Created in COMSOL Multiphysics 5.5

Flow of Oldroyd-B Viscoelastic Fluid

Introduction

Many complex fluids of interest exhibit a combination of viscous and elastic behavior under strain. Examples of such fluids are polymer solutions and melts, oil, toothpaste, and clay, among many others. The Oldroyd-B fluid presents one of the simplest constitutive models capable of describing the viscoelastic behavior of dilute polymeric solutions under general flow conditions. Despite the apparent simplicity of the constitutive relation, the dynamics that arise in many flows are complicated enough to present a considerable challenge to numerical simulations.

Model Definition

This example studies a flow of Oldroyd-B fluid past a cylinder between two parallel plates. The flow is considered as being two-dimensional (2D). The aspect ratio of the cylinder radius to the to the channel half-width is 1/2.

The fluid is a dilute solution of polymer in a Newtonian liquid solvent of viscosity μ_s. The total stress is presented as

$$\sigma = -p\mathbf{I} + 2\mu_s\mathbf{S}(\mathbf{u}) + \mathbf{T}_e$$

where $\mathbf{u} = (u, v)$ is the flow velocity vector, p is the pressure, and

$$\mathbf{S}(\mathbf{u}) = \frac{1}{2}[\nabla\mathbf{u} + (\nabla\mathbf{u})^T]$$

is the strain rate. The extra stress contribution due to the polymer is given by the following Oldroyd-B constitutive relation:

$$\mathbf{T}_e + \lambda\overset{\nabla}{\mathbf{T}_e} = 2\mu_p\mathbf{S}(\mathbf{u}) \qquad (1)$$

where the upper convective derivative operator is defined as

$$\overset{\nabla}{\mathbf{T}_e} \equiv \frac{\partial\mathbf{T}_e}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{T}_e - [(\nabla\mathbf{u})\mathbf{T}_e + \mathbf{T}_e(\nabla\mathbf{u})^T]$$

The polymer is characterized by two physical parameters: the viscosity μ_p and the relaxation time λ.

NONDIMENSIONAL FORMULATION

The Weissenberg number is defined as:

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

where U_{in} is the average fluid velocity at the inlet, R is the radius of the cylinder, and λ is the polymer relaxation time.

A zero Weissenberg number gives a pure viscous fluid (no elasticity), while the infinite Weissenberg number limit corresponds to purely elastic response. Due to the convective nature of the constitutive relation, solution stability is lost with increasing fluid elasticity. In practice, already values $\mathrm{Wi} > 1$ are considered as a high for many flows of an Oldroyd-B fluid.

The flow is stationary, and the problem becomes dimensionless by using R, U_{in}, and the total viscosity $\mu = \mu_{\mathrm{s}} + \mu_{\mathrm{p}}$. The Reynolds number is defined as

$$\mathrm{Re} = \frac{R\mu U_{\mathrm{in}}}{\rho} \tag{2}$$

BOUNDARY CONDITIONS

Because of the flow symmetry, it is sufficient to model only the upper halves of the channel and the cylinder. At the channel centerline, use the symmetry conditions of zero normal flow and zero total tangential stress. At the channel walls and the cylinder surface, the model uses no slip boundary conditions. At the inlet, the fully developed parabolic velocity profile and the corresponding extra stresses components are specified:

$$u = \frac{3}{2}(1 - y^2)$$

$$T_{11} = 2\mu_p \mathrm{Wi}\left(\frac{\partial u}{\partial y}\right)^2$$

$$T_{12} = \mu_p \frac{\partial u}{\partial y}$$

$$T_{22} = 0$$

At the outlet, use the pressure boundary condition for developed flow; the only stress acting at the boundary is due to the pressure force p_{out}:

$$\sigma \cdot \mathbf{n} = -p_{\mathrm{out}}\mathbf{n}$$

Results

The analysis gradually increases the Weissenberg number from 0 to 1 using the parametric solver. Figure 1 and Figure 2 show the flow field and stress distribution for the value Wi = 0.7. Figure 3 shows the drag coefficient as a function of the Weissenberg number. The result is in good agreement with the experimental and simulation results presented in Ref. 2.

