Model created in COMSOL Multiphysics 6.4

Elastohydrodynamic Lubrication in a Cylindrical Journal Bearing

Introduction

It is commonly known that elastic deformations can be significant in heavily loaded bearings. Under these conditions, it is important to include the elastic effect of the lubricated surfaces. Disregarding this effect can potentially lead to incorrect estimations of the fluid pressure and underestimation of the film thickness. As a consequence, a suboptimal design might be the result of using the simpler hydrodynamic approach.

To show how important the effect of elastic deformations can be when simulating heavily loaded bearings, this model compares the results obtained considering the elastic behavior of the components with those obtained with a standard hydrodynamic analysis.

Model Definition

The model geometry is visualized in Figure 1. The geometry consists of a journal and the surrounding bearing. The bearing is equipped with two grooves intended for lubrication supply. Note that symmetry conditions will be used, which means that only half of the geometry will be used in the simulations.

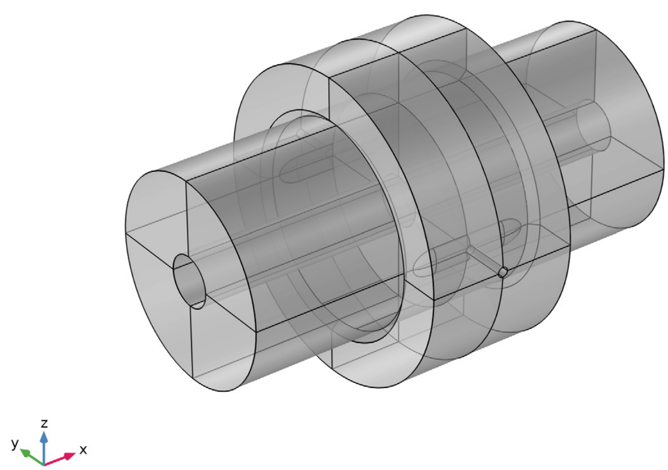


Figure 1: Model geometry of the full journal bearing assembly.

The shaft as well as the bearing are assumed to be made of aluminum. The lubricant used for the simulations will be a generic mineral oil.

Figure 2 shows half of the journal and lubricated surface. A constant pressure will be imposed the groove regions. These regions are therefore not considered as a part of the lubricated surface. The journal has a radius of 10 cm, and a width of 6.7 cm. The cylindrical bearing has a uniform initial clearance of 200 μm. The speed of the journal will be simulated in the interval $\Omega \in [100, 10000]$ rpm, while being loaded with a total force of 20 kN.

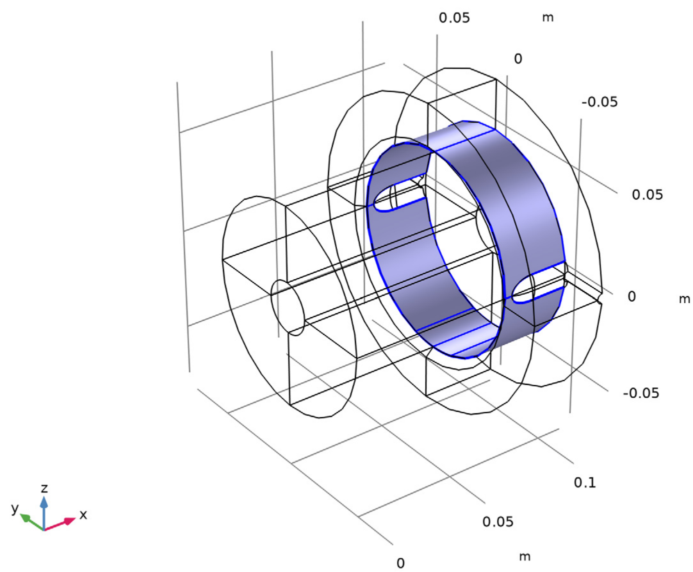


Figure 2: Lubricated surface of the journal.

In many cases, a liquid lubricant can be assumed to possess pressure-independent fluid properties. However, in these heavily loaded conditions, the extreme pressure encountered will cause the lubricant to compress. As a consequence, the density and viscosity will increase. It is therefore of great importance that the density–pressure and viscosity–pressure relations are correctly described. In this model, the viscosity of the lubricant is assumed to follow the isothermal relation suggested by Barus (Ref. 1):

$$\mu = \mu_0 e^{\xi p}$$

where μ_0 is the viscosity at zero pressure, ξ is a lubricant dependent pressure-viscosity coefficient, and p is the pressure in the fluid.

In addition, the isothermal density–pressure relation proposed by Dowson and Higginson (Ref. 1) will be used:

$$\rho = \rho_0\left(1 + \frac{0.6p}{1.7p + 10^9\,\mathrm{Pa}}\right)$$

where ρ_0 is the density at zero pressure. The assumed parameters used in model is summarized in Table 1.

TABLE 1: FLUID PARAMETER

Parameter	Value	Unit
μ_0	75	mPa·s
ξ	25	GPa^{-1}
ρ_0	850	kg/m^3

In this analysis, the outermost bearing surface is assumed to be fixed, while the fluid loads are applied to the lubricated surfaces as shown in Figure 3, while a constant pressure of 2 bar is assigned to the groove regions.

