Model created in COMSOL Multiphysics 6.4

Homogenization of Non-Newtonian Porous Media Flow

Introduction

Understanding the dynamics of non-Newtonian flow in porous media is crucial for a variety of scientific and industrial applications. Non-Newtonian fluids exhibit complex flow characteristics that depend on the shear rate. This makes their behavior in porous media — such as soil, rock, and engineered materials — particularly challenging to model and predict. Pore-scale modeling is an essential tool to capture these flow patterns and interactions at a microscopic level, and to find patterns that can be used to derive homogenized properties that can be used at the macroscale.

This model shows how to use pore-scale modeling of non-Newtonian flow through a porous structure to find homogenized quantities for modeling at the macroscopic scale. It also compares the results and proves that this is a valuable approach that can be carried on to other applications.

Model Definition

The modeling process starts with opening the model file from *Analyzing Porous Structures on the Microscopic Scale* in the Porous Media Module Application Library. This model (Figure 1) was used to compute the porosity ε_p and permeability κ (m^2) of the porous structure. The results are listed in Table 1. These values are intrinsic values of the porous structure and do not depend on the fluid flow.

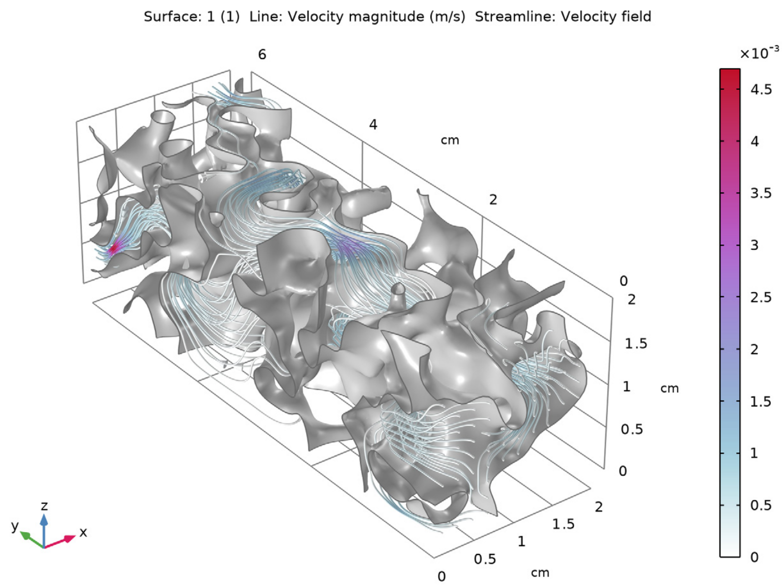


Figure 1: Results from the initial model for computing the porosity and permeability.

Describing non-Newtonian flow in porous media with mathematical models remains an active research area due to the complexity of these flows. No universally applicable model exists yet. The flow behavior in porous media strongly depends on pore geometry, and one approach to describe it uses the concept of an apparent shear rate. Unlike free non-Newtonian flow, where shear rate can be calculated directly from flow velocity, the apparent shear rate depends on pore geometry and fluid properties. Therefore, it must be determined experimentally or numerically for each configuration.

The apparent shear rate $\dot{\gamma}_{\mathrm{app}}$ (1/s) is often assumed to take the following form (Ref. 1):

$$\dot{\gamma}_{\mathrm{app}} = \alpha\frac{v}{\sqrt{\kappa\varepsilon_{\mathrm{p}}}} \tag{1}$$

The term $v/(\sqrt{\kappa\varepsilon_{\mathrm{p}}})$ is called the normalized velocity with the velocity magnitude v.

The correction factor α can be assumed to be constant within a certain velocity range for a specific fluid, so it must be determined individually for each setup. The advantage of this approach is that well-known non-Newtonian fluid models can be used as they are. In this context, a Carreau fluid is used with an apparent viscosity of

$$\mu_{app} = \mu_{inf} + (\mu_0 - \mu_{inf})[1 + (\lambda\dot{\gamma}_{app})^2]^{\frac{n-1}{2}} \tag{2}$$

The idea is now to compute the apparent viscosity at the pore scale using Darcy's Law:

$$-\nabla p = \frac{\mu_{app}}{\kappa}\mathbf{u} \tag{3}$$

After rearranging Equation 2, compute the apparent shear rate:

$$\dot{\gamma}_{app} = \frac{1}{l}\sqrt{\left(\frac{\mu_{app} - \mu_{inf}}{\mu_0 - \mu_{inf}}\right)^{\frac{2}{n-1}} - 1} \tag{4}$$

Using a Least-Squares Fit function to fit Equation 1 to the computed values gives α, which then can be used for the Brinkman Equations on the macroscale.