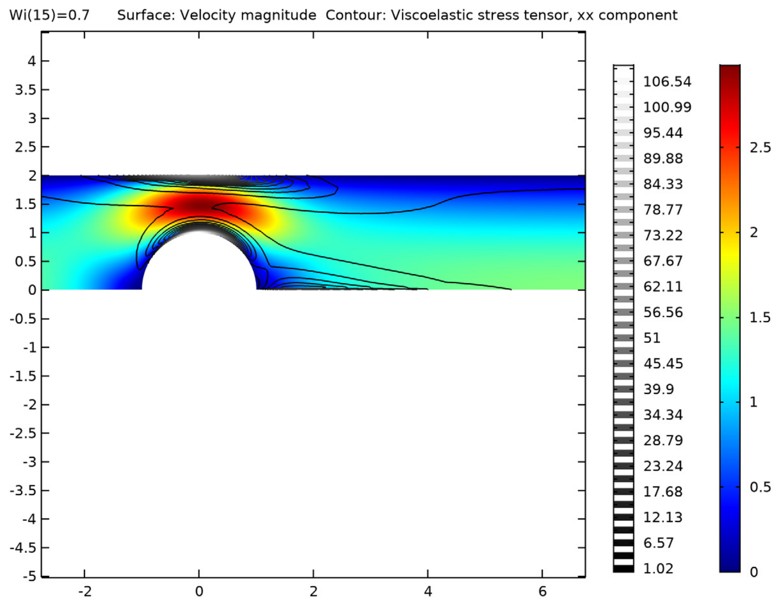

Figure 1: Flow field near cylinder and stress distribution for Wi = 0.7.

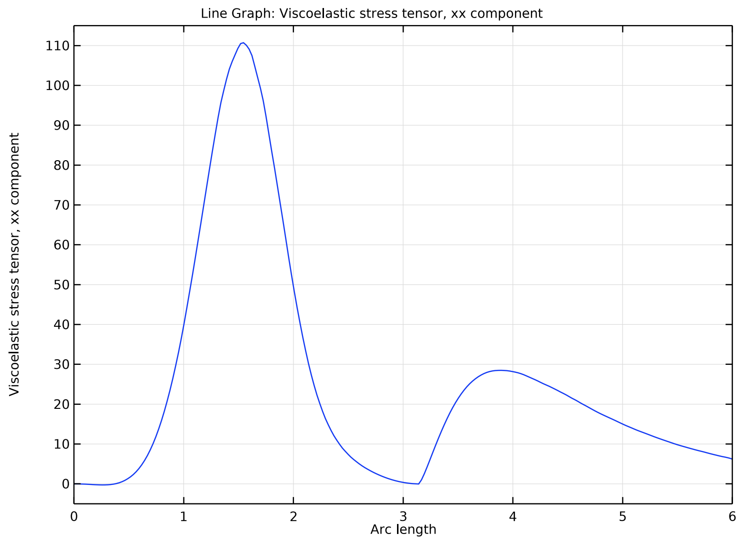

Figure 2: Stress distribution along the cylinder surface and wake centerline for Wi = 0.7.

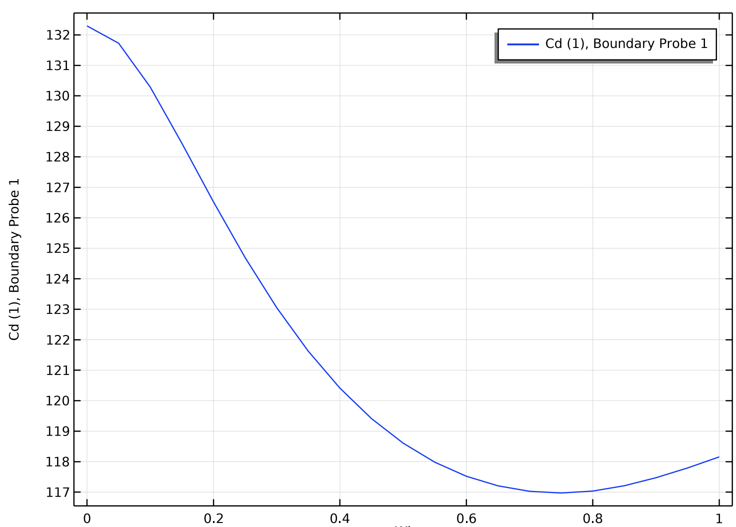

Figure 3: Drag on the cylinder.

References

1. T.J. Craven, J.M. Rees, and W.B. Zimmerman, "Stabilized finite element modelling of Oldroyd-B viscoelastic flow," *COMSOL Conference 2006*, Birmingham, U.K., 2006.

2. M.A. Alves, F.T. Pinho, and P.J. Oliveira, "The flow of viscoelastic fluids past a cylinder: finite-volume high-resolution methods," *J. Non-Newtonian Fluid Mech.*, vol. 97, pp. 207–232, 2001.

Application Library path: `CFD_Module/Single-Phase_Flow/oldroyd_b_viscoelastic`

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>Viscoelastic Flow (vef)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

ROOT

1. In the **Model Builder** window, click the root node.
2. In the root node's **Settings** window, locate the **Unit System** section.
3. From the **Unit system** list, choose **None**.

   The equations you will solve are formulated in dimensionless form.

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 25.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. In the **x** text field, type -10.

6 Click **Build Selected**.

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 **Width** text field, type 6.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. In the **x** text field, type -2.

6 Click **Build Selected**.

Circle 1 (c1)

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

2 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **r1** and **r2** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate selection** toggle button.

5 Select the object **c1** only.

6 Click **Build Selected**.

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
Re	1e-3	0.001	Reynolds number
Wi	0.05	0.05	Weissenberg number
mu_s	0.59	0.59	Solvent relative viscosity
mu_p	1-mu_s	0.41	Polymer relative viscosity

VISCOELASTIC FLOW (VEF)

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 Re.

