Non-Newtonian Flow

Introduction

This example shows the influence of shear rate dependent viscosity on the flow of a linear polystyrene solution. For this type of flow, you can use the Carreau viscosity model. Due to rotational symmetry, it is possible to reduce the model dimensions from 3D to axisymmetric 2D (see Figure 1).

Model Definition

For Carreau fluid, the constitutive relation between viscous stress **K** and strain-tensor **S** is given by

$$\mathbf{K} = \left(\mu_\infty + (\mu_0 - \mu_\infty)[1 + (\lambda\dot{\gamma})^2]^{\frac{(n-1)}{2}}\right)\mathbf{S}$$

where $\dot{\gamma}$ is the shear rate, μ_∞ is the infinite shear rate viscosity, μ_0 is the zero shear rate viscosity, λ is a parameter with units of time, and n is a dimensionless parameter.

A solution of linear polystyrene in 1-chloronaphthalene has the properties listed in Table 1 (Ref. 1).

TABLE 1: PROPERTIES OF A SOLUTION OF LINEAR POLYSTYRENE IN 1-CHLORONAPHTALENE.

PARAMETER	VALUE
μ_∞	0
μ_0	166 Pa·s
λ	$1.73 \cdot 10^{-2}$ s
n	0.538
ρ	450 kg/m^3

The model domain is shown in Figure 1.

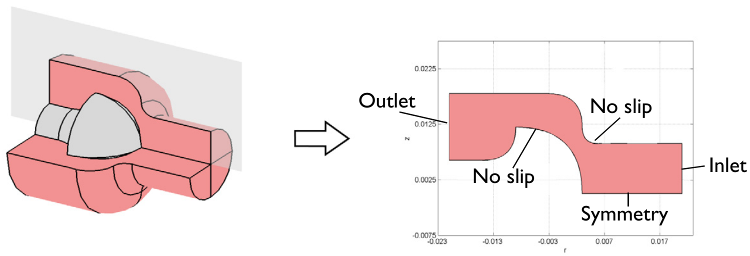


Figure 1: Model domain. The geometry can be simplified assuming axisymmetry.

The boundary conditions at the inlet and the outlet are set to fixed pressures and vanishing viscous stresses:

$$p = p_{\mathrm{in}}$$
$$p = 0$$

and

$$\mathbf{n} \cdot \mathbf{K} = 0$$

To study the effect on viscosity at different inlet pressures, the model makes use of the parametric solver to vary p_{in} from 10 kPa to 210 kPa. The axis of rotation requires the axial symmetry condition:

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

while all other boundaries impose the no slip condition:

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

Results and Discussion

Figure 2 shows that the velocity distribution is more pronounced at the inlet compared to the outlet. This is because the cross section is greater at the outlet. The figure also shows that the region with greatest velocity gradient is in the contraction, which means that the shear rate is largest there.

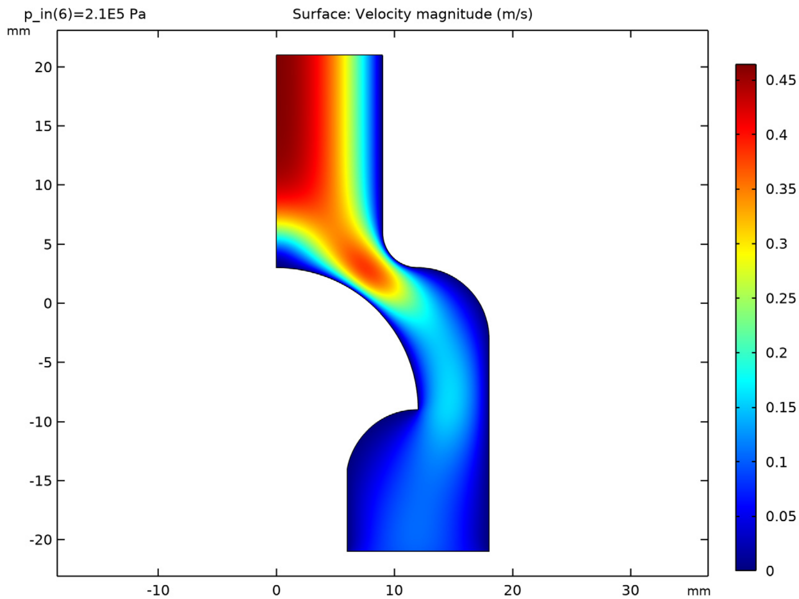


Figure 2: Velocity field in the modeling domain.

Because the fluid is shear thinning, the viscosity depends on the shear rate and is shown in Figure 3. It reaches its lowest value close to the wall in the contraction between the piston and the wall.

