Created in COMSOL Multiphysics 6.1

Shape Optimization of a Step Thrust Bearing

Introduction

This model is based on the *Step Thrust Bearing* model in the Rotordynamics Module Application Library. In the present 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.

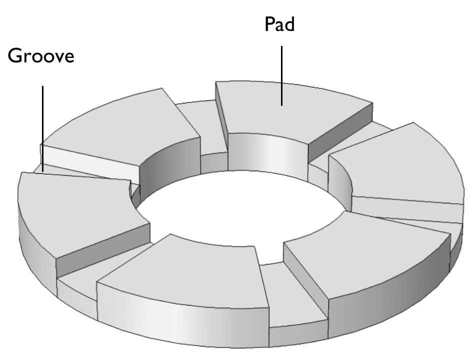

Figure 1: Initial step thrust bearing geometry.

The shape is optimized by deforming the geometry in the azimuthal direction. This deformation varies depending on the radial coordinate. It is implemented using the **Control Function** feature and the **Deformed Geometry** interface. Different functions are used for the leading and trailing edges of the pads, but all pads have the same shape. The **Coefficient Form Boundary PDE** and **Coefficient Form Edge PDE** interfaces are used to compute the displacement field for the interior mesh element of the pads and grooves.

The bearing load capacity is used as objective function. The amount of design freedom is determined by the order of the control functions and the maximum azimuthal deformation. 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 five grooves.

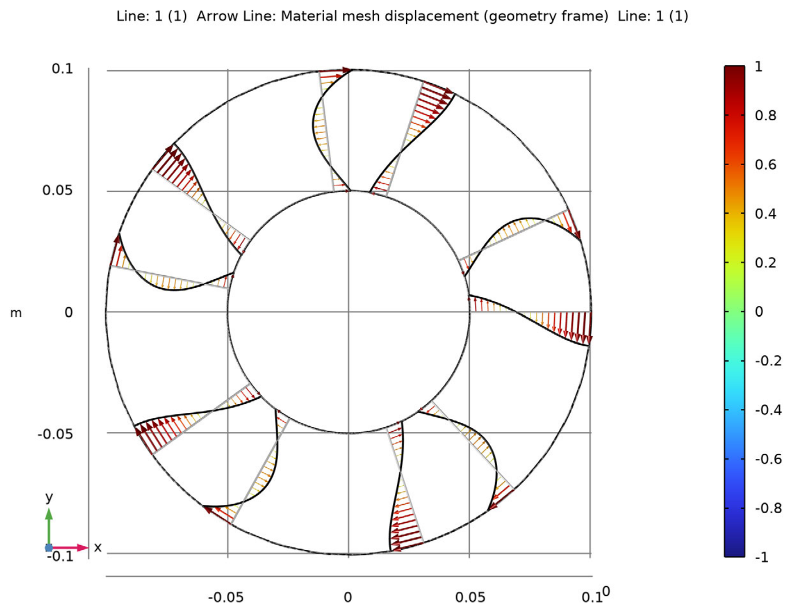

Figure 2: The edges of the optimized bearing is shown for the case of five grooves.

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 a optimization study with a **Parametric Sweep**, the result of which is shown in Figure 4.

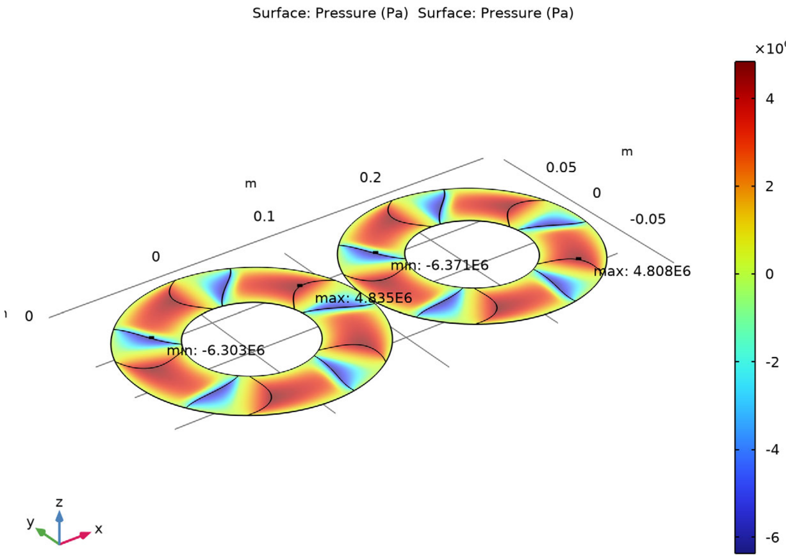

Figure 3: A comparison between the pressure distribution on the optimization mesh and on a mesh generated in the deformed configuration.

