Shape Optimization of a Step Thrust Bearing

Introduction

This model is inspired by the Step Thrust Bearing model. In this example, shape optimization is applied to identify the optimal shape of grooves and pads as well as their optimal number.

Model Definition

The geometry and mesh are fixed, as is the norm for shape optimization. The out-of-plane geometry is defined in terms of a spatially varying bearing clearance; see Figure 1.

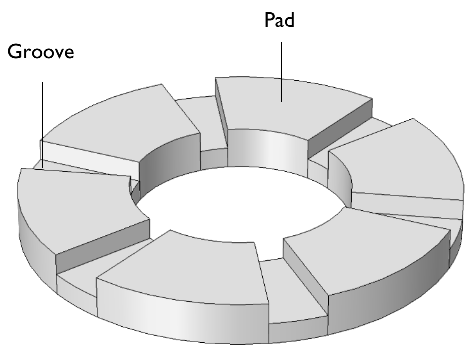


Figure 1: Initial step thrust bearing geometry.

Initially, a model of a classic six step bearing is set up. This constitutes a benchmark for the subsequent shape optimization. The shape is optimized using the **Polynomial Shell** feature.

The bearing load capacity is used as objective function. The amount of design freedom is determined by the order and maximum displacement of the **Polynomial Shell**. Too aggressive settings can, however, be expected to introduce problems with inverted elements.

Results and Discussion

Figure 2 shows the optimized bearing for the case of four grooves.

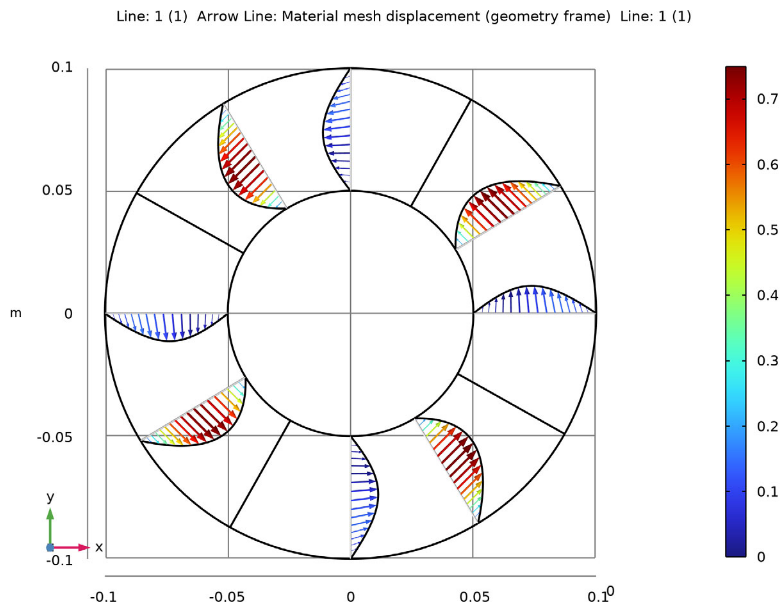


Figure 2: The edges of the optimized bearing is shown for the case of four grooves.

The shape optimization deforms the mesh, and this can impact the simulation accuracy, so it is good practice to remesh the geometry in the deformed configuration and recompute the result. Comparing the raw optimization results with such a verification simulation can reveal if the optimization relies on unphysical effects rooted in numerical errors, but this does not seem to be the case as illustrated in Figure 3.

Finally, one can identify the optimal number of pads by wrapping an optimization study with a **Parametric Sweep**, the result of which is shown in Figure 4.

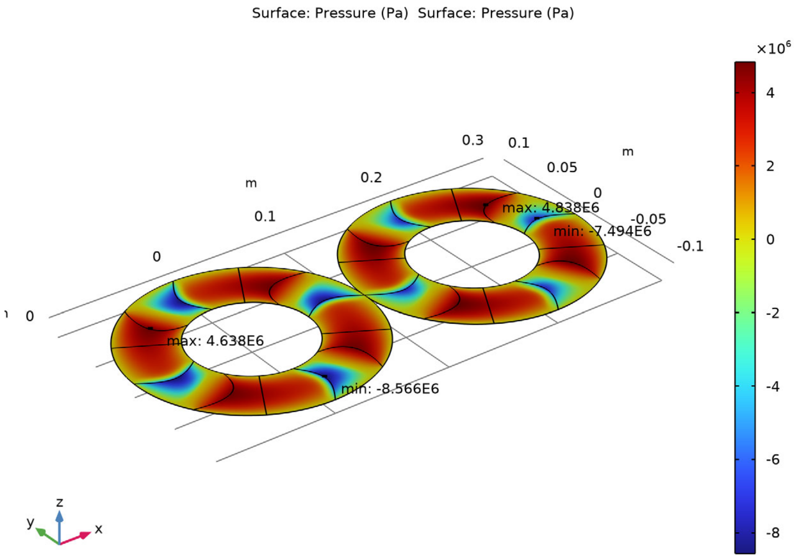


Figure 3: A comparison between the pressure distribution on the optimization mesh and on a mesh generated in the deformed configuration.

