Created in COMSOL Multiphysics 6.2

Thin-Film BAW Resonator with Equivalent Circuit

Introduction

This tutorial demonstrates how to

1. Model a thin-film BAW resonator
2. Set up an equivalent-circuit model with lumped parameters
3. Determine the values of the lumped parameters using a Parameter Estimation Study

The tutorial outlines the steps from FEM modeling to deriving a compact circuit model for system-level simulations.

From FEM to Lumped Parameter Model

While FEM can compute eigenfrequencies, eigenmodes, and frequency responses, extending the method to model complex circuits would be computationally costly and inefficient. In circuit design, components are represented by equivalent circuits with lumped parameters. In this tutorial you will create a FEM model of an FBAR in 3D and derive its equivalent circuit model based on the FEM simulation and a Parameter Estimation Study. The FBAR is a building block in circuits used in mobile devices.

Model Definition

FEM Model of the FBAR

The model is based on a five-sided apodized FBAR design for maximizing confinement of vibrational energy. The device is suspended from the anchor points. More information on the structure and operation of such a resonator can be found in the Thin-Film BAW Composite Resonator model documentation. In this tutorial, the FBAR geometry is parameterized following the first table in the Modeling Instructions section. Because of the symmetry of the device structure and the mode of interest, only a 36-degree sector needs to be modeled (see Figure 1), which reduces the computational time significantly. This is a good strategy to use in this particular model where a fine mesh is needed to compute the eigenfrequency and to generate a smooth surface plot of the mode shape. The FBAR is designed to have a series resonance at about 3.25 GHz and comprises a stack of a 0.2 µm silicon nitride supporting layer, a 0.1 µm molybdenum bottom electrode, a 1 µm aluminum nitride piezoelectric layer, and a 0.2 µm aluminum top electrode. The FEM model is used to compute the frequency response of the device using a Frequency Domain study. In the following steps, this FEM data is used as reference or the Experimental data.

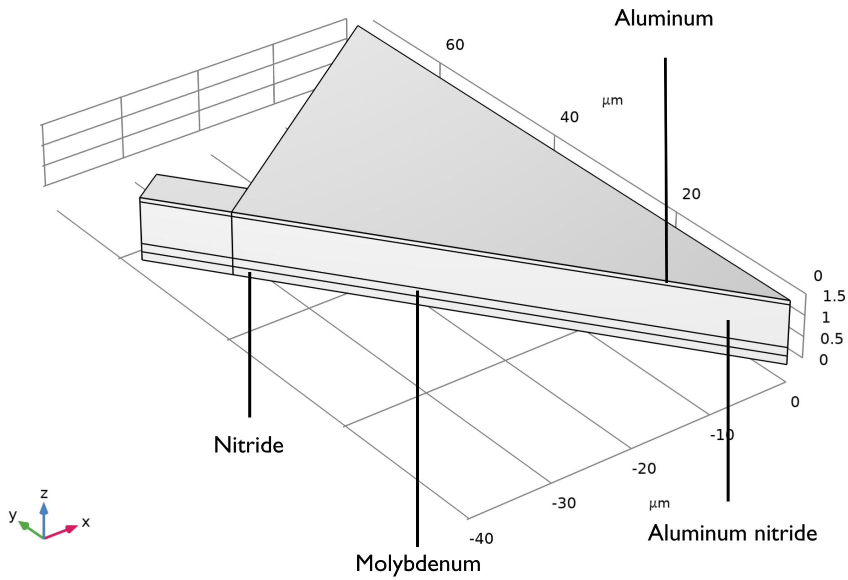


Figure 1: 36-degree sector of the five-sided thin-film BAW resonator.

The Modified Butterworth–Van Dyke Equivalent Circuit Model

An electromechanical device such as the FBAR can be represented by the Modified Butterworth–Van Dyke circuit with six lumped elements illustrated in Figure 2. The circuit is essentially the aluminum nitride capacitance Co in parallel with the series circuit Rm-Lm-Cm (the motional branch) for the piezoelectric effect. The Electrical Circuit interface is used to specify and excite the circuit using the initial values listed in Table 1. These values are estimated using equations in Ref. 1 and are based on the resonance and antiresonance frequencies of the FEM model and the Co value from the device geometry. As in the FEM model, the circuit model is used to compute the frequency response of the device using a Frequency Domain study. With the appropriate initial values, the frequency response from FEM and the circuit model should be close.

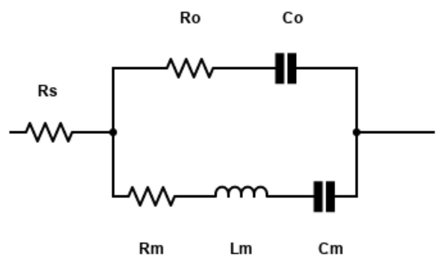


Figure 2: Diagram of the mBVD circuit model.

TABLE 1: INITIAL VALUES USED IN THE CIRCUIT SIMULATION.

