Created in COMSOL Multiphysics 6.2

Aluminum Nitride Lamb Wave Resonator — 3D

Introduction

This tutorial demonstrates how to model an aluminum nitride Lamb wave resonator in 3D. The Lamb wave resonator (LWR) is an important building block in circuits used in mobile communication systems. Its operational characteristics are controlled by the piezoelectric material properties, the pattern of thin conducting layer on its surface, and its dimensions. Using the Piezoeletricity multiphysics interface and material models from the Piezoelectric Material Library, you can create a digital prototype of an LWR.

Model Definition

In this tutorial, the LWR is an aluminum nitride (piezoelectric) block with an interdigitated electrode (IDE) on its surface as shown in Figure 1. Suspended from four anchor points, the 0.4 μm piezoelectric block responds to AC excitation through the platinum IDE (80 nm). The LWR is designed to resonate at 8 GHz. The thin platinum layer was specified using the Thin Film feature in the Solid Mechanics Interface.

The LWR geometric parameters are summarized in the first table in the Modeling Instructions section so the geometry can be easily modified for design optimization. The geometry is rebuilt automatically when geometric parameters are changed if the rules are followed. For example, `n` must be chosen so `(n+1)/4` is an integer, or `n` = {3, 7, 11, …}. Also, the overlap parameter `op` should be smaller than the length of the finger `l`, that is, `op` < `l`. Keep in mind, however, that changes in the geometry will shift the resonant frequency. An outline of the steps in the fabrication of an actual LWR can be found in Ref. 1.

In this model, the fully coupled structural and electrostatic equations are solved in the piezoelectric domain. The model also includes mechanical loss through an isotropic structural loss factor of 0.002 for the piezoelectric and the platinum layers.

In this tutorial we are interested in the symmetric S0 mode, which, in typical applications, results in maximal electromechanical coupling efficiency. Because of the symmetry of the device structure and the symmetry of the S0 mode, only a half of the device needs to be modeled. This reduces computational time, which is important because this model requires a fine mesh (Figure 2) to compute accurate eigenmodes.

This tutorial shows you the setup of Eigenfrequency and Frequency Domain studies. In the Eigenfrequency study you investigate the eigenmodes of the structure to find the S0 mode. In the subsequent Frequency Domain study, a 1 V drive signal is applied to the Signal electrode and the resonator's frequency response from 7.95 to 8.05 GHz is analyzed.

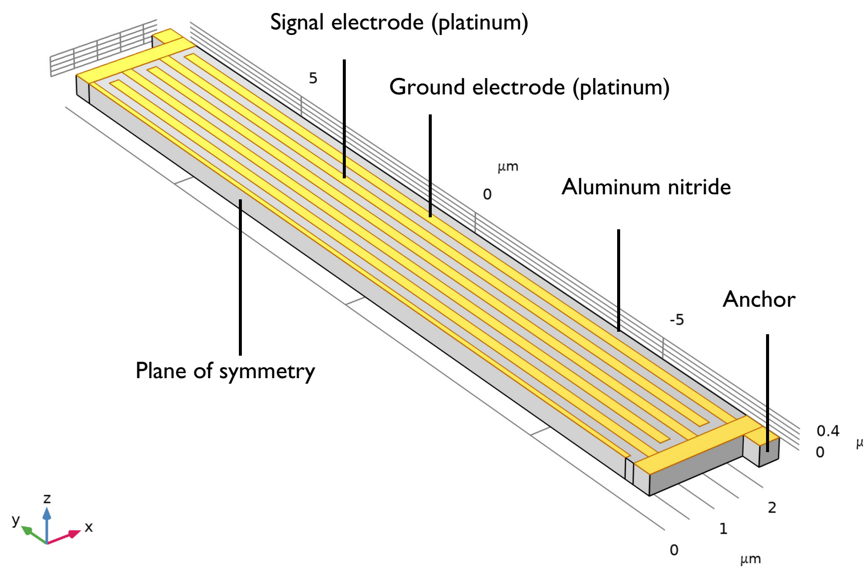


Figure 1: The geometric model of one half of the Lamb wave resonator. The interdigitated electrodes (IDE) are patterned from a thin layer of platinum (80 nm) on the top surface of the piezoelectric aluminum nitride block (0.4 µm).

