Model created in COMSOL Multiphysics 6.4

Birdbath Resonator Gyroscope

Introduction

This tutorial shows how to model and analyze a birdbath resonator gyroscope (BRG), a type of Coriolis vibratory gyroscope invented at the University of Michigan and commercialized by Enertia Microsystems as a millimeter-scale navigation-grade gyroscope. This tutorial shows how the gyroscope's axisymmetric shape allows coupling between drive and sense modes and how the Coriolis force is measured.

Used in an inertial measurement unit (IMU), a gyroscope provides information on its position and orientation for navigation. A BRG has an axisymmetric structure made of fused quartz that gives it high sensitivity and high quality factor. A BRG can be much more accurate than commercial MEMS gyroscopes and match the performance of larger and more expensive gyroscopes. The BRG design minimizes cost, weight, size, and power consumption while delivering higher accuracy and stability. Based on Ref. 1, the model in this tutorial demonstrates the principles of BRG operation.

Model Definition

Figure 1 shows a cut-away diagram of a typical BRG (not of the actual model) comprising a conductive fused-quartz shell, four pairs of electrodes, and a substrate. The center of the shell is anchored to the substrate whereas its outer rim is free. The electrodes are positioned around it, forming parallel plate capacitors with small gaps. In this model, two electrodes are used to drive the gyroscope; one is used for sensing and the remaining are grounded. FEM can be used to compute the shell's deformation and frequency response under the effects of the Coriolis force due to the angular velocity Ω.

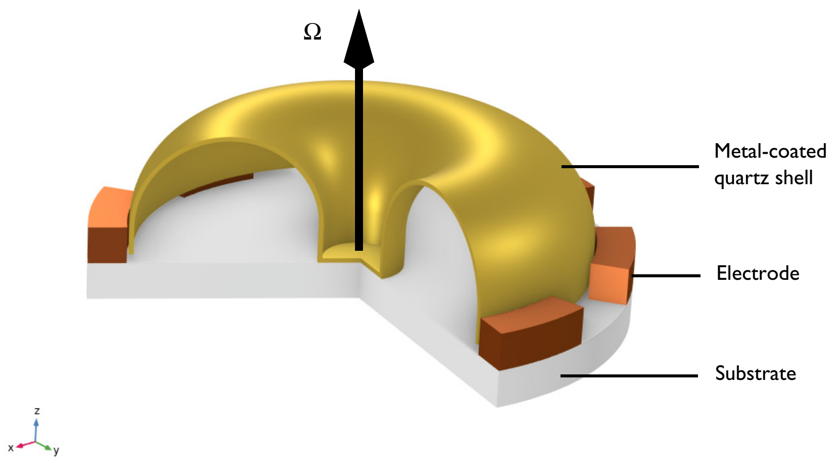


Figure 1: Cut-away diagram of a typical BRG comprising metal-coated quartz shell, electrodes, and substrate. The angular velocity of the rotating frame is Ω*, as indicated by the arrow through the center of the BRG.*

Due to the axisymmetry of the shell, this type of gyroscopes has two eigenmodes with nearly identical frequencies oriented 45° apart. These so-called $n = 2$ wine-glass modes are shown in Figure 2. Typically, the positions of drive and sense electrodes are aligned with the lower and the higher frequency modes, respectively. However, the higher frequency is applied to the drive electrodes to facilitate excitation of the second mode under Coriolis force. Its sensitivity to the Coriolis force arises from how easily the two modes can become coupled.

As the drive mode vibrates within the rotating frame, the Coriolis body force acts on the structure which excites the sense mode. The Coriolis force ($\mathbf{F}_{\mathrm{cor}}$) is given by

$$\mathbf{F}_{\mathrm{cor}} = -2\rho\Omega \times \frac{\partial \mathbf{u}}{\partial t}$$

where ρ is the density of the material, Ω is the angular velocity of the rotating frame, and $\mathbf{u}$ is the local displacement of the structure. From the above equation, it is clear that the Coriolis force is maximal when the angular velocity of the frame is parallel to the symmetry axis of the shell. In this case, the resulting force is in the in plane direction and produces a corresponding in-plane motion. This motion causes reaction moments along the rim of the shell, thereby exciting the sense mode. The sense electrodes measure the floating potential, converting the motion to electrical signal with a characteristic waveform.

