INTRODUCTION TO

Polymer Flow Module

COMSOL

Introduction to the Polymer Flow Module

Version: COMSOL 6.2

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Contents

The Polymer Flow Module

Non-Newtonian fluids are found in a great variety of processes in the polymer, food, pharmaceutical, cosmetics, household, and fine chemicals industries. Examples of these fluids are coatings, paints, yogurt, ketchup, colloidal suspensions, aqueous suspensions of drugs, lotions, creams, shampoo, suspensions of peptides, and proteins, to mention a few. The Polymer Flow Module is an optional add-on package for COMSOL Multiphysics® that is designed to aid engineers and scientists in simulating flows of non-Newtonian fluids with viscoelastic, thixotropic, shear-thickening, or shear-thinning properties. Simulations can be used to gain physical insight into the behavior of complex fluids, reduce prototyping costs, and speed up development. The Polymer Flow Module allows users to quickly and accurately model single-phase flows, multiphase flows, nonisothermal flows, and reacting flows of Newtonian and non-Newtonian fluids.
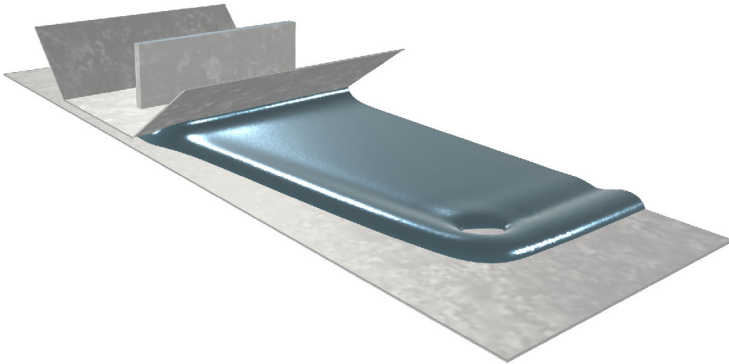

Figure 1: Coating flow simulation with a power-law fluid. The thickness of the coating layer can be controlled by varying the speed of the lower wall relative to that of the injection slot. Span-wise thickness variations and edge effects may be minimized by optimizing the polymer composition in the coating fluid.

The Polymer Flow Module can solve stationary and time-dependent flows in two-dimensional and three-dimensional domains. Formulations suitable for different types of flow are set up as predefined Fluid Flow interfaces, referred to as

physics interfaces. These Fluid Flow interfaces use physical quantities, such as velocity and pressure, and physical properties, such as density and viscosity, to define a fluid flow problem. There are different physics interfaces available for a wide range of flows. For instance, there are the Laminar Flow, Creeping Flow, Viscoelastic Flow, Brinkman Equations, Heat Transfer, and Transport of Diluted Species interfaces. The physics interfaces can be combined with the interfaces in the Mathematics branch (Level Set, Phase Field in Fluids and Ternary Phase Field), or defined on arbitrary Lagrangian-Eulerian (ALE) frame to simulate two- and three-phase flows, and rotating flows. The Polymer Flow Module includes a set of predefined multiphysics couplings for facilitating the setup of multiphysics simulations: Nonisothermal Flow; Reacting Flow; Two-Phase Flow, Level Set; Two-Phase Flow, Phase Field; Three-Phase Flow, Phase Field; Brinkman Equations, Level Set; and Rotating Machinery, Fluid Flow.

For each of the physics interfaces, the underlying physical principles are expressed in the form of partial differential equations, together with corresponding initial conditions and boundary conditions. The COMSOL Multiphysics software was designed to emphasize the physics by providing users with the equations solved by each feature and offering the user full access to the underlying equation system. There is also tremendous flexibility in adding user-defined equations and expressions to the system. For example, to model the curing state during a mold injection, a Stabilized Convection Diffusion Equation interface can be added from the Mathematics branch — no scripting or coding is required. When COMSOL Multiphysics compiles the equations, the complex couplings generated by these user-defined expressions are automatically included in the equation system. The equations are then solved using the finite element method and a range of industrial-strength solvers. Once a solution is obtained, a vast range of postprocessing tools are available for analyzing the data, and predefined plots are automatically generated for visualizing the results. COMSOL Multiphysics offers the flexibility to evaluate a wide range of physical quantities, including predefined quantities such as the pressure, velocity, shear rate, or the vorticity (available through easy-to-use menus), as well as arbitrary user-defined expressions.

To set up a fluid flow simulation, the geometry is first defined in the software. Then appropriate materials are selected and suitable physics interfaces, together with the appropriate multiphysics couplings, are added. Initial conditions and boundary conditions are set up within the physics interfaces. Next, the mesh is defined — in many cases the default mesh, which is produced from physics-dependent defaults, will be appropriate for the problem. A solver is selected, again with defaults appropriate for the relevant physics interfaces, and the problem is solved. Finally, the results are visualized. All of the features needed for these steps can be accessed from within the COMSOL Desktop.

Newtonian and Non-Newtonian Fluids

A fluid may be characterized according to its response under the action of a shear stress. A Newtonian fluid has a linear relationship between shear stress and shear rate, with the line passing through the origin. The constant of proportionality is referred to as the viscosity of the fluid and may depend on temperature, pressure, and composition. For a non-Newtonian fluid, the curve for shear stress versus shear rate is nonlinear or does not pass through the origin. If the curve is shifted away from the origin, the fluid has a yield stress. If the curve bends toward the shear-rate axis, the fluid is said to be shear thinning (pseudoplastic), but if it instead bends toward the shear-stress axis, it is said to be shear thickening (dilatant). The relationship may also contain time derivatives of the shear rate to model memory effects (thixotropy), and even parallel viscous and elastic responses. We refer to the latter as *viscoelastic non-Newtonian fluids* and all others as *inelastic non-Newtonian fluids*. For the inelastic non-Newtonian models, it is possible to define an apparent viscosity from a generalized Newtonian relationship between the deviatoric stress tensor and the strain-rate tensor.
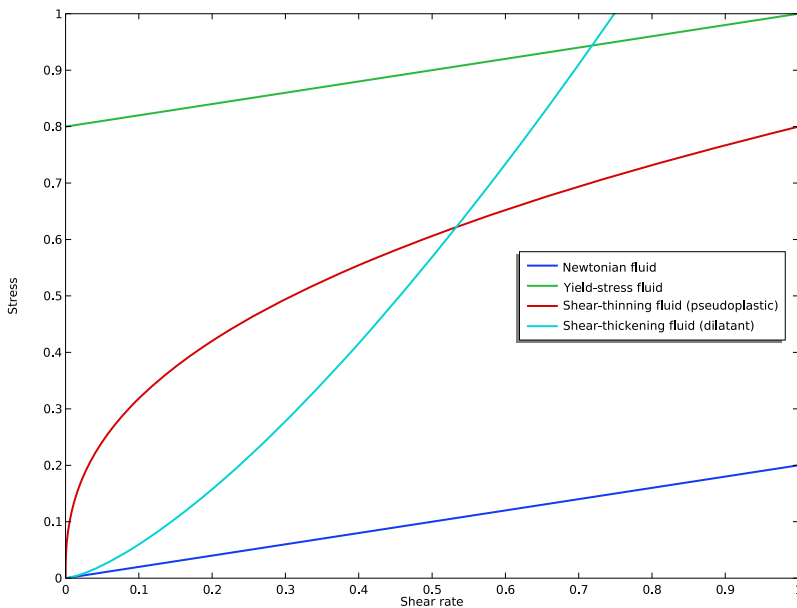

Figure 2: Conceptual behavior of shear stress versus shear rate for Newtonian, yield-stress, shear-thinning, and shear-thickening fluids.

The apparent viscosity will then be a function of the first invariant of the strain-rate tensor. In simple shear flows, this invariant reduces to the shear rate.

