Created in COMSOL Multiphysics 6.1

Topology Optimization of a Step Thrust Bearing

Introduction

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

Model Definition

The geometry and mesh is fixed as is the norm for topology optimization. The out-of-plane geometry is defined in terms of a spatially varying bearing clearance. Topology optimization is performed by letting the variables from the **Density Model** feature control this clearance. The load capacity of the bearing is a built-in variable, and this is used as objective function. The model is unconstrained and there is no need for projection or special material interpolation, because the problem is self-penalizing for the chosen objective. Regularization is not needed either, but a Helmholtz filter does give smoother results in postprocessing, so this is enabled. See *Topology Optimization of an MBB Beam* in the Optimization Module Application Library for more details.

Results and Discussion

The topology optimized design is shown in Figure 1 below.

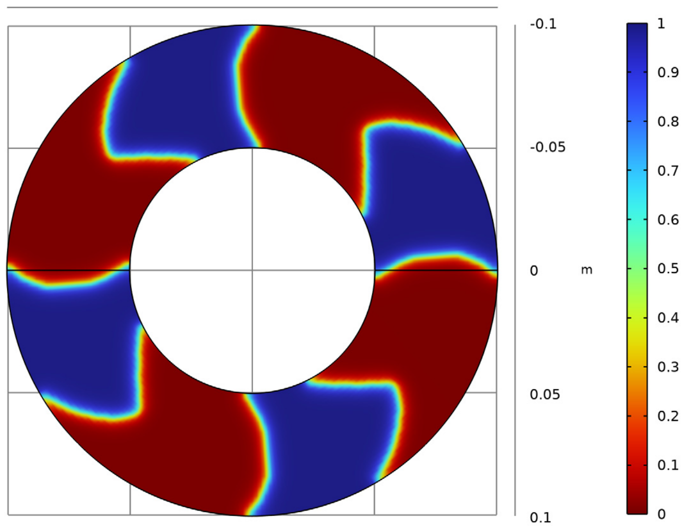


Figure 1: The topology optimized step thrust bearing with grooves (blue) and pads (red).

The optimization finds a design with four grooves when it is initialized with a uniform design. To investigate whether this is indeed the global optimum, the optimization is started with different non-uniform designs to provoke local optima corresponding to three and five grooves. Comparing the performance shows that the design with four grooves has the highest objective function, see Figure 2.

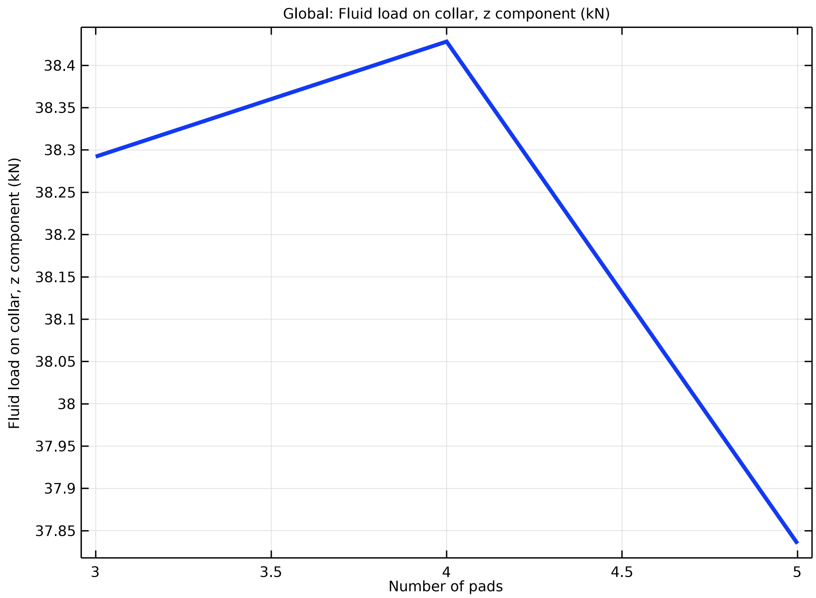


Figure 2: The optimized bearing capacity is plotted versus the number of grooves.

