Created in COMSOL Multiphysics 5.6

Pore-Scale Flow

Introduction

This example uses Creeping Flow (Stokes Flow) to solve the flow in the interstices of a porous medium. The example comes from pore-scale flow experiments conducted by Arturo Keller, Maria Auset, and Sanya Sirivithayapakorn of the University of California, Santa Barbara. To produce the example geometry they used electron microscope images. This type of nonconventional pore-scale modeling with COMSOL Multiphysics sheds new light on the movement of large particulates and colloids moving through variable-pore geometries in the subsurface. Several of these researchers have published results from their COMSOL Multiphysics modeling in the publication *Water Resources Research* (Ref. 1 and Ref. 2).

Keller, Auset, and Sirivithayapakorn designed their lab experiments on the basis of scanning electron microscope (SEM) images of thinly sliced rock sections (Figure 1). They etched the geometric patterns from the images onto a solid with an elaborate process similar to the etching of silicon wafers. They then transferred these images to DXF files, which they finally imported into COMSOL Multiphysics.

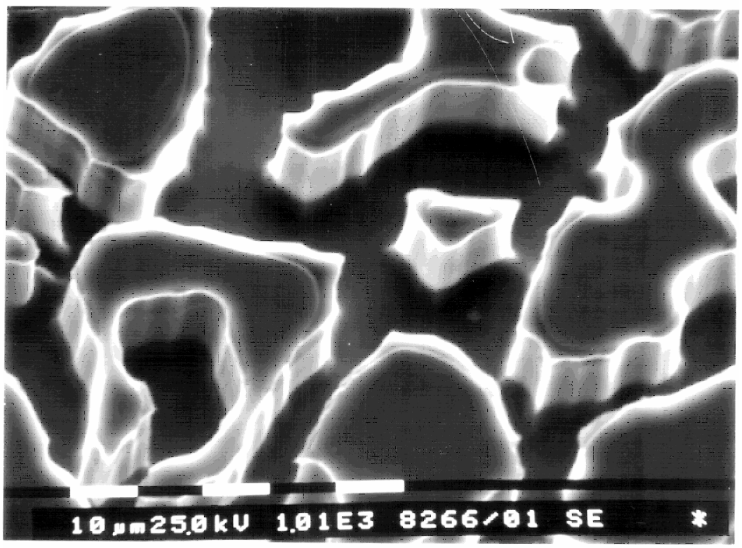

*Figure 1: Scanning electron microscope image of the repeat pattern in the silicon wafer. The scale at bottom indicates that pore throat and body dimensions are on the order of 1 μm–100 μm (*Ref. 1*).*

It is typical to represent fluid flow in the subsurface as a continuum process using average or "continuous" properties for the bulk rather than detailing the shape and orientation of each solid particle within a porous medium. Inserting the bulk properties into an equation, such as Darcy's law or Brinkman equations, gives an average flow rate for the total volume. While bulk approximations typically produce excellent estimates sufficient for considering flow over large areas, they might miss the between-grain nuances that a close-up Stokes flow analysis would give.

This exercise is divided in two models: The first model takes one of the SEM images of Keller, Auset, and Sirivithayapakorn and solves for the flow velocity and pressure drop in the pore throats using the Creeping Flow interface. The geometry is imported as a binary file and only the pore space is meshed but not the solid regions. The second model is devoted to the modeling of the whole slice, by importing the SEM image and deriving porous medium properties, such as porosity and permeability for further use in the Brinkman Equations interface.

Model Definition

The entire model covers 640 μm by 320 μm. Water moves from right to left across the geometry. The flow in the pores does not penetrate the solid grains. The inlet and outlet fluid pressures are known. Assume no flow at the top and bottom boundaries. The primary zone of interest is the rectangular region with an upper left corner at (0, 0) μm and lower right coordinates at (581.6, –265.0) μm.

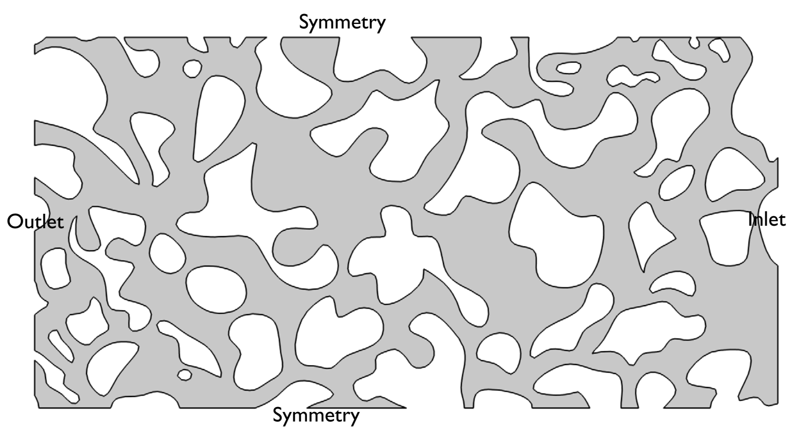

Figure 2: A 640 μm by 320 μm geometry and boundary conditions.