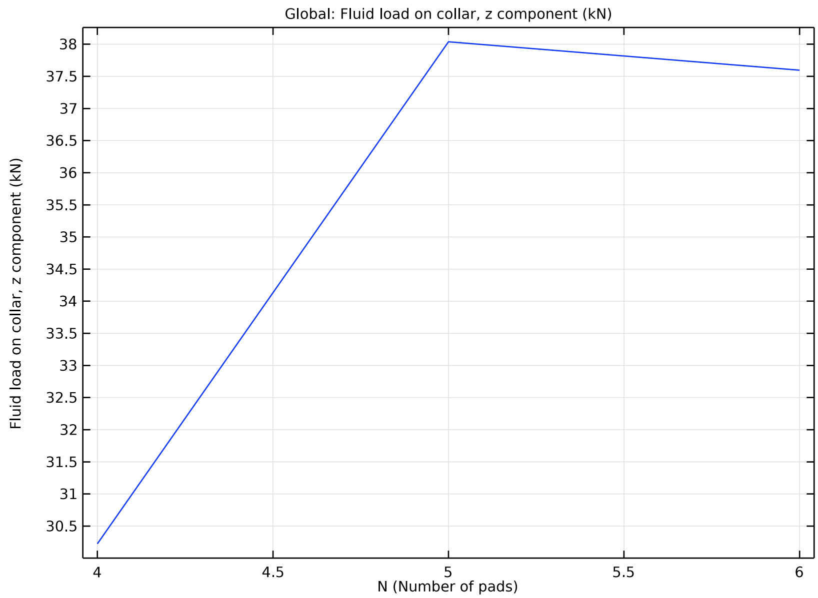

Figure 4: The optimized bearing load capacity is plotted versus the number of pads.

Application Library path: `Optimization_Module/Shape_Optimization/step_thrust_bearing_shape_optimization`

Modeling Instructions

This example starts from an existing model from the Rotordynamics Module Application Library.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Rotordynamics Module>Tutorials>step_thrust_bearing** in the tree.

3 Click **Open**.

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	5	5	Number of pads
gAng	25[deg]	0.43633 rad	Groove angle (deg)
maxAngle	8[deg]	0.13963 rad	Limit on pad profile angle

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`.

Trailing Edge Points

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 **Edge**.

4 Click **Add**.

5 In the **Add** dialog box, select **Trailing Edges of the Pads** 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 points**.

9 In the **Label** text field, type `Trailing Edge Points`.

Leading Edge Points

1 Right-click **Trailing Edge Points** and choose **Duplicate**.

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

3 Click **Build Preceding State**.

4 In the **Input selections** list, select **Trailing Edges of the Pads**.

5 Click **Delete**.

6 Click **Add**.

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

8 Click **OK**.

9 In the **Settings** window for **Adjacent Selection**, type `Leading Edge Points` in the **Label** text field.

10 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.

Add a couple of control functions to control the leading and trailing edge shapes of the pads.

COMPONENT 1 (COMP1)

Control Function 1 (cfunc1)

1 In the **Definitions** toolbar, click **Optimization** and choose **Control Variables>Control Function**.

2 In the **Settings** window for **Control Function**, locate the **Input** section.

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

4 Locate the **Output** section. In the f_{min} text field, type `-maxAngle`.

5 In the f_{max} text field, type `maxAngle`.

6 In the c_0 text field, type `maxAngle/10`.

It is possible to initialize the **Control Function** to zero, but then the machinery for automatic scaling of the coefficient PDEs fails, so one has to resort to manually scaling.

7 Locate the **Units** section. In the **Argument** text field, type `1`.

8 In the **Function** text field, type `rad`.

Control Function 2 (cfunc2)

Right-click **Control Function 1 (cfunc1)** and choose **Duplicate**.

