



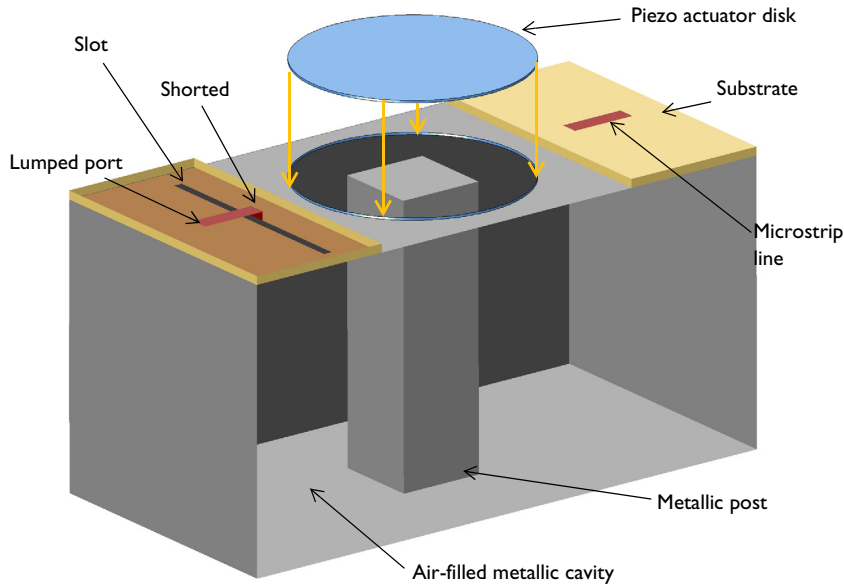
Model created in COMSOL Multiphysics 6.4

# Tunable Evanescent Mode Cavity Filter Using a Piezo Actuator

## Introduction

---

An evanescent mode cavity filter can be realized by adding a structure inside of the cavity. This structure changes the resonant frequency below that of the dominant mode of the unfilled cavity. A piezo actuator is used to control the size of a small air gap which provides the tunability of the resonant frequency.



*Figure 1: A tunable evanescent mode cavity filter is composed of a rectangular cavity with a metallic post, a piezo actuator disk, and slot-coupled microstrip lines. There is a small gap between the top of the post and the bottom side of the piezo actuator. The front part of the cavity wall is removed for visualization purposes.*

---

**Note:** In addition to the RF Module, this example requires one of the Acoustics Module, the MEMS Module, or the Structural Mechanics Module.

---

## Model Definition

---

This example starts from a basic rectangular cavity filter, whose resonant frequencies are given by

$$f_{nml} = \frac{c}{2\pi\sqrt{\epsilon_r\mu_r}} \sqrt{\left(\frac{m\pi}{a}\right)^2 + \left(\frac{n\pi}{b}\right)^2 + \left(\frac{l\pi}{d}\right)^2}$$

where  $a$  and  $b$  are the waveguide aperture dimensions and  $d$  is the length of the waveguide cavity. In this example, the cavity width, height, and length are  $a = 100$  mm,  $b = 50$  mm, and  $d = 50$  mm, respectively. The resulting resonant frequency of the dominant mode,  $TE_{101}$ , is 3.354 GHz.

By adding a metallic post and creating reactance inside the cavity, the resonance frequency can be lowered. The cavity is air filled and the height of the post is slightly smaller than  $b$ , which creates a small gap between the top of the post and the cavity where the electric fields are confined. Two shorted  $50 \Omega$  microstrip lines on a dielectric substrate, fed by a lumped port, are coupled into the cavity. The dimensions and locations of the slots can be adjusted to improve input matching properties and power transfer between input and output ports. The air box around the microstrip lines are enclosed by a scattering boundary condition representing the infinite air space. A circular aperture at the top of the cavity is closed with a piezo actuator and the bottom surface of the disk is finished with a layer of a highly conductive material that is several skin depths in thickness.

Model all metal parts — the cavity walls, post, substrate ground planes, microstrip lines, and the bottom surface of the piezo device — as perfect electric conductors (PECs). The material for the piezo actuator is Lead Zirconate Titanate (PZT-5H). It is  $z$ -polarized and generates mainly  $z$ -directional deflection of the device.

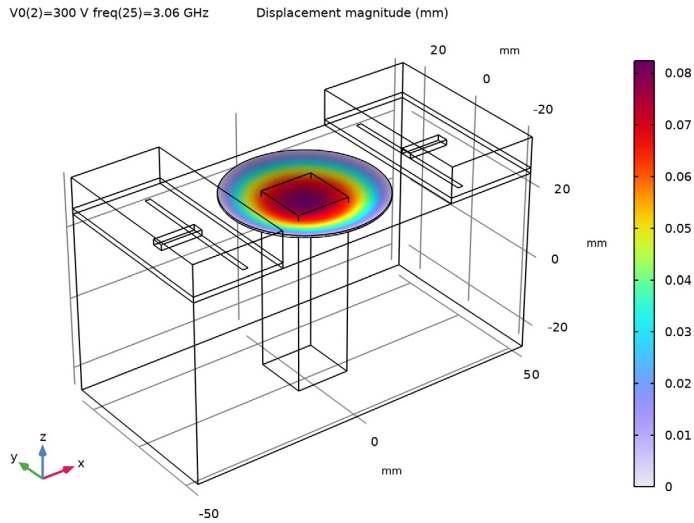
Mesh the model using a tetrahedral mesh with approximately five elements per wavelength in each material at the highest simulation frequency. When the piezo device deforms due to the input bias, the Moving Mesh interface is used to deform the mesh for the Electromagnetic Waves physics.

## *Results and Discussion*

---

A +300 V potential is applied across the piezo actuator, which causes the device to deflect  $\sim 90 \mu\text{m}$  toward the bottom; see [Figure 2](#). This makes the reactance stronger and shifts the resonant frequency lower than the negative bias case. [Figure 3](#) plots the electric field norm

at the resonance. At the center of the cavity as well as in the gap between the top of the post and the bottom of the piezo device, strong electric fields are observed.



*Figure 2: This plot shows the total piezo displacement when 300 V is applied on the actuator. The visualization is exaggerated to emphasize the deflection.*

The S-parameters plotted in [Figure 4](#) show the effect of the piezo device deflection on the filter's resonant frequency. The tunable frequency range of this example is  $\sim 40$  MHz. This range can be adjusted by different choices of the piezo disk size and the input bias voltage.