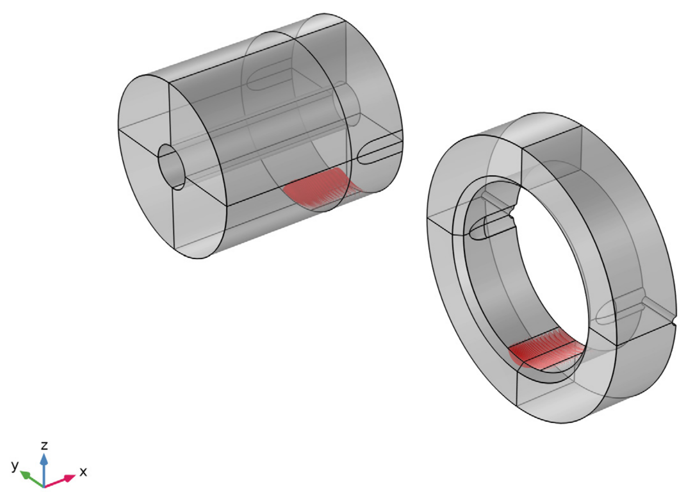


Figure 3: Load distribution from the fluid film on journal and bearing surfaces.

Results and Discussion

Figure 4 shows the physical pressure in the journal bearing at 100 rpm. Under these heavily loaded condition, the journal will be located close to the bushing in the very bottom of the bearing. This causes the pressure to increase dramatically in this region. This is also indicated in the figure, which predicts the pressure to be well above 160 bar.

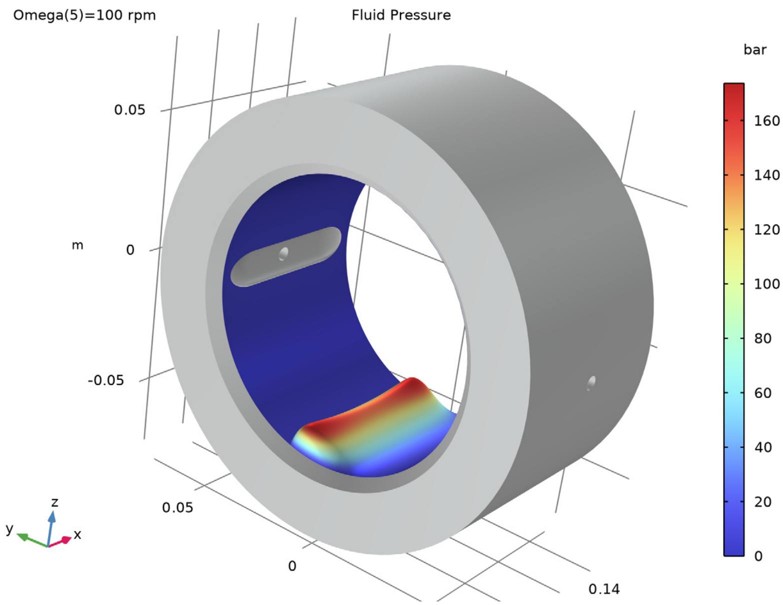


Figure 4: Physical fluid pressure at Ω = 100 rpm.

In Figure 5, you will find a visualization of the displacements normalized with respect to the initial clearance C. The plot depicts the conditions at a speed of 100 rpm. The deformations in the lower part of the bearing is more than 3 µm (approximately 0.15% of the initial clearance). What dominates the displacement field of the journal is naturally the translations toward the bottom of the bearing. However, it is clear that the displacements are smaller in the lower part the journal, which suggests that in that location the greatest deformations are expected.

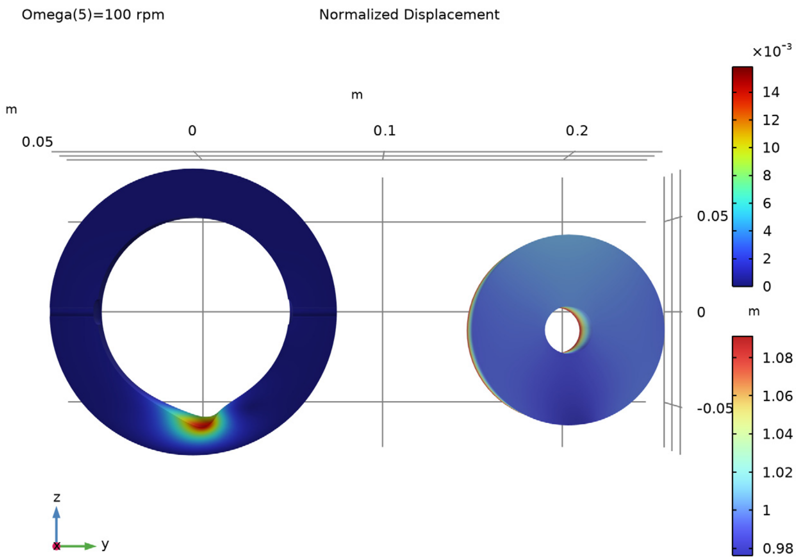


Figure 5: Displacements of journal and bearing at Ω = 100 rpm, normalized with respect to clearance.

