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.

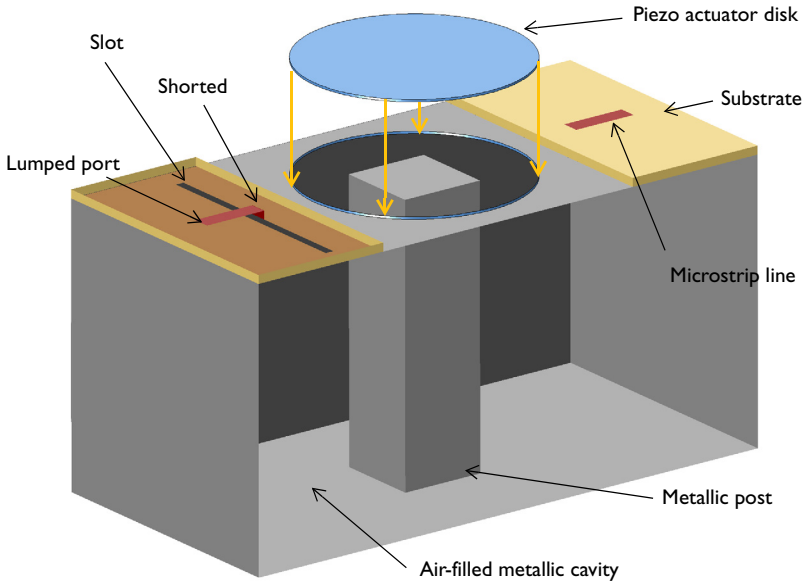


Figure 1: A tunable evanescent mode cavity filter is composed of a rectangular cavity with a metallic post, a piezo actuator disk, and slot-coupled microstrip lines. There is a small gap between the top of the post and the bottom side of the piezo actuator. The front part of the cavity wall is removed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

Note: In addition to the RF Module, this model 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

$$f_{nml} = \frac{c}{2\pi\sqrt{\varepsilon_r\mu_r}}\sqrt{\left(\frac{m\pi}{a}\right)^2+\left(\frac{n\pi}{b}\right)^2+\left(\frac{l\pi}{d}\right)^2} \quad (1)$$

where a and b are the waveguide aperture dimensions and d is the length of the waveguide cavity. In this model, 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 Ω 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 ~90 µm 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

 ©2013 COMSOL

between the top of the post and the bottom of the piezo device, strong electric fields are observed.

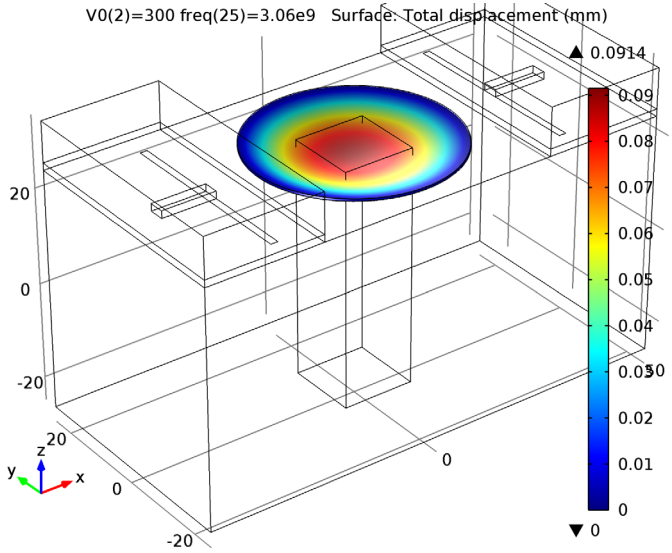


Figure 2: This plot shows the total piezo displacement when 300 V is applied on the actuator. The visualization is exaggerated to emphasize the deflection.

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 model is ~40MHz. This range can be adjusted by different choices of the piezo disk size and the input bias voltage.

