Step Thrust Bearing

Introduction

In this example the pressure distribution in a step thrust bearing is analyzed. A step thrust bearing consists of a stepped bearing surface on which the end of the shaft rotates. The entire assembly is submerged in a lubricant. A six step thrust bearing is considered in this example. The shaft collar is assumed to be spinning without any axial motion in the bearing. The simulation is performed using the Rotordynamics Module's Hydrodynamic Bearing interface. This interface solves the Reynolds equation to compute the pressure developed in a thin fluid film.

Model Definition

A six-pad step thrust bearing is considered. The inner and outer pad diameters are 0.1 m and 0.2 m, respectively. The grooves span 15° and have a depth of 200 μm.

The bearing is analyzed under various conditions. This includes a parametric study of the angular speed and the film thickness. Here the angular speed and film thickness are varied in the intervals $\Omega \in [500,1000]$ rad/s and $h_{\mathrm{film}} \in [60,160]$ μm, respectively.

The bearing geometry is shown in Figure 1 below.

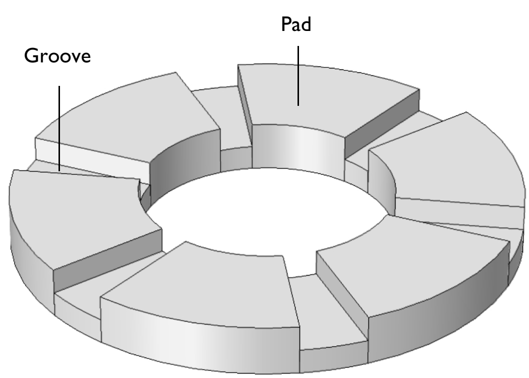

Figure 1: Step thrust bearing geometry.

The effect of cavitation is also included in the computation of the fluid-film pressure distribution. The following fluid properties are needed — the dynamic viscosity, the density at cavitation pressure, and the compressibility. The utilized fluid parameters are summarized in Table 1. The selected values are close to those of lubricating oils used in real bearings.

TABLE 1: FLUID PROPERTIES.

Property	Value
Density ρ	866 kg/m^3
Dynamic viscosity μ	0.072 Pa·s
Compressibility β	10^{-7} Pa^{-1}

Results and Discussion

Figure 2 shows the fluid pressure profile in the bearing for the highest values of angular speed and film thickness.

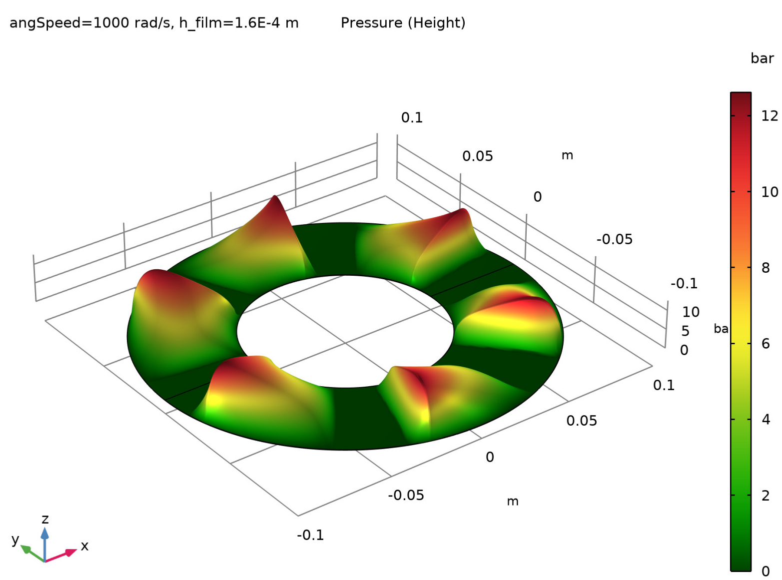

Figure 2: Fluid-film pressure profile at Ω = *1000 rad/s with a film thickness of 160* μ*m.*

The resulting vertical force on the collar associated with the pressure distribution shown above is 7500 N. This is the load carrying capacity of the bearing at the specified speed.

The mass fraction of the lubricant, which is a measure of the cavitation, is shown in Figure 3. It is clear that amount of cavitation is very small and is localized near the trailing edge of the pads.

