Created in COMSOL Multiphysics 6.2

Capacitive Micromachined Ultrasonic Transducer

Introduction

A capacitive micromachined ultrasonic transducer (CMUT) is a microscale receiver that converts ultrasound to an electrical signal for high-resolution imaging applications. This model analyzes a CMUT design with optimized force-displacement characteristics for increased efficiency. An important metric to improve is the ***displacement uniformity factor***, which can be calculated using a Frequency Domain, Prestressed study. This particular design improves upon a well-established medical imaging technology dominated by piezoelectric transducers and promises miniaturization and higher resolution.

Model Definition

The device is based on the work reported in Ref. 1. It can be fabricated using well-established 0.35 μm CMOS-MEMS process technology. On a silicon substrate, alternating layers of dielectric (silicon dioxide) and metal (aluminum) are deposited and lithographically patterned in sequence. All dielectric and metal thicknesses are 1.0 μm and 0.64 μm, respectively, except for the topmost metal layer which is 1.28 μm. There are three dielectric layers, four metal layers, and one final passivation layer (silicon nitride) which is 1.0 μm. The passivation layer serves as a membrane that can respond to external pressure and protects the device from the external environment. After the deposition steps are completed, a selective and isotropic etching process removes the sacrificial material to release the membrane and the suspended mass.

The model is a 3D model of one quadrant of the device. Because of its four-fold symmetry and the analyses needed, it is not necessary to explicitly model the entire device. Figure 1 shows the model comprising the dielectric, metal, and passivation layers. The model dimensions are:

- Length: 31.75 μm (half the length of the device)
- Height: 6.56 μm

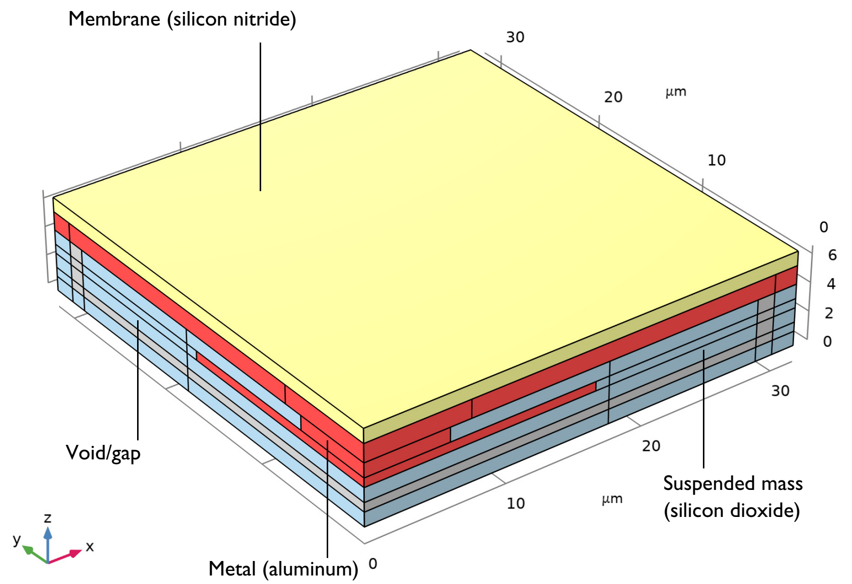


Figure 1: The model of one quadrant of the device. This 3D plot shows the materials used in the model. The membrane is silicon nitride (yellow), the top electrode and supports are metal (red), and the suspended mass is silicon dioxide (blue). The void or gap is shown in gray.

Figure 2 shows the boundary conditions. The ground plane is represented by a Ground boundary condition. The first layer of silicon dioxide is deposited over the ground plane. The gap is defined by the thickness of the sacrificial metal layer. Embedded within the suspended mass is the electrode plate connected to the silicon nitride membrane/ passivation layer. As the membrane is deflected by an external pressure differential, the gap between the electrodes changes. The external stimulation is represented by a Boundary Load boundary condition using a Harmonic Perturbation. This condition is applied to the top surface of the membrane. With small bias of 5 V between the electrodes, the change in capacitance will be reflected in the output current.