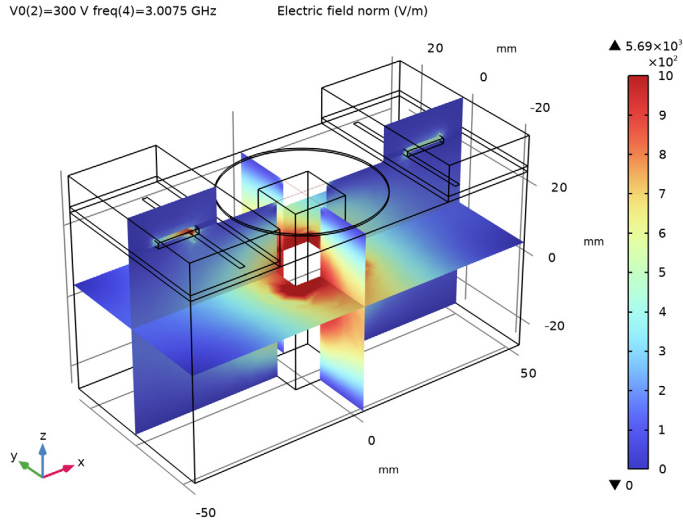


Figure 3: The dominant mode inside the cavity is observed from the electric field distribution plot.

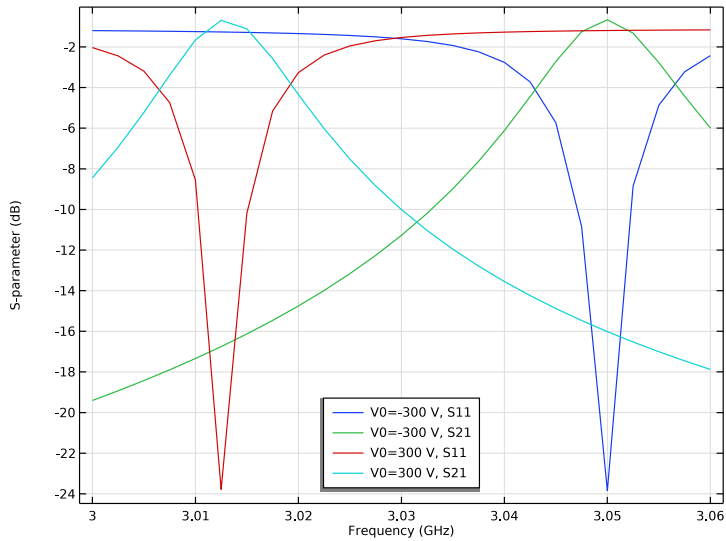


Figure 4: The deflection for the piezo device controlled by the input bias can shift the resonance frequency of the filter.

## *Notes About the COMSOL Implementation*

---

This example uses the built-in Piezoelectricity multiphysics interface which couples the Solid Mechanics and Electrostatics interfaces through the Piezoelectric Effect node located under the Multiphysics branch. Other physics interfaces used are Moving Mesh and Electromagnetic Waves, Frequency Domain. A Stationary study is used for the Solid Mechanics, Electrostatics and Moving Mesh interfaces, and a Frequency Domain study is used for the Electromagnetic Waves interface.

---

**Application Library path:** RF\_Module/Filters/tunable\_cavity\_filter


---

## *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**.
- 2 In the **Select Physics** tree, select **Structural Mechanics > Electromagnetics–Structure Interaction > Piezoelectricity > Piezoelectricity, Solid**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Radio Frequency > Electromagnetic Waves, Frequency Domain (emw)**.
- 5 Click **Add**.
- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **Empty Study**.
- 8 Click  **Done**.

### **STUDY 1**

*Step 1: Stationary*

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

### Step 2: Frequency Domain

1 In the **Study** toolbar, click  **Frequency Domain**.

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the highest frequency value in the specified range.

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


3 In the **Frequencies** text field, type `range(3[GHz], 2.5[MHz], 3.06[GHz])`.


### GEOMETRY I

The model geometry is available as a parameterized geometry sequence in a separate MPH file. If you want to build it from scratch, follow the instructions in the section [Appendix: Geometry Modeling Instructions](#). Otherwise load it from file with the following steps.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

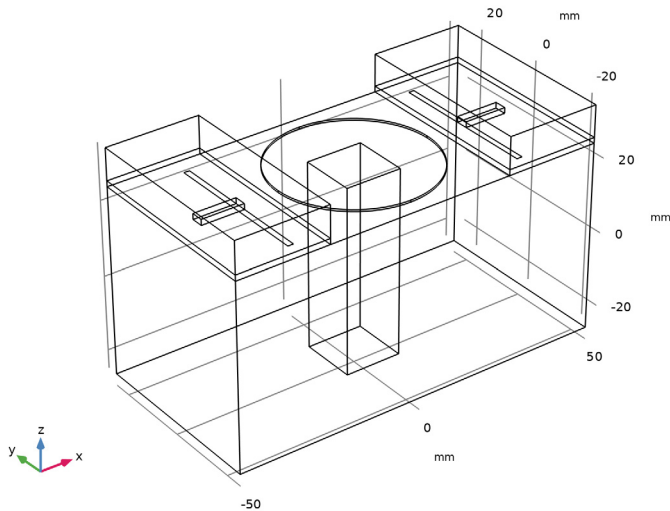
2 Browse to the model's Application Libraries folder and double-click the file `tunable_cavity_filter_geom_sequence.mph`.

3 In the **Geometry** toolbar, click  **Build All**.

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

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

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



## GLOBAL DEFINITIONS

### Parameters I

Add parameters that are not related to geometry.

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:


Name	Expression	Value	Description
V0	300[V]	300 V	Piezo actuator bias

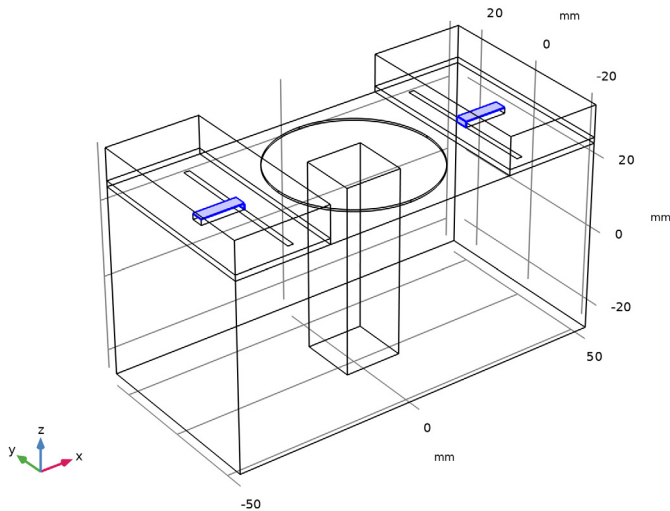
Here,  $c\_const$  is a predefined COMSOL constant for the speed of light in vacuum.