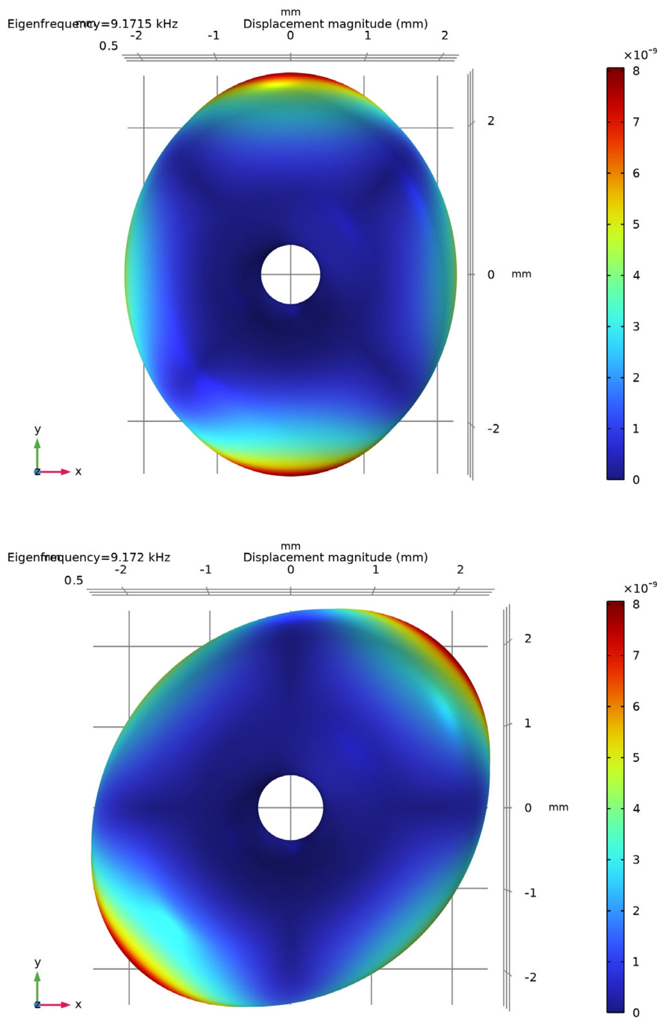


Figure 2: The two mode shapes of the BRG have nearly identical frequencies and are oriented 45° apart.

Rather than modeling the 30 μm-thick shell as a 3D volume, here you model the vibrating structure as a **Linear Elastic Material** shell under the **Shell** interface (requires a Structural

Mechanics Module license) as shown in Figure 3. For convenience, create the geometry and the mesh based on an 11.25° sector. The shell is defined as a perfect conductor with mechanical properties of fused quartz. In a real device, the shell could be made of fused quartz and coated with metal such as gold or platinum. Use the SiO_2 material model from the MEMS material library because it most closely matches the mechanical properties of fused quartz.

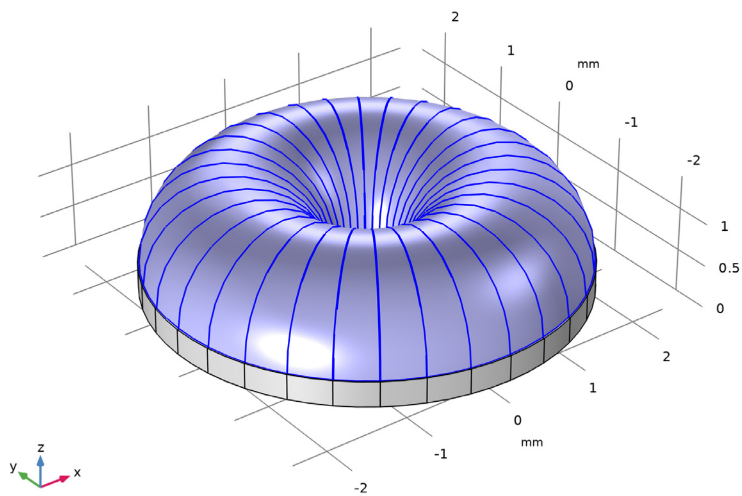


Figure 3: The shell domain of the BRG modeled using the Shell interface (requiring a Structural Mechanics Module license).

The shell and the electrodes form parallel-plate capacitors with 20 µm gaps which are very small relative to their areas so edge effects can be neglected to simplify geometry and mesh creation. Use **Boundary Terminal** features to represent the electrode surfaces opposite of the shell rather than model the electrodes as 3D objects. To minimize the size of the model, do not include the free space around the gyroscope and specify the electrostatics domain as a 20 µm-thick ring between the shell and the terminals. Set a 25 V potential difference with 1 V harmonic perturbation between the shell and drive electrodes. As previously mentioned, the drive electrodes are aligned with the lower-frequency mode shape, as seen in Figure 4.

