

# Rock Fracture Flow

Understanding how fracture flow develops has been an active research area since the 19<sup>th</sup> 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

$$\mathbf{q} = -K_{\rm s}\nabla H$$

which depends on the hydraulic conductivity  $K_s$ , the hydraulic head, H = H(x, y), and the hydraulic gradient  $\nabla H$ . 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  $\nabla H$  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.

Surface: Velocity magnitude (m/s) Arrow Surface:



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

# Notes About the COMSOL Implementation

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.

# Reference

1. P. Witherspoon et.al., "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

- I 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.
- IO In the Model Wizard window, click 🛒 Select Source Term Quantity.
- II In the Physical Quantity dialog box, type timechangeinpressurehead in the text field.
- 12 Click 🔫 Filter.
- I3 In the tree, select Transport>Time change in pressure head (m/s).
- I4 Click OK.
- **I5** In the **Model Wizard** window, click  $\bigcirc$  **Study**.
- **I6** In the Select Study tree, select General Studies>Stationary.
- 17 Click 🗹 Done.

# GLOBAL DEFINITIONS

#### Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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)

- I In the Home toolbar, click f∞ Functions and choose Global>Interpolation.
  Define an interpolation function using the aperture data available in a file.
- 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 Arguments text field, type mm.

**9** In the **Function** text field, type mm.

IO Click 🗿 Plot.

## GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 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)

- I 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 I

- I In the Model Builder window, under Component I (compl) 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            |
|------|--------------------------|------|------------------------|
| а    | 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    | <pre>sqrt(u^2+v^2)</pre> | m/s  | Velocity magnitude     |

## CONVECTION-DIFFUSION EQUATION (CDEQ)

Convection-Diffusion Equation 1

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

- I In the Physics toolbar, click Boundaries and choose Dirichlet Boundary Condition.
- **2** Select Boundary 2 only.

Dirichlet Boundary Condition 2

I 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].

Proceed to set up the solvers.

## MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- **3** From the **Element size** list, choose **Extra fine**.

## STUDY I

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

## RESULTS

Surface 1

- I In the Model Builder window, expand the 2D Plot Group I node, then click Surface I.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type U.
- 4 In the 2D Plot Group I toolbar, click 💽 Plot.

## Height Expression 1

- I Right-click Surface I 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 2D Plot Group I toolbar, click 💽 Plot.
- **8** Click the 4 **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**

- I In the Settings window for Geometry, locate the Units section.
- 2 From the Length unit list, choose mm.

#### Parametric Surface 1 (ps1)

- I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Parametric Surface.
- 2 In the Settings window for Parametric Surface, locate the Expressions section.
- 3 In the x text field, type s1.
- **4** Locate the **Parameters** section. Find the **First parameter** subsection. In the **Minimum** text field, type **10**.
- 5 In the Maximum text field, type 90.
- 6 Locate the **Expressions** section. In the **y** text field, type **s2**.
- 7 Locate the Parameters section. Find the Second parameter subsection. In the Minimum text field, type 20.
- 8 In the Maximum text field, type 70.
- 9 Locate the Expressions section. 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.
- II In the Maximum number of knots text field, type 100.
- 12 Click 🟢 Build All Objects.
- **I3** Click the  $\longleftrightarrow$  **Zoom Extents** button in the **Graphics** toolbar.

## DEFINITIONS (COMP2)

Variables 2

- I 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            |
|------|--------------------|------|------------------------|
| а    | 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  |

| Name | Expression        | Unit | Description           |
|------|-------------------|------|-----------------------|
| W    | -Ks*H2Tz          |      | Velocity, z component |
| U    | sqrt(u^2+v^2+w^2) |      | Velocity magnitude    |

# ADD PHYSICS

- I 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 I.
- 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 Add to Component 2 in the window toolbar.
- 8 In the Home toolbar, click 🙀 Add Physics to close the Add Physics window.

# COEFFICIENT FORM BOUNDARY PDE (CB)

- I In the Settings window for Coefficient Form Boundary PDE, locate the Units section.
- 2 Click Select Dependent Variable Quantity.
- 3 In the Physical Quantity dialog box, type head in the text field.
- 4 Click 👎 Filter.
- 5 In the tree, select Transport>Head (m).
- 6 Click OK.
- 7 In the Settings window for Coefficient Form Boundary PDE, locate the Units section.
- 8 Click **Select Source Term Quantity**.
- 9 In the Physical Quantity dialog box, type timechangeinpressurehead in the text field.
- 10 Click 🔫 Filter.

II In the tree, select Transport>Time change in pressure head (m/s).

I2 Click OK.

#### Coefficient Form PDE 1

- I In the Model Builder window, under Component 2 (comp2)> Coefficient Form Boundary PDE (cb) click Coefficient Form PDE I.
- 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 I

- I In the Physics toolbar, click 🔚 Edges and choose Dirichlet Boundary Condition.
- 2 Select Edge 2 only.

#### Dirichlet Boundary Condition 2

- I 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

- I In the Home toolbar, click 2 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  $\sim\sim$  Add Study to close the Add Study window.

## STUDY 2

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

#### RESULTS

Surface 1

- I In the Model Builder window, expand the 3D Plot Group 2 node, then click Surface I.
- 2 In the Settings window for Surface, locate the Expression section.

3 In the Expression text field, type U.

# Arrow Surface 1

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

14 | ROCK FRACTURE FLOW