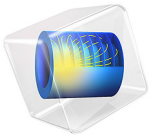


Model created in COMSOL Multiphysics 6.4

Micromachined Gyroscope with Mixed Formulation

Introduction

This tutorial shows how to model a capacitively-driven gyroscope using electromechanics coupling. The model solves for electromechanical force, Coriolis force and structure deformation without using analytic formulas used in A Micromachined Comb-Drive Tuning Fork Rate Gyroscope. The electromechanics coupling is specified at the start and the relevant physics are added automatically. The original device is loosely based on Ref. 1.

Model Definition

For convenience, the geometry is imported into the model. The structural part (excluding the free space around it) is the same as in A Micromachined Comb-Drive Tuning Fork Rate Gyroscope shown in the figure below. Various selection features are used for the setup of domain or boundary-dependent features and mesh.

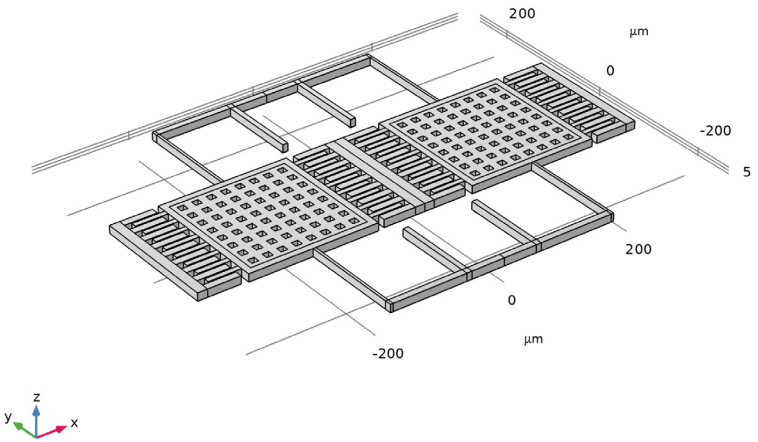


Figure 1: The geometry of the gyroscope.

The gyroscope is composed of two proof masses supported by springs anchored to the substrate (not explicitly modeled). The comb drive excites the drive mode with the two masses oscillating along the X-axis in opposite directions. The device is designed to sense rotations around the Y-axis. The combination of such rotations and the drive-mode motion causes a Coriolis force in the positive and negative Z directions, which excites the out-of-plane sense-mode oscillation of the two masses. The sense-mode oscillation is picked up capacitively with electrodes in the substrate.

The combs are assumed to be DC-biased at 20 V and AC-excited at 1.5 V. The sense electrodes are assumed to be DC-biased at 5 V.

To save time and file size, a relatively coarse mesh is used. Nevertheless, the mesh is parameterized to be ready for refinement studies.

Results and Discussion

Figure 2 shows the stationary response of the device. The masses are pulled down slightly by the bias voltage of the sense electrodes. The masses do not move horizontally since the DC part of the comb drive forces for each mass are equal and in opposite directions so they cancel out.

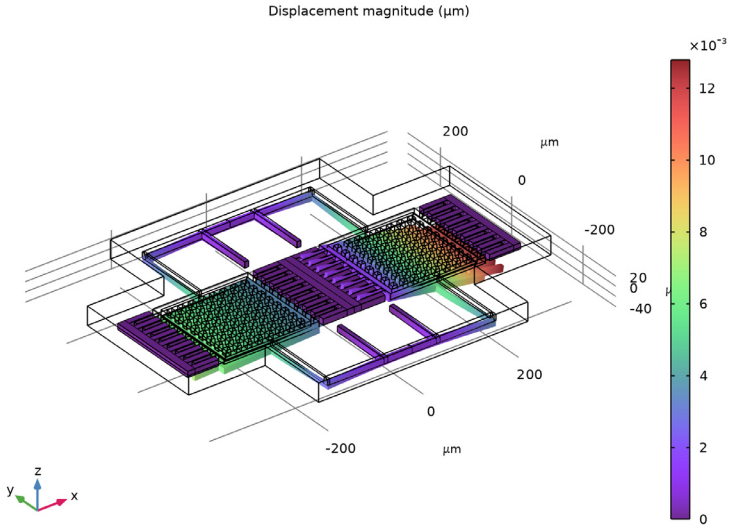


Figure 2: Stationary response of the device.

Figure 3 and Figure 4 show the prestressed eigenfrequencies and mode shapes of the in-plane drive mode and the out-of-plane sense mode, respectively.

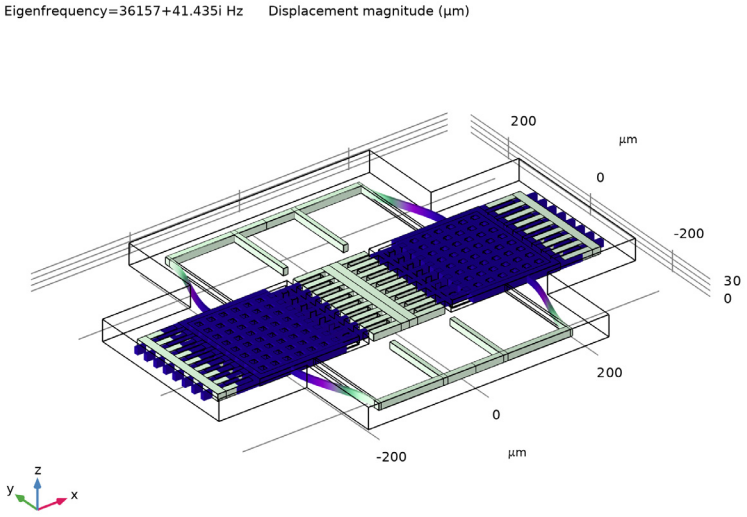


Figure 3: Drive mode shape.