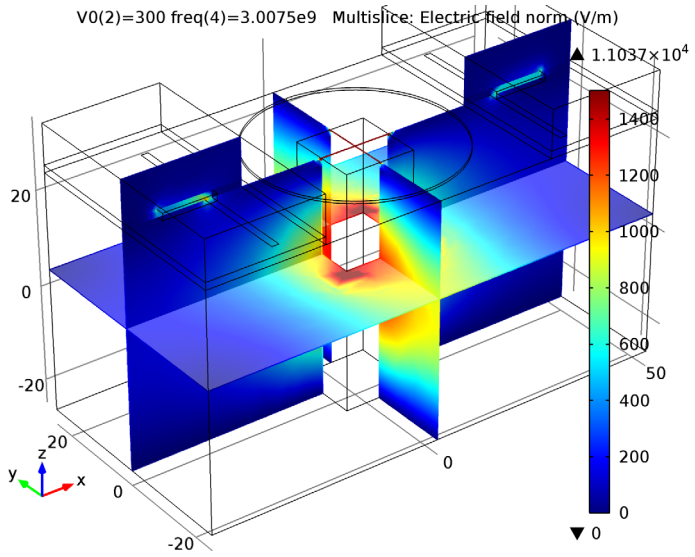


Figure 3: The dominant mode inside the cavity is observed from the electric field distribution plot.

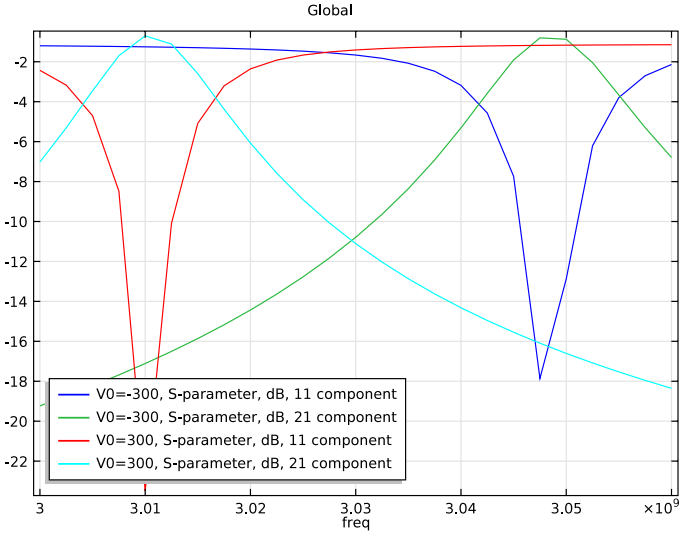


Figure 4: The deflection for the piezo device controlled by the input bias can shift the resonance frequency of the filter.

 ©2013 COMSOL

Notes About the COMSOL Implementation

Solve this model with three physics interfaces: Piezoelectric Devices; Moving Mesh; and Electromagnetic Waves, Frequency Domain. Use a Stationary study for the Piezoelectric Devices and Moving Mesh interfaces, and a Frequency Domain study for the Electromagnetic Waves interface.

Model Library path: `RF_Module/Passive_Devices/tunable_cavity_filter`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Piezoelectric Devices (pzd)**.

4 Click **Add Selected**.

5 In the **Add physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

6 Click **Add Selected**.

7 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

8 Click **Add Selected**.

9 Click **Next**.

10 Find the **Studies** subsection. In the tree, select **Custom Studies>Empty Study**.

11 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
thickness	60[mil]	Substrate thickness
l_slot	40[mm]	Slot length
w_slot	1.5[mm]	Slot width
x_slot	32[mm]	Slot location
l_feed	10[mm]	Feed line length
h_post	49.85[mm]	Post height
V0	300[V]	Piezo actuator bias
f_min	3[GHz]	Minimum frequency in sweep
f_max	3.06[GHz]	Maximum frequency in sweep
lda0	c_const/f_max	Minimum wavelength, air
h_max	0.2*lda0	Maximum element size, air

Here, `c_const` is a predefined COMSOL constant for the speed of light in vacuum.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

First, create a block for the cavity.

Block 1

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

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type 100.

4 In the **Depth** edit field, type 50.

5 In the **Height** edit field, type 50.

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

7 Right-click **Model 1>Geometry 1>Block 1** and choose **Rename**.

8 Go to the **Rename Block** dialog box and type `Cavity` in the **New name** edit field.

9 Click **OK**.

Add a substrate block.