TABLE 1: MATERIAL PROPERTIES.

Property	**Symbol**	**Value**
Porosity	ε_p	0.3709
Permeability	κ	3.029e-8
Zero shear rate viscosity	μ_0	0.1[Pa]
Infinite shear rate viscosity	μ_{inf}	0.005[Pa]
Relaxation time	λ	1.5[s]
Power index	n	0.65

Results and Discussion

After computing the pore-scale non-Newtonian flow for different pressure drops, inspect the viscosity (Figure 2). The fluid's shear-thinning characteristics are clearly visible, especially at high-pressure drops and high velocities. In narrow regions with higher velocities, the viscosity is low, while in areas with lower velocities, the viscosity is high.

.

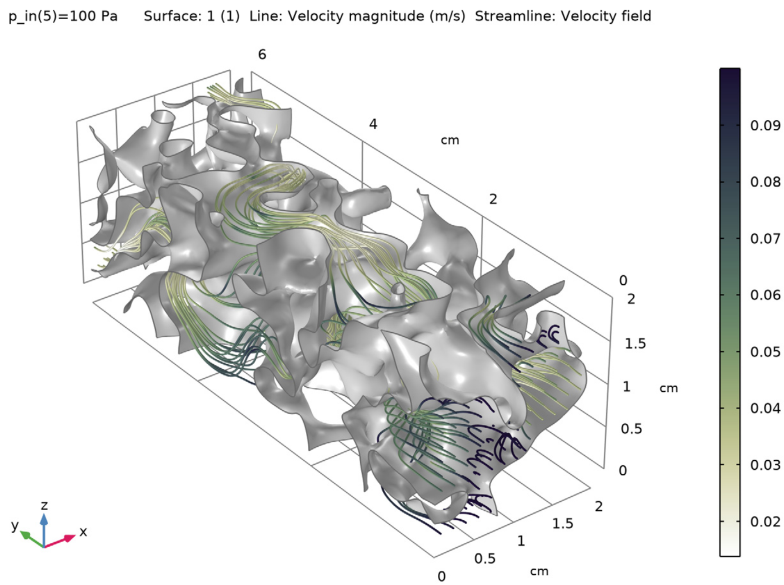


Figure 2: Dynamic viscosity for inlet pressure of 100 Pa.

With these results, the apparent shear rate is computed using Equation 4. The resulting values are then fitted using Equation 1 to find the correction factor α. Figure 3 shows that the linear relationship is a good approximation for the considered pressure range.

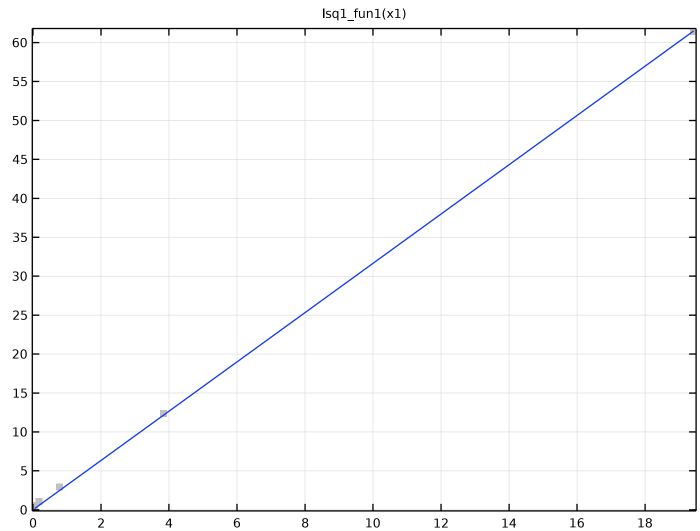


Figure 3: Least-square fitted data.

The value for α is approximately 3.24. With this, the homogenization is completed. To verify the results, a model using Brinkman Equations is set up and the results are compared. Figure 4 and Figure 5 show that the results using the homogenized approach match the ones from the pore scale very well.