The simplest viscoelastic model in the Polymer Flow Module is the Oldroyd-B model which has a constant viscosity and a constant first normal-stress coefficient. This is conceptually visualized as a Hookean spring in series with a dashpot in Figure 3.

Figure 3: A Hookean spring connected to a dashpot.

To ensure material-frame indifference, an upper-convected time derivative is used to describe the transport of the additional stress due to the spring and dashpot. Multiple branches with different relaxation times and viscosities can be added in order to include multimodal responses. In the Oldroyd-B model, the springs are infinitely extensible, which can lead to unphysical behavior. The other viscoelastic models in the module include nonlinear-spring effects and can also model shear-thinning behavior.

When applying a viscoelastic model, it is important to keep in mind that the mathematical formulation may become ill-posed at sharp corners. One remedy is to use a fillet operation to smooth the geometry at such points. As always when working with complex fluid flow simulations, it is advisable to start with a simple model and then switch to a more complicated model if the simple model does not capture the expected flow phenomena. In most industrial applications, inelastic non-Newtonian models produce sufficiently accurate results at a reasonable computational cost.

The physics interfaces in this module are based on the laws for conservation of momentum, mass, and energy in fluids. The different flow models contain different combinations and formulations of the conservation laws that apply to the physics of the flow field. These laws of physics are translated into partial differential equations and are solved together with the specified initial conditions and boundary conditions.

A physics interface defines a number of features. These features are used to specify the fluid properties, initial conditions, boundary conditions, and possible constraints. Each feature represents an operation describing a term or condition in the conservation equations. Such a term or condition can be defined on a geometric entity of the component, such as a domain, boundary, edge (for 2D components), or point.

Figure 4 shows the Model Builder, including a Laminar Flow interface, and the Settings window for the selected Fluid Properties 1 feature node. The Fluid Properties 1 node adds the marked terms to the component equations in a selected geometry domain. Furthermore, the Fluid Properties 1 feature may link to the Materials feature node to obtain physical properties such as density and constitutive parameters — in this case, rubber modeled with a power-law fluid. The fluid properties, defined by the Rubber, Power law material, can be functions of the modeled physical quantities, such as pressure and temperature. In the same way, the Wall 1 node adds the boundary conditions at the walls of the fluid domain.
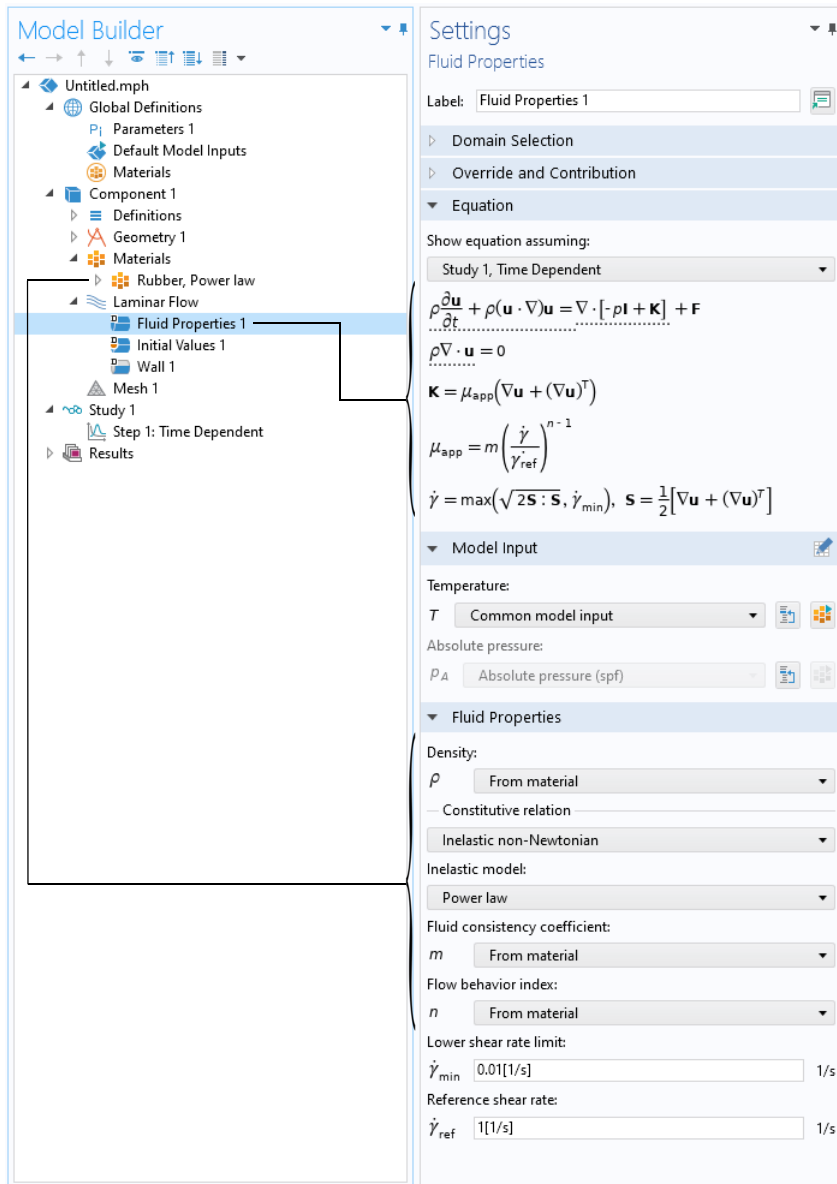
Figure 4: The Model Builder including a Laminar Flow interface (left), and the Settings window for Fluid Properties for the selected feature node (right). The Equation section in the Settings window shows the component equations and the terms added by the Fluid Properties 1 node. The added terms are underlined with a dotted line. The arrows also explain the link between the Materials node and the values for the fluid properties.

The Polymer Flow Module includes a number of Fluid Flow interfaces for different types of flow. It also includes Chemical Species Transport interfaces for reacting flows in multicomponent solutions, and physics interfaces for heat transfer in solids and in fluids found under the Heat Transfer branch.

Figure 5 shows the Polymer Flow Module interfaces as they are displayed when you add a physics interface (see also Physics Interface Guide by Space Dimension and Study Type for further information). A short description of the physics interfaces follows.

SINGLE-PHASE FLOW

The Creeping Flow interface (≡) approximates the Navier–Stokes equations for very low Reynolds numbers. This is often referred to as *Stokes flow* and is applicable when viscous effects are dominant, such as in very small channels or microfluidics devices.

The Laminar Flow interface (≋) is primarily applied to flows at low to intermediate Reynolds numbers. This physics interface solves the Navier–Stokes equations for incompressible, weakly compressible, and compressible flows (up to Mach 0.3). The Laminar Flow interface also allows for simulation of non-Newtonian fluids.

The Rotating Machinery, Laminar Flow interface (≋) combines the Laminar Flow interface and a Rotating Domain, and is applicable to fluid flow problems where one or more of the boundaries rotate in, for example, mixers and around propellers. The physics interface supports incompressible, weakly compressible and compressible (Mach < 0.3) laminar flows of Newtonian and non-Newtonian fluids.

The Viscoelastic Flow interface (≋) is used for simulating incompressible isothermal flow of viscoelastic fluids. It solves the continuity equation, the momentum equation, and a constitutive equation that defines the elastic stresses. The physics interface supports the formulation of constitutive equation in terms of stress or conformation. There are several predefined models to model the elastic stresses: Oldroyd-B, FENE-P, FENE-CR, Giesekus, Rolie-Poly, LPTT, and EPTT.

The Potential Flow interface (⊙) can be used to model irrotational flow, or to get initial values for other fluid-flow interfaces.

POROUS MEDIA AND SUBSURFACE FLOW

The Brinkman Equations interface (▣) models flow through a porous medium where shear stresses cannot be neglected. The physics interface supports both the Stokes–Brinkman formulation, suitable for very low flow velocities, and Forchheimer drag, which is used to account for effects at higher velocities. The

fluid can be either incompressible or compressible, provided that the Mach number is less than 0.3.