 ©2013 COMSOL

Block 2

1 Right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type `25`.

4 In the **Depth** edit field, type `50`.

5 In the **Height** edit field, type `thickness`.

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

7 In the **x** edit field, type `-37.5`.

8 In the **z** edit field, type `25+thickness/2`.

9 Right-click **Model 1>Geometry 1>Block 2** and choose **Rename**.

10 Go to the **Rename Block** dialog box and type `Substrate` in the **New name** edit field.

11 Click **OK**.

Add a block for the air domain.

Block 3

1 Right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type `25`.

4 In the **Depth** edit field, type `50`.

5 In the **Height** edit field, type `10`.

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

7 In the **x** edit field, type `-37.5`.

8 In the **z** edit field, type `30`.

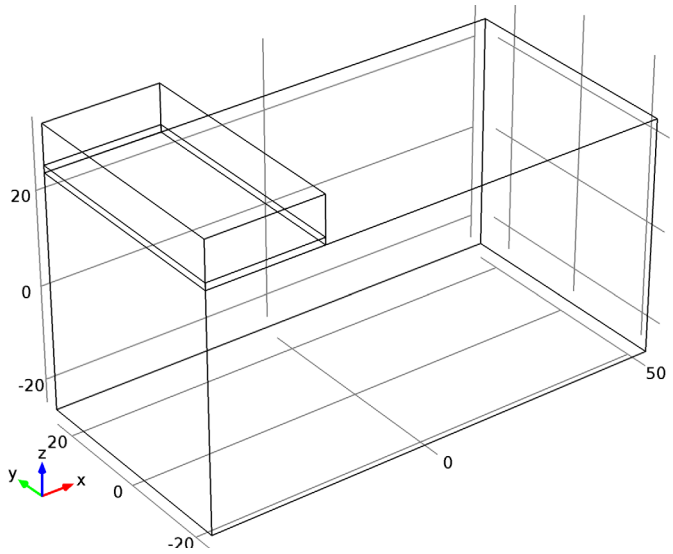
9 Right-click **Model 1>Geometry 1>Block 3** and choose **Rename**.

10 Go to the **Rename Block** dialog box and type `Air block` in the **New name** edit field.

11 Click **OK**.

12 Right-click **Model 1>Geometry 1>Block 3** and choose **Build Selected**.

13 Click the **Wireframe Rendering** button on the Graphics toolbar.



Add a block for the microstrip line feed.

Block 4

1 Right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type `l_feed+w_slot`.

4 In the **Depth** edit field, type `3.2`.

5 In the **Height** edit field, type `thickness`.

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

7 In the **x** edit field, type `-x_slot-l_feed/2`.

8 In the **z** edit field, type `25+thickness/2`.

9 Right-click **Model 1>Geometry 1>Block 4** and choose **Rename**.

10 Go to the **Rename Block** dialog box and type `Feed line` in the **New name** edit field.

11 Click **OK**.

Add a work plane where you will draw a slot.

Work Plane 1

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

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

 ©2013 COMSOL

3 In the **z-coordinate** edit field, type `25`.

Plane Geometry

Add a rectangle for the slot.

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 Click the **Zoom Extents** button on the Graphics toolbar.

Rectangle 1

1 In the **Rectangle** settings window, locate the **Size** section.

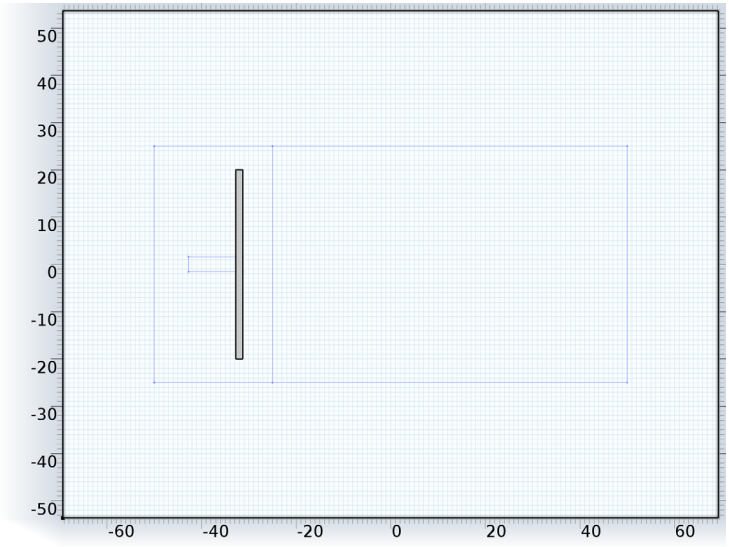
2 In the **Width** edit field, type `w_slot`.