The von Mises stress is depicted in Figure 6. The maximum stress in the bearing appears beneath the surface chose to location with increased fluid pressure. This is commonly known to be the case in, for example, Hertzian contact theory, which shares many similarities with elastohydrodynamic problems. The stress field in the journal, on the other hand, is a combination of the same phenomenon and stress due to bending.

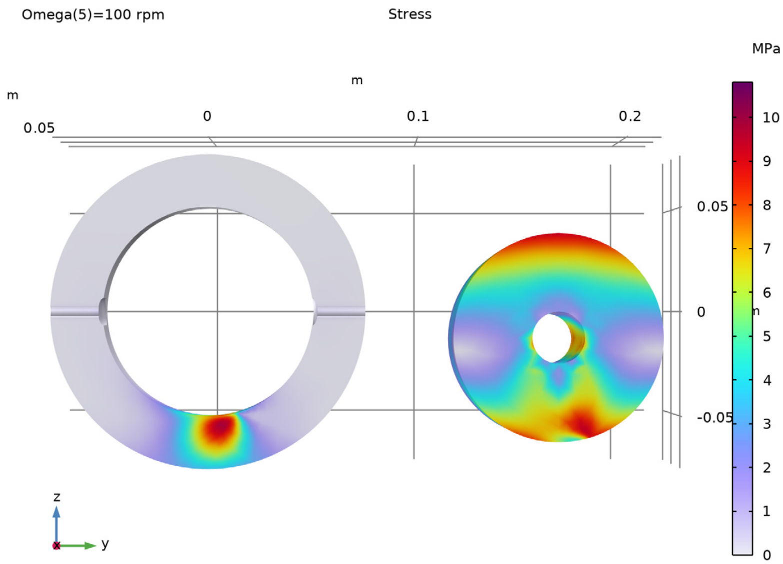


Figure 6: von Mises stress in journal and bearing at Ω = *100 rpm.*

Figure 7 shows a comparison between the physical fluid pressure in the circumferential direction along the center of the bearing. The comparison is shown for five different rotational speeds in the investigated interval. At higher speeds, more oil is forced through the converging zone in the bearing, which leads to more distributed pressure profiles with a lower maximum pressure. In many cases, the more moderate pressure encountered at this high speed is replicated to an acceptable agreement by the simplified hydrodynamic simulation. However, it is evident that as the speed is reduced, the disagreement between the pressure estimates increases. This shows how important it is to include the effect of deformations when these become significant.

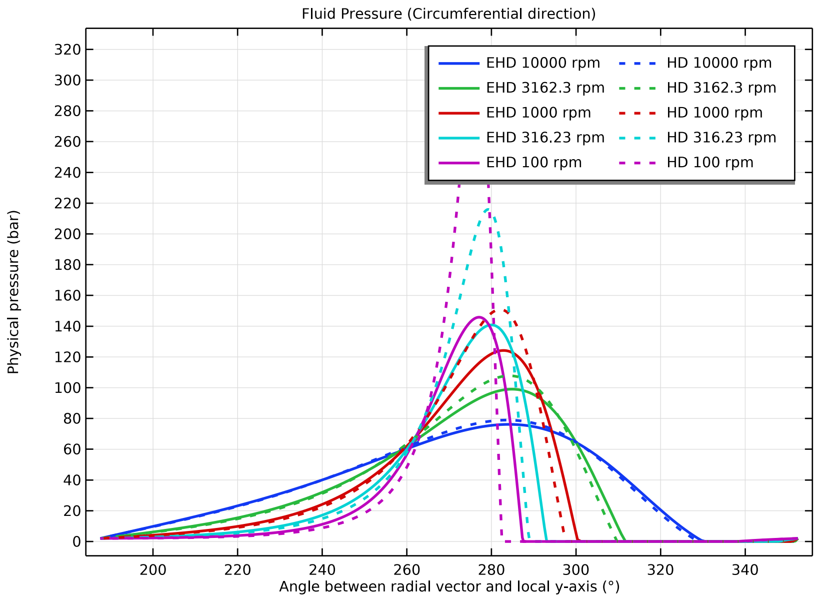


Figure 7: Comparison between physical pressure at bearing center along the circumferential direction.

The film thickness is visualized in Figure 8. It compares the results obtained with the two different modeling strategies for five different speeds in the simulated interval. The minimal clearance in the bearing reduces with decreasing speeds. Note that as the journal approaches the bushing, and the pressure increases dramatically, the film thickness flattens out due to the elastic deformations, while a small bump starts to form at the exit of the closure.