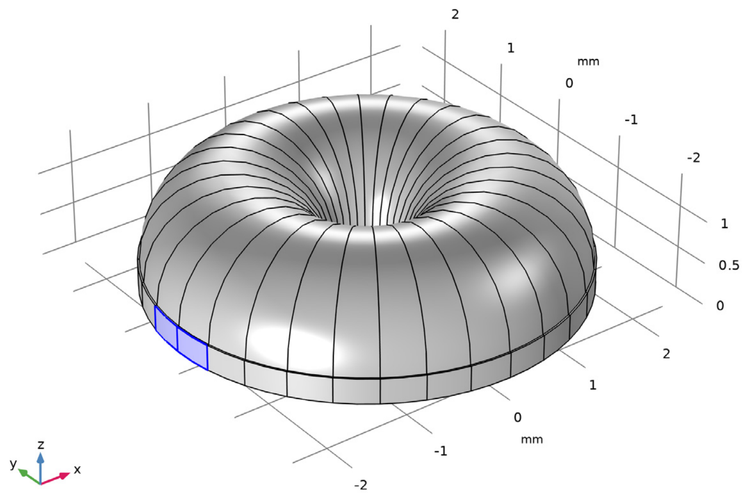


Figure 4: The electrostatic domain comprising terminal boundaries, the opposing surface of the shell, and a 20 um gap between them.

Rotation is modeled using the **Rotating Frame** feature of the **Shell** interface with the Coriolis force and Spin softening features enabled. In general, resonant structures require a very fine mesh to achieve an accurate frequency response. To save time, use a relatively coarse mesh for this tutorial and anticipate resonant peaks to shift if a finer mesh is used instead. The mesh is parameterized as a preparation for refinement studies. Follow the detailed steps in the Modeling Instructions section below.

Figure 2 above shows the $n = 2$ wine glass eigenmodes computed with a **Prestressed Eigenfrequency** study. Based on this result, use the higher frequency in the subsequent **Frequency Domain Perturbation** study to analyze the effect of angular velocity on the gyroscope. Figure 5 shows the plots of voltage versus phase measured by the sense (floating) electrode for $\Omega = 0$, 10, 20, 30, 40 and 50 rad/s. This result shows how the angular velocity can be read by an electronic circuit that measures and amplifies the voltage at the floating electrodes as a signal with an amplitude that is proportional to the Coriolis force.

Note that a real BRG can be connected to a complex control loop that generates a signal to negate the effect of Coriolis force. In this mode of operation, the amplitude and phase of this control signal become the measure of the Coriolis force.

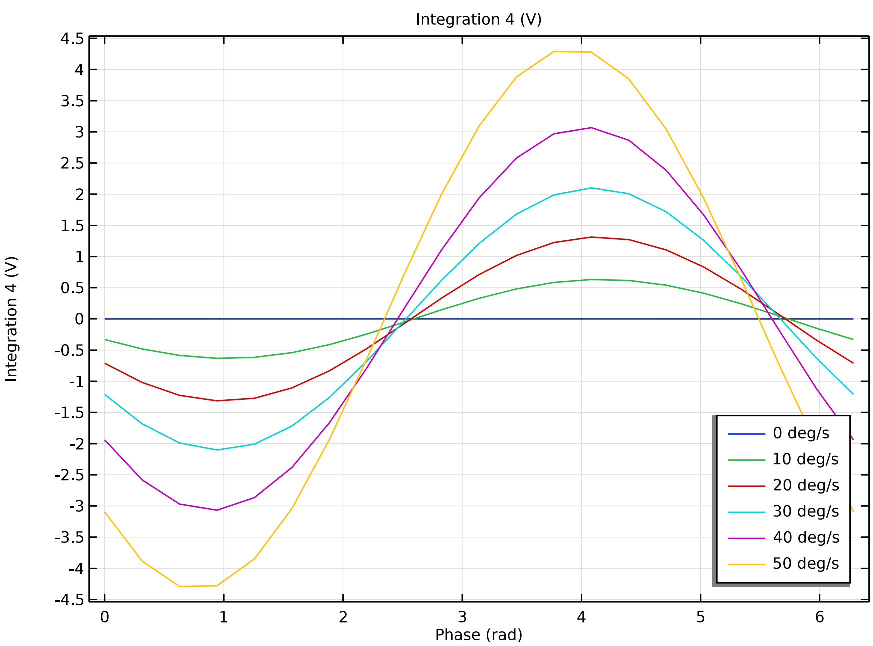


Figure 5: Floating potential measured at the sense electrode as a function of phase for angular velocities 0, 10, 20, 30, 40 and 50 rad/s.

