Created in COMSOL Multiphysics 6.2

Beads-on-String Structure of Viscoelastic Filaments

Introduction

In this example, the thinning of a viscoelastic filament under the action of surface tension is studied. The evolution of the filament radius depends on the relative magnitude of capillary, viscous, and elastic stresses. The interplay of capillary and elastic stresses leads to the formation of very thin and stable filaments between drops, a so-called beads-on-a-string structure. The transformation of the filament shape can be divided into two regimes with distinct time scales. First, for times smaller than the polymer relaxation time, the beads-on-string structure develops. This is followed by exponential thinning of the threads. The fluid is expelled from the threads to the connected beads leading to almost spherical drops. The numerical results show that both transient regimes compare well with experimental measurements.

Model Definition

This example studies the evolution of a long, initially unstretched filament of Oldroyd-B fluid. The flow is considered as axisymmetric. The fluid filament is modeled as a liquid cylinder with a small perturbation of the initial radius of the cylinder, R_0 (Figure 1, $t = 0$). The radius of the column is given by

$$r(z, 0) = R_0\left(1 + \varepsilon\cos\frac{z}{2R_0}\right), \qquad 0 \le \frac{z}{R_0} \le 8\pi$$

where z is the z-coordinate and ε is the perturbation magnitude.

The fluid is a dilute solution of polymer in a Newtonian liquid with solvent of viscosity μ_s. The polymer is characterized by two physical parameters: the viscosity μ_p and the relaxation time λ.

NONDIMENSIONAL FORMULATION

The problem is made dimensionless by using an initial radius of the cylinder, R_0, surface tension coefficient σ, and fluid density ρ. The corresponding inertial-capillary time scale is

$$\tau = \sqrt{\frac{\rho R_0^{\ 3}}{\sigma}}$$

Dynamics of the filament thinning is governed by two dimensionless parameters: Deborah number and Ohnesorge number. The nondimensional polymer solution relaxation time is called the Deborah number:

$$\mathrm{De} = \frac{\lambda_e}{\tau}$$

The Ohnesorge number is the ratio between the inertia-capillary and viscous-capillary time scales:

$$\mathrm{Oh} = \frac{\mu_0}{\sqrt{\rho\sigma R_0}}$$

where $\mu_0 = \mu_\mathrm{s} + \mu_\mathrm{p}$ is the total viscosity of the polymer.

The relative importance of viscous stresses from the solvent can be characterized by the solvent viscosity ratio $\beta = \mu_\mathrm{s}/\mu_0$. The dimensionless viscosities of the solvent and the polymer contribution are $\eta_\mathrm{s} = \beta\ \mathrm{Oh}$ and $\eta_\mathrm{p} = (1 - \beta)\ \mathrm{Oh}$, respectively.

Results and Discussion

Figure 1 shows the evolution of the filament at the different time steps. The results are in good agreement with the experimental and simulation results presented in Ref. 1 for the following set of the dimensionless parameters: $\beta = 0.25$, $\mathrm{Oh} = 3.16$, and $\mathrm{De} = 94.9$.

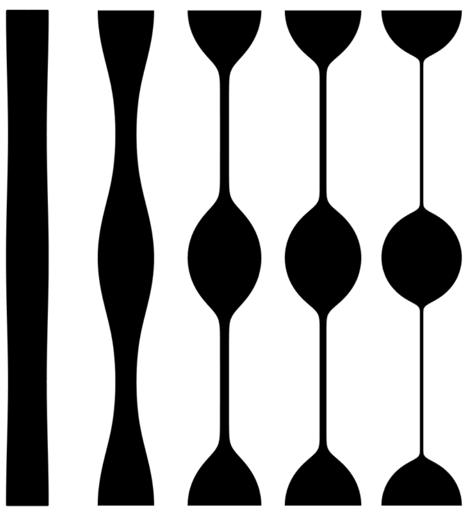

Figure 1: Filament profiles at 5 different dimensionless times: 0, 20, 30, 100, and 300.

The minimum filament radius as a function of time is plotted in Figure 2. After a rapid formation of the beads-on-string structure, the figure shows the slow thinning of the thread.