DEFINITIONS (COMP1)

Define variables to specify the motions of the pad boundaries.

Variables: Trailing edge motion

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

2 Right-click **Component 1 (comp1)>Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, type `Variables: Trailing edge motion` in the **Label** text field.

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Edge**.

5 From the **Selection** list, choose **Trailing Edges of the Pads**.

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

Name	**Expression**	**Unit**	**Description**
`Re`	`sqrt(Xg^2+Yg^2)`	m	`Radial coordinate`
`phie`	`atan2(Yg, Xg)`	rad	`Circumferential coordinate`
`se`	`(Re-Ri)/(Ro-Ri)`		`Scaled radial coordinate`
`movexef`	`cos(phie+cfunc1(se))*Re-Xg`	m	`Displacement, x direction`
`moveyef`	`sin(phie+cfunc1(se))*Re-Yg`	m	`Displacement, y direction`

Variables: Leading edge motion

1 Right-click **Variables: Trailing edge motion** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Leading Edges of the Pads**.

4 In the **Label** text field, type `Variables: Leading edge motion`.

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

Name	**Expression**	**Unit**	**Description**
`movexef`	`cos(phie+cfunc2(se))*Re-Xg`	m	`Displacement, x direction`
`moveyef`	`sin(phie+cfunc2(se))*Re-Yg`	m	`Displacement, y direction`

Variables: Circular boundary motion

1 Right-click **Variables: Leading edge motion** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Circular Boundaries**.

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

Name	**Expression**	**Unit**	**Description**
`se`	`(Re-Ri)/(Ro-Ri)`		`Scaled radial coordinate`

Name	Expression	Unit	Description
movexef	cos(phie+phieMove)*Re-Xg		Displacement, x direction
moveyef	sin(phie+phieMove)*Re-Yg		Displacement, y direction

5 Click **Delete**.

6 In the **Label** text field, type `Variables: Circular boundary motion`.

Add boundary and edge PDEs to solve for the mesh deformation in the pad.

ADD PHYSICS

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

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

3 In the tree, select **Mathematics>PDE Interfaces>Lower Dimensions> Coefficient Form Boundary PDE (cb)** and **Mathematics>PDE Interfaces>Lower Dimensions> Coefficient Form Edge PDE (ce)**.

4 Click **Add to Component 1** in the window toolbar.

5 In the tree, select **Mathematics>Deformed Mesh>Deformed Geometry> Prescribed Deformation**.

6 Click **Add to Component 1** in the window toolbar.

7 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

COEFFICIENT FORM BOUNDARY PDE (CB)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Coefficient Form Boundary PDE (cb)**.

2 In the **Settings** window for **Coefficient Form Boundary PDE**, click to expand the **Dependent Variables** section.

3 In the **Field name** text field, type `movef`.

4 In the **Number of dependent variables** text field, type `2`.

5 In the **Dependent variables** table, enter the following settings:

movexf
moveyf

6 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

7 In the **Physical Quantity** dialog box, type `length` in the text field.

8 Click Filter.

9 In the tree, select **General>Length (m)**.

10 Click **OK**.

11 In the **Settings** window for **Coefficient Form Boundary PDE**, locate the **Units** section.

12 Click **Select Source Term Quantity**.

13 In the **Physical Quantity** dialog box, type `dimensionless` in the text field.

14 Click **Filter**.

15 In the tree, select **General>Dimensionless (1)**.

16 Click **OK**.

Coefficient Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Coefficient Form Boundary PDE (cb)** click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Source Term** section.

3 In the f text-field array, type `0` on the first row.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Edges** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Circular & pad edges**.

4 Locate the **Dirichlet Boundary Condition** section. In the r_1 text field, type `movexef`.

5 In the r_2 text field, type `moveyef`.

COEFFICIENT FORM EDGE PDE (CE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Coefficient Form Edge PDE (ce)**.

2 In the **Settings** window for **Coefficient Form Edge PDE**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Circular Boundaries**.