Reference

1. J.Y. Cho and others, "1.5-Million Q-Factor Vacuum-Packaged Birdbath Resonator Gyroscope (BRG)," *2019 IEEE 32nd International Conference on Micro Electro Mechanical Systems (MEMS)*, Seoul, Korea (South), pp. 210–213, 2019.

Application Library path: `MEMS_Module/Sensors/birdbath_resonator_gyroscope`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, Start by creating a new 3D model with an **Electromechanics, Shell** multiphysics interface.

2 click **3D**.

3 In the **Select Physics** tree, select **AC/DC** > **Electromagnetics and Mechanics** > **Electromechanics** > **Electromechanics, Shell**.

4 Click **Add**.

5 Click **Study**.

6 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Shell** > **Eigenfrequency, Prestressed**.

7 Click **Done**.

GEOMETRY 1

The Model Wizard starts the COMSOL Desktop at the **Geometry** node. Take the opportunity to set the length unit to millimeter for convenience.

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

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

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

Define and specify the parameters of the model.

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
r	1[mm]	0.001 m	Radius
rc	0.4[mm]	4E-4 m	Center radius
ha	0.4[mm]	4E-4 m	Anchor height
he	0.3[mm]	3E-4 m	Electrode height
d	0.02[mm]	2E-5 m	Air gap
ts	0.03[mm]	3E-5 m	Shell thickness
Vdc	25[V]	25 V	DC bias
Vdrive	1[V]	1 V	Drive voltage
Omega	0[deg/s]	0 rad/s	Angular velocity
Q	5E5	5E5	Quality factor

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Work Plane 1 (wp1) > Plane Geometry

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

Work Plane 1 (wp1) > Circular Arc 1 (ca1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **xw** text field, type `r+rc`.

4 In the **yw** text field, type `ha`.

5 Locate the **Radius** section. In the **Radius** text field, type `r`.

6 Locate the **Angles** section. In the **End angle** text field, type `180`.

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 **ca1**, select Point 2 only.

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

4 From the **Specify** list, choose **Coordinates**.

5 In the **xw** text field, type `rc`.

Work Plane 1 (wp1) > Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `d`.

4 In the **Height** text field, type `he`.

5 Locate the **Position** section. In the **xw** text field, type `rc+2*r`.

6 In the **yw** text field, type `ha-he`.

7 Click **Build Selected**.

Revolve 1 (rev1)

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

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **Start angle** text field, type `78.75`.

5 In the **End angle** text field, type `90`.

6 Click **Build Selected**.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **rev1** only.

3 In the **Settings** window for **Rotate**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

5 Locate the **Rotation** section. Click **Range**.

6 In the **Range** dialog, type `11.25` in the **Start** text field.

7 In the **Step** text field, type `11.25`.

8 In the **Stop** text field, type `348.75`.

9 Click **Replace**.

10 In the **Settings** window for **Rotate**, click **Build Selected**.

11 Click **Build Selected**.

Define selections for the electrodes and other boundaries and domains to make specifying material models and physics interface features easier.

DEFINITIONS

Gap

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

2 In the **Settings** window for **Box**, type `Gap` in the **Label** text field.

Shell

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

2 In the **Settings** window for **Cylinder**, type `Shell` 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 `rc+2*r+0.01`.

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

Drive

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `2 3 222 223` in the **Selection** text field.

6 Click **OK**.

Sense

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `182 192` in the **Selection** text field.

6 Click **OK**.

Ground

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Ground` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog, type `37, 41, 49, 53, 85, 89, 113, 125, 183, 193` in the **Selection** text field.
6. Click **OK**.

Fixed Boundary

1. In the **Definitions** toolbar, click **Box**.
2. In the **Settings** window for **Box**, type `Fixed Boundary` in the **Label** text field.
3. Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
4. Locate the **Box Limits** section. In the **z maximum** text field, type `0.01`.
5. Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Roller Bottom

1. Right-click **Fixed Boundary** and choose **Duplicate**.
2. In the **Settings** window for **Box**, type `Roller Bottom` in the **Label** text field.
3. Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4. Locate the **Box Limits** section. In the **z maximum** text field, type `ha-he+0.001`.

Roller Top

1. Right-click **Roller Bottom** and choose **Duplicate**.
2. In the **Settings** window for **Box**, type `Roller Top` in the **Label** text field.
3. Locate the **Box Limits** section. In the **z minimum** text field, type `ha-0.001`.
4. In the **z maximum** text field, type `ha+0.001`.