## DEFINITIONS

Create a set of selections for use when setting up the physics. First, create a selection for the microstrip feed line.


### Feed line

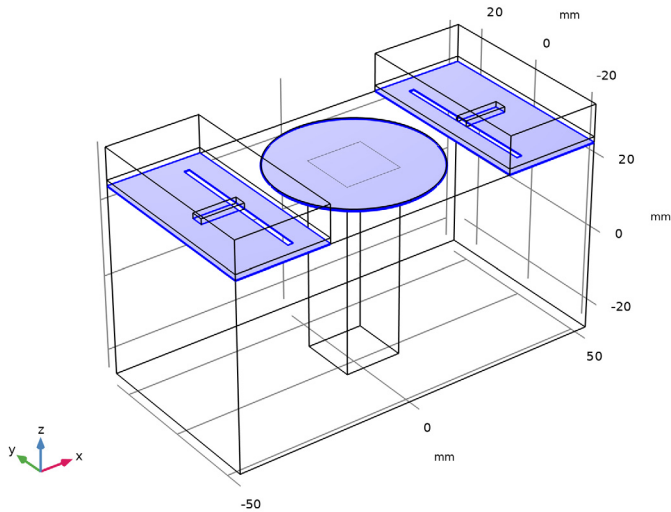
- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Feed line in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 17, 22, 47, and 50 only.



Add a selection for the ground.

#### Ground

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Ground in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 6, 16, 28, 39, and 53 only.

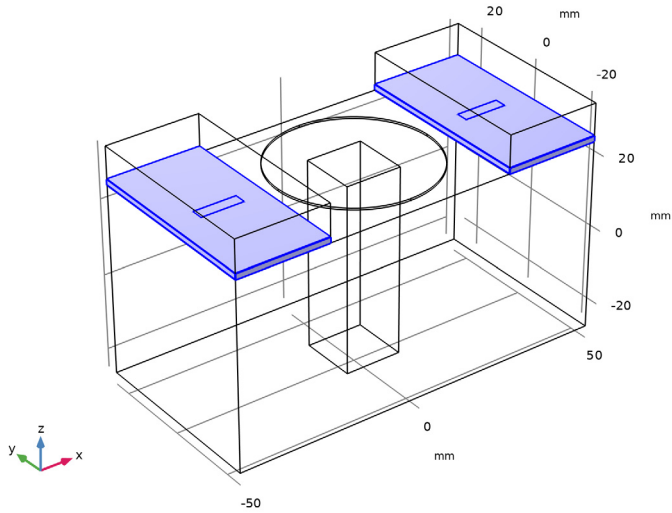


Add a selection for the substrate.

#### Substrate


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Substrate in the **Label** text field.

3 Select Domains 2, 4, 6, and 8 only.

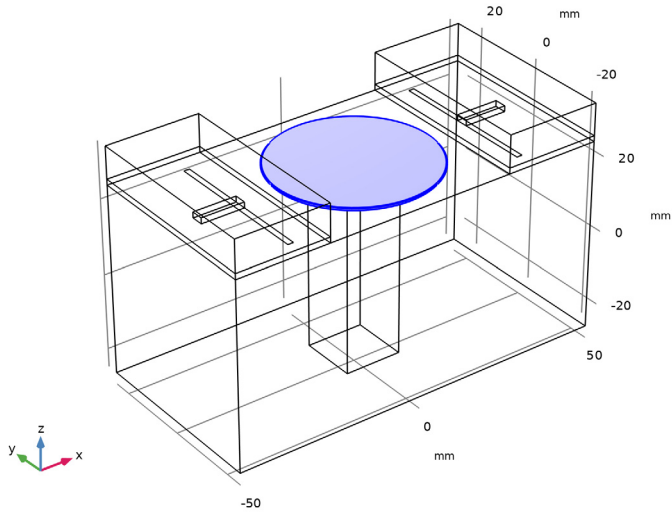


Add a selection for the piezo actuator disk.

*Piezo actuator*


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Piezo actuator in the **Label** text field.

3 Select Domain 5 only.

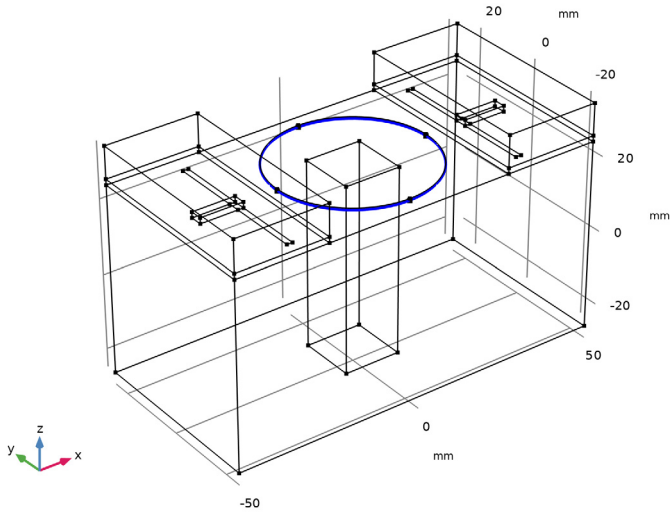


Add a selection for fixed edges of the piezo disk.

*Piezo fixed edges*


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Piezo fixed edges in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edges 50, 51, 63, and 66 only.