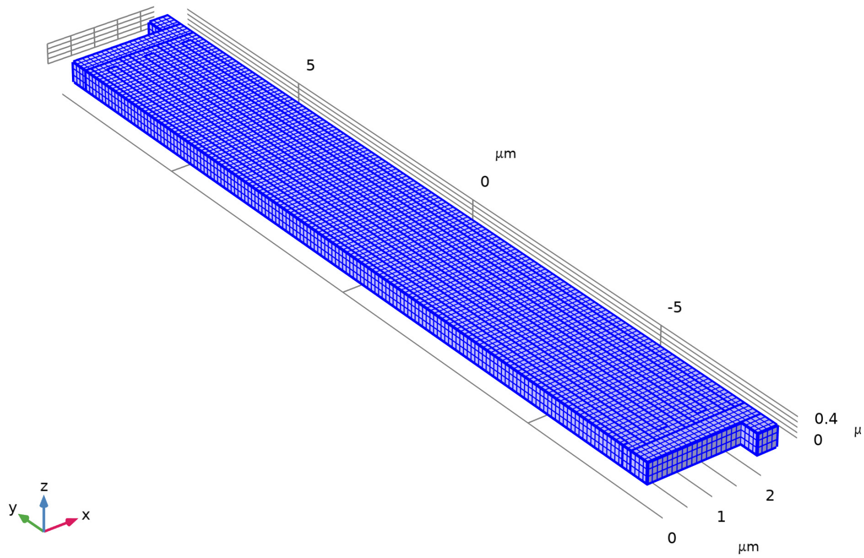


Figure 2: The top surface of the LWR is meshed using Free Quad with maximum mesh size of 0.1 µm for accuracy and smooth surface plots.

Results and Discussion

Figure 3 shows the displacement in the z direction for the S0 mode at 7.993 GHz. In high-frequency resonant devices such as LWRs, the Eigenfrequency study returns many extraneous solutions very close to the mode of interest. Only through visual inspection can you identify the correct mode from among the spurious modes. Figure 4 shows the solid displacement across the XZ cut plane through the center of the resonator to verify the S0 mode.

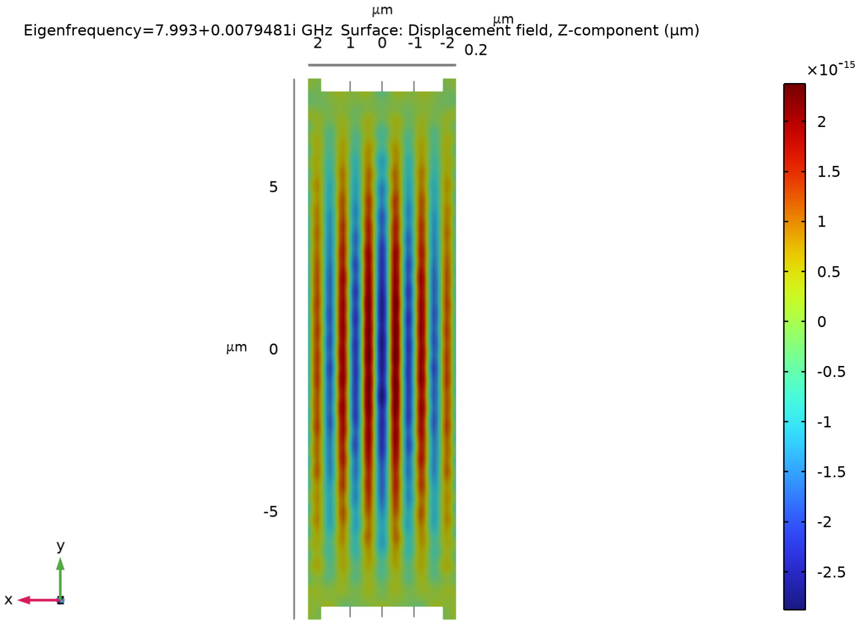


Figure 3: 3D plot showing the pattern of the displacement along the z-axis for the S0 mode at 7.99 GHz.

