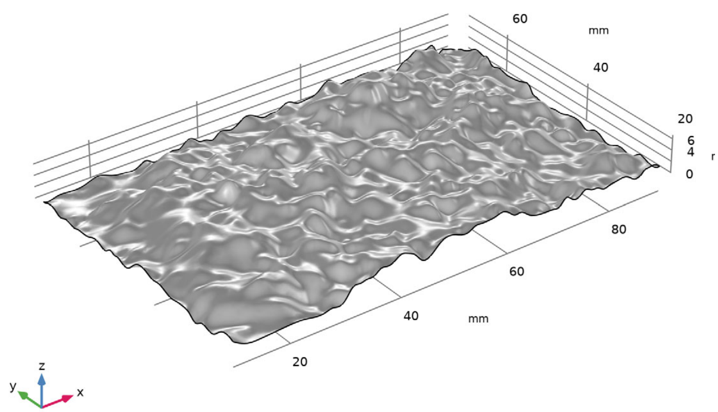
8 In the **y** text field, type `s2`.

9 In the **z** text field, type `data(s1,s2)`.

10 Locate the **Advanced Settings** section. In the **Relative tolerance** text field, type `1.0E-2`.

11 In the **Maximum number of knots** text field, type `100`.

12 Click **Build All Objects**. Compare with the image below.



DEFINITIONS (COMP2)

Variables 2

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
a	data(x,y)/1000	m	Aperture
Ks	a^2*rho*g_const/mu	m/s	Hydraulic conductivity
Ts	a*Ks	m²/s	Transmissivity
u	-Ks*H2Tx		Velocity, x component
v	-Ks*H2Ty		Velocity, y component
w	-Ks*H2Tz		Velocity, z component
U	sqrt(u^2+v^2+w^2)		Velocity magnitude

ADD PHYSICS

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

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

3 In the tree, select **Mathematics>PDE Interfaces>Lower Dimensions> Coefficient Form Boundary PDE (cb)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click to expand the **Dependent Variables** section. In the **Field name** text field, type H2.

6 In the **Dependent variables** table, enter the following settings:

H2

7 Click the **Units** tab.

8 Click **Select Dependent Variable Quantity**.

9 In the **Physical Quantity** dialog box, type head in the text field.

10 Click **Filter**.

11 In the tree, select **Transport>Head (m)**.

12 Click **OK**.

13 Go to the **Add Physics** window.

14 Locate the **Dependent Variables** section. Click **Select Source Term Quantity**.

15 In the **Physical Quantity** dialog box, select **Transport>Time change in pressure head (m/s)** in the tree.

16 Click **OK**.

17 Go to the **Add Physics** window.

18 Click **Add to Component 2** in the window toolbar.

COEFFICIENT FORM BOUNDARY PDE (CB)

Coefficient Form PDE 1

1. In the **Model Builder** window, under **Component 2 (comp2)>** **Coefficient Form Boundary PDE (cb)** click **Coefficient Form PDE 1**.
2. In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.
3. In the c text field, type `-Ts`.
4. Locate the **Source Term** section. In the f text field, type `0`.
5. Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `0`.

Dirichlet Boundary Condition 1

1. In the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.
2. Select Edge 2 only.

Dirichlet Boundary Condition 2

1. In the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.
2. Select Edge 3 only.
3. In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Dirichlet Boundary Condition** section.
4. In the r text field, type `20[mm]`.

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 **Convection-Diffusion Equation (cdeq)**.
5. Click **Add Study** in the window toolbar.
6. In the **Model Builder** window, click the root node.
7. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **3D Plot Group 2** node, then click **Surface 1**.

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

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

Arrow Surface 1

1 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Arrow Surface**.

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

3 In the **X component** text field, type u.

4 In the **Y component** text field, type v.

5 In the **Z component** text field, type w.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

7 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

8 In the **3D Plot Group 2** toolbar, click **Plot**.