Note that the optimization results in a design specific to the chosen rotation direction. Some practical applications will require that both directions are taken into account, so that symmetry with respect to the rotation can be achieved.

Notes About the COMSOL Implementation

The model demonstrates verification on a body fitted mesh in a new component, as this is good practice for topology optimization. The performance is slightly better with the body fitted mesh.

The model indicates that four grooves is the optimal topology, but running the optimization with a finer mesh and a higher projection slope can give different topologies with significantly better performance (`N=1, beta=8, meshsz=1[mm]` is one example). This design is qualitatively identical to the design with four grooves, but the ratio between the length and width of the grooves is significantly higher than what is seen in common bearing designs.

Application Library path: `Rotordynamics_Module/Optimization/step_thrust_bearing_topology_optimization`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Rotordynamics>Hydrodynamic Bearing (hdb)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

Start by importing the parameters for the thrust bearing.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `step_thrust_bearing_topology_optimization.txt`.

Next, create the geometry for the bearing. While doing so, define selections for later use.

GEOMETRY 1

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `Ro`.

4 Click **Build Selected**.

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

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Circle 1 (c1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `Ri`.

Work Plane 1 (wp1)>Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

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

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **c2** only.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **dif1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Find the **End vertex** subsection. Click to select the **Activate Selection** toggle button.

5 On the object **dif1**, select Point 8 only.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Partition Objects 1 (par1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **dif1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Find the **Tool objects** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **ls1** only.

6 Click **Build Selected**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

Interior 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 **Work Plane 1** 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 Select the **Interior edges** check box.

10 Clear the **Exterior edges** check box.

11 In the **Label** text field, type `Interior Edges`.

Circumferential Edges

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

2 In the **Settings** window for **Complement Selection**, type `Circumferential Edges` 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 **Interior Edges** in the **Selections to invert** list.

6 Click **OK**.

MATERIALS

Material 1 (mat1)

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

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**.

4 In the **Settings** window for **Hydrodynamic Bearing**, click to expand the **Cavitation** section.

5 Select the **Cavitation** check box.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.

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

Add a **Topology Optimization** node to optimize the shape of the pads.

COMPONENT 1 (COMP1)

Density Model 1 (dtopo1)

1 In the **Definitions** toolbar, click **Optimization** and choose **Topology Optimization>Density Model**.

2 In the **Settings** window for **Density Model**, 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 **Projection** section. From the **Projection type** list, choose **Hyperbolic tangent projection**.

6 In the β text field, type `beta`.

7 Locate the **Interpolation** section. From the **Interpolation type** list, choose **Linear**.

8 Locate the **Control Variable Discretization** section. From the **Element order** list, choose **Constant**.

Define a non-uniform initial value, so that the optimization can be started close to different local optima.

9 Locate the **Control Variable Initial Value** section. In the θ_0 text field, type `if(initUniform,volfrac,0.5+0.5*sin(N*atan2(Yg,Xg)))`.

HYDRODYNAMIC BEARING (HDB)

Hydrodynamic Thrust Bearing 1

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

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 `hg+hf*dtopo1.theta_p`.

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

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 **Orientation vector defining local y direction** vector as

1	x
0	y
0	z

MESH 1

Mapped 1

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

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

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

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

4 Click **Build All**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz`.

7 In the **Minimum element size** text field, type `meshsz/2`.

8 Click **Build All**.

STUDY 1

Topology Optimization

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

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

3 In the **Settings** window for **Study**, type `Study: Sweep Initial Condition` in the **Label** text field.

Add a **Parametric Sweep** for the number of pads, initial density distribution of the pad profile, mesh size, and projection slope.

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**twice.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
N (Number of pads)	3 3 4 5	
initUniform (Uniform initialization)	1 0 0 0	

Topology Optimization

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

2 In the **Settings** window for **Topology Optimization**, 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 - N> comp1.hdb.htb1.Fcz - Fluid load on collar, z component**.

Scale the objective for better behavior of the optimization solver.

3 Locate the **Objective Function** section. From the **Objective scaling** list, choose **Initial solution based**.

4 From the **Type** list, choose **Maximization**.

Initialize the study to generate plots to show while solving.