4 Click to expand the **Dependent Variables** section. In the **Field name** text field, type `phieMove`.

5 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

6 In the **Physical Quantity** dialog box, type `plane angle` in the text field.

7 Click **Filter**.

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

9 Click **OK**.

10 In the **Settings** window for **Coefficient Form Edge PDE**, locate the **Units** section.

11 Click **Select Source Term Quantity**.

12 In the **Physical Quantity** dialog box, click **OK**.

13 In the **Settings** window for **Coefficient Form Edge PDE**, locate the **Dependent Variables** section.

14 In the **Dependent variables** table, enter the following settings:

phieMove

Coefficient Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Coefficient Form Edge PDE (ce)** click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Source Term** section.

3 In the *f* text field, type `0`.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Points** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Trailing Edge Points**.

4 Locate the **Dirichlet Boundary Condition** section. In the *r* text field, type `cfunc1(se)`.

Dirichlet Boundary Condition 2

1 Right-click **Dirichlet Boundary Condition 1** and choose **Duplicate**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Leading Edge Points**.

4 Locate the **Dirichlet Boundary Condition** section. In the *r* text field, type `cfunc2(se)`.

Prescribe the mesh deformation in the **Prescribed Displacement** node in **Deformed Geometry**.

DEFORMED GEOMETRY

Prescribed Deformation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Deformed Geometry** node, then click **Prescribed Deformation 1**.

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

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

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

5 Locate the **Prescribed Deformation** section. Specify the *dx* vector as

`movexf`	X
`moveyf`	Y
`0`	Z

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: INITIAL DESIGN

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

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

STUDY 2

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

5 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.

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

Scale the objective for better behavior of the optimization solver.

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

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

9 Select the **Plot** check box.

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

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

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

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

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study: 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.

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

Shape Optimization

1 In the **Model Builder** window, under **Study: 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 plots for the pad motion.

3D Plot Group 11, 3D Plot Group 12

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **3D Plot Group 11** and **3D Plot Group 12**.
2 Right-click 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 (Shape Optimization)

1 In the **Model Builder** window, expand the **Results>Datasets** node.
2 Right-click **Results>Datasets>Surface 1** and choose **Duplicate**.
3 In the **Settings** window for **Surface**, locate the **Data** section.
4 From the **Dataset** list, choose **Study: Shape Optimization/Solution 2 (sol2)**.
5 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: 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: 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: 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: 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: 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**, locate the **Expression** section.

3 In the **Expression** text field, type `if(isdefined(se),abs(atan2(Y,X)-atan2(Yg, Xg))/maxAngle,nan)`.

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

5 In the **Minimum** text field, type `-1`.

6 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.

Study: Shape Optimization/Solution 2 (sol2)

The optimization study is now completed. Next, remesh the optimized configuration and recompute the pressure profile and compare it with the optimized results. Start by generating the mesh for the deformed configuration.

In the **Results** toolbar, click **Attributes** and choose **Remesh Deformed Configuration**.

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 boxes for **Coefficient Form Boundary PDE (cb)** and **Coefficient Form Edge PDE (ce)**.

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

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

STUDY: REMESH AND VERIFY

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

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

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

Step 1: Stationary

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

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

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

4 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study: Shape Optimization, Stationary**.

7 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: Remesh and Verify/Solution 3 (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 2

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

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

3 From the **Dataset** list, choose **Study: 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 Click **Add Study** in the window toolbar.

5 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**, locate 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)	4 5 6	

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: 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 **Study** toolbar, click **Compute**.

RESULTS

Shape Optimization Sweep

1 In the **Model Builder** window, right-click **Shape Optimization** and choose **Duplicate**.
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: Shape Optimization Sweep/ Parametric Solutions 1 (sol5)**.
4 From the **Parameter value (N)** list, choose **4**.
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: Shape Optimization Sweep/ Parametric Solutions 1 (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.

Thumbnail

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

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

3 Right-click **Thumbnail** and choose **Move Out**.

4 Right-click **Thumbnail** and choose **Move Down**.

5 Right-click **Thumbnail** and choose **Move Down**.

Surface 1

1 In the **Model Builder** window, expand the **Thumbnail** node.

2 Right-click **Results>Thumbnail>Surface 1** and choose **Delete**.

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.

Line 2

1 Right-click **Results>Thumbnail>Line 1** and choose **Duplicate**.

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 **Results>Thumbnail>Line 2>Deformation 1** and choose **Delete**.

Thumbnail

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

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