3 In the **Height** edit field, type `l_slot`.

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

5 In the **xw** edit field, type `-x_slot`.

6 Click the **Build Selected** button.



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

Mirror 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Mirror**.

2 Select the objects **blk3**, **blk4**, **blk2**, and **wp1** only.

3 In the **Mirror** settings window, locate the **Input** section.

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

5 Locate the **Normal Vector to Plane of Reflection** section. In the **x** edit field, type `1`.

6 In the **z** edit field, type `0`.

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

Block 5

1 Right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type `15`.

4 In the **Depth** edit field, type `15`.

5 In the **Height** edit field, type `h_post`.

6 Locate the **Position** section. In the **x** edit field, type `-7.5`.

7 In the **y** edit field, type `-7.5`.

8 In the **z** edit field, type `-25`.

9 Right-click **Model 1>Geometry 1>Block 5** and choose **Rename**.

10 Go to the **Rename Block** dialog box and type `Post` in the **New name** edit field.

11 Click **OK**.

Add a cylinder for the piezo actuator disk.

Cylinder 1

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `21`.

4 In the **Height** edit field, type `0.5`.

5 Locate the **Position** section. In the **z** edit field, type `25`.

6 Right-click **Model 1>Geometry 1>Cylinder 1** and choose **Rename**.

7 Go to the **Rename Cylinder** dialog box and type `Piezo actuator` in the **New name** edit field.

8 Click **OK**.

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

Difference 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

 ©2013 COMSOL

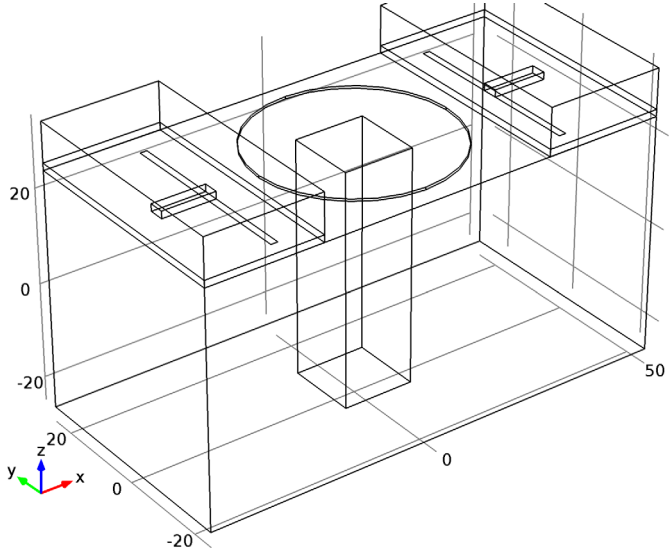
2 Select the object **blk1** only.

3 In the **Difference** settings window, locate the **Difference** section.

4 Under **Objects to subtract**, click **Activate Selection**.

5 Select the object **blk5** only.

6 Click the **Build All** button.



DEFINITIONS

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

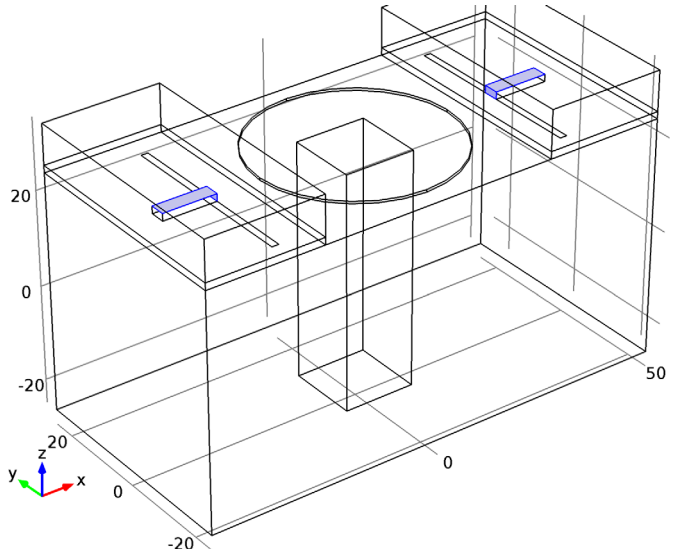
Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

4 Select Boundaries 17, 22, 47, and 50 only.



5 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Feed line` in the **New name** edit field.