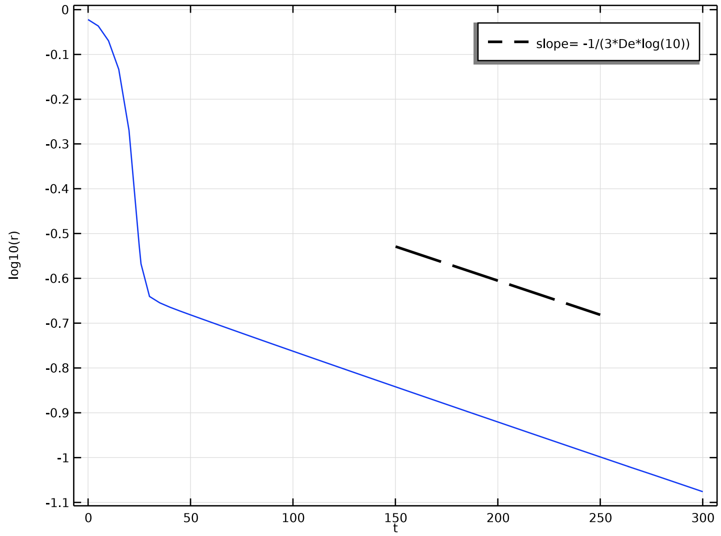

Figure 2: The minimum radius of the filament as a function of time.

The rate of thinning is determined by the balance of surface tension and the elastic forces. Under the assumption of a spatially constant and slender profile, the asymptotic solution of the problem can be derived as

$$r = r_0 \exp\left[-\frac{t}{3\lambda_e}\right] \tag{1}$$

where r_0 is an integration constant that depends on initial conditions. Figure 2 shows that minimum radius evolution curve follows the asymptotic prediction well for large times ($t/\tau > 40$).

For low viscosities or high surface tensions, the formation of the satellite drops can be observed (Ref. 2). To compute the right plot in Figure 3, the finer mesh should be used.

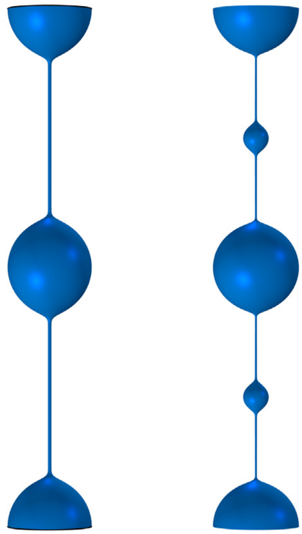

Figure 3: Filament shapes for β = *0.25,* Oh = *3.16,* De = *94.9, and t/*τ = *300 (left) and* β = *0.25,* Oh = *0.4,* De = *0.8, and t/*τ = *19.75 (right).*

Notes About the COMSOL Implementation

The arbitrary Lagrangian–Eulerian (ALE) method is used to handle the dynamics of the deforming geometry and moving boundaries. The Navier–Stokes equations for fluid flow are formulated in the moving coordinates.

The viscosity of the air is neglected, and only the air pressure it taken into account on the interface between the polymer and the air. Therefore, the **Free Surface** feature can be used.

The **Periodic Flow Condition** feature is used on the top and bottom boundaries to mimic the effect of the filament being infinitely long.

References

1. C. Clasen, J. Eggers, M.A. Fontelos, J. Li, and G. McKinley, “The beads-on-string structure of viscoelastic threads,” *J. Fluid Mech.*, vol. 556, pp. 283–308, 2006.

2. E. Turkoz, *High-Resolution Printing of Complex Fluids Using Blister-Actuated Laser-Induced Forward Transfer*, PhD thesis, Department of Mechanical and Aerospace Engineering, Princeton University, 2019.

Application Library path: `Polymer_Flow_Module/Verification_Examples/beads_on_string`

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>Viscoelastic Flow (vef)**.

3 Right-click and choose **Add Physics**.

4 Click **Study**.

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

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 are formulated in dimensionless form.

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
epsilon	0.05	0.05	Radius perturbation
beta	0.25	0.25	Solvent viscosity ratio
Oh	3.16	3.16	Ohnesorge number
De	94.9	94.9	Deborah number
mus	beta*Oh	0.79	Solvent viscosity
mup	(1-beta)*Oh	2.37	Elastic viscosity

GEOMETRY 1

Parametric Curve 1 (pc1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `8*pi`.

4 Locate the **Expressions** section. In the **r** text field, type `1+epsilon*cos(s/2)`.

5 In the **z** text field, type `s`.

6 Click **Build Selected**.

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. In the table, enter the following settings:

r	z
1+epsilon	0
0	0
0	8*pi
1+epsilon	8*pi

5 Click **Build All Objects**.

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 both objects.

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

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Free Deformation**.

2 Select Domain 1 only.

3 In the **Settings** window for **Deforming Domain**, locate the **Smoothing** section.

4 From the **Mesh smoothing type** list, choose **Hyperelastic**.

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**, click to expand the **Discretization** section.

3 From the **Discretization of fluids** list, choose **P1+P1**.

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 `1`.

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

5 In the table, enter the following settings:

Branch	Viscosity	Relaxation time
1	mup	De

Free Surface 1

The viscosity of the air is negligible compared to that of the viscoelastic fluid. Only the air pressure is accounted for on the exterior side of the free surface.

