Journal Bearing

Introduction

Journal bearings are used to carry radial loads, for example, to support a rotating shaft.

A simple journal bearing consists of two rigid cylinders. The outer cylinder (bearing) wraps the inner rotating journal (shaft). Normally, the position of the journal center is eccentric with the bearing center. A lubricant fills the small annular gap or clearance between the journal and the bearing. The amount of eccentricity of the journal is related to the pressure that will be generated in the bearing to balance the radial load. The lubricant is supplied through a hole or a groove and may or may not extend all around the journal.

Under normal operating conditions, the gases dissolved in the lubricant cause cavitation in the diverging clearance between the journal and the bearing. This happens because the pressure in the lubricant drops below the saturation pressure for the release of dissolved gases. The saturation pressure is normally similar to the ambient pressure. The following model does not account for cavitation and therefore predicts sub-ambient pressures. Such sub-ambient pressures are the result of the so-called Sommerfeld boundary condition. For practical purposes, these sub-ambient pressures should be neglected. Future versions of the Lubrication Shell interface will offer additional tools for modeling cavitation.

Model Definition

The pressure in the lubricant (SAE 10 at 70° C) is governed by the Reynolds equation. For an incompressible fluid with no-slip condition, the stationary Reynolds equation in the continuum range is given by

$$\nabla_T \cdot \left(\frac{-\rho h^3}{12\eta} \nabla_T p + \frac{\rho h}{2}(v_a + v_b) \right) - \rho((\nabla_T b \cdot v_b) - (\nabla_T a \cdot v_a)) = 0 \qquad (1)$$

In this equation, ρ is the density (kg/m^3), h is the lubricant thickness (m), η is the viscosity (Pa·s), p is the pressure (Pa), a is the location (m) of the channel base, v_a is the tangential velocity (m/s) of the channel base, b is the location (m) of the solid wall, and v_b is the tangential velocity (m/s) of the solid wall. Here the rotating journal is considered to be the solid wall. Figure 1 shows the rotating journal wall on which you solve the Reynolds equation. Because the pressure is constant through the lubricant

film thickness, COMSOL uses the tangential projection of the gradient operator, ∇_T, to calculate the pressure distribution on the lubricant surface. Note that in this case the term $\rho((\nabla_T b \cdot v_b) - (\nabla_T a \cdot v_a))$ equates to 0, so the governing equation simplifies to

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

The lubricant thickness, h, is defined as

$$h = c(1 + \varepsilon \cos\theta)$$

where $c \equiv R_B - R_J$ is the difference between the bearing radius and the journal radius, ε is the eccentricity, and θ is the polar angular coordinate of a point on the lubricant. Figure 2 shows the converging and diverging lubricant thickness around the journal.

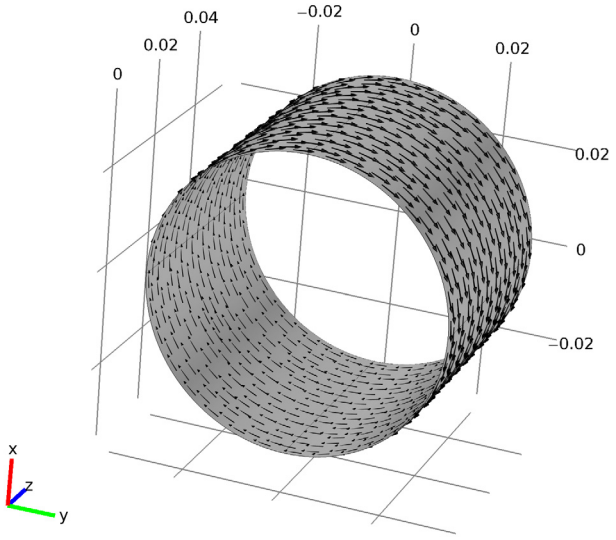

Figure 1: Geometry (cylindrical journal) showing the velocity direction with black arrows.

©2012 COMSOL

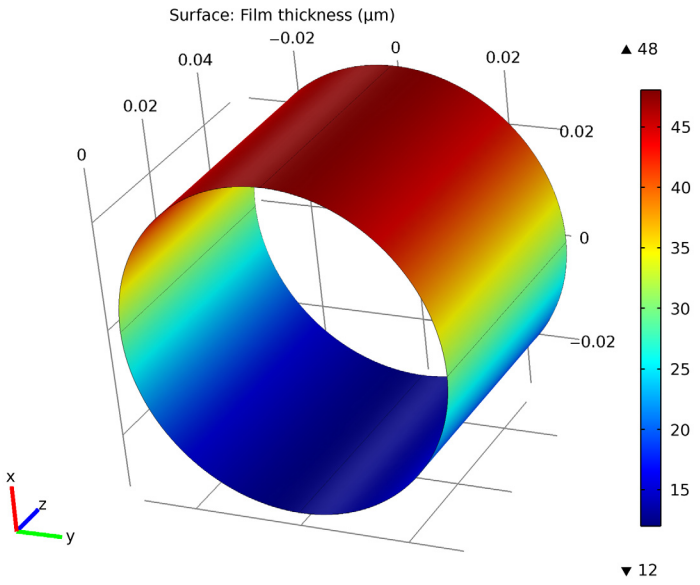

Figure 2: The lubricant thickness around the rotating journal.