7 Click **OK**.

Add a selection for the ground.

Explicit 2

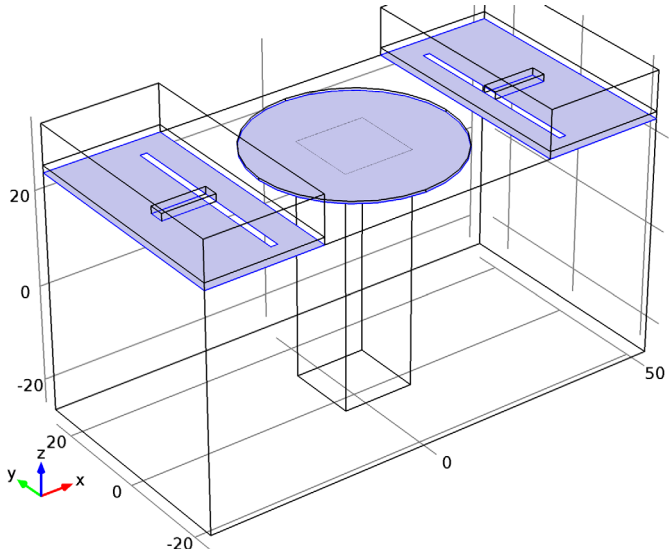
1 Right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

 ©2013 COMSOL

4 Select Boundaries 6, 16, 28, 39, and 53 only.



5 Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Ground` in the **New name** edit field.

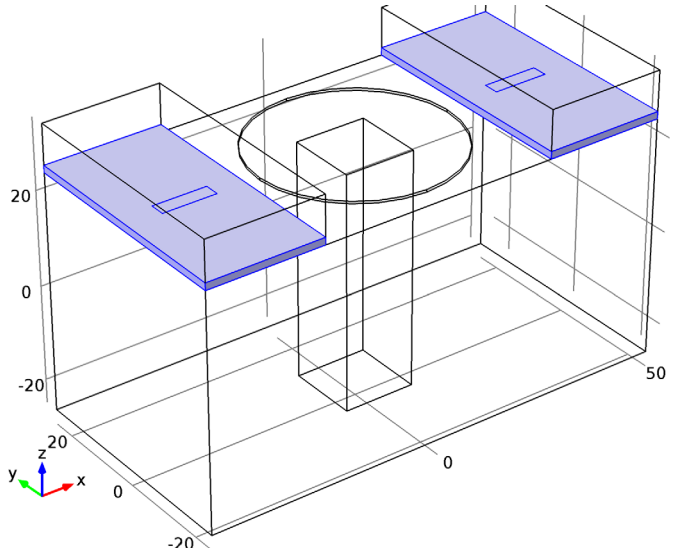
7 Click **OK**.

Add a selection for the substrate.

Explicit 3

1 Right-click **Definitions** and choose **Selections>Explicit**.

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



3 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Substrate` in the **New name** edit field.

5 Click **OK**.

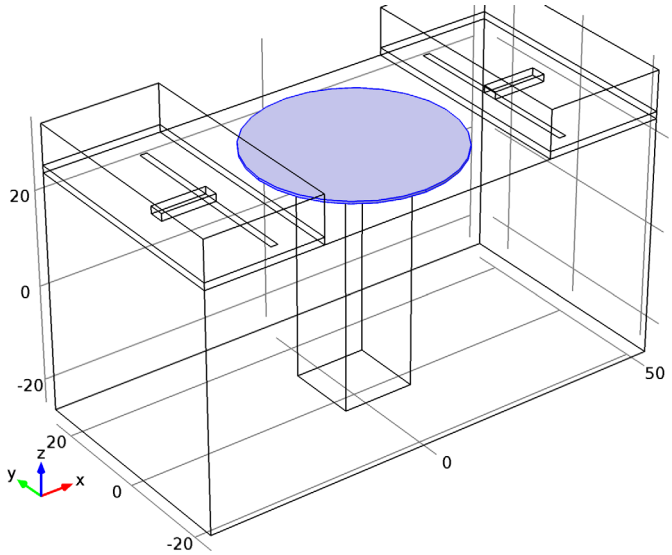
Add a selection for the piezo actuator disk.