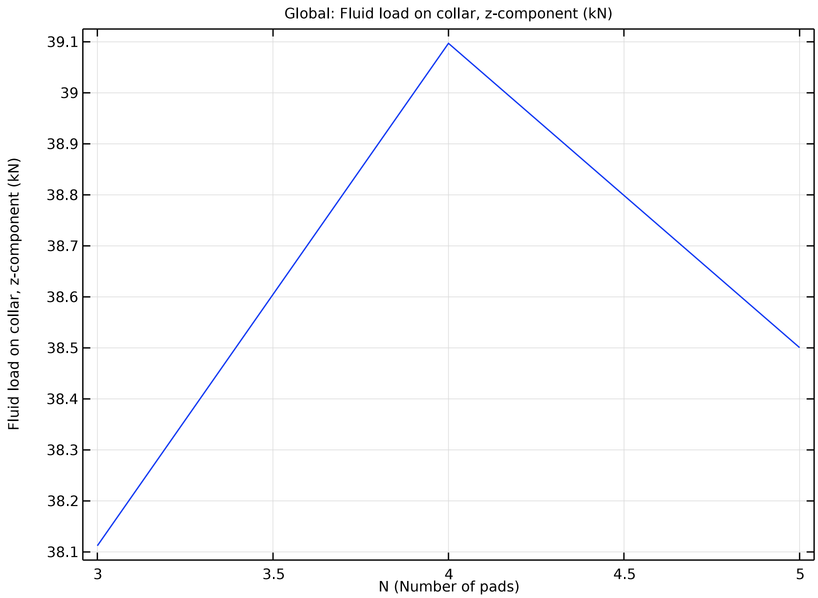


Figure 4: The optimized bearing load capacity is plotted versus the number of pads.

Application Library path: `Rotordynamics_Module/Optimization/step_thrust_bearing_shape_optimization`

Modeling Instructions

Start by setting up an analysis of a classic step thrust bearing.

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

Load a set of parameters used to define the geometry of the thrust bearing.

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_shape_optimization.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 In the **Sector angle** text field, type `gAng`.

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

Layer name	Thickness (m)
Layer 1	Ro-Ri

6 Right-click **Circle 1 (c1)** and choose **Duplicate**.

Work Plane 1 (wp1)>Circle 2 (c2)

1 In the **Model Builder** window, click **Circle 2 (c2)**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Sector angle** text field, type `padAng/2`.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `gAng`.

5 Right-click **Circle 2 (c2)** and choose **Duplicate**.

Work Plane 1 (wp1)>Circle 3 (c3)

1 In the **Model Builder** window, click **Circle 3 (c3)**.

2 In the **Settings** window for **Circle**, locate the **Rotation Angle** section.

3 In the **Rotation** text field, type `gAng+padAng/2`.

Domains to Delete

1 In the **Work Plane** toolbar, click **Selections** and choose **Disk Selection**.

2 In the **Settings** window for **Disk Selection**, type `Domains to Delete` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Ri*1.01`.

4 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

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

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Delete Entities**.

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

3 Click in the **Graphics** window and then press Ctrl+D to clear all objects.

4 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

6 From the **Selection** list, choose **Domains to Delete**.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 Click **Build Selected**.

Disk Selection: Leading Edge

1 In the **Work Plane** toolbar, click **Selections** and choose **Disk Selection**.

2 In the **Settings** window for **Disk Selection**, type `Disk Selection: Leading Edge` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.01*Ro`.

5 In the **Inner radius** text field, type `0.99*Ri`.

6 In the **Start angle** text field, type `gAng-1`.

7 In the **End angle** text field, type `gAng+1`.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

9 Click **Build Selected**.

10 Right-click **Disk Selection: Leading Edge** and choose **Duplicate**.

Work Plane 1 (wp1)>Disk Selection: Leading Edge 1 (disksel3)

1 In the **Model Builder** window, click **Disk Selection: Leading Edge 1 (disksel3)**.

2 In the **Settings** window for **Disk Selection**, locate the **Size and Shape** section.

3 In the **Start angle** text field, type `secAng-1`.

4 In the **End angle** text field, type `secAng+1`.

5 Click **Build Selected**.

6 In the **Label** text field, type `Disk Selection: Trailing Edge`.

Disk Selection: Groove

1 In the **Work Plane** toolbar, click **Selections** and choose **Disk Selection**.

2 In the **Settings** window for **Disk Selection**, locate the **Size and Shape** section.

3 In the **Outer radius** text field, type `1.01*Ro`.

4 In the **Inner radius** text field, type `0.99*Ri`.

5 In the **End angle** text field, type `gAng`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

7 Click **Build Selected**.

8 In the **Label** text field, type `Disk Selection: Groove`.

9 Right-click **Disk Selection: Groove** and choose **Duplicate**.

Work Plane 1 (wp1)>Disk Selection: Groove 1 (disksel5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** click **Disk Selection: Groove 1 (disksel5)**.

2 In the **Settings** window for **Disk Selection**, type `Disk Selection: Pad` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `gAng`.