Since the channels are at most 0.1 mm in width and the maximum velocity is lower than 10^{-4} m/s, the maximum Reynolds number is less than 0.01. Since Reynolds number is far less than one, the example uses the Creeping Flow (Stokes Flow) interface, instead of the Laminar Flow (Navier-Stokes) interface. The fluid is considered isothermal and with constant density. Owing to the problem's small scale, the example does not consider gravity.

The Creeping Flow interface solves Stokes equations in the channels. The incompressible assumption together with the stationary condition reads

$$0 = -\nabla p + \nabla \cdot \mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)$$

$$\nabla \cdot \mathbf{u} = 0$$

here, p is the pressure (SI unit: Pa), $\mathbf{u}$ is the velocity field (SI unit: m/s), and μ is the dynamic viscosity of the fluid (SI unit: Pa·s).

At the physical boundaries, the inlet pressure and the outlet pressure are known. Velocities are zero at the grain boundaries, which implies a no-slip condition. The flow is symmetric about the top and bottom boundaries. Table 1 summarizes the boundary conditions.

TABLE 1: BOUNDARY CONDITIONS.

BOUNDARY TYPE	BOUNDARY CONDITION	VALUE
Inlet	Pressure, no viscous stress	$p = p_0$
Outlet	Pressure, no viscous stress	$p = 0$
Grain walls	Wall	no slip
Symmetry sides	Symmetry	-

Here p_0 is a specified pressure drop. Table 2 collects the relevant model data.

TABLE 2: MODEL DATA.

QUANTITY	VALUE	DESCRIPTION
ρ_0	1000 kg/m^3	Fluid density
μ_0	0.001 kg/(m·s)	Fluid dynamic viscosity
p_0	0.715 Pa	Pressure drop

Results and Discussion

Figure 3 shows the COMSOL Multiphysics solution predicted with the creeping flow analysis for the fluid velocity field in the pore spaces of a microscale porous slice. The velocity magnitude is higher in the narrowest pores than at the inlet, tending to decrease in stretches where the channels' cross-sectional area increases.

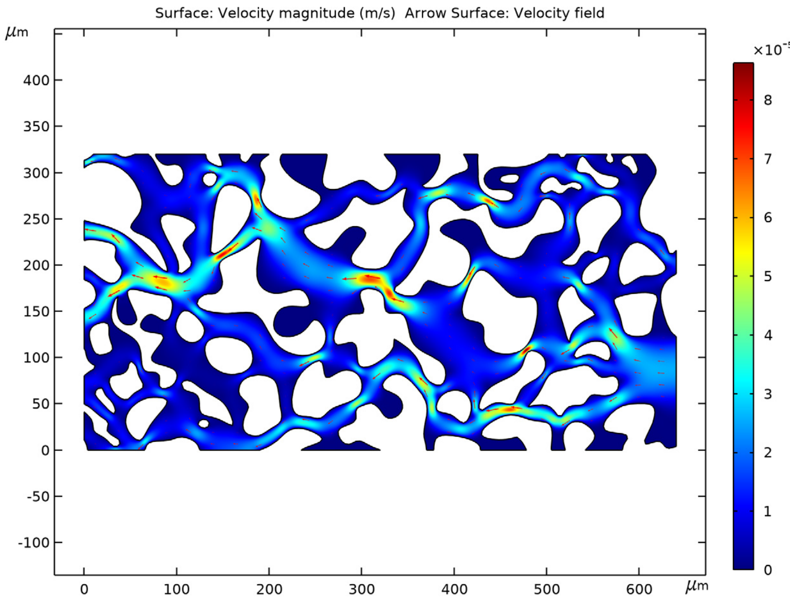

Figure 3: Surface and arrow plots of the velocity field calculated by the Creeping Flow interface.

Model Definition

The second model takes a completely different approach than the first model. Here, the scanning electron microscope image is imported and physical properties are derived from the color scale. As opposed to consumer cameras, SEM images are grayscale, but in this example, the color code is binary; see Figure 4.

