Created in COMSOL Multiphysics 6.1

Journal Bearing with Cavitation

Introduction

Journal bearings are used to carry radial loads to, for example, 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.

If the bearing is not designed correctly, the gases dissolved in the lubricant can 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. Cavitation can cause damage to the bearing components leading to premature failure.

The following model predicts the onset and extent of cavitation in the lubrication layer. The onset and extent of gaseous cavitation in a journal bearing determine the load that can be applied to the bearing.

This example is based on the Journal Bearing model, that does not include cavitation effects; review that model before beginning this one.

Model Definition

The governing equation, geometry and boundary conditions are discussed for the Journal Bearing model.

With the cavitation feature enabled, the flow in the journal bearing is divided in two regions:

- A full film region where the pressure varies but is limited from below by the cavitation pressure.
- A cavitation region where only part of the volume is occupied by the fluid. Because of the presence of the gas in the void fraction, the pressure in this region is assumed to be constant and equal to the cavitation pressure.

Elrod and Adams derived a general form of the Reynolds equation by introducing a switch function, g, equal to 1 in the full film region ($\theta \geq 1$) and 0 in the cavitation region ($\theta < 1$). This switch function allows for solving a single equation for both the full film and the cavitation region and leads to a modified version of the average velocity used in the Reynold's equation:

$$\mathbf{v}_{av} = \mathbf{v}_{av,c} - g v_{av,p} \nabla_t p_f$$

where the first and second terms on the right-hand side correspond to the average Couette and average Poiseuille velocities, respectively. This switch function sets the average Poiseuille velocity is to zero in the cavitation region.

Because the average Poiseuille velocity is set to zero in the cavitation region, the density needs to be a function of the pressure variable and could be defined as

$$\rho = \rho_c e^{\beta p_f}$$

A density that is not pressure dependent would lead to empty equations in the cavitation region since the pressure variable p would no longer be present in the governing equations.

Results and Discussion

While the pressure is constant and equal to the cavitation pressure in the cavitation region, the computed pressure, `pfilm`, is negative in this region. The value of this negative pressure can be used to derive the volume fraction of fluid in the cavitation region. The actual or physical pressure, available in the postprocessing section as `tff.p`, is equal to the computed pressure in the full film region and equal to the cavitation pressure in the cavitation region. Figure 1 shows this physical pressure, `tff.p`. The maximum pressure is reached in a region closer to the minimum lubricant thickness.

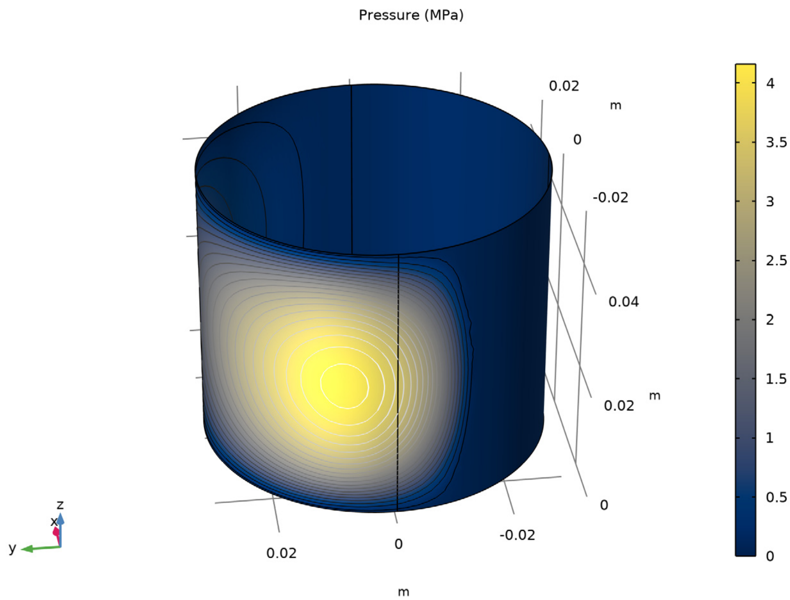

Figure 1: Pressure distribution and pressure contours on the journal.

Figure 2 shows the fluid mass fraction. The mass fraction is equal to 1 in the full film region and less than 1 in the cavitation region (where only part of the volume is occupied by the fluid). It is computed as the minimum value between 1 and the ratio ρ/ρ_{cav}, where ρ and ρ_{cav} represent the fluid density and the density at the cavitation pressure, respectively.

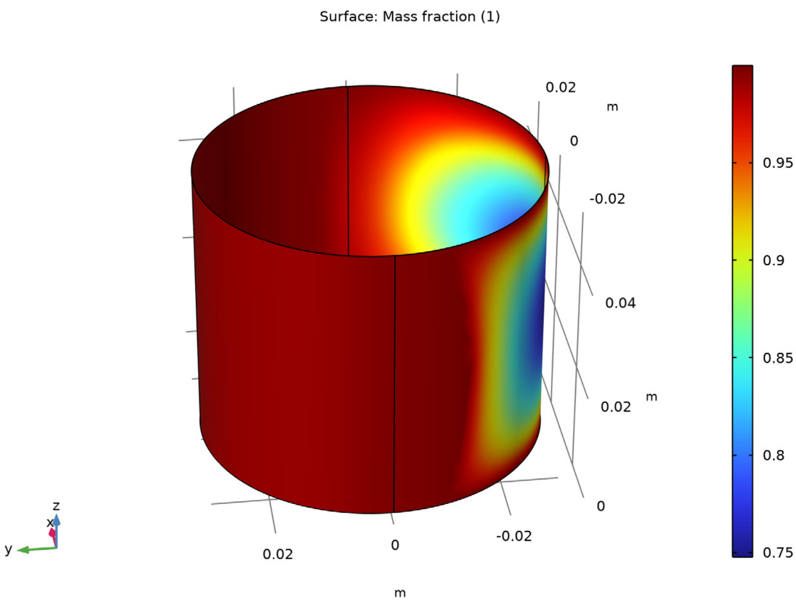

Figure 2: Fluid mass fraction.

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

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)

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

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

3 Click **OK**.

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

5 In the **Settings** window for **Thin-Film Flow**, locate the **Physical Model** section.

6 From the **Fluid type** list, choose **Liquid with cavitation**.

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

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

5 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

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

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 1.

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. In the **Expression** text field, type `tff.p`.
4. From the **Unit** list, choose **MPa**.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.
6. In the **Color Table** dialog box, select **Linear>Cividis** in the tree.
7. Click **OK**.

Surface 1

1. In the **Model Builder** window, click **Surface 1**.
2. In the **Fluid Pressure (tff)** toolbar, click **Plot**.

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. In the **Expression** text field, type `tff.p`.
4. From the **Unit** list, choose **MPa**.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.
6. In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.
7. Click **OK**.
8. In the **Settings** window for **Contour**, locate the **Coloring and Style** section.
9. 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**.

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

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

Mass Fraction

Reproduce Figure 2 by the following these steps.

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

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

Surface 1

1 In the **Mass Fraction** toolbar, click **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> Cavitation>tff.theta - Mass fraction**.

3 In the **Mass Fraction** toolbar, click **Plot**.