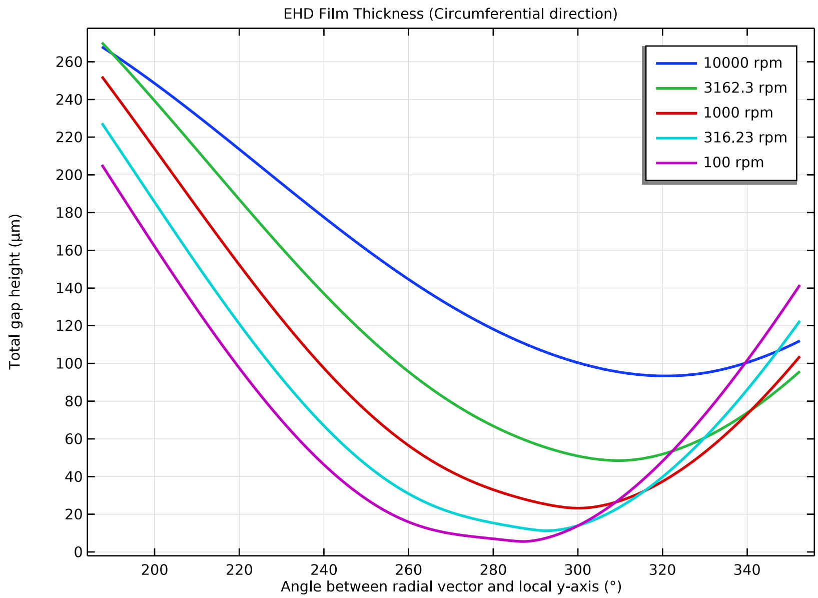


Figure 8: Comparison between film thickness at bearing center along the circumferential direction.

In Figure 9, an unwrapped plot of the physical fluid pressure is compared at 100 rpm. The maximum pressure computed by the hydrodynamic simulation is nearly twice as high as what is predicted in the elastohydrodynamic simulation. In addition, the shape of the pressure distribution is clearly different.

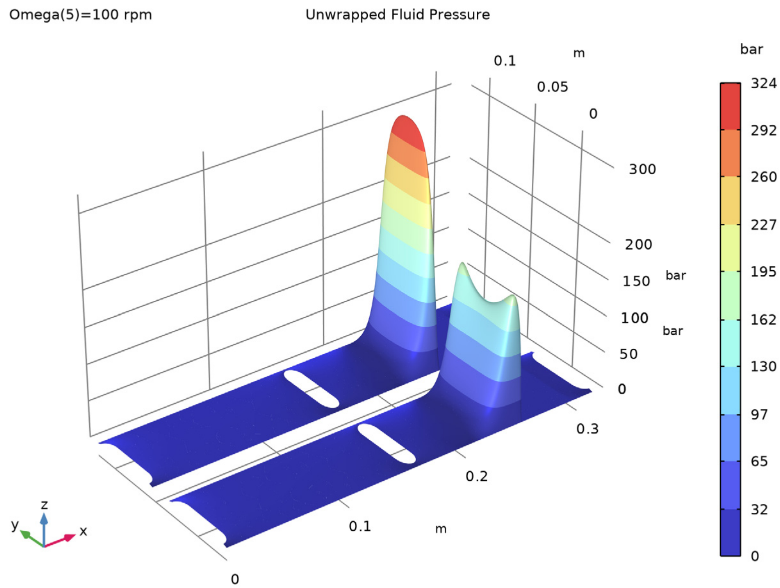


Figure 9: Comparison of unwrapped fluid pressure at Ω = *100 rpm.*

Notes About the COMSOL Implementation

- Use the **Solid Mechanics** interface along with the **Hydrodynamic Bearing** interface to set up the elastohydrodynamic model.
- This model shows how to set up a manual coupling between the **Solid Mechanics** and **Hydrodynamic Bearing** interfaces. As an alternative approach, the built-in multiphysics coupling **Solid-Bearing Coupling** automatically couples the **Solid Mechanics** and **Hydrodynamic Bearing** interface. Note that the built-in coupling uses an approximated displacement field for the journal and bearing displacements, which makes it incapable to represent local deformations which are important for this model.

Reference

1. B.J. Hamrock, S.R. Schmid, and B.O. Jacobson, *Fundamentals of Fluid Film Lubrications*, Marcel Dekker, 2004.

Application Library path: `Rotordynamics_Module/Tutorials/elastohydrodynamic_journal_bearing`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

The first step to build the model is to add the required physics interfaces and study.

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics > Solid Mechanics (solid)** and **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

Next, define the parameters that are needed for setting up the model.

1 In the **Model Builder** window, 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
Omega	100[rpm]	1.6667 1/s	Rotational speed
C	200[um]	2E-4 m	Bearing clearance
W	20[kN]	20000 N	Bearing load
ps	2[bar]	2E5 Pa	Supply pressure

Name	Expression	Value	Description
mu0	75[mPa*s]	0.075 Pa·s	Dynamic viscosity at zero pressure
xi	2.5e-8[m^2/N]	2.5E-8 1/Pa	Viscosity-pressure coefficient
rho0	850[kg/m^3]	850 kg/m³	Density at zero pressure

ROOT

Now, define a set of analytic functions for the density-pressure and viscosity-pressure dependencies.

DEFINITIONS

Fluid Density

1 In the **Definitions** toolbar, click **Analytic**.
2 In the **Settings** window for **Analytic**, type `Fluid Density` in the **Label** text field.
3 In the **Function name** text field, type `rho`.
4 Locate the **Definition** section. In the **Expression** text field, type `rho0*(1+0.6*x/(1.7*x+1e9))`.
5 Locate the **Units** section. In the **Function** text field, type `kg/m^3`.
6 In the table, enter the following settings:

Argument	Unit
x	Pa

Fluid Viscosity

1 Right-click **Fluid Density** and choose **Duplicate**.
2 In the **Settings** window for **Analytic**, type `Fluid Viscosity` in the **Label** text field.
3 In the **Function name** text field, type `mu`.
4 Locate the **Definition** section. In the **Expression** text field, type `mu0*exp(xi*x)`.
5 Locate the **Units** section. In the **Function** text field, type `Pa*s`.

