Model created in COMSOL Multiphysics 6.4

Transient Elastohydrodynamic Squeeze-Film Interaction

Introduction

This benchmark model computes the transient pressure distribution and film height in a squeeze-film bearing for lubrication in a nonconformal conjunction of a solid sphere and an elastic wall separated by a lubricant film.

Lubrication between mechanical parts prevents wear and tear due to friction. Elastohydrodynamic contact between mechanical parts refers to the interaction between a lubricant and elastic bodies. The pressure developed in the lubricant and the mechanical stresses near the contact center are important concerns in elastohydrodynamic interaction. Solving such problems numerically involves modeling the elastohydrodynamic interaction by solving the Reynolds equation and solid mechanics as a coupled problem.

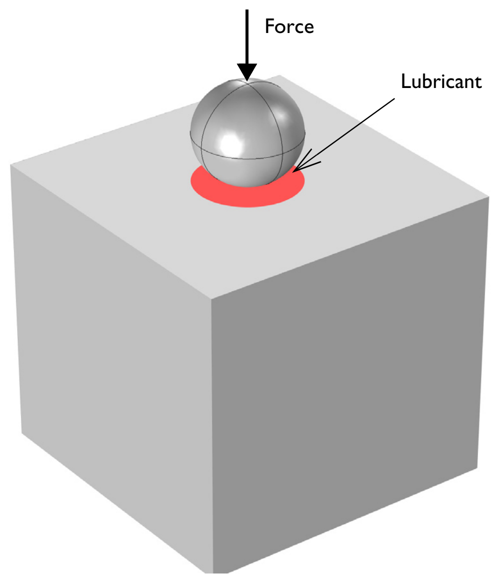


Figure 1: This example considers the case of an equivalent rigid sphere and an elastic wall. The equivalent model does not require modeling the rigid sphere. Because the model is symmetric, it is sufficient to model one quarter of the above geometry.

This example solves the benchmark case of hydrodynamic interaction between a solid sphere and a wall separated by a lubricant film, and extends the benchmark case to include elastic deformation and stresses on the contacting wall. The model setup involves a solid sphere being pushed by an external force toward a solid plane wall. The lubricant layer gets squeezed by the approaching ball, which leads to a rise in the pressure in the lubricant. The calculated maximum lubricant pressure and the change in film height with time are compared with analytical solutions.

Model Definition

Figure 1 shows the scenario of an elastic sphere pushed by an external force toward an elastic wall covered by a thin lubricant layer. This model computes the time-dependent pressure developed in the lubricant and the position of the sphere relative to the elastic wall. The scenario in Figure 1 is reduced to an equivalent model with a rigid sphere and an elastic wall with an equivalent Young's modulus given by (Ref. 1)

$$E = \frac{2}{\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2}}$$

where E_1 and E_2 are the Young's moduli and v_1 and v_2 are the Poisson's ratios of the two elastic bodies. Because of symmetry, the model uses only one quarter of the geometry shown in Figure 1.

For no-slip boundary conditions at the wall and the base, Reynolds equation takes the form

$$\frac{\partial}{\partial t}(\rho h) + \nabla_t \cdot (\rho h \mathbf{v}_{\mathrm{av}}) + \rho(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) = 0$$

$$\mathbf{v}_{\mathrm{av}} = \frac{1}{2}(\mathbf{I} - \mathbf{n}_r \mathbf{n}_r^T)(\mathbf{v}_w + \mathbf{v}_b) - \frac{h^2}{12\mu}\nabla_t p_f$$

where h is the film thickness, ρ is the fluid density, μ is the viscosity, and p_f — the dependent variable in the Thin-Film Flow user interface — is the pressure developed as a result of the flow. For further details, see the theory section for the Thin-Film Flow interfaces in the *CFD Module User's Guide*.

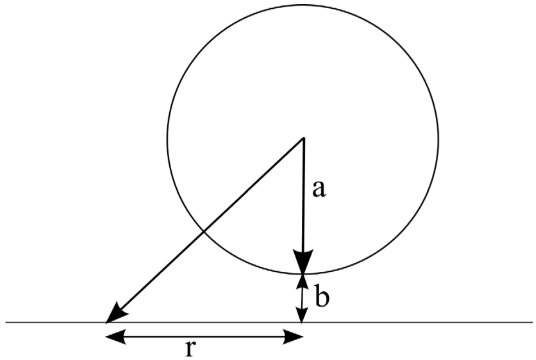


Figure 2: 2D representation of the distance of the sphere from the solid wall.