Explicit 4

1 Right-click **Definitions** and choose **Selections>Explicit**.

 ©2013 COMSOL

2 Select Domain 5 only.



3 Right-click **Model 1>Definitions>Explicit 4** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Piezo actuator` in the **New name** edit field.

5 Click **OK**.

Now, set up the physics. Start by specifying the Piezoelectric Devices domain.

PIEZOELECTRIC DEVICES

1 In the **Piezoelectric Devices** settings window, locate the **Domain Selection** section.

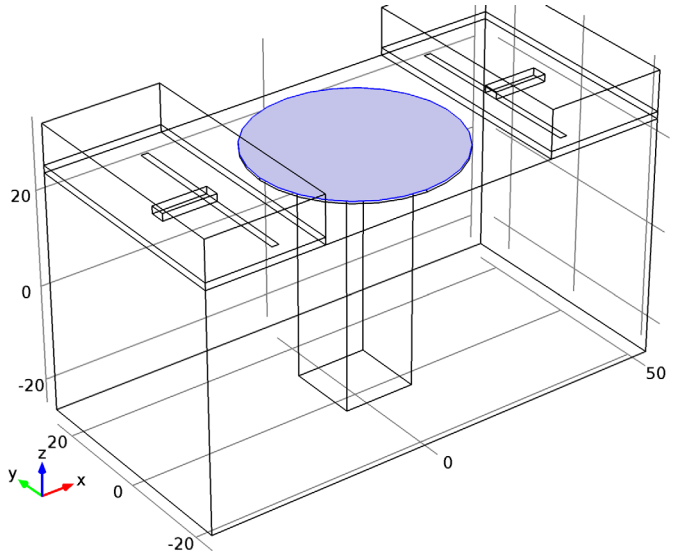
2 From the **Selection** list, choose **Piezo actuator**.

Next, define the electric potential and ground on the piezo actuator.

Electric Potential 1

1 In the **Model Builder** window, right-click **Piezoelectric Devices** and choose the boundary condition **Electrical>Electric Potential**.

2 Select Boundary 29 only.



3 In the **Electric Potential** settings window, locate the **Electric Potential** section.

4 In the V_0 edit field, type `-V0`.

Ground 1

1 Right-click **Piezoelectric Devices** and choose the boundary condition **Electrical>Ground**.

2 In the **Ground** settings window, locate the **Boundary Selection** section.

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

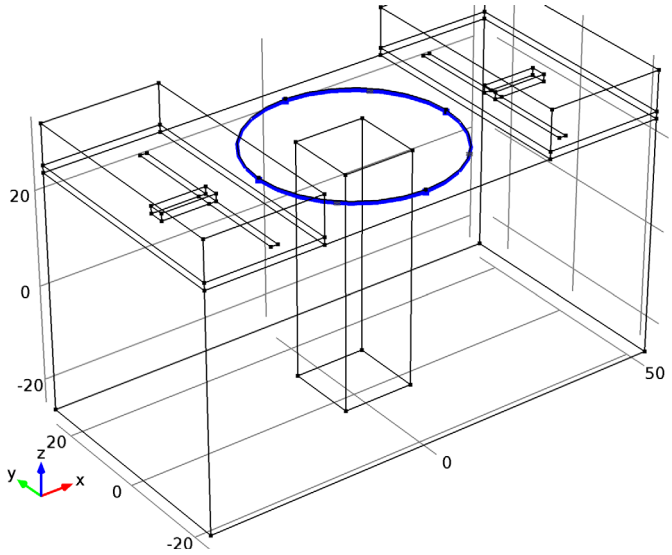
Assume this bottom rim part is attached on the same size circular aperture of the cavity top and no deflection is expected.