Import the geometry for this model. The file is prepared in the COMSOL Multiphysics geometry file format (`.mphbin`).

GEOMETRY 1

Import 1 (imp1)

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

2 In the **Settings** window for **Import**, locate the **Source** section.

3 Click **Browse**.

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

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

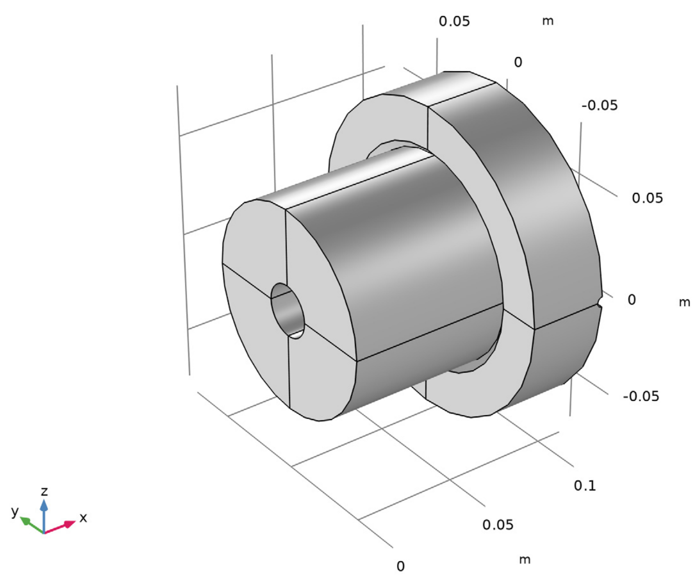
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Select the **Create imprints** checkbox.

5 In the **Geometry** toolbar, click **Build All**.

The geometry is now imported and appears in the **Graphics** window.



Next, define a set of selections. These will be useful when setting up the physics.

DEFINITIONS

Hydrodynamic Bearing (Journal)

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Hydrodynamic Bearing (Journal)` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.
5 Click **Paste Selection**.
6 In the **Paste Selection** dialog, type `14-19` in the **Selection** text field.
7 Click **OK**.

Hydrodynamic Bearing (Bearing)

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Hydrodynamic Bearing (Bearing)` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select the **Group by continuous tangent** checkbox.
5 Click **Paste Selection**.
6 In the **Paste Selection** dialog, type `41` in the **Selection** text field.
7 Click **OK**.

Fixed Boundary

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Fixed Boundary` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select the **Group by continuous tangent** checkbox.
5 Click **Paste Selection**.
6 In the **Paste Selection** dialog, type `30` in the **Selection** text field.
7 Click **OK**.

Boundary (Supply Pressure)

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Boundary (Supply Pressure)` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `20-23, 47-58` in the **Selection** text field.

6 Click **OK**.

Symmetry Plane

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Symmetry Plane` in the **Label** text field.

3 In the **Graphics** window toolbar, click ▼ next to **View Unhidden**, then choose **View All**.

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

5 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

6 Click **Paste Selection**.

7 In the **Paste Selection** dialog, type `24-27, 59-62` in the **Selection** text field.

8 Click **OK**.

Rigid Connector Edges

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Rigid Connector Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select the **Group by continuous tangent** checkbox.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `5` in the **Selection** text field.

7 Click **OK**.

Bearing Load

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Bearing Load` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** checkbox.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `1` in the **Selection** text field.

7 Click **OK**.

Inlet Edges

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Inlet Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `33 34 36 37 39-42` in the **Selection** text field.

6 Click **OK**.

COMSOL Multiphysics is equipped with built-in material properties for a range of different materials. In this model, you will use the aluminum for the structural parts, whereas a user-defined material will be used for the lubricant.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Aluminum**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Oil

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

2 In the **Settings** window for **Material**, type `Oil` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Hydrodynamic Bearing (Journal)**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	`mu(hdb.p)`	Pa·s	Basic
Density	rho	`rho(hdb.p)`	kg/m³	Basic

The next step is to set up the **Solid Mechanics** interface. The outermost surface of the bearing is assumed to be fixed, a symmetry condition is used at the symmetry plane, and a **Rigid Connector** is applied to prevent the journal from rotating around the x-axis. The supply pressure is applied to the groove regions and feeding channels, and the fluid forces will be applied to the lubricated surfaces. Lastly, mapping operators provided by the

Identity Boundary Pair are used to apply the fluid forces from the **Hydrodynamic Bearing** to the bearing modeled using the **Solid Mechanics** interface.

SOLID MECHANICS (SOLID)

Continuity 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Continuity 1**.
2 In the **Settings** window for **Continuity**, locate the **Advanced** section.
3 Select the **Disconnect pair** checkbox.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Fixed Boundary**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Symmetry Plane**.