Figure 2 shows a 2D representation of the sphere at some distance from the solid wall with an exaggerated view of the film height between the sphere and the solid wall. The sphere has a radius a and the center of the sphere is initially located at a distance $a + b$ from the surface of the solid. For a thin-film approximation, the film thickness h is given by the expression

$$h = b(t) + \frac{r^2}{2a}$$

where $r = \sqrt{x^2 + y^2}$ is the horizontal radial distance measured from the center of the sphere.

The external force, $\boldsymbol{F}$, is counterbalanced by the pressure in the lubricant. This is imposed as a constraint:

$$\int_{\partial\Omega} p_f dS - \boldsymbol{F} = 0 \qquad (1)$$

The hydrodynamic pressure exerted by the lubricant causes elastic deformation of the two surfaces containing the lubricant. In this model, the surface of interest is the elastic wall. The hydrodynamic pressure is therefore used as a mechanical load on the elastic wall to calculate its deformation using a Solid Mechanics interface. However, the elastic deformation in this example is negligibly small in comparison with the change in film height due to the squeezing motion of the sphere against the elastic wall. Therefore, the results for pressure developed in the lubricant and the change in film height can be compared with the solution to the benchmark hydrodynamic problem of a solid sphere pushed against a wall with a lubricant layer between the sphere and the wall. Because the problem is axisymmetric, the Reynolds equation can be greatly simplified and written as

$$\frac{1}{r}\frac{\partial}{\partial r}\left(rh^3\frac{\partial p_f}{\partial r}\right) = 12\mu\frac{\partial h}{\partial t} \tag{2}$$

Restricting the lubrication calculations to the range $0 < r < a$, Equation 2 is solved with boundary conditions $\partial p_f/\partial r = 0$ at $r = 0$ and $p_f = 0$ at $r = a$ to give the pressure developed in the film as (see Ref. 2)

$$p_f(r) = -6\mu\frac{\partial b}{\partial t}\left(\frac{2a^3}{(2ab + r^2)^2} - \frac{2a^3}{(2ab + a^2)^2}\right) \tag{3}$$

Given this expression for the pressure distribution, the hydrodynamic force can be calculated using Equation 1 and is given by the following expression (see Ref. 2)

$$F = \frac{6\pi\mu\dot{b}a^2}{b}\left(1 - \frac{2ab}{(a^2 + 2ab)} - \frac{2a^3b}{(a^2 + 2ab)^2}\right) \tag{4}$$

Equation 4 is an ordinary differential equation that can be solved for the analytical change in the film height, b. The values of b thus obtained can then be substituted in Equation 3 to solve for the analytical pressure developed in the lubricant film.

Results and Discussion

Figure 3 shows that the pressure distribution in the lubricant is concentrated near the center of the wall with the maximum pressure due to the squeezing action developing at the center. Figure 4 shows the von Mises stress distribution on the elastic wall resulting from the fluid load due to increased pressure in the lubricant.

Figure 5 shows the results for the maximum film pressure and the change in film height with time, respectively, together with the corresponding analytical solutions obtained by solving Equation 3 and Equation 4. As expected, as the gap between the sphere and the wall decreases, the film pressure increases. The figures also show a very good match between the numerical and analytical solutions.

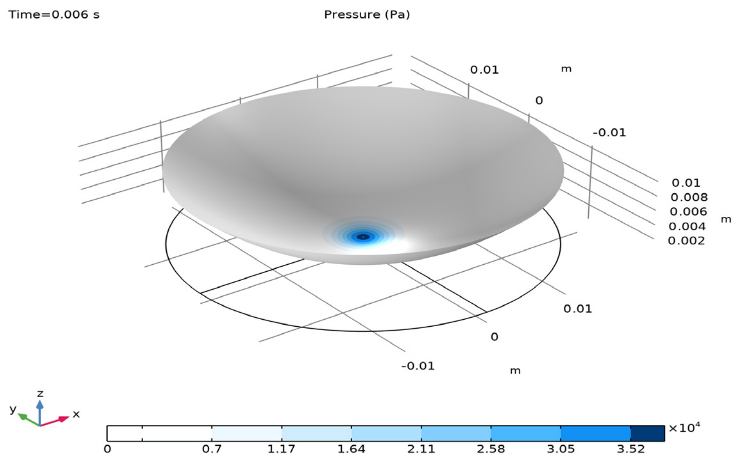


Figure 3: Pressure distribution in the lubricant. The height represents the total gap height

