Model created in COMSOL Multiphysics 6.4

Viscoelastic Flow Through a Channel with a Flexible Wall

Introduction

The correct description of blood circulation requires an understanding of its complex rheological behavior and its interaction with the blood vessel walls. Depending on the size of the vessel, we may consider microcirculation as the flow in arterioles, venules, and capillaries, and macrocirculation as the flow in larger arteries. In macrocirculation, inertia effects tend to be dominant and the non-Newtonian behavior of the blood plays a lesser role. In microcirculation, inertia effects are smaller and we need to include the rheological behavior of blood.

In Ref. 1 and Ref. 2, a benchmark geometry for fluid-structure interaction with a Newtonian liquid in a thin two-dimensional channel, partially bounded by an elastic wall, was extended to viscoelastic flow to serve as a preliminary study of microcirculation in vessels where capturing the elastic nature of the wall and the viscoelastic character of blood is important. This models reproduces their results using the Viscoelastic Flow interface and the Fluid-Structure Interaction multiphysics coupling.

Note: This application requires the Polymer Flow Module and the Structural Mechanics Module.

Model Definition

The geometry of the model is depicted in Figure 1 and Table 1. The flow is from the left to the right, and it is affected by an elastic section of the wall which is loaded with a uniform external pressure $P_{\mathrm{ext}} = 1.755$ Pa. The upper wall has a thickness $b = 0.1$ mm, and is made of a solid material obeying Hooke's law with a Young modulus of 35.9 kPa and a Poison ratio of 0.45. The flow enters from the left with an average velocity $U_{\mathrm{in}} = 0.03$ m/s, and reaches an outlet pressure $P_{\mathrm{out}} = 0$ Pa at the right-hand side of the channel. A no-slip boundary condition is enforced at the walls.

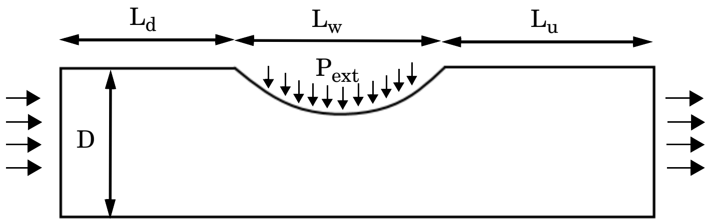

Figure 1: Channel geometry.

TABLE 1: DIMENSIONS OF THE GEOMETRY.

PARAMETER	VALUE
D	0.01[m]
b	0.01*D
Ld	5*D
Lw	5*D
Lu	30*D

The fluid is assumed to be an Oldroyd-B fluid with density 1000 kg/m^3. The solvent viscosity ratio is $\beta = \mu_s/\mu_0 = 0.5$, with $\mu_p = \mu_s = 0.5$ mPa·s. These parameters correspond to an inlet Reynolds number of

$$\mathrm{Re} = \frac{D\rho U_{\mathrm{in}}}{\mu_0} = 300$$

Studies of viscoelastic flows use the Weissenberg number to compare the magnitude of the elastic forces to the viscous forces. In this case, it can be defined as

$$\mathrm{Wi} = \lambda\frac{U_{\mathrm{in}}}{D}$$

where λ represents the relaxation time of the polymer. In this model, three different relaxation times are used — 0.05 s, 0.1 s, and 0.2 s — corresponding to $\mathrm{Wi} = 0.15$, $\mathrm{Wi} = 0.3$, and $\mathrm{Wi} = 0.6$, respectively.

Results and Discussion

The deformation of the plate and the boundary loads on the wall are plotted in Figure 2 for a Weissenberg number of 0.6. Arrows pointing upward represent the loads due to the interaction with the fluid, while the downward arrows represent the external pressure on the elastic plate. As commented in Ref. 1 and formally proven in Ref. 3, the normal component of the viscous and elastic stresses are negligible, and the normal forces on the wall are mainly due to the pressure on the elastic plate, which is plotted for all studied Weissenberg numbers in Figure 3. The pressure from the fluid, pointing upward, acts against the external pressure on the plate, but has smaller magnitude. This results in a downward deformation of the plate, as plotted in Figure 4 for all Weissenberg numbers. Even though the normal component of the viscoelastic stresses does not have a direct effect on the plate, different relaxation times result in different distributions of pressure and plate displacement. Specifically, increasing the Weissenberg number results in an increased pressure downstream of the narrowest gap in the channel and a slightly smaller plate displacement. Narrowing the channel also affects the velocity distribution as seen in Figure 5.