Rigid Connector 1

1 In the **Physics** toolbar, click **Edges** and choose **Rigid Connector**.
2 In the **Settings** window for **Rigid Connector**, locate the **Edge Selection** section.
3 From the **Selection** list, choose **Rigid Connector Edges**.
4 Locate the **Prescribed Rotation** section. From the **By** list, choose **Constrained rotation**.
5 Select the **Constrain rotation around x-axis** checkbox.

Boundary Load (Supply Pressure)

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, type `Boundary Load (Supply Pressure)` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Boundary (Supply Pressure)**.
4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
5 In the p text field, type `ps`.

Boundary Load (Fluid Load, Journal)

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, type `Boundary Load (Fluid Load, Journal)` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hydrodynamic Bearing (Journal)**.
4 Locate the **Force** section. Specify the $\mathbf{f}_A$ vector as

hdb.fjx	x
hdb.fjy	y
hdb.fjz	z

Boundary Load (Fluid Load, Bearing)

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, type `Boundary Load (Fluid Load, Bearing)` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hydrodynamic Bearing (Bearing)**.
4 Locate the **Force** section. Specify the $\mathbf{f}_A$ vector as

src2dst_ap1(hdb.fbx)	x
src2dst_ap1(hdb.fby)	y
src2dst_ap1(hdb.fbz)	z

Boundary Load (Bearing Load)

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, type `Boundary Load (Bearing Load)` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Bearing Load**.
4 Locate the **Force** section. From the **Load type** list, choose **Total force**.
5 Specify the $\mathbf{F}_{tot}$ vector as

-W/2	z

SOLID MECHANICS (SOLID)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics (solid)** node.

HYDRODYNAMIC BEARING (HDB)

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

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

3 From the **Selection** list, choose **Hydrodynamic Bearing (Journal)**.

Now, set up the **Hydrodynamic Bearing** interface. Here a symmetry condition is applied, and the supply pressure is assigned to the inlet edges. In addition, the displacements are used to describe the journal and foundation movements. The mapping operators are used to map the displacement field from the bearing.

Hydrodynamic Journal Bearing 1

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

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Bearing Properties** section.

3 In the C text field, type `C`.

4 Specify the vector as

`0.1125[m]`	x

5 Locate the **Journal Properties** section. From the $\mathbf{u}_\mathrm{j}$ list, choose **Displacement field (solid)**.

6 From the **Velocity of the journal** list, choose **Revolutions per time**.

7 In the f_j text field, type `Omega`.

Moving Foundation 1

1 In the **Model Builder** window, click **Moving Foundation 1**.

2 In the **Settings** window for **Moving Foundation**, locate the **Foundation Motion** section.

3 Specify the $\mathbf{u}_\mathrm{f}$ vector as

`dst2src_ap1(u)`	x
`dst2src_ap1(v)`	y
`dst2src_ap1(w)`	z

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Click the **View Unhidden** button in the **Graphics** toolbar.

3 Select Edges 46–48, 52, 56, and 58 only.

Inlet 1

1. In the **Physics** toolbar, click **Edges** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Inlet Edges**.
4. Locate the **Inlet Settings** section. From the **Inlet condition** list, choose **Pressure**.
5. In the p_{film0} text field, type `ps`.

Now, follow the instructions below to generate a mesh which is significantly refined in the lower part of the bearing. This is desirable, since the pressure gradients are expected to be greatest in this region.

HYDRODYNAMIC BEARING (HDB)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Hydrodynamic Bearing (hdb)** node.

MESH 1

Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 14 and 19 only.
5. Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Identical Mesh 1

1. In the **Mesh** toolbar, click **More Attributes** and choose **Identical Mesh**.
2. In the **Settings** window for **Identical Mesh**, locate the **First Entity Group** section.
3. From the **Selection** list, choose **Hydrodynamic Bearing (Journal)**.
4. Locate the **Second Entity Group** section. From the **Selection** list, choose **Hydrodynamic Bearing (Bearing)**.

Mapped 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.
2. Select Boundaries 16 and 17 only.

Distribution 1

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

2 Select Edges 48 and 52 only.

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

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

Distribution 2

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

2 Select Edges 25, 27, and 31 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 30.

6 In the **Element ratio** text field, type 20.

7 From the **Growth rate** list, choose **Exponential**.

Convert 1

1 In the **Mesh** toolbar, click **Modify** and choose **Convert**.

2 In the **Settings** window for **Convert**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

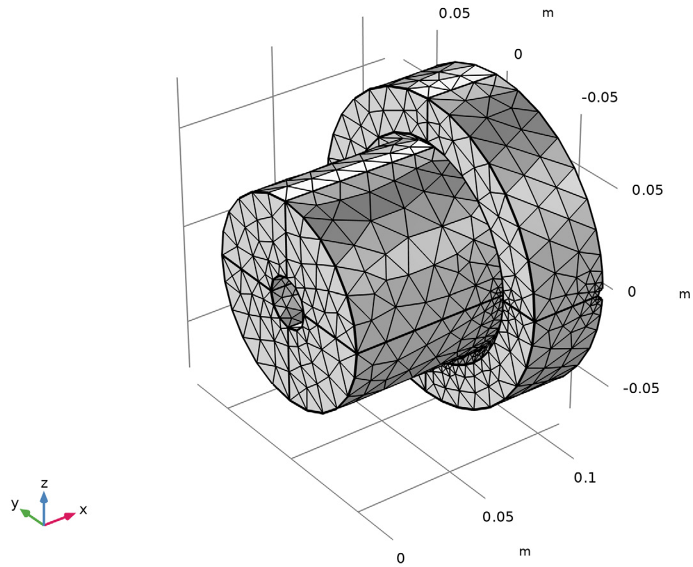
4 Select Boundaries 16 and 17 only.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

The mesh should look similar to the mesh shown below.