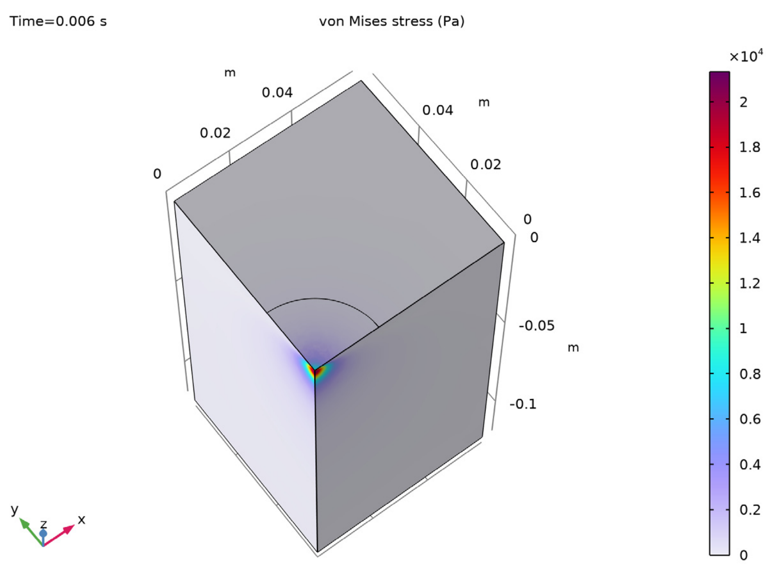


Figure 4: von Mises stress plot on the boundaries of the elastic solid.

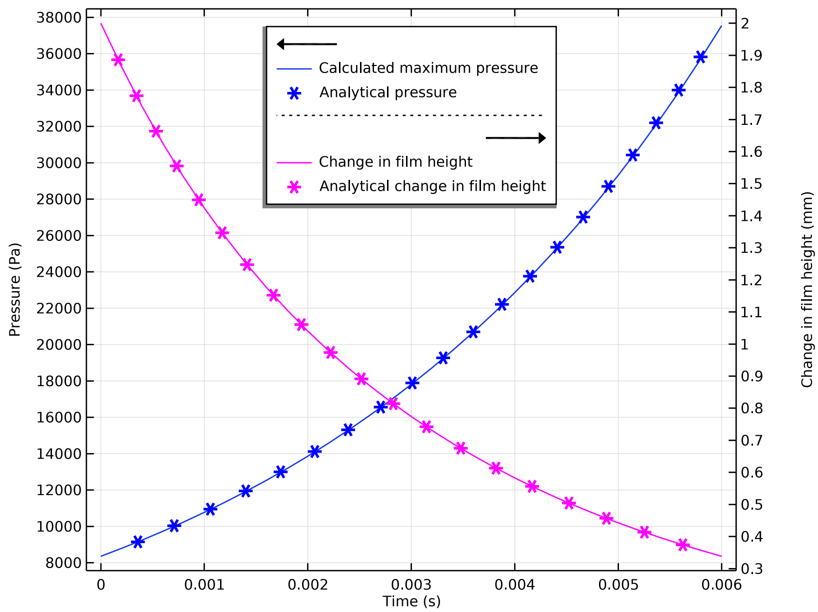


Figure 5: Comparison between calculated and analytical values of maximum pressure (blue) and change in film height (magenta).

Notes About the COMSOL Implementation

To resolve the high pressure gradients at the center of the wall, the mesh is customized to be fine in this region. This is important for getting results with higher accuracy. As the wall deforms, the film height changes by an additional amount equal to the wall displacement along the surface normal. This is accounted for in the settings of the Thin-Film Flow, Shell user interface by choosing the displacement field as an additional wall displacement. However, in this example this change in film height is negligibly small in comparison to the change in film height due to the external force.

References

1. A.Z. Szeri, *Fluid Film Lubrication: Theory and Design*, Cambridge University Press, 1998.

2. L.G. Leal, *Advanced Transport Phenomena: Fluid Mechanics and Convective Transport Processes,* Cambridge University Press, 2007.

Application Library path: `CFD_Module/Thin-Film_Flow/elastohydrodynamic_interaction`

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** > **Thin-Film Flow** > **Thin-Film Flow (tff)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

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

8 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
a	0.02[m]	0.02 m	Sphere radius
extent	a	0.02 m	Extent of lubricated area
Force	1.5[N]	1.5 N	Applied force
b0	a/10	0.002 m	Initial film height
visc_mat2	0.8[Pa*s]	0.8 Pa·s	Lubricant viscosity

Name	Expression	Value	Description
density_mat2	860[kg/m^3]	860 kg/m³	Lubricant density
timescale	6*pi*visc_mat2*a^2/Force	0.0040212 s	Time scale
nu_steel	0.28	0.28	Poisson's ratio
E_steel	205e9[Pa]	2.05E11 Pa	Young's modulus
dens_steel	7850[kg/m^3]	7850 kg/m³	Density
E_eqv	E_steel/(1-nu_steel^2)	2.2244E11 Pa	Equivalent Young's modulus

GEOMETRY 1

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `3*a`.