Roller Side

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Roller Side` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog, type `12, 16, 36, 40, 60, 64, 84, 88, 154, 158, 174, 178, 194, 198, 214, 218` in the **Selection** text field.

6 Click **OK**.

Add SiO2 material to the model and specify the regions it belongs to.

ADD MATERIAL

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

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

3 In the tree, select **MEMS** > **Insulators** > **SiO2 - Silicon oxide**.

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

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

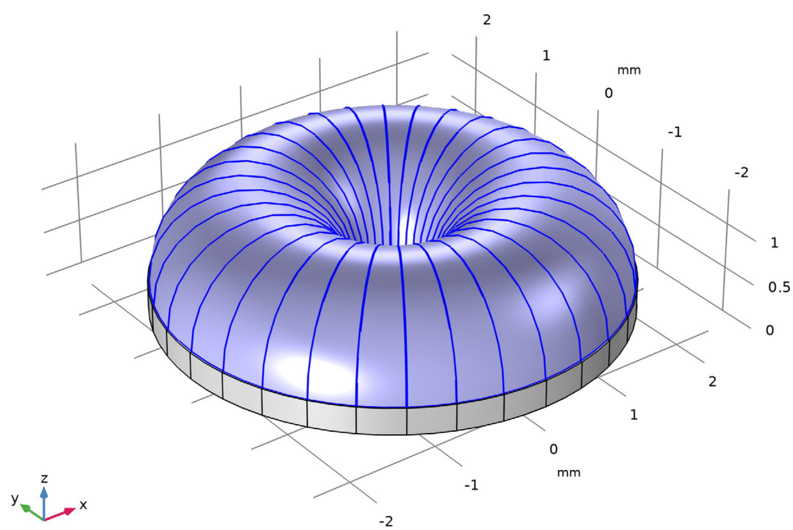
MATERIALS

SiO2 - Silicon oxide (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

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

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



MOVING MESH

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Deforming Domain 1**.
2 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Gap**.

Symmetry/Roller 1

1 In the **Model Builder** window, click **Symmetry/Roller 1**.
2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Roller Top**.

Symmetry/Roller 2

1 Right-click **Component 1 (comp1)** > **Moving Mesh** > **Symmetry/Roller 1** and choose **Duplicate**.
2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Roller Bottom**.

Symmetry/Roller 3

Right-click **Symmetry/Roller 2** and choose **Duplicate**.

Symmetry/Roller 2

From the **Selection** list, choose **Roller Side**.

Specify the settings for the **Electrostatics** interface.

ELECTROSTATICS (ES)

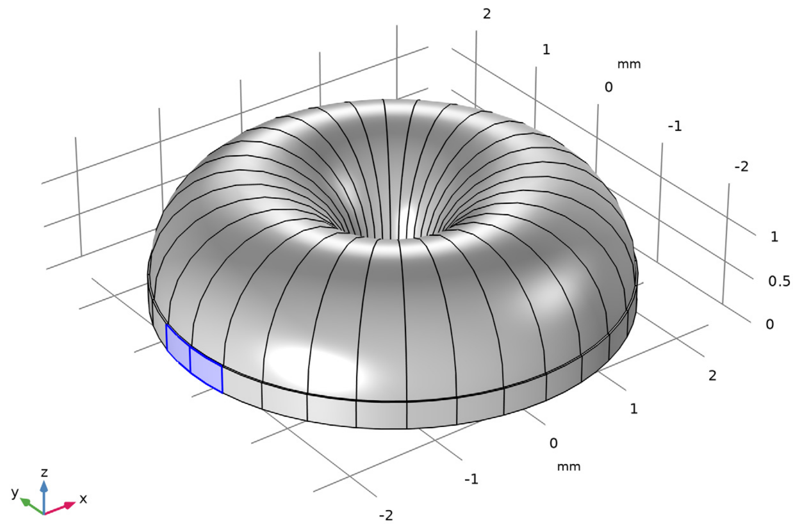
Shell

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Terminal**.
2 In the **Settings** window for **Boundary Terminal**, type `Shell` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Shell**.
4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.
5 In the V_0 text field, type `Vdc`.

Drive electrodes

1 Right-click **Shell** and choose **Duplicate**.
2 In the **Settings** window for **Boundary Terminal**, type `Drive electrodes` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Drive**.



4 Locate the **Terminal** section. In the V_0 text field, type `0`.

Harmonic Perturbation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Harmonic Perturbation**.

2 In the **Settings** window for **Harmonic Perturbation**, locate the **Terminal** section.

3 In the V_0 text field, type `Vdrive`.

Ground 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Ground**.