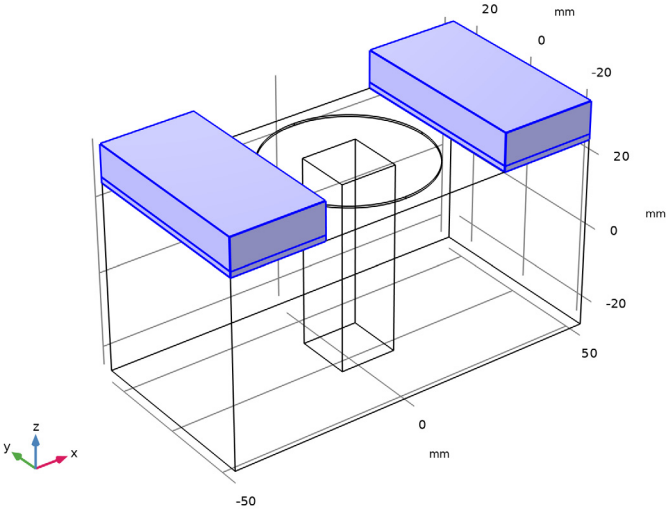


Add a selection for the open boundaries of RF domain.

*Open boundaries*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Open boundaries in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4, 5, 7, 8, 10, 12, 13, 23, 25, 37, 38, 40, 41, 43–45, 56, and 57 only.



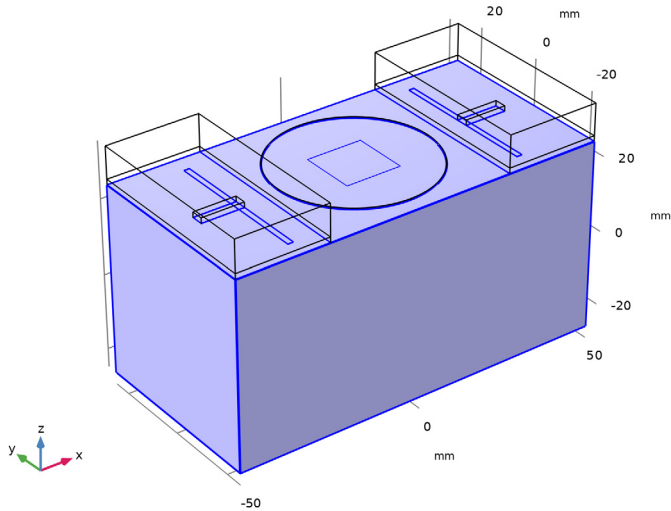
Set up the Moving Mesh. Because the substrate and top air domains are deflection free regions, do not include them.

**COMPONENT 1 (COMP1)**


*Deforming Domain 1*

1 In the **Physics** toolbar, click  **Moving Mesh** and choose **Free Deformation**.

2 Select Domain 1 only.



#### Prescribed Mesh Displacement I

- 1 In the **Moving Mesh** toolbar, click  **Prescribed Mesh Displacement**.
- 2 Select Boundary 28 only.
- 3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.
- 4 Specify the  $dx$  vector as

u	X
v	Y
w	Z

Before defining materials, specify domains for each physics.

#### SOLID MECHANICS (SOLID)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
- 2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Piezo actuator**.

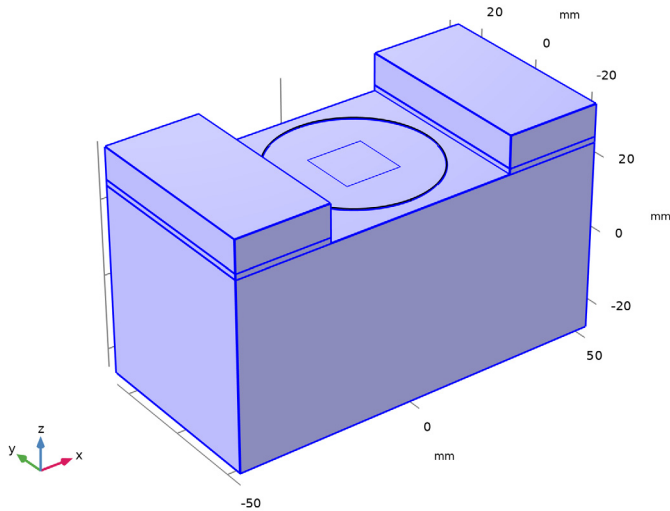
#### ELECTROSTATICS (ES)

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

- 2 In the **Settings** window for **Electrostatics**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Piezo actuator**.



### ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
- 2 Select Domains 1–4 and 6–8 only.



Assign material properties. Use three materials for this model: PZT-5H, air, and a user-defined substrate.

### ADD MATERIAL

- 1 In the **Materials** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Piezoelectric > Lead Zirconate Titanate (PZT-5H)**.
- 4 Click the **Add to Component** button in the window toolbar.
- 5 In the tree, select **Built-in > Air**.
- 6 Click the **Add to Component** button in the window toolbar.
- 7 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

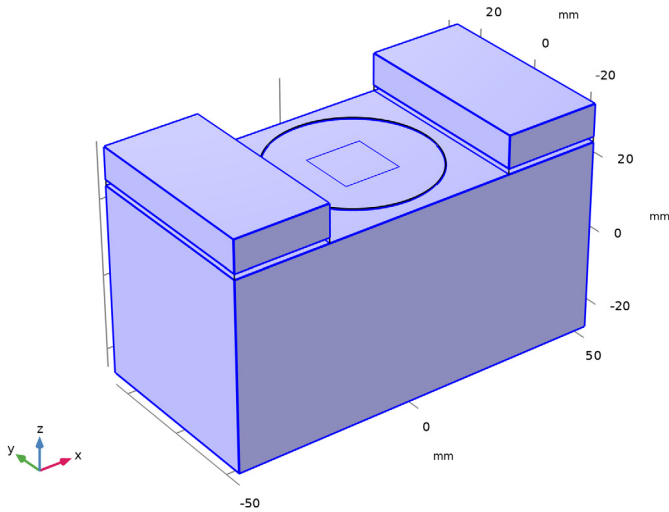
## MATERIALS

*Lead Zirconate Titanate (PZT-5H) (mat1)*

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

*Air (mat2)*

- 1 In the **Model Builder** window, click **Air (mat2)**.
- 2 Select Domains 1, 3, and 7 only.



*Material 3 (mat3)*

- 1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Substrate**.


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

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilon <sub>r_</sub> iso ; epsilon <sub>r_</sub> ii = epsilon <sub>r_</sub> iso, epsilon <sub>r_</sub> ij = 0	3.38		Basic
Relative permeability	mu <sub>r_</sub> iso ; mu <sub>r_</sub> ii = mu <sub>r_</sub> iso, mu <sub>r_</sub> ij = 0	1		Basic
Electric conductivity	sigma <sub>iso</sub> ; sigma <sub>ii</sub> = sigma <sub>iso</sub> , sigma <sub>ij</sub> = 0	0	S/m	Basic

### SOLID MECHANICS (SOLID)

Now, set up the physics constraints. Start by assuming the bottom rim of piezo actuator part is attached on the same size circular aperture of the cavity top and no deflection is expected.


