



Tilted Pad Thrust Bearing

Introduction

Tilted pad thrust bearings are used in a variety of rotating machineries that have to withstand high thrust loading. The thrust load is transferred from a sliding part to a stationary part through hydrodynamic oil films. The tilted pad thrust bearing consists of a series of flat surfaces sliding over stationary tilted pads. The motion of the sliding flat surface drags the lubricant in toward the diminishing gap between the tilted pad and the sliding surface. The pressure developed in the lubricant between the sliding surface and the tilted pad counteracts the external load applied to the sliding surface and thus prevents contact between the two surfaces. Reynolds equation is solved on the surface of the tilted pad to find the pressure distribution on the surface of the tilted pad. This load is used in the structural analysis of the tilted pad to find its elastic deformation.

Model Definition

The pressure in the lubricant (engine oil) is governed by Reynolds equation. For an incompressible fluid with no slip condition, Reynolds equation in the continuum range is given by

$$\nabla_t \cdot \left(\frac{-\rho h^3}{12\eta} \nabla_t p + \frac{\rho h}{2} (v_a + v_b) \right) - \rho ((\nabla_t b \cdot v_b) - (\nabla_t a \cdot v_a)) = 0$$

In the above equation, ρ is the density (SI unit: kg/m^3), h is the lubricant thickness, η is the viscosity (SI unit: $\text{Pa}\cdot\text{s}$), p is the pressure (SI unit: Pa), a is the location (SI unit: m) of the channel base, v_a is the tangential velocity (SI unit: m/s) of the channel base, b is the location (SI unit: m) of the solid wall, and v_b is the tangential velocity (SI unit: m/s) of the solid wall. Here the sliding surface is considered the base. Because the pressure is constant through the lubricant film thickness, COMSOL uses the tangential derivative operator ∇_t (that is, the projection of the gradient operator to the tangent plane) to calculate the pressure distribution on the lubricant surface. The Reynolds equation is solved on the surface of the tilted pad. The external load (W_{ext}) applied to the sliding surface is counterbalanced by the pressure in the lubricant. This is imposed as the constraint:

$$\int_{\partial\Omega} p dS - W_{\text{ext}} = 0 \quad (1)$$

The hydrodynamic pressure exerted by the lubricant causes elastic deformation of the two surfaces containing the lubricant. In this example, the surface of interest is the stationary tilted pad. The hydrodynamic pressure is used as a mechanical load on the tilted pad to

calculate the elastic deformation of the pad. This example also performs a parametric study on the degree of tilt.

Figure 1 shows the lubricant thickness distribution on the surface of the tilted pad. The arrows indicate the direction of rotation of a flat surface over the tilted pad.

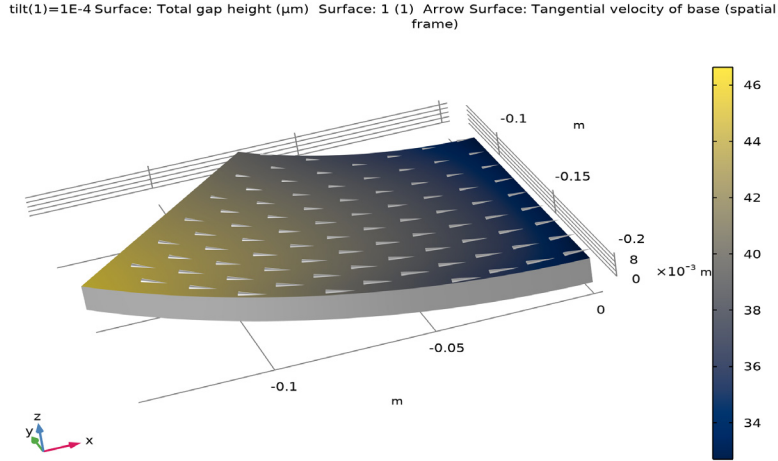


Figure 1: Lubricant thickness and the direction of rotation of the flat surface.