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

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

Sense electrode

1 In the **Physics** toolbar, click **Boundaries** and choose **Floating Potential**.

2 In the **Settings** window for **Floating Potential**, type `Sense electrode` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Sense**.

Define some integration operators for results processing.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 Select Point 3 only.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 Select Point 93 only.

Integration 3 (intop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 Select Point 157 only.

Integration 4 (intop4)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 Select Point 161 only.

Specify the settings for the **Shell** interface.

SHELL (SHELL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

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

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

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2 In the **Settings** window for **Damping**, locate the **Damping Settings** section.
3 From the **Damping type** list, choose **Isotropic loss factor**.
4 From the η_s list, choose **User defined**. In the associated text field, type `1/Q`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Thickness and Offset 1**.
2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
3 In the d_0 text field, type `ts`.
4 From the **Position** list, choose **Top surface on boundary**.

Rotating Frame 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rotating Frame**.
2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.
3 In the ω_r text field, type `Omega`.
4 Locate the **Frame Acceleration Effect** section. Select the **Coriolis force** checkbox.

Fixed Constraint 1

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

Create the mesh for the model.

MESH 1

Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2 In the **Settings** window for **Edge**, locate the **Edge Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type `203 205 207 274` in the **Selection** text field.
5 Click **OK**.

Distribution 1

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

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

3 In the list box, select **203**.

4 Click **Remove from Selection**.

5 Select Edges 205, 207, and 274 only.

6 In the list box, select **274**.

7 Click **Remove from Selection**.

8 Select Edges 205 and 207 only.

9 Locate the **Distribution** section. In the **Number of elements** text field, type 8.

Distribution 2

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

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

3 In the list box, select **205**.

4 Click **Remove from Selection**.

5 Select Edges 203, 207, and 274 only.

6 In the list box, select **207**.

7 Click **Remove from Selection**.

8 Select Edges 203 and 274 only.

9 Locate the **Distribution** section. In the **Number of elements** text field, type 6.

Mapped 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 122 in the **Selection** text field.

5 Click **OK**.

Edge 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 199-202, 215, 218 in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Edge 2** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.
3 In the list, choose **199** and **200**.
4 Click **Remove from Selection**.
5 Select Edges 201, 202, 215, and 218 only.
6 In the list, choose **202** and **215**.
7 Click **Remove from Selection**.
8 Select Edges 201 and 218 only.
9 Locate the **Distribution** section. In the **Number of elements** text field, type 60.

Distribution 2

1 In the **Model Builder** window, right-click **Edge 2** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.
3 In the list box, select **201**.
4 Click **Remove from Selection**.
5 Select Edges 199, 200, 202, 215, and 218 only.
6 In the list box, select **218**.
7 Click **Remove from Selection**.
8 Select Edges 199, 200, 202, and 215 only.
9 Locate the **Distribution** section. In the **Number of elements** text field, type 8.

Mapped 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.
2 Select Boundaries 119 and 120 only.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.
2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 18 only.
5 Click to expand the **Source Faces** section. Click **Paste Selection**.
6 In the **Paste Selection** dialog, type 122 in the **Selection** text field.
7 Click **OK**.

8 In the **Settings** window for **Swept**, click to expand the **Destination Faces** section.

9 Click **Paste Selection**.

10 In the **Paste Selection** dialog, type 125 in the **Selection** text field.

11 Click **OK**.

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 4.

4 Click **Build Selected**.

Copy Face 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 120 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type 10, 11, 21, 23, 33, 35, 45, 47, 57, 59, 69, 71, 81, 83, 93, 95, 117, 127, 129, 131, 133, 135, 137, 139, 141, 143, 145, 147, 149, 151, 153 in the **Selection** text field.

9 Click **OK**.

Copy Face 2

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 119 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type 96-111, 118, 126, 128, 130, 132, 134, 136, 138, 140, 142, 144, 146, 148, 150, 152 in the **Selection** text field.

9 Click **OK**.

10 In the **Settings** window for **Copy Face**, click **Build Selected**.

Copy 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy**.

2 In the **Settings** window for **Copy**, locate the **Dimension** section.

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

4 Locate the **Source Entities** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 18 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Copy**, locate the **Destination Entities** section.