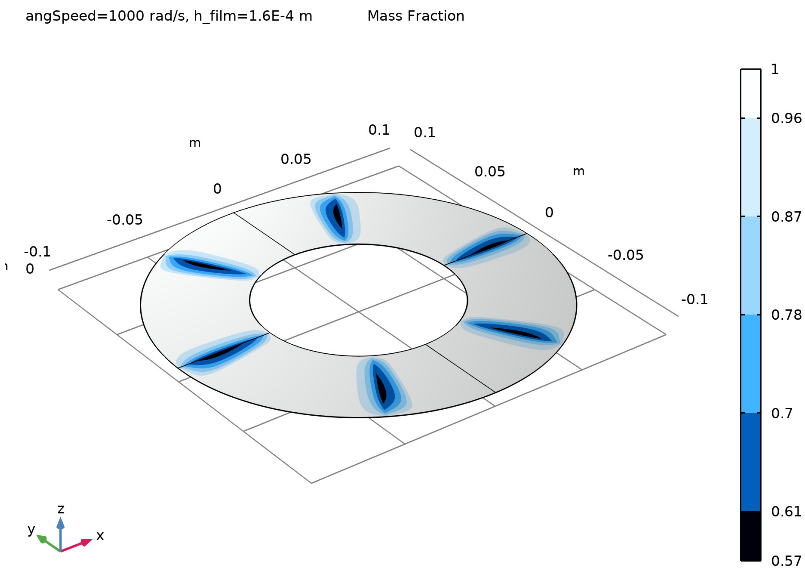

Figure 3: Mass fraction at Ω = 1000 rad/s with a film thickness of 160 μm.

The pressure distribution in radial and circumferential directions on a single pad are shown in Figure 4 and Figure 5, respectively. The pressure at the inner and outer radius locations are zero with the distribution marginally biased toward the outer edge. The velocity of the collar varies linearly from the inner to the outer radius. Since the pressure is proportional to the shear velocity in the film, it is expected to increase linearly from the inner point to the outer point. However, along the inner and outer edge the film pressure is set to zero. Therefore, the maximum pressure occurs toward the outer side relative to the mid position. In the circumferential direction, the pressure suddenly rises at the step location, that is, the leading edge of the pad, and reduces slowly toward the trailing edge.