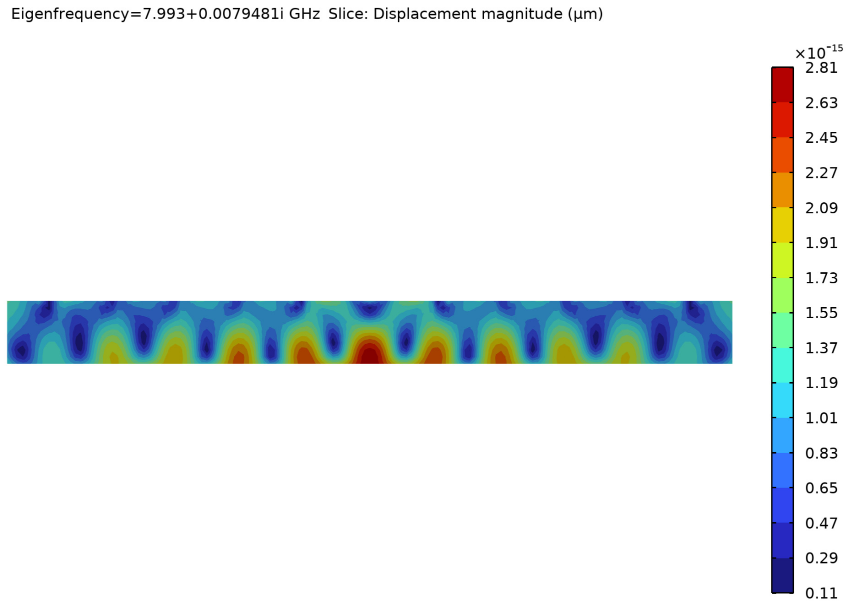


Figure 4: Slice plot of solid displacement across the XZ cut plane through the center of the resonator for the S0 mode at 7.99 GHz.

Figure 5 is a log plot of the admittance magnitude versus frequency from 7.95 to 8.05 GHz. The plot is annotated with the positions of resonance and anti-resonance peaks for the calculation of the electromechanical coupling coefficient.

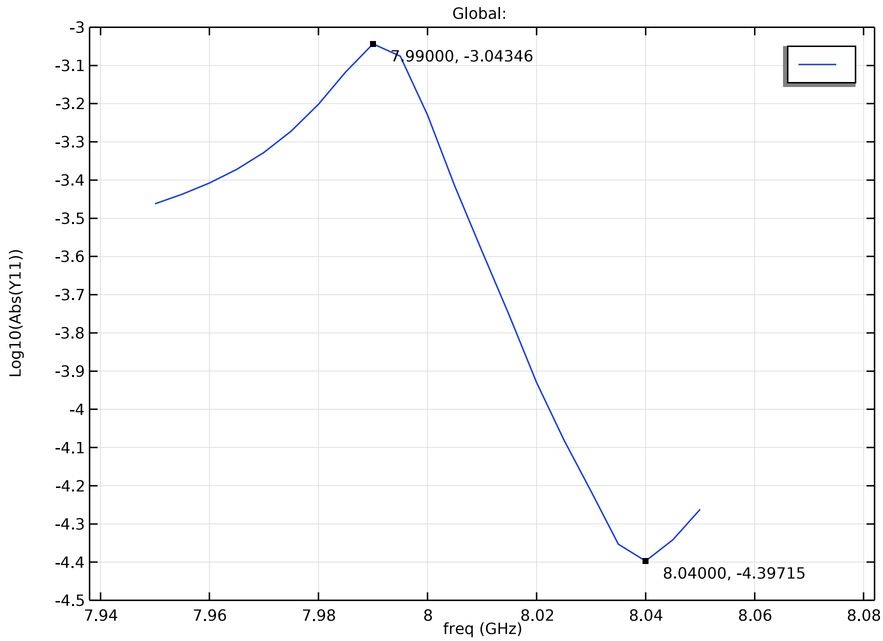


Figure 5: Admittance versus frequency.