BOUNDARY CONDITIONS

Reynolds Equation

The edges of the top surface of the tilted pad are assigned a boundary condition of ambient pressure

$$p = 0$$

Structural Mechanics

The top face of the tilted pad has a face load

$$F_{\Gamma} = -p$$

The bottom boundary is fixed with zero displacements in the x , y , and z directions. All the other boundaries are free.

RESULTS AND DISCUSSION

The pressure distribution in the engine oil is shown in Figure 2. As one would expect, the pressure is at maximum toward the region of diminishing film height. The corresponding elastic deformation (exaggerated) and displacement of the tilted pad is shown in Figure 3.

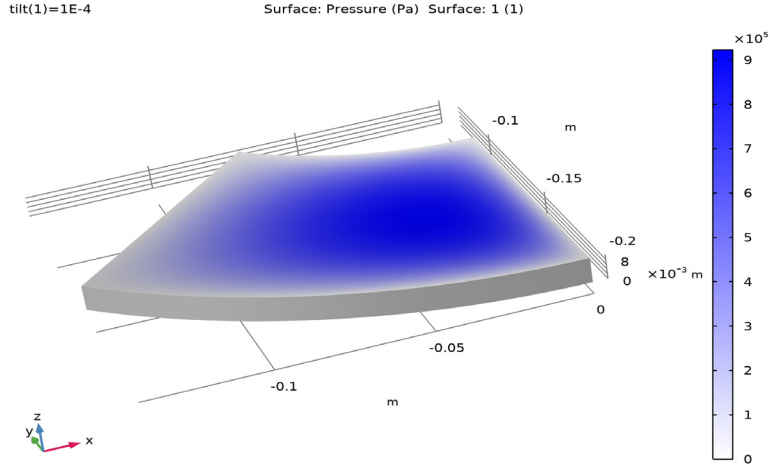


Figure 2: Pressure distribution in engine oil.

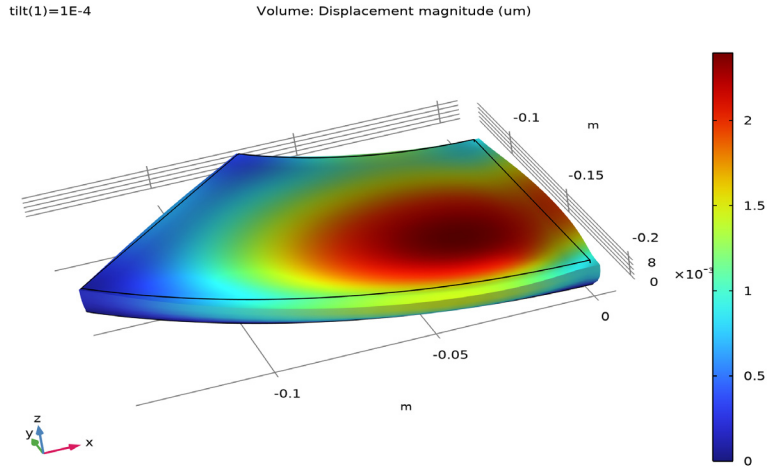



Figure 3: Elastic deformation of the tilted pad.

Application Library path: CFD_Module/Thin-Film_Flow/tilted_pad_bearing



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>Stationary**.

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
tilt	1e-4	1E-4	Tilt parameter
h0	10[um]	1E-5 m	Initial film thickness
W_ext	5[kN]	5000 N	External load
omega	1[rad/s]	1 rad/s	Angular velocity

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click  **Go to Plane Geometry**.

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click  **Circle**.

