Model created in COMSOL Multiphysics 6.4

Flow in an Internal Mixer

Introduction

When mixing highly viscous non-Newtonian polymers and rubbers, it is important to keep track of both the temperature and the torque that is applied to the rotors in the vessel. The process must be tuned so that the temperature during mixing is below the disassociation temperature, but high enough so that mixing can occur. In this example, a HDPE polymer is stirred in an internal mixer with nonisothermal flow and a non-Newtonian Carreau model for the apparent viscosity of the fluid. Thermal effects are included in the form of an exponential law.

Model Definition

The geometry is a chamber with two counter-rotating impellers as shown in Figure 1.

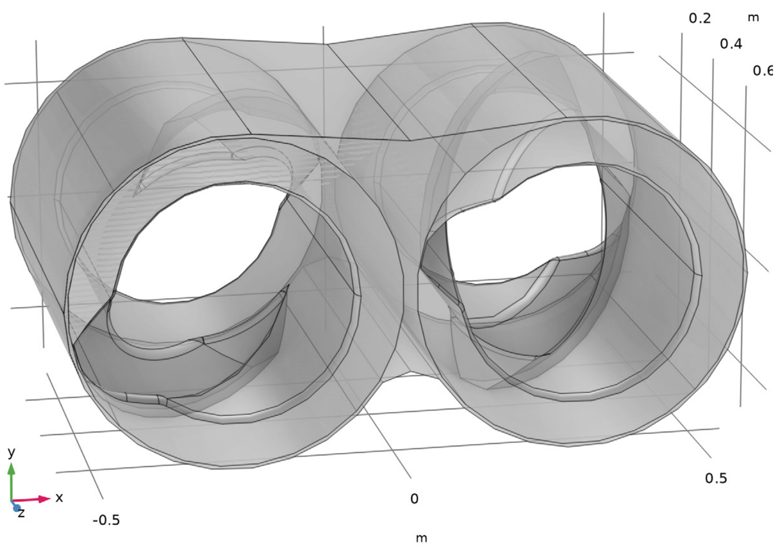

Figure 1: Mixer with two counter-rotating impellers.

The viscous fluid is a HDPE (high density polyethylene) with a non-Newtonian Carreau model for the apparent viscosity. The equation for the shear, and temperature, dependent viscosity is

$$\mu = \mu_\infty + (\alpha_T\mu_0 - \mu_\infty)[1 + (\lambda\dot{\gamma})^2]^{\frac{n-1}{2}} \qquad (1)$$

where $\alpha_T = e^{-b(T - T_0)}$ is the thermal contribution to the viscosity defined by thermal coefficients b and T_0. Moreover μ_∞, μ_0, λ, and n are model parameters while $\dot{\gamma}$ is the strain rate in the fluid. The outside of the cavity has a convective heat flux condition, while the other boundaries are thermally insulated. Viscous heating is a major contribution for such viscous fluids and is included as a source in the heat transfer equation as $Q_{\text{vd}} = \tau : \nabla\mathbf{u}$, where τ is the viscous tensor and $\mathbf{u}$ is the velocity vector.

The model is solved in two different studies. The first part uses a frozen-rotor approach where the rotation of the impellers is modeled in a stationary manner. This gives a stationary solution for the temperature, velocity and pressure. In order to analyze the start up and mixing behavior, a time dependent study is included. Transient results for torque on the rotors and the temperature at specific locations in the mixing cavity are presented.

Results and Discussion

The velocity together with velocity arrows obtained from the Frozen Rotor study can be seen in Figure 2.