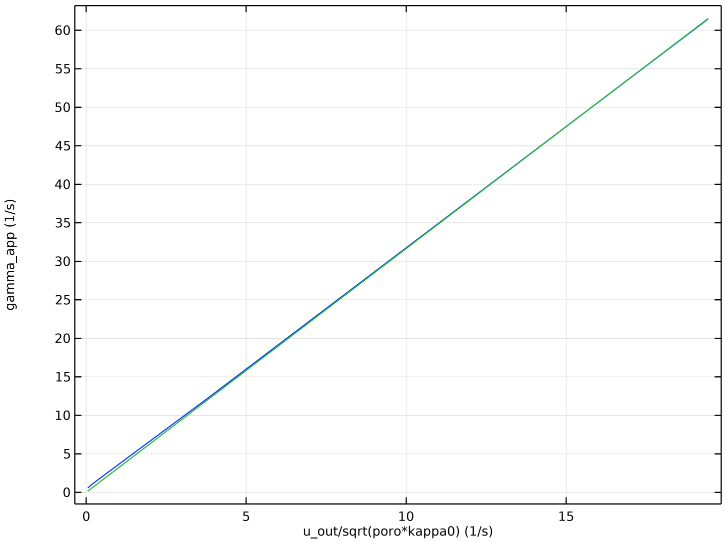


Figure 4: Apparent shear rate.

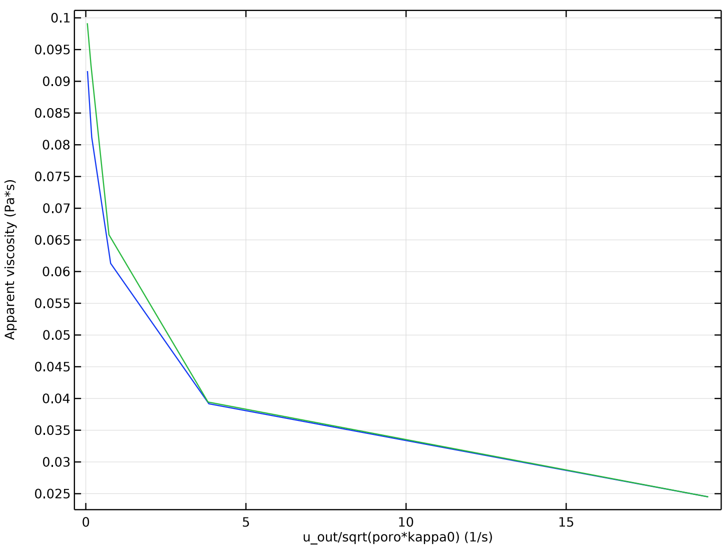


Figure 5: Apparent viscosity.

Reference

1. N. Zamani, I. Bondino, R. Kaufmann, and A. Skauge, "Computation of polymer in-situ rheology using direct numerical simulation," *J. Pet. Sci. Eng.*, vol. 159, pp. 92–102, 2017.

Application Library path: `Polymer_Flow_Module/Tutorials/homogenization_non_newtonian_porous`

Modeling Instructions

ROOT

Start by opening the model `pore_scale_flow_3d` from the Porous Media Flow Module Application Library. If you have not installed the Application Library you can alternatively download the model from the COMSOL website.

1 From the **File** menu, choose **Open**.

2 From the Application Libraries root, browse to the folder `Porous_Media_Flow_Module/Fluid_Flow` and double-click the file `pore_scale_flow_3d.mph`.

RESULTS

Velocity

The model has been used to calculate the porosity and permeability of the porous structure. These values, which represent the intrinsic properties of the porous medium, now serve as inputs for the model. To utilize them throughout the whole model, create parameters based on the values provided in **Table 1** under the **Graphics** window. Additionally, add parameters for the Carreau fluid.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, expand the **Results** > **Tables** node, then click **Global Definitions** > **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
poro	0.3709	0.3709	Porosity
kappa0	3.029e-8[m^2]	3.029E-8 m²	Permeability
muc0	0.1[Pa*s]	0.1 Pa·s	Zero shear rate viscosity, Carreau fluid
muc_inf	0.005[Pa*s]	0.005 Pa·s	Infinite shear rate viscosity
lambda	1.5[s]	1.5 s	Relaxation time
n	0.65	0.65	Power index

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

CREEPING FLOW (SPF)

Fluid Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Creeping Flow (spf)** node, then click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 Find the **Constitutive relation** subsection. From the list, choose **Inelastic non-Newtonian**.

4 From the **Inelastic model** list, choose **Carreau**.

Edit the material properties using the parameters.

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, expand the **Global Definitions** > **Materials** node, then click **Material 1 (mat1)**.