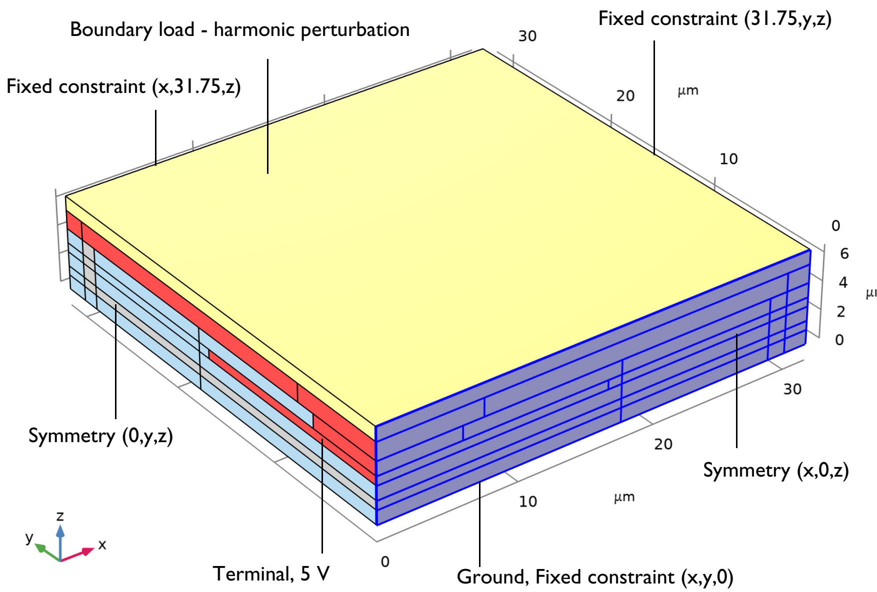


Figure 2: This 3D plot shows boundary conditions used in the model. At the bottom surface of the model is Ground plate and top electrode is set to 5 V. Applied to the top surface is the Boundary Load using a Harmonic Perturbation.

The work reported in Ref. 1 aimed to increase amplified output voltage, V_{out}, of the CMUT as given by

$$V_{\mathrm{out}} = \left[\frac{C_{\mathrm{CMUT}}}{h_{1\mathrm{MD}}/\varepsilon_{\mathrm{r}} + d_{\mathrm{o}}} V_{\mathrm{P}}\right] \cdot \frac{A_{\mathrm{BF}}}{C_{\mathrm{F}}} \cdot \Delta d_{\mathrm{avg}}$$

where C_{CMUT} is the static capacitance of the device, C_{F} is the capacitance of the CI, A_{BF} is the gain of the buffer circuit, d_{o} is the capacitor gap, $h_{1\mathrm{MD}}$ is the total thickness of the dielectric between the electrodes, ε_{r} is the permittivity of the dielectric, and Δd_{avg} is the average displacement of the electrode. The new design aims to increase V_{out} while keeping the same process technology so that yield can be maintained. This means that the layer thickness (d_{o} , $h_{1\mathrm{MD}}$) and the material properties (ε_{r}) are unchanged as are the external buffer circuit parameters (C_{F}, A_{BF}). Δd_{avg} is the only parameter that can be changed through mask design; it increases when the electrode curvature is reduced and the suspended mass design reduces the electrode curvature. The improvement is reflected in the CMUT displacement uniformity factor, F, given by

$$F = \Delta d_{\mathrm{avg}} / \Delta d_{\max}$$

which can be calculated using Average and Minimum coupling operators. A key feature in this design is that the suspended mass is only connected to the membrane where metal supports overlap. This means that the top electrode is partly decoupled from the membrane so the curvature on the top electrode is reduced, which in turn increases the average displacement over the electrode area.

The model geometry is fully parameterized to allow for easy changes in the device structure to target a specific resonant frequency. The model solves a multiphysics problem involving electromechanical force and calculates the frequency response of the device using a Frequency Domain, Prestressed study. The device is connected to an external circuit using the Electrical Circuit interface for calculating the output current. The model uses built-in operators to compute average and minimum displacements over the electrode surface to calculate the displacement uniformity factor.

Results and Discussion

Figure 3 shows the frequency response over the range1.7–2.5 MHz to measure electrode displacement and output current.

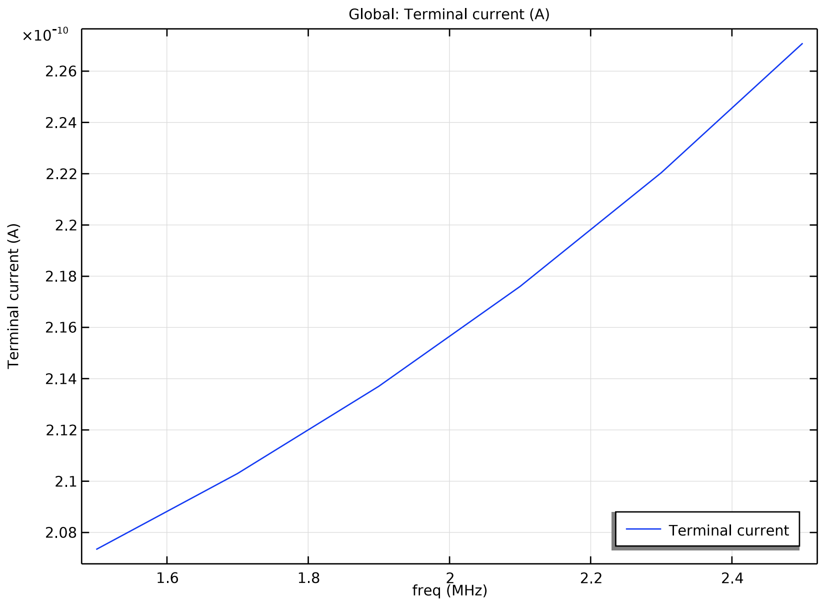


Figure 3: Current versus frequency.