The Free and Porous Media Flow, Brinkman interface (🔵) models both porous media (using the Brinkman Equations) and laminar flow, automatically providing the coupling between them.
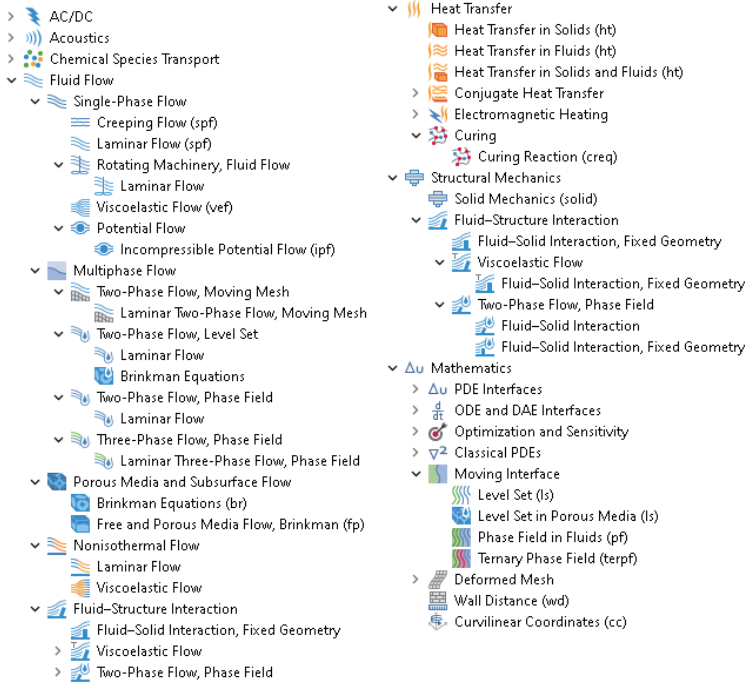

Figure 5: The physics interfaces for the Polymer Flow Module, as shown in the Model Wizard.

MULTIPHASE FLOW

The Two-Phase Flow, Level Set interface (🌊); the Two-Phase Flow, Phase Field interface (🌊); and the Two-Phase Flow, Moving Mesh interface (🌊) are used to model two fluids separated by a fluid–fluid interface. The moving interface is tracked in detail using the level set method, the phase field method, or by a moving mesh, respectively. The level set and phase field methods use a fixed mesh and solve additional equations to track the interface location. The moving mesh method solves the Navier–Stokes equations on a moving mesh with boundary conditions to represent the interface. In this case, equations must be solved for the mesh deformation. Since a surface in the geometry is used to represent the interface between the two fluids in the Moving Mesh interface, the interface itself cannot break up into multiple disconnected surfaces. This means that the Moving

Mesh interface cannot be applied to problems such as droplet formation in inkjet devices (in these applications, the Level Set or Phase Field in Fluids interfaces are appropriate). These physics interfaces support incompressible flows, where one or both fluids can be non-Newtonian.

The Two-Phase Flow, Level Set, Brinkman Equations interface ( ) is designed to track the interface between two immiscible fluids within a porous medium.

The Laminar Three-Phase Flow, Phase Field interface ( ) models laminar flow of three incompressible phases, which can be either Newtonian or non-Newtonian. The moving fluid–fluid interfaces between the three phases are tracked in detail using the phase-field method.

NONISOTHERMAL FLOW

The Nonisothermal Flow, Laminar Flow interface ( ) is primarily applied for modeling flow at low to intermediate Reynolds numbers in situations where the temperature and flow fields have to be coupled. A typical example is natural convection, where thermal buoyancy forces drive the flow. This is a multiphysics interface for which the nonlocal couplings between fluid flow and heat transfer are set up automatically.

The Nonisothermal Flow, Viscoelastic Flow interface ( ) is used to model nonisothermal flow of viscoelastic fluid. The couplings between fluid flow and heat transfer are set up automatically. The irreversible losses due to transformation of mechanical energy into internal energy can be added.

REACTING FLOW

The Laminar Flow interface ( ) under the Reacting Flow branch combines the functionality of the Single-Phase Flow and Transport of Diluted Species interfaces. The physics interface is primarily applied to model flow at low to intermediate Reynolds numbers in situations where the mass transport and flow fields have to be coupled.

CURING

The Curing Reaction interface ( ) is intended for modeling curing of polymers during the heat treatment. For the thermosets, the viscosity depends both on the temperature and on the degree of cure. The Curing Reaction interface includes several chemoviscosity models to relate the viscosity to the degree of cure (conversion) and temperature.

FLUID–STRUCTURE INTERACTION

The Fluid–Structure Interaction interfaces ( ) couple a Single-Phase Flow, Viscoelastic Flow or Two-Phase Flow, Phase Field interface to a Solid Mechanics interface for studies of deformation induced by fluid forces.

Physics Interface Guide by Space Dimension and Study Type

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
⠿ Chemical Species Transport				
Transport of Diluted Species[1]		tds	all dimensions	stationary; time dependent
≋ Reacting Flow				
Laminar Flow, Diluted Species[1]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
≋ Fluid Flow				
≋ Single-Phase Flow				
Creeping Flow	≋	spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Laminar Flow	≋	spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Viscoelastic Flow	≋	vef	3D, 2D, 2D axisymmetric	stationary; time dependent
≋ Rotating Machinery, Fluid Flow				
Rotating Machinery, Laminar Flow		spf	3D, 2D	frozen rotor; time dependent
● Potential Flow				
Incompressible Potential Flow	●	ipf	3D, 2D, 2D axisymmetric	stationary

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Multiphase Flow				
Two-Phase Flow, Moving Mesh				
Laminar Two-Phase Flow, Moving Mesh		—	3D, 2D, 2D axisymmetric	time dependent
Two-Phase Flow, Level Set				
Laminar Two-Phase Flow, Level Set		—	3D, 2D, 2D axisymmetric	time dependent with phase initialization
Two-Phase Flow, Level Set, Brinkman Equations		—	3D, 2D, 2D axisymmetric	time dependent with initialization
Two-Phase Flow, Phase Field				
Laminar Two-Phase Flow, Phase Field		—	3D, 2D, 2D axisymmetric	time dependent with phase initialization
Three-Phase Flow, Phase Field				
Laminar, Three-Phase Flow, Phase Field		—	3D, 2D, 2D axisymmetric	time dependent with phase initialization
Porous Media and Subsurface Flow				
Brinkman Equations		br	3D, 2D, 2D axisymmetric	stationary; time dependent
Free and Porous Media Flow, Brinkman		fp	3D, 2D, 2D axisymmetric	stationary; time dependent
Nonisothermal Flow				
Laminar Flow[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; Stationary, one-way NITF; time dependent, one-way NITF

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Viscoelastic Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF
Fluid-Structure Interaction				
Fluid–Solid Interaction, Fixed Geometry[2]		—	3D, 2D, 2D axisymmetric	stationary; stationary, one way; time dependent; time dependent, one way
Fluid–Solid Interaction, Viscoelastic Flow, Fixed Geometry[2]		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
Fluid–Solid Interaction, Two-Phase Flow, Phase Field, Fixed Geometry[2]		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
Heat Transfer				
Heat Transfer in Fluids[1]		ht	all dimensions	stationary; time dependent
Heat Transfer in Solids and Fluids[1]		ht	all dimensions	stationary; time dependent
Curing				
Curing Reaction		creq	3D, 2D, 2D axisymmetric, 1D	stationary; time dependent
Moving Interface				
Level Set		ls	all dimensions	time dependent with phase initialization
Level Set in Porous Media		ls	all dimensions	time dependent with phase initialization
Phase Field in Fluids		pf	all dimensions	time dependent; time dependent with phase initialization

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Ternary Phase Field		terpf	3D, 2D, 2D axisymmetric	time dependent

[1] This physics interface is included with the core COMSOL Multiphysics software but has added functionality for this module.