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

3 In the **Radius** text field, type 0.2.


4 In the **Sector angle** text field, type 45.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type -135.

6 Click to expand the **Layers** section. In the table, enter the following settings:

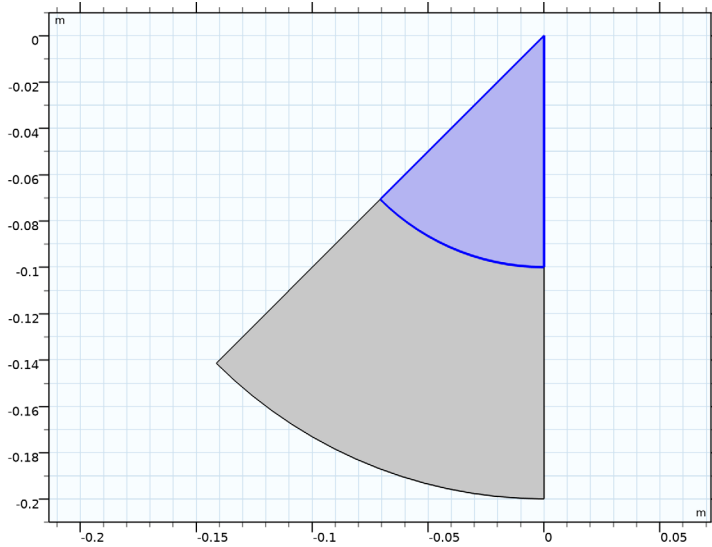
Layer name	Thickness (m)
Layer 1	0.1

7 Click  **Build Selected**.


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

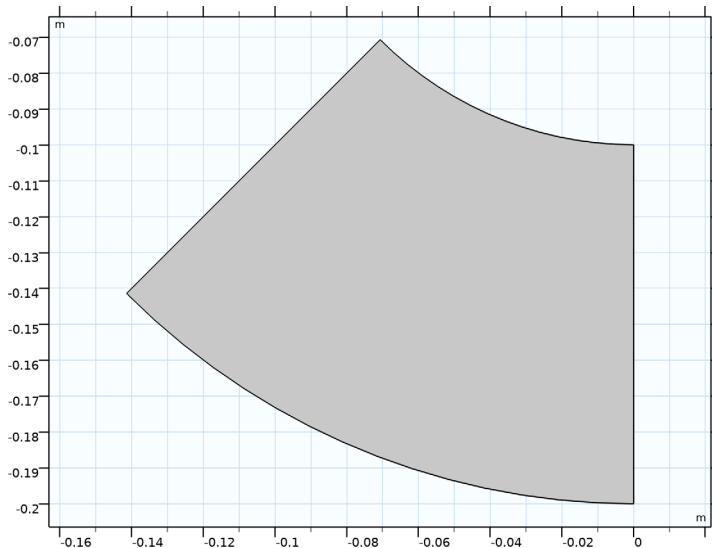
Work Plane 1 (wpl)>Delete Entities 1 (del)

- 1** Right-click **Plane Geometry** and choose **Delete Entities**.
- 2** In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
- 3** From the **Geometric entity level** list, choose **Domain**.
- 4** On the object **c1**, select Domain 1 only.



- 5** Click  **Build Selected**.


- 6 Click the  **Zoom Extents** button in the **Graphics** toolbar.






Extrude 1 (ext1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.
- 2 In the **Settings** window for **Extrude**, locate the **Distances** section.
- 3 In the table, enter the following settings:

Distances (m)
0.01

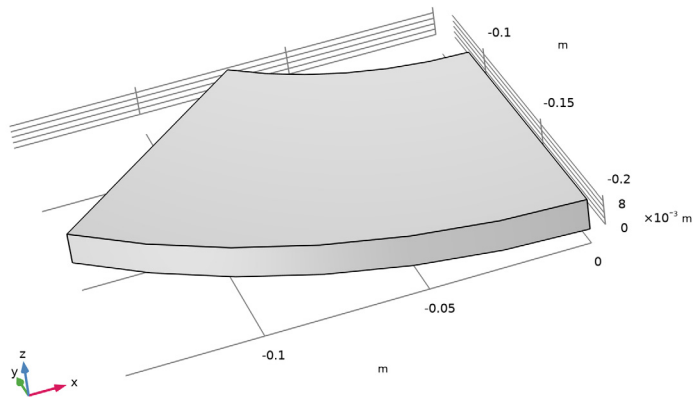
- 4 In the **Geometry** toolbar, click  **Build All**.

Form Union (fin)

- 1 Click  **Build All**.
- 2 Click the  **Go to Default View** button in the **Graphics** toolbar.
- 3 Click the  **Zoom Extents** button in the **Graphics** toolbar.