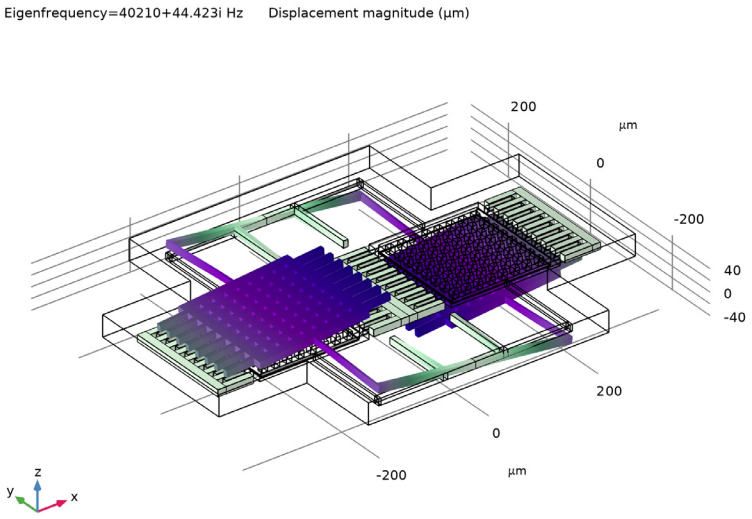


Figure 4: Sense mode shape.

The drive-mode and sense-mode resonant frequencies can be estimated with analytic formulas from standard textbooks (for example, Ref. 2). The agreement between the numerical and analytic results is good; see Table 1.

TABLE 1: NUMERICAL AND ANALYTIC RESULTS OF THE DRIVE-MODE AND SENSE-MODE FREQUENCIES.

	Drive mode	**Sense mode**
Numerical	36 kHz	40 kHz
Analytic	40 kHz	45 kHz

Figure 5 shows the drive-mode displacement under the simulated operation. The amplitude can be read off from the color legend.

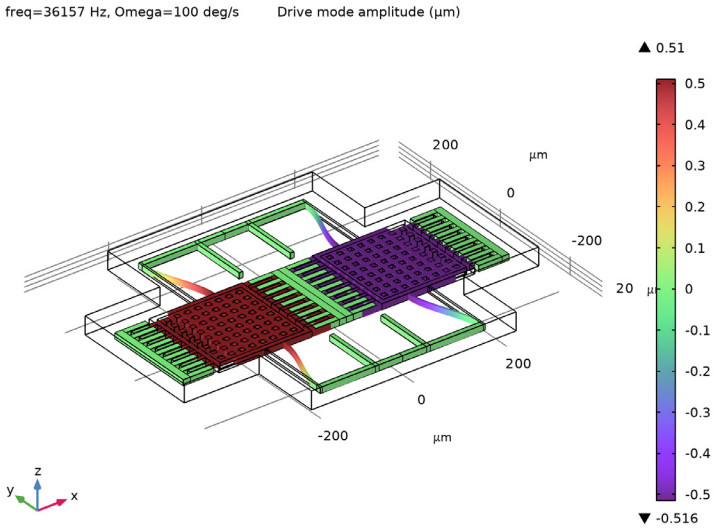


Figure 5: The drive mode displacement.

Figure 6 shows the sense-mode displacement. Due to the tilt of the masses, the amplitude is either read off the Evaluation 3D table after clicking around the centers of the masses, or evaluated using **Join** dataset as detailed in the Modeling Instructions section.

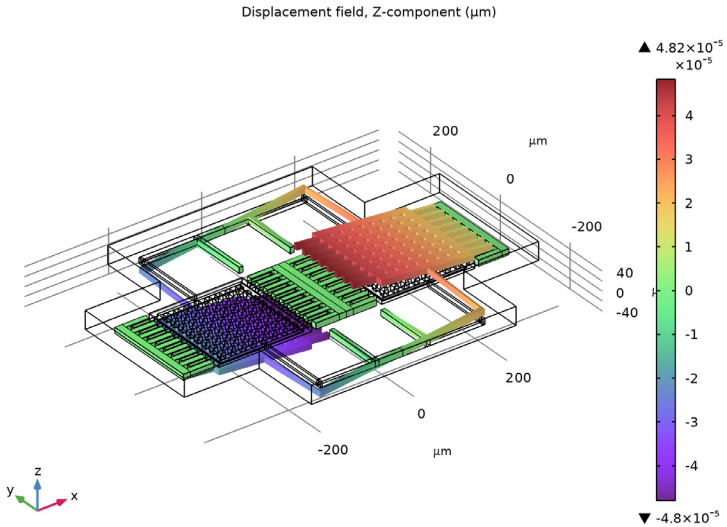


Figure 6: The sense-mode displacement.

Reference

1. J. Bernstein, S. Cho, A.T. King, A. Kourepenis, P. Maciel, and M. Weinberg, *"A micromachined comb-drive tuning fork rate gyroscope*," Proceedings IEEE Micro Electro Mechanical Systems, Fort Lauderdale, FL, USA, 1993, pp. 143–148.

2. V. Kaajakari, *Practical MEMS*, Small Gear Pub. (Las Vegas, Nev.), 2009.

Application Library path: `MEMS_Module/Sensors/gyroscope_mixed_formulation`

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 a **Electromechanics** multiphysics interface.

2 click **3D**.

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

4 Click **Add**.

5 Click **Study**.

6 In the **Select Study** tree, select **General Studies** > **Stationary**.

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 microns 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 **µm**.

