Created in COMSOL Multiphysics 5.3

Forchheimer Flow

Introduction

This is a tutorial of the coupling between flow of a fluid in an open channel and a porous block attached to one of the channel walls. The flow is described by the Navier-Stokes equation in the free region and a Forchheimer-corrected version of the Brinkman equations in the porous region.

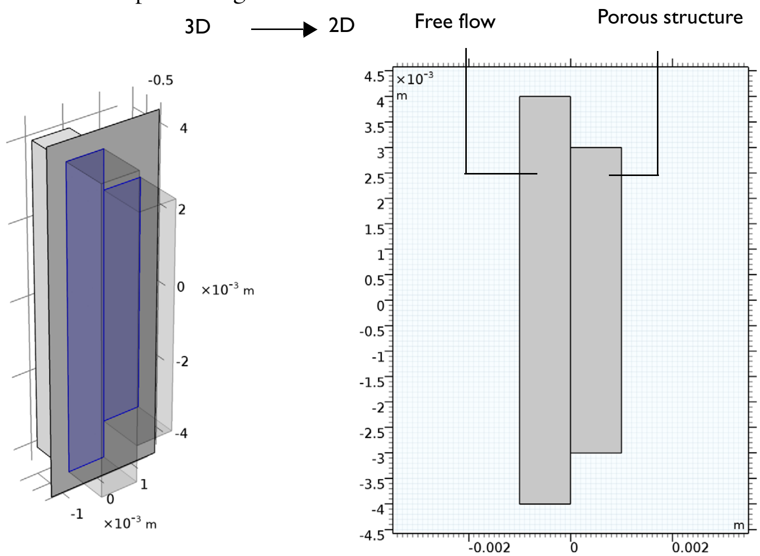

Figure 1: Depiction of the modeling geometry and domain. The 3D geometry can be reduced to a 2D representation assuming that changes through the thickness are negligible.

The coupling of free media flow with porous media flow is common in the fields of earth science and chemical engineering. Perhaps the most frequent way to deal with coupled free and porous media flow is to incorporate Darcy's law adjacent to Navier-Stokes because it is usually numerically easy to solve. However, this approach does not account for viscous effects arising from the free media flow, which may still be important in the region close to the free-porous structure interface. Depending on the pore size and pore distribution, but also the fluid's properties, it can therefore be an oversimplification to employ Darcy's law. The Brinkman equations account for momentum transport by macroscopic viscous effects as well as pressure gradients (stemming from microscopic shear effects inside each pore channel) and can be considered an extension of Darcy's Law.

Still, the Brinkman equations assume laminar flow. Looking at processes in relatively open structures, like gas flow through packed beds, there is also a turbulent contribution to the resistance to flow. In those cases, an additional term accounts for the turbulent contribution to the resistance to flow in the porous domain. The Forchheimer equation (also accredited to Ergun) is widely used to predict pressure drops in packed beds. This equation can generally be written as

$$\frac{\Delta p}{L} = \alpha_1 u + \alpha_2 u^2$$

The left-hand side is the pressure drop per unit length of traveled distance through the bed. The first term on the right-hand side represents the Blake-Kozeny equation for laminar flow. The pressure drop depends linearly on the average linear velocity u for laminar flow, corresponding to Darcy flow. The second term is from the purely turbulent Burke-Plummer equation where the pressure drop is proportional to the square of the velocity. Description of an intermediate flow, where both the laminar and turbulent effects are important, requires the two-term Forchheimer equation. The coefficients α_1 and α_2 are functions of porosity, viscosity, average pore diameter, and fluid density.

Model Definition

Figure 2 below shows the example domain and notations for the boundary conditions.