[2] This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.

Tutorial Model: 2D Non-Newtonian Slot-Die Coating

Achieving uniform coating quality is important in several different areas, such as optical coatings, semiconductor and electronics industry, technologies utilizing thin membranes, and surface treatment of metals. Bad coating quality will compromise the performance of the products or lead to complete failure in some cases.

Several different coating processes exist. This tutorial investigates the performance of a slot-die coating process, a so-called premetered coating method. In this process, the coating fluid is suspended from a thin slot die to a moving substrate. The final coating layer thickness is evaluated from the continuity relationship for a coating liquid. Therefore, the thickness of the liquid layer is determined by the slot gap, the coating fluid inlet velocity, and the substrate speed.

The final goal of coating processes is to achieve a defect-free film of a desired thickness. However, manufacturing the uniform coating is not a trivial task, as various flow instabilities or defects such as bubbles, ribbing, and rivulets are frequently observed in the process. The die geometry, the size of the slot and height above the substrate, and the non-Newtonian fluid nature of the coating fluid are important to consider.

This tutorial demonstrates how to model the fluid flow in a polymer slot-die coating process using the Laminar Two-Phase Flow, Phase Field interface and an inelastic non-Newtonian power law model for the polymer fluid.

Model Geometry
A typical setup of the slot-die coating process is shown in Figure 6.
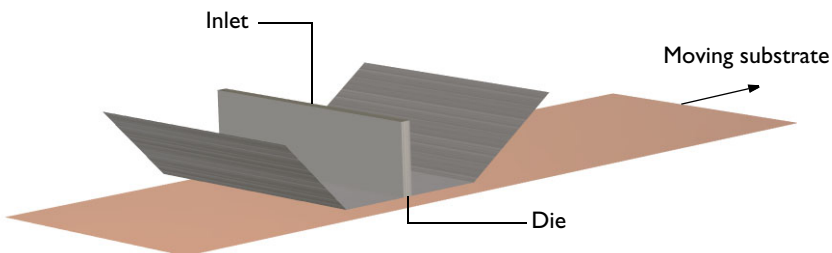
Figure 6: Typical geometry for a slot-die coating process with the slot die positioned over a substrate.

This model uses a 2D cross section of the die shown in Figure 6, assuming out-of-plane invariance. See also *Slot-Die Coating with Channel Defect* in the Polymer Flow Module Application Library for a 3D model of slot-die coating. The

inlet for the coating fluid is at the top of the die, as shown in Figure 7, and there are open boundaries at both ends. The bottom boundary is the coating substrate, which is moving at the coating velocity.
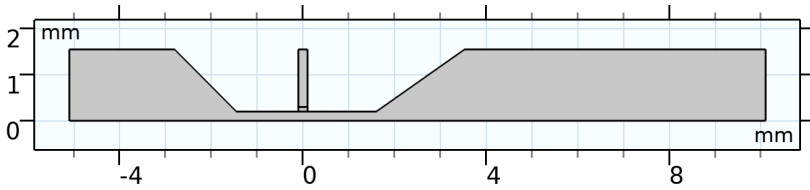
Figure 7: Model geometry. 2D cross section of a slot die.

The geometrical and material parameters in this model are taken from Ref. 1.

Domain Equations and Boundary Conditions

The flow in this model is laminar, so a Laminar Flow interface will be used together with a Phase Field in Fluids interface to track the interface between the air and the polymer fluid. The coupling of these two interfaces is handled by the Two-Phase Flow, Phase Field multiphysics interface. In this model, the air is specified as a Newtonian fluid, and the coating fluid is a non-Newtonian power law fluid.

The inlet fluid velocity is increases smoothly from 0 m/s to 0.1 m/s. Both the upstream and downstream boundaries of the model are specified as open boundaries. The corresponding inlet and outlet boundary conditions must also be set in the Phase Field in Fluids interface together with the initial values for both fluids to correctly define the position of the initial interface. For the moving substrate, a Moving Wall boundary condition with a Navier slip condition is used.

Results and Discussion

shows the evolution of the coating fluid interface for $t = 0.03$ s, $t = 0.06$ s, and $t = 0.2$ s.
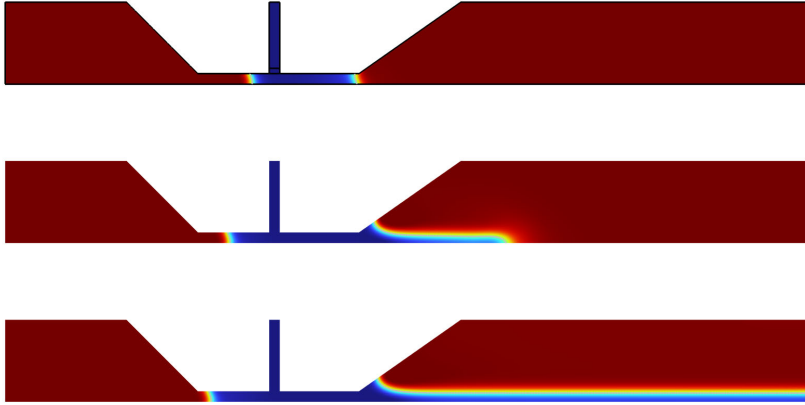
Figure 8: Coating fluid interface at $t = 0.03$ s, $t = 0.06$ s, and $t = 0.2$ s (top to bottom).

The coating film attains a constant thickness downstream of the die at $t = 0.2$ s. The film forms upstream and downstream menisci with the upstream and downstream walls of the die. As the substrate speed increases or the inlet velocity decreases, the upstream meniscus is pulled closer to the slot, eventually causing

defects in the coating film. The evolution of the film thickness and position of the upstream meniscus as a function of time is shown in Figure 9.
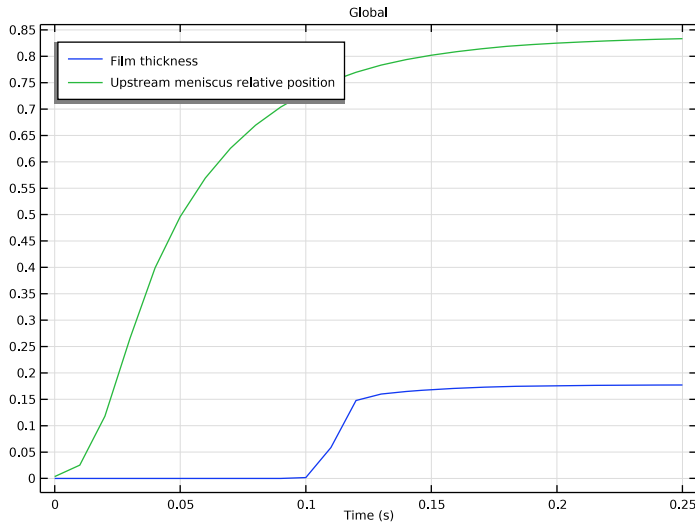
Figure 9: Film thickness and upstream meniscus position as a function of time.

By changing the geometry, the inlet velocity, and the wall velocity, it is easy to explore the sensitivity of the design parameters toward the film thickness and coating velocity for a variety of fluid properties in a fast and efficient manner.

Reference

1. K.L. Bhamidipati, *Detection and elimination of defects during manufacture of high-temperature polymer electrolyte membranes*, PhD Thesis, Georgia Institute of Technology, 2011.

Model Wizard

1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new

COMSOL Multiphysics model or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

If COMSOL Multiphysics is already open, you can start the Model Wizard by selecting New 🗋 from the File menu and then click Model Wizard ◈.

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.

2 In the Model Wizard window, click ⚘ 2D.

3 In the Select Physics tree, select
Fluid Flow>Multiphase Flow>Two-Phase Flow, Phase Field>Laminar Flow
≋ .

4 Click Add.

5 Click → Study.

6 In the tree, under Preset Studies for Selected Multiphysics, click
Time Dependent with Phase Initialization ⎍ .

7 Click ☑ Done.

Global Definitions

Load the model parameters from a text file.