4 In the **Depth** text field, type `3*a`.

5 In the **Height** text field, type `6*a`.

6 Locate the **Position** section. In the **z** text field, type `-6*a`.

7 Click **Build All Objects**.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Circular Arc 1 (ca1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Circular Arc**.

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

3 From the **Specify** list, choose **Endpoints and radius**.

4 Locate the **Starting Point** section. In the **xw** text field, type `extent`.

5 Locate the **Endpoint** section. In the **yw** text field, type `extent`.

6 Locate the **Radius** section. In the **Radius** text field, type `extent`.

DEFINITIONS

Modify the view settings.

View 1

Use the mouse to rotate the image so that you can see the lubricant boundary.

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

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

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

4 Select the **Lock camera** checkbox.

Lubricant

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

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

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

4 Select Boundary 4 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 **Lubricant**.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`forcetot`	`4*intop1(pfilm)`	N	`Net force in lubricant`
`r`	`sqrt(x^2+y^2)`	m	`Radial distance`

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E_eqv	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu_steel	1	Young's modulus and Poisson's ratio
Density	rho	dens_steel	kg/m³	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

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

4 From the **Selection** list, choose **Lubricant**.

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

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	visc_mat2	Pa·s	Basic
Density	rho	density_mat2	kg/m³	Basic

THIN-FILM FLOW (TFF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thin-Film Flow (tff)**.

2 In the **Settings** window for **Thin-Film Flow**, locate the **Boundary Selection** section.

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

Fluid-Film Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Thin-Film Flow (tff)** click **Fluid-Film Properties 1**.

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

3 In the h_{w1} text field, type b+r^2/(2*a).

4 From the $\mathbf{u}_w$ list, choose **Displacement field (solid)**.

Symmetry 1

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

2 Select Edges 4 and 5 only.

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

4 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

5 Click **OK**.

Global Equations 1 (ODE1)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
b	Force-forcetot	b0	b0/ timescale	Change in film height

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog, type `length` in the text field.

6 In the tree, select **General** > **Length (m)**.

7 Click **OK**.

8 In the **Settings** window for **Global Equations**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog, type `force` in the text field.

11 In the tree, select **General** > **Force (N)**.

12 Click **OK**.

Global Equations 2 (ODE2)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
k	timescale*kt+k/(1-2*a*k/(extent^2+2*a*k)-extent^2*(2*a*k)/(extent^2+2*a*k)^2)	b0	b0/ timescale	Analytical change in film height

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog, type `length` in the text field.

6 In the tree, select **General** > **Length (m)**.

7 Click **OK**.

8 In the **Settings** window for **Global Equations**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog, type `length` in the text field.

11 In the tree, select **General** > **Length (m)**.

12 Click **OK**.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
analytical_p	-6*visc_mat2*kt*(2*a^3/(2*a*k+r^2)^2-2*a^3/(2*a*k+extent^2)^2)	Pa	Analytical pressure

SOLID MECHANICS (SOLID)

Boundary Load 1

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

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

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

4 Locate the **Force** section. From the **Load type** list, choose **Force per deformed area**.

5 From the $\mathbf{f}_a$ list, choose **Fluid load on wall (tff/ffp1)**.

Fixed Constraint 1

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

2 Select Boundary 3 only.

Symmetry 1

The symmetry boundary condition applied in the next step requires either the Structural Mechanics module or the MEMS module. An alternative is to use prescribed displacement boundary conditions to constrain displacements normal to the symmetry boundaries.

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Select Boundaries 1 and 2 only.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.
2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Lubricant**.

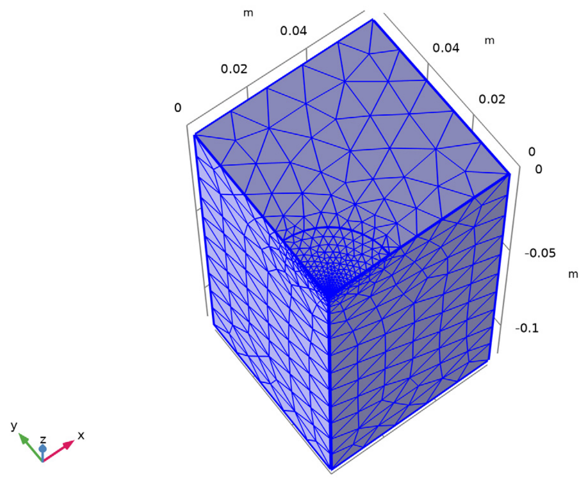
Size 1

1 Right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Point**.
4 Select Point 2 only.
5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
6 Click the **Custom** button.
7 Locate the **Element Size Parameters** section.
8 Select the **Maximum element size** checkbox. In the associated text field, type `1.92e-4`.
9 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.15`.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.