Fixed Constraint 1

1 Right-click **Piezoelectric Devices** and choose **Edges>Fixed Constraint**.

 ©2013 COMSOL

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



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

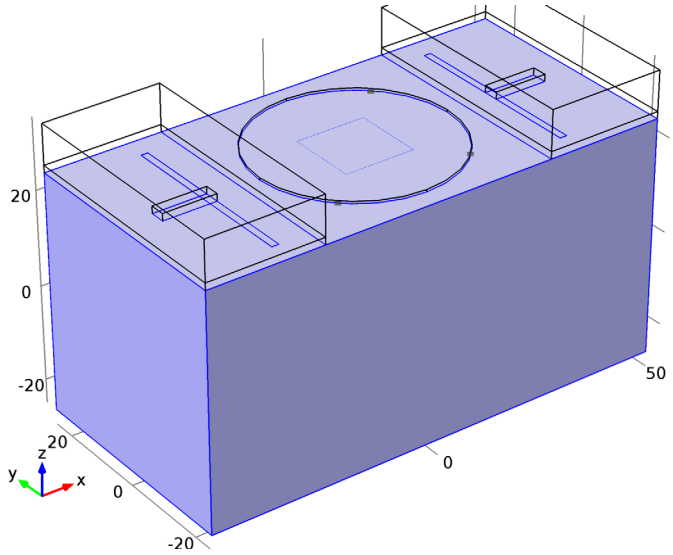
MOVING MESH

1 In the **Model Builder** window, under **Model 1** click **Moving Mesh**.

2 In the **Moving Mesh** settings window, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.



Prescribed Mesh Displacement 1

1 In the **Model Builder** window, under **Model 1>Moving Mesh** click **Prescribed Mesh Displacement 1**.

2 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.

3 In the d_x edit field, type u.

4 In the d_y edit field, type v.

5 In the d_z edit field, type w.

Free Deformation 1

1 In the **Model Builder** window, right-click **Moving Mesh** and choose **Free Deformation**.

2 Select Domain 1 only.

Prescribed Mesh Displacement 2

1 Right-click **Moving Mesh** and choose **Prescribed Mesh Displacement**.

2 In the **Prescribed Mesh Displacement** settings window, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 Go to the **Paste Selection** dialog box.

 ©2013 COMSOL

5 In the **Selection** edit field, type `1-27,29-57`.

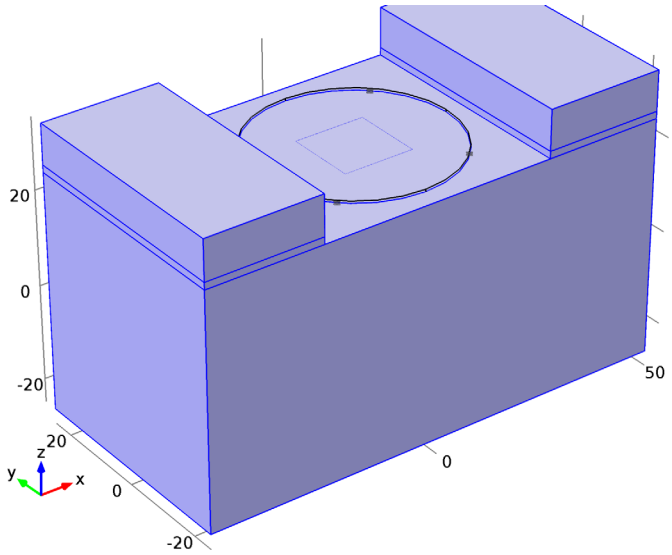
6 Click the **OK** button.

Set up the Electromagnetic Waves, Frequency Domain interface. Suppress the piezo actuator disk domain.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