The frequency response yields important information relating to device performance such as the effective electromechanical coupling coefficient given by

$$k_t^2 = \frac{\pi^2}{4} \times \frac{f_s(f_p - f_s)}{f_p^2}$$

where f_s and f_p are the resonance and anti-resonance frequencies, respectively. With $f_\mathrm{s} = 7.99$ and $f_\mathrm{p} = 8.04$, $k_\mathrm{t}^2 = 1.5\%$. The effective electromechanical coupling coefficient is known to be a measure of transduction efficiency for conversion of electrical into mechanical energy.

Reference

1. J. Zou, C.M. Lin, A. Gao, and A.P. Pisano, “The Multi-Mode Resonance in AlN Lamb Wave Resonators,” *J. Microelectromech. Syst.*, vol. 27, no. 6, pp. 973–84, 2018.

Application Library path: `MEMS_Module/Piezoelectric_Devices/aln_lamb_wave_resonator_3d`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

Start by creating a new 3D model with a **Piezoelectricity** multiphysics interface.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

Use microns as 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 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
tp	0.4[um]	4E-7 m	Thickness of piezoelectric layer
l	15[um]	1.5E-5 m	Length of finger
wf	0.2[um]	2E-7 m	Width of finger
op	14.6[um]	1.46E-5 m	Length of overlap, for op less than l
dy	(l-op)/2	2E-7 m	Electrode separation
n	11	11	Number of fingers, where (n+1)/4 = integer
we	0.2[um]	2E-7 m	Width of edge
la	0.4[um]	4E-7 m	Length of anchor
Vapp	1[V]	1 V	Applied voltage
eta0	2.0e-3	0.002	Loss factor for electrode layer
eta1	2.0e-3	0.002	Loss factor for piezoelectric layer

Create the geometry model for half of the lamb wave resonator.

GEOMETRY 1

Work Plane 1 (wp1)

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

2 Right-click **Geometry 1** and choose **Work Plane**.

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 `wf`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **yw** text field, type `dy/2`.

Work Plane 1 (wp1)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **xw size** text field, type `(n+1)/4`.

5 Locate the **Displacement** section. In the **xw** text field, type `4*wf`.

Work Plane 1 (wp1)>Move 1 (mov1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Move**.

2 Click the **Select All** button in the **Graphics** toolbar.

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

4 Select the **Keep input objects** check box.

5 Locate the **Displacement** section. In the **xw** text field, type `2*wf`.

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

Work Plane 1 (wp1)>Rectangle 2 (r2)

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 `(n-1/2)*wf+we`.

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

5 Locate the **Position** section. In the **yw** text field, type `(l+dy)/2`.

Work Plane 1 (wp1)>Rectangle 3 (r3)

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 `2*we`.

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

5 Locate the **Position** section. In the **xw** text field, type `(n-1/2)*wf-we`.

6 In the **yw** text field, type `(l+dy)/2+2*wf`.

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

1 In the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **r2** and **r3** only.

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

4 Select the **Keep input objects** check box.

5 Locate the **Normal Vector to Line of Reflection** section. In the **xw** text field, type `0`.

6 In the **yw** text field, type `-1`.

Work Plane 1 (wp1)>Rectangle 4 (r4)

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 `(n+1/2)*wf`.

4 In the **Height** text field, type `l+dy`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **xw** text field, type `(n+1/2)*wf/2`.

Work Plane 1 (wp1)>Partition Objects 1 (par1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **arr1(1,1)** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Click to select the **Activate Selection** toggle button for **Tool objects**.

5 Select the object **r4** only.

6 Select the **Keep tool objects** check box.

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

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **par1**, select Domain 1 only.

Extrude 1 (ext1)

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

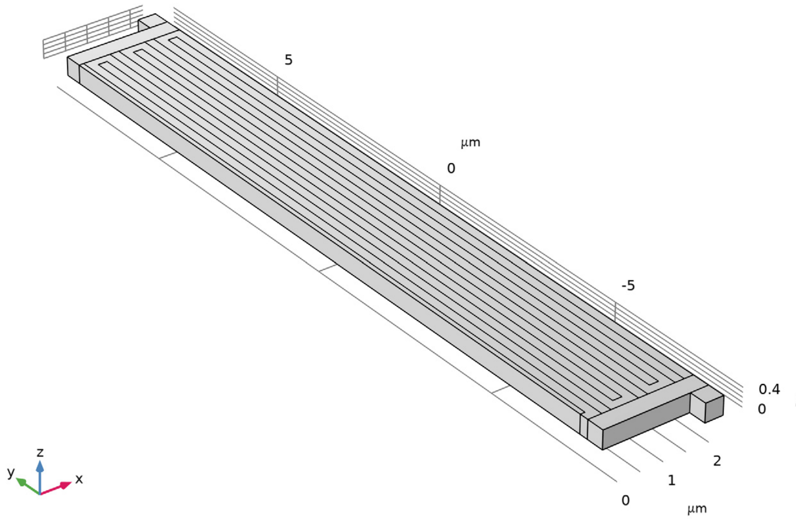
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (µm)
tp