#### *Fixed Constraint 1*

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

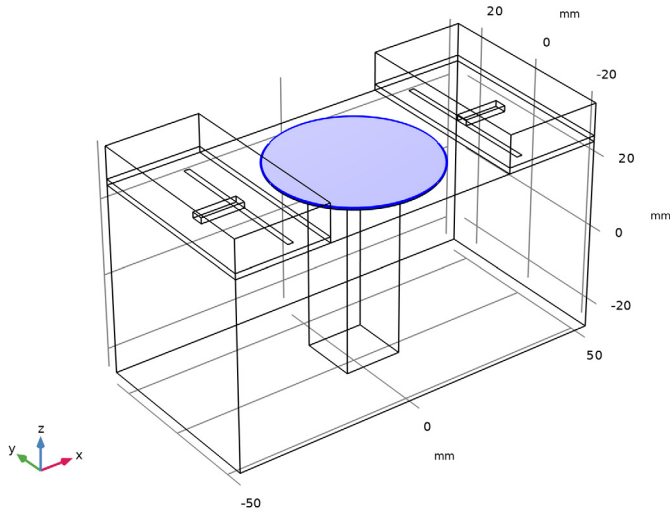
### ELECTROSTATICS (ES)

Next, define the Electrostatics constraints.


#### *Electric Potential 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Electric Potential**.
- 2 Select Boundary 29 only.
- 3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the  $V_0$  text field, type -V0.




#### *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**.

### **ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)**

#### *Perfect Electric Conductor 2*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Perfect Electric Conductor**.
- 2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Feed line**.

#### *Perfect Electric Conductor 3*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Perfect Electric Conductor**.
- 2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Ground**.

#### *Scattering Boundary Condition 1*

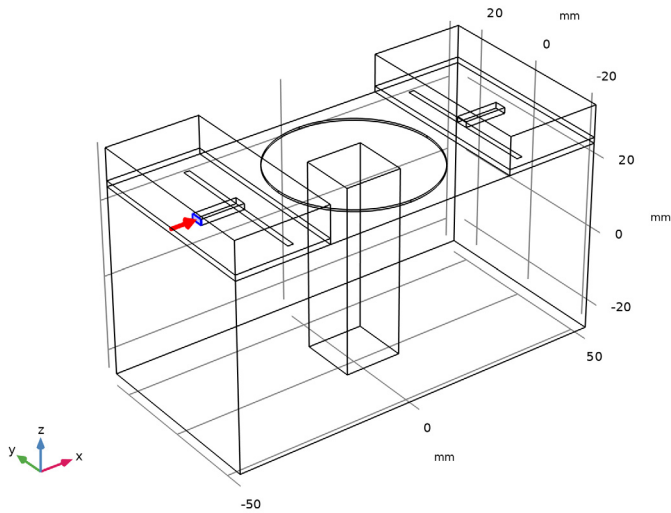
- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Scattering Boundary Condition**.

- 2 In the **Settings** window for **Scattering Boundary Condition**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Open boundaries**.

#### *Lumped Port 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Lumped Port**.
- 2 Select Boundary 14 only.

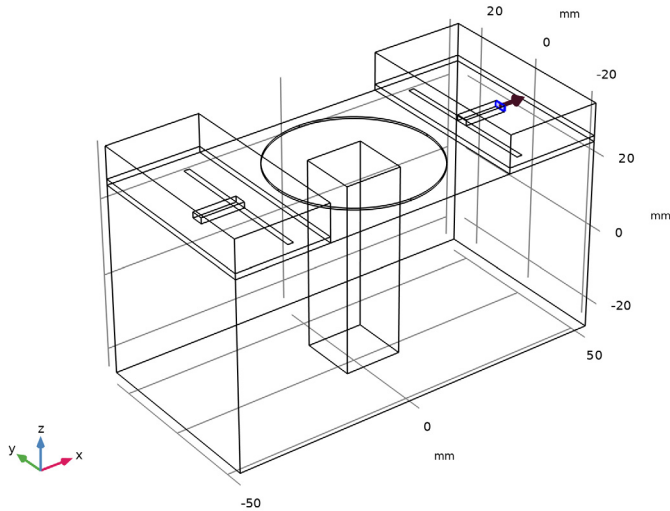
For the first port, wave excitation is **on** by default.



#### *Lumped Port 2*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Lumped Port**.

2 Select Boundary 54 only.



## MESH 1

Adjust the maximum and minimum element size manually on the piezo actuator to reduce the memory cost of running the study without compromising the accuracy. Then, use swept mesh for the piezo actuator to handle the structural deformation more efficiently.

- 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

- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size 1**.
- 2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.
- 3 In the **Maximum element size** text field, type 10.
- 4 In the **Minimum element size** text field, type 1.


### Free Tetrahedral 1

- 1 In the **Model Builder** window, click **Free Tetrahedral 1**.
- 2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1–4 and 6–8 only.

Select all domains except for the piezo actuator.

### *Swept 1*

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

### *Distribution 1*

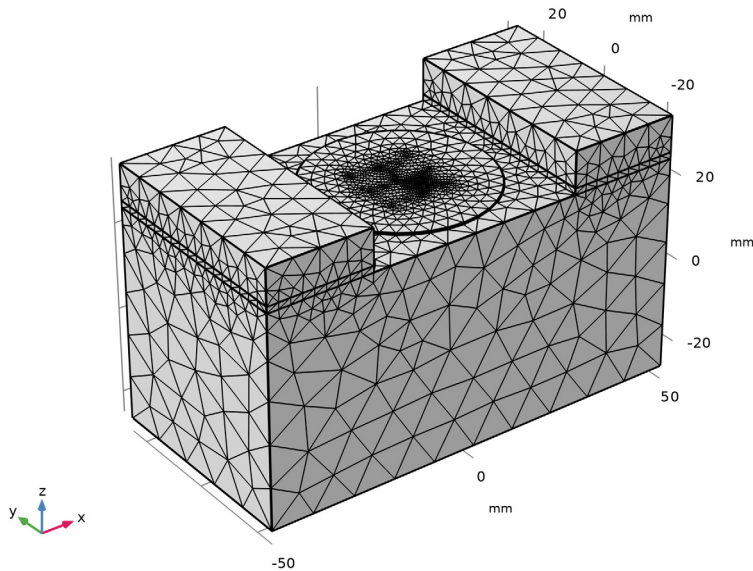
1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 2.