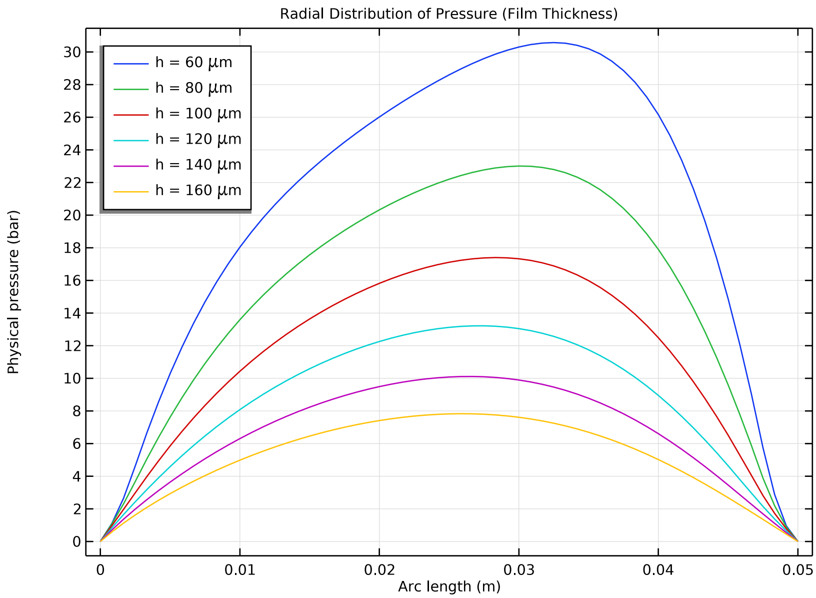

Figure 4: Pressure profiles in radial direction at Ω *= 1000 rad/s.*

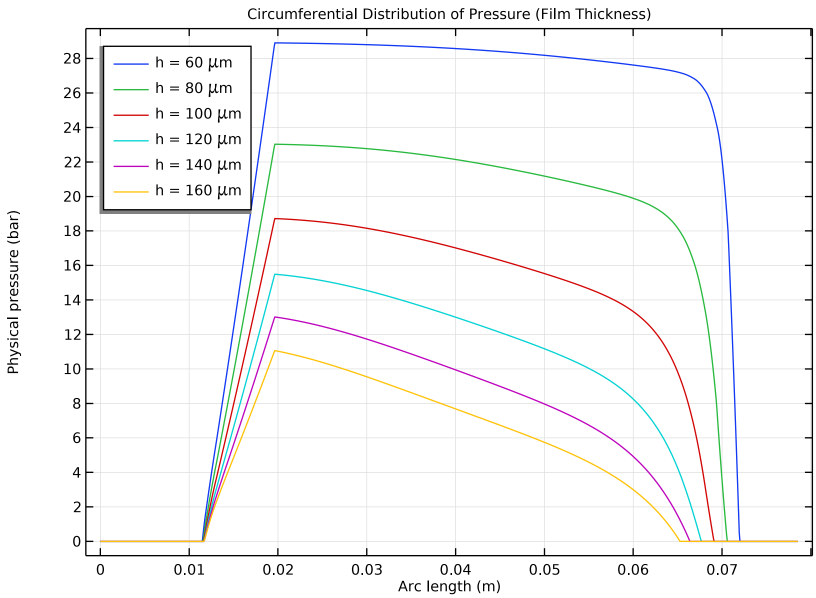

Figure 5: Pressure profiles in circumferential direction at Ω *= 1000 rad/s.*

Figure 6 shows bearing profile computed using the film thickness.

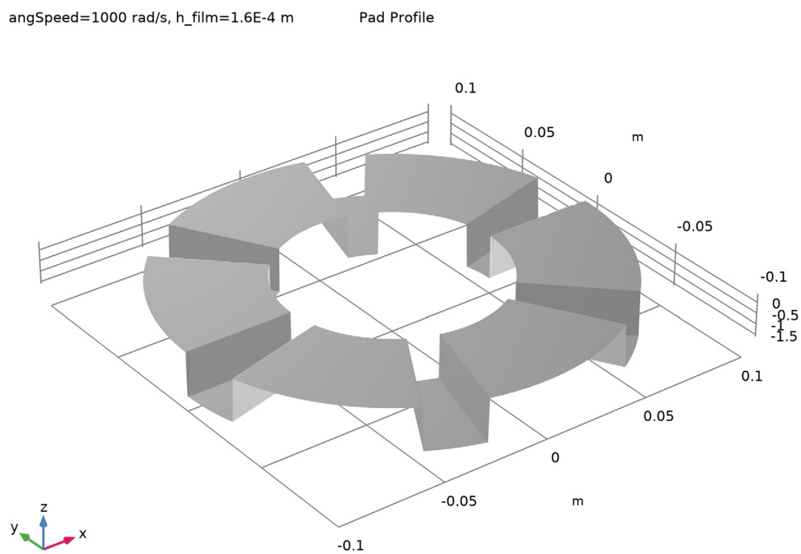

Figure 6: Bearing profile.

Application Library path: `Rotordynamics_Module/Tutorials/step_thrust_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 **Structural Mechanics** > **Rotordynamics** > **Hydrodynamic Bearing (hdb)**.
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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `step_thrust_bearing_parameters.txt`.

GEOMETRY 1

Work Plane 1 (wp1)

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

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 `Ro`.

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

Layer name	Thickness (m)
Layer 1	Ro-Ri

5 Click **Build Selected**.

Work Plane 1 (wp1) > Delete Entities 1 (del1)

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 5 only.

5 Click **Build Selected**.

HYDRODYNAMIC BEARING (HDB)

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

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

3 Click **OK**.

Enable the **Cavitation** formulation in the bearing.

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Hydrodynamic Bearing (hdb)**.

5 In the **Settings** window for **Hydrodynamic Bearing**, locate the **Physical Model** section.

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

Reduce the **Cavitation transition width** for the sharper transition between the cavitated and noncavitated regions.

7 In the Δp_{sw} text field, type `0.5[MPa]`.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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
Dynamic viscosity	mu	mu_f	Pa·s	Basic

HYDRODYNAMIC BEARING (HDB)

Hydrodynamic Thrust Bearing 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Hydrodynamic Thrust Bearing**.