Figure 4 shows profile of the top-electrode cross section as a function of the frequency. The maximum displacement is at the center of the device and is 13 nm at 2.1 MHz.

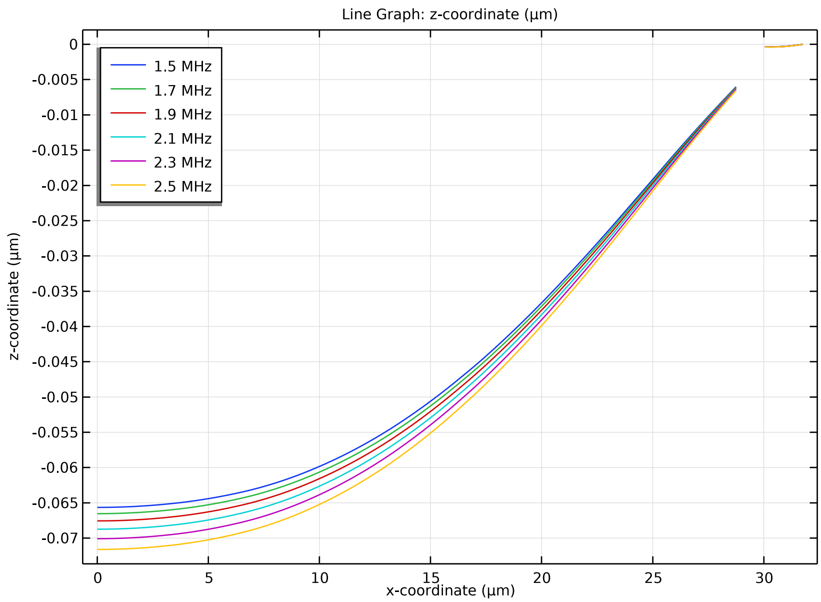


Figure 4: Profile of the top electrode s a function of frequency. Maximum displacement is at the center of the device and is 13 nm at 2.1 MHz.

Figure 5shows the z-displacement of the structure with a harmonic perturbation at 1.7 MHz. As expected, the displacement is maximal on the symmetry planes at the center of the device. Due to the design, the corners of the suspended mass along the diagonal of the device also have large displacement, leading to the lobe extending to the corner of the device. The conventional device would have a circular pattern at the center of the device.

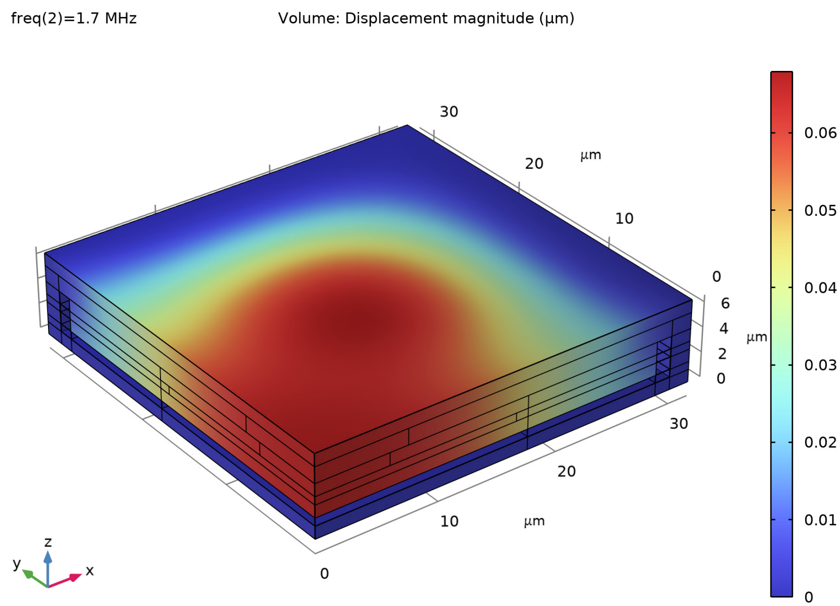


Figure 5: The z-displacement of the structure with a harmonic perturbation at 1.7 MHz.

The model uses an Eigenfrequency study to determine the resonant frequency. The eigenfrequency of 7.5011 MHz matches the reported value of 7.52 MHz (Ref. 1).