4 Click  **Build All**.

5 In the **Model Builder** window, click **Mesh 1**.



## **STUDY 1**

### *Parametric Sweep*

1 In the **Study** toolbar, click  **Parametric Sweep**.


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

3 Click  **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0 (Piezo actuator bias)	-300, 300	V

### Step 2: Frequency Domain

- 1 In the **Model Builder** window, click **Step 2: Frequency Domain**.
- 2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.
- 3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Moving Mesh**, **Solid Mechanics (solid)**, and **Electrostatics (es)**.
- 4 In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkbox for **Piezoelectricity 1 (pze1)**.
- 5 In the **Study** toolbar, click  **Compute**.



## RESULTS

### Displacement

Replace the default stress plot by displacement plot.

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

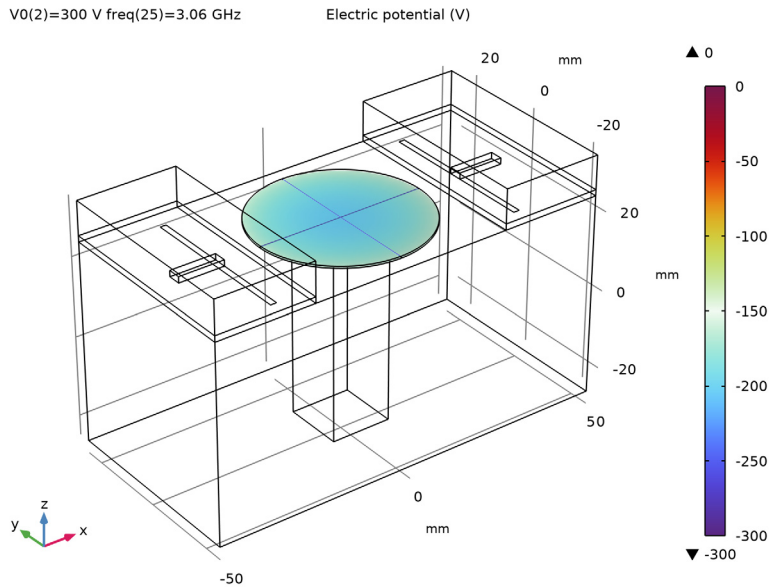
### Volume 1

- 1 In the **Model Builder** window, expand the **Displacement** node, then click **Volume 1**.
- 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 In the **Displacement** toolbar, click  **Plot**.
- 4 Click the  **Zoom Extents** button in the **Graphics** toolbar.

This plot shows the deflected piezo actuator disk; compare with [Figure 2](#).


### Electric Potential (es)

- 1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.



The electric potential inside the piezo actuator disk.

### Electric Field (emw)

- 1 In the **Model Builder** window, click **Electric Field (emw)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (freq (GHz))** list, choose **3.0075**.
- 4 In the **Electric Field (emw)** toolbar, click  **Plot**.

### Multislice 1

- 1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice 1**.
- 2 In the **Settings** window for **Multislice**, click to expand the **Range** section.
- 3 Select the **Manual color range** checkbox.
- 4 In the **Maximum** text field, type 1000.


- 5 In the **Electric Field (emw)** toolbar, click  **Plot**.

The resulting plot shows strong electric fields resulting from the dominant resonance at the center of the cavity as well as in the gap between the metallic post and the ceiling of the cavity. Compare the plot with that shown in [Figure 3](#).

#### *S-Parameter (emw)*

- 1 In the **Model Builder** window, expand the **Results > S-Parameter (emw)** node, then click **S-Parameter (emw)**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Lower middle**.

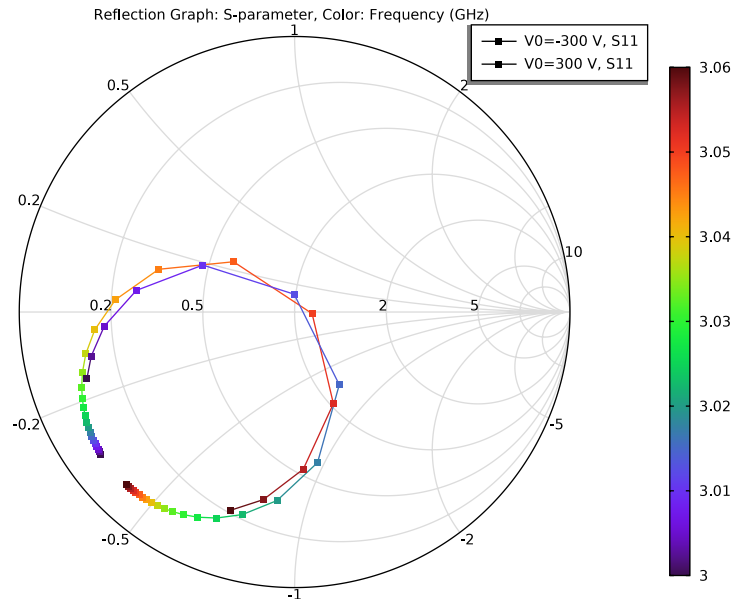
#### *Global 1*

- 1 In the **Model Builder** window, click **Global 1**.
- 2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.
- 3 From the **Axis source data** list, choose **freq**.
- 4 In the **S-Parameter (emw)** toolbar, click  **Plot**.

The plotted S-parameters show the frequency shift as a function of the input bias on the piezo actuator; compare with [Figure 4](#).

### Smith Plot (emw)


In the **Model Builder** window, expand the **Results > Smith Plot (emw)** node, then click **Smith Plot (emw)**.




Analyze the same model with a much finer frequency resolution using an **Adaptive Frequency Sweep** based on asymptotic waveform evaluation (AWE). When a device presents a slowly varying frequency response, the AWE provides a faster solution time when running the simulation on many frequency points. The following example with the AWE can be computed 10 times faster than regular Frequency Domain sweeps with the same frequency resolution.

## ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)



### Lumped Port 1

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Electromagnetic Waves, Frequency Domain (emw)** click **Lumped Port 1**.
- 2 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.
- 3 Click  **Create Selection**.
- 4 In the **Create Selection** dialog, type Lumped port 1 in the **Selection name** text field.
- 5 Click **OK**.