Parameter	Value
Cm	62 fF
Lm	39 nH
Co	1 pF
Rs	1 Ω
Ro	500 Ω
Rm	1 Ω

Parameter Estimation Study

To derive the precise values of the model parameters, use a Parameter Estimation study. In the Parameter Estimation study, a model expression is computed from the Frequency Domain study of the circuit model defined by the six parameters, which are varied to fit the experimental data (from FEM). To fit the model expression to the experimental data, Parameter Estimation minimizes an objective function defined as the error between model expression and experimental data. The output of the Parameter Estimation (PE) is the set of circuit parameters that best approximates the frequency response from FEM.

Results and Discussion

Figure 3 shows the absolute value of the terminal current as a function of frequency from the Frequency Domain study between 2.8 and 3.8 GHz. The series resonance frequency at 3.25 GHz corresponds to the thickness-extension mode.

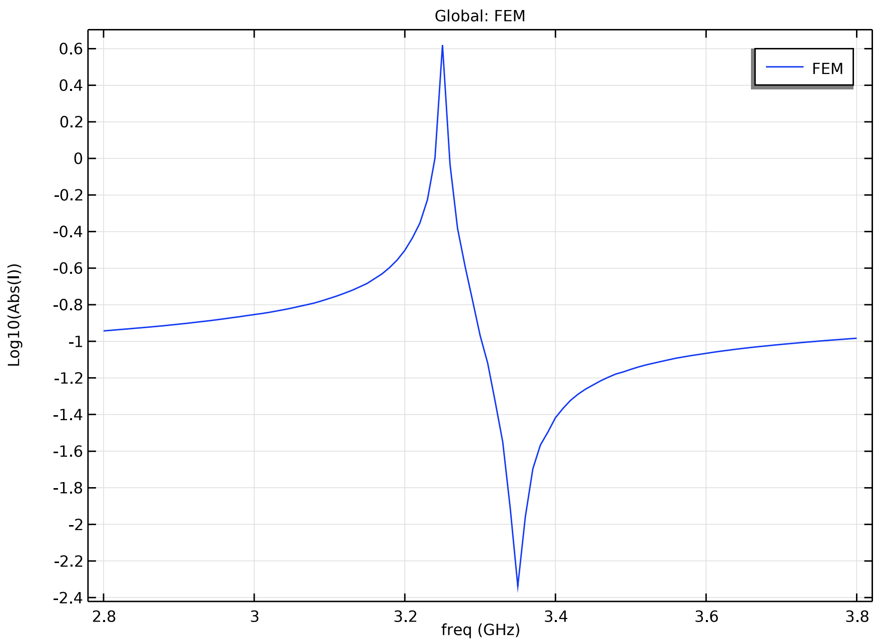


Figure 3: Plot of current versus frequency from FEM simulation showing series resonance at 3.25 GHz of the thickness-extension mode.

A second Frequency Domain study analyzes the frequency response of the circuit as defined by parameter initial values listed in Table 1. Figure 4 shows the plot for the circuit simulation superimposed onto the plot from the FEM simulation.

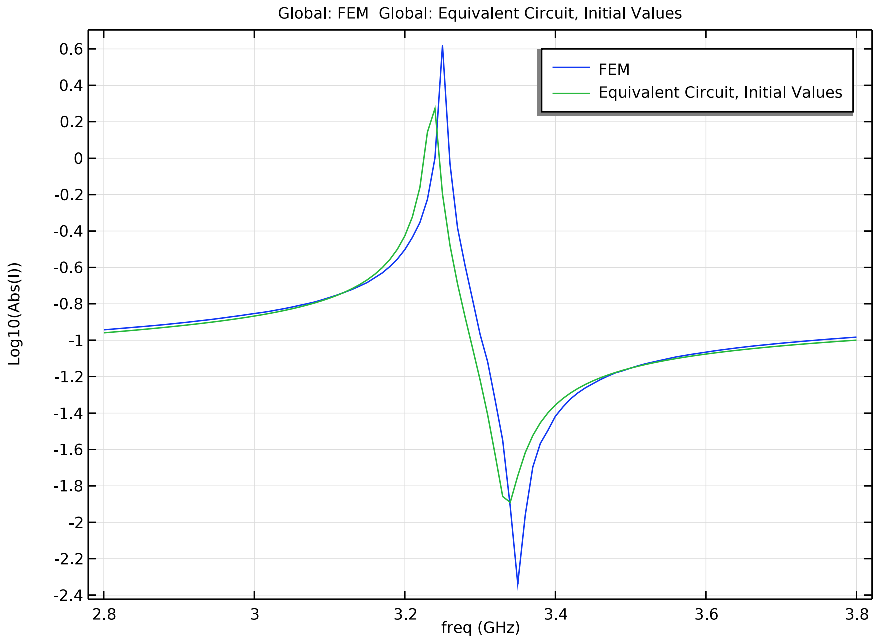


Figure 4: Plot of current versus frequency from FEM simulation and circuit simulation using the parameter initial values.

