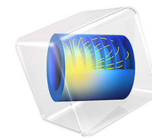

Journal Bearing

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 is 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.

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

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

In this equation, ρ is the density (SI unit: $\mathrm{kg/m^3}$), h is the lubricant thickness (SI unit: m), μ is the viscosity (SI unit: Pa·s), p is the pressure (SI unit: Pa), a is the location (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.

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 $\rho((\nabla_t b \cdot v_\mathrm{b}) - (\nabla_t a \cdot v_\mathrm{a}))$ equates to 0, so the governing equation simplifies to

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

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.

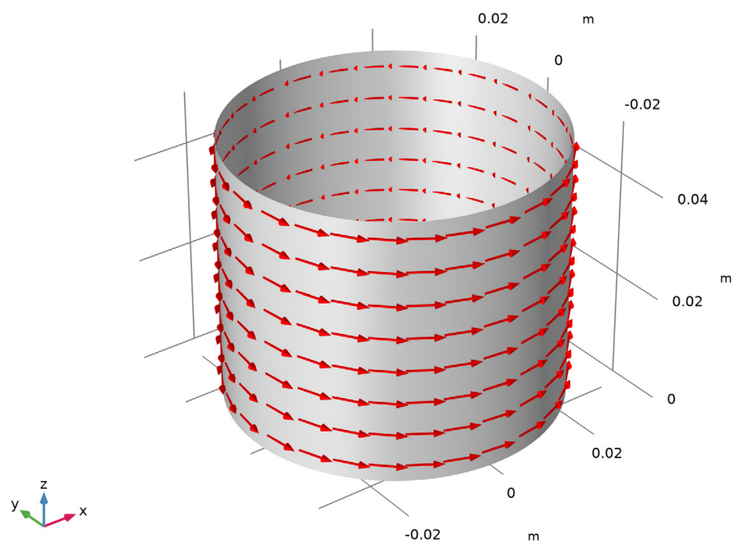

Figure 1: Geometry (cylindrical journal) showing the base velocity direction with red arrows.

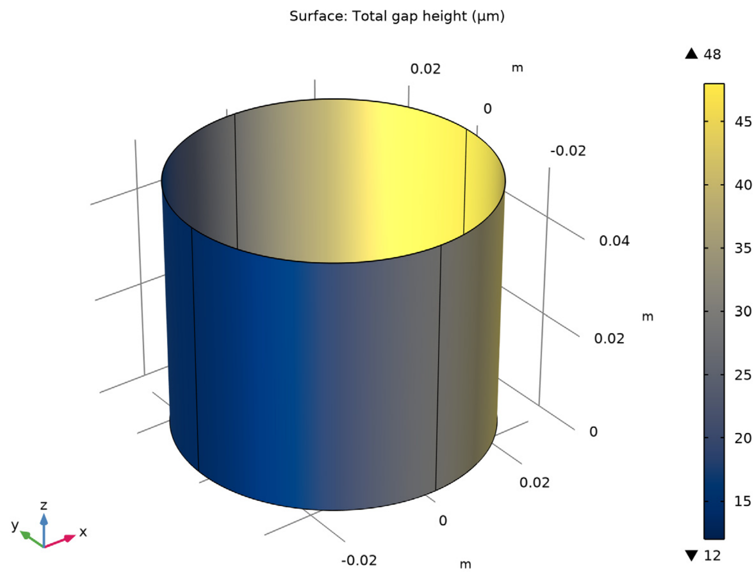

Figure 2: The lubricant thickness around the rotating journal.

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

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

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.

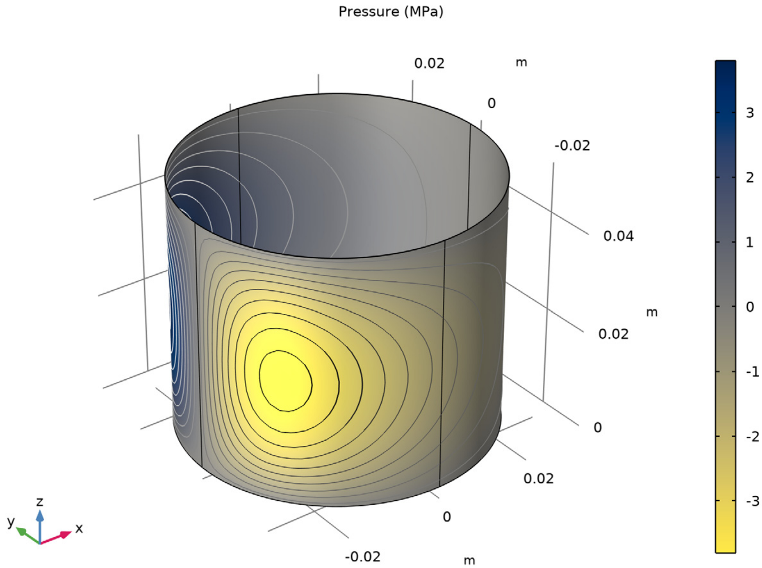

Figure 3: Pressure distribution and pressure contours on the journal.

Application Library path: `CFD_Module/Thin-Film_Flow/journal_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 Click **Study**.

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

6 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
R	0.03[m]	0.03 m	Journal radius
H	0.05[m]	0.05 m	Journal height
c	0.03[mm]	3E-5 m	Clearance between the bearing and the journal
omega	1500/60*2*pi[rad/s]	157.08 rad/s	Journal angular velocity

GEOMETRY 1

Cylinder 1 (cyl1)

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

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

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

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

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

6 Click **Build All Objects**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **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
angle	atan2(y,x)[rad]	rad	Angle along circumference
th	c*(1+0.6*cos(angle))	m	Lubricant film thickness

Name	Expression	Unit	Description
u_b	-omega*R*sin(angle)	m/s	x-component of journal velocity
v_b	omega*R*cos(angle)	m/s	y-component of journal velocity

THIN-FILM FLOW (TFF)

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 **Fluid Properties** section.

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

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

5 Locate the **Wall Properties** section. In the h_{w1} text field, type th.

6 Locate the **Base Properties** section. From the $\mathbf{v}_b$ list, choose **User defined**. Specify the vector as

u_b	x
v_b	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**.

STUDY 1

In the **Home** toolbar, click **=** **Compute**.

RESULTS

Fluid Pressure (tff)

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

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 **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

8 From the **Color table transformation** list, choose **Reverse**.

Contour 1

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

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

3 From the **Unit** list, choose **MPa**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Contour**, locate the **Coloring and Style** section.

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

Fluid Pressure (tff)

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

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Pressure (MPa)`.

5 In the **Fluid Pressure (tff)** toolbar, click **Plot**.

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

The following steps reproduce Figure 1.

Velocity direction

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Surface 1

1 Right-click **Velocity direction** and choose **Surface**.

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

3 From the **Coloring** list, choose **Uniform**.

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

Arrow Surface 1

1 In the **Model Builder** window, right-click **Velocity direction** 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.vbx,tff.vby,tff.vbz - Velocity of base**.

Velocity direction

1 In the **Model Builder** window, click **Velocity direction**.

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

3 Clear the **Plot dataset edges** check box.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Reproduce Figure 2 by the following steps.

Lubricant thickness

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Surface 1

1 Right-click **Lubricant thickness** and choose **Surface**.

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 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

6 Click **OK**.

Surface 1

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

2 In the **Lubricant thickness** toolbar, click **Plot**.

Lubricant thickness

1 In the **Model Builder** window, click **Lubricant thickness**.

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

3 Select the **Show maximum and minimum values** check box.