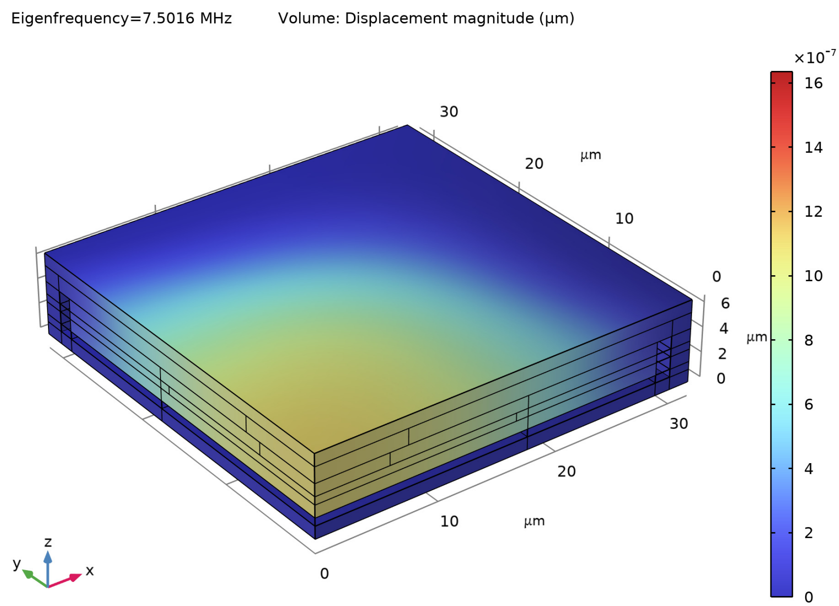


Figure 6: Profile of the top electrode s a function of frequency. Maximum displacement is at the center of the device and is 13 nm at 2.1 MHz.

According Ref. 1, the ***F*** value for a conventional CMUT is 0.6 so the piston design improves the output of the device by about 40%.

Reference

1. C. Chou,P. Chen, H. Wu, T. Hsu, and M. Li, "Piston-Shaped CMOS-MEMS CMUT Front-End Featuring Force-Displacement Transduction Enhancement," *Proceedings of the 21st International Conference on Solid-State Sensors, Actuators and Microsystems (Transducers)*, pp. 26–29, 2021.

Application Library path: `MEMS_Module/Sensors/capacitive_micromachined_ultrasonic_transducer`

Modeling Instructions

Start by creating a new 3D model with Electromechanics and Electrical Circuit interfaces.

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 **AC/DC>Electromagnetics and Mechanics>Electromechanics>Electromechanics**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Solid Mechanics>Frequency Domain, Prestressed**.
8 Click **Done**.

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
l	63.5[um]	6.35E-5 m	Length of device
l_et	33[um]	3.3E-5 m	Length of top electrode
l_eb	35[um]	3.5E-5 m	Length of bottom electrode
t_ox	1[um]	1E-6 m	Thickness of oxide
t_m2	0.64[um]	6.4E-7 m	Thickness of sacrificial metal, M2

Name	Expression	Value	Description
t_m3	0.64[um]	6.4E-7 m	Thickness of top electrode, M3
t_m4	1.28[um]	1.28E-6 m	Thickness of support metal, M4
t_w	3.4[um]	3.4E-6 m	Thickness of wall
w_ox	11.25[um]	1.125E-5 m	Width of oxide around top electrode
l_v43	12[um]	1.2E-5 m	Length of via 4-3
l_m4	15[um]	1.5E-5 m	Length of support metal
w_b	3.6[um]	3.6E-6 m	Width of support beam
t_np	1[um]	1E-6 m	Thickness of nitride passivation layer
p_max	1[MPa]	1E6 Pa	Maximum pressure
R_load	1[Gohm]	1E9 Ω	Load resistance
V_a	5[V]	5 V	Applied voltage

Use microns as the geometry unit.

GEOMETRY 1

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**.

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**.

Device

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

2 In the **Settings** window for **Square**, type `Device` in the **Label** text field.

3 Locate the **Size** section. In the **Side length** text field, type `l/2`.

Piston

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

2 In the **Settings** window for **Square**, type `Piston` in the **Label** text field.

3 Locate the **Size** section. In the **Side length** text field, type `l_eb/2+w_ox`.

Inside Wall

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

2 In the **Settings** window for **Square**, type `Inside Wall` in the **Label** text field.

3 Locate the **Size** section. In the **Side length** text field, type `l/2-t_w/2`.

Bottom Electrode

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

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

3 Locate the **Size** section. In the **Side length** text field, type `l_eb/2`.

Oxide 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, type `Oxide 1` in the **Label** text field.

3 Locate the **Distances** section. In the table, enter the following settings:

Distances (µm)
t_ox

Metal 2

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type `Metal 2` in the **Label** text field.