Parameters 1

1 In the Home toolbar click Parameters P¡ and select Parameter 1 P¡ .

Note: On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

2 Click 📂 Load from File.

3 Browse to the model's Application Libraries folder and double-click the file slot_die_coating_2d_parameters.txt.

Follow the steps below to calculate the parameters for the power-law fluid model based on measurement data in this example.

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 Click 📂 Load from File.

4 Browse to the model's Application Libraries folder and double-click the file
`slot_die_coating_2d_viscosity_input.txt`.

5 Locate the Column Settings section. In the table, click to select the cell at row number 1 and column number 1.

6 In the Name text field, type `gammadot`.

7 In the Unit text field, type `1`.

8 In the table, click to select the cell at row number 2 and column number 1.

9 In the Name text field, type `mu_app`.

10 In the Expression text field, type `m*max(gammadot,0.01)^(n-1)`.

11 In the Unit text field, type `Pa*s`.

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

PARAMETER	VALUE
m	1
n	1

13 Click ⊿ Fit Parameters. The computed parameter values from the least-squares fit appear in the Parameters table.

14 Click 🔳 Create Plot to see how close data points are to the fitted function.

mu_app(gammadot) (Pa*s)
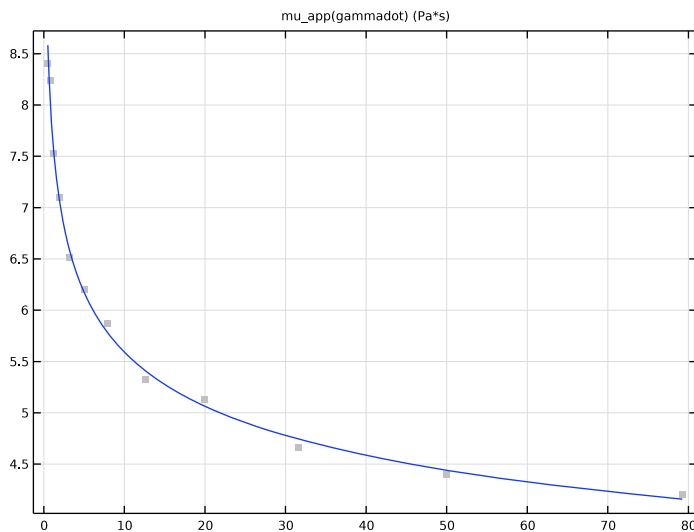
Create a step function to use for ramping up the inlet velocity. To improve convergence, define a smoothing transition zone to gently increase the inlet velocity from zero.

Step 1

1 In the Home toolbar, click ꜰ(ₓ) Functions and choose Global>Step ⌐ .

2 Locate the Parameters section. In the Location text field, type `0.01`.

3 Click to expand the Smoothing section. In the Size of transition zone text field, type `0.02`.

Geometry 1

You can build the slot-die geometry from geometric primitives. Here, you instead import the geometry sequence from the model file.

Note: The location of the file used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path might be similar to
`C:\Program Files\COMSOL\COMSOL62\Multiphysics\applications\`.

1 In the Geometry toolbar choose Insert Sequence 📑.

2 Browse to the application library folder and double-click the file
`\Polymer_Flow_Module\Tutorials\slot_die_coating_2d.mph`.

3 Go to the Home toolbar and click Build All 🔳 .

Compare the resulting geometry to Figure 7.

Definitions

Next define integration operators. First, define an integration coupling that integrates along the outlet boundary to calculate the film thickness. Then, define a coupling operator that integrates along the upstream die lip. You will use it later for the integration of the volume fraction along the boundary to evaluate the location of the upstream meniscus.

Integration 1 (intop1)

1 In the Model Builder window, expand the Component 1 (comp1)>Definitions node ≡ .

2 Right-click Definitions and choose Nonlocal Couplings>Integration ∫du .

3 In the Settings window for Integration, locate the Source Selection section.

4 From the Geometric entity level list, choose Boundary.

5 Select Boundary 16 only.

Integration 2 (intop2)

1 In the Definitions toolbar, click 🔗 Nonlocal Couplings and choose Integration.

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

3 From the Geometric entity level list, choose Boundary.

4 Select Boundary 5 only.

Now specify the material properties for the model. The material can be created by clicking the Add Multiphase Material in the Two-phase flow, Phase Field-coupling.

Two-Phase Flow, Phase Field

1 In the Model Builder window, under Component 1 (comp1)>Multiphysics click Two-Phase Flow, Phase Field 1 (tpf1) 🌊.

2 In the Settings window for Two-Phase Flow, Phase Field, locate the Material Properties section.

3 Click ▦ Add Multiphase Material.

▾ Material Properties

Multiphase material:

| Multiphase Material 1 (mpmat1) | ∨ | 🗐 ▦ |
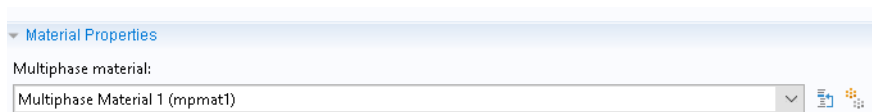
Materials

Now, define the two fluid for the model: air and coating fluid.

1 In the Model Builder window, under
Component 1 (comp1)>Materials>Multiphase Material 1 (mpmat1) click Phase 1 (mpmat1.phase1).

2 In the Settings window for Phase, locate the Link Settings section.

3 Click Add Material from Library . This button is found when expanding the options next to the Material list.

4 Add Material to Phase 1 (mpmat1.phase1)

5 Go to the Add Material to Phase 1 (mpmat1.phase1) window.

6 In the tree, select Built-in>Air. Click OK.

Settings

Phase

Label: Phase 1

Name: phase1

▼ Volume Fraction

$$V_{f,i} = 1 - \sum_{j \neq i}^{n} V_{f,j}$$

▼ Link Settings

Material:

Locally defined

　　Add Material from Library

　　Blank Material

▼ Material Properties

▼ Material Contents

	Property	Variable	Value	Unit	Property group
⊘	Density	rho		kg/m³	Basic
⊘	Dynamic viscosity	mu		Pa·s	Basic

Continue by adding the coating fluid.

1 In the Model Builder window, under Component 1 (comp1)>Materials>Multiphase Material 1 (mpmat1) click Phase 2 (mpmat1.phase2).

2 In the Settings window for Phase, locate the Link Settings section.

3 Click Blank Material . This button is found when expanding the options next to the Material list.

Coating Fluid Parameters

1 In the Model Builder window, under Global Definitions>Materials click Material 2 (mat2).

2 In the Settings window for Material, type `Coating Fluid` in the Label text field.

3 Locate the Material Properties section. In the Material properties tree, select Fluid Flow>Inelastic Non-Newtonian>Power Law.

4 Right-click and choose Add to Material.

Settings

Material

Label: Coating Fluid

Name: mat2

▼ Material Properties

> ▦ Electromagnetic Models
> ▦ Equilibrium Discharge
> ▦ External Material Parameters
> ▦ Fluid Flow
>> ▦ Inelastic Non-Newtonian
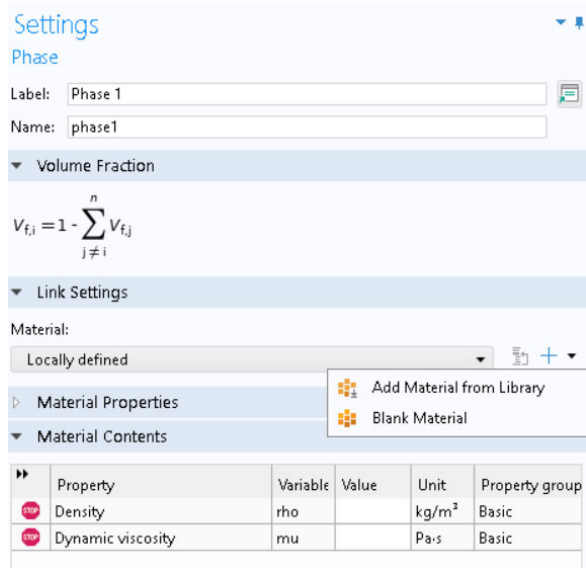>>> ▦ Bingham-Papanastasiou
>>> ▦ Carreau Model
>>> ▦ Carreau-Yasuda
>>> ▦ Casson-Papanastasiou
>>> ▦ Cross Model
>>> ▦ Cross-Williamson
>>> ▦ DeKee Model
>>> ▦ Ellis Model
>>> ▦ Herschel-Bulkley-Papanastasiou
>>> ▦ Power Law
>>> ▦ Robertson-Stiff-Papanastasiou
>>> ▦ Sisko

Use the parameters from the least-squares fit `lsq1.m` and `lsq1.n` to define material coefficients.