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
Vcomb	20[V]	20 V	Comb drive voltage
V_ac	1.5[V]	1.5 V	AC voltage
Vbase	5[V]	5 V	Base voltage
Q	500	500	Quality factor
t_anchor	2[um]	2E-6 m	Anchor layer thickness

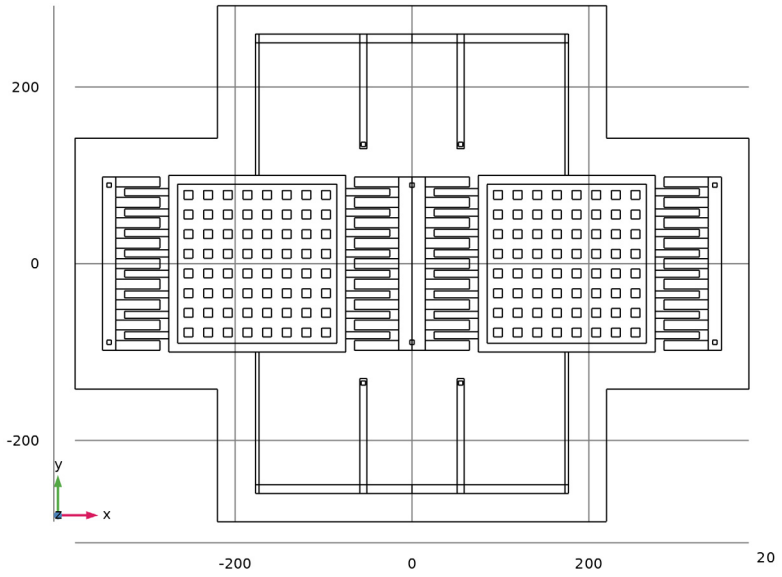
Name	Expression	Value	Description
mf	10	10	Mesh factor
Omega	0[deg/s]	0 rad/s	Angular speed

Instead of creating the geometry, use the **Import** feature.

GEOMETRY 1

Import 1 (imp1)

1. In the **Geometry** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Source** section.
3. Click the **Browse** button. From the menu, choose **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `gyroscope_mixed_formulation_geometry.mphbin`.
5. Click **Import**.
6. Click the **Go to XY View** button in the **Graphics** toolbar.
7. Click the **Orthographic Projection** button in the **Graphics** toolbar.
8. Click the **Wireframe Rendering** button in the **Graphics** toolbar.



Define selections for the electrodes and other boundaries and domains. This will make specifying the material models and physics interface settings easier.

DEFINITIONS

Base

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

2 In the **Settings** window for **Box**, type `Base` 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 `-t_anchor`.

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

Bottom electrode

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

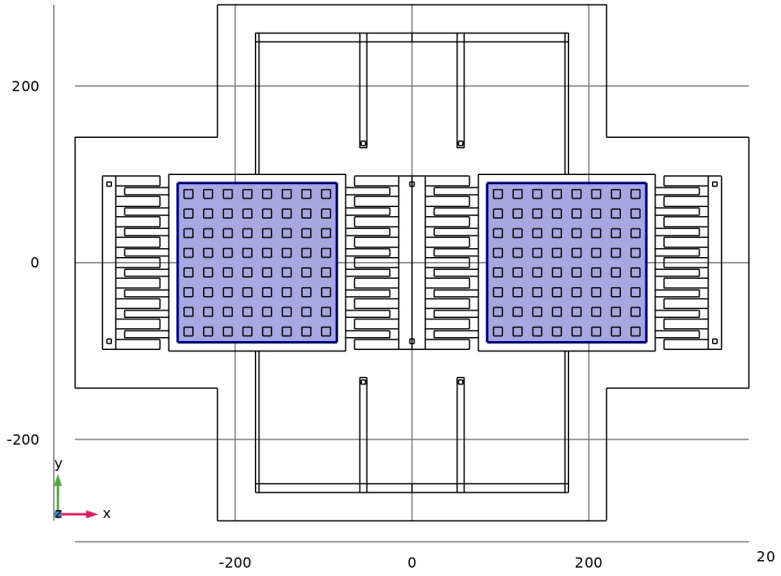
2 In the **Settings** window for **Explicit**, type `Bottom electrode` in the **Label** text field.

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 `146 813` in the **Selection** text field.

6 Click **OK**.



Ground

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

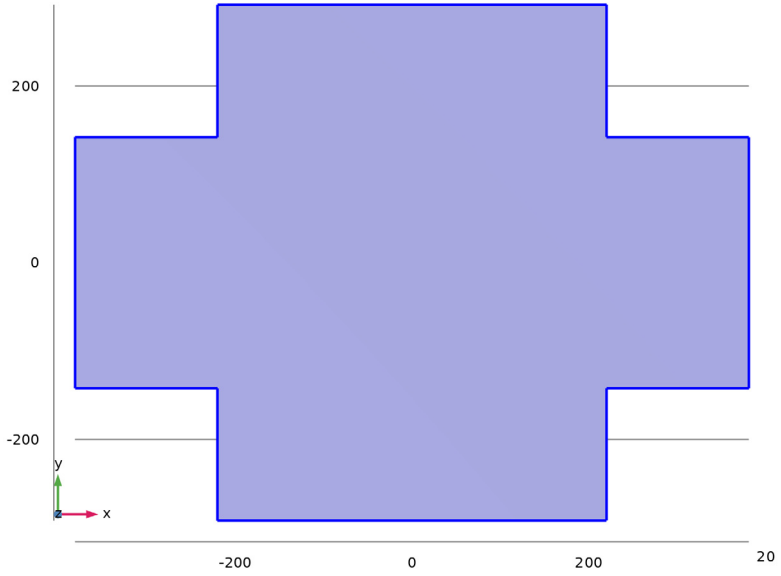
2 In the **Settings** window for **Explicit**, type `Ground` in the **Label** text field.

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 `1 2 4 5 216 217 218 219 1036 1037 1038` in the **Selection** text field.

