Model created in COMSOL Multiphysics 6.4

Rubber Injection Molding

Introduction

Injection molding is a popular way of producing devices and equipment. A polymer fluid is injected into a mold, where it eventually will set to form the desired shape.

This tutorial demonstrates how to model the fluid flow in a rubber injection molding process of an automotive vibration damper using the **Laminar Two-Phase Flow, Phase Field** interface and an inelastic non-Newtonian power-law model for the rubber. The model geometry and material parameters are inspired from Ref. 1.

Model Definition

MODEL GEOMETRY

The geometry of the vibration damper is an imported CAD file where only the fluid domains are present. A quarter symmetry is used to reduce the computation time. The geometry can be seen in Figure 1.

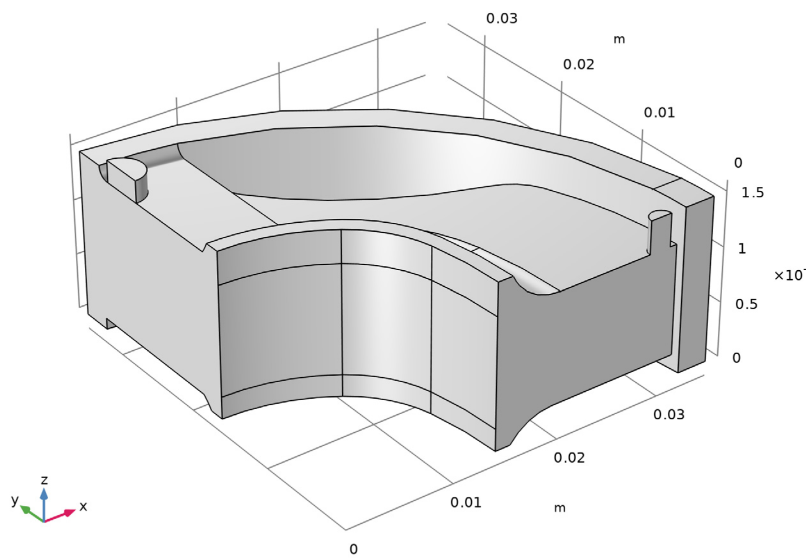


Figure 1: Model geometry. A quarter piece of the fluid volume of an automotive vibration damper. The fluid inlet can be seen at the top of the small cylinder to the left. While the outlets are to the right in the figure.

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** interface for the tracking of the interface between the air and the rubber fluid. The coupling of these two interfaces is handled by the **Two-Phase Flow, Phase Field** multiphysics interface. This model uses the **Multiphase Material** to define the effective material properties for a material composed of two phases: the air and the rubber. The rubber is a non-Newtonian power-law fluid. The non-Newtonian material properties are specified by using the **Power Law** material property group.

A small portion of the inlet channel is set up to contain rubber initially. This will automatically define the initial interface between rubber and air.

At the rubber inlet, the fluid velocity is increased smoothly from 0 at $t = 0$, and the outlets have a uniform pressure, $p = 0$. The corresponding inlet and outlet boundary conditions must also be set in the **Phase Field** interface together with the initial values for both fluids, so that the initial interface is correctly defined.

The symmetry planes are defined as **Symmetry** in both the Laminar flow interface and the Phase field interface. All other boundaries are solid walls described by a no-slip boundary condition.

Results and Discussion

Figure 2-Figure 5 show the propagation of the rubber front as time progresses from $t = 1$s, $t = 4$s, $t = 8$s and $t = 16$s.