4 In the **End angle** text field, type `360/N`.

5 Click **Build Selected**.

Work Plane 1 (wp1)>Rotate 1 (rot1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Rotate**.
2 In the **Settings** window for **Rotate**, locate the **Input** section.
3 From the **Input objects** list, choose **Delete Entities 1**.
4 Locate the **Rotation** section. In the **Angle** text field, type `range(0,secAng,360-secAng)`.
5 Click **Build Selected**.
6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Leading Edges of the Pads

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Selections>Union Selection**.
2 In the **Settings** window for **Union Selection**, type `Leading Edges of the Pads` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
4 Locate the **Input Entities** section. Click **Add**.
5 In the **Add** dialog box, select **Disk Selection: Leading Edge (Work Plane 1)** in the **Selections to add** list.
6 Click **OK**.
7 Right-click **Leading Edges of the Pads** and choose **Duplicate**.

Trailing Edges of the Pads

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Leading Edges of the Pads 1 (unisel2)**.
2 In the **Settings** window for **Union Selection**, type `Trailing Edges of the Pads` in the **Label** text field.
3 Locate the **Input Entities** section. Click **Build Preceding State**.
4 In the **Selections to add** list, select **Disk Selection: Leading Edge (Work Plane 1)**.
5 Click **Delete**.
6 Click **Add**.
7 In the **Add** dialog box, select **Disk Selection: Trailing Edge (Work Plane 1)** in the **Selections to add** list.
8 Click **OK**.
9 In the **Settings** window for **Union Selection**, click **Build Selected**.

Grooves

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Disk Selection: Groove (Work Plane 1)** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Grooves` in the **Label** text field.

8 Right-click **Grooves** and choose **Duplicate**.

Pads

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Grooves 1 (unisel4)**.

2 In the **Settings** window for **Union Selection**, type `Pads` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Build Preceding State**.

4 In the **Selections to add** list, select **Disk Selection: Groove (Work Plane 1)**.

5 Click **Delete**.

6 Click **Add**.

7 In the **Add** dialog box, select **Disk Selection: Pad (Work Plane 1)** in the **Selections to add** list.

8 Click **OK**.

9 In the **Settings** window for **Union Selection**, click **Build Selected**.

Groove Edges

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

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

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

4 Click **Add**.

5 In the **Add** dialog box, select **Grooves** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

8 From the **Geometric entity level** list, choose **Adjacent edges**.

9 In the **Label** text field, type `Groove Edges`.

Groove Inner Edges

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.

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

3 From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog box, select **Groove Edges** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, locate the **Size and Shape** section.

9 In the **Outer radius** text field, type `1.01*Ri`.

10 In the **Inner radius** text field, type `0.99*Ri`.

11 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

12 In the **Label** text field, type `Groove Inner Edges`.

Groove Edges (adjsel1), Groove Inner Edges (cylsel1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Groove Edges (adjsel1)** and **Groove Inner Edges (cylsel1)**.

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

Pad Edges

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Groove Edges 1 (adjsel2)**.

2 In the **Settings** window for **Adjacent Selection**, type `Pad Edges` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Build Preceding State**.

4 In the **Input selections** list, select **Grooves**.

5 Click **Delete**.

6 Click **Add**.

7 In the **Add** dialog box, select **Pads** in the **Input selections** list.

8 Click **OK**.

9 In the **Settings** window for **Adjacent Selection**, click **Build Selected**.

Pad Inner Edges

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Groove Inner Edges 1 (cylsel2)**.

2 In the **Settings** window for **Cylinder Selection**, type `Pad Inner Edges` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Selections** list, select **Groove Edges**.

4 Click **Delete**.

5 Click **Add**.

6 In the **Add** dialog box, select **Pad Edges** in the **Selections** list.

7 Click **OK**.

Control Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.

2 In the **Settings** window for **Cylinder Selection**, type `Control Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `INf`.

5 In the **Start angle** text field, type `-padAng*0.51`.

6 In the **End angle** text field, type `gAng+0.51*padAng`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

Sector Symmetry

1 In the **Geometry** toolbar, click **Selections** and choose **Complement Selection**.

2 In the **Settings** window for **Complement Selection**, type `Sector Symmetry` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Control Boundaries** in the **Selections to invert** list.

6 Click **OK**.

Fixed Edges

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Fixed Edges` in the **Label** text field.

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

4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent edges**.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog box, select **Control Boundaries** in the **Input selections** list.

7 Click **OK**.

DEFINITIONS

In the step bearing, the film thickness varies in steps with one value in the groove and another on the pad. Define a film thickness variable `hf` in two separate Variable nodes with complementary selections to specify different values in different regions.