### Lumped Port 2

- 1 In the **Model Builder** window, click **Lumped Port 2**.
- 2 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.
- 3 Click  **Create Selection**.
- 4 In the **Create Selection** dialog, type Lumped port 2 in the **Selection name** text field.
- 5 Click **OK**.


### 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 the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

### STUDY 2


- 1 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 2 Clear the **Generate default plots** checkbox.

### Parametric Sweep


- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click **+ Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0 (Piezo actuator bias)	-300 300	V

#### Step 1: Stationary

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

#### Step 2: Adaptive Frequency Sweep

- 1 In the **Study** toolbar, click  **More Study Steps** and choose **Frequency Domain > Adaptive Frequency Sweep**.
- 2 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Study Settings** section.

**3** In the **Frequencies** text field, type range (3 [GHz] , 0.25 [MHz] , 3.06 [GHz] ).  
 Use a 10 times finer frequency resolution.  
 A slowly varying scalar value curve works well for AWE expressions. When **AWE expression type** is set to **Physics controlled** in the **Adaptive Frequency Sweep** study settings, `abs (comp1 . emw . S21)` is used automatically for two-port devices.

**4** In the **Relative tolerance** text field, type 0.001.

Use a smaller value for the **Relative tolerance** to get more accurate results.

**5** Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Moving Mesh**, **Solid Mechanics (solid)**, and **Electrostatics (es)**.

**6** In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkbox for **Piezoelectricity 1 (pze1)**.

Because such a fine frequency step generates a memory-intensive solution, the model file size will increase tremendously when it is saved. When only the frequency response of port related variables are of interest, it is not necessary to store all of the field solutions. By selecting the **Store in Output** checkbox in the **Values of Dependent Variables** section, we can control the part of the model on which the computed solution is saved. We only add the selection containing these boundaries where the port variables are calculated. The lumped port size is typically very small compared to the entire modeling domain, and the saved file size with the fine frequency step is more or less that of the regular discrete frequency sweep model when only the solutions on the lumped port boundaries are stored.

**7** Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Solid Mechanics (solid)	Selection

**8** Click to select row number 1 in the table.

**9** Under **Selections**, click **+ Add**.

**10** In the **Add** dialog, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

**11** Click **OK**.

It is necessary to include the lumped port boundaries to calculate S-parameters. By choosing only the lumped port boundaries for **Store in Output** settings, it is possible to reduce the size of a model file a lot.

**12** In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Store in Output** section.

13 In the table, enter the following settings:

Interface	Output
Electrostatics (es)	Selection

14 Click to select row number 2 in the table.

15 Under **Selections**, click **+ Add**.

16 In the **Add** dialog, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

17 Click **OK**.

18 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Store in Output** section.

19 In the table, enter the following settings:

Interface	Output
Electromagnetic Waves, Frequency Domain (emw)	Selection

20 Click to select row number 3 in the table.

21 Under **Selections**, click **+ Add**.

22 In the **Add** dialog, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

23 Click **OK**.

24 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Store in Output** section.

25 In the table, enter the following settings:

Interface	Output
Moving mesh (Component 1)	Selection

26 Click to select row number 4 in the table.

27 Under **Selections**, click **+ Add**.

28 In the **Add** dialog, in the **Selections** list, choose **Lumped port 1** and **Lumped port 2**.

29 Click **OK**.

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

## RESULTS

*S-Parameter (emw) 1*

In the **Model Builder** window, right-click **S-Parameter (emw)** and choose **Duplicate**.

*Global 1*

1 In the **Model Builder** window, expand the **S-Parameter (emw) 1** node, then click **Global 1**.

- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	dB	S11 Regular Sweep
emw.S21dB	dB	S21 Regular Sweep


- 4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

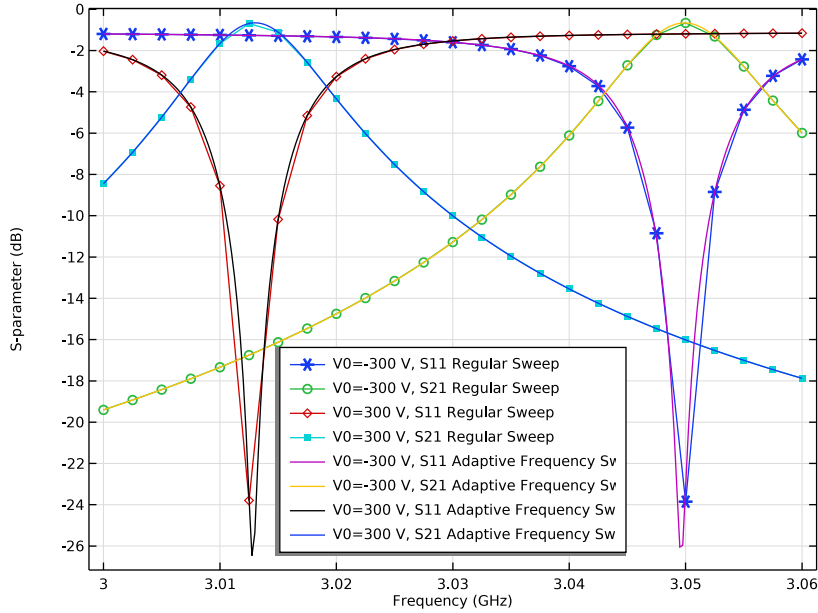
#### *Global 2*

- 1 Right-click **Results > S-Parameter (emw) 1 > Global 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 2/Parametric Solutions 2 (sol8)**.
- 4 Locate the **y-Axis Data** section. In the table, enter the following settings:


Expression	Unit	Description
emw.S11dB	dB	S11 Adaptive Frequency Sweep
emw.S21dB	dB	S21 Adaptive Frequency Sweep

- 5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **None**.


6 In the **S-Parameter (emw)** I toolbar, click  **Plot**.



#### *Streamline Multislice I*

- 1 In the **Model Builder** window, expand the **Results > Electric Potential (es)** node, then click **Streamline Multislice I**.
- 2 In the **Settings** window for **Streamline Multislice**, locate the **Streamline Positioning** section.
- 3 In the **Density level** text field, type 11.
- 4 In the **Electric Potential (es)** toolbar, click  **Plot**.

