



Model created in COMSOL Multiphysics 6.4

# Shape Optimization of a Microphone Diaphragm

## *Introduction*

---

Shape optimization can be applied to achieve different design objectives. This tutorial demonstrates how to tune the lowest natural frequency of a MEMS microphone diaphragm using the **Shape Optimization** interface and study. In this example, **Stationary** and **Eigenfrequency** study steps are used in succession to deform the suspension springs and decrease the lowest natural frequency to 125 kHz from around 150 kHz of the initial design. Such investigations can occur during the design phase. Although the model is in 3D, shape optimization preserves the thickness of the structural layer in accordance with standard MEMS fabrication processes.

---

**Note:** This example requires the Optimization Module.

---

Manufacturing constraints might prevent all of the performance improvements gained by the shape optimization, but in many cases the deformed design can be used to fine tune the design and/or the manufacturing process to achieve improved performance. Having some idea about the magnitude of the potential performance improvements can also be useful when deciding whether to change the manufacturing tool/process.

This tutorial includes the use of the following features:

- **Mirror Symmetry** — to preserve the symmetry of the design
- **Symmetry/Roller** — to keep the thickness of the structural layer constant during shape changes, since actual MEMS fabrication results in uniform layer thickness

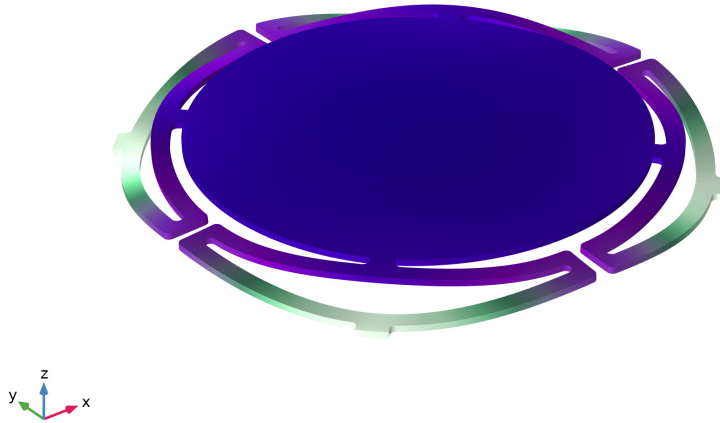
## *Model Definition*

---

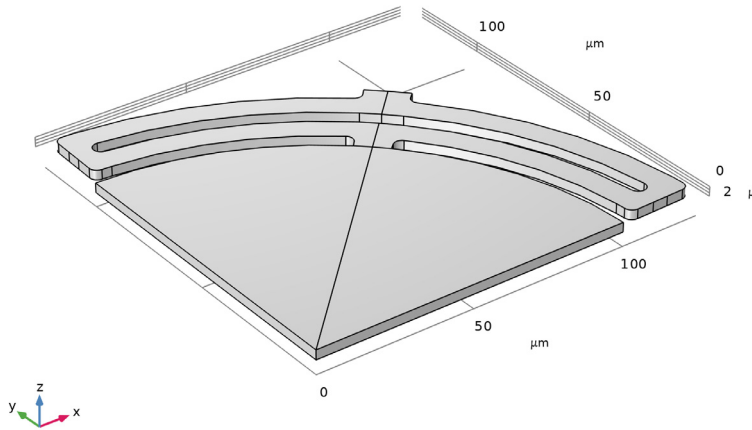
The MEMS microphone diaphragm responds to acoustics pressure with motion along the  $z$  direction. The structure has a twofold symmetry comprising a disc and suspension springs with four anchor points, as shown in [Figure 1](#). Because of the symmetry of the structure and the mode of interest, it is sufficient to model one quadrant, as shown in [Figure 2](#). The structure is made of silicon, a linear elastic material. The first study in this tutorial solves for the lowest natural frequency of the initial design, around 150 kHz, as a reference.

Eigenfrequency=143.64 kHz

Displacement magnitude ( $\mu\text{m}$ )



*Figure 1: 3D plot of a MEMS microphone diaphragm with suspension springs anchored at four points. The plot shows displacement in the z direction.*