Variables: Grooves

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

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

5 From the **Selection** list, choose **Grooves**.

6 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
hf	hg+h_film	m	

7 In the **Label** text field, type `Variables: Grooves`.

8 Right-click **Variables: Grooves** and choose **Duplicate**.

Variables: Pads

1 In the **Model Builder** window, click **Variables: Grooves 1**.

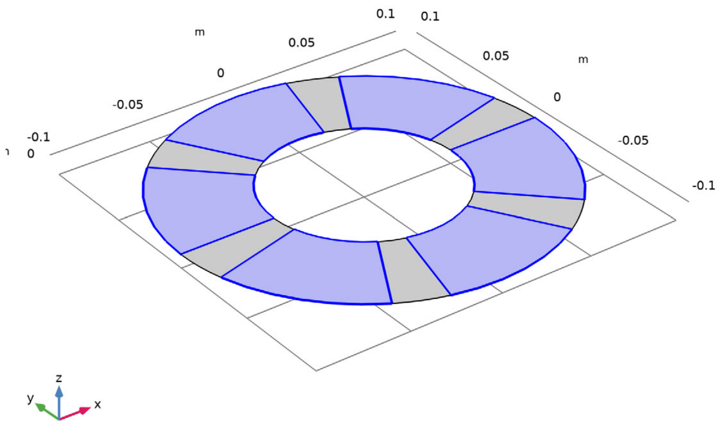
2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Pads**.

4 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
hf	h_film	m	

5 In the **Label** text field, type `Variables: Pads`.



HYDRODYNAMIC BEARING (HDB)

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**.

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 **User defined**.

6 In the h_{b1} text field, type `hf`.

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

8 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

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. In the **Settings** window for **Distribution**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Groove Inner Edges**.
4. Locate the **Distribution** section. In the **Number of elements** text field, type `round(gAng)`.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Pad Inner Edges**.
4. Locate the **Distribution** section. In the **Number of elements** text field, type `round(padAng/2)`.

Distribution 3

1. Right-click **Mapped 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Leading Edges of the Pads**.
4. Locate the **Distribution** section. 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** check box.
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 **Model Builder** window, click **Study 1**.

10 In the **Settings** window for **Study**, type `Study 1: Initial Design` in the **Label** text field.

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

A set of default plot are generated. These show the pressure distribution in the bearing. To generate a height plot of the pressure distribution, start by creating a **Surface dataset**.

RESULTS

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**.

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.

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. Click **Change Color Table**.

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

6 Click **OK**.

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** check box. In the associated text field, type `2e-8`.

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

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

Follow the instructions below to create visualize the mass fraction of the lubricant.

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.

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. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

7 Click **OK**.

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

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

Now, generate a plot which shows the bearing profile.

2D Plot Group 4

In the **Home** toolbar, click **Add Plot Group** and choose **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** check box. 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 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

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

Generate a plot of the pressure distributions along the radial and circumferential directions of the bearing. 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** check box.

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**.

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

Radial Distribution of Pressure (Angular Speed)

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

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 **Parameterized Curve** to create the circumferential sector line.

Parameterized Curve 3D: Circumferential Line

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

2 In the **Settings** window for **Parameterized 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 `Parameterized 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 **Model Builder** window, under **Results** click **Radial Distribution of Pressure (Film Thickness) 1**.

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

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

4 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 **Parameterized Curve 3D: Circumferential Line**.

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

Lift Force

1 In the **Home** toolbar, click **Add Plot Group** and choose **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** check box. In the associated text field, type `Angular speed of the shaft (rad/s)`.

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

GLOBAL DEFINITIONS

Parameters 1

Introduce a parameter for the maximum azimuthal deformation and increase the groove angle to allow larger mesh deformation.

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
N	4	4	Number of pads
gAng	125[deg]/N	0.54542 rad	Groove angle (deg)

Define some selections for later use.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

GEOMETRY 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

Work Plane 1 (wp1)

In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** node.

Circular Boundaries

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** node.

2 Right-click **Geometry 1** and choose **Selections>Adjacent Selection**.

3 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

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

5 Click **Add**.

6 In the **Add** dialog box, in the **Input selections** list, choose **Grooves** and **Pads**.

7 Click **OK**.

8 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

9 From the **Geometric entity level** list, choose **Adjacent edges**.

10 In the **Label** text field, type `Circular Boundaries`.

11 Click **Build Selected**.

Circular & pad edges

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

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

3 From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Leading Edges of the Pads**, **Trailing Edges of the Pads**, and **Circular Boundaries**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Circular & pad edges` in the **Label** text field.

8 In the **Model Builder** window, collapse the **Geometry 1** node.

HYDRODYNAMIC BEARING (HDB)

Manually set the location of the bearing center. This avoids high memory consumption when the compensation of `nojac` terms is enabled on the **Optimization Solver** node.

Hydrodynamic Thrust Bearing 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Hydrodynamic Bearing (hdb)** node, then click **Hydrodynamic Thrust Bearing 1**.
2 In the **Settings** window for **Hydrodynamic Thrust Bearing**, locate the **Bearing Properties** section.
3 From the $\mathbf{X}_c$ list, choose **User defined**.

DEFINITIONS

In the **Model Builder** window, collapse the **Definitions** node.

COMPONENT 1 (COMP1)

Free Shape Domain 1

In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization**.

The periodicity is not supported by the **Shape Optimization** interface, so the geometry is constructed such that the trailing and leading edges of the pads are internal to the selection of the **Polynomial Shell** feature. Then the exterior edges can be fixed and the deformation can be copied to the other sectors using the **Sector Symmetry** feature.