6 Click **OK**.



Mass

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

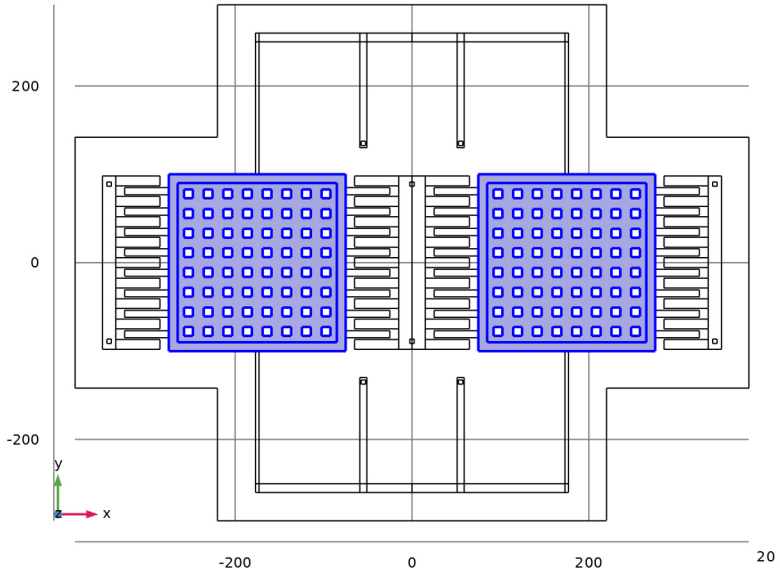
2 In the **Settings** window for **Explicit**, type `Mass` in the **Label** text field.

3 Click the **Select Box** button in the **Graphics** toolbar.

4 Select Domains 22 and 23 only.

5 Click the **Select Box** button in the **Graphics** toolbar.

6 Select Domains 22, 23, 88, and 89 only.



Rotor

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

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

3 Click the **Select Box** button in the **Graphics** toolbar.

4 Select Domains 14–21 only.

5 Click the **Select Box** button in the **Graphics** toolbar.

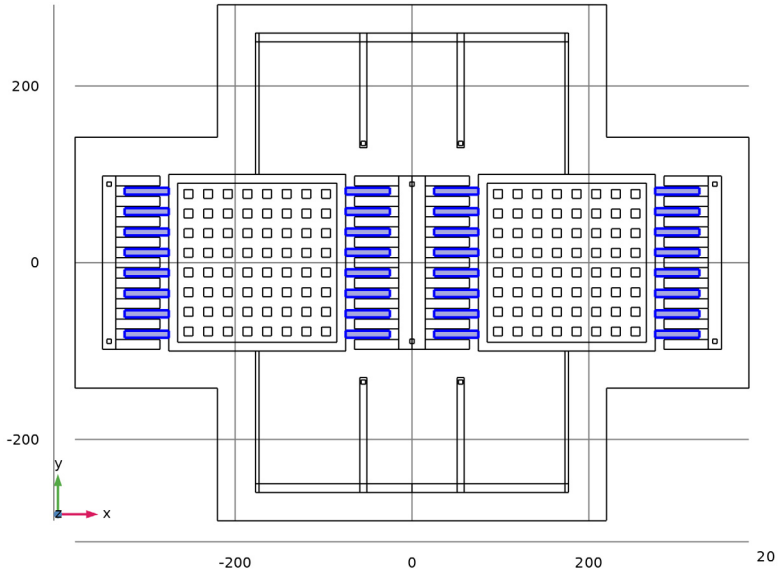
6 Select Domains 14–21 and 30–37 only.

7 Click the **Select Box** button in the **Graphics** toolbar.

8 Select Domains 14–21, 30–37, and 72–79 only.

9 Click the **Select Box** button in the **Graphics** toolbar.

10 Select Domains 14–21, 30–37, 72–79, and 94–101 only.



Outer stator

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

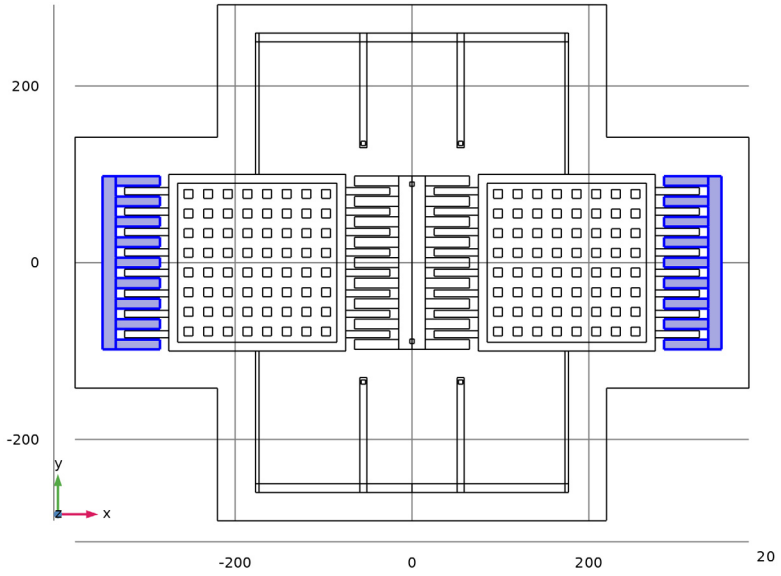
2 In the **Settings** window for **Explicit**, type `Outer stator` in the **Label** text field.