2 In the **Settings** window for **Hydrodynamic Thrust Bearing**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Because the reference surface is assumed to be located on the collar, change the **Reference normal orientation** to align it with the collar normal.

4 Locate the **Reference Surface Properties** section. From the **Reference normal orientation** list, choose **Opposite direction to geometry normal**.

5 Locate the **Bearing Properties** section. From the **Bearing type** list, choose **Step**.

6 In the γ_p text field, type `padAng`.

7 In the d_i text field, type `2*Ri`.

8 In the d_o text field, type `2*Ro`.

9 In the h_s text field, type `hg`.

10 In the h_{film} text field, type `h_film`.

11 From the **Groove type** list, choose **Constant arc**.

12 Locate the **Collar Properties** section. In the Ω text field, type `angSpeed`.

13 Locate the **Fluid Properties** section. In the ρ_c text field, type `rho_c`.

Bearing Orientation 1

1 In the **Model Builder** window, click **Bearing Orientation 1**.

2 In the **Settings** window for **Bearing Orientation**, locate the **Bearing Orientation** section.

3 From the **Axis** list, choose **z-axis**.

4 Specify the V vector as

1	x
0	y
0	z

5 In the ϕ text field, type `gAng`.

Initial Values 1

An auxiliary sweep will be used in the stationary study, which does not support parameter dependencies in initial expressions. Therefore, apply a constant initial value for the pressure.

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *pfilm* text field, type `100000[Pa]`.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Create one element per degree in the azimuthal direction to capture the pressure accurately.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 4, 5, 8, and 10 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 90.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 1, 6, 9, and 12 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 20.

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

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
h_film (Film thickness)	range(6e-5,2e-5,16e-5)	m

6 Click + **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
angSpeed (Angular speed of shaft)	range(500,100,1000)	rad/s

8 From the **Sweep type** list, choose **All combinations**.

9 In the **Study** toolbar, click **Compute**.

Set preferred units for the pressure by following the instructions below.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, type `pres` in the text field.

5 In the tree, select **General** > **Pressure (Pa)**.

6 Click **OK**.

7 In the **Settings** window for **Preferred Units**, locate the **Units** section.

8 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Pressure	Pa	bar

9 Click **Apply**.

Default plot shows the pressure distribution in the bearing. To generate the height plot for the pressure distribution shown in Figure 2 start by creating a **Surface dataset**.

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Fluid Pressure (hdb)

1 In the **Model Builder** window, under **Results** click **Fluid Pressure (hdb)**.

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

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

4 Locate the **Color Legend** section. Select the **Show units** checkbox.

Pressure (Height)

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure (Height)` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

1 Right-click **Pressure (Height)** and choose **Surface**.

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

3 In the **Expression** text field, type `hdb.p`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.

Height Expression 1

1 Right-click **Surface 1** and choose **Height Expression**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `2e-3`.

4 Click the **Go to Default View** button in the **Graphics** toolbar.

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

The following instructions are to plot the mass fraction of the lubricant shown in Figure 3.

Mass Fraction

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.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Contour 1

1 Right-click **Mass Fraction** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Hydrodynamic Bearing** > **Cavitation** > **hdb.theta - Mass fraction - 1**.

3 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

4 Locate the **Levels** section. In the **Total levels** text field, type `5`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

6 Click the **Go to Default View** button in the **Graphics** toolbar.

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

Figure 6 shows the bearing profile. Follow the instructions below to replicate it.

2D Plot Group 4

In the **Results** toolbar, click **2D Plot Group**.

Surface 1

1 Right-click **2D Plot Group 4** and choose **Surface**.

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

3 In the **Expression** text field, type `hg-hdb.h`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

Height Expression 1

1 Right-click **Surface 1** and choose **Height Expression**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `100`.

Pad Profile

1 In the **Model Builder** window, under **Results** click **2D Plot Group 4**.

2 In the **Settings** window for **2D Plot Group**, type `Pad Profile` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

6 In the **Pad Profile** toolbar, click **Plot**.

You can also analyze the pressure distributions along the radial and circumferential directions of the bearing shown in Figure 4 and Figure 5. Start by creating a **Cut line** along the radial line.