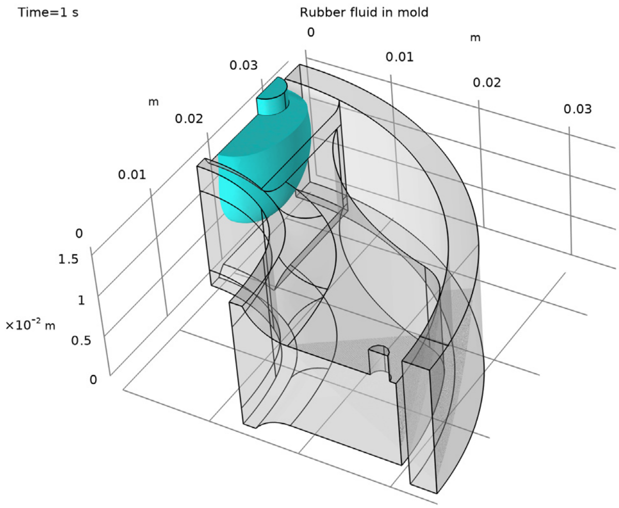


Figure 2: Rubber front after $t = 1$*s.*

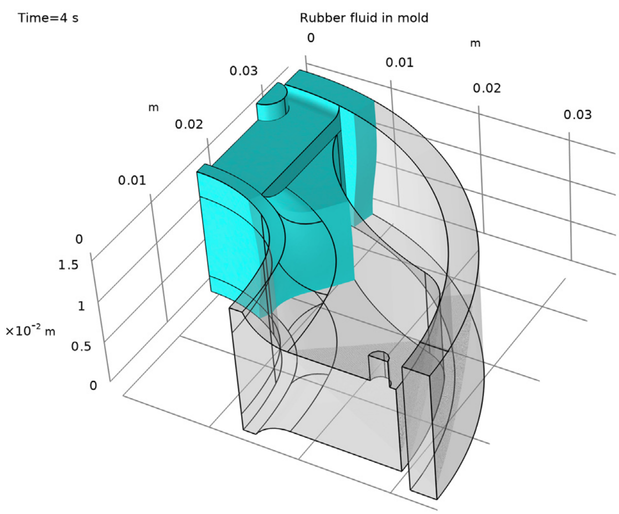


Figure 3: Rubber front after $t = 4$*s*

.

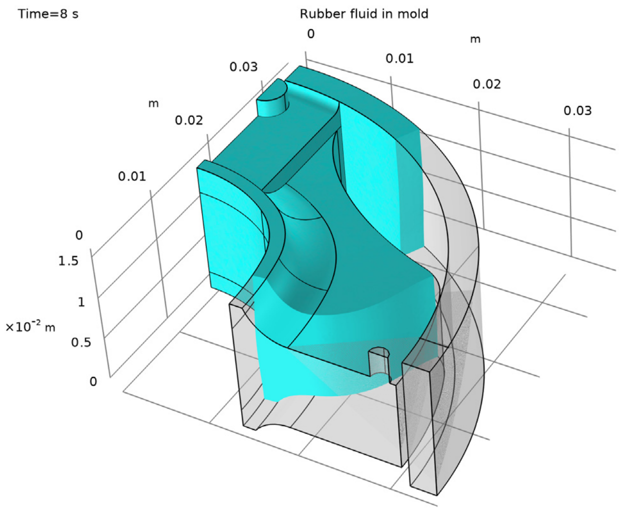


Figure 4: Rubber front after t = 8s.

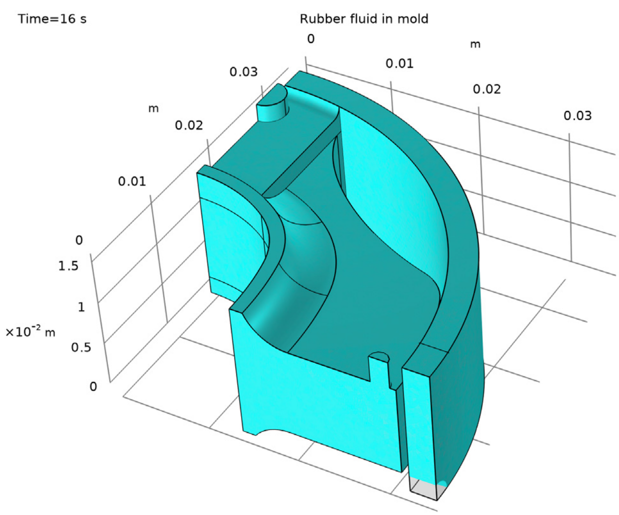


Figure 5: Rubber front after t = 16s.