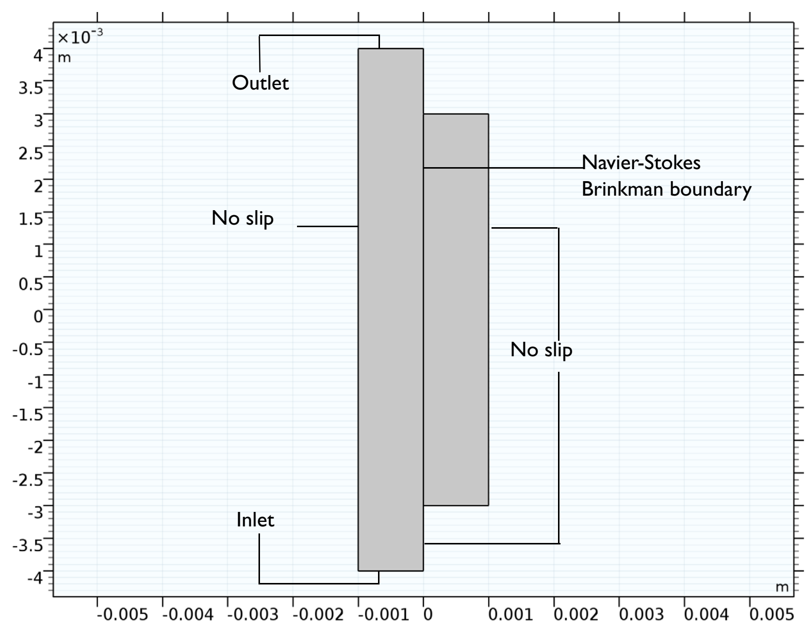

Figure 2: Modeled domain and boundary notations. Flow enters at the bottom and leaves at the top. The region of porous structure is not as long as the free channel.

Flow in the free channel is described by the stationary, incompressible Navier-Stokes equations:

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla\cdot[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)] \qquad (1)$$
$$\nabla\cdot\mathbf{u} = 0$$

where μ denotes the dynamic viscosity (Pa·s), $\mathbf{u}$ refers to the velocity in the open channel (m/s), ρ is the fluid's density (kg/m^3), and p is the pressure (Pa). In the porous domain, the Brinkman equations with Forchheimer correction describe the flow:

$$\frac{\mu}{k}\mathbf{u} = \nabla\cdot\left[-p\mathbf{I}+\frac{\mu}{\varepsilon_p}(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)\right]-\frac{\rho\varepsilon_p C_\mathrm{f}}{\sqrt{k}}\mathbf{u}|\mathbf{u}| \qquad (2)$$
$$\nabla\cdot\mathbf{u} = 0$$

Here k denotes the permeability of the porous medium (m^2), ε_p is the porosity (dimensionless), and the dimensionless friction coefficient is (Ref. 1)

$$C_\mathrm{f} = \frac{1{,}75}{\sqrt{150\varepsilon_p^3}}$$

As Equation 1 and Equation 2 reveal, the momentum transport equations are closely related. The term on the left-hand side of the Navier-Stokes formulation corresponds to momentum transferred by convection in free flow. The Brinkman formulation replaces this term by a contribution associated with the drag force experienced by the fluid flowing through a porous medium. In addition, the last term in the right-hand side of Equation 2 presents the Forchheimer correction for turbulent drag contributions.

In COMSOL Multiphysics, it is easy to set up and solve such a coupled flow regime. The implementation of the extra drag is done with a Forchheimer coefficient (kg/m^4) equal to

$$\beta_F = \frac{\rho\varepsilon_p C_\mathrm{f}}{\sqrt{k}}$$

The boundary conditions are

$$\mathbf{u} = (u_0,\ v_0) \qquad \text{Inlet: velocity}$$
$$\mathbf{u} = \mathbf{0} \qquad \text{Wall:no slip}$$
$$p = 0,\ \ \mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T) = 0 \qquad \text{Outlet: Pressure, no viscous stress}$$

where the pressure level at the outlet is used as a reference value.

The following table lists the input data for the example:

PROPERTY	VALUE	DESCRIPTION
μ	10^{-3} kg/(m·s)	Dynamic viscosity
ρ	1000 kg/m^3	Density
k	10^{-7} m^2	Permeability
ε_p	0.4	Porosity
v_0	2 cm/s	Inlet velocity

Results and Discussion

Figure 3 shows the velocity field in the open channel and porous structure. The plot reveals that there are slight disturbances in the velocity at the porous wall, which suggests momentum transport by viscous effects.