Now, set up the stationary study with an auxiliary sweep for the rotational speed.

STUDY 1

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

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** checkbox.

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
Omega (Rotational speed)	`10^range(4,-0.5,2)`	`rpm`

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** and choose **Fully Coupled**.
4 In the **Study** toolbar, click **Compute**.

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

RESULTS

Preferred Units 1

1 In the **Model Builder** window, expand the **Results** node.
2 Right-click **Results** and choose **Preferred Units**.
3 In the **Settings** window for **Preferred Units**, locate the **Units** section.
4 Click **Add Physical Quantity**.
5 In the **Physical Quantity** dialog, type `Pressure` in the text field.
6 In the tree, select **General** > **Pressure (Pa)**.
7 Click **OK**.
8 In the **Settings** window for **Preferred Units**, locate the **Units** section.
9 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Pressure	Pa	bar

10 Click **Add Physical Quantity**.
11 In the **Physical Quantity** dialog, type `Stress` in the text field.
12 In the tree, select **Solid Mechanics** > **Stress tensor (N/m^2)**.
13 Click **OK**.
14 In the **Settings** window for **Preferred Units**, locate the **Units** section.
15 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

16 Click **Add Physical Quantity**.
17 In the **Physical Quantity** dialog, type `angle` in the text field.

18 In the tree, select **General** > **Plane angle (rad)**.

19 Click **OK**.

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

21 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Plane angle	rad	°

22 Click **Apply**.

Next, use the Mirror 3D dataset to mirror the solution around the symmetry plane.

Mirror 3D 1

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

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 In the **X-coordinate** text field, type `0.1125[m]`.

Now, follow the instructions below to visualize the physical fluid pressure on the bearing surface.

Fluid Pressure

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

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

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

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

Pressure

1 Right-click **Fluid Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Pressure` in the **Label** text field.

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

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

Deformation 1

1 Right-click **Pressure** and choose **Deformation**.

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

3 In the **x-component** text field, type `-hdb.nrefx*hdb.p`.

4 In the **y-component** text field, type `-hdb.nrefx*hdb.p`.

5 In the **z-component** text field, type `-hdb.nrefz*hdb.p`.

Bearing

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

2 In the **Settings** window for **Surface**, type `Bearing` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `1`.

Material Appearance 1

Right-click **Bearing** and choose **Material Appearance**.

Selection 1

1 In the **Model Builder** window, right-click **Bearing** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `28-37, 39, 40, 47-58` in the **Selection** text field.

5 Click **OK**.

6 In the **Fluid Pressure** toolbar, click **Plot**.

Follow the instructions below to generate a plot of the normalized displacements.

Normalized Displacement

1 In the **Results** toolbar, click **3D Plot Group**.

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

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

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

5 Locate the **Color Legend** section. From the **Position** list, choose **Right double**.

6 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

7 From the **Array axis** list, choose **y**.

Bearing

1 Right-click **Normalized Displacement** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Bearing` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `solid.disp/C`.

Deformation 1

Right-click **Bearing** and choose **Deformation**.

Selection 1

1 In the **Model Builder** window, right-click **Bearing** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 Click **Clear Selection**.
4 From the **Geometric entity level** list, choose **Domain**.
5 Select Domain 2 only.
6 In the **Normalized Displacement** toolbar, click **Plot**.

Material Appearance 1

1 Right-click **Bearing** and choose **Material Appearance**.
2 In the **Settings** window for **Material Appearance**, locate the **Color** section.
3 Select the **Use the plot's color** checkbox.

Journal

1 Right-click **Bearing** and choose **Duplicate**.
2 In the **Settings** window for **Surface**, type `Journal` in the **Label** text field.
3 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Selection 1

1 In the **Model Builder** window, expand the **Journal** node, then click **Selection 1**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 Click **Clear Selection**.
4 Select Domain 1 only.
5 In the **Normalized Displacement** toolbar, click **Plot**.

Normalized Displacement

In the **Model Builder** window, collapse the **Results** > **Normalized Displacement** node.

The following instructions can be used to generate a plot of the von Mises stress in the journal and the bearing.

Stress

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Stress` in the **Label** text field.
3 Locate the **Title** section. From the **Title type** list, choose **Label**.
4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
5 Locate the **Color Legend** section. Select the **Show units** checkbox.

6 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

7 From the **Array axis** list, choose **y**.

8 In the **Relative padding** text field, type `0.1`.

Bearing

1 Right-click **Stress** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Bearing` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **solid.misesGp - von Mises stress - N/m²**.

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

Deformation 1

Right-click **Bearing** and choose **Deformation**.

Selection 1

1 In the **Model Builder** window, right-click **Bearing** and choose **Selection**.

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

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

Journal

1 Right-click **Bearing** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Journal` in the **Label** text field.

3 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Bearing**.