The model geometry is now complete.

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




DEFINITIONS

Cylindrical System 2 (sys2)

In the **Definitions** toolbar, click  **Coordinate Systems** and choose **Cylindrical System**.

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
U	$-\omega \cdot \text{sys2}.r \cdot \sin(\text{sys2}.\phi)$	m/s	Slider velocity, x-component
V	$\omega \cdot \text{sys2}.r \cdot \cos(\text{sys2}.\phi)$	m/s	Slider velocity, y-component
F_fluid	intop1(pfilm)		Total fluid load


Here intop1 refers to a nonlocal integration coupling for the sliding surface; define this coupling as follows.

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 Select Boundary 4 only.

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 Click  **Clear Selection**.
- 4 Select Boundary 4 only.

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 $h0\text{-}t\text{ilt}\text{*}x$.
- 4 From the u_w list, choose **Displacement field (solid)**.
- 5 Locate the **Base Properties** section. Specify the u_b vector as


0	x
0	y
Deltax_b	z


Here, Deltax_b is a constant displacement that takes a value such that the total load constraint in [Equation 1](#) is fulfilled. You will define Deltax_b using a Global Equation feature node added to the Structural Mechanics user interface shortly.

- 6 From the v_b list, choose **User defined**. Specify the vector as

U	x
V	y
0	z

ADD MATERIAL

- 1 In the **Home** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-in>Nylon**.
- 4 Click **Add to Component** in the window toolbar.

- 5 In the tree, select **Liquids and Gases>Liquids>Engine oil**.
- 6 Click **Add to Component** in the window toolbar.
- 7 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

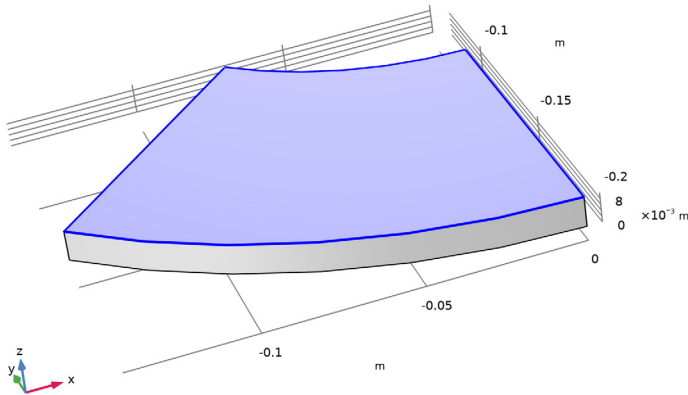
Nylon (mat1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Nylon (mat1)**.
- 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
Density	rho	0.33	kg/m ³	Basic

Engine oil (mat2)

- 1 In the **Model Builder** window, click **Engine oil (mat2)**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 4 only.





SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.


Fixed Constraint 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundary 3 only.





Boundary Load 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 From the \mathbf{F}_A list, choose **Fluid load on wall (tff/ffp1)**.
- 5 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.
- 7 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	$f(u,ut,utt,t)$ (l)	Initial value (u_0) (l)	Initial value (u_t0) (l/s)	Description
Deltax_b	F_fluid-W_ext	0	0	Channel base displacement enforcing total load constraint

- 4 Locate the **Units** section. Click  **Select Dependent Variable Quantity**.
- 5 In the **Physical Quantity** dialog box, type displacement in the text field.
- 6 Click  **Filter**.
- 7 In the tree, select **General>Displacement (m)**.
- 8 Click **OK**.
- 9 In the **Settings** window for **Global Equations**, locate the **Units** section.
- 10 Click  **Select Source Term Quantity**.
- 11 In the **Physical Quantity** dialog box, type force in the text field.
- 12 Click  **Filter**.
- 13 In the tree, select **General>Force (N)**.


14 Click **OK**.

MESH I

Mapped I

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Mapped**.
- 2 Select Boundary 4 only.

Size

- 1 In the **Model Builder** window, click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Finer**.
- 4 Click  **Build All**.