4 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	Relaxation time
1	mu_p	Wi

Inlet 1

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

2 Select Boundary 1 only.

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

4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

1.5*(1-(y/2)^2)	x
0	y

6 Locate the **Viscoelastic Stress** section. From the list, choose **Symmetric**.

7 In the T_{e0} table, enter the following settings:

2*Wi*mu_p*uy^2	mu_p*uy	0
mu_p*uy	0	0
0	0	0

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2 Select Boundary 11 only.
3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.
4 Clear the **Suppress backflow** check box.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Click the **Zoom Extents** button in the **Graphics** toolbar.
3 Select Boundaries 2, 5, 7, and 9 only.

Proceed to set up boundary probe to compute the drag coefficient.

Boundary Probe 1 (bnd1)

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.
2 In the **Settings** window for **Boundary Probe**, locate the **Probe Type** section.
3 From the **Type** list, choose **Integral**.
4 Locate the **Source Selection** section. Click **Clear Selection**.
5 Select Boundaries 12 and 13 only.
6 In the **Variable name** text field, type `Cd`.
7 Locate the **Expression** section. In the **Expression** text field, type `-2*(vef.T_stressx+ vef.Te_stressx)`.
8 Select the **Description** check box.
9 In the associated text field, type `Cd`.

MESH 1

Size

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 1

1 Select Boundaries 5, 7, 12, and 13 only.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Extra fine**.
4 Click **Build Selected**.

Mapped 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.
2. In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1 and 3 only.

Distribution 1

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 2 and 3 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. From the **Distribution type** list, choose **Predefined**.
5. In the **Number of elements** text field, type `20`.
6. In the **Element ratio** text field, type `5`.
7. Click **Build Selected**.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 9 and 10 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. From the **Distribution type** list, choose **Predefined**.
5. In the **Number of elements** text field, type `25`.
6. In the **Element ratio** text field, type `5`.
7. Select the **Reverse direction** check box.
8. 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**, click to expand the **Study Extensions** section.
3. Select the **Auxiliary sweep** check box.
4. Click **Add**.
5. From the list in the **Parameter name** column, choose **Wi (Weissenberg number)**.
6. Click **Range**.
7. In the **Range** dialog box, type `0` in the **Start** text field.

8 In the **Stop** text field, type `1`.

9 In the **Step** text field, type `0.05`.

10 Click **Replace**.

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

12 From the **Run continuation for** list, choose **No parameter**.

13 From the **Reuse solution from previous step** list, choose **Yes**.

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

RESULTS

Velocity (vef)

To monitor the variation of the drag on the cylinder due to the flow, click on the **Probe Plot** tab once it becomes available.

Once the solution is complete, the plot of the flow field appears. Adjust the view to magnify the region around the cylinder, then add a contour plot for the extra stresses. Follow these steps:

DEFINITIONS

View 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 1**.

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

3 Select the **Lock axis** check box.

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

3 In the **x minimum** text field, type `-2`.

4 In the **x maximum** text field, type `6`.

5 In the **y minimum** text field, type `-4`.

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

RESULTS

Velocity (vef)

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

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

3 From the **View** list, choose **View 1**.

4 Locate the **Data** section. From the **Parameter value (Wi)** list, choose **0.7**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Contour 1

1 Right-click **Velocity (vef)** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Viscoelastic Flow> Viscoelastic variables>Viscoelastic stress tensor - kg/(m·s²)>vef.Te_1xx - Viscoelastic stress tensor, xx component**.

3 Locate the **Levels** section. In the **Total levels** text field, type 40.

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

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

You should now obtain the plot shown in Figure 1.

To plot the stress variation along the cylinder surface and in the wake, follow these steps:

1D Plot Group 4

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Line Graph 1

1 Right-click **1D Plot Group 4** and choose **Line Graph**.

2 Select Boundaries 7, 12, and 13 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1> Viscoelastic Flow>Viscoelastic variables>Viscoelastic stress tensor - kg/(m·s²)>vef.Te_1xx - Viscoelastic stress tensor, xx component**.

4 In the **1D Plot Group 4** toolbar, click **Plot**.

1D Plot Group 4

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

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (Wi)** list, choose **From list**.

4 In the **Parameter values (Wi)** list, select **0.7**.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type 0.

7 In the **x maximum** text field, type 6.

8 In the **y minimum** text field, type `-5`.

9 In the **y maximum** text field, type `115`.

10 In the **1D Plot Group 4** toolbar, click **Plot**.

This will produce the stress plot shown in Figure 2.

Finally, check the complete probe plot of the drag coefficient and compare it to that shown in Figure 3.

Probe Plot Group 3

1 In the **Model Builder** window, right-click **Probe Plot Group 3** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Drag coefficient` in the **New label** text field.

3 Click **OK**.