4 Click **Build Selected**.

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



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

DEFINITIONS

Signal

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.
2. Right-click **Definitions** and choose **Selections>Box**.
3. In the **Settings** window for **Box**, type `Signal` in the **Label** text field.
4. Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
5. Locate the **Box Limits** section. In the **y minimum** text field, type `-l/2`.
6. In the **z minimum** text field, type `tp-0.01`.
7. Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
8. Right-click **Signal** and choose **Duplicate**.

Ground

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Signal 1**.

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

3 Locate the **Box Limits** section. In the **y minimum** text field, type `-Inf`.

4 In the **y maximum** text field, type `1/2`.

Electrodes

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

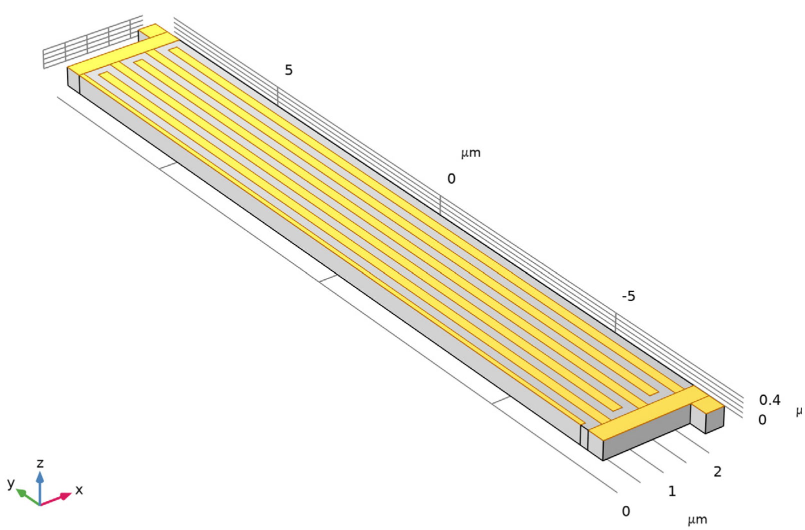
2 In the **Settings** window for **Union**, type `Electrodes` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog box, in the **Selections to add** list, choose **Signal** and **Ground**.

6 Click **OK**.



Symmetry

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

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

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

Top Surface

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

2 In the **Settings** window for **Box**, type `Top Surface` 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 minimum** text field, type `tp-0.01`.

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

Device

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

2 In the **Settings** window for **Box**, type `Device` 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 **y minimum** text field, type `-((l+dy)/2+2*wf+la)+0.01`.

5 In the **y maximum** text field, type `(l+dy)/2+2*wf+la-0.01`.

Fixed Constraints

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

2 In the **Settings** window for **Complement**, type `Fixed Constraints` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

5 In the **Add** dialog box, select **Device** in the **Selections to invert** list.

6 Click **OK**.

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

ELECTROSTATICS (ES)

Terminal 1

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

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

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

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

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**.

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

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

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

Mechanical Damping 1

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

2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.

3 From the **Damping type** list, choose **Isotropic loss factor**.

Thin Layer 1

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

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

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

4 Locate the **Boundary Properties** section. In the L_{th} text field, type `80[nm]`.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)>Thin Layer 1>Linear Elastic Material 1** node, then 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**.

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 Constraints**.

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**.

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 **MEMS>Metals>Pt - Platinum**.