5 Locate the Material Contents section. In the table, enter the following settings:

PROPERTY	VARIABLE	VALUE	UNIT	PROPERTY GROUP
Fluid consistency coefficient	m_pow	`lsq1.m`	Pa·s	Power law
Flow behavior index	n_pow	`lsq1.n`	l	Power law
Density	rho	`1400`	kg/m³	Basic

Mixing Rules

When working with fluids that have large viscosity and density ratios, switching from the default linear method for the properties averaging can increase the performance. In this model, smoothed Heaviside functions are used to average the fluid properties

$$\rho = \rho_1 - (\rho_2 - \rho_1)H\left(\frac{V_{f,2} - 0.5}{l_\rho}\right)$$

where $V_{f,2}$ is the volume fraction of fluid 2 and l_ρ is a mixing parameter. A similar expression is used for the dynamic viscosity.

1 In the Model Builder window, under Component 1 (comp1)>Materials click Multiphase Material 1 (mpmat1).

2 In the Settings window for Multiphase Material, locate the Material Content section.

3 Click to select row number 1 in the table, to select mixing rule for density.

4 In the Edit Mixing Rule dialog box, choose Heaviside function from the Mixing rule list. In the l_{mix} text field, type 0.9.

5 Click ⬇ Next Row to continue with mixing rule for viscosity.

6 From the Mixing rule list, choose Heaviside function. Click OK.
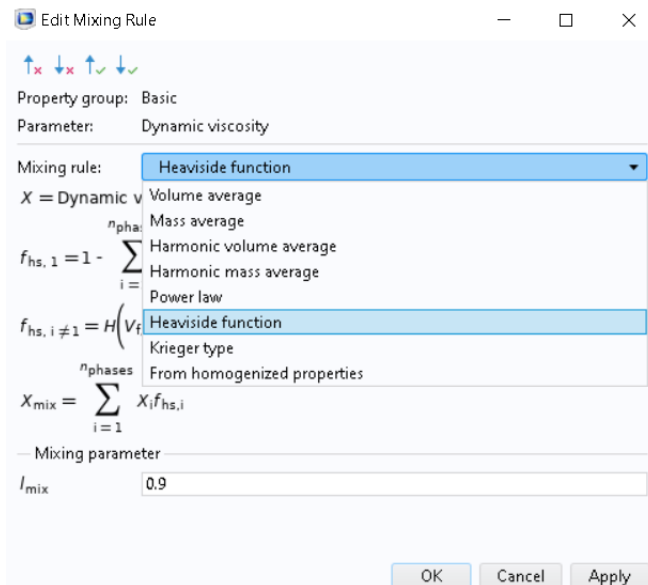
The mixing parameter can be decreased to sharpen the interface, but that will increase the computation time for this example.

Laminar Flow

Now, set up the physics of the problem by defining the domain physics conditions and the boundary conditions.

Wall 2

1 In the Model Builder window, under Component 1 (comp1) right-click Laminar Flow (spf) and choose Wall.

2 Select Boundary 2 only.

3 In the Settings window for Wall, locate the Boundary Condition section.

4 From the Wall condition list, choose Navier slip.

5 Click to expand the Wall Movement section. Select the Sliding wall check box.

6 In the U_w text field, type `-U_wall`.

Inlet 1

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

2 Select Boundary 10 only.

3 n the Settings window for Inlet, locate the Boundary Condition section.

4 From the list, choose Fully developed flow.

5 Locate the Fully Developed Flow section. In the U_{av} text field, type `step1(t[1/s])*U_in`.

Open Boundary 1

1 In the Physics toolbar, click ⬭ Boundaries and choose Open Boundary.

2 Select Boundaries 1 and 16.

Phase Field in Fluids (pf)

The initial interface between the coating fluid and the air is automatically assigned to the boundaries between the two initial value domains. Set up the initial coating fluid domain in the inlet channel.

Initial Values, Fluid 2

1 In the Model Builder window, under Component 1 (comp1)>Phase Field (pf) click Initial Values, Fluid 2.

2 Select Domain 3 only.

Wetted Wall 1

1 In the Model Builder window, click Wetted Wall 1.

2 In the Settings window for Wetted Wall, locate the Wetted Wall section.

3 In the θ_w text field, type `68.5[deg]`.

Inlet 1

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

2 In the Settings window for Inlet, locate the Phase Field Condition section.

3 From the list, choose Fluid 2 ($\phi = 1$).

4 Select Boundary 10 only.

Outlet 1

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

2 Select Boundaries 1 and 16.

Wetted Wall 2

1 In the Physics toolbar, click ⊖ Boundaries and choose Wetted Wall.

2 Select Boundary 2 only.

3 In the Settings window for Wetted Wall, locate the Wetted Wall section.

4 In the θ_w text field, type 74[deg].

The default surface tension for the phase field model is evenly distributed across the fluid–fluid interface. In cases with a large difference in density between the two phases, significant spurious oscillations in the velocity field can occur for the lighter phase which results in smaller time steps and longer computing time. Thus, it may be advantageous to shift the surface tension toward the heavy phase to avoid such oscillations. This is done by multiplying the surface tension force by

$$f_s = \frac{2}{(\rho_1 + \rho_2)}\left(\rho_1 H\left(\frac{V_{f,1} - 0.5}{d_{s,Fst}}\right) + \rho_2 H\left(\frac{V_{f,2} - 0.5}{d_{s,Fst}}\right)\right)$$

where $d_{s,Fst}$ is a mixing parameter defining the size of the transition zone.

To change averaging methods, you must first activate Advanced Physics Options.

1 Click the ☞ Show More Options button in the Model Builder toolbar.

2 In the Show More Options dialog box, in the tree, select the check box for the node Physics>Advanced Physics Options.

3 Click OK.

4 In the Model Builder window, under Component 1 (comp1)>Multiphysics click Two-Phase Flow, Phase Field 1 (tpf1).

5 Select the Shift surface tension force to the heaviest phase check box.

Study 1

If you want to inspect the progress of the fluids during the simulation, you can enable the plot while solving option in the Step 2: Time Dependent node. By calculating the initial values first, the solver sequence and default plots will be generated. In the following section, you generate the default plot groups and use one of them for plotting the volume fraction while solving. Note that the plot while solving feature, in general, will affect the computation time slightly since the plot needs to be updated in each time step.