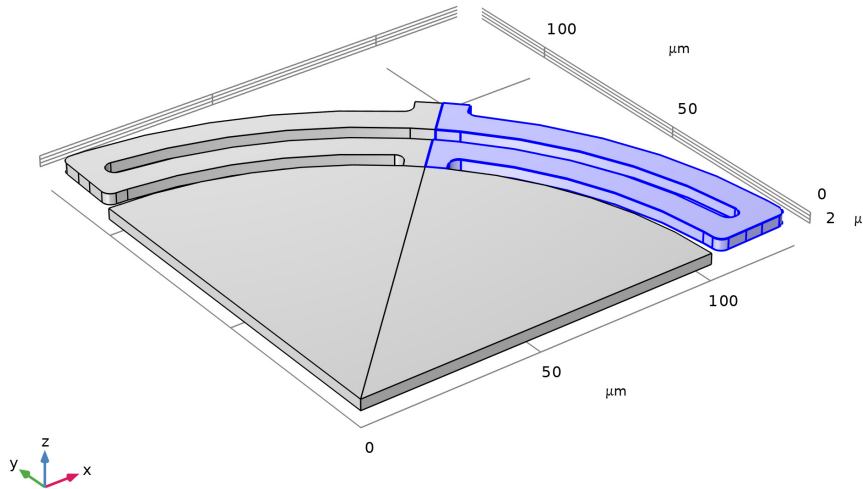


*Figure 2: 3D model of one quadrant of the diaphragm and the spring suspension.*

The **Shape Optimization** interface is added to specify the domain to be optimized, i.e., the spring suspension springs, as shown in [Figure 3](#). Other features are added here to control the shape deformation:

- **Free Shape Boundary** — to impose a maximum displacement
- **Mirror Symmetry** — to preserve symmetry of the design
- **Symmetry/Roller** — to restrict the shape deformation to displacements in the lateral direction in order to preserve the thickness of the silicon layer

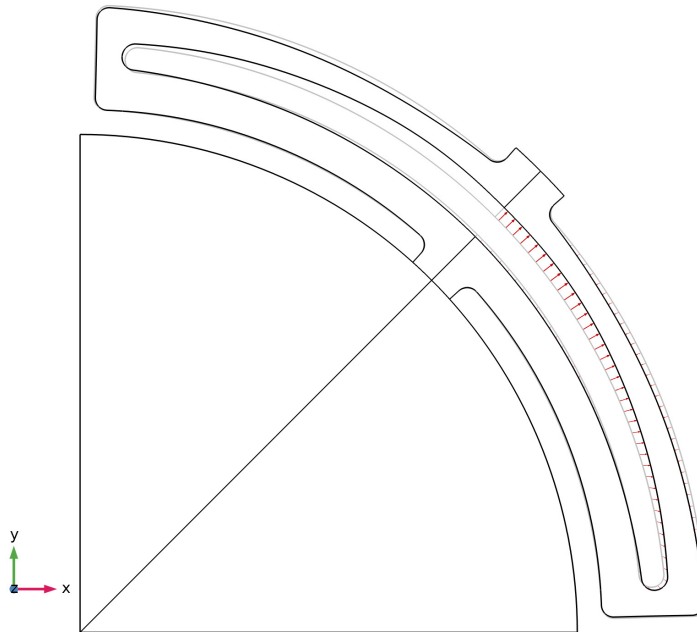
Note that although the layer thickness is fixed, the lateral deformation can vary in the vertical direction. This model performs a verification study where these variations are removed, but one can also use equation-based modeling to avoid it as demonstrated in the Optimization Module Application Library model *Optimization of an Extruded MBB Beam*.



*Figure 3: The free-shape domain, where shape deformation is allowed, is highlighted in blue.*

In the **Shape Optimization** study, the problem is a root-finding problem. It is defined as minimization of the objective function  $(\text{freq} - \text{f\_opt}[\text{kHz}])^2$ , where `freq` is the natural frequency and `f_opt` is the target for the first eigenfrequency. You can also add constraints such as, in this example, requiring that the deformed area not exceed the original area. Although the spring is optimized with respect to only the lowest natural frequency, the first six natural frequencies are computed in every optimization iteration. The IPOPT optimization algorithm is used to decrease the lowest natural frequency to 125 kHz. Note, however, that there are many designs that can achieve this result, so the solution to this optimization problem is not unique.

Finally, this tutorial also includes a procedure for exporting the data that defines the optimized shape and a third and last study that uses the optimized geometry for verification. Detailed information is found in the [Modeling Instructions](#) section.



*Figure 4: The shape-optimized suspension spring and displacement vectors (in red) from the boundaries of the original design.*

Shape optimization deforms the free-shape domain to obtain the natural frequency of 125 kHz. [Figure 4](#) shows a 2D plot of the shape-optimized suspension spring with displacement from the original boundaries to the initial design. This optimized geometry can then be exported in a text format for other uses, such as in a separate verification study.