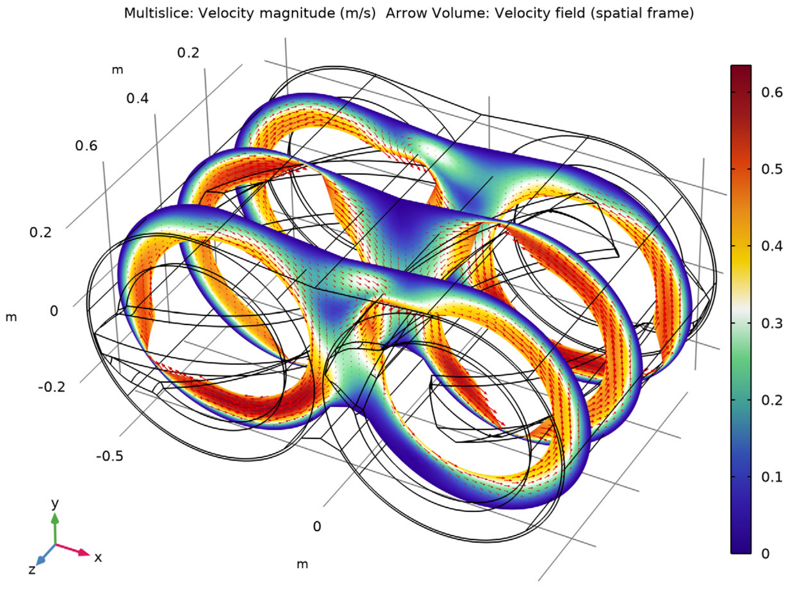

Figure 2: Velocity and velocity arrows for the Frozen Rotor study.

Figure 3 shows the pressure.

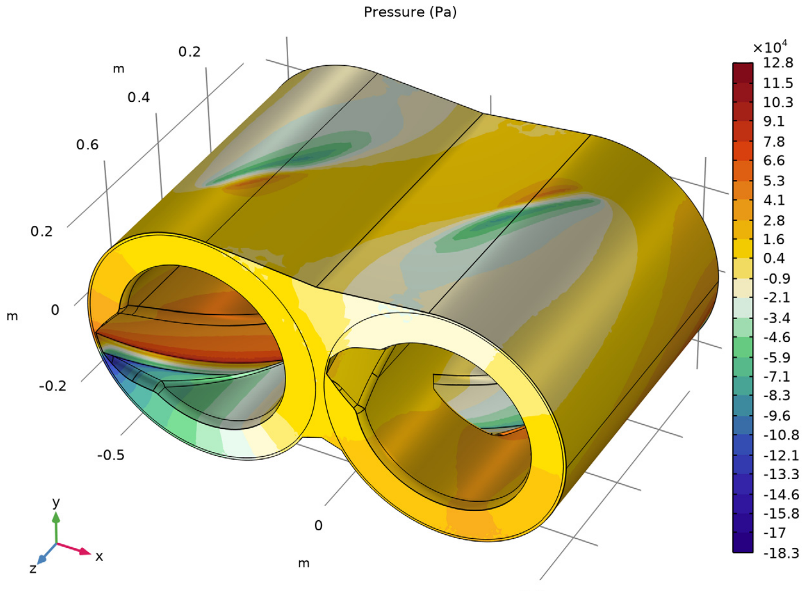

Figure 3: Pressure distribution for the Frozen Rotor study.

Figure 4 shows the shear rate.

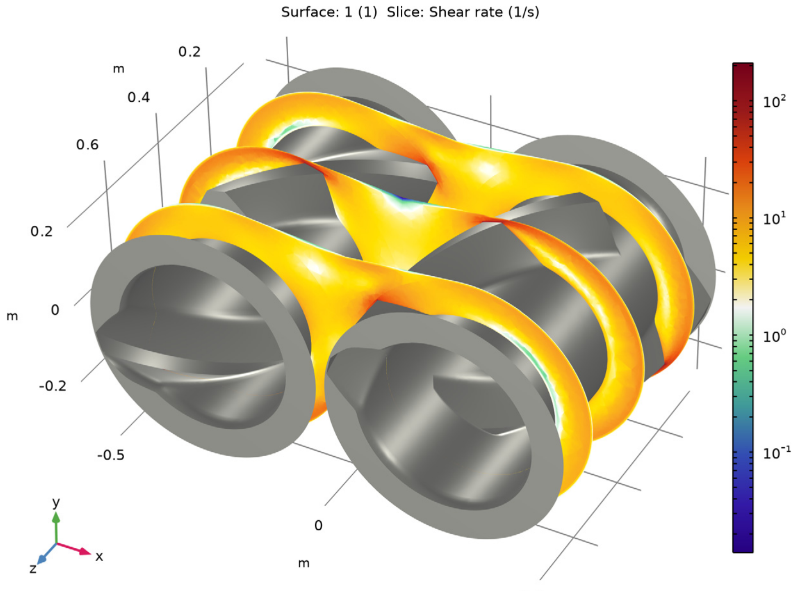

Figure 4: Shear rate for the Frozen Rotor study.