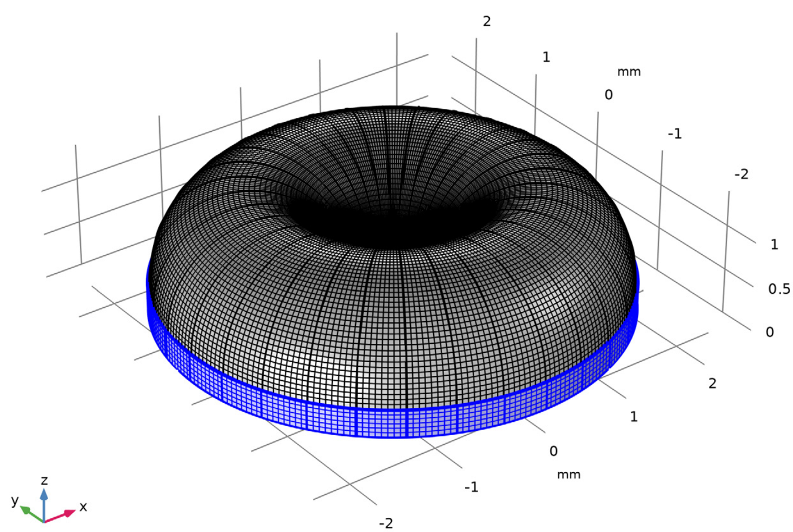
8 Click to select the **Activate Selection** toggle button.

9 Click **Paste Selection**.

10 In the **Paste Selection** dialog, type 1-17, 19-32 in the **Selection** text field.

11 Click **OK**.

12 In the **Settings** window for **Copy**, click **Build Selected**.



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

Study 1 - Prestressed Eigenfrequency

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

2 In the **Settings** window for **Study**, type `Study 1 - Prestressed Eigenfrequency` in the **Label** text field.

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

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 1 - Prestressed Eigenfrequency** click **Step 2: Eigenfrequency**.

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

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

4 In the **Search for eigenfrequencies around shift** text field, type `1`.

For the study to solve, disable the **Floating Potential** boundary.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

6 In the tree, select **Component 1 (comp1)** > **Electrostatics (es)** > **Sense electrode**.

7 Right-click and choose **Disable**.

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

Set up a **Frequency Domain Perturbation** study using the sense mode frequency from the previous study.

Add Study

1 In the **Study** 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 **Preset Studies for Selected Physics Interfaces** > **Shell** > **Frequency Domain, Prestressed**.

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

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

Study 2 - Frequency Domain Perturbation

1 In the **Settings** window for **Study**, type `Study 2 - Frequency Domain Perturbation` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 2 - Frequency Domain Perturbation** click **Step 2: Frequency-Domain Perturbation**.

2 In the **Settings** window for **Frequency-Domain Perturbation**, locate the **Study Settings** section.

3 From the **Frequency unit** list, choose **kHz**.

4 In the **Frequencies** text field, type `9.172`.

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Omega (Angular velocity)	`range(0, 10, 50)[deg/s]`	`rad/s`

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

RESULTS

Create plot to show the shapes of the eigenmodes.

Mode Shape

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

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

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

Surface 1

1 Right-click **Mode Shape** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Shell** > **Displacement** > **shell.disp - Displacement magnitude - m**.

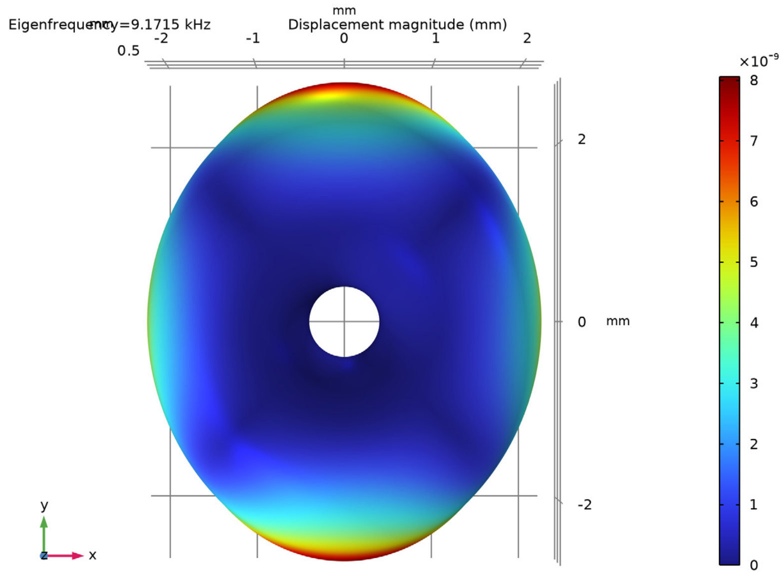
3 Locate the **Expression** section. Select the **Description** checkbox.