---

**Application Library path:** MEMS\_Module/Actuators/  
microphone\_diaphragm\_shape\_optimization

---


### *Modeling Instructions*

---




Start by creating a new 3D model with a **Solid Mechanics** interface and then add an **Eigenfrequency** study.

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 > Solid Mechanics (solid)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies > Eigenfrequency**.
- 6 Click  **Done**.

## OPTIMIZATION

- 1 In the **Model Builder** window, click **Component 1 (comp1)**.
- 2 In the **Settings** window for **Component**, type Optimization in the **Label** text field.  
Define and enter the values for the following parameters.

## 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 In the table, enter the following settings:

Name	Expression	Value	Description
rd	100[um]	1E-4 m	Diaphragm radius
d	5[um]	5E-6 m	Gap
ws	8[um]	8E-6 m	Spring width
td	3[um]	3E-6 m	Diaphragm thickness
f_opt	125[kHz]	1.25E5 Hz	Optimized lowest natural frequency
alpha	45[deg]	0.7854 rad	Sector angle
beta	3[deg]	0.05236 rad	Rotation angle


## GEOMETRY 1

Create the geometry for microphone diaphragm using some of the parameters defined previously.

Use micrometers as the geometry unit.

- 1 In the **Model Builder** window, under **Optimization (comp1)** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Units** section.
- 3 From the **Length unit** list, choose  **$\mu\text{m}$** .


*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  $rd+2*ws+2*d$ .
- 4 In the **Sector angle** text field, type  $\alpha\text{-}\beta/2$ .
- 5 Locate the **Rotation Angle** section. In the **Rotation** text field, type  $\beta/2$ .
- 6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness ( $\mu\text{m}$ )
Layer 1	ws
Layer 2	d
Layer 3	ws

- 7 Click  **Build Selected**.

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


- 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  $rd+2*ws+3*d$ .
- 4 In the **Sector angle** text field, type  $\beta$ .
- 5 Locate the **Rotation Angle** section. In the **Rotation** text field, type  $\alpha\text{-}\beta$ .

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


Layer name	Thickness ( $\mu\text{m}$ )
Layer 1	ws+d
Layer 2	d
Layer 3	ws+d

7 Click  **Build Selected**.


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

- 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  $rd+2*ws+2*d$ .
- 4 In the **Sector angle** text field, type  $\beta$ .
- 5 Locate the **Rotation Angle** section. In the **Rotation** text field, type  $\beta/2$ .
- 6 Locate the **Layers** section. In the table, enter the following settings:


Layer name	Thickness ( $\mu\text{m}$ )
Layer 1	$2*ws+d$

7 Click  **Build Selected**.

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



- 1 Right-click **Plane Geometry** and choose **Delete Entities**.
- 2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 On the object **c1**, select Domains 1 and 3 only.
- 5 On the object **c2**, select Domains 1 and 3 only.
- 6 On the object **c3**, select Domain 1 only.
- 7 Click  **Build Selected**.

*Work Plane 1 (wp1) > Union 1 (uni1)*



- 1 In the **Work Plane** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 In the **Settings** window for **Union**, locate the **Union** section.
- 3 From the **Input objects** list, choose **All objects**.
- 4 Clear the **Keep interior boundaries** checkbox.

- 5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
- 6 Find the **Cumulative selection** subsection. Click **New**.
- 7 In the **New Cumulative Selection** dialog, type Spring in the **Name** text field.
- 8 Click **OK**.



*Work Plane 1 (wp1) > Circle 4 (c4)*

- 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 rd.
- 4 In the **Sector angle** text field, type alpha.
- 5 Click  **Build Selected**.
- 6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
- 7 Find the **Cumulative selection** subsection. Click **New**.
- 8 In the **New Cumulative Selection** dialog, type Diaphragm in the **Name** text field.
- 9 Click **OK**.


*Work Plane 1 (wp1) > Union 2 (uni2)*


- 1 In the **Work Plane** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 In the **Settings** window for **Union**, locate the **Union** section.
- 3 From the **Input objects** list, choose **All objects**.
- 4 Click  **Build Selected**.

*Work Plane 1 (wp1) > Fillet 1 (fil1)*

- 1 In the **Work Plane** toolbar, click  **Fillet**.
- 2 On the object **uni2**, select Points 5, 12, 16, 17, 19, and 22 only.
- 3 In the **Settings** window for **Fillet**, locate the **Radius** section.
- 4 In the **Radius** text field, type d/2.
- 5 Click  **Build Selected**.

*Work Plane 1 (wp1) > Mirror 1 (mir1)*

- 1 In the **Work Plane** toolbar, click  **Transforms** and choose **Mirror**.
- 2 In the **Settings** window for **Mirror**, locate the **Input** section.
- 3 From the **Input objects** list, choose **All objects**.