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

MATERIALS

Pt - Platinum (mat1)

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

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

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

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

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	eta0	1	Basic

ADD MATERIAL

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

2 In the tree, select **Piezoelectric>Aluminum 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

Aluminum Nitride (mat2)

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

2 From the **Selection** list, choose **All domains**.

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	eta1	1	Basic

Create the mesh for the model.

MESH 1

Free Quad 1

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

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

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

Size 1

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

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** check box. In the associated text field, type `0.1`.

6 Click **Build Selected**.

Swept 1

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

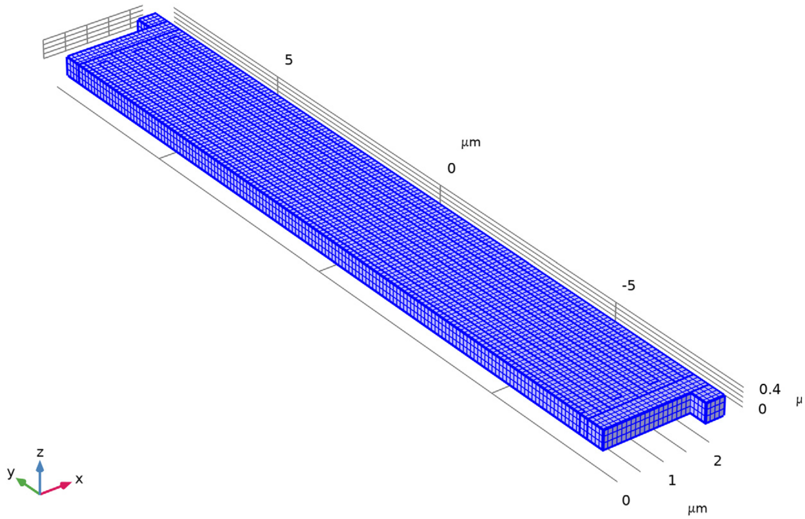
Distribution 1

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

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

3 In the **Number of elements** text field, type `3`.

4 Click **Build Selected**.



Set up an **Eigenfrequency** study to search for an eigenfrequency around 8 GHz.

EIGENFREQUENCY

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

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

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

Step 1: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `20`.

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

5 In the **Search for eigenfrequencies around shift** text field, type `8`.

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

Add a **Mirror** dataset to complete the device structure. This dataset will be used to plot the result of the **Eigenfrequency** study.

RESULTS

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, click **Plot**.

With the solution from the **Eigenfrequency** study, create a 3D plot to display the shape of the eigenmode at 8 GHz. Use the **Mirror** dataset previously created.

Mode Shape

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

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

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

4 From the **Eigenfrequency (GHz)** list, choose **7.993+0.0079481i**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

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

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

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

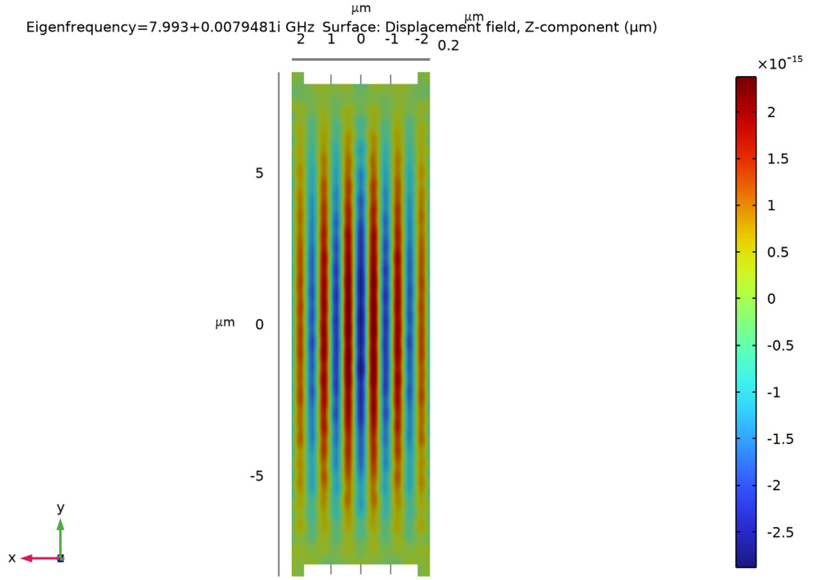
4 Select the **Description** check box.