3 Click the **Select Box** button in the **Graphics** toolbar.

4 Select Domains 2–13 only.

5 Click the **Select Box** button in the **Graphics** toolbar.

6 Select Domains 2–13 and 102–113 only.



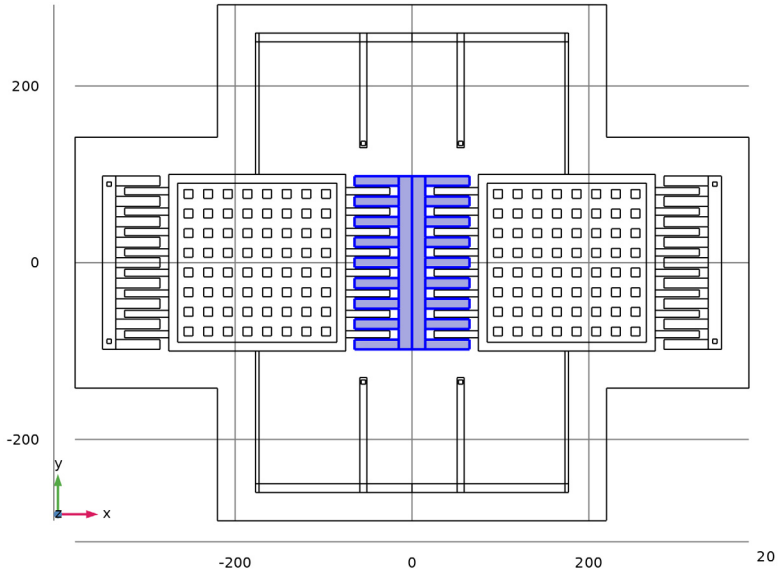
Center stator

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

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

3 Click the **Select Box** button in the **Graphics** toolbar.

4 Select Domains 38–46, 55–57, 59–61, and 63–71 only.



Spring

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

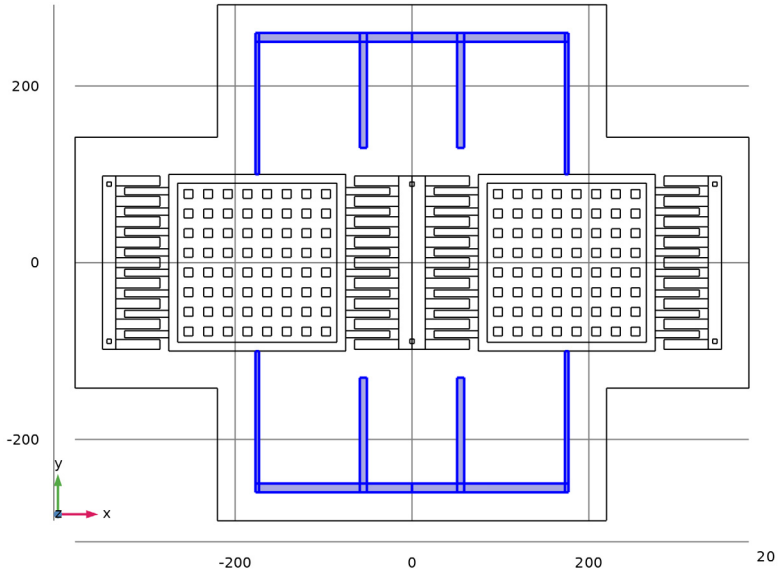
2 In the **Settings** window for **Explicit**, type `Spring` in the **Label** text field.

3 Click the **Select Box** button in the **Graphics** toolbar.

4 Select Domains 26, 27, 29, 49, 50, 52, 54, 62, 82, 83, 85, 87, 92, and 93 only.

5 Click the **Select Box** button in the **Graphics** toolbar.

6 Select Domains 24–29, 47–54, 58, 62, 80–87, and 90–93 only.



All domain

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

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

3 Locate the **Input Entities** section. Select the **All domains** checkbox.

Stator

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

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

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **Outer stator** and **Center stator**.

5 Click **OK**.

Rotor, stator

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

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

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **Rotor**, **Outer stator**, and **Center stator**.

5 Click **OK**.

Polysilicon

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

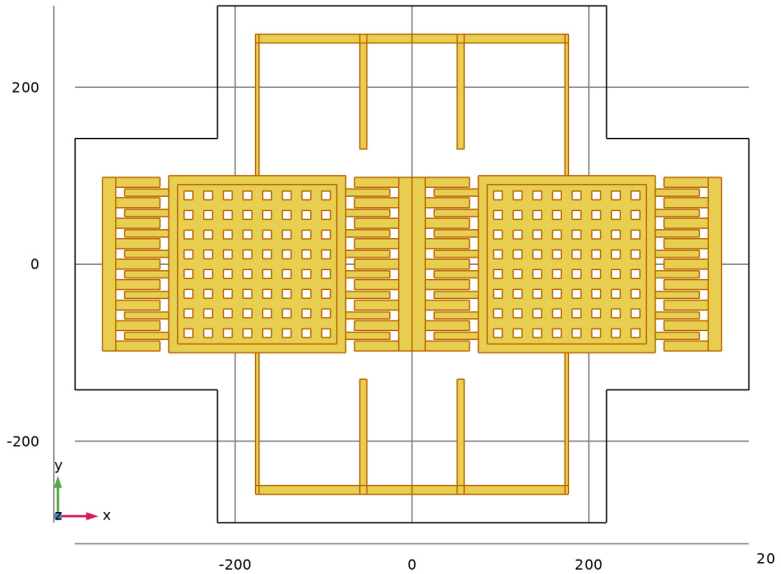
2 In the **Settings** window for **Union**, locate the **Input Entities** section.

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **Mass**, **Rotor**, **Outer stator**, **Center stator**, and **Spring**.

5 Click **OK**.

6 In the **Settings** window for **Union**, type `Polysilicon` in the **Label** text field.



Mass, rotor, spring

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

2 In the **Settings** window for **Union**, type `Mass, rotor, spring` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **Mass**, **Rotor**, and **Spring**.

5 Click **OK**.

Free space

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

2 In the **Settings** window for **Difference**, type `Free space` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.

4 In the **Add** dialog, select **All domain** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click + **Add**.

8 In the **Add** dialog, select **Polysilicon** in the **Selections to subtract** list.

9 Click **OK**.

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `164` in the **Selection** text field.