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

RESULTS

Output material volume factor

1 In the **Model Builder** window, expand the **Topology Optimization** node, then click **Output material volume factor**.

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

3 From the **Dataset** list, choose **Study: Sweep Initial Condition/Solution 1 (sol1)**.

4 Click the **Go to XY View** button in the **Graphics** toolbar, so that the plot updates while optimizing.

STUDY: SWEEP INITIAL CONDITION

Topology Optimization

1 In the **Model Builder** window, under **Study: Sweep Initial Condition** click **Topology Optimization**.

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

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Output material volume factor**.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Solver Configurations** node.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study: Sweep Initial Condition>Solver Configurations>Solution 1 (sol1)>Optimization Solver 1>Stationary 1>Segregated 1** node.

4 Right-click **Optimization** and choose **Move Up**to decrease the computational time.

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

The default plots show the pressure in the bearing, volume fraction, and optimization threshold.

RESULTS

Fluid Pressure (hdb)

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

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

3 From the **Parameter value (N,initUniform)** list, choose **1: N=3, initUniform=1**.

4 In the **Fluid Pressure (hdb)** toolbar, click **Plot**.

1D Plot Group 4

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

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

Surface 1

1 In the **Model Builder** window, expand the **Output material volume factor** node, then click **Surface 1**.

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

3 From the **Color table transformation** list, choose **Reverse**, so that the thicker oil layer becomes blue.

Output material volume factor

1 In the **Model Builder** window, click **Output material volume factor**.

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

3 From the **Dataset** list, choose **Study: Sweep Initial Condition/Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (N,initUniform)** list, choose **3: N=4, initUniform=0**to show the best design.

5 In the **Output material volume factor** toolbar, click **Plot**.

6 From the **Dataset** list, choose **Study: Sweep Initial Condition/Solution 1 (sol1)**, so that the plot still updates while optimizing.

Threshold

You can plot the bearing load capacity versus number of pads for the optimized shape of the bearings. Use the instructions below to generate this plot.

Bearing Capacity vs Number of Pads

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

2 In the **Settings** window for **1D Plot Group**, type `Bearing Capacity vs Number of Pads` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Sweep Initial Condition/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (N, initUniform)** list, choose **From list**.

5 In the **Parameter values (N,initUniform)** list, choose **2: N=3, initUniform=0**, **3: N=4, initUniform=0**, and **4: N=5, initUniform=0**.

Global 1

1 Right-click **Bearing Capacity vs Number of Pads** 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 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 **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `N`.

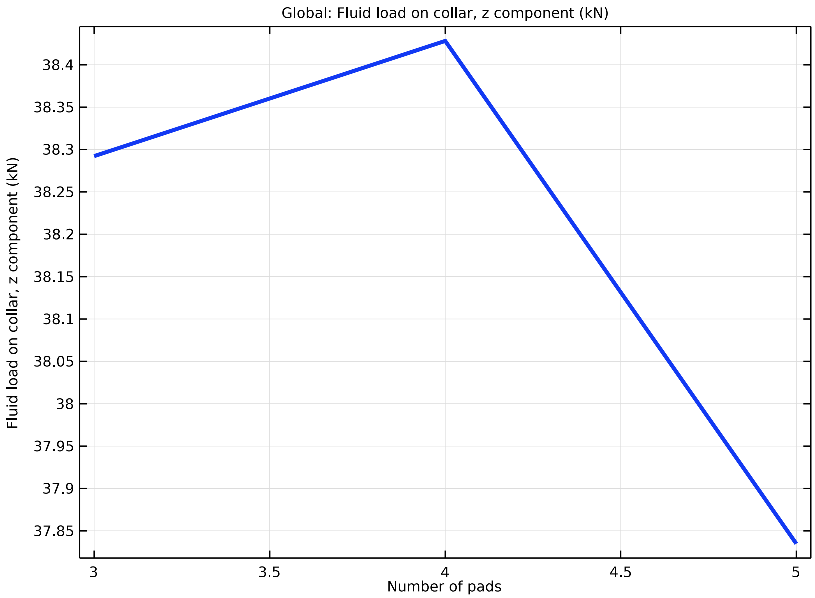
6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

Bearing Capacity vs Number of Pads

1 In the **Model Builder** window, click **Bearing Capacity vs Number of Pads**.

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

3 Clear the **Show legends** check box.



The performance seems to peak for four grooves. You can see that this a numerical effect by running the optimization with `meshsz=` 1[mm], `beta=`8, `N=`16. This gives a better objective, but the design is very different from conventional bearings. In the following we perform a verification in a new component, as this good practice in the context of topology optimization.