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
Zero shear rate viscosity	mu0	muc0	Pa·s	Carreau model
Infinite shear rate viscosity	mu_inf	muc_inf	Pa·s	Carreau model

Property	Variable	Value	Unit	Property group
Relaxation time	lam_car	lambda	s	Carreau model
Power index	n_car	n	1	Carreau model

DEFINITIONS

Variables 1

Add the expressions for the apparent viscosity and apparent shear rate to the table using Equation 3 and Equation 4 respectively.

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node, then click **Variables 1**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
mu_app	kappa0/u_out*dPdL	Pa·s	Apparent viscosity
gamma_app	sqrt(((mu_app-muc_inf)/(muc0-muc_inf))^(2/(n-1))-1)/lambda	1/s	

CREEPING FLOW (SPF)

Inlet 1

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

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

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

4 Locate the **Pressure Conditions** section. Click to select the p_0 text field. Right-click and choose **Create Parameter**.

5 In the **Create Parameter** dialog, type p_in in the **Name** text field.

6 In the **Expression** text field, type 1[Pa].

7 In the **Description** text field, type Inlet pressure.

8 Click **OK**.

ADD STUDY

1 In the **Study** 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 Click the **Add Study** button in the window toolbar.

5 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

2 Select the **Auxiliary sweep** checkbox.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_in (Inlet pressure)	`10^range(0,0.5,2)`	Pa

5 In the table, click to select the cell at row number 1 and column number 3.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** checkbox.

9 In the **Study** toolbar, click **Compute**.

Reuse the velocity plot to visualize the viscosity. The streamlines color table currently indicates velocity magnitude, but switch it to show viscosity instead. To keep the original Velocity plot in the model, duplicate it first, then make the modifications.

RESULTS

Viscosity

1 In the **Model Builder** window, right-click **Velocity** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Viscosity` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Streamline 1

1 In the **Model Builder** window, expand the **Viscosity** node, then click **Streamline 1**.

2 In the **Settings** window for **Streamline**, locate the **Coloring and Style** section.

3 Find the **Point style** subsection. From the **Type** list, choose **None**.

4 Locate the **Streamline Positioning** section. In the **Number** text field, type `80`.

Color Expression 1

1 In the **Model Builder** window, expand the **Streamline 1** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `spf.mu`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Lichen**.

5 In the **Viscosity** toolbar, click **Plot**. Compare with Figure 2.

Next, compute the normalized velocity, apparent shear rate, and apparent viscosity as previously defined under **Variables 1**.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`u_out/sqrt(poro*kappa0)`	`1/s`	
`gamma_app`	`1/s`	
`mu_app`	`Pa*s`	`Apparent viscosity`

5 Click next to **Evaluate**, then choose **New Table**.

TABLE 2

1 Go to the **Table 2** window.

2 Click the **Table Graph** button in the window toolbar.

RESULTS

Table Graph 1

1 In the **Settings** window for **Table Graph**, locate the **Data** section.

2 From the **x-axis data** list, choose **u_out/sqrt(poro*kappa0) (1/s)**.

3 From the **Plot columns** list, choose **Manual**.

4 In the **Columns** list box, select **gamma_app (1/s)**.

Apparent Shear Rate vs. Normalized Velocity

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Apparent Shear Rate vs. Normalized Velocity` in the **Label** text field.

3 In the **Apparent Shear Rate vs. Normalized Velocity** toolbar, click **Plot**.

Next, set up a Least-Squares Fit function to find α in Equation 1.

GLOBAL DEFINITIONS

Least-Squares Fit 1 (lsq1_fun1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Least-Squares Fit**.

2 In the **Settings** window for **Least-Squares Fit**, locate the **Data** section.

3 From the **Data source** list, choose **Result table**.

4 From the **Result table** list, choose **Table 2**.

5 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
p_in (Pa)	Ignored column	
u_out/sqrt(poro*kappa0) (1/s)	Argument	Name=x1
gamma_app (1/s)	Function values	Name=lsq1_fun1, Expression=a1*x1+a0
Apparent viscosity (Pa*s)	Ignored column	

6 Locate the **Parameters** section. Click **Clear Table**.

7 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 3 and column number 3.