Figure 5 shows the output of the Parameter Estimation study comparison between experimental data and model expression. The fitted parameters are listed in Table 2.

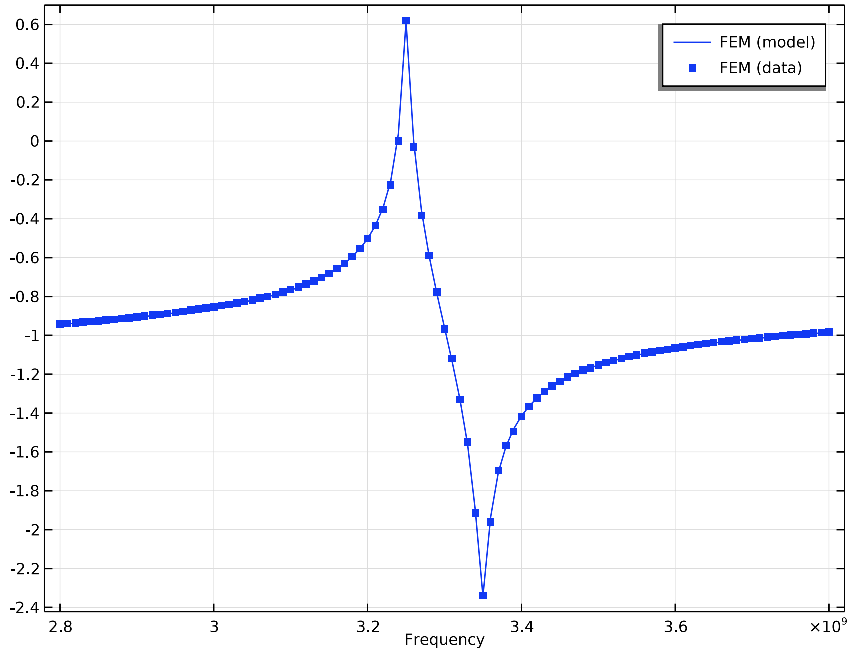


Figure 5: Plot of FEM result and equivalent circuit using final parameters from the Parameter Estimation Study.

TABLE 2: VALUES OF CIRCUIT PARAMETERS COMPUTED FROM PARAMETER ESTIMATION STUDY.

Parameter	**Value**
Cm	64.49 fF
Lm	37.18 nH
Co	1.04 pF
Rs	0.87 Ω
Ro	1340 Ω
Rm	0.325 Ω

Figure 6 is a surface plot of the solid displacement and the shape of the thickness-extension mode.

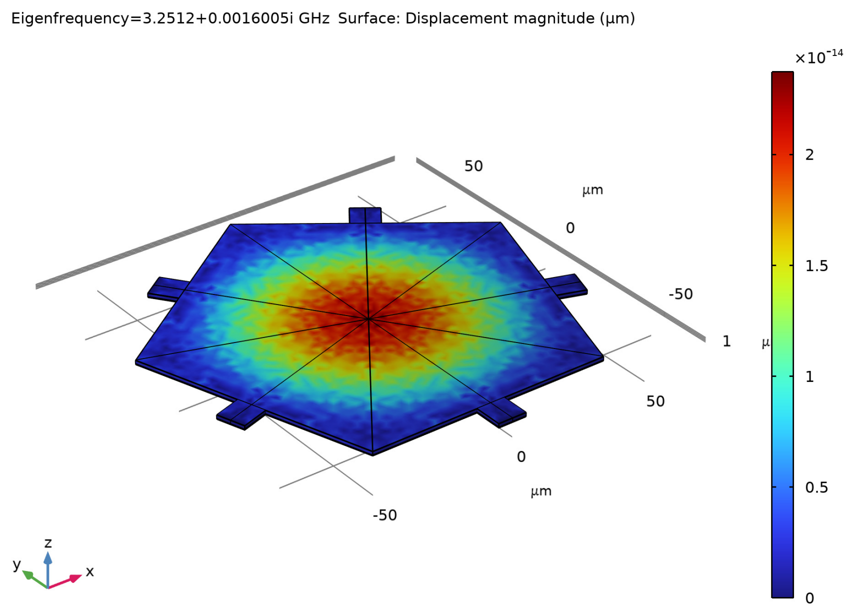


Figure 6: Surface plot showing the pattern of displacement for the thickness-extension mode.

Figure 6 shows the mode shape at 3.25 GHz from the FEM Eigenfrequency study. For high-resonance devices, an Eigenfrequency study often returns many solutions that are very close to the resonance frequency at 3.25 GHz. These are called spurious modes and need to be discarded based on visual inspection. As an alternative to visual verification, you can compute the displacement phase uniformity and plot against frequency. For the true resonance the value will be close to 1 whereas for spurious modes, they will be much smaller than 1. A plot of displacement phase uniformity versus frequency is shown in Figure 7.