The mold is almost completely filled after 16s. The small remaining void near the bottom corner might indicate that the mold design needs to be modified.

Notes About the COMSOL Implementation

The default method for averaging the fluid properties across the interface between the two phases is linear with respect to the volume fraction. When working with fluids that have a large difference in viscosities, switching to a different averaging method increases the performance. This example utilizes the Heaviside averaging method.

Reference

1. M. Erfanian, M. Anbarsooz, and M. Moghima, "A three dimensional simulation of a rubber curing process considering variable order of reaction," *Applied Mathematical Modeling*, vol. 40, pp. 8592–8604, 2016.

Application Library path: `Polymer_Flow_Module/Tutorials/rubber_injection_molding`

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 **3D**.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Time Dependent with Phase Initialization**.

6 Click **Done**.

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
U0	0.1[m/s]	0.1 m/s	Inlet velocity

GEOMETRY 1

Import 1 (imp1)

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

2 In the **Settings** window for **Import**, locate the **Source** section.

3 Click **Browse**.

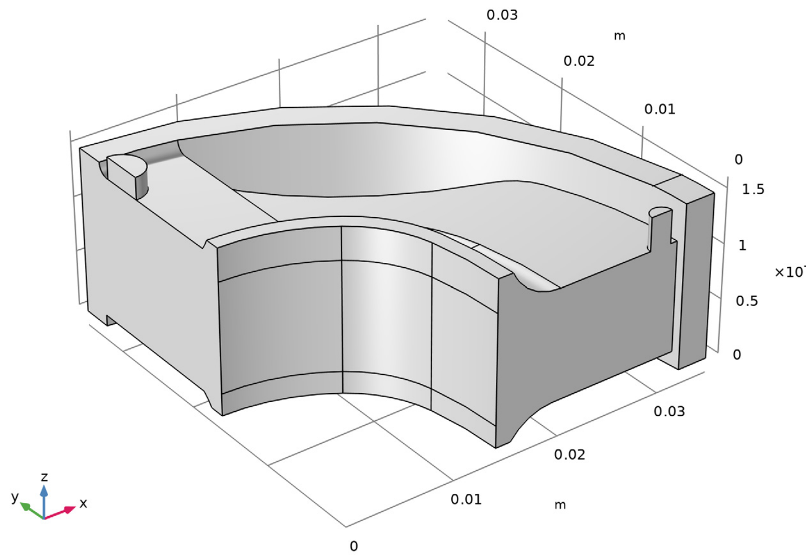
4 Browse to the model's Application Libraries folder and double-click the file `rubber_injection_molding.mphbin`.

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.



Explicit Selection 1 (sel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 14 only.

5 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog, type `Inlet` in the **Name** text field.

8 Click **OK**.

Explicit Selection 2 (sel2)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 41 and 43 only.

5 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog, type `Outlet` in the **Name** text field.

8 Click **OK**.

Explicit Selection 3 (sel3)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 1, 11, 35, and 44 only.

5 Locate the **Resulting Selection** section. From the **Show in physics** list, choose **Off**.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog, type `Symmetry` in the **Name** text field.

8 Click **OK**.

Disable the analysis of the geometry as the remaining small geometric details can be kept.

9 In the **Model Builder** window, click **Geometry 1**.

10 In the **Settings** window for **Geometry**, locate the **Cleanup** section.

11 Clear the **Automatic detection of small details** checkbox.

12 In the **Geometry** toolbar, click **Build All**.

Now, add the material properties for the two phases, air and rubber.

MULTIPHYSICS

Two-Phase Flow, Phase Field 1 (tpf1)

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**.

MATERIALS

Phase 1 (mpmat1.phase1)

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.

ADD MATERIAL TO PHASE 1 (MPMAT1.PHASE1)

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

2 In the tree, select **Built-in** > **Air**.

3 Click **Add Material**.

MATERIALS

Phase 2 (mpmat1.phase2)

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**, click to expand the **Material Properties** section.