- 4 Select the **Keep input objects** checkbox.
- 5 Locate the **Normal Vector to Line of Reflection** section. In the **xw** text field, type -1.
- 6 In the **yw** text field, type 1.
- 7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
- 8 Locate the **Selections on Input Objects** section. Clear the **Propagate selections to resulting objects** checkbox.
- 9 Click  **Build Selected**.

#### *Extrude 1 (ext1)*

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
- 2 In the **Settings** window for **Extrude**, locate the **Distances** section.
- 3 In the table, enter the following settings:


<b>Distances (<math>\mu\text{m}</math>)</b>
td

- 4 Click  **Build Selected**.

Add some coordinate-based selections.

## **DEFINITIONS**

### *Spring Domain*

- 1 In the **Definitions** toolbar, click  **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, type Spring Domain in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Outer radius** text field, type Inf.
- 4 In the **Inner radius** text field, type rd-0.1.
- 5 In the **End angle** text field, type alpha.
- 6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

### *Spring Boundaries*

- 1 Right-click **Spring Domain** and choose **Duplicate**.
- 2 In the **Settings** window for **Cylinder**, type Spring 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  $rd+2*ws+3*d-0.1$ .
- 5 In the **Inner radius** text field, type  $rd+0.1$ .
- 6 In the **Top distance** text field, type  $td-0.1$ .
- 7 In the **Bottom distance** text field, type  $0.1$ .
- 8 In the **End angle** text field, type  $\alpha-0.1$ .
- 9 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity intersects cylinder**.

#### *Mirror Symmetry*

- 1 In the **Model Builder** window, right-click **Spring Domain** and choose **Duplicate**.
- 2 In the **Settings** window for **Cylinder**, type **Mirror Symmetry** in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Start angle** text field, type  $\alpha$ .
- 4 In the **End angle** text field, type  $2*\alpha$ .


#### *Spring Mirror Plane*

- 1 In the **Model Builder** window, right-click **Spring Boundaries** and choose **Duplicate**.
- 2 In the **Settings** window for **Cylinder**, type **Spring Mirror Plane** in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Outer radius** text field, type  $rd+2*ws+3*d+0.1$ .
- 4 In the **Inner radius** text field, type  $rd-0.1$ .
- 5 In the **Top distance** text field, type  $td+0.1$ .
- 6 In the **Bottom distance** text field, type  $-0.1$ .
- 7 In the **Start angle** text field, type  $\alpha-0.1$ .
- 8 In the **End angle** text field, type  $\alpha+0.1$ .
- 9 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

#### *Fixed Boundaries*



- 1 Right-click **Spring Mirror Plane** and choose **Duplicate**.
- 2 In the **Settings** window for **Cylinder**, type **Fixed Boundaries** in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Inner radius** text field, type  $rd+2*ws+3*d-0.1$ .
- 4 In the **Start angle** text field, type  $0$ .
- 5 In the **End angle** text field, type  $360$ .

### *Symmetry Planes*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Symmetry Planes** in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 1 and 3 only.

Add a material model.



### **ADD MATERIAL**

- 1 In the **Home** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **MEMS > Semiconductors > Si - Silicon (single-crystal, isotropic)**.
- 4 Click the **Add to Component** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.


Create a mesh.

### **MESH 1**




#### *Free Triangular 1*

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.
- 2 Click the  **Zoom Extents** button in the **Graphics** toolbar.
- 3 Select Boundaries 6 and 12 only.


#### *Size 1*

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Finer**.
- 4 Click  **Build Selected**.


#### *Copy Face 1*

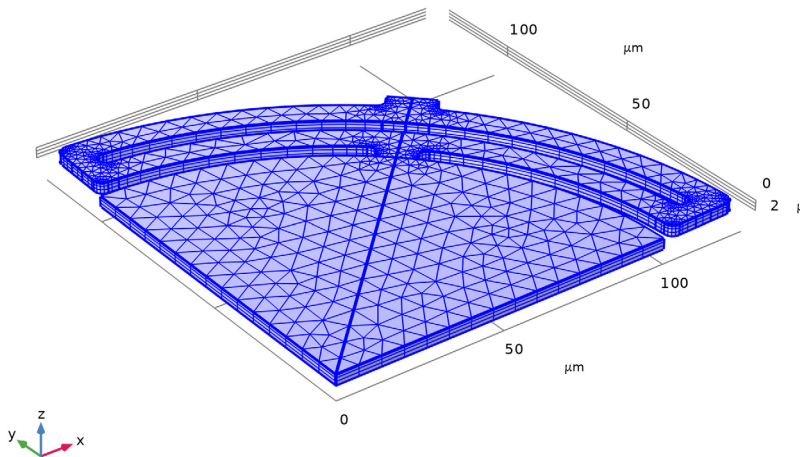
- 1 In the **Mesh** toolbar, click  **Copy** and choose **Copy Face**.
- 2 Select Boundaries 6 and 12 only.
- 3 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.
- 4 Click to select the  **Activate Selection** toggle button.
- 5 Select Boundaries 7 and 30 only.
- 6 Click  **Build Selected**.