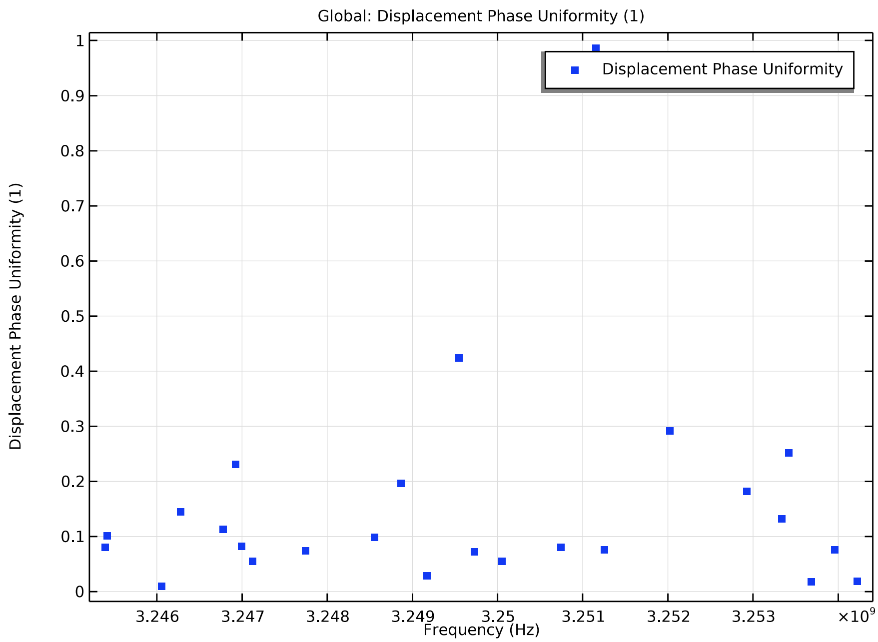


Figure 7: Plot of displacement phase uniformity versus frequency. Its value is close to 1 only at the resonance frequency of 3.25 GHz.

References

1. J. D. Larson, P. D. Bradley, S. Wartenberg and R. C. Ruby, “Modified Butterworth-Van Dyke circuit for FBAR resonators and automated measurement system,” *Proc. IEEE Ultrason. Symp.*, pp. 863–868, 2000.

Application Library path: `MEMS_Module/Piezoelectric_Devices/thin_film_baw_resonator_3d_equivalent_circuit`

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 **Piezoelectricity** multiphysics interface.

2 click **3D**.

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

4 Click **Add**.

5 Click **Study**.

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

7 Click **Done**.

GEOMETRY 1

Use microns to define the geometry unit.

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 for the FEM model.

GLOBAL DEFINITIONS

Parameters - FEM Model

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Parameters - FEM Model` in the **Label** text field.

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

Name	Expression	Value	Description
L	85[um]	8.5E-5 m	Length of resonator side
L_a	10[um]	1E-5 m	Length of anchor
W_a	10[um]	1E-5 m	Width of anchor
t_te	0.1[um]	1E-7 m	Thickness of top electrode
t_piezo	1[um]	1E-6 m	Thickness of piezoelectric layer
t_be	0.2[um]	2E-7 m	Thickness of bottom electrode
t_nitride	0.2[um]	2E-7 m	Thickness of nitride membrane
Vapp	5[V]	5 V	Applied voltage
L_apo	L/(2*tan(36[deg]))	5.8496E-5 m	Apothem of pentagon

Define and specify the parameters for the circuit model.

Parameters - Equivalent Circuit Model

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Parameters - Equivalent Circuit Model` in the **Label** text field.

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

Name	Expression	Value	Description
Cm	62[fF]	6.2E-14 F	Capacitor, motional
Lm	39[nH]	3.9E-8 H	Inductor, motional
Rm	1[ohm]	1 Ω	Resistor, motional
Co	1[pF]	1E-12 F	Capacitor
Ro	500[ohm]	500 Ω	Resistor
Rs	1[ohm]	1 Ω	Resistor
Vsrc	5[V]	5 V	Voltage Source

Create the geometry model of a 36-degree sector of the resonator.

GEOMETRY 1

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>Square 1 (sq1)

1. In the **Work Plane** toolbar, click **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.
3. In the **Side length** text field, type `L_apo`.

Work Plane 1 (wp1)>Square 2 (sq2)

1. In the **Work Plane** toolbar, click **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.
3. In the **Side length** text field, type `2*L`.
4. Locate the **Rotation Angle** section. In the **Rotation** text field, type `-36`.

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

1. In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.
2. Select the object **sq1** only.
3. In the **Settings** window for **Difference**, locate the **Difference** section.
4. Click to select the **Activate Selection** toggle button for **Objects to subtract**.
5. Select the object **sq2** only.

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 `W_a/2`.
4. In the **Height** text field, type `L_a`.
5. Locate the **Position** section. In the **yw** text field, type `L_apo`.

Work Plane 1 (wp1)>Rotate 1 (rot1)

1. In the **Work Plane** toolbar, click **Transforms** and choose **Rotate**.
2. In the **Settings** window for **Rotate**, locate the **Rotation** section.
3. In the **Angle** text field, type `36`.
4. Click in the **Graphics** window and then press Ctrl+A to select both objects.