Polynomial Shell 1

1 In the **Shape Optimization** toolbar, click **Polynomial Shell**.
2 In the **Settings** window for **Polynomial Shell**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Control Boundaries**.
4 Locate the **Control Variable Settings** section. From the d_{max} list, choose **User defined**.
5 In the table, enter the following settings:

	Lock	Lower bound (m)	Upper bound (m)
X		-maxDisp	maxDisp
Y		-maxDisp	maxDisp
Z	√	-maxDisp	maxDisp

6 Locate the **Polynomial** section. In the n text field, type 3.

Sector Symmetry 1

1 In the **Shape Optimization** toolbar, click **Sector Symmetry**.
2 In the **Settings** window for **Sector Symmetry**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Sector Symmetry**.

5 Locate the **Sector** section. From the **Transformation** list, choose **Rotation**.

Free Shape Domain 1

In the **Model Builder** window, right-click **Free Shape Domain 1** and choose **Delete**.

Fixed Edge 1

1 In the **Shape Optimization** toolbar, click **Fixed Edge**.

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

3 From the **Selection** list, choose **Fixed Edges**.

HYDRODYNAMIC BEARING (HDB)

Add an optimization study to maximize the net lift force in the bearing.

ROOT

From the **Home** menu, choose **Add Study**.

ADD STUDY

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

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

3 Click **Add Study** in the window toolbar.

4 From the **Home** menu, choose **Add Study**.

STUDY 2

Shape Optimization

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

2 Right-click **Study 2** and choose **Optimization>Shape Optimization**.

3 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

4 In the **Maximum number of iterations** text field, type 15.

5 In the **Model Builder** window, click **Shape Optimization**.

6 Click **Replace Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Hydrodynamic Bearing>Fluid loads>Fluid load on collar (spatial and material frames) - N>comp1.hdb.htb1.Fcz - Fluid load on collar, z-component**.

7 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.

Scale the objective for better behavior of the optimization solver.

8 From the **Objective scaling** list, choose **Initial solution based**.

9 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

10 Select the **Plot** check box.

11 In the **Model Builder** window, click **Study 2**.

12 In the **Settings** window for **Study**, type `Study 2: Shape Optimization` in the **Label** text field.

Initialize the study to regenerate default plot for use while optimizing.

13 In the **Study** toolbar, click **Get Initial Value**.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2: Shape Optimization>Solver Configurations** node.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Optimization Solver 1**.

3 In the **Settings** window for **Optimization Solver**, click to expand the **Advanced** section.

4 Locate the **Optimization Solver** section. Clear the **Globally Convergent MMA** check box.

Switching off the compensation for nojac terms allows to reduce the memory consumption if you compute the bearing center from geometry.

5 Locate the **Advanced** section. From the **Compensate for nojac terms** list, choose **Off**.

6 In the **Model Builder** window, expand the **Study 2: Shape Optimization>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1>Stationary Solver 1>Segregated 1** node.

7 Right-click **Merged Variables** and choose **Move Down**.

8 In the **Model Builder** window, click **Merged Variables**.

9 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

10 From the **Nonlinear method** list, choose **Automatic (Newton)**.

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

Shape Optimization

1 In the **Model Builder** window, under **Study 2: Shape Optimization** click **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Pressure (Height)**.

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

RESULTS

Fluid Pressure, Shape Optimization (hdb)

In the **Settings** window for **3D Plot Group**, type `Fluid Pressure, Shape Optimization (hdb)` in the **Label** text field.

Delete the default shape optimization plot.

Shape Optimization

In the **Model Builder** window, right-click **Shape Optimization** and choose **Delete**.

Duplicate the **Pressure (Height)**, **Mass Fraction**, and **Pad Profile** plots from the previous study and change the settings to plot the shape optimized results.

Mass Fraction, Pad Profile, Pressure (Height)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Pressure (Height)**, **Mass Fraction**, and **Pad Profile**.

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Surface 1** and choose **Duplicate**.

Surface (Shape Optimization)

1 In the **Model Builder** window, click **Surface 2**.

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

3 From the **Dataset** list, choose **Study 2: Shape Optimization/Solution 2 (sol2)**.

4 In the **Label** text field, type `Surface (Shape Optimization)`.

Pressure, Shape Optimization (Height)

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

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

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

4 In the **Label** text field, type `Pressure, Shape Optimization (Height)`.

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

Mass Fraction, Shape Optimization

1 In the **Model Builder** window, click **Mass Fraction 1**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2: Shape Optimization/Solution 2 (sol2)**.
4 In the **Label** text field, type `Mass Fraction, Shape Optimization`.
5 In the **Mass Fraction, Shape Optimization** toolbar, click **Plot**.

Pad Profile, Shape Optimization

1 In the **Model Builder** window, click **Pad Profile 1**.
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Surface (Shape Optimization)**.
4 In the **Label** text field, type `Pad Profile, Shape Optimization`.
5 In the **Pad Profile, Shape Optimization** toolbar, click **Plot**.

Similarly, duplicate the radial and circumferential distribution of the pressure plots and change the settings for the shape optimization study to generate the plots. Note that this will also require duplicating the corresponding datasets.

Circumferential Distribution of Pressure (Film Thickness), 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 **Circumferential Distribution of Pressure (Film Thickness)**.
2 Right-click and choose **Duplicate**.