Figure 5 shows the viscosity and Figure 6 shows the temperature for the Frozen Rotor study.

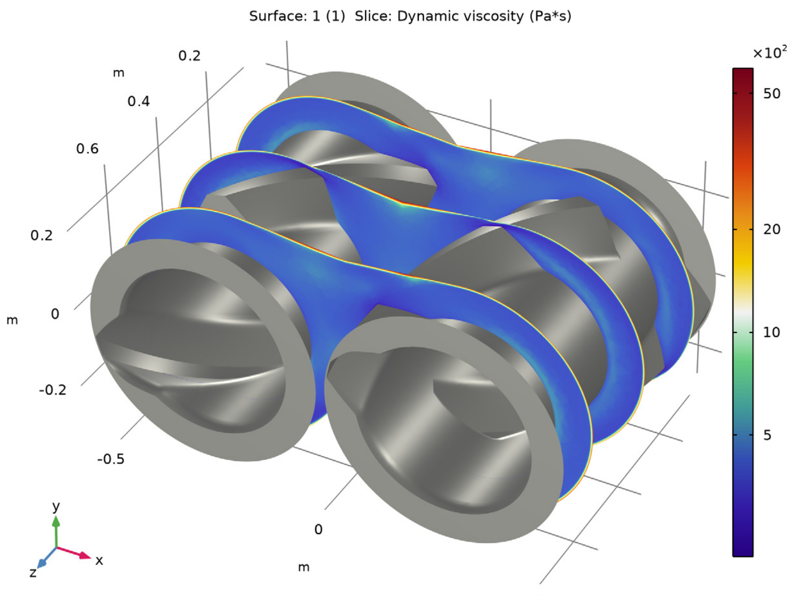

Figure 5: Dynamic viscosity for the Frozen Rotor study.

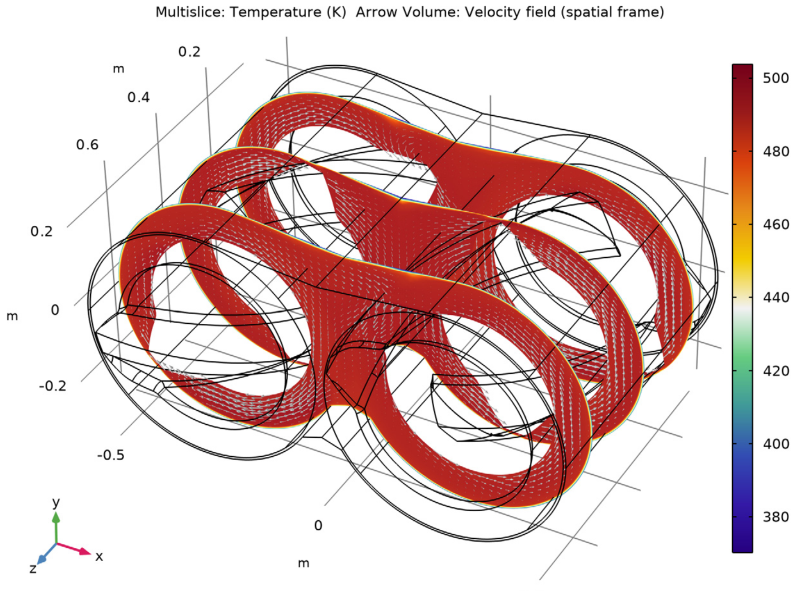

Figure 6: Temperature and velocity arrows for the Frozen Rotor study.

For the time-dependent study, it is interesting to investigate the torque from the rotors onto the fluid (which is the reciprocal of the torque on the rotors). The rotors have a forced rotation velocity which increases slowly from 0 to 20 RPM during the first second. The torque as a function of time from the two rotors is shown in Figure 7. Moreover, the temperature after two seconds is shown in Figure 8 and the temperature as a function of time at a couple of points located at the center of the cavity is shown in Figure 9. Note that a stationary result is not achieved for the transient simulations as we have focused on investigating only the first couple of revolutions.