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Free Surface**, locate the **Surface Tension** section.

4 From the **Surface tension coefficient** list, choose **User defined**. In the σ text field, type `1`.

Contact Angle 1

1 In the **Model Builder** window, expand the **Free Surface 1** node, then click **Contact Angle 1**.

2 In the **Settings** window for **Contact Angle**, locate the **Normal Wall Velocity** section.

3 Select the **Constrain wall-normal velocity** check box.

Periodic Flow Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Flow Condition**.

2 Select Boundaries 2 and 3 only.

MOVING MESH

Additionally, mesh boundary conditions are needed on all exterior boundaries (except for the **Free Surface**, which automatically includes the necessary mesh constraint).

Symmetry/Roller 1

1 In the **Moving Mesh** toolbar, click **Symmetry/Roller**.

2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 2 3` in the **Selection** text field.

5 Click **OK** (top, bottom and symmetry boundaries).

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Extremely coarse**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type `0.001`.

7 In the **Resolution of narrow regions** text field, type `8`.

8 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1. In the **Model Builder** window, expand the **Study 1>Step 1: Time Dependent** node, then click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Output times** text field, type `range(0,5,300)`.
4. From the **Tolerance** list, choose **User controlled**.
5. In the **Relative tolerance** text field, type `0.005`.
6. Click to expand the **Study Extensions** section. Select the **Automatic remeshing** check box.
7. In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Before solving the problem, prepare a plot of the velocity. The plot will be shown and updated during the computations.

Mirror 2D 1

1. In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.
2. In the **Settings** window for **Mirror 2D**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Remeshed Solution 1 (sol2)**.
4. Click **Plot**.

Velocity (vef)

1. In the **Model Builder** window, expand the **Results>Velocity (vef)** node, then click **Velocity (vef)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Mirror 2D 1**.

STUDY 1

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.
3. Select the **Plot** check box.
4. From the **Update at** list, choose **Time steps taken by solver**.

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

3 From the **Method** list, choose **Generalized alpha**.

4 From the **Maximum step constraint** list, choose **Expression**.

5 In the **Maximum step** text field, type `comp1.vef.dt_CFL`.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Automatic Remeshing**.

7 In the **Settings** window for **Automatic Remeshing**, locate the **Condition for Remeshing** section.

8 From the **Condition type** list, choose **Distortion**.

9 In the **Stop when distortion exceeds** text field, type `1.05`.

10 From the **Remesh at** list, choose **Last output from solver before stop**.

11 Locate the **Remesh** section. From the **Consistent initialization** list, choose **On**.

12 Click **Compute**.

RESULTS

Velocity (vef)

The default plots show the velocity and the pressure. Continue with a visualization of the filament shape at selected times.

1 In the **Model Builder** window, under **Results** right-click **Velocity (vef)** and choose **Duplicate**.

Shape

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

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

Surface

1 In the **Model Builder** window, expand the **Shape** node, then click **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

Compare the resulting plot with Figure 1 at t = 0, 20, 30, 100, and 300.

Line Minimum 1

Reproduce the plot of the minimum filament radius (Figure 2) as follows:

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Minimum>Line Minimum**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Minimum**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`log10(r)`		

5 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Remeshed Solution 1 (sol2)**.

6 Click **Evaluate**.

TABLE 1

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

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

RESULTS

Minimum Radius

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

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

3 Click to expand the **Title** section. Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 Select the **y-axis label** check box.

5 In the **x-axis label** text field, type `t`.

6 In the **Minimum Radius** toolbar, click **Plot**.

Grid 1D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

2 In the **Settings** window for **Grid 1D**, locate the **Parameter Bounds** section.

3 In the **Minimum** text field, type 50.

4 In the **Maximum** text field, type 300.

Minimum Radius

1 In the **Model Builder** window, collapse the **Results>Minimum Radius** node.

2 In the **Model Builder** window, click **Minimum Radius**.

Function 1

1 In the **Minimum Radius** toolbar, click **More Plots** and choose **Function**.

2 In the **Settings** window for **Function**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `-1/(3*De*log(10))*x-0.3`.

4 Locate the **x-Axis Data** section. In the **Expression** text field, type x.

5 In the **Lower bound** text field, type 150.

6 In the **Upper bound** text field, type 250.

7 Locate the **Data** section. From the **Dataset** list, choose **Grid 1D 1**.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

9 From the **Width** list, choose **2**.

10 From the **Color** list, choose **Black**.

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

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

13 In the table, enter the following settings:

Legends
slope= -1/(3*De*log(10))

14 Click to expand the **Title** section. From the **Title type** list, choose **None**.

15 In the **Minimum Radius** toolbar, click **Plot**.