### *Swept 1*

In the **Mesh** toolbar, click  **Swept**.

### *Distribution 1*


- 1 Right-click **Swept 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 3.
- 4 Click  **Build All**.




Specify the settings for the **Solid Mechanics** interface.

## **SOLID MECHANICS (SOLID)**

### *Fixed Constraint 1*

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

### *Symmetry 1*

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

3 From the **Selection** list, choose **Symmetry Planes**.

Set up and compute the **Eigenfrequency** study.

### INITIAL DESIGN

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

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

*Step 1: Eigenfrequency*

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

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

3 From the **Unit** list, choose **kHz**.

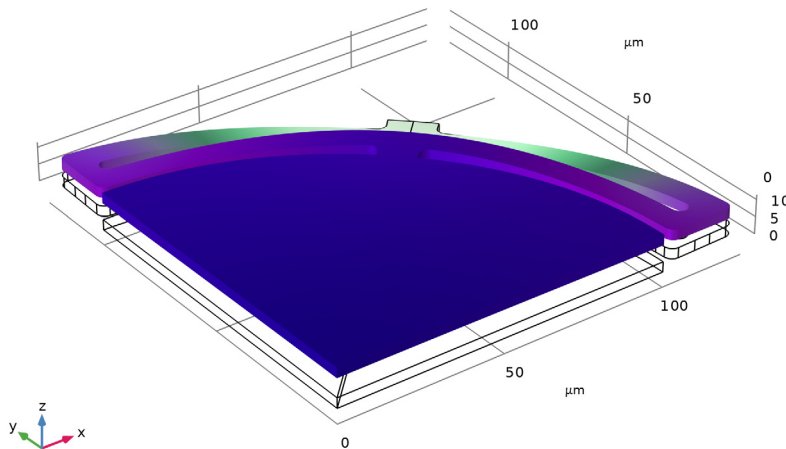
4 In the **Study** toolbar, click **Compute**.

### RESULTS

*Mode Shape (solid)*

Eigenfrequency=143.64 kHz


Displacement magnitude ( $\mu\text{m}$ )




Next, add a **Shape Optimization** interface and the necessary features.

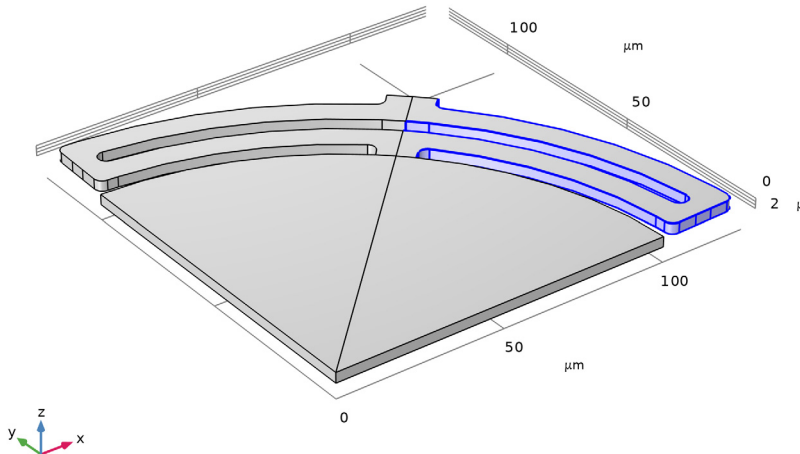
## OPTIMIZATION (COMPI)

### Free Shape Domain I


- 1 In the **Physics** toolbar, click  **Optimization** and choose **Shape Optimization**.
- 2 In the **Settings** window for **Free Shape Domain**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Spring Domain**.

### Free Shape Boundary I


- 1 In the **Shape Optimization** toolbar, click  **Free Shape Boundary**.
- 2 In the **Settings** window for **Free Shape Boundary**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Spring Boundaries**.