#### *Streamline Multislice I*

- 1 In the **Model Builder** window, expand the **Results > Electric Field (es)** node, then click **Streamline Multislice I**.
- 2 In the **Settings** window for **Streamline Multislice**, locate the **Streamline Positioning** section.
- 3 In the **Density level** text field, type 11.
- 4 In the **Electric Field (es)** toolbar, click  **Plot**.


### **ADD COMPONENT**

In the **Home** toolbar, click  **Add Component** and choose **3D**.

### **GLOBAL DEFINITIONS**

Load geometrical parameters from a file.

#### *Parameters I*


- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `tunable_cavity_filter_parameters.txt`.

### **GEOMETRY I**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry I**.
- 2 In the **Settings** window for **Geometry**, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.


First, create a block for the cavity.

#### *Cavity*

- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, type Cavity in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type 100.
- 4 In the **Depth** text field, type 50.
- 5 In the **Height** text field, type 50.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.

Add a substrate block.




#### *Substrate*

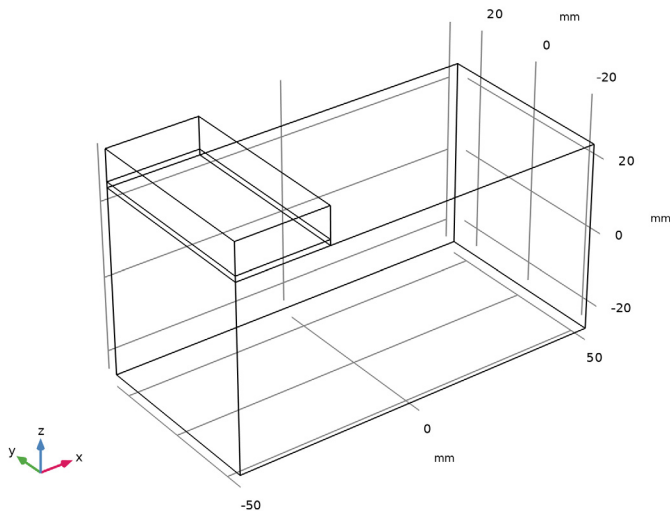
- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, type Substrate in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type 25.
- 4 In the **Depth** text field, type 50.

- 5 In the **Height** text field, type thickness.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** text field, type  $-37.5$ .
- 8 In the **z** text field, type  $25 + \text{thickness} / 2$ .

Add a block for the air domain.


#### *Air block*

- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, type Air block in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type 25.
- 4 In the **Depth** text field, type 50.
- 5 In the **Height** text field, type 10.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** text field, type  $-37.5$ .
- 8 In the **z** text field, type 30.
- 9 Click  **Build Selected**.
- 10 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.





Add a block for the microstrip line feed.

### *Feed\_line*

- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, type *Feed\_line* in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type  $l\_feed + w\_slot$ .
- 4 In the **Depth** text field, type 3.2.
- 5 In the **Height** text field, type thickness.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** text field, type  $-x\_slot - l\_feed/2$ .
- 8 In the **z** text field, type  $25 + thickness/2$ .

Add a work plane where you will draw a slot.



### *Work Plane 1 (wp1)*


- 1 In the **Geometry** toolbar, click  **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
- 3 In the **z-coordinate** text field, type 25.
- 4 Click  **Go to Plane Geometry**.

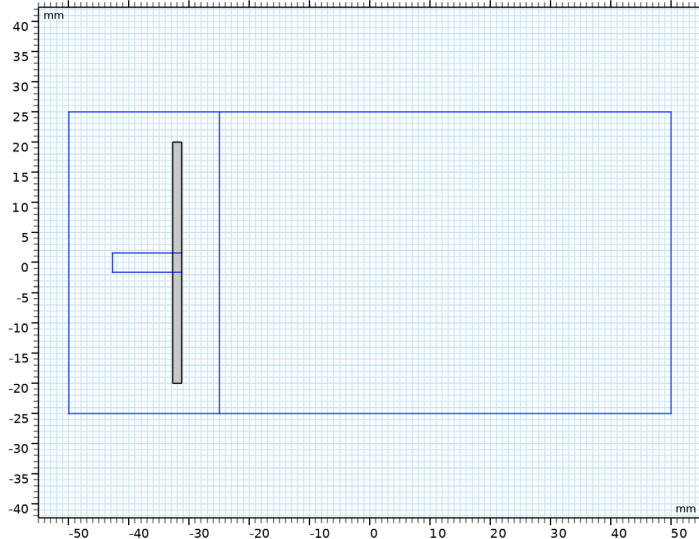
### *Work Plane 1 (wp1) > Plane Geometry*

Add a rectangle for the slot.

### *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  $w\_slot$ .
- 4 In the **Height** text field, type  $l\_slot$ .
- 5 Locate the **Position** section. From the **Base** list, choose **Center**.
- 6 In the **xw** text field, type  $-x\_slot$ .
- 7 Click  **Build Selected**.

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




Generate the 2nd slot coupled microstrip line by mirroring some geometries.


*Mirror 1 (mir1)*

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms > Mirror**.
- 2 Select the objects **blk2**, **blk3**, **blk4**, and **wp1** only.
- 3 In the **Settings** window for **Mirror**, locate the **Input** section.
- 4 Select the **Keep input objects** checkbox.
- 5 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type 1.
- 6 In the **z** text field, type 0.
- 7 Click  **Build All Objects**.

Add a block for the metal post in the middle of the cavity.



*Post*

- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, type **Post** in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type 15.
- 4 In the **Depth** text field, type 15.
- 5 In the **Height** text field, type 50-gap\_post.
- 6 Locate the **Position** section. In the **x** text field, type -7.5.

- 7 In the **y** text field, type -7.5.
- 8 In the **z** text field, type -25.
- 9 Click  **Build Selected**.





Add a cylinder for the piezo actuator disk.

#### *Piezo actuator*

- 1 In the **Geometry** toolbar, click  **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, type Piezo actuator in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Radius** text field, type 21.
- 4 In the **Height** text field, type 0.5.
- 5 Locate the **Position** section. In the **z** text field, type 25.
- 6 Click  **Build Selected**.

The inside of the metal post is not part of the modeling domain. Therefore, subtract it from the cavity.

#### *Difference 1 (dif1)*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **blk1** 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 **blk5** only.
- 6 Click  **Build All Objects**.
- 7 Click the  **Zoom Extents** button in the **Graphics** toolbar.