3 Locate the **General** section. From the **Extrude from** list, choose **Faces**.

4 On the object **ext1**, select Boundaries 4, 8, 12, and 16 only.

5 From the **Input object handling** list, choose **Keep**.

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

Distances (µm)
t_m2

Oxide 2

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type `Oxide 2` in the **Label** text field.

3 Locate the **General** section. From the **Extrude from** list, choose **Faces**.

4 On the object **ext2**, select Boundaries 4, 8, 12, and 16 only.

5 From the **Input object handling** list, choose **Keep**.

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

Distances (µm)
`t_ox`

Metal 3

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type `Metal 3` in the **Label** text field.

3 Locate the **General** section. From the **Extrude from** list, choose **Faces**.

4 On the object **ext3**, select Boundaries 4, 8, 12, and 16 only.

5 From the **Input object handling** list, choose **Keep**.

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

Distances (µm)
`t_m3`

Oxide 3

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type `Oxide 3` in the **Label** text field.

3 Locate the **General** section. From the **Extrude from** list, choose **Faces**.

4 On the object **ext4**, select Boundaries 4, 8, 12, and 16 only.

5 From the **Input object handling** list, choose **Keep**.

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

Distances (µm)
`t_ox`

Work Plane 2 (wp2)

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

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

3 In the **z-coordinate** text field, type `t_ox+t_m2+t_ox+t_m3+t_ox`.

Work Plane 2 (wp2)>Plane Geometry

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

Support, Center

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

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

3 Locate the **Size** section. In the **Side length** text field, type `l_m4/2`.

Support, x

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

2 In the **Settings** window for **Rectangle**, type `Support, x` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `l/2-l_m4/2`.

4 In the **Height** text field, type `w_b/2`.

5 Locate the **Position** section. In the **xw** text field, type `l_m4/2`.

Support, y

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

2 In the **Settings** window for **Rectangle**, type `Support, y` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_b/2`.

4 In the **Height** text field, type `l/2-l_m4/2`.

5 Locate the **Position** section. In the **yw** text field, type `l_m4/2`.

Device, Upper

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

2 In the **Settings** window for **Square**, type `Device, Upper` in the **Label** text field.

3 Locate the **Size** section. In the **Side length** text field, type `l/2`.

Inside Wall, Upper

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

2 In the **Settings** window for **Square**, type `Inside Wall, Upper` in the **Label** text field.

3 Locate the **Size** section. In the **Side length** text field, type `l/2-t_w/2`.

Metal 4

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, type `Metal 4` in the **Label** text field.

3 Locate the **General** section. From the **Input object handling** list, choose **Keep**.

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

Distances (µm)
`t_m4`

Nitride

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Nitride` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `l/2`.

4 In the **Depth** text field, type `l/2`.

5 In the **Height** text field, type `t_np`.

6 Locate the **Position** section. In the **z** text field, type `t_ox+t_m2+t_ox+t_m3+t_ox+ t_m4`.

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

8 Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Passivation` in the **Name** text field.

10 Click **OK**.

11 In the **Settings** window for **Block**, locate the **Selections of Resulting Entities** section.

12 From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.

13 Click **Define custom colors**.

14 Set the RGB values to 255, 255, and 155, respectively.

15 Click **Add to custom colors**.

16 Click **Show color palette only** or **OK** on the cross-platform desktop.

Via

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Via` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `l_v43/2`.

4 In the **Depth** text field, type `l_v43/2`.

5 In the **Height** text field, type `t_ox`.

6 Locate the **Position** section. In the **z** text field, type `t_ox+t_m2+t_ox+t_m3`.

Top Electrode

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Top Electrode` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `l_et/2`.

4 In the **Depth** text field, type `l_et/2`.

5 In the **Height** text field, type `t_m3`.

6 Locate the **Position** section. In the **z** text field, type `t_ox+t_m2+t_ox`.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Define the following selections for the Gap, Oxide, and Electrode regions.

DEFINITIONS

Gap

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2 12 17 18 19 20 27` in the **Selection** text field.

5 Click **OK**.

Oxide

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 3 8 10 11 13 14 15 16 21 22 23 24 25 28` in the **Selection** text field.

5 Click **OK**.

Electrode

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4 5 6 9 26 29 30` in the **Selection** text field.

5 Click **OK**.

Edge, Center

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

2 In the **Settings** window for **Box**, type `Edge, Center` 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 **x maximum** text field, type `0`.

5 In the **y maximum** text field, type `0`.

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

Define the Average and Minimum operators needed for evaluation.

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 box, type 12 in the **Selection** text field.

6 Click **OK**.

Minimum 1 (minop1)

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 12 in the **Selection** text field.

6 Click **OK**.

Add materials to the 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>Metals>Al - Aluminum**.

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

MATERIALS

Al - Aluminum (mat1)

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

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

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **MEMS>Insulators>SiO2 - Silicon oxide**.
3. Click **Add to Component** in the window toolbar.