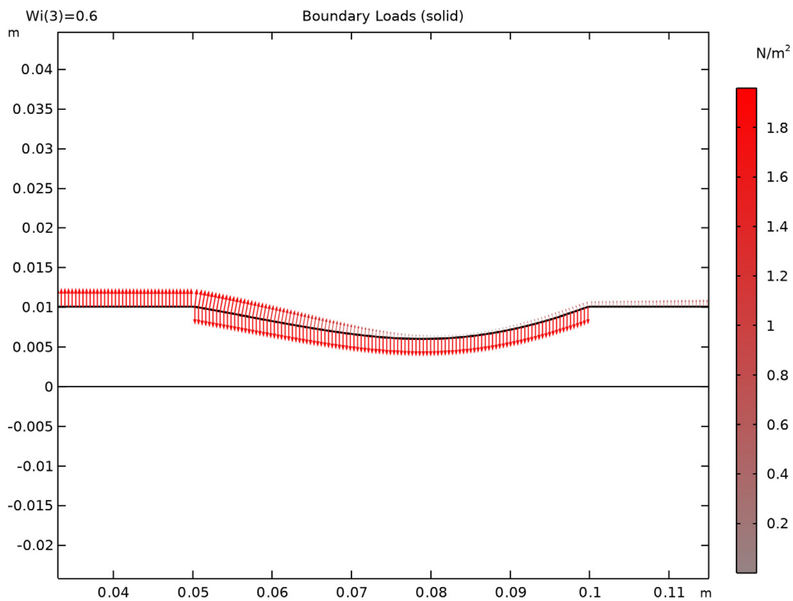

Figure 2: Boundary loads applied to the wall for $\mathrm{Wi} = 0.6$*. The upper arrows represent the fluid forces exerted on the wall. The downward pointing arrows represent the uniform pressure on the flexible wall.*

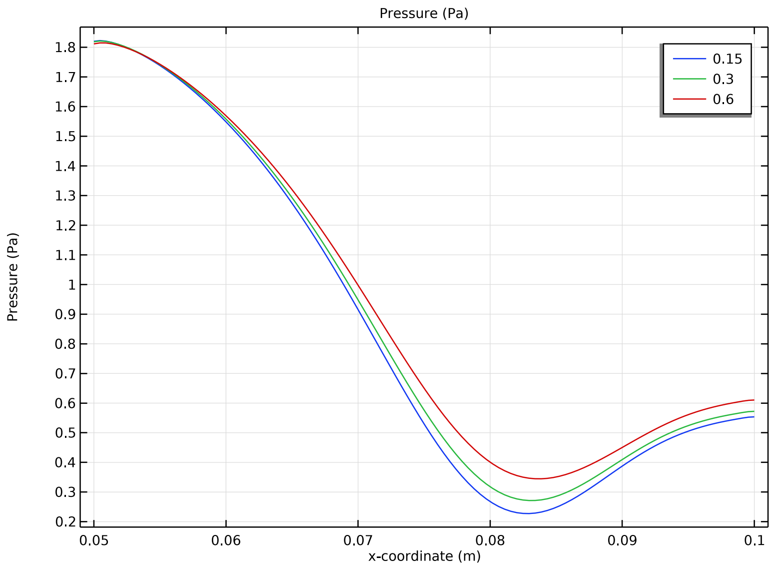

Figure 3: Distribution of fluid pressure on the elastic wall for the different Weissenberg numbers.