6 Click **OK**.

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `759` in the **Selection** text field.

6 Click **OK**.

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 **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `149 816` in the **Selection** text field.

6 Click **OK**.

Average 1 (aveop1)

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `149 816` in the **Selection** text field.

6 Click **OK**.

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

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 - Polycrystalline silicon**.

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.

MATERIALS

Si - Polycrystalline silicon (mat1)

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

2 From the **Selection** list, choose **Polysilicon**.

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 **Free space**.

Symmetry/Roller 1

In the **Model Builder** window, right-click **Symmetry/Roller 1** and choose **Disable**.

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

ELECTROSTATICS (ES)

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

2 In the **Settings** window for **Electrostatics**, click to expand the **Discretization** section.

Charge Conservation in Solids 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electrostatics (es)** click **Charge Conservation in Solids 1**.

2 In the **Settings** window for **Charge Conservation in Solids**, locate the **Domain Selection** section.

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

Domain Terminal 1

1 In the **Physics** toolbar, click **Domains** and choose **Domain Terminal**.

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

3 From the **Selection** list, choose **Mass, rotor, spring**.

4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.

5 In the V_0 text field, type `0`.

Domain Terminal 2

1 Right-click **Domain Terminal 1** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Outer stator**.

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

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 `V_ac`.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Outer stator**.

Domain Terminal 3

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electrostatics (es)** right-click **Domain Terminal 2** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Center stator**.

Harmonic Perturbation 1

1 In the **Model Builder** window, expand the **Domain Terminal 3** node, then click **Harmonic Perturbation 1**.

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

3 From the **Selection** list, choose **Center stator**.

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

Boundary Terminal 4

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

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

3 From the **Selection** list, choose **Bottom electrode**.

4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.

5 In the V_0 text field, type `Vbase`.

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**.

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

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

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

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** 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`.

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 **Base**.

Rotating Frame 1

1 In the **Physics** toolbar, click **Domains** 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 From the **Axis of rotation** list, choose **y-axis**.

5 Locate the **Frame Acceleration Effect** section. Select the **Coriolis force** checkbox.

Create the mesh for the model.

MESH 1

Free Tetrahedral 1

1 In the **Mesh** toolbar, click  **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Entire geometry**.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

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

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

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

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `mf`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `mf/5`.

Size 2

1 Right-click **Size 1** and choose **Duplicate**.

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

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

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `mf*4`.

5 In the **Minimum element size** text field, type `mf*4/5`.

Size 3

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

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

3 From the **Selection** list, choose **Rotor, stator**.

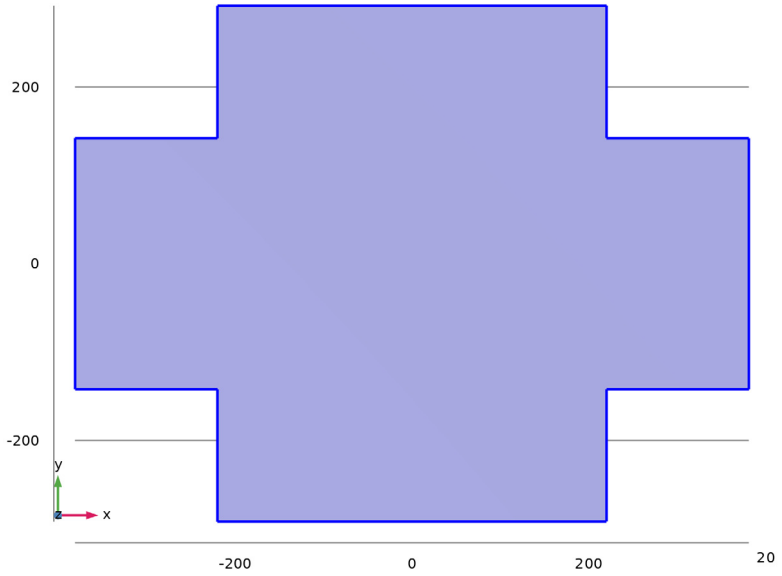
Size 4

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Free space**.



Compute the **Stationary** study.

STUDY 1

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

RESULTS

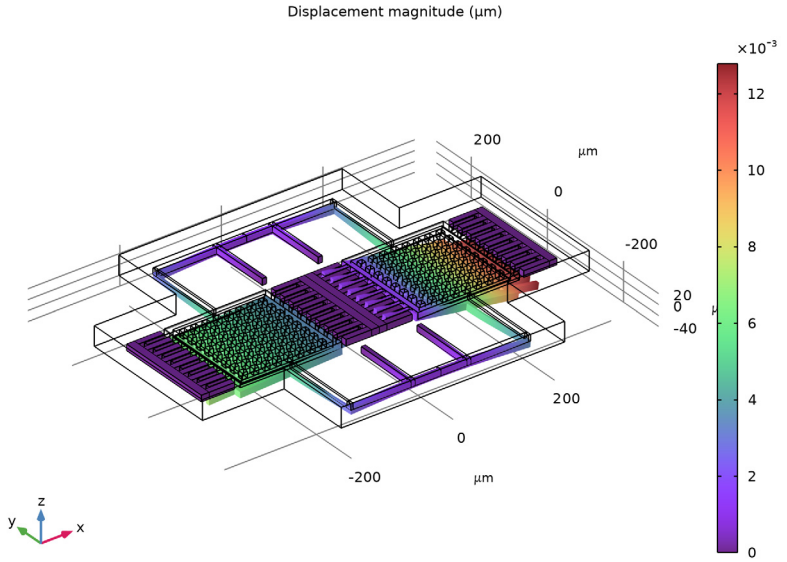
Displacement (solid)

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

Deformation 1

1 In the **Model Builder** window, expand the **Displacement (solid)** node.

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



Set up an **Eigenfrequency Prestressed** study to search for an eigenfrequency around 36 kHz.