Swept I

- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, click  **Build All**.



STUDY I

Step 1: Stationary

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.


In the first step, solve only for the Lubrication Shell variables to get an initial solution for the coupled problem. Add a separate study step for the latter solution stage.

Step 2: Stationary 2


- 1 In the **Study** toolbar, click  **Study Steps** and choose **Stationary>Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
- 3 Select the **Auxiliary sweep** check box.
- 4 Click  **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
tilt (Tilt parameter)	range (1e-4, (5e-4-1e-4) / 19, 5e-4)	

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Dependent Variables 2**.
- 4 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.
- 5 From the **Method** list, choose **Manual**.
- 6 In the **Scale** text field, type $1e-5$.
- 7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** click **Pressure (comp1.pfilm)**.
- 8 In the **Settings** window for **Field**, locate the **Scaling** section.
- 9 From the **Method** list, choose **Initial value based**.

The pressure differs in scale by about 11 orders of magnitude. It therefore needs a separate scaling factor, which you can take from the initial solution.

The default segregated solver does not work for this problem, which is better solved fully coupled.
- 10 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node.
- 11 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** and choose **Fully Coupled**.
- 12 In the **Study** toolbar, click  **Compute**.

RESULTS

The first default plot group shows the fluid pressure as a surface plot for the final tilt value in the parametric sweep. Modify this plot to display the one shown in [Figure 2](#) as follows:

Fluid Pressure (tff)

- 1 In the **Model Builder** window, under **Results** click **Fluid Pressure (tff)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (tilt)** list, choose **1E-4**.
- 4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1



- 1 In the **Model Builder** window, expand the **Fluid Pressure (tff)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

- 3 From the **Coloring** list, choose **Gradient**.
- 4 From the **Bottom color** list, choose **Blue**.
- 5 From the **Color table transformation** list, choose **Reverse**.

Surface 2


- 1 In the **Model Builder** window, right-click **Fluid Pressure (tff)** 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**.

Selection 1



- 1 Right-click **Surface 2** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.
- 4 Select Boundaries 1–3, 5, and 6 only.
- 5 In the **Fluid Pressure (tff)** toolbar, click  **Plot**.
- 6 Click the  **Zoom Extents** button in the **Graphics** toolbar.

The second default plot group shows a surface plot of the von Mises stress and a deformation plot of the displacement. Change this plot group to display the deformation plot in [Figure 3](#) by following these steps:

Stress (solid)

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, click  **Plot First**.

Volume 1

- 1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.
- 2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>solid.disp - Displacement magnitude - m**.
- 3 Locate the **Expression** section. In the **Unit** field, type um.
- 4 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.
- 6 Click **OK**.
- 7 In the **Stress (solid)** toolbar, click  **Plot**.

The following steps reproduce [Figure 1](#).



Fluid Pressure (tff)

In the **Model Builder** window, under **Results** right-click **Fluid Pressure (tff)** and choose **Duplicate**.


Lubricant thickness

- 1 In the **Model Builder** window, under **Results** click **Fluid Pressure (tff) 1**.
- 2 In the **Settings** window for **3D Plot Group**, type **Lubricant thickness** in the **Label** text field.

Surface 1

- 1 In the **Model Builder** window, expand the **Lubricant thickness** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Thin-Film Flow>Wall and base properties>tff.h - Total gap height - m**.
- 3 Locate the **Expression** section. From the **Unit** list, choose **µm**.
- 4 In the **Lubricant thickness** toolbar, click  **Plot**.
- 5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Color table**.
- 6 Click  **Change Color Table**.
- 7 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.
- 8 Click **OK**.
- 9 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 10 From the **Color table transformation** list, choose **None**.

Arrow Surface 1

- 1 In the **Model Builder** window, right-click **Lubricant thickness** and choose **Arrow Surface**.
- 2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Thin-Film Flow>Wall and base properties>tff.vbtx,...,tff.vbtz - Tangential velocity of base (spatial frame)**.
- 3 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
- 4 From the **Color** list, choose **White**.
- 5 In the **Lubricant thickness** toolbar, click  **Plot**.