MATERIALS

SiO2 - Silicon oxide (mat2)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Oxide**.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **MEMS>Insulators>Si3N4 - Silicon nitride**.
3. Click **Add to Component** in the window toolbar.

MATERIALS

Si3N4 - Silicon nitride (mat3)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Passivation**.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Liquids and Gases>Gases>Air**.
3. Click **Add to Component** in the window toolbar.
4. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat4)

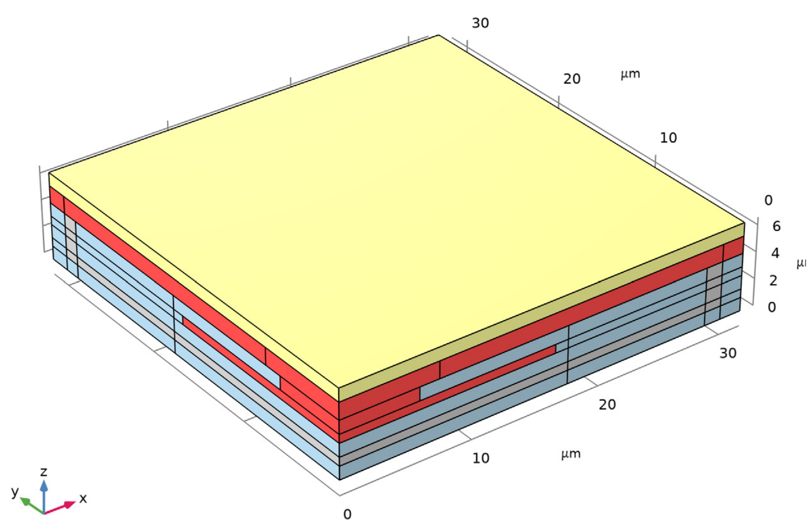
1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

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

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

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



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 Select Boundaries 4, 5, 38, 54, 57, 60, 63, 117, 128, 130, 132, and 134 only.

Set up Electrostatics boundary conditions.

ELECTROSTATICS (ES)

The default **Charge Conservation** feature was set to use solid material type. Add one more **Charge Conservation** feature to represent the nonsolid (air) domains.

Charge Conservation 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose **Charge Conservation**.

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

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

Terminal 1

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

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

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

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

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 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3` in the **Selection** text field.

5 Click **OK**.

Symmetry Plane 1

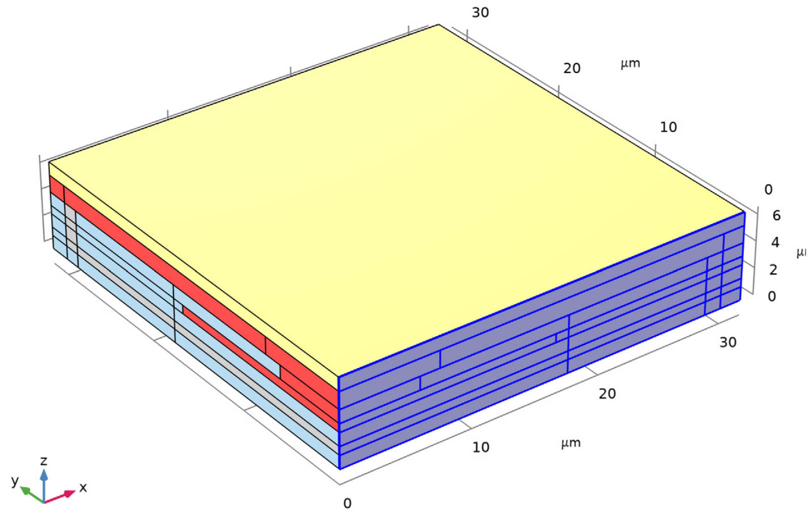
1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2 5 8 11 14 17 20 105 107 113 115 117 119 121 123 126 128 130 132 134 137 139 141 143 145 147` in the **Selection** text field.

5 Click **OK**.



Symmetry Plane 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 1 4 7 10 13 16 19 23 27 31 35 38 41 44 47 51 54 57 60 63 67 70 73 76 79 82 in the **Selection** text field.

5 Click **OK**.

Set up Solid Mechanics boundary conditions.

SOLID MECHANICS (SOLID)

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

Symmetry 1

In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

ELECTROSTATICS (ES)

Symmetry Plane 1

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

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

3 In the list, choose **2**, **5**, **8**, **11**, **14**, **17**, **20**, **105**, **107**, **113**, **115**, **117**, **119**, **121**, **123**, **126**, **128**, **130**, **132**, **134**, **137**, **139**, **141**, **143**, **145**, and **147**.