Instead of solving for the creeping flow in the channels, the incompressible, stationary Brinkman equations, with the Stokes-Brinkman assumption is used

$$0 = -\nabla p + \nabla \cdot \frac{\mu}{\varepsilon_p}(\nabla \mathbf{u} + \nabla \mathbf{u}^T) - \frac{\mu}{\kappa}\mathbf{u}$$

$$\nabla \cdot \mathbf{u} = 0$$

here, p is the pressure, $\mathbf{u}$ is Darcy's velocity field, μ is the dynamic viscosity of the fluid, ε_p is the porosity, and κ is the permeability of the medium.

In order to define physical properties from the image color code, the following relations has been implemented for the porosity and permeability

$$\kappa(x, y) = \frac{\kappa_0}{100 * \mathtt{im1}(x, y) + 0.1} \tag{1}$$

$$\varepsilon_p(x, y) = 1 - 0.99 * \mathtt{im1}(x, y) \tag{2}$$

here, `im1` is a image function derived from the SEM image, which in this example ranges from 0 to 1 as a function of position. Other expressions can be implemented when importing RGB or grayscale images.

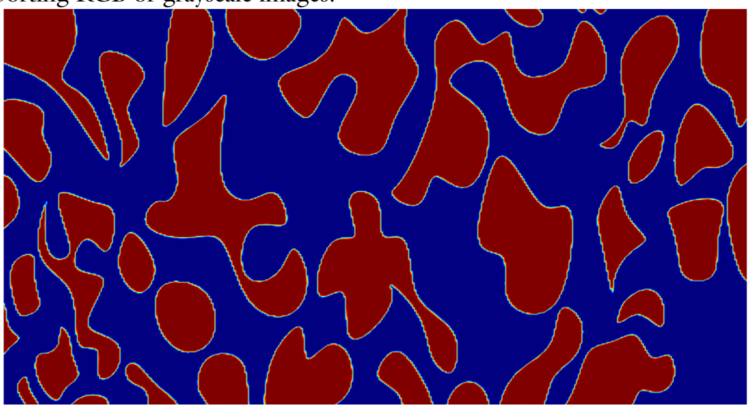

Figure 4: SEM image. The color code is blue for 0 and red for 1. COMSOL Multiphysics can handle grayscale and RGB images.

After solving Brinkman equations with the material parameters derived from the SEM image, it is possible to observe a very similar pressure and velocity profiles in the porous slice as obtained with the Creeping flow model. Compare Figure 5 with the results in Figure 3.

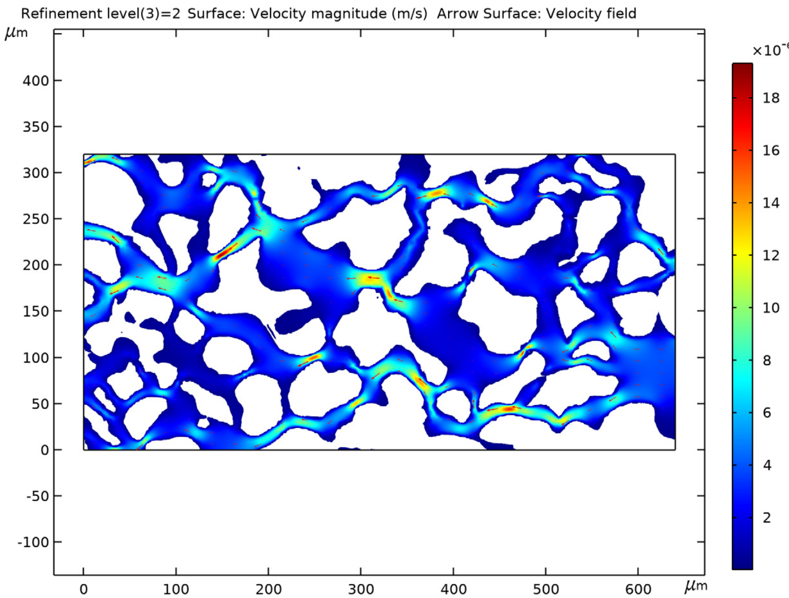

Figure 5: Surface and arrow plots of the velocity field calculated by the Brinkman Equations interface. The porosity and permeability are taken from the SEM image, as written in Equation 1 *and* Equation 2.

Notes About the COMSOL Implementation

In this model, an external image is used to infer material properties, such as porosity and permeability. The same technique can be used to derive other material properties, like density and thermal or electric conductivity.

To find out more about how to import images, see the chapter about the Definitions node in the *COMSOL Multiphysics Reference Manual.*

References

1. M. Auset and A.A. Keller, "Pore-scale Processes that Control Dispersion of Colloids in Saturated Porous Media", *Water Resources Research*, vol. 40, no. 3, W03503, 2004.