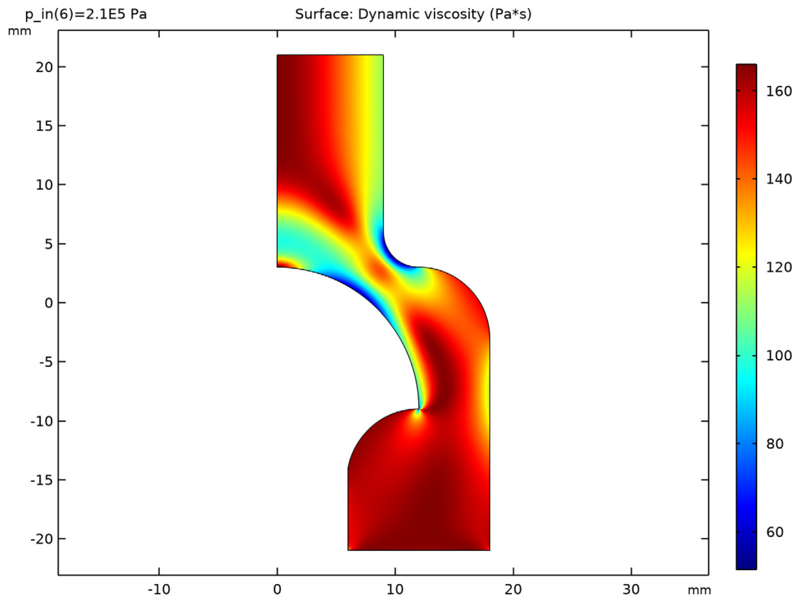


Figure 3: Viscosity distribution in the domain. The lowest viscosity occurs at the wall in the contraction region.

Showing the result of a parametric study of the inlet pressure, Figure 4 contains a cross-sectional plot of the viscosity across the contraction. Sweeping through a range of inlet pressures imposes greater velocities on the non-Newtonian fluid. As the velocity increases, the shear rate also increases and the viscosity decreases. An optimal condition is to have as flat a viscosity profile as possible. This is hindered by also wanting to put through as high a flow rate as possible.

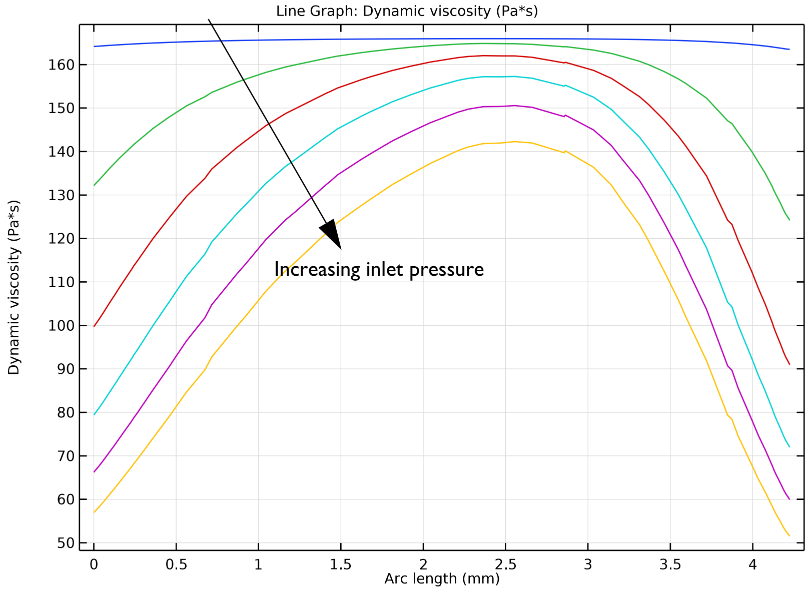


Figure 4: Parametric study of the process, sweeping the inlet pressure from 10 kPa to 210 kPa, while investigating a cross-sectional viscosity plot. A greater inlet pressure (and pressure differential) results in lower values for the viscosity and greater variations in its distribution through the cross section.

Reference

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, John Wiley & Sons, 1960.

Application Library path: `CFD_Module/Single-Phase_Flow/non_newtonian_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 Axisymmetric**.
2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar 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 **mm**.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
5 In the **r** text field, type `0 0 9 9`.
6 In the **z** text field, type `3 21 21 6`.

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.
2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.
3 From the **Specify** list, choose **Endpoints and start angle**.
4 Locate the **Starting Point** section. In the **r** text field, type `9`.
5 In the **z** text field, type `6`.
6 Locate the **Endpoint** section. In the **r** text field, type `12`.
7 In the **z** text field, type `3`.
8 Locate the **Angles** section. In the **Start angle** text field, type `180`.