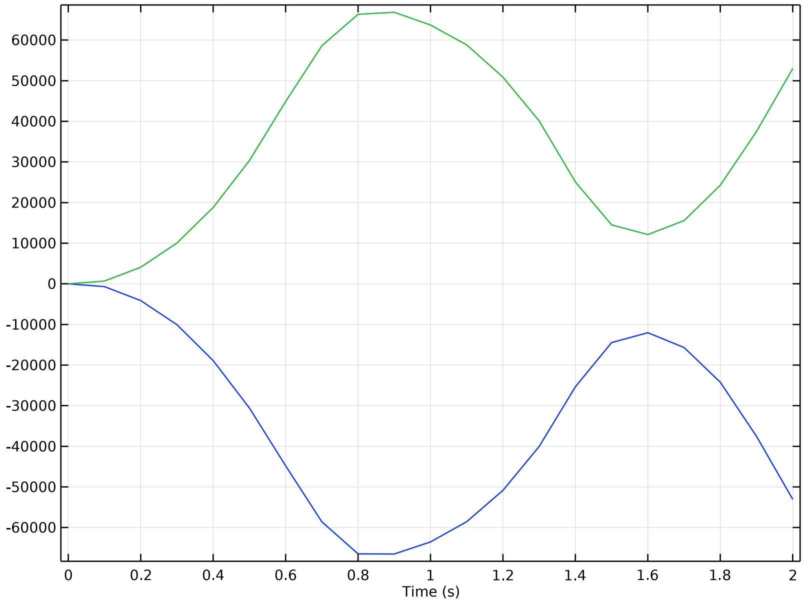

Figure 7: Impeller torque as a function of time during startup.

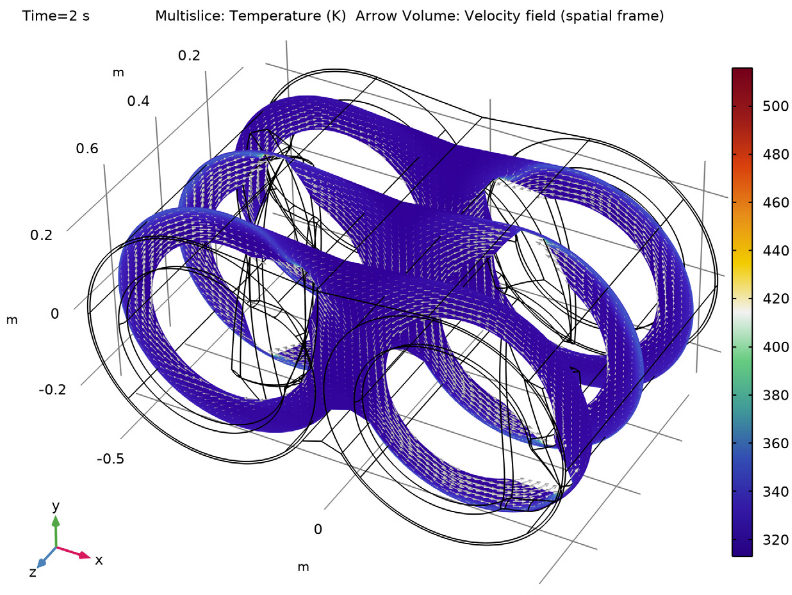

Figure 8: Temperature after two seconds.