4 Click **Copy Selection**.

SOLID MECHANICS (SOLID)

Symmetry 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 2 5 8 11 14 17 20 105 107 113 115 117 119 121 123 126 128 130 132 134 137 139 141 143 145 147 in the **Selection** text field.

5 Click **OK**.

Symmetry 2

In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

ELECTROSTATICS (ES)

Symmetry Plane 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Symmetry Plane 2**.

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

3 In the list, choose **1**, **4**, **7**, **10**, **13**, **16**, **19**, **23**, **27**, **31**, **35**, **38**, **41**, **44**, **47**, **51**, **54**, **57**, **60**, **63**, **67**, **70**, **73**, **76**, **79**, and **82**.

4 Click **Copy Selection**.

SOLID MECHANICS (SOLID)

Symmetry 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Symmetry 2**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 4 7 10 13 16 19 23 27 31 35 38 41 44 47 51 54 57 60 63 67 70 73 76 79 82` in the **Selection** text field.

5 Click **OK**.

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 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3 37 53 69 86 87 88 89 90 91 92 103 152 153 154 155 156 157 158 159` in the **Selection** text field.

5 Click **OK**.

Specify a **Boundary Load** feature with **Harmonic Perturbation** to model an excitation by a pressure wave.

Boundary Load 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `22` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Load**, locate the **Force** section.

7 From the **Load type** list, choose **Pressure**.

8 In the p text field, type `p_max`.

9 Right-click **Boundary Load 1** and choose **Harmonic Perturbation**.

Set up Electrical Circuit parameters and specify how the device terminals are connected to the external circuit.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

External I vs. U 1 (IvsU1)

1 In the **Electrical Circuit** toolbar, click **External I vs. U**.

2 In the **Settings** window for **External I vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
n	0

4 Locate the **External Device** section. From the V list, choose **Terminal voltage (es/term1)**.

Resistor 1 (R1)

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `R_load`.

Voltage Source 1 (V1)

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the v_{src} text field, type `V_a`.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Extra coarse**.

Free Tetrahedral 1

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

Set up the steps for a **Frequency Domain, Prestressed** study.

STUDY 1

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

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

3 Clear the **Generate default plots** check box.

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 1** 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 **MHz**.

4 In the **Frequencies** text field, type `range(1.5,0.2,2.5)`.

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

From the results of Study 1, plot the terminal current versus frequency.

RESULTS

Terminal Current vs. Frequency

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Terminal Current vs. Frequency` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

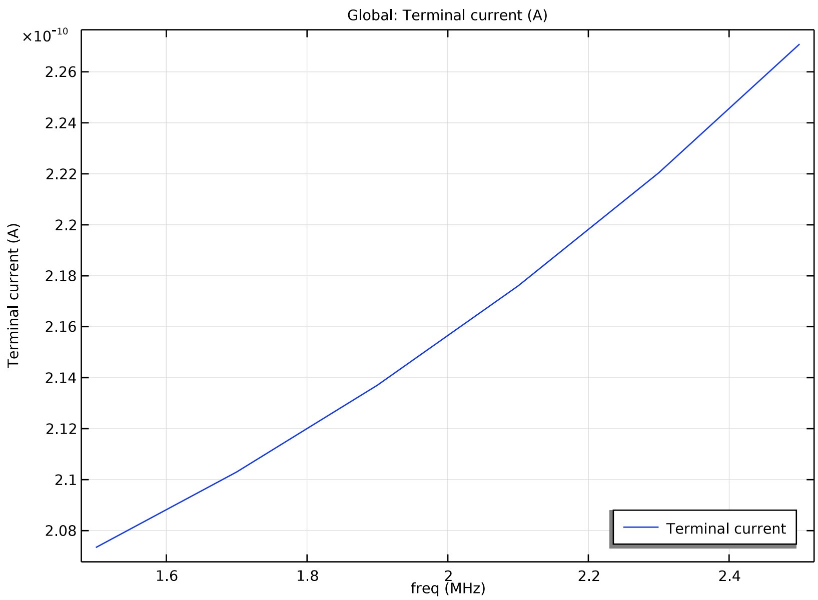
1 Right-click **Terminal Current vs. Frequency** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`es.I0_1`	`A`	`Terminal current`

4 In the **Terminal Current vs. Frequency** toolbar, click **Plot**.



From the results of Study 1, plot the electrode profile.

Electrode Profile

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Electrode Profile` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Electrode Profile** and choose **Line Graph**.