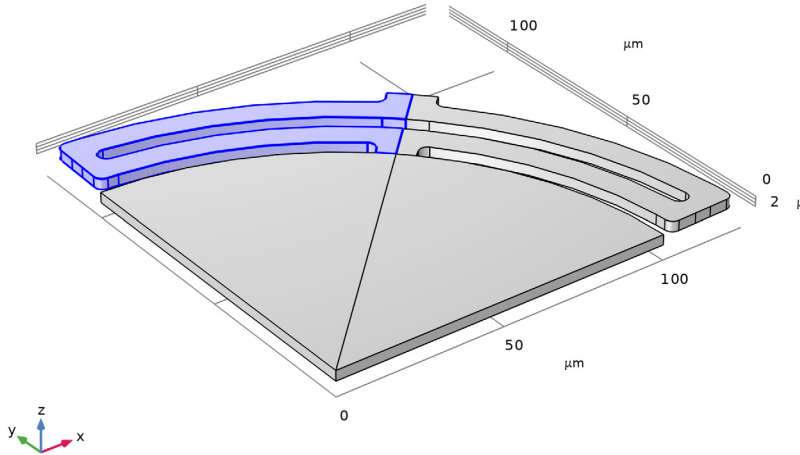
### Symmetry/Roller I

- 1 In the **Shape Optimization** toolbar, click  **Symmetry/Roller**.
- 2 Select Boundaries 29 and 30 only.


### Mirror Symmetry I

- 1 In the **Shape Optimization** toolbar, click  **Mirror Symmetry**.
- 2 In the **Settings** window for **Mirror Symmetry**, locate the **Geometric Entity Selection** section.

- 3 From the **Selection** list, choose **Mirror Symmetry**.





#### *Symmetry/Roller 2*

- 1 In the **Shape Optimization** toolbar, click  **Symmetry/Roller**.
- 2 In the **Settings** window for **Symmetry/Roller**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Spring Mirror Plane**.


Next, add an empty study for the **Shape Optimization** study using the **Stationary then Eigenfrequency** study step.

#### **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 **Empty Study**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

## STUDY 2

### Shape Optimization

- 1 In the **Study** toolbar, click  **Optimization** and choose **Shape Optimization**.
- 2 In the **Settings** window for **Study**, type Shape Optimization in the **Label** text field.

### Step 1: Stationary Then Eigenfrequency

- 1 In the **Study** toolbar, click  **More Study Steps** and choose **Eigenfrequency > Stationary Then Eigenfrequency**.
- 2 In the **Settings** window for **Stationary Then Eigenfrequency**, locate the **Study Settings** section.
- 3 From the **Unit** list, choose **kHz**.

Define the objective function. Replace the default GCMMA with IPOPT and choose to use initial solution for objective scaling to reduce the computational time.

### Shape Optimization

- 1 In the **Model Builder** window, click **Shape Optimization**.
- 2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.
- 3 From the **Method** list, choose **IPOPT**.
- 4 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description
$(\text{real}(\text{freq}) - f_{\text{opt}})^2$	Error



- 5 From the **Solution** list, choose **Use first**.
- 6 Find the **Objective settings** subsection. From the **Objective scaling** list, choose **Initial solution based**.
- 7 Click **Replace Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Optimization (comp1) > Definitions > Free Shape Domain 1 > comp1.fsd1.relVolume - Material volume divided by geometry volume - 1**.
- 8 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound
comp1.fsd1.relVolume		1

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





## RESULTS

### *Mode Shape (solid)*


- 1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, click  **Plot Next**.
- 3 Click  **Plot Previous**.

Data that defines the optimized geometry can be exported for other uses, for example to define a new lithographic mask. The following steps describe how to export the data in a text format to create a new geometry for a separate FEM study. The procedure starts with (a) creating a 2D plot of the displaced boundaries, (b) exporting this as a text with sectionwise data format, (c) creating a mesh part based on the data, and (d) importing this mesh part when creating a new geometry.


### *Shape Optimization*

- 1 Click the  **Go to XY View** button in the **Graphics** toolbar.
- 2 Click the  **Orthographic Projection** button in the **Graphics** toolbar.
- 3 Click the  **Show Grid** button in the **Graphics** toolbar.
- 4 Click the  **Show Legends** button in the **Graphics** toolbar.


### *Cut Plane 1*

- 1 In the **Model Builder** window, expand the **Results > Datasets** node.
- 2 Right-click **Results > Datasets** and choose **Cut Plane**.
- 3 In the **Settings** window for **Cut Plane**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Shape Optimization/Solution 2 (sol2)**.
- 5 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.
- 6 In the **Z-coordinate** text field, type  $0.5 * t_d$ .
- 7 Click  **Plot**.