Selection 1

1 In the **Model Builder** window, expand the **Journal** node, then click **Selection 1**.

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

5 In the **Stress** toolbar, click **Plot**.

Next, follow the instructions listed below to set up a study without the effect of elasticity. This will be used to compare with the results obtained in the previous study.

HYDRODYNAMIC BEARING (HDB)

Hydrodynamic Journal Bearing 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Hydrodynamic Bearing (hdb)** right-click **Hydrodynamic Journal Bearing 1** and choose **Duplicate**.

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Journal Properties** section.

3 From the **Specify** list, choose **Load**.

4 Specify the $\mathbf{W}_j$ vector as

-W/2	z

Moving Foundation 1

1 In the **Model Builder** window, expand the **Hydrodynamic Journal Bearing 2** node, then click **Moving Foundation 1**.

2 In the **Settings** window for **Moving Foundation**, locate the **Foundation Motion** section.

3 Specify the $\mathbf{u}_f$ vector as

0	x
0	y
0	z

Now, disable the new **Hydrodynamic Journal Bearing** feature in the current study to allow for future re-runs.

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**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Hydrodynamic Bearing (hdb)** > **Hydrodynamic Journal Bearing 2**.

5 Click **Disable**.

STUDY 1

In the **Model Builder** window, collapse the **Study 1** node.

Add a new stationary study for the case without the effects of structural deformations.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** checkbox.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)**.

5 Click **Disable in Model**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Omega (Rotational speed)	`10^range(4,-0.5,2)`	`rpm`

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

Create a plot for comparing the physical pressure with and without the effect of elastic deformations.

RESULTS

Fluid Pressure (Circumferential direction)

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

2 In the **Settings** window for **1D Plot Group**, type `Fluid Pressure (Circumferential direction)` in the **Label** text field.

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

4 Locate the **Legend** section. In the **Number of columns** text field, type `2`.

5 In the **Maximum relative width** text field, type `1`.

EHD

1 Right-click **Fluid Pressure (Circumferential direction)** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, type `EHD` in the **Label** text field.
3 Select Edges 46, 48, 52, and 58 only.
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `hdb.p`.
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6 In the **Expression** text field, type `hdb.Th`.
7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.
8 Click to expand the **Legends** section. Select the **Show legends** checkbox.
9 Find the **Include** subsection. Select the **Label** checkbox.

HD

1 Right-click **EHD** and choose **Duplicate**.
2 In the **Settings** window for **Line Graph**, type `HD` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
5 From the **Color** list, choose **Cycle (reset)**.

Now, create a plot for visualizing the film thickness in the lower part of the bearing.

EHD Film Thickness (Circumferential direction)

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `EHD Film Thickness (Circumferential direction)` in the **Label** text field.
3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Line Graph 1

1 Right-click **EHD Film Thickness (Circumferential direction)** and choose **Line Graph**.
2 Select Edges 46, 48, 52, and 58 only.
3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `hdb.h`.
5 From the **Unit** list, choose **µm**.
6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
7 In the **Expression** text field, type `hdb.Th`.

8 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

9 Locate the **Legends** section. Select the **Show legends** checkbox.

10 In the **EHD Film Thickness (Circumferential direction)** toolbar, click **Plot**.

Use the result template **Unwrapped Fluid Pressure** to compare the fluid pressure with and without the effect of elastic deformations.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Hydrodynamic Bearing** > **Unwrapped Plots (hjb1)** > **Unwrapped Fluid Pressure (hjb1)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the tree, select **Study 2/Solution 2 (sol2)** > **Hydrodynamic Bearing** > **Unwrapped Plots (hjb2)** > **Unwrapped Fluid Pressure (hjb2)**.

6 Click the **Add Result Template** button in the window toolbar.

7 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Mirror 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

2 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

3 From the **Axis entry method** list, choose **Point and direction**.

4 Find the **Direction** subsection. In the **x** text field, type 1.

5 In the **y** text field, type 0.

Mirror 2D 3

1 Right-click **Mirror 2D 2** and choose **Duplicate**.

2 In the **Settings** window for **Mirror 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Surface (hjb2)**.

Unwrapped Fluid Pressure (hjb2)

In the **Model Builder** window, under **Results** right-click **Unwrapped Fluid Pressure (hjb2)** and choose **Delete**.

Unwrapped Fluid Pressure

1 In the **Model Builder** window, under **Results** click **Unwrapped Fluid Pressure (hjb1)**.

2. In the **Settings** window for **2D Plot Group**, type `Unwrapped Fluid Pressure` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 2**.
4. Click to expand the **Title** section. From the **Title type** list, choose **Label**.
5. Locate the **Color Legend** section. Select the **Show units** checkbox.
6. Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.
7. From the **Array axis** list, choose **y**.

EHD

1. In the **Model Builder** window, expand the **Unwrapped Fluid Pressure** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, type `EHD` in the **Label** text field.

HD

1. Right-click **EHD** and choose **Duplicate**.
2. In the **Settings** window for **Surface**, type `HD` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 3**.
4. Click to expand the **Inherit Style** section. From the **Plot** list, choose **EHD**.
5. In the **Unwrapped Fluid Pressure** toolbar, click **Plot**.