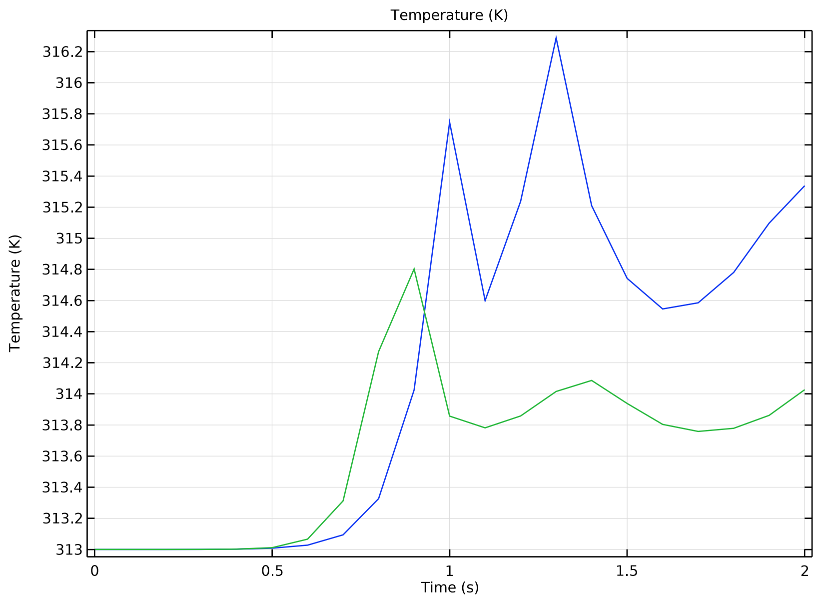

Figure 9: Temperature at a couple of points in the middle of the cavity as a function of time.

Application Library path: `Polymer_Flow_Module/Tutorials/internal_mixer`

Notes About the COMSOL Implementation

The geometry is made as an assembly to make it easier to go from Frozen Rotor to a Time Dependent study. Note that to run the frozen rotor simulation, the rotational velocity must be set to 20 RPM.

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 > Nonisothermal Flow > Rotating Machinery, Nonisothermal Flow > Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Frozen Rotor**.
6 Click **Done**.

GEOMETRY 1

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

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Cleanup** section.
3 Clear the **Automatic detection of small details** checkbox.

Import the geometry from a file.

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 `internal_mixer.mphbin`.
5 Click **Build All Objects**.

Add a centroid measurement node to find the rotation axis of the two rotors. The geometry parameters are later used in the definition of the rotating domains.

Centroid Measurement 1 (cm1)

1 In the **Geometry** toolbar, click **Measurements** and choose **Centroid Measurement**.
2 On the object **imp1(3)**, select Points 4, 6, 10, and 12 only.
3 In the **Settings** window for **Centroid Measurement**, click **Build Selected**.

Centroid Measurement 2 (cm2)

1 Right-click **Centroid Measurement 1 (cm1)** and choose **Duplicate**.
2 In the **Settings** window for **Centroid Measurement**, locate the **Vertex Selection** section.

3 Click the **Clear Selection** button for **Vertices**.

4 On the object **imp1(3)**, select Points 22, 24, 28, and 30 only.

5 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Click **Build Selected**.

Create a selection for the boundaries of the rotors that you can use later when postprocessing.

DEFINITIONS

Rotors

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

2 Right-click **Definitions** and choose **Selections** > **Explicit**.

3 In the **Settings** window for **Explicit**, type `Rotors` in the **Label** text field.

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select the **Group by continuous tangent** checkbox.

6 In the **Angular tolerance** text field, type `50`.

7 Select Boundaries 19–35, 39–59, 61–63, 65–86, 91–114, and 116–118 only.

MOVING MESH

Rotating Domain 1

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

2 Select Domain 2 only.

3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

4 In the f text field, type `-1/3`.

5 Locate the **Axis** section. Specify the $\mathbf{r}_{\mathrm{ax}}$ vector as

geom1.cm1.x	X

`geom1.cm1.y`	Y
`geom1.cm1.z`	Z

Rotating Domain 2

1 Right-click **Component 1 (comp1)** > **Moving Mesh** > **Rotating Domain 1** and choose **Duplicate**.

2 Select Domain 3 only.

3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

4 In the *f* text field, type `1/3`.

5 Locate the **Axis** section. Specify the $\mathbf{r}_{\mathrm{ax}}$ vector as

`geom1.cm2.x`	X
`geom1.cm2.y`	Y
`geom1.cm2.z`	Z

LAMINAR FLOW (SPF)

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Laminar Flow (spf)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 Find the **Constitutive relation** subsection. From the list, choose **Inelastic non-Newtonian**.

4 From the **Inelastic model** list, choose **Carreau**.

MATERIALS

HDPE

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `HDPE` in the **Label** text field.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Zero shear rate viscosity	mu0	1130	Pa·s	Carreau model
Infinite shear rate viscosity	mu_inf	50	Pa·s	Carreau model

Property	Variable	Value	Unit	Property group
Relaxation time	lam_car	0.95	s	Carreau model
Power index	n_car	0.85	1	Carreau model
Heat capacity at constant pressure	Cp	250	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.2	W/(m·K)	Basic
Density	rho	760	kg/m³	Basic

LAMINAR FLOW (SPF)

Fluid Properties 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Laminar Flow (spf)** click **Fluid Properties 1**.
2. In the **Settings** window for **Fluid Properties**, click to expand the **Thermal Effects** section.
3. From the **Thermal function** list, choose **Exponential**.
4. In the T_0 text field, type 453.
5. In the b text field, type 0.02.