BORDER CONDITIONS

The pressure at the ends of the cylindrical journal is assumed to be similar to the ambient pressure. Therefore, the border conditions are

$$p = 0 \quad \text{at} \quad z = 0, L \tag{3}$$

where L is the length of the cylindrical journal.

Results and Discussion

Figure 3 shows the calculated pressure distribution and pressure contours. As expected, the maximum pressure is reached in a region closer to the minimum lubricant thickness. Sub-ambient or negative pressure also results due to approximate boundary conditions. For a more accurate modeling of pressure distribution, gaseous cavitation has to be taken into account.

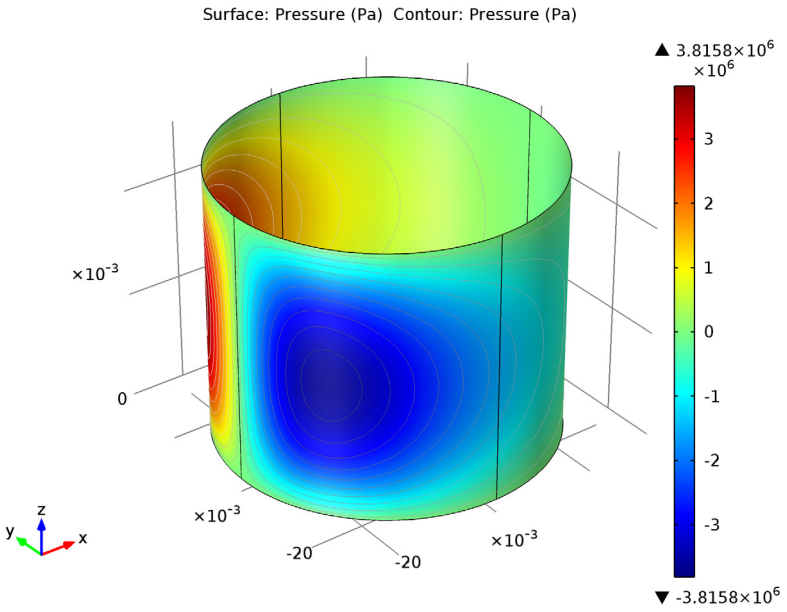

Figure 3: Pressure distribution and pressure contours on the journal.

Model Library path: `CFD_Module/Thin-Film_Flow/journal_bearing`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Fluid Flow>Thin-Film Flow>Thin-Film Flow, Shell (tffs)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

 ©2012 COMSOL

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
R	0.03[m]	Journal radius
H	0.05[m]	Journal height
c	0.03[mm]	Clearance between the bearing and the journal
omega	1500/60*2*pi[rad/s]	Journal angular velocity

GEOMETRY 1

Cylinder 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size and Shape** section. In the **Radius** edit field, type R.

5 In the **Height** edit field, type H.

6 Click the **Build All** button.

7 Click the **Zoom Extents** button on the Graphics toolbar.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
angle	atan2(y,x)[rad]	Angle along circumference
h	c*(1+0.6*cos(angle))	Lubricant film thickness

Name	Expression	Description
u	-omega*R*sin(angle)	x-component of journal velocity
v	omega*R*cos(angle)	y-component of journal velocity

THIN-FILM FLOW, SHELL

Fluid-Film Properties 1

1 In the **Model Builder** window, expand the **Model 1>Thin-Film Flow, Shell** node, then click **Fluid-Film Properties 1**.

2 In the **Fluid-Film Properties** settings window, locate the **Fluid Properties** section.

3 From the ρ list, choose **User defined**. In the associated edit field, type `860[kg/m^3]`.

4 From the μ list, choose **User defined**. In the associated edit field, type `0.01[Pa*s]`.

5 Locate the **Wall Properties** section. In the h_{w1} edit field, type `h`.

6 Click to expand the **Base Properties** section. From the $\mathbf{v}_b$ list, choose **User defined**. In the table, enter the following settings:

u	x
v	y
0	z

Border 1

As you can see in the **Border Settings** section, the default condition that applies at the cylinder ends is **Zero pressure**.

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Fluid Pressure

The default plot group shows the pressure field as a surface plot and the displacement of the solid wall as a deformation plot. Add a contour plot of the same quantity to reproduce the plot in Figure 3.

 ©2012 COMSOL

1 In the **Model Builder** window, under **Results** right-click **Fluid Pressure** and choose **Contour**.

2 In the **Contour** settings window, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **GrayScale**.

4 Clear the **Color legend** check box.

To see the bearing from different angles just click and drag in the Graphics window.

 ©2012 COMSOL