5 Click **Build Selected**.

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

Extrude 1 (ext1)

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (µm)
`t_nitride`
`t_nitride+t_be`
`t_nitride+t_be+t_piezo`
`t_nitride+t_be+t_piezo+t_te`

4 Click **Build Selected**.

Adjust the view scale in the *z* direction to better see the layers.

DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Camera

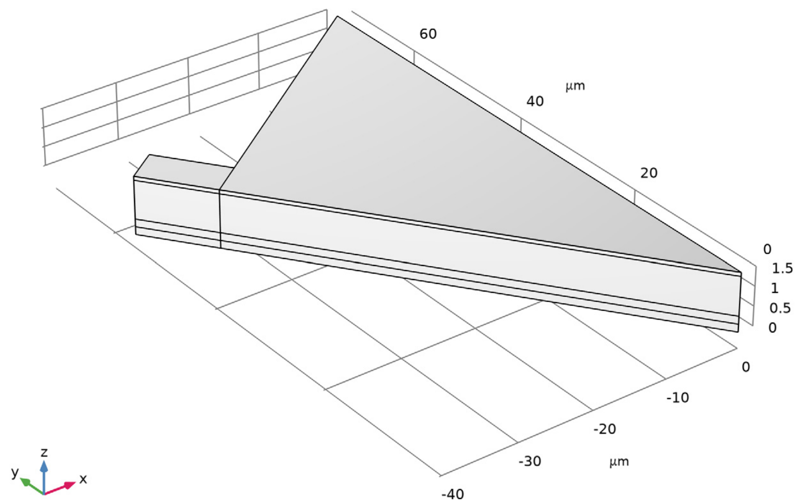
1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **Camera**.

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

3 From the **View scale** list, choose **Manual**.

4 In the **z scale** text field, type `5`.

5 Click **Update**.



Define selections for the piezoelectric, top, and bottom electrodes, and nitride layers as well as the symmetry and fixed boundaries. This will make specifying the materials and physics interfaces settings easier.

Fixed

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

2 In the **Settings** window for **Explicit**, type `Fixed` 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 box, type `1 4 7 10` in the **Selection** text field.

6 Click **OK**.

Nitride

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1 5` in the **Selection** text field.

6 Click **OK**.

Top Electrode

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

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

3 Select Domains 4 and 8 only.

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 Select Domains 2 and 6 only.

Piezoelectric

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

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

3 Select Domains 3 and 7 only.

Symmetry

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

2 In the **Settings** window for **Explicit**, type `Symmetry` 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 box, type `2 5 8 11 19 22 25 28 35 36 37 38` in the **Selection** text field.

6 Click **OK**.

Define an integration operator for evaluation of the displacement.

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 **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `30` in the **Selection** text field.

6 Click **OK**.

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

ELECTROSTATICS (ES)

Charge Conservation, Piezoelectric 1

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

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

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

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

Top Terminal

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

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

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Top Electrode**.

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

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

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 `9 26` in the **Selection** text field.

5 Click **OK**.

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

SOLID MECHANICS (SOLID)

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**.

Piezoelectric Material 1

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

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

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

Mechanical Damping 1

In the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.

Fixed Constraint 1

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

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

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

Symmetry 1

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

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

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

Create the mesh for the model.

MESH 1

Free Triangular 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `12 29` in the **Selection** text field.

5 Click **OK**.

Size 1

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

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

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

Swept 1

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

Distribution 1

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

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 3.

Distribution 2

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

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 2.

Distribution 3

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

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 2.

Distribution 4

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

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 1.

5 Click **Build All**.

MATERIALS

Add materials to the model and specify the regions they belong to.

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 **Piezoelectric>Aluminum Nitride**.

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

MATERIALS

Aluminum Nitride (mat1)

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

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

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

Property	Variable	Value	Unit	Property group
Loss factor for elasticity matrix cE	eta_cE_iso ; eta_cEii = eta_cE_iso, eta_cEij = 0	1e-3	1	Stress-charge form

ADD MATERIAL

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

2 In the tree, select **MEMS>Metals>Al - Aluminum**.

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

MATERIALS

Al - Aluminum (mat2)

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

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

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

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	1e-4	1	Basic

ADD MATERIAL

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

2 In the tree, select **Built-in>Molybdenum**.

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

MATERIALS

Molybdenum (mat3)

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

2 From the **Selection** list, choose **Bottom 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
Isotropic structural loss factor	eta_s	1e-4	1	Basic

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.

4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Si3N4 - Silicon nitride (mat4)

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

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

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

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	1e-2	1	Basic

Set up a Frequency Domain study for the FEM model. The range is chosen to include the features of interest: the series and parallel resonance frequencies. Also, disable the generation of default plots from this study.

FREQUENCY DOMAIN - FEM

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

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

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

Step 1: Frequency Domain

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

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

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

4 In the **Frequencies** text field, type `range(2.8, 0.01, 3.8)`.

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

From the FEM simulation in Study 1, plot the terminal current versus frequency.