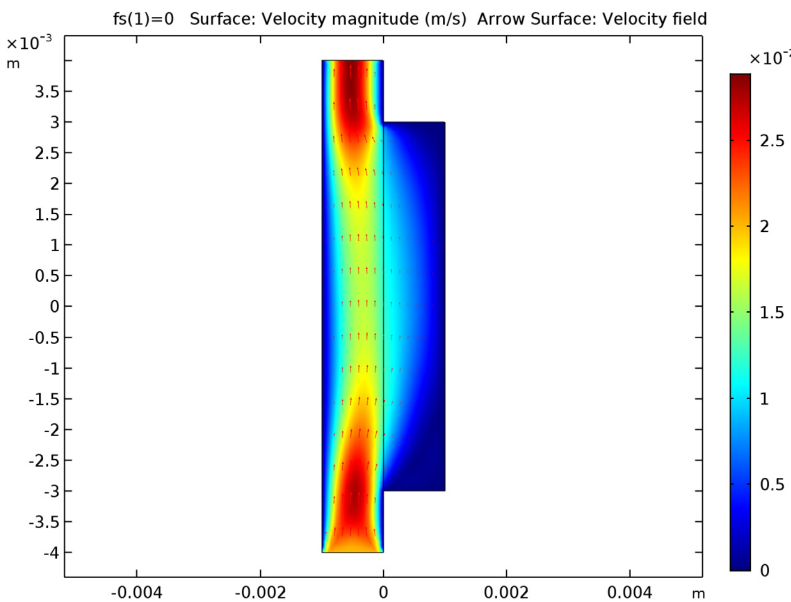

Figure 3: Velocity field without Forchheimer correction.

Figure 4 shows the corresponding field in the porous medium, as modeled by the Forchheimer-corrected Brinkman equation.

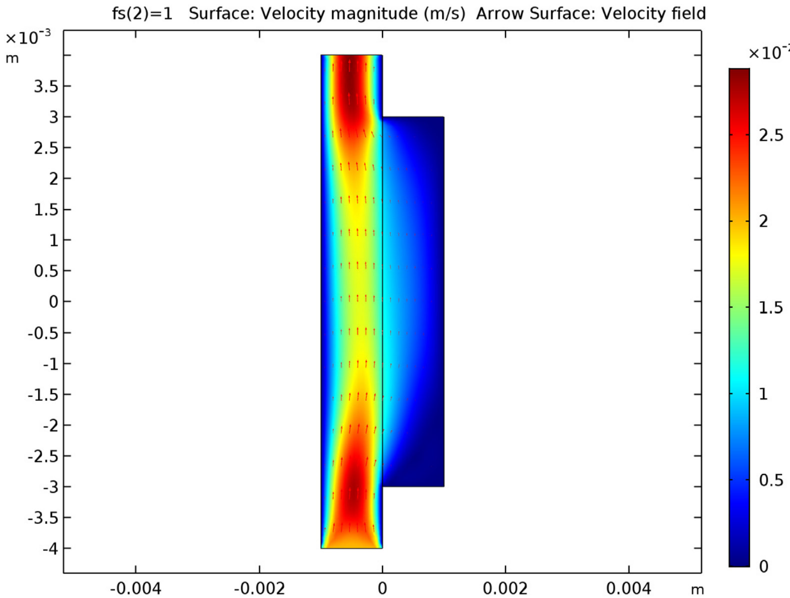

Figure 4: Velocity field in the both free flow and porous medium.

Figure 5 contains a cross-sectional velocity plot. It shows that without the Forchheimer correction, the resistance to flow is underestimated in the porous domain. The added correction gives a solution with slower flow in the porous domain and a faster flow in the free domain, due to incompressibility.

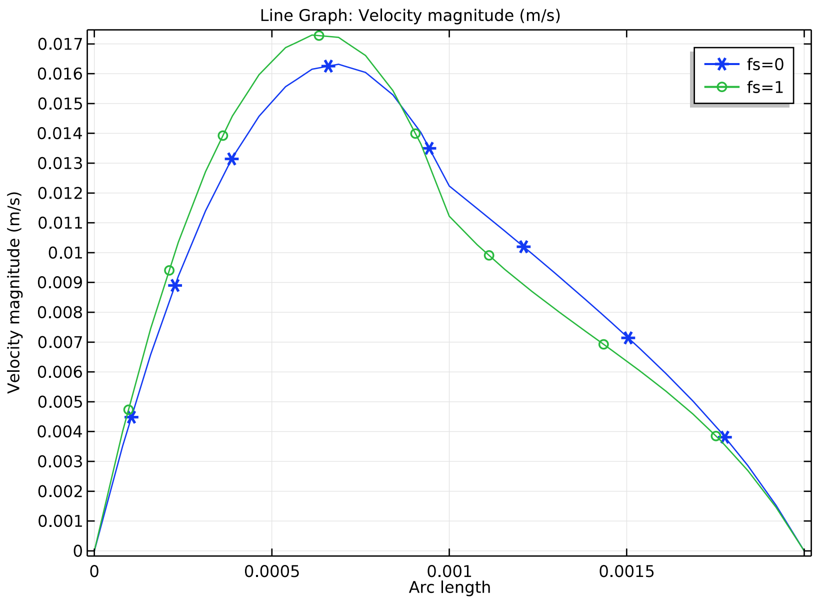

Figure 5: Cross section of the velocity field (velocity magnitude) in the middle of the modeling domain with and without the Forchheimer correction (stars and circles markers, respectively).