5 In the **Mode Shape** toolbar, click **Plot**.

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

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

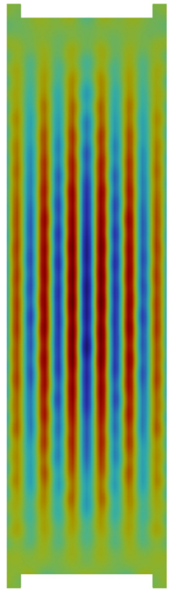
8 Click the **Go to XY View** button in the **Graphics** toolbar.



With the solution from the **Eigenfrequency** study, create a **Slice** plot to display the eigenmode in the xz cut plane through the center.

Mode Shape

In the **Model Builder** window, click **Mode Shape**.



Mode Shape, Center XZ Plane

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

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

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

4 From the **Eigenfrequency (GHz)** list, choose **7.993+0.0079481i**.

Slice 1

1 Right-click **Mode Shape, Center XZ Plane** and choose **Slice**.

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

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

4 Select the **Description** check box.

5 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

6 In the **Planes** text field, type `1`.

7 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Discrete**.

8 In the **Number of bands** text field, type `15`.

Mode Shape, Center XZ Plane

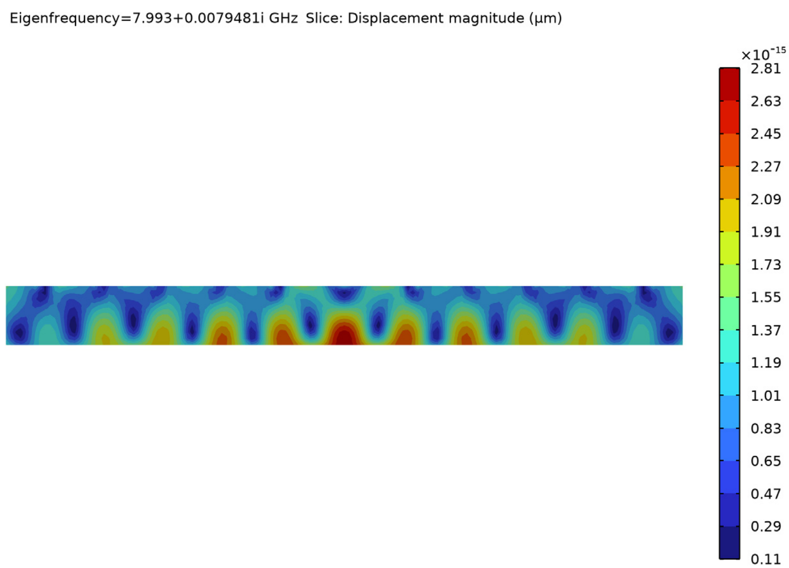
1 In the **Model Builder** window, click **Mode Shape, Center XZ Plane**.

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

3 Clear the **Plot dataset edges** check box.

4 In the **Mode Shape, Center XZ Plane** toolbar, click **Plot**.

5 Click the **Go to XZ View** button in the **Graphics** toolbar.



Set up a **Frequency Domain** study with a range that includes the features of interest, for example, resonance and anti-resonance peaks. Disable the option to generate default plots from this study.

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 - 7.95 TO 8.05 GHZ

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

2 In the **Settings** window for **Study**, type `Frequency Domain - 7.95 to 8.05 GHz` 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 - 7.95 to 8.05 GHz** 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 Click **Range**.

5 In the **Range** dialog box, type `7.95` in the **Start** text field.

6 In the **Step** text field, type `0.005`.

7 In the **Stop** text field, type `8.05`.

8 Click **Add**.

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

With the solution from **Frequency Domain** study, plot the admittance versus frequency and add a **Graph Marker** to return the coordinates of the maximum and minimum values.

RESULTS

Frequency Domain

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 Domain` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Frequency Domain - 7.95 to 8.05 GHz/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Log10(Abs(Y11))`.

Global 1

1 Right-click **Frequency Domain** 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
`log10(abs(es.Y11))`		

4 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

Graph Marker 1

1 Right-click **Global 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Text Format** section.

3 Select the **Show x-coordinate** check box.

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