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** > **Solid Mechanics** > **Eigenfrequency, 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

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

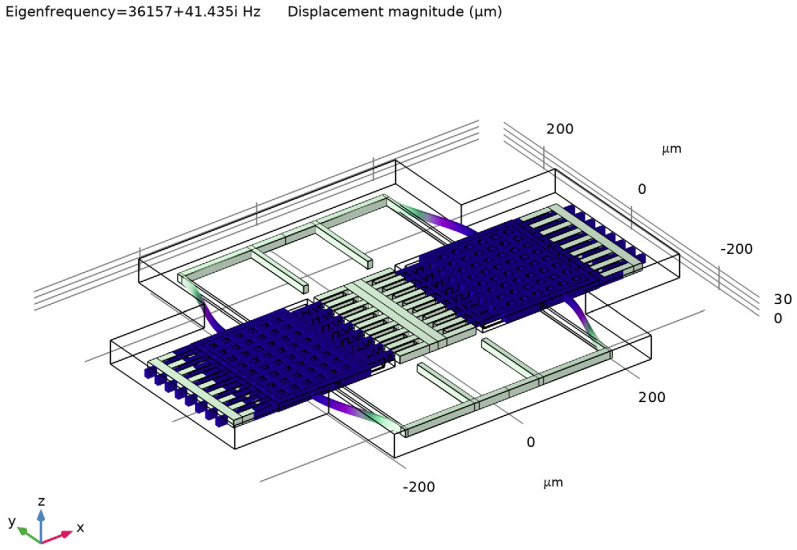
RESULTS

Mode Shape (solid)

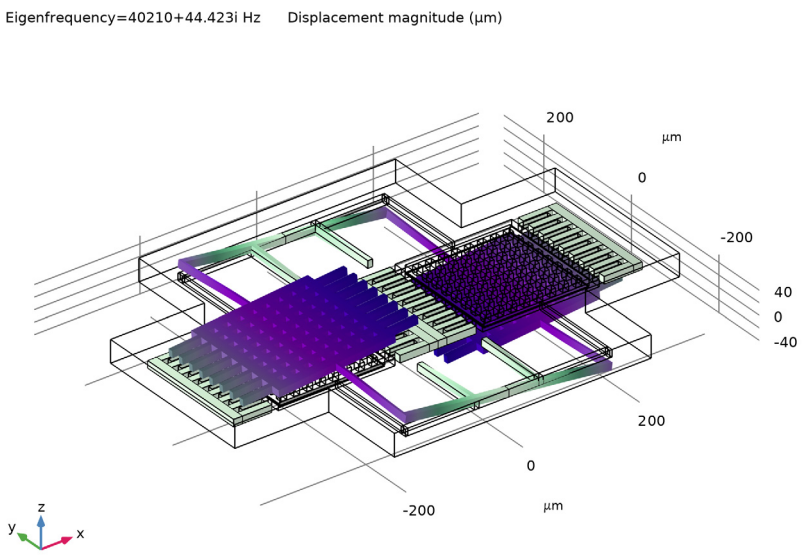
1 In the **Settings** window for **3D Plot Group**, click **Plot Next**.

2 Click → **Plot Next**.

3 Click → **Plot Next**.



4 Click → **Plot Next**.



Set up a **Frequency Domain Perturbation** study at the drive frequency.

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 **Preset Studies for Selected Physics Interfaces** > **Solid Mechanics** > **Frequency Domain, Prestressed**.

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 3

Step 2: Frequency-Domain Perturbation

In the **Settings** window for **Frequency-Domain Perturbation**, locate the **Study Settings** section and type in the value of the drive frequency.

1 In the **Settings** window for **Frequency-Domain Perturbation**, click to expand the **Study Extensions** section.

2 Select the **Auxiliary sweep** checkbox.

3 Click + **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Omega (Angular speed)	0 100	deg/s

For the study to solve, set null-space function to **Sparse**.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.

3 In the **Model Builder** window, expand the **Study 3** > **Solver Configurations** > **Solution 4 (sol4)** > **Stationary Solver 2** node, then click **Advanced**.

4 In the **Settings** window for **Advanced**, locate the **General** section.

5 From the **Null-space function** list, choose **Sparse**.

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

RESULTS

Imag X displacement - Drive mode amplitude

1 In the **Model Builder** window, right-click **Displacement (solid) 1** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Imag X displacement - Drive mode amplitude` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Drive mode amplitude (μm)`.

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Material (X, Y, Z)**.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Volume 1

1 In the **Model Builder** window, expand the **Imag X displacement - Drive mode amplitude** node, then click **Volume 1**.

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

3 In the **Expression** text field, type `imag(u)`.

Deformation 1

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

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

3 In the **x-component** text field, type `imag(u)`.

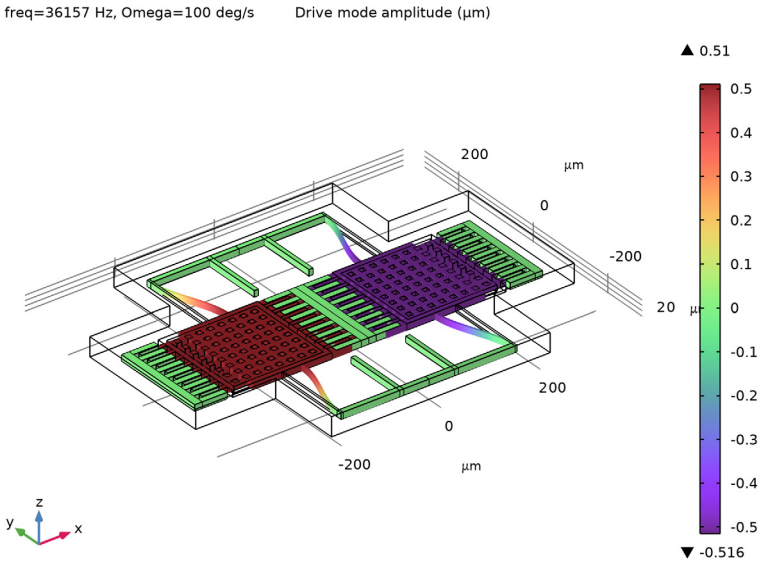
4 In the **y-component** text field, type `imag(v)`.