Deformation 1

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

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

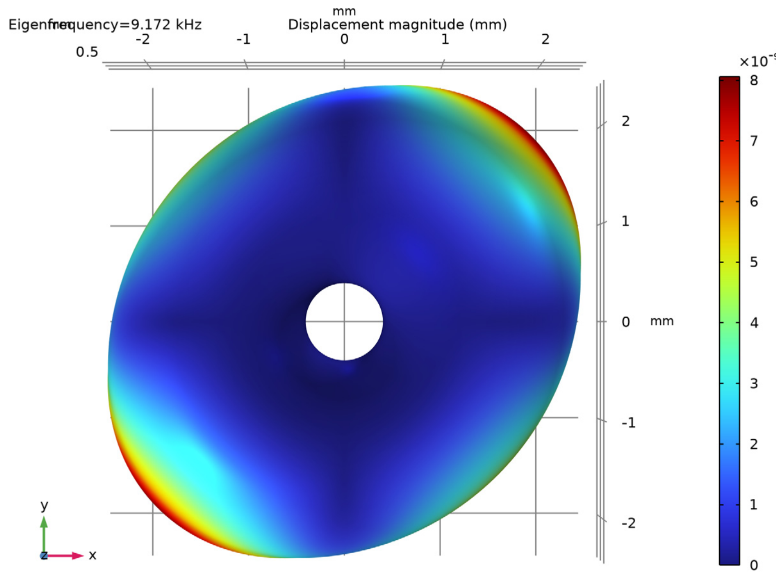
3 Click the **Go to XY View** button in the **Graphics** toolbar.



Surface 1

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

2 In the **Settings** window for **Surface**, click ➜ **Plot Next**.



Create plot to show the displacement from the frequency domain study.

Displacement

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Domain Perturbation/Solution 3 (sol3)**.

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

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Shell** > **Displacement** > **shell.disp - Displacement magnitude - m**.

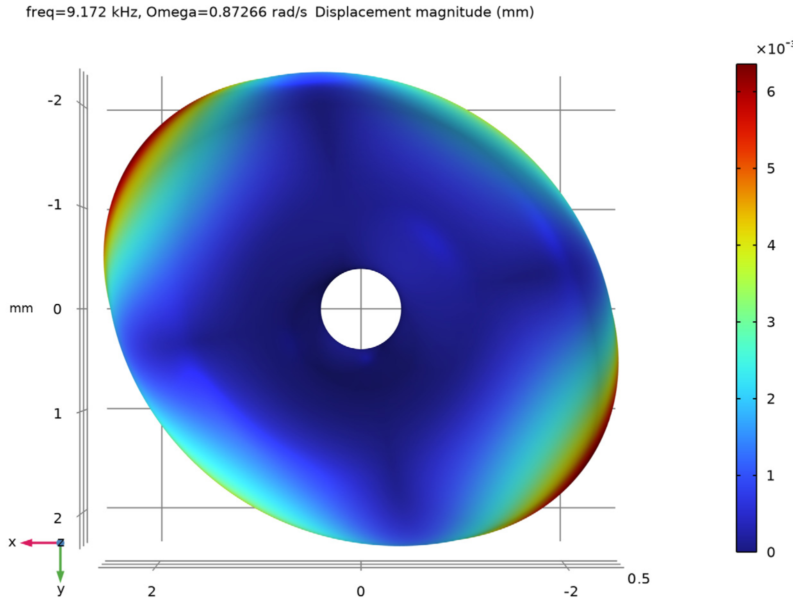
3 Locate the **Expression** section. Select the **Description** checkbox.

Deformation 1

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

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

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



Create a 1D plot of potential at the sense terminal as a function of phase.

Sense Electrode Potential

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

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

3 From the **Dataset** list, choose **Study 2 - Frequency Domain Perturbation/Solution 3 (sol3)**.

4 In the **Label** text field, type `Sense Electrode Potential`.

Global 1

1 Right-click **Sense Electrode Potential** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
intop4(V)	V	Integration 4

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Phase**.

5 Click to expand the **Legends** section. From the **Legends** list, choose **Automatic**.

6 Find the **Include** subsection. Clear the **Description** checkbox.

7 In the **Sense Electrode Potential** toolbar, click **Plot**.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
0 deg/s
10 deg/s
20 deg/s
30 deg/s
40 deg/s
50 deg/s

Sense Electrode Potential

1 In the **Model Builder** window, click **Sense Electrode Potential**.

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

3 From the **Position** list, choose **Lower right**.

4 In the **Sense Electrode Potential** toolbar, click **Plot**.