Circular Arc 2 (ca2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.
2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.
3 From the **Specify** list, choose **Endpoints and start angle**.

4 Locate the **Starting Point** section. In the **r** text field, type `18`.

5 In the **z** text field, type `-3`.

6 Locate the **Endpoint** section. In the **r** text field, type `12`.

7 In the **z** text field, type `3`.

Polygon 2 (pol2)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **r** text field, type `18 18 6 6`.

6 In the **z** text field, type `-3 -21 -21 -14`.

Circular Arc 3 (ca3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

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

3 From the **Specify** list, choose **Endpoints and start angle**.

4 Locate the **Starting Point** section. In the **r** text field, type `12`.

5 In the **z** text field, type `-9`.

6 Locate the **Endpoint** section. In the **r** text field, type `6`.

7 In the **z** text field, type `-14`.

8 Locate the **Angles** section. In the **Start angle** text field, type `90`.

Circular Arc 4 (ca4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

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

3 From the **Specify** list, choose **Endpoints and start angle**.

4 Locate the **Starting Point** section. In the **r** text field, type `12`.

5 In the **z** text field, type `-9`.

6 Locate the **Endpoint** section. In the **z** text field, type `3`.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Convert to Solid**, click **Build All Objects**.

4 Click **Build Selected**.

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

Form Union (fin)

In the **Model Builder** window, right-click **Form Union (fin)** and choose **Build Selected**.

GLOBAL DEFINITIONS

Add a global parameter for the inlet pressure. You will use this as the parameter in a parametric sweep.

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
p_in	10[kPa]	10000 Pa	Inlet pressure

LAMINAR FLOW (SPF)

Fluid Properties 1

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

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

3 From the ρ list, choose **User defined**. In the associated text field, type 450.

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

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

6 In the μ_0 text field, type 166.

7 In the λ text field, type 1.73e-2.

8 In the n text field, type 0.538.

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 4 only.

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**.

4 Click **Build All**.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_in (Inlet pressure)	range(10000,40000,210000)	Pa

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

RESULTS

The default plot groups shows the velocity magnitude in 2D (compare with Figure 2) and 3D as well as the pressure field in 2D.

Visualize the viscosity distribution in a separate plot group with the following steps:

2D Plot Group 4

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Surface 1

1 Right-click **2D Plot Group 4** and choose **Surface**.

2 In the **2D Plot Group 4** toolbar, click **Plot**.

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

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

5 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Material properties>spf.mu - Dynamic viscosity - Pa·s**.

6 In the **2D Plot Group 4** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 3.

Viscosity

1 In the **Model Builder** window, right-click **2D Plot Group 4** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Viscosity` in the **New label** text field.

3 Click **OK**.

Finally, plot the viscosity across the contraction as in Figure 4.

Cut Line 2D 1

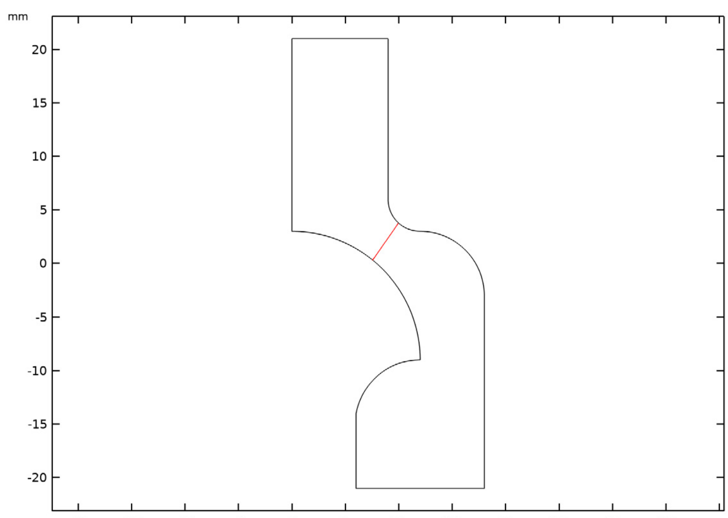
1 In the **Results** toolbar, click **Cut Line 2D**.

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

3 In row **Point 1**, set **r** to `7.55` and **z** to `0.32`.

4 In row **Point 2**, set **r** to `9.97` and **z** to `3.79`.

5 Click **Plot**.



1D Plot Group 5

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

Line Graph 1

1 Right-click **1D Plot Group 5** and choose **Line Graph**.

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

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

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Material properties>spf.mu - Dynamic viscosity - Pa·s**.

5 Click to expand the **Quality** section. From the **Recover** list, choose **Within domains**.

6 In the **1D Plot Group 5** toolbar, click **Plot**.