

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/m³), h is the lubricant thickness, η is the viscosity (SI unit: Pa.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 constraint:

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

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

2 | TILTED PAD THRUST BEARING

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

- I In the Model Wizard window, click 间 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 \bigcirc Study.

7 In the Select Study tree, select General Studies>Stationary.

8 Click 🗹 Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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	IE-4	Tilt parameter
h0	10[um]	IE-5 m	Initial film thickness
W_ext	5[kN]	5000 N	External load
omega	1[rad/s]	l rad/s	Angular velocity

GEOMETRY I

Work Plane I (wp1)

I In the Geometry toolbar, click 📥 Work Plane.

2 In the Settings window for Work Plane, click Show Work Plane.

Work Plane I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

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

- I 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 틤 Build Selected.

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

Work Plane I (wp1)>Delete Entities I (del1)

- I 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.





Extrude I (extI)

- In the Model Builder window, under Component I (compl)>Geometry I right-click
 Work Plane I (wpl) 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)

- I Click 📗 Build All.
- 2 Click the View button in the Graphics toolbar.

3 Click the 4 **Zoom Extents** button in the **Graphics** toolbar.

The model geometry is now complete.



DEFINITIONS

Cylindrical System 2 (sys2) In the **Definitions** toolbar, click $\swarrow^{z \to y}$ **Coordinate Systems** and choose **Cylindrical System**.

Variables I

- I In the **Definitions** toolbar, click $\partial =$ **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*sys2.r*sin(sys2.phi)	m/s	Slider velocity, x-component
V	omega*sys2.r*cos(sys2.phi)	m/s	Slider velocity, y-component
F_fluid	<pre>intop1(pfilm)</pre>		Total fluid load

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

Integration 1 (intop1)

I In the Definitions toolbar, click Ronlocal 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)

- I In the Model Builder window, under Component I (compl) 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

- I In the Model Builder window, under Component I (compl)>Thin-Film Flow (tff) click Fluid-Film Properties I.
- 2 In the Settings window for Fluid-Film Properties, locate the Wall Properties section.
- **3** In the h_{w1} text field, type h0-tilt*x.
- **4** From the \mathbf{u}_w list, choose **Displacement field (solid)**.
- 5 Locate the Base Properties section. Specify the **u**_b vector as

0	x
0	у
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 \mathbf{v}_b list, choose **User defined**. Specify the vector as

U	x
V	у
0	z

ADD MATERIAL

I 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)

- I In the Model Builder window, under Component I (compl)>Materials click Nylon (matl).
- 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/m³	Basic

Engine oil (mat2)

- I 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 I (compl) click Solid Mechanics (solid).

Fixed Constraint I

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

Boundary Load I

- I 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 **F**_A list, choose Fluid load on wall (tff/ffpl).
- 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

- I 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) (1)	Initial value (u_t0) (1/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.
- IO Click Select Source Term Quantity.
- II In the Physical Quantity dialog box, type force in the text field.

12 Click 🖶 Filter.

I3 In the tree, select **General>Force (N)**.

I4 Click OK.

MESH I

Mapped I

- I In the Mesh toolbar, click \triangle Boundary and choose Mapped.
- **2** Select Boundary 4 only.

Size

- I 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

- I In the Mesh toolbar, click 🆓 Swept.
- 2 In the Settings window for Swept, click 📗 Build All.

STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: 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.

Stationary 2

- I 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 (soll)

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll) 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 I>Solver Configurations>Solution I (soll)> Dependent Variables 2 click Pressure (compl.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.

- IO In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Stationary Solver 2 node.
- II Right-click Study I>Solver Configurations>Solution I (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)

- I 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 IE-4.

4 Locate the Plot Settings section. Clear the Plot dataset edges check box.

Surface 1

- I In the Model Builder window, expand the Fluid Pressure (tff) node, then click Surface I.
- 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

- I 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

- I 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 \longleftrightarrow **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)

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

Volume 1

- I In the Model Builder window, expand the Stress (solid) node, then click Volume I.
- 2 In the Settings window for Volume, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>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.

Lubricant thickness

- I In the Model Builder window, right-click Fluid Pressure (tff) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Lubricant thickness in the Label text field.

Surface 1

- I In the Model Builder window, expand the Lubricant thickness node, then click Surface I.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>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 **I** 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.
- **IO** From the **Color table transformation** list, choose **None**.

Arrow Surface 1

- I 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 I (compl)>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 **OM Plot**.

17 | TILTED PAD THRUST BEARING

18 | TILTED PAD THRUST BEARING