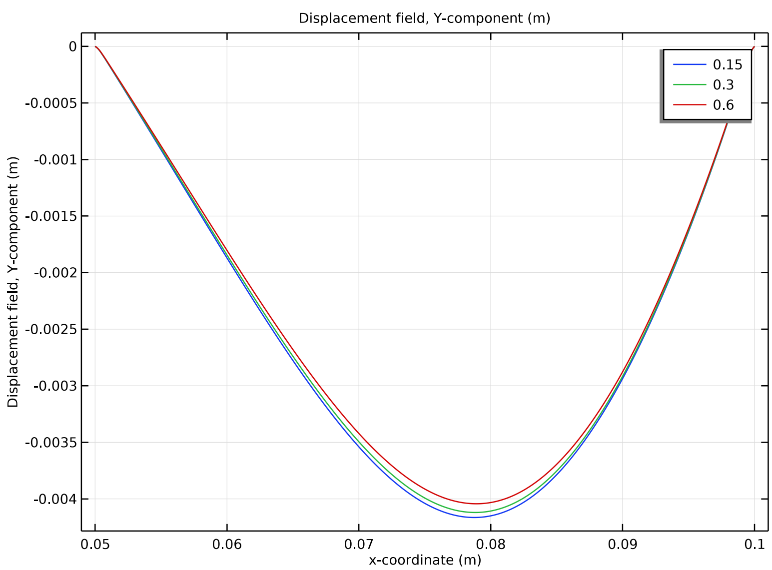

Figure 4: Elastic wall displacement for the different Weissenberg numbers.

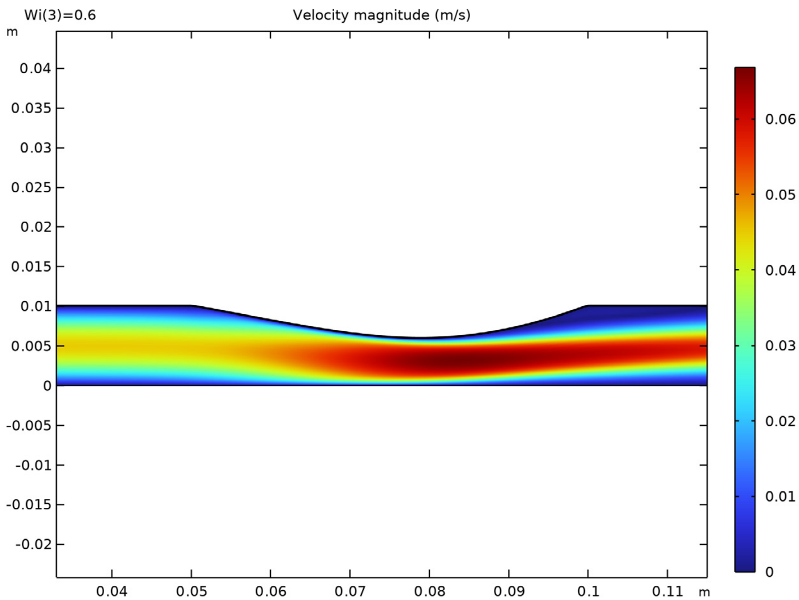

Figure 5: Velocity magnitude in the deformed channel for $\mathrm{Wi} = 0.6$.

Notes About the COMSOL Implementation

The present application solves a coupled fluid-structure interaction problem with a stationary study. A robust way to solve this model is to provide a good initial solution for the nonlinear problem by first solving the fluid and solid parts separately. First, solve the viscoelastic flow in the deformed channel. Then, solve for the structural displacements using the fluid loads for an undeformed channel. Finally, solve the coupled problem for the different Weissenberg numbers using the previous solution fields as the initial values.

References

1. E. Turkoz, *Numerical Modeling of Fluid-Structure Interaction with Rheologically Complex Fluids*, PhD thesis, Department of Mechanical Engineering, Technische Universität, 2014.

2. D. Chakraborty and others, “Viscoelastic Flow in a Two-Dimensional Collapsible Channel,” *J. Non-Newtonian Fluid Mech.*, vol. 165, pp. 1204–1218, 2010.

3. N.A. Patankar and others, "Normal Stresses on the Surface of a Rigid Body in an Oldroyd-B Fluid," *J. Fluid Eng.*, vol. 124, pp. 279–280, 2002.