5 In the **z-component** text field, type `imag(w)`.

Imag X displacement - Drive mode amplitude

1 In the **Model Builder** window, under **Results** click **Imag X displacement - Drive mode amplitude**.

2 In the **Imag X displacement - Drive mode amplitude** toolbar, click **Plot**.



Real Z displacement - No rotation

1 In the **Model Builder** window, right-click **Displacement (solid) 1** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Real Z displacement - No rotation` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (Omega (deg/s))** list, choose **0**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Material (X, Y, Z)**.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Volume 1

1 In the **Model Builder** window, expand the **Real Z displacement - No rotation** node, then click **Volume 1**.

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

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

Deformation 1

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

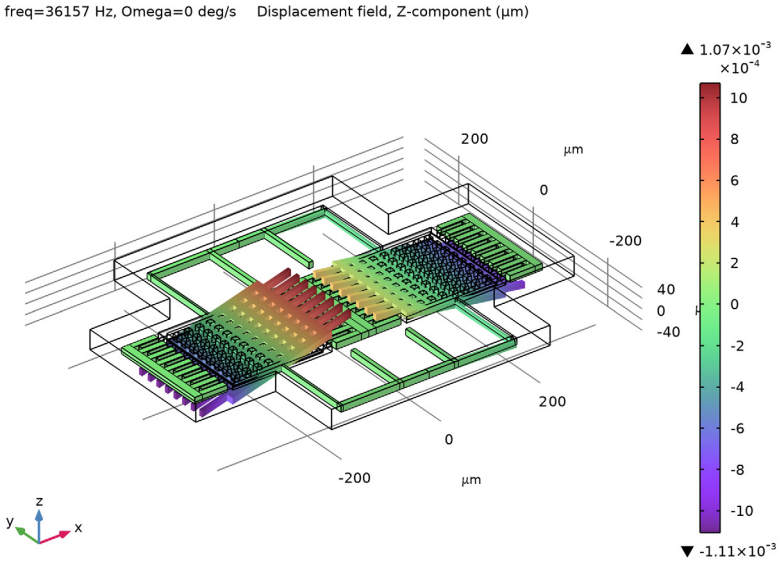
2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **z-component** text field, type `w*1e3`.

Real Z displacement - No rotation

1 In the **Model Builder** window, under **Results** click **Real Z displacement - No rotation**.

2 In the **Real Z displacement - No rotation** toolbar, click **Plot**.



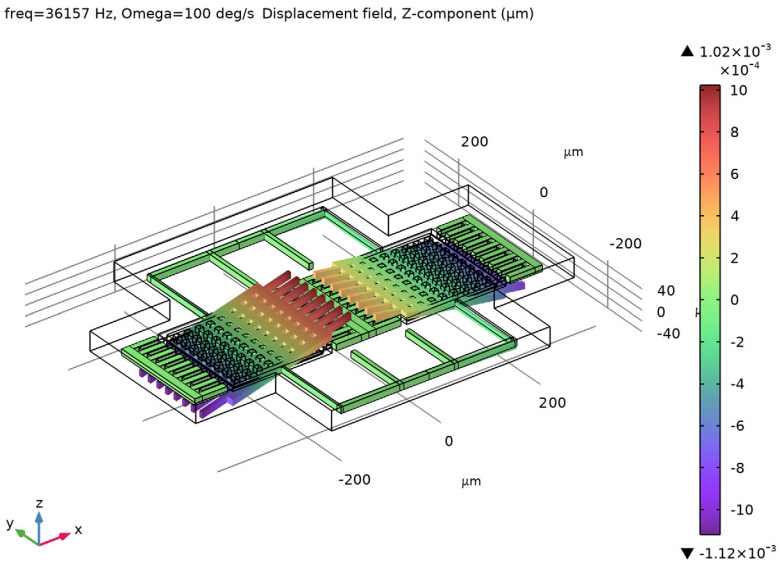
Real Z displacement - Rotation

1 Right-click **Real Z displacement - No rotation** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Real Z displacement - Rotation` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (Omega (deg/s))** list, choose **100**.

4 In the **Real Z displacement - Rotation** toolbar, click **Plot**.



Use **Join** dataset to calculate the very small difference between rotation and no rotation.

Join 1

1 In the **Results** toolbar, click **More Datasets** and choose **Join**.

2 In the **Settings** window for **Join**, locate the **Data 1** section.

3 From the **Data** list, choose **Study 3/Solution 4 (sol4)**.

4 From the **Solutions** list, choose **One**.

5 Locate the **Data 2** section. From the **Data** list, choose **Study 3/Solution 4 (sol4)**.

6 From the **Solutions** list, choose **One**.

7 From the **Parameter value (freq (Hz),Omega (deg/s))** list, choose **1: freq=36157 Hz, Omega=0 deg/s**.

Real Z displacement - Net sense signal

1 In the **Model Builder** window, right-click **Real Z displacement - Rotation** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Real Z displacement - Net sense signal` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Join 1**.

4 In the **Real Z displacement - Net sense signal** toolbar, click **Plot**.