Pressure Point Constraint 1

1. In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2. Select Point 17 only.

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)** click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the T text field, type 313.

Heat Flux 1

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

   Add a convective heat flux to all boundaries.
2. Select Boundary 8 only.

3 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

6 In the h text field, type 40.

7 In the T_{ext} text field, type 313.

Now apply "Thermal insulation" boundary condition to the boundaries of the rotors. This overrides the previous boundary condition on these boundaries.

Thermal Insulation 2

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

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

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

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, select the **Use** checkbox for **Geometric Analysis, Detail Size**.

4 From the **Element size** list, choose **Fine**.

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

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type 5.

4 In the **Stretching factor** text field, type 1.15.

5 In the **Thickness adjustment factor** text field, type 1.5.

6 Click **Build All**.

7 Click the **Mesh Rendering** button in the **Graphics** toolbar.

STUDY 1

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

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **z-planes** subsection. In the **Planes** text field, type `3`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Tectocoris**.

7 In the **Velocity (spf)** toolbar, click **Plot**.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `60`.

4 Find the **y grid points** subsection. In the **Points** text field, type `40`.

5 Find the **z grid points** subsection. In the **Points** text field, type `3`.

6 In the **Velocity (spf)** toolbar, click **Plot**.

Viscosity

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

2 In the **Settings** window for **3D Plot Group**, type `Viscosity` in the **Label** text field.

Surface 1

1 Right-click **Viscosity** and choose **Surface**.

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

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

Material Appearance 1

1 Right-click **Surface 1** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

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

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

Slice 1

1 In the **Model Builder** window, right-click **Viscosity** and choose **Slice**.

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

3 In the **Expression** text field, type `spf.mu`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **Planes** text field, type `3`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Tectocoris**.

7 From the **Scale** list, choose **Logarithmic**.

Viscosity

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

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

3 Clear the **Plot dataset edges** checkbox.

4 In the **Viscosity** toolbar, click **Plot**.

Shear Rate

1 Right-click **Viscosity** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Shear Rate` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Shear Rate** node, then click **Slice 1**.

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

3 In the **Expression** text field, type `spf.sr`.

4 In the **Shear Rate** toolbar, click **Plot**.

Temperature

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

2 In the **Settings** window for **3D Plot Group**, type `Temperature` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Temperature** node, then click **Multislice 1**.

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

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

Arrow Volume 1

1 In the **Model Builder** window, click **Arrow Volume 1**.

2 In the **Settings** window for **Arrow Volume**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Gray**.

4 In the **Temperature** toolbar, click **Plot**.

Define a step function so that you can specify a smooth startup of the rotation velocity for the rotors.

DEFINITIONS

Step 1 (step1)

1 In the **Definitions** toolbar, click f(X) **More Functions** and choose **Step**.

2 In the **Settings** window for **Step**, click to expand the **Smoothing** section.

3 In the **Size of transition zone** text field, type `0.5`.

4 Click **Plot**.

5 Locate the **Parameters** section. In the **Location** text field, type `.5`.

6 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `1`.

7 Click **Plot**.

MOVING MESH

Rotating Domain 1

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

2 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

3 In the *f* text field, type `-1/3*step1(t)`.

Rotating Domain 2

1 In the **Model Builder** window, click **Rotating Domain 2**.

2 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

3 In the *f* text field, type `1/3*step1(t)`.

ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,0.1,2)`.

LAMINAR FLOW (SPF)

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

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

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

5 In the **Show More Options** dialog, select **Physics** > **Stabilization** in the tree.

6 In the tree, select the checkbox for the node **Physics** > **Stabilization**.

7 Click **OK**.

8 In the **Settings** window for **Laminar Flow**, click to expand the **Consistent Stabilization** section.

9 Find the **Navier–Stokes equations** subsection. Select the **Limit small time steps effect on stabilization time scale** checkbox.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

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

3 Clear the **Generate default plots** checkbox to disable the generation of default plots for this study. Instead, the plots from the first study will be copied and adapted for this study.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Time-Dependent Solver 1** node.