2 Select Edges 12, 144, 159, and 207 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `z`.

5 Select the **Description** check box.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

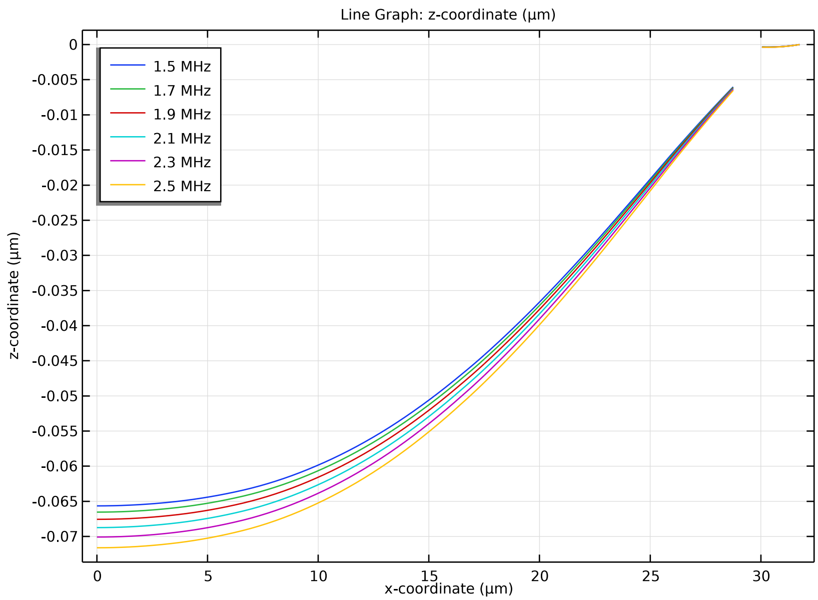
7 In the **Expression** text field, type x.

8 Select the **Description** check box.

9 Locate the **y-Axis Data** section. Select the **Compute differential** check box.

10 Click to expand the **Legends** section. Select the **Show legends** check box.

11 In the **Electrode Profile** toolbar, click **Plot**.



From the results of Study 1, plot the displacement magnitude.

Displacement Magnitude

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Parameter value (freq (MHz))** list, choose **1.7**.

Volume 1

1 Right-click **Displacement Magnitude** and choose **Volume**.

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

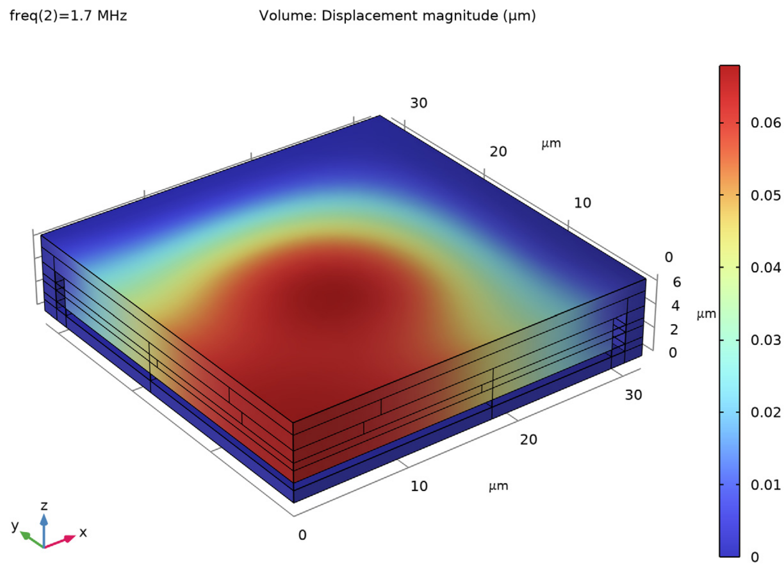
3 Locate the **Expression** section. Select the **Description** check box.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

6 Click **OK**.

7 In the **Displacement Magnitude** toolbar, click **Plot**.



From the results of Study 1, evaluate the displacement uniformity factor.

Displacement Uniformity Factor

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

2 In the **Settings** window for **Evaluation Group**, type `Displacement Uniformity Factor` in the **Label** text field.

Global Evaluation 1

1 Right-click **Displacement Uniformity Factor** and choose **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`aveop1(w)/minop1(w)`	1	`Displacement uniformity factor`

4 In the **Description** text field, type `Displacement uniformity factor`.

5 In the **Displacement Uniformity Factor** toolbar, click **Evaluate**.

Set up an **Eigenfrequency** study to search for an eigenfrequency around 7.5 MHz.

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 Multiphysics>Eigenfrequency**.

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

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

STUDY 2

Step 1: Eigenfrequency

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

2 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `1`.

3 Find the **Elliptic search region** subsection. From the **Unit** list, choose **MHz**.

4 In the **Search for eigenfrequencies around shift** text field, type `7.5`.

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

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

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

From the results of Study 2, plot the eigenmode.

RESULTS

Eigenmode

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

4 From the **Eigenfrequency (MHz)** list, choose **7.5016**.

Volume 1

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

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

3 Locate the **Expression** section. Select the **Description** check box.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

6 Click **OK**.

7 In the **Eigenmode** toolbar, click **Plot**.