Cut Line 3D: Radial Line, Parameterized Curve 3D: Circumferential Line

1 In the **Model Builder** window, under **Results>Datasets**, Ctrl-click to select **Cut Line 3D: Radial Line** and **Parameterized Curve 3D: Circumferential Line**.
2 Right-click and choose **Duplicate**.

Cut Line 3D: Radial line (Optimization)

1 In the **Model Builder** window, under **Results>Datasets**, Ctrl-click to select **Cut Line 3D: Radial Line 1** and **Parameterized Curve 3D: Circumferential Line 1**.
2 In the **Settings** window for **Cut Line 3D**, type `Cut Line 3D: Radial line (Optimization)` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Shape Optimization/ Solution 2 (sol2)**.

Parameterized Curve 3D: Circumferential line (Optimization)

1 In the **Model Builder** window, under **Results>Datasets** click **Parameterized Curve 3D: Circumferential Line 1**.

2 In the **Settings** window for **Parameterized Curve 3D**, type `Parameterized Curve 3D: Circumferential line (Optimization)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Shape Optimization/ Solution 2 (sol2)**.

Radial Distribution of Pressure (Shape Optimization)

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

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

3 From the **Dataset** list, choose **Cut Line 3D: Radial line (Optimization)**.

4 In the **Label** text field, type `Radial Distribution of Pressure (Shape Optimization)`.

5 In the **Radial Distribution of Pressure (Shape Optimization)** toolbar, click **Plot**.

6 Click the **Show Legends** button in the **Graphics** toolbar.

Circumferential Distribution of Pressure (Shape Optimization)

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

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

3 From the **Dataset** list, choose **Parameterized Curve 3D: Circumferential line (Optimization)**.

4 In the **Label** text field, type `Circumferential Distribution of Pressure (Shape Optimization)`.

5 In the **Circumferential Distribution of Pressure (Shape Optimization)** toolbar, click **Plot**.

6 Click the **Show Legends** button in the **Graphics** toolbar.

Use the following instructions to plot the deformed mesh shape after the optimization.

Mesh

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

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

3 From the **Dataset** list, choose **Study 2: Shape Optimization/Solution 2 (sol2)**.

4 In the **Label** text field, type `Mesh`.

Mesh 1

1 Right-click **Mesh** and choose **Mesh**.

2 In the **Mesh** toolbar, click **Plot**.

You can highlight the optimized pad shape and change from the original shape by following the instructions below.

Shape Optimization

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Shape Optimization/ Solution 2 (sol2)**.

Line 1

1 Right-click **Shape Optimization** and choose **Line**.

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

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

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

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

Deformation 1

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

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

3 In the **X-component** text field, type `-material.dX`.

4 In the **Y-component** text field, type `-material.dY`.

5 In the **Z-component** text field, type `0`.

6 Locate the **Scale** section.

7 Select the **Scale factor** check box. In the associated text field, type `1`.

8 In the **Shape Optimization** toolbar, click **Plot**.

Arrow Line 1

1 In the **Model Builder** window, right-click **Shape Optimization** and choose **Arrow Line**.

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

3 In the **X-component** text field, type `material.dX`.

4 In the **Y-component** text field, type `material.dY`.

5 In the **Z-component** text field, type `material.dZ`.

6 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

7 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

8 Select the **Scale factor** check box.

Color Expression 1

1 Right-click **Arrow Line 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Polynomial Shell 1>pls1.rel_disp - Relative displacement - 1**.

3 Click to expand the **Range** section. Select the **Manual color range** check box.

4 In the **Minimum** text field, type `0`.

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

Surface 1

1 In the **Model Builder** window, right-click **Shape Optimization** and choose **Surface**.

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

3 In the **Expression** text field, type `hf`.

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

5 In the **Shape Optimization** toolbar, click **Plot**.

Shape Optimization

1 In the **Model Builder** window, click **Shape Optimization**.

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

3 From the **Position** list, choose **Right double**.

Circumferential Distribution of Pressure (Angular Speed), Circumferential Distribution of Pressure (Film Thickness), Fluid Pressure (hdb), Lift Force, Mass Fraction, Pad Profile, Pressure (Height), Radial Distribution of Pressure (Angular Speed), Radial Distribution of Pressure (Film Thickness)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Fluid Pressure (hdb)**, **Pressure (Height)**, **Mass Fraction**, **Pad Profile**, **Radial Distribution of Pressure (Film Thickness)**, **Radial Distribution of Pressure (Angular Speed)**, **Circumferential Distribution of Pressure (Film Thickness)**, **Circumferential Distribution of Pressure (Angular Speed)**, and **Lift Force**.

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

Initial Design

In the **Settings** window for **Group**, type `Initial Design` in the **Label** text field.

Circumferential Distribution of Pressure (Shape Optimization), Fluid Pressure, Shape Optimization (hdb), Mass Fraction, Shape Optimization, Mesh, Pad Profile, Shape Optimization, Pressure, Shape Optimization (Height), Radial Distribution of Pressure (Shape Optimization), Shape Optimization

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Fluid Pressure, Shape Optimization (hdb)**, **Pressure, Shape Optimization (Height)**, **Mass Fraction, Shape Optimization**, **Pad Profile, Shape Optimization**, **Radial Distribution of Pressure (Shape Optimization)**, **Circumferential Distribution of Pressure (Shape Optimization)**, **Mesh**, and **Shape Optimization**.
2 Right-click and choose **Group**.