4 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Dependent Variables 1** node, then click **Pressure (comp1.p)**.

5 Drag and drop below **Pressure (comp1.p)**.

6 In the **Settings** window for **Field**, locate the **Residual Scaling** section.

7 From the **Method** list, choose **Manual**.

8 In the **Scale** text field, type `1e5`.

9 In the **Model Builder** window, under **Study 2 > Solver Configurations > Solution 2 (sol2) > Dependent Variables 1** click **Temperature (comp1.T)**.

10 In the **Settings** window for **Field**, locate the **Scaling** section.

11 From the **Method** list, choose **Manual**.

12 In the **Scale** text field, type `350`.

13 In the **Model Builder** window, under **Study 2 > Solver Configurations > Solution 2 (sol2) > Dependent Variables 1** click **Velocity Field (Spatial Frame) (comp1.u)**.

14 In the **Settings** window for **Field**, locate the **Scaling** section.

15 From the **Method** list, choose **Manual**.

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

RESULTS

Velocity (spf) 1

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

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Velocity (spf) 1** toolbar, click **Plot**.

Also duplicate the **Temperature** and **Shear Rate** plot.

Shear Rate, Temperature

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Shear Rate** and **Temperature**.

2 Right-click and choose **Duplicate**.

Temperature 1

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

2 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

3 In the **Temperature 1** toolbar, click **Plot**.

Shear Rate 1

1 In the **Model Builder** window, click **Shear Rate 1**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Shear Rate 1** toolbar, click **Plot**.

Set up some selections and integration operators to evaluate the torque exerted on the rotors.

DEFINITIONS

Rotor 1

1 In the **Definitions** toolbar, click **Explicit**.

By using the continuous tangent and a large angle tolerance you will be able to select all the boundaries on the rotor by just clicking at one surface.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select the **Group by continuous tangent** checkbox.

5 In the **Angular tolerance** text field, type `50`.

6 Select Boundaries 19–35, 39–59, 61–63, and 65–68 only.

7 In the **Label** text field, type `Rotor 1`.

Rotor 2

1 Right-click **Rotor 1** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Rotor 2` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Clear Selection**.

4 Select Boundaries 69–86, 91–114, and 116–118 only.

Integration 1 (intop1)

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 From the **Selection** list, choose **Rotor 1**.

Integration 2 (intop2)

1 Right-click **Integration 1 (intop1)** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Rotor 2**.

Add the expression for the torque on the two rotors by using the integration operator from the previous step.

Variables 1

1 In the **Model Builder** window, 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
T1	intop1(x*spf.T_stressy-y*spf.T_stressx)	N·m	
T2	intop2(x*spf.T_stressy-y*spf.T_stressx)	N·m	

Update the solution to incorporate the newly defined operators and expressions.

STUDY 2

In the **Study** toolbar, click **Update Solution**.

RESULTS

Rotor Torque

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

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Label** text field, type `Rotor Torque`.

Global 1

1 Right-click **Rotor Torque** 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
T1	N*m	
T2	N*m	

4 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

5 In the **Rotor Torque** toolbar, click **Plot**.

Add a couple of cut points in the middle of the cavity for monitoring temperature, pressure and shear rate as functions of time.

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Point Data** section. In the **x** text field, type `0`.

5 In the **y** text field, type `0`.

6 In the **z** text field, type `0.3 0.6`.

7 Click **Plot**.

Cavity Temperature

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

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

3 From the **Dataset** list, choose **Cut Point 3D 1**.

4 In the **Label** text field, type `Cavity Temperature`.

Point Graph 1

1 Right-click **Cavity Temperature** and choose **Point Graph**.

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

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

4 In the **Cavity Temperature** toolbar, click **Plot**.

Cavity Shear Rate

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

2 In the **Settings** window for **1D Plot Group**, type `Cavity Shear Rate` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

Point Graph 1

1 Right-click **Cavity Shear Rate** and choose **Point Graph**.

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

3 In the **Expression** text field, type `spf.sr`.

4 In the **Cavity Shear Rate** toolbar, click **Plot**.