RESULTS

Frequency Response

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

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Log10(Abs(I))`.

Because the 36 degree sector model is 1/10 of the device geometry, multiply terminal current by a factor of 10.

FEM

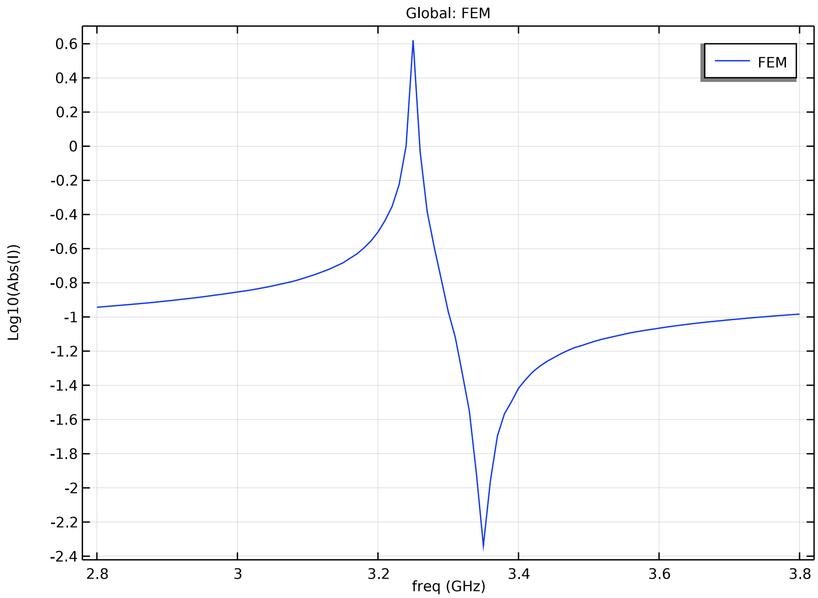
1 Right-click **Frequency Response** and choose **Global**.

2 In the **Settings** window for **Global**, type `FEM` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
log10(abs(10*es.I0_1))		FEM

4 In the **Frequency Response** toolbar, click **Plot**.



Add an **Electrical Circuit** interface. Add the components of the equivalent circuit and specify their parameter values.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **AC/DC>Electrical Circuit (cir)**.

4 Click **Add to Component 1** in the window toolbar.

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

ELECTRICAL CIRCUIT (CIR)

Voltage Source 1 (V1)

1 Right-click **Component 1 (comp1)>Electrical Circuit (cir)** and choose **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
n	0

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

R_s

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

2 In the **Settings** window for **Resistor**, type `R_s` in the **Label** text field.

3 Locate the **Node Connections** section. 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 `Rs`.

R_m

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

2 In the **Settings** window for **Resistor**, type `R_m` in the **Label** text field.

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

Label	Node names
p	2
n	3

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

L_m

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

2 In the **Settings** window for **Inductor**, type `L_m` in the **Label** text field.

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

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the L text field, type `Lm`.

C_m

1 In the **Electrical Circuit** toolbar, click **Capacitor**.
2 In the **Settings** window for **Capacitor**, type `C_m` in the **Label** text field.
3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	4
n	0

4 Locate the **Device Parameters** section. In the C text field, type `Cm`.

R_o

1 In the **Electrical Circuit** toolbar, click **Resistor**.
2 In the **Settings** window for **Resistor**, type `R_o` in the **Label** text field.
3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	2
n	0

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

C_o

1 In the **Electrical Circuit** toolbar, click **Capacitor**.
2 In the **Settings** window for **Capacitor**, type `C_o` in the **Label** text field.
3 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the C text field, type `Co`.

Set up a **Frequency Domain** study for the circuit model. Disable the **Electrostatics** and **Solid Mechanics** interfaces and the **Piezoelectricity** multiphysics coupling. Also, disable generation of default plots.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Frequency Domain**.

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

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

FREQUENCY DOMAIN - EQUIVALENT CIRCUIT

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

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Frequency Domain - Equivalent Circuit** click **Step 1: Frequency Domain**.

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

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

4 In the **Frequencies** text field, type `range(2.8,0.01,3.8)`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Electrostatics (es)** and **Solid Mechanics (solid)**.

6 In the table, clear the **Solve for** check box for **Piezoelectricity 1 (pze1)**.

7 In the **Model Builder** window, click **Frequency Domain - Equivalent Circuit**.

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

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

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

From the circuit simulation in Study 2, plot the terminal current versus frequency and add to the **Frequency Domain** plot group to compare the FEM and equivalent-circuit results using initial values of the circuit parameters.