Shape Optimization

In the **Settings** window for **Group**, type `Shape Optimization` in the **Label** text field.

Filter 1

The optimization study is now completed. Next, remesh the optimized configuration and recompute the pressure profile and compare it with the optimized results. If the number of pads and other geometry parameters were fixed, this could be achieved by remeshing in the deformed configuration, but we wish to vary the number of pads in a parametric sweep later on and therefore it is necessary to perform the verification in a separate component.

1 In the **Results** toolbar, click **More Datasets** and choose **Filter**.
2 In the **Settings** window for **Filter**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2: Shape Optimization/Solution 2 (sol2)**.
4 Locate the **Expression** section. In the **Expression** text field, type `1`.
5 Right-click **Filter 1** and choose **Create Mesh Part**.

MESH PART 1

Import 1

1 In the **Settings** window for **Import**, locate the **Import** section.
2 From the **Boundary partitioning** list, choose **Minimal**.
3 Click **Build All**.
4 In the **Model Builder** window, right-click **Mesh Part 1** and choose **Create Geometry**.

GEOMETRY 2

Import 1 (imp1)

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

2 Clear the **Simplify mesh** check box.

3 Clear the **Form solids from surface objects** check box.

4 Click **Build Selected**.

MATERIALS

Material 1 (mat1)

In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Material 1 (mat1)** and choose **Copy**.

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Paste Material**.

HYDRODYNAMIC BEARING (HDB)

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

COMPONENT 2 (COMP2)

In the **Model Builder** window, right-click **Component 2 (comp2)** and choose **Paste Hydrodynamic Bearing**.

HYDRODYNAMIC BEARING (HDB2)

In the **Messages from Paste** dialog box, click **OK**.

Initial Values 1

1 In the **Model Builder** window, expand the **Hydrodynamic Bearing (hdb2)** node, then 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]*hdb2.max(hdb2.hB1)/(0.1*hdb2.max(hdb2.hB1)+hdb2.hB1)`.

DEFINITIONS (COMP1)

Variables: Grooves, Variables: Pads

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions**, Ctrl-click to select **Variables: Grooves** and **Variables: Pads**.

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

DEFINITIONS (COMP2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Definitions** and choose **Paste Multiple Items**.

Variables: Grooves

1 In the **Model Builder** window, under **Component 2 (comp2)>Definitions** click **Variables: Grooves**.

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

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

4 From the **Selection** list, choose **Grooves (Import 1)**.

Variables: Pads

1 In the **Model Builder** window, click **Variables: Pads**.

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

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

4 From the **Selection** list, choose **Pads (Import 1)**.

MESH 2

Free Triangular 1

1 In the **Model Builder** window, expand the **Component 2 (comp2)> Hydrodynamic Bearing (hdb2)** node.

2 Right-click **Component 2 (comp2)>Mesh 2** and choose **More Generators>Free Triangular**.

3 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

4 From the **Geometric entity level** list, choose **Remaining**.

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `2*pi*Ro/360`.

4 In the **Minimum element size** text field, type `2*pi*Ro/720`.

5 Click **Build All**.

STUDY 2: SHAPE OPTIMIZATION

Step 1: Stationary

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

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

3 In the table, clear the **Solve for** check box for **Hydrodynamic Bearing (hdb2)**.

STUDY 1: INITIAL DESIGN

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

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

3 In the table, clear the **Solve for** check box for **Hydrodynamic Bearing (hdb2)**.

Add a new study for verification.

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Hydrodynamic Bearing (hdb)**.

5 Click **Add Study** in the window toolbar.

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

STUDY 3: REMESH AND VERIFY

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3: Remesh and Verify` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

Follow the instructions below to compare the results of the optimization study with the solution on the optimized configuration.

RESULTS

Verification

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

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

3 From the **Dataset** list, choose **Study 3: Remesh and Verify/Solution 3 (4) (sol3)**.

4 In the **Label** text field, type `Verification`.

Surface 1

Right-click **Verification** and choose **Surface**.

Marker 1

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

2 In the **Settings** window for **Marker**, locate the **Text Format** section.

3 In the **Display precision** text field, type `4`.

Surface 1

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

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 From the **Dataset** list, choose **Study 2: Shape Optimization/Solution 2 (sol2)**.

Translation 1

1 Right-click **Surface 2** and choose **Translation**.

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

3 In the **x** text field, type `0.2`.

4 In the **Verification** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Surface 1**.

4 In the **Verification** toolbar, click **Plot**.

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

ROOT

You can now optimize the pad shapes for different numbers of pads in the bearings. Add a new study with shape optimization and a parametric sweep over the number of pads.

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Hydrodynamic Bearing (hdb2)**.

5 Click **Add Study** in the window toolbar.

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

STUDY 4

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

2 From the **Probes** list, choose **None**.

Shape Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type 15.