1 In the Study toolbar, click $\underset{t=0}{\cup}$ Get Initial Value.

2 In the Model Builder window, under Study 1 click Step 2: Time Dependent.

3 In the Settings window for Time Dependent, click to expand the Results While Solving section.

4 Select the Plot check box.

5 From the Plot group list, choose Volume Fraction of Fluid 1 (pf).

6 From the Update at list, choose Time steps taken by solver.

7 Locate the Study Settings section. In the Output times text field, type `range(0,0.01,0.25)`.

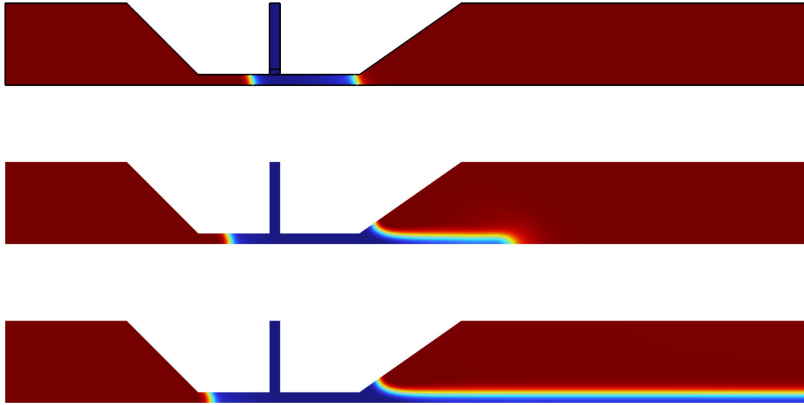**8** In the Study toolbar, click $=$ Compute.

Results

Examine the default plot.

Volume Fraction of Fluid 1 (pf)

1 In the Model Builder window, under Results click Volume Fraction of Fluid 1 (pf).

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

3 From the Time (s) list, choose 0.03.

4 You can step through time using the Plot Next → buttons at the top of the plot group.

Coating fluid interface at $t = 0.03$ s, $t = 0.06$ s, and $t = 0.2$ s (top to bottom).

Proceed to reproduce the plot of the film thickness and the upstream meniscus position.

Film Thickness and Upstream Meniscus Position

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

2 In the Settings window for 1D Plot Group, type Film Thickness and Upstream Meniscus Position in the Label text field.

3 Locate the Legend section. From the Position list, choose Upper left.

Global 1

1 Right-click Film Thickness and Upstream Meniscus Position and choose Global.

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

3 In the table, enter the following settings:

Expression	Unit	Description
intop1(pf.Vf2)	mm	Film Thickness
intop2(pf.Vf2)/intop2(1)	1	Uptream meniscus relative position

4 In the Film Thickness and Upstream Meniscus Position toolbar, click Plot. The plot in the Graphics window should look like that in Figure 9.

Tutorial Model: Beads-on-String Structure of Viscoelastic Filaments

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.
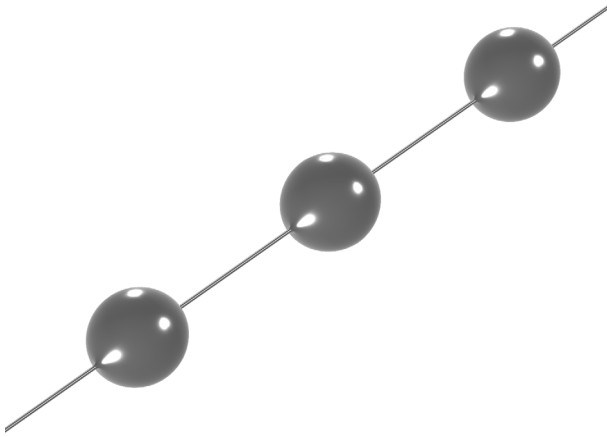
Model Geometry

This example studies the evolution of a long, initially unstretched axisymmetric filament of an Oldroyd-B fluid. The fluid filament is modeled as a liquid cylinder with a small perturbation of the initial radius of the cylinder, R_0 (Figure 10, $t = 0$). The initial 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.

Domain Equations and Boundary Conditions

The Oldroyd-B fluid used in this example is viscoelastic and you should therefore use the Viscoelastic Flow interface. An 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 and the evolution

equations for the elastic stress tensor components are formulated in the coordinates of the moving frame.

A Free Surface feature is applied at the interface between the polymer fluid and the air. This feature sets up the surface-tension force and specifies the normal velocity of the free surface. The outside air pressure is assumed to be constant, and the tangential stress on the free surface is neglected. A Periodic Flow Condition feature is used at the top and bottom boundaries to mimic the effect of an infinitely long filament.

The problem is made dimensionless by the initial radius of the cylinder, R_0; the surface tension coefficient, σ; the fluid density, ρ; and the total viscosity, μ_0, of the polymer. The dynamics of the filament thinning is governed by two dimensionless parameters: the Deborah number (the dimensionless relaxation time of the polymer solution) and the Ohnesorge number (the ratio between the inertia-capillary and viscous-capillary time scales). The relative importance of viscous stresses from the solvent is characterized by the solvent viscosity ratio, β.

Results

Figure 10 shows the evolution of the filament at different times for the following set of the dimensionless parameters: $\beta = 0.25$, Oh = 3.16, and De = 94.9.
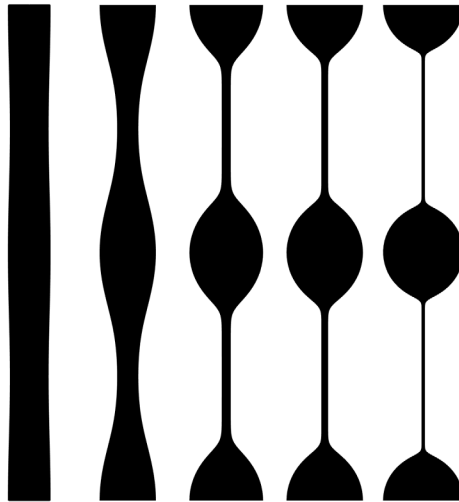

Figure 10: Filament profiles at 5 different dimensionless times: 0, 20, 30, 100, and 300.

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 an 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 the literature (Ref. 1).

Reference
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.

The following instructions show how to set up the model, solve it, and plot the results.

Model Wizard

Note: These instructions are for the user interface on Windows but also apply, with minor differences, to Linux and Mac.

1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new COMSOL Multiphysics model or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

If COMSOL Multiphysics is already open, you can start the Model Wizard by selecting New ▯ from the File menu and then clicking Model Wizard ◈.

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.

2 In the Select Space Dimension window, click the 2D Axisymmetric button ⬚.
3 In the Select Physics tree, select
Fluid Flow>Single-Phase Flow>Viscoelastic Flow (vef) ≋.
4 Click Add and then the Study button ➔ .
5 In the tree, under General Studies, click Time Dependent ⬚ .
6 Click the Done button ☑ .

Root

The equations are formulated in dimensionless form. Proceed to change the Unit System for the model.

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.

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 Enter the values of the perturbation magnitude and the dimensionless parameters:

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

Create the geometry by using a parametric curve and a polygon.

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.

6 Convert to Solid 1 (csol1).

7 In the Geometry toolbar, click ▦ Conversions and choose Convert to Solid.

8 Click in the Graphics window and then press Ctrl+A to select both objects.

9 In the Settings window for Convert to Solid, click ▦ Build All Objects.
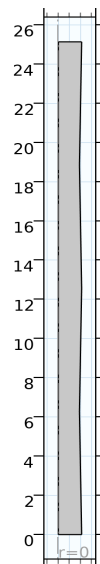
Definitions

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.
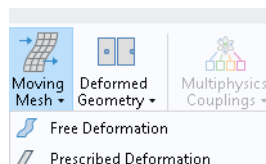
Viscoelastic Flow (vef)

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.

Note: Several default nodes are added automatically to the model tree. The 'D' in the upper-left corner of the node indicates that these are default nodes.

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 r list, choose User defined. In the associated text field, type 1.

4 For the Viscoelastic interface, Viscoelastic, Newtonian, and Inelastic non-Newtonian constitutive relations are available. By default, the viscoelastic Oldroyd-B model is selected.

5 Choose User defined from the ms list. In the associated text field, type mus.

6 In the table, enter values for elastic viscosity and relaxation time:

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 s 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.