RESULTS

Equivalent Circuit, Initial Values

1 In the **Model Builder** window, right-click **Frequency Response** and choose **Global**.

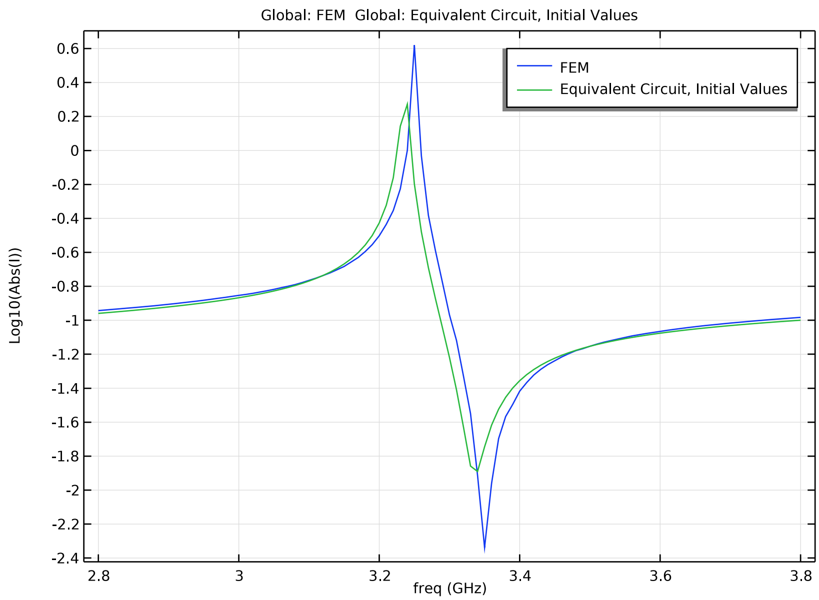
2 In the **Settings** window for **Global**, type `Equivalent Circuit, Initial Values` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Frequency Domain - Equivalent Circuit/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`log10(abs(cir.R1_i))`		`Equivalent Circuit, Initial Values`

5 In the **Frequency Response** toolbar, click **Plot**.



Copy the result of Study 1 to a table for use as reference data in a **Parameter Estimation** study.

FEM

In the **Model Builder** window, right-click **FEM** and choose **Copy Plot Data to Table**.

FEM Reference Data

1 In the **Model Builder** window, under **Results>Tables** click **Table 1**.

2 In the **Settings** window for **Table**, type `FEM Reference Data` in the **Label** text field.

3 Click **Update**.

Set up a **Parameter Estimation** study based on the previous **Frequency Domain** study for the circuit model.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

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

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

PARAMETER EXTRACTION

In the **Settings** window for **Study**, type `Parameter Extraction` in the **Label** text field.

FREQUENCY DOMAIN - EQUIVALENT CIRCUIT

Step 1: Frequency Domain

In the **Model Builder** window, under **Frequency Domain - Equivalent Circuit** right-click **Step 1: Frequency Domain** and choose **Copy**.

PARAMETER EXTRACTION

In the **Model Builder** window, right-click **Parameter Extraction** and choose **Paste Frequency Domain**.

Parameter Estimation

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

Define the **Experimental Data** for the **Parameter Estimation** study.

2 In the **Settings** window for **Parameter Estimation**, locate the **Experimental Data** section.

3 From the **Data source** list, choose **Result table**.

4 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
freq (GHz)	Frequency	Frequency unit=GHz

5 From the **Frequency unit** list, choose **GHz**.

Next, define the **Model expression** to be computed by the **Frequency Domain - Equivalent Circuit** study.

6 In the table, click to select the cell at row number 2 and column number 2.

7 In the **Model expression** text field, type `log10(abs(comp1.cir.R1_i))`.

8 In the **Unit** text field, type `1`.

9 From the **Scale** list, choose **Manual**.

10 In the **Scale value** text field, type 1.

Select the circuit parameters to be included in the study. Specify their initial values, scaling, and the lower and upper bounds. For this study, a default plot will be generated automatically comparing the FEM reference data and the circuit model using the final values of the circuit parameters.

11 Locate the **Estimated Parameters** section. Click **Add** six times.

12 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
Cm (Capacitor, motional)	62[fF]	60[fF]	45[fF]	75[fF]
Lm (Inductor, motional)	39[nH]	40[nH]	10[nH]	70[nH]
Co (Capacitor)	1[pF]	1[pF]	0.1[pF]	2[pF]
Rs (Resistor)	1[ohm]	10[ohm]	0.001[ohm]	20[ohm]
Rm (Resistor, motional)	1[ohm]	10[ohm]	0.01[ohm]	20[ohm]
Ro (Resistor)	500[ohm]	2500[ohm]	100[ohm]	5000[ohm]

13 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **SNOPT**.

14 Find the **Solver settings** subsection. From the **Least-squares time/parameter method** list, choose **Use only least-squares data points**.

Because in the **Frequency Domain** the variables are complex, the option for Split complex variables in real and imaginary parts must be enabled.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Compile Equations: Frequency Domain**.