4 Click **Replace Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Hydrodynamic Bearing>Fluid loads> Fluid load on collar (spatial and material frames) - N>comp1.hdb.htb1.Fcz - Fluid load on collar, z-component**.

5 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.

6 From the **Objective scaling** list, choose **Initial solution based**.

7 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
N (Number of pads)	3 4 5	

5 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

6 In the **Model Builder** window, click **Study 4**.

7 In the **Settings** window for **Study**, type `Study 4: Shape Optimization Sweep` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 4 (sol4)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Optimization Solver 1**.

3 In the **Settings** window for **Optimization Solver**, locate the **Optimization Solver** section.

4 Clear the **Globally Convergent MMA** check box.

5 Locate the **Advanced** section. From the **Compensate for nojac terms** list, choose **Off**.

6 In the **Model Builder** window, expand the **Study 4: Shape Optimization Sweep>Solver Configurations>Solution 4 (sol4)>Optimization Solver 1>Stationary Solver 1>Segregated 1** node.

7 Right-click **Study 4: Shape Optimization Sweep>Solver Configurations>Solution 4 (sol4)>Optimization Solver 1>Stationary Solver 1>Segregated 1>Merged Variables** and choose **Move Down**.

8 In the **Model Builder** window, under **Study 4: Shape Optimization Sweep>Solver Configurations>Solution 4 (sol4)>Optimization Solver 1>Stationary Solver 1>Segregated 1** click **Merged Variables**.

9 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

10 From the **Nonlinear method** list, choose **Automatic (Newton)**.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 4: Shape Optimization Sweep** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 1

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

RESULTS

Shape Optimization

In the **Model Builder** window, under **Results>Shape Optimization** right-click **Shape Optimization** and choose **Duplicate**.

Shape Optimization Sweep

1. In the **Model Builder** window, under **Results>Shape Optimization** click **Shape Optimization 1**.
2. In the **Settings** window for **3D Plot Group**, type `Shape Optimization Sweep` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 4: Shape Optimization Sweep/Parametric Solutions 1 (7) (sol5)**.
4. From the **Parameter value (N)** list, choose **3**.
5. In the **Shape Optimization Sweep** toolbar, click **Plot**.

Objective vs Number of Pads

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Objective vs Number of Pads` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 4: Shape Optimization Sweep/Parametric Solutions 1 (7) (sol5)**.

Global 1

1. Right-click **Objective vs Number of Pads** and choose **Global**.
2. In the **Settings** window for **Global**, click **Add 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 (spatial and material frames) - N> hdb.htb1.Fcz - Fluid load on collar, z-component**.
3. In the **Objective vs Number of Pads** toolbar, click **Plot**.
4. 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`

5. Click to expand the **Legends** section. Clear the **Show legends** check box.
6. In the **Objective vs Number of Pads** toolbar, click **Plot**.

Duplicate the **Shape Optimization** plot once again to generate the thumbnail for the model.

Shape Optimization

In the **Model Builder** window, under **Results>Shape Optimization** right-click **Shape Optimization** and choose **Duplicate**.

Thumbnail

1. In the **Model Builder** window, under **Results>Shape Optimization** click **Shape Optimization 1**.
2. In the **Settings** window for **3D Plot Group**, type `Thumbnail` in the **Label** text field.
3. Right-click **Results>Shape Optimization>Thumbnail** and choose **Move Out**.
4. Right-click **Results>Shape Optimization>Thumbnail** and choose **Move Down**.
5. Right-click **Results>Shape Optimization>Thumbnail** and choose **Move Down**.

Surface 1

1. In the **Model Builder** window, expand the **Thumbnail** node.
2. Right-click **Surface 1** and choose **Delete**.

Arrow Line 1

1. In the **Model Builder** window, under **Results>Thumbnail** click **Arrow Line 1**.
2. In the **Settings** window for **Arrow Line**, locate the **Arrow Positioning** section.
3. From the **Placement** list, choose **Uniform**.
4. In the **Number of arrows** text field, type `500`.

Color Expression 1

1. In the **Model Builder** window, expand the **Arrow Line 1** node, then click **Color Expression 1**.
2. In the **Settings** window for **Color Expression**, click to expand the **Range** section.
3. Clear the **Manual color range** check box.
4. In the **Thumbnail** toolbar, click **Plot**.

Line 1

1. In the **Model Builder** window, under **Results>Thumbnail** click **Line 1**.
2. In the **Settings** window for **Line**, locate the **Coloring and Style** section.
3. From the **Line type** list, choose **Tube**.
4. In the **Tube radius expression** text field, type `4e-4`.
5. Select the **Radius scale factor** check box.
6. Right-click **Results>Thumbnail>Line 1** and choose **Duplicate**.

Line 2

1 In the **Model Builder** window, click **Line 2**.

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

3 From the **Color** list, choose **Black**.

Deformation 1

1 In the **Model Builder** window, expand the **Line 2** node.

2 Right-click **Deformation 1** and choose **Delete**.

Thumbnail

1 Click the **Go to XY View** button in the **Graphics** toolbar.

2 Click the **Zoom In** button in the **Graphics** toolbar.