Cut Line 3D: Radial Line

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 2**, set **X** to `0`.

4 In row **Point 2**, set **Y** to `Ro`.

5 Click **Plot**.

6 In the **Label** text field, type `Cut Line 3D: Radial Line`.

Radial Distribution of Pressure (Film Thickness)

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Line 3D: Radial Line**.

4 From the **Parameter selection (angSpeed)** list, choose **Last**.

5 In the **Label** text field, type `Radial Distribution of Pressure (Film Thickness)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Line Graph 1

1 Right-click **Radial Distribution of Pressure (Film Thickness)** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `hdb.p`.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

5 From the **Legends** list, choose **Evaluated**.

6 In the **Legend** text field, type `h = eval(h_film, um) \mu m`.

7 In the **Radial Distribution of Pressure (Film Thickness)** toolbar, click **Plot**.

Radial Distribution of Pressure (Film Thickness)

1 In the **Model Builder** window, click **Radial Distribution of Pressure (Film Thickness)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

Radial Distribution of Pressure (Angular Speed)

1 Right-click **Radial Distribution of Pressure (Film Thickness)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (h_film)** list, choose **Last**.

4 From the **Parameter selection (angSpeed)** list, choose **All**.

5 In the **Label** text field, type `Radial Distribution of Pressure (Angular Speed)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Radial Distribution of Pressure (Angular Speed)** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

3 In the **Legend** text field, type `\Omega = eval(angSpeed) rad/s`.

4 In the **Radial Distribution of Pressure (Angular Speed)** toolbar, click **Plot**.

Radial Distribution of Pressure (Film Thickness)

In the **Model Builder** window, collapse the **Results** > **Radial Distribution of Pressure (Film Thickness)** node.

Use the **Parametric Curve** to create the circumferential sector line.

Parametric Curve 3D: Circumferential Line

1 In the **Results** toolbar, click **More Datasets** and choose **Parametric Curve 3D**.

2 In the **Settings** window for **Parametric Curve 3D**, locate the **Parameter** section.

3 In the **Maximum** text field, type `2*pi/N`.

4 Locate the **Expressions** section. In the **x** text field, type `0.5*(Ro+Ri)*cos(s)`.

5 In the **y** text field, type `0.5*(Ro+Ri)*sin(s)`.

6 In the **Label** text field, type `Parametric Curve 3D: Circumferential Line`.

7 Click **Plot**.

Radial Distribution of Pressure (Angular Speed), Radial Distribution of Pressure (Film Thickness)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Radial Distribution of Pressure (Film Thickness)** and **Radial Distribution of Pressure (Angular Speed)**.

2 Right-click and choose **Duplicate**.

Circumferential Distribution of Pressure (Film Thickness)

1 In the **Settings** window for **1D Plot Group**, type `Circumferential Distribution of Pressure (Film Thickness)` in the **Label** text field.

2 Locate the **Data** section. From the **Dataset** list, choose **Parametric Curve 3D: Circumferential Line**.

3 In the **Circumferential Distribution of Pressure (Film Thickness)** toolbar, click **Plot**.

Circumferential Distribution of Pressure (Angular Speed)

1 In the **Model Builder** window, under **Results** click **Radial Distribution of Pressure (Angular Speed) 1**.

2 In the **Settings** window for **1D Plot Group**, type `Circumferential Distribution of Pressure (Angular Speed)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Parametric Curve 3D: Circumferential Line**.

4 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

5 In the **Circumferential Distribution of Pressure (Angular Speed)** toolbar, click **Plot**.

Lift Force

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Lift Force` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 Right-click **Lift Force** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Hydrodynamic Bearing** > **Fluid loads** > **Fluid load on collar - N** > **hdb.htb1.Fcz - Fluid load on collar, z-component**.

3 Click to expand the **Legends** section. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`hdb.htb1.Fcz`	`kN`	`Fluid load on collar, z-component`

4 Locate the **Legends** section. From the **Legends** list, choose **Evaluated**.

5 In the **Legend** text field, type `h = eval(h_film, um) \mu m`.

6 In the **Lift Force** toolbar, click **Plot**.

Lift Force

1 In the **Model Builder** window, click **Lift Force**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Angular speed of the shaft (rad/s)`.

6 In the **Lift Force** toolbar, click **Plot**.