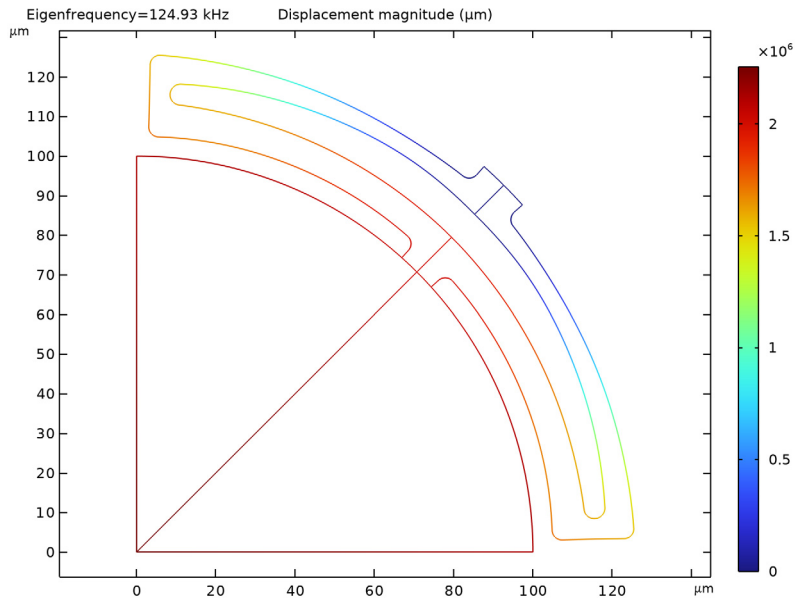
### *For Export*

- 1 In the **Results** toolbar, click  **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type For Export in the **Label** text field.


### *Line 1*

- 1 Right-click **For Export** and choose **Line**.
- 2 In the **For Export** toolbar, click  **Plot**.

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



*Plot 1*

- 1** In the **Results** toolbar, click  **Data** and choose **Plot**.
- 2** In the **Settings** window for **Plot**, locate the **Plot** section.
- 3** From the **Plot group** list, choose **For Export**.
- 4** Locate the **Output** section. In the **Filename** text field, type `microphone_diaphragm_shape_optimization_optimized_geometry.txt`.
- 5** From the **Data format** list, choose **Sectionwise**, then click **Export**.

## GLOBAL DEFINITIONS

In the **Model Builder** window, right-click **Global Definitions** and choose **Mesh Parts > 2D Part**.

## MESH PART 1

*Import 1*

- 1** In the **Settings** window for **Import**, locate the **Import** section.
- 2** From the **Source** list, choose **Sectionwise file**.
- 3** In the **Filename** text field, type `microphone_diaphragm_shape_optimization_optimized_geometry.txt`.

4 Click  **Import**.

#### ADD COMPONENT

Right-click **Global Definitions > Mesh Parts > Mesh Part 1 > Import 1** and choose **3D**.

#### VERIFICATION

In the **Settings** window for **Component**, type **Verification** in the **Label** text field.


#### GEOMETRY 2

1 In the **Model Builder** window, under **Verification (comp2)** click **Geometry 2**.

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

3 From the **Length unit** list, choose **µm**.

*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) > Import 1 (imp1)*

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

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

3 From the **Source** list, choose **Mesh**.

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

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

*Work Plane 1 (wp1) > Convert to Solid 1 (csol1)*

1 In the **Work Plane** toolbar, click  **Conversions** and choose **Convert to Solid**.

2 In the **Settings** window for **Convert to Solid**, locate the **Input** section.

3 Click the  **Paste Selection** button for **Input objects**.

4 In the **Paste Selection** dialog, type **imp1** in the **Selection** text field.



5 Click **OK**.

6 In the **Settings** window for **Convert to Solid**, locate the **Selections of Resulting Entities** section.



7 Select the **Resulting objects selection** checkbox.

8 Click  **Build Selected**.



*Work Plane 1 (wp1) > Adjacent Selection 1 (adjsel1)*

- 1 In the **Work Plane** toolbar, click  **Selections** and choose **Adjacent Selection**.
- 2 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.
- 3 Click **+ Add**.
- 4 In the **Add** dialog, select **Convert to Solid 1** in the **Input selections** list.
- 5 Click **OK**.
- 6 In the **Settings** window for **Adjacent Selection**, click  **Build Selected**.


*Work Plane 1 (wp1) > Adjacent Selection 2 (adjsel2)*

- 1 In the **Work Plane** 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, select **Adjacent Selection 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 domains**.
- 9 Click  **Build Selected**.