Mesh Import Parameters 1

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

2 Right-click **Filter** and choose **Mesh Import Parameters**.

3 In the **Settings** window for **Mesh Import Parameters**, locate the **Data** section.

4 From the **Parameter value (N,initUniform)** list, choose **3: N=4, initUniform=0**.

Filter

Right-click **Filter** 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 **Import**.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

GEOMETRY 2

Import 1 (imp1)

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

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

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Geometry 1**.

Grooves

1 Right-click **Import 1 (imp1)** and choose **Duplicate**.

2 In the **Settings** window for **Import**, type `Grooves` in the **Label** text field.

3 Locate the **Import** section. From the **Source** list, choose **Mesh or 3D printing file (STL, 3MF, PLY)**.

4 From the **Mesh** list, choose **Mesh Part 1**.

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

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

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

8 From the **Show in physics** list, choose **Boundary selection**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

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

3 From the **Repair tolerance** list, choose **Relative**.

4 In the **Relative repair tolerance** text field, type `1.0E-4`.

5 Click **Build Selected**.

Pads

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

2 In the **Settings** window for **Complement Selection**, type `Pads` 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 **Grooves** in the **Selections to invert** list.

6 Click **OK**.

DEFINITIONS (COMP2)

Groove Variables

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

2 In the **Settings** window for **Variables**, type `Groove Variables` in the **Label** text field.

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

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

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

Name	Expression	Unit	Description
hfilm	hg+hf	m	Film thickness

Pad Variables

1 Right-click **Groove Variables** and choose **Duplicate**.

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Pads**.

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

Name	Expression	Unit	Description
hfilm	hf	m	Film thickness

MATERIALS

Material 1 (mat1)

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

MATERIALS

Material 1 (mat2)

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**.

HYDRODYNAMIC BEARING (HDB2)

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

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

Hydrodynamic Thrust Bearing 1

1 In the **Model Builder** window, expand the **Hydrodynamic Bearing (hdb2)** node, then click **Hydrodynamic Thrust Bearing 1**.

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

3 In the h_{b1} text field, type `hfilm`.

MESH 2

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

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

3 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 `meshsz`.

4 In the **Minimum element size** text field, type `meshsz/2`.

5 In the **Curvature factor** text field, type `10`.

6 Click **Build All**.

ROOT

In the **Home** toolbar, click **Windows** and 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 In the **Model Builder** window, click the root node.

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

STUDY 2

Step 1: Stationary

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

2 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Hydrodynamic Bearing (hdb)		Automatic (Stationary)
Hydrodynamic Bearing (hdb2)	√	Automatic (Stationary)
Topology Optimization (Component 1)		Automatic

STUDY: SWEEP INITIAL CONDITION

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study: Sweep Initial Condition** node, then click **Step 1: Stationary**.

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

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Hydrodynamic Bearing (hdb)	√	Automatic (Stationary)
Hydrodynamic Bearing (hdb2)		Automatic (Stationary)
Topology Optimization (Component 1)	√	Automatic

VERIFICATION

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

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

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

RESULTS

Topology Optimization 1

In the **Model Builder** window, under **Results** right-click **Topology Optimization 1** and choose **Delete**.

Objective Comparison

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Objective Comparison` in the **Label** text field.

Global Evaluation 1

1 Right-click **Objective Comparison** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Add Expression** in the upper-right corner of the **Expressions** 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**.

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Verification/Solution 7 (4) (sol7)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 2 (comp2)>Hydrodynamic Bearing>Fluid loads>Fluid load on collar - N>hdb2.htb1.Fcz - Fluid load on collar, z component**.

5 In the **Objective Comparison** toolbar, click **Evaluate**.

The performance is slightly higher for the verification in the new component.