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

4 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Fluid consistency coefficient	m_pow	5503	Pa·s	Power law
Flow behavior index	n_pow	0.2929	1	Power law
Density	rho	1200	kg/m³	Basic

The default method for averaging the viscosity across the interface between the two phases is linear with respect to the volume fraction. When working with fluids that have a large difference in viscosities, switching to a different averaging method increases the performance. In this model we use the Heaviside averaging method. The mixing parameter can be made smaller to sharpen the interface, but that will increase the computation time.

Multiphase Material 1 (mpmat1)

1 In the **Model Builder** window, click **Multiphase Material 1 (mpmat1)**.

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

3 Click to select row number 1 in the table.

4 Right-click and choose **Edit Mixing Rule**.

5 In the **Edit Mixing Rule** dialog, choose **Heaviside function** from the **Mixing rule** list.

6 In the l_{mix} text field, type `0.8`.

7 Click **Next Row (Store Changes)**.

8 From the **Mixing rule** list, choose **Heaviside function**.

9 Click **OK**.

MULTIPHYSICS

Two-Phase Flow, Phase Field 1 (tpf1)

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

2 In the **Show More Options** dialog, in the tree, select the checkbox 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 In the **Settings** window for **Two-Phase Flow, Phase Field**, locate the **Surface Tension** section.

6 Clear the **Include surface tension force in momentum equation** checkbox.

At the startup of a transient simulation of a filling process you can assume that the initial velocity in the domain is zero. To get consistent initial values, the initial velocity at the inlet should also be zero. Use a step function as described in the section below to ramp up the inlet velocity.

DEFINITIONS

Step 1 (step1)

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

2 Right-click **Definitions** and choose **Functions** > **Step**.

3 In the **Settings** window for **Step**, locate the **Parameters** section.

4 In the **Location** text field, type `0.05`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.095`.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
U_in	U0*(step1(t[1/s])-step1(t[1/s]-15))	m/s	Inlet velocity

LAMINAR FLOW (SPF)

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet**.
4. Locate the **Velocity** section. In the U_0 text field, type U_in.

Outlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Outlet**.

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Symmetry**.

   The initial interface between the rubber fluid and air is automatically assigned to the boundaries between the two initial value domains. Assign the initial values of fluid two to be in the small domain close to the inlet.

PHASE FIELD IN FLUIDS (PF)

Initial Values, Fluid 2

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Phase Field in Fluids (pf)** click **Initial Values, Fluid 2**.
2. Select Domain 2 only.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet**.

4 Locate the **Phase Field Condition** section. From the list, choose **Fluid 2 (φ = 1)**.

Outlet 1

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

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

3 From the **Selection** list, choose **Outlet**.

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry**.

When working with two-phase flows, the mesh should be fine enough to resolve the initial fluid interface. It is also advantageous to have a relatively uniform mesh element size in the domain. Modify the default mesh sequence to achieve a better mesh for this model.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.00125/2`.

Size 1

1 In the **Model Builder** window, click **Size 1**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** checkbox. In the associated text field, type `0.00125/2`.

6 Click **Build All**.

STUDY 1

Step 2: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,1,16)`.

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

RESULTS

Injected fluid

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **8**.

4 In the **Label** text field, type `Injected fluid`.

5 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

6 In the **Injected fluid** toolbar, click **Plot** to generate a dedicated view for this plot that you can rotate independently of the default view.

7 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

Volume 1

1 Right-click **Injected fluid** and choose **Volume**.

2 In the **Settings** window for **Volume**, 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 **Cyan**.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Rubber fluid in mold`.

Filter 1

1 Right-click **Volume 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `pf.Vf2>0.5`.

Injected fluid

1 In the **Model Builder** window, under **Results** click **Injected fluid**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

Surface 1

1. Right-click **Injected fluid** and choose **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 **Gray**.
6. Click to expand the **Title** section. From the **Title type** list, choose **None**.

Transparency 1

1. Right-click **Surface 1** and choose **Transparency**.
2. In the **Settings** window for **Transparency**, locate the **Transparency** section.
3. Find the **Transparency** subsection. Set the **Transparency** value to **0.75**.

Injected fluid

1. In the **Model Builder** window, under **Results** click **Injected fluid**.
2. In the **Injected fluid** toolbar, click **Plot**.