*Work Plane 1 (wp1) > Complement Selection 1 (comsel1)*

- 1 In the **Work Plane** toolbar, click  **Selections** and choose **Complement Selection**.
- 2 In the **Settings** window for **Complement Selection**, locate the **Input Entities** section.
- 3 Click **+ Add**.
- 4 In the **Add** dialog, select **Adjacent Selection 2** in the **Selections to invert** list.
- 5 Click **OK**.
- 6 In the **Settings** window for **Complement Selection**, click  **Build Selected**.

*Work Plane 1 (wp1) > Delete Entities 1 (dell)*

- 1 Right-click **Plane Geometry** and choose **Delete Entities**.
- 2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Complement Selection 1**.
- 5 Click  **Build Selected**.

### *Extrude 1 (ext1)*

- 1 In the **Model Builder** window, right-click **Geometry 2** and choose **Extrude**.
- 2 In the **Settings** window for **Extrude**, locate the **Distances** section.
- 3 In the table, enter the following settings:

<b>Distances (µm)</b>
td

- 4 Click  **Build Selected**.

Copy a coordinate-based selection from the **Optimization** component.

### **DEFINITIONS (COMP1)**

#### *Fixed Boundaries*

In the **Model Builder** window, under **Optimization (comp1) > Definitions > Selections** right-click **Fixed Boundaries** and choose **Copy**.

### **DEFINITIONS (COMP2)**

- 1 In the **Model Builder** window, expand the **Verification (comp2) > Definitions** node.
- 2 Right-click **Verification (comp2) > Definitions** and choose **Paste Cylinder**.

### **DEFINITIONS (COMP1)**

#### *Symmetry Planes*

In the **Model Builder** window, under **Optimization (comp1) > Definitions > Selections** right-click **Symmetry Planes** and choose **Copy**.


### **DEFINITIONS (COMP2)**

In the **Model Builder** window, under **Verification (comp2) > Definitions** right-click **Selections** and choose **Paste Explicit**.



### **ADD MATERIAL FROM LIBRARY**

In the **Home** toolbar, click  **Windows** and choose **Add Material from Library**.

### **ADD MATERIAL**


- 1 Go to the **Add Material** window.
- 2 In the tree, select **MEMS > Semiconductors > Si - Silicon (single-crystal, isotropic)**.
- 3 Click the **Add to Component** button in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

## 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 **Structural Mechanics > Solid Mechanics (solid)**.
- 4 Click the **Add to Verification** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

## SOLID MECHANICS 2 (SOLID2)

### *Fixed Constraint 1*

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

### *Symmetry 1*

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

## MESH 2



### *Free Triangular 1*

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.
- 2 Select Boundaries 6 and 12 only.


### *Size 1*

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Finer**.


### *Copy Face 1*

- 1 In the **Mesh** toolbar, click  **Copy** and choose **Copy Face**.
- 2 Select Boundaries 6 and 12 only.
- 3 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.
- 4 Click to select the  **Activate Selection** toggle button.
- 5 Select Boundaries 7 and 32 only.



### *Swept 1*

In the **Mesh** toolbar, click  **Swept**.

### *Distribution 1*

- 1 Right-click **Swept 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 3.
- 4 Click  **Build Selected**.


### **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** > **Eigenfrequency**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

### **VERIFICATION**

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

### *Step 1: Eigenfrequency*

- 1 In the **Model Builder** window, under **Verification** click **Step 1: Eigenfrequency**.
- 2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
- 3 From the **Unit** list, choose **kHz**.
- 4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Optimization (comp1)**.
- 5 In the **Study** toolbar, click  **Compute**.

### **RESULTS**

#### *Mode Shape (solid2)*

Next, delete the Shape Optimization plot associated with the last study.

#### *Shape Optimization 1*

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

### *Eigenfrequencies (Initial Design)*

Next, clear the selection of the Verification component in the first two Eigenfrequency study steps.

## **INITIAL DESIGN**

### *Step 1: Eigenfrequency*

- 1** In the **Model Builder** window, under **Initial Design** click **Step 1: Eigenfrequency**.
- 2** In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
- 3** In the **Solve for** column of the table, under **Verification (comp2)**, clear the checkbox for **Solid Mechanics 2 (solid2)**.

## **SHAPE OPTIMIZATION**

### *Step 1: Stationary Then Eigenfrequency*

- 1** In the **Model Builder** window, under **Shape Optimization** click **Step 1: Stationary Then Eigenfrequency**.
- 2** In the **Settings** window for **Stationary Then Eigenfrequency**, locate the **Physics and Variables Selection** section.
- 3** In the **Solve for** column of the table, clear the checkbox for **Verification (comp2)**.