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**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog, type `2e-4` in the **Step** text field.

5 In the **Stop** text field, type `6e-3`.

6 Click **Replace**.

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

8 From the **Tolerance** list, choose **User controlled**.

9 In the **Relative tolerance** text field, type `0.0001`.

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `1e-5`.

The fully coupled solver performs better for this model and is therefore enabled in the next step.

6 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** and choose **Fully Coupled**.

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

RESULTS

Add millimeter as the preferred unit for length in results.

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **General** > **Length (m)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Length	m	mm

8 Click **Apply**.

Fluid Pressure (tff)

The first default plot group shows a surface plot of the fluid pressure for the final time step. To create the plot as shown in Figure 3 proceed as follows:

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

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

Sector 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 2D**.

2 In the **Settings** window for **Sector 2D**, locate the **Symmetry** section.

3 In the **Number of sectors** text field, type `4`.

Fluid Pressure, 2D

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Fluid Pressure, 2D` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Sector 2D 1**.
4 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.
5 Click to expand the **Number Format** section. Select the **Manual axis settings** checkbox.

Contour 1

1 Right-click **Fluid Pressure, 2D** and choose **Contour**.
2 In the **Settings** window for **Contour**, locate the **Levels** section.
3 In the **Total levels** text field, type `8`.
4 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.
5 From the **Color table** list, choose **JupiterAuroraBorealis**.
6 From the **Color table transformation** list, choose **Reverse**.

Height Expression 1

1 Right-click **Contour 1** and choose **Height Expression**.
2 In the **Settings** window for **Height Expression**, locate the **Expression** section.
3 From the **Height data** list, choose **Expression**.
4 In the **Expression** text field, type `tff.h`.
5 From the **Unit** list, choose **m**.
6 Locate the **Axis** section.
7 Select the **Scale factor** checkbox. In the associated text field, type `1`.
8 In the **Fluid Pressure, 2D** toolbar, click **Plot**.
9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Stress (solid)

The second default plot group shows a surface plot of the von Mises stress and a deformation plot (exaggerated) of the elastic wall displacement. To reproduce Figure 4 do as follows.

1 In the **Model Builder** window, expand the **Stress (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results** > **Stress (solid)** > **Volume 1** node.

2 Right-click **Deformation** and choose **Disable**.

To reproduce Figure 5 do as follows.

1D Plot Group 4

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

Point Graph 1

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

2 Select Point 2 only.

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

4 Select the **Description** checkbox. In the associated text field, type `Calculated maximum pressure`.

5 Click to expand the **Legends** section. Select the **Show legends** checkbox.

6 Find the **Include** subsection. Clear the **Point** checkbox.

7 Select the **Description** checkbox.

Point Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Point Graph**.

2 Select Point 2 only.

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

4 In the **Expression** text field, type `analytical_p`.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Blue**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

8 From the **Positioning** list, choose **Interpolated**.

9 In the **Number** text field, type `20`.

10 Locate the **Legends** section. Select the **Show legends** checkbox.

11 Find the **Include** subsection. Clear the **Point** checkbox.

12 Select the **Description** checkbox.

Global 1

1 Right-click **1D Plot Group 4** 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
`b`	`mm`	`Change in film height`

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Magenta**.

Global 2

1 Right-click **1D Plot Group 4** 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
`k`	`mm`	`Analytical change in film height`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

6 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

7 From the **Positioning** list, choose **Interpolated**.

8 In the **Number** text field, type `20`.

Maximum Pressure and Change in Film Height

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

2 In the **Settings** window for **1D Plot Group**, type `Maximum Pressure and Change in Film Height` in the **Label** text field.

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

4 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

5 In the table, select the **Plot on secondary y-axis** checkboxes for **Global 1** and **Global 2**.

6 Select the **y-axis label** checkbox. In the associated text field, type `Pressure (Pa)`.

7 Select the **Secondary y-axis label** checkbox.

8 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

9 In the **Maximum Pressure and Change in Film Height** toolbar, click **Plot**.