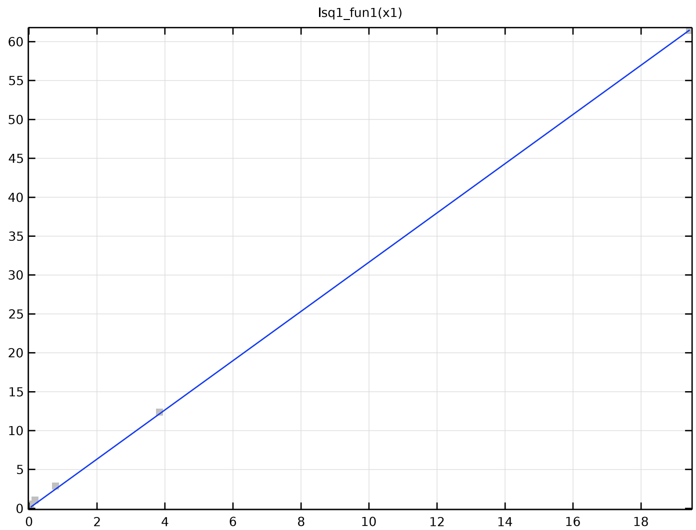
8 In the **Expression** text field, type `alpha*x1`.

9 Locate the **Parameters** section. In the table, enter the following settings:

Parameter	Values	Scale	Lower bound	Upper bound
alpha	0	1		

10 Click **Fit Parameters**.

11 Click **Plot**.



ADD COMPONENT

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

GEOMETRY 2

Block 1 (blk1)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type `2[cm]`.
4. In the **Depth** text field, type `6[cm]`.
5. In the **Height** text field, type `2[cm]`.

ADD PHYSICS

1. In the **Physics** 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** checkboxes for **Study 1** and **Study 2**.

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

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

BRINKMAN EQUATIONS (BR)

Fluid 1

1 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

2 From the ρ list, choose **User defined**. In the associated text field, type `rho_f`.

3 Find the **Constitutive relation** subsection. From the list, choose **Inelastic non-Newtonian**.

4 From the **Inelastic model** list, choose **Carreau**.

5 Find the **Apparent shear rate** subsection. In the α text field, type `lsq1.alpha`.

MATERIALS

Material Link 2 (matlnk2)

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

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

3 Click **Go to Material**.

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions > Materials** click **Material 1 (mat1)**.

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
Porosity	epsilon	poro	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	kappa0	m²	Basic

BRINKMAN EQUATIONS (BR)

Inlet 1

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

2 Select Boundary 2 only.

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 p_in.

Outlet 1

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

2 Select Boundary 5 only.

Symmetry 1

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

2 Select Boundaries 1, 3, 4, and 6 only.

Define variables which are later used to compare the porous approach with the pore-scale modeling.

DEFINITIONS (COMP2)

Average 3 (aveop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

Variables 2

1 In the **Model Builder** window, 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
u_out	br.out1.massFlowRate/rho_f/width^2	m/s	Darcy velocity
mu_app	aveop3(br.mu)	Pa·s	Apparent viscosity
gamma_app	aveop3(br.porous.fluid.gamma_app)	1/s	Apparent shear rate

ADD STUDY

1 In the **Study** 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** checkbox for **Creeping Flow (spf)**.

5 Click the **Add Study** button in the window toolbar.

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

STUDY 3

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

2 Select the **Auxiliary sweep** checkbox.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_in (Inlet pressure)	`10^range(0,0.5,2)`	Pa

5 In the **Model Builder** window, click **Study 3**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** checkbox.

8 In the **Study** toolbar, click **Compute**.

RESULTS

Global Evaluation 3

1 In the **Model Builder** window, under **Results** > **Derived Values** right-click **Global Evaluation 2** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3/Solution 3 (4) (sol3)**.

4 Click next to **Evaluate**, then choose **New Table**.

Table Graph 2

1 In the **Model Builder** window, under **Results** > **Apparent Shear Rate vs. Normalized Velocity** right-click **Table Graph 1** and choose **Duplicate**.

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

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

4 In the **Apparent Shear Rate vs. Normalized Velocity** toolbar, click **Plot**. Compare with Figure 4.

Apparent Viscosity vs. Normalized Velocity

1 In the **Model Builder** window, right-click **Apparent Shear Rate vs. Normalized Velocity** and choose **Duplicate**.

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

Table Graph 1

1 In the **Model Builder** window, expand the **Apparent Viscosity vs. Normalized Velocity** node, then click **Table Graph 1**.

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

3 In the **Columns** list box, select **Apparent viscosity (Pa*s)**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

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

3 In the **Columns** list box, select **Apparent viscosity (Pa*s)**.

4 In the **Apparent Viscosity vs. Normalized Velocity** toolbar, click **Plot**. Compare with Figure 5.