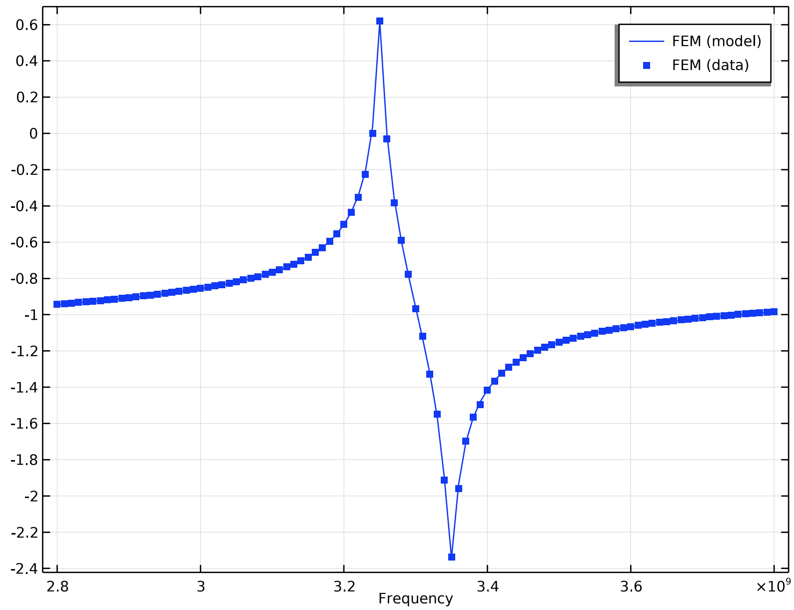
3 In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.

4 Select the **Split complex variables in real and imaginary parts** check box.

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

RESULTS

Parameter estimation



Set up an **Eigenfrequency** study to search for an eigenfrequency around 3.25 GHz. Deselect the **Electrical Circuit** interface from the list of physics interfaces solved for.

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 Multiphysics>Eigenfrequency**.

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

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

STUDY 4

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 25.

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

4 In the **Search for eigenfrequencies around shift** text field, type `3.25`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electrical Circuit (cir)**.

Disable the generation of default plots from this **Eigenfrequency** study.

6 In the **Model Builder** window, click **Study 4**.

7 In the **Settings** window for **Study**, type `Eigenfrequency` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

Add a **Mirror** and a **Sector** dataset to complete the device structure. These datasets will be used in a plot from the **Eigenfrequency study**.

RESULTS

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

3 From the **Dataset** list, choose **Eigenfrequency/Solution 4 (sol4)**.

4 Click **Plot**.

Sector 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Symmetry** section. In the **Number of sectors** text field, type `5`.

5 Click **Plot**.

From the **Eigenfrequency** study, create a 3D plot to show the shape of the eigenmode at 3.25 GHz. Use the dataset previously created.

Modes

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

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

Surface 1

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

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

3 In the **Expression** text field, type `solid.disp`.

4 Select the **Description** check box.

Modes

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

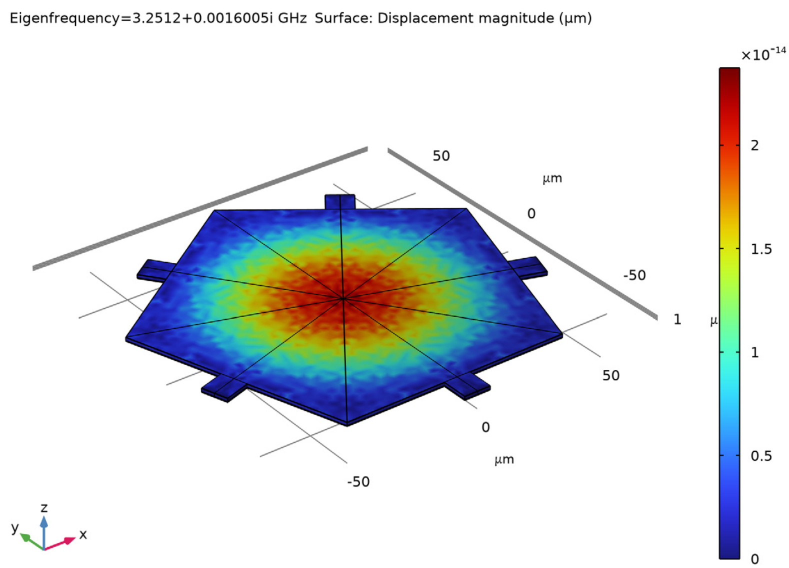
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Sector 3D 1**.

4 In the **Modes** toolbar, click **Plot**.

5 From the **Eigenfrequency (GHz)** list, choose **3.2512+0.0016005i**.

6 In the **Modes** toolbar, click **Plot**.



From the results of Study 1, evaluate the displacement phase uniformity. To do this, use the integral operator `intop1` defined earlier to access the z-displacement at the top surface of the device.

Displacement Phase Uniformity

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

2 In the **Settings** window for **1D Plot Group**, type `Displacement Phase Uniformity` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Eigenfrequency/Solution 4 (sol4)**.

Global 1

1 Right-click **Displacement Phase Uniformity** 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
`abs(intop1(w))/intop1(abs(w))`	1	`Displacement Phase Uniformity`

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

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

6 In the **Displacement Phase Uniformity** toolbar, click **Plot**.

7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.