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

Definitions

Symmetry/Roller 1

1 In the Moving Mesh toolbar choose Symmetry/Roller.

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

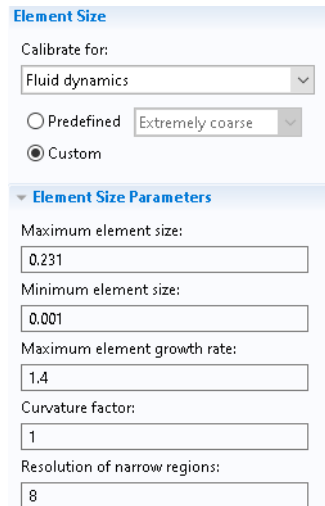**3** Select Boundaries 1, 2, and 3 on (top, bottom, and symmetry boundaries).

Symmetry/Roller

Label: Symmetry/Roller 1

Boundary Selection

Selection: Manual

```
1
2
3
```

Mesh 1
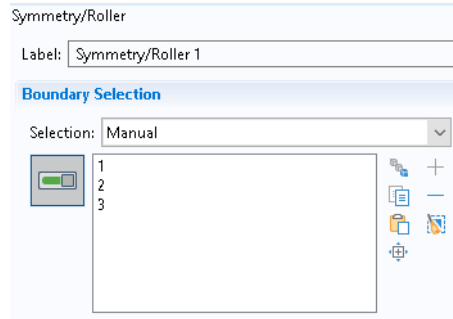
Free Triangular 1

In the Mesh toolbar, click Free Triangular.

Size

4 In the Model Builder window, click Size.

5 In the Settings window for Size, locate the Element Size section.

6 From the Calibrate for list, choose Fluid dynamics.

7 From the Predefined list, choose Extremely coarse.

8 Click to expand the Element Size Parameters section. In the Minimum element size text field, type 0.001.

9 In the Resolution of narrow regions text field, type 8.

10 Click Build All.

Element Size

Calibrate for:

Fluid dynamics

○ Predefined   Extremely coarse
◉ Custom

▾ **Element Size Parameters**

Maximum element size:

0.231

Minimum element size:

0.001

Maximum element growth rate:

1.4

Curvature factor:

1

Resolution of narrow regions:

8

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. In the Output times text field, type range(0,5,300).

3 From the Tolerance list, choose User controlled.

4 In the Relative tolerance text field, type 0.005.

5 Click to expand the Study Extensions section. Select the Automatic remeshing check box.
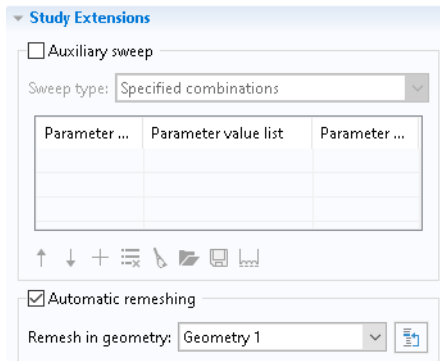
Before solving the problem, prepare a plot of the velocity. The plot will be shown and updated during the computations. In the Study toolbar, click $\frac{u}{t=0}$ Get Initial Value. Default plots for the velocity and pressure are generated automatically.

Results

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.

Getting the initial values for the step also creates the solver suggestions based on the existing physics.

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 Reduce the default value. Type `1.05` in the Stop when distortion exceeds field.

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

After the computation, the velocity plot for the final time will be shown
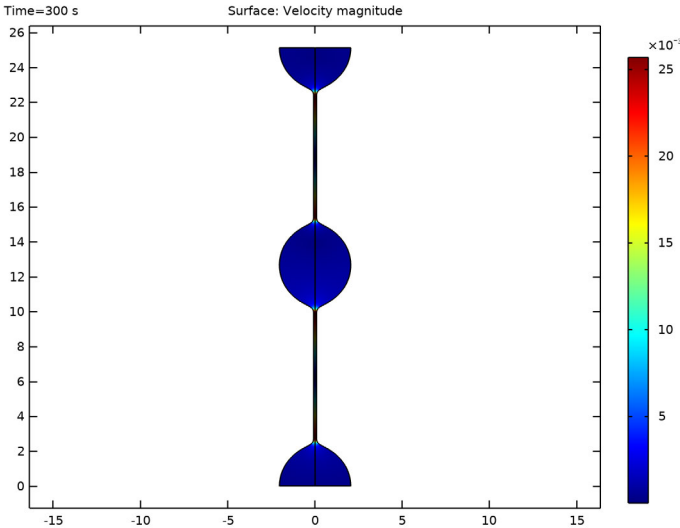
Continue with a visualization of the filament shape at selected times (Figure 10).

Shape

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

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. The plot shows the shape at the last time step, t=300.

6 In the Settings window for Surface, locate the Data section.

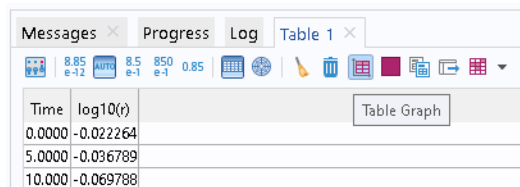**7** From the Time (s) list, choose 0, 20, 30, 100 to plot the shape evolution at different times.

Now, plot the minimum filament radius.

Line Minimum 1

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

2 Select the Boundary 4 only.

3 In the Settings window for Line Minimum, locate the Expressions section, enter log10(r).

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

5 Click = Evaluate.

Table

The result appears in the Table window at the bottom of the COMSOL Desktop. Click Table Graph in the window toolbar.

Minimum Radius

1 In the Model Builder window, under Results click 1D Plot Group 5.

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 ⊡ .

The plot shows the minimum filament radius as a function of time. After a rapid formation of the beads-on-string structure, the figure shows the slow thinning of the threads. 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, an asymptotic solution to the problem can be derived. Plot the asymptotic solution using the following steps.
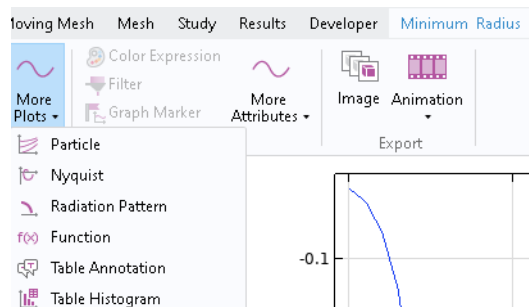
Grid 1D 1

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; in the Maximum text field, type 300.
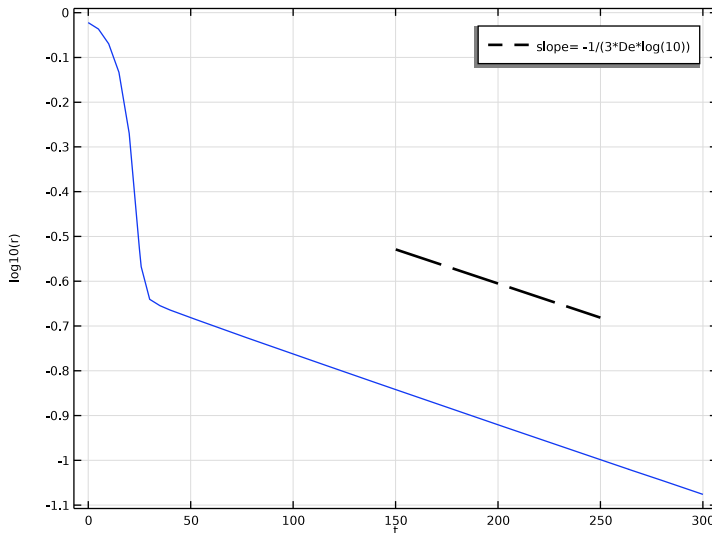
Minimum Radius

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, enter the asymptotic expression -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 In the Width text field, type 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 Type `slope=-1/(3*De*log(10))` in the table.

14 In the Minimum Radius toolbar, click Plot ⊡ .

The figure shows that the evolution of the minimum radius agrees well with the asymptotic prediction for large times.