Application Library path: `Polymer_Flow_Module/Verification_Examples/viscoelastic_fsi_flexible_wall`

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** > **Fluid–Structure Interaction** > **Viscoelastic Flow** > **Fluid–Solid Interaction**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

Load the model parameters from a text file.

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 Click **Load from File**.

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

GEOMETRY 1

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 `D`.

Rectangle 2 (r2)

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

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

3 In the **Height** text field, type `b`.

4 Locate the **Position** section. In the **y** text field, type `D`.

5 Click to expand the **Layers** section. Clear the **Layers on bottom** checkbox.

6 Select the **Layers to the left** checkbox.

7 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Lu
Layer 2	Lw

8 Click **Build All Objects**.

MOVING MESH

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Deforming Domain 1**.

2 Select Domain 1 only.

GEOMETRY 1

Rectangle 2 (r2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Rectangle 2 (r2)**.

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

3 In the **Width** text field, type `L`.

4 Click **Build All Objects**.

VISCOELASTIC FLOW (VEF)

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

2 In the **Settings** window for **Viscoelastic Flow**, locate the **Domain Selection** section.

3 In the list, choose **2**, **3**, and **4**.

4 Click **Remove from Selection**.

5 Select Domain 1 only.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Viscoelastic Flow (vef)** 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 `rho_f`.

4 Find the **Constitutive relation** subsection. From the μ_s list, choose **User defined**. In the associated text field, type `mu_s`.

5 In the table, enter the following settings:

Branch	Viscosity (Pa*s)	Relaxation time (s)
1	mu_p	lambda

Inlet 1

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

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

3 Select Boundary 1 only.

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

5 From the list, choose **Fully developed flow**.

6 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `Uin`.

Outlet 1

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

2 Select Boundary 12 only.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

3 From the E list, choose **User defined**. In the associated text field, type E_s.

4 From the ν list, choose **User defined**. In the associated text field, type nu_s.

5 From the ρ list, choose **User defined**. In the associated text field, type rho_s.

Fixed Constraint 1

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

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 3 4 10 13 in the **Selection** text field.

5 Click **OK**.

Boundary Load 1

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

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 8 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Load**, locate the **Force** section.

7 Specify the $\mathbf{f}_A$ vector as

0	x
-p_ext*1[m]	y

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 **Fine**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** right-click **Free Triangular 1** and choose **Delete**.

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type 6 in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Distribution**, locate the **Distribution** section.
7 In the **Number of elements** text field, type 4.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type 1 in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Distribution**, locate the **Distribution** section.
7 From the **Distribution type** list, choose **Predefined**.
8 In the **Number of elements** text field, type 30.
9 In the **Element ratio** text field, type 10.
10 Select the **Symmetric distribution** checkbox.
11 Click **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics (solid)** and **Moving Mesh**.
4 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkbox for **Fluid–Structure Interaction 1 (fsi1)**.

Step 2: Stationary 2

1. In the **Study** toolbar, click **Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Viscoelastic Flow (vef)**.

Step 3: Stationary 3

1. In the **Study** toolbar, click **Stationary**.
2. In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
3. Select the **Auxiliary sweep** checkbox.
4. Click **Add**.
5. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Wi (Weissenberg number)	`0.15 0.3 0.6`	

6. In the **Study** toolbar, click **Show Default Plots**.

Solution 1 (sol1)

1. In the **Study** toolbar, click **Show Default Solver**.
2. Click **Compute**.

RESULTS

Plate Deformation

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Plate Deformation` in the **Label** text field.

Line Graph 1

1. Right-click **Plate Deformation** and choose **Line Graph**.
2. Select Boundary 7 only.
3. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `v_solid`.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `x`.
7. Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Automatic**.

9 In the **Plate Deformation** toolbar, click **Plot**.

Pressure at the Plate

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

2 In the **Settings** window for **1D Plot Group**, type `Pressure at the Plate` in the **Label** text field.

Line Graph 1

1 Right-click **Pressure at the Plate** and choose **Line Graph**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `p`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Automatic**.

9 In the **Pressure at the Plate** toolbar, click **Plot**.