Further postprocessing would show that the shear rate perpendicular to the flow is also continuous. This implies that there is significant viscous momentum transfer across the interface and into the porous material, a transport that is not accounted for by Darcy's law.

Notes About the COMSOL Implementation

To implement the Forchheimer pressure drop relation in a differential equation framework, this example uses an approach suggested in Ref. 1 in which the Brinkman momentum balance is amended with a Forchheimer term. The system studied in this example is that of a 2D cross section of a rectangular channel with a porous layer attached to one of its walls. Flow enters the volume with a uniform velocity profile and develops throughout the length of the channel.

Reference

1. A. Amiri and K. Vafai, "Transient Analysis of Incompressible Flow Through a Packed Bed," *Int. J. Heat and Mass Transfer*, vol. 41, pp. 4259–4279, 1998.

Application Library path: `Subsurface_Flow_Module/Fluid_Flow/forchheimer_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>Laminar Flow (spf)**.

3 Click **Add Physics**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1e-3`.

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

5 Locate the **Position** section. In the **y** text field, type `-3e-3`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1e-3`.

4 In the **Height** text field, type `8e-3`.

5 Locate the **Position** section. In the **x** text field, type `-1e-3`.

6 In the **y** text field, type `-4e-3`.

7 On the **Geometry** toolbar, click **Build All**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
v0	2[cm/s]	0.02 m/s	Inlet velocity
eps_p	0.4	0.4	Porosity
Cf	1.75/sqrt(150*eps_p^3)	0.56481	Friction coefficient
fs	1	1	Switch for Forchheimer terms

MATERIALS

Add blank materials for the fluid and porous matrix. You will specify their properties after the set up of the physics interface.

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**, type `Fluid` in the **Label** text field.

Material 2 (mat2)

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

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

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 Select the **Enable porous media domains** check box.

Fluid and Matrix Properties 1

1 On the **Physics** toolbar, click **Domains** and choose **Fluid and Matrix Properties**.

2 Select Domain 2 only.

3 In the **Settings** window for **Fluid and Matrix Properties**, locate the **Fluid Properties** section.

4 From the **Fluid material** list, choose **Fluid (mat1)**.

5 Locate the **Porous Matrix Properties** section. From the **Porous material** list, choose **Porous Matrix (mat2)**.

Now, COMSOL recognizes which material properties are required to solve this model.

MATERIALS

Fluid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Fluid (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	`1000[kg/m^3]`	kg/m³	Basic
Dynamic viscosity	mu	`1e-3[Pa*s]`	Pa·s	Basic

Porous Matrix (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous Matrix (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Porosity	epsilon	`eps_p`	1	Basic
Permeability	kappa	`1e-7[m^2]`	m²	Basic

LAMINAR FLOW (SPF)

Fluid and Matrix Properties 1

In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid and Matrix Properties 1**.

Forchheimer Drag 1

1. On the **Physics** toolbar, click **Attributes** and choose **Forchheimer Drag**.
2. In the **Settings** window for **Forchheimer Drag**, locate the **Forchheimer Drag** section.
3. In the β_F text field, type `fs*Cf*eps_p*spf.rho/sqrt(spf.kappaxx)`.

   Recall that `fs` is a parameter switching on and off the Forchheimer drag. Moreover, `spf.rho` is the density variable for the Laminar Flow interface and `spf.kappabr` the permeability. To find these names, click **Show** in the Model Builder window's toolbar and choose **Equation View**. Then go to the **Equation View** nodes for the Fluid and Matrix Properties.

Inlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Inlet**, locate the **Velocity** section.
4. In the U_0 text field, type `v0`.

Outlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. Select Boundary 3 only.

MESH 1

Size

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `1e-4`.
5. Click **Build All**.

STUDY 1

Step 1: Stationary

1. In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
2. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
3. Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
fs	0 1	

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

Visualize the velocity fields as in Figure 3 and Figure 4.

RESULTS

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Velocity (spf)** and choose **Arrow Surface**.

2 On the **Velocity (spf)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (fs)** list, choose **0**.

4 On the **Velocity (spf)** toolbar, click **Plot**.

Create the plot from Figure 5.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `-1e3`.

4 In row **Point 2**, set **X** to `1e3`.

1D Plot Group 3

1 On the **Results** toolbar, click **1D Plot Group**.

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

Line Graph 1

1 Right-click **Velocity magnitude** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 On the **Velocity magnitude** toolbar, click **Plot**.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Marker** list, choose **Cycle**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`fs=0`
`fs=1`

9 On the **Velocity magnitude** toolbar, click **Plot**.