2. S. Sirivithayapakorn and A.A. Keller, "Transport of Colloids in Saturated Porous Media: A Pore-scale Observation of the Size Exclusion Effect and Colloid Acceleration", *Water Resources Research*, vol. 39, no. 4, 2003.

Application Library path: `Subsurface_Flow_Module/Fluid_Flow/pore_scale_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 **Fluid Flow>Single-Phase Flow>Creeping Flow (spf)**.

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 **Units** section.

3 From the **Length unit** list, choose **µm**.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `pore_scale_flow.mphbin`.

5 Click **Import**.

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
rho0	1000[kg/m^3]	1000 kg/m³	Fluid density
eta0	0.001[kg/(m*s)]	0.001 kg/(m·s)	Dynamic viscosity
p0	0.715[Pa]	0.715 Pa	Pressure drop

MATERIALS

Material 1 (mat1)

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

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
Density	rho	rho0	kg/m³	Basic
Dynamic viscosity	mu	eta0	Pa·s	Basic

CREEPING FLOW (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Creeping Flow (spf)** and choose **Inlet**.

2 Select Boundaries 2231 and 2232 only (the straight edges on the far right).

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

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type p0.

Outlet 1

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

2 Select Boundaries 1, 4, 7, 10, 13, and 16 only (the straight edges on the far left).

Symmetry 1

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

2 Select Boundaries 31, 59, 247, 342, 517, 601, 720, 825, 878, 1149, 1300, 1421, 1488, 1748, 1756, 1823, 1912, and 2025 only.

Use the **Select Box** control in the **Graphics** toolbar to select all straight upper boundaries at once. Repeat to select all lower boundaries at once. Alternatively use the **Paste Selection** button next to the **Selection** list and enter the numbers listed above.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

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

RESULTS

Velocity (spf)

The first default plot shows the magnitude of the velocity field. Add an arrow plot to visualize the velocity field with the following steps.

Arrow Surface 1

1 Right-click **Velocity (spf)** and choose **Arrow Surface**.

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

3 Find the **x grid points** subsection. In the **Points** text field, type 25.

4 Find the **y grid points** subsection. In the **Points** text field, type 25.

5 In the **Velocity (spf)** toolbar, click **Plot**.

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

Next, set up the second model using the Brinkman equations.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>2D**.

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 **Fluid Flow>Porous Media and Subsurface Flow>Brinkman Equations (br)**.

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

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

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

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Creeping Flow (spf)**.

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

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.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 Click **Load from File**.

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

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

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

3 From the **Length unit** list, choose **µm**.

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 `L`.

4 In the **Height** text field, type `H`.

5 Click **Build Selected**.

GLOBAL DEFINITIONS

Define a function that will be used when setting up the material properties.

Image 1 (im1)

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

2 In the **Settings** window for **Image**, locate the **Coordinates** section.

3 In the **x maximum** text field, type `L`.

4 In the **y maximum** text field, type `H`.

5 Locate the **File** section. Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `pore_scale_flow_structure.png`.

7 Click **Import**.

8 Click **Create Plot**.

RESULTS

SEM image

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

2 In the **SEM image** toolbar, click **Plot**.

MATERIALS

The functions for porosity and permeability are defined according to Equation 1 and Equation 2.

Material 2 (mat2)

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

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
Density	rho	`rho0`	kg/m^3	Basic
Dynamic viscosity	mu	`eta0`	Pa·s	Basic
Porosity	epsilon	`1-0.99*im1(x,y)`	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	`k0/(100*im1(x,y)+0.1)`	m^2	Basic

BRINKMAN EQUATIONS (BR)

In the **Model Builder** window, under **Component 2 (comp2)** click **Brinkman Equations (br)**.

Symmetry 1

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

2 Select Boundaries 2 and 3 only.

Inlet 1

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

2 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

3 From the list, choose **Pressure**.

4 Select Boundary 4 only.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p0`.

Outlet 1

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

2 Select Boundary 1 only.

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

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

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

Compute the problem using adaptive mesh refinement.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Adaptation and Error Estimates** section.

3 From the **Adaptation and error estimates** list, choose **Adaptation and error estimates**.

4 From the **Adaptation in geometry** list, choose **Geometry 2**.

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

RESULTS

Arrow Surface 1

1 Right-click **Velocity (br)** and choose **Arrow Surface**.

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

3 Find the **x grid points** subsection. In the **Points** text field, type `25`.

4 Find the **y grid points** subsection. In the **Points** text field, type `25`.

Adjust the color range to reproduce Figure 5.

Surface

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

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual data range** check box.

4 In the **Minimum** text field, type `1.8e-7`.

5 In the **Velocity (br)** toolbar, click **Plot**.