1 In the **Model Builder** window, under **Model 1** click **Electromagnetic Waves, Frequency Domain**.

2 Select Domains 1–4 and 6–8 only.



Perfect Electric Conductor 2

1 Right-click **Model 1>Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Feed line**.

Perfect Electric Conductor 3

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

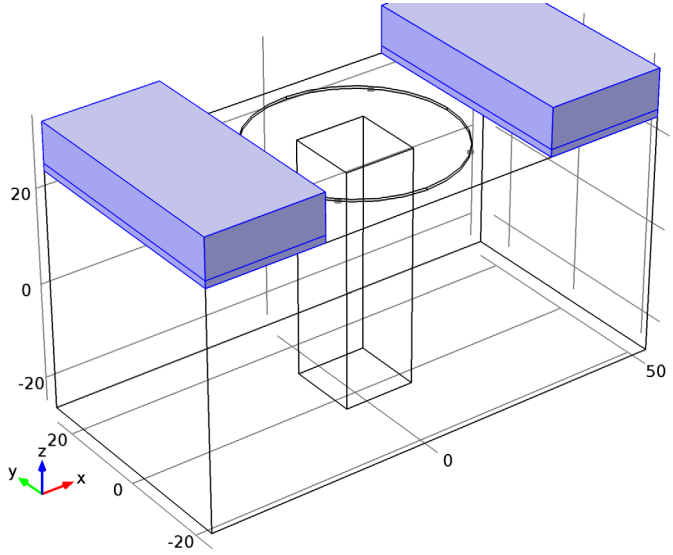
2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.

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

Scattering Boundary Condition 1

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Scattering Boundary Condition**.

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

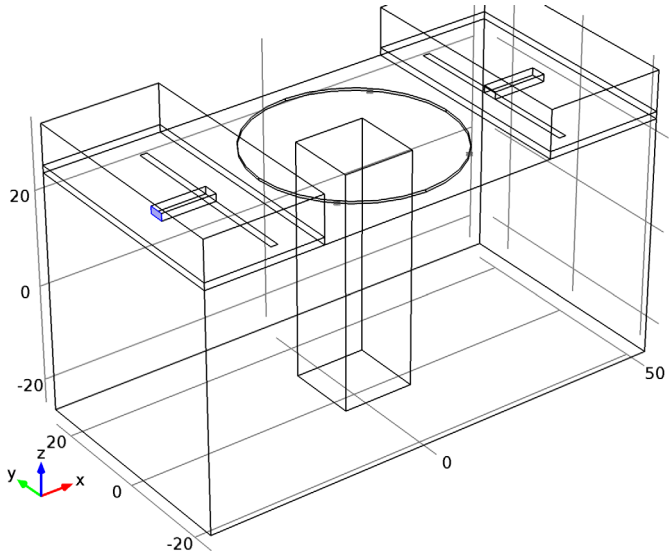


Lumped Port 1

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.

 ©2013 COMSOL

2 Select Boundary 14 only.



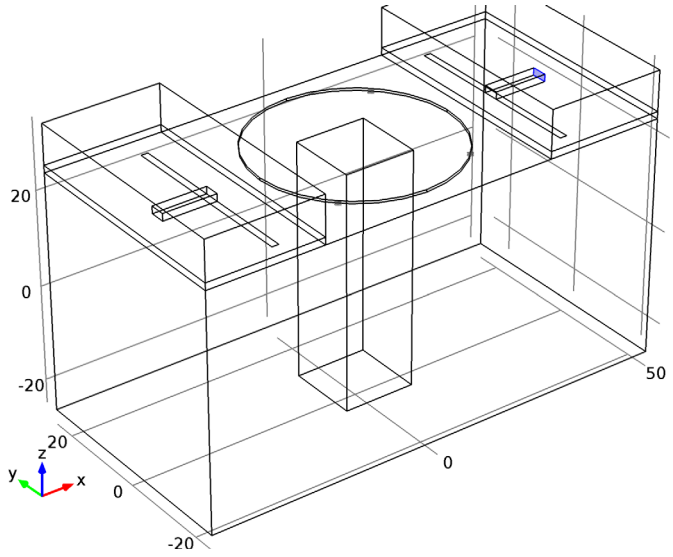
3 In the **Lumped Port** settings window, locate the **Port Properties** section.

4 From the **Wave excitation at this port** list, choose **On**.

Lumped Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.

2 Select Boundary 54 only.



MATERIALS

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

Material Browser

1. In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
2. In the **Material Browser** settings window, In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.
3. Click **Add Material to Model**.

Lead Zirconate Titanate (PZT-5H)

1. In the **Model Builder** window, under **Model 1>Materials** click **Lead Zirconate Titanate (PZT-5H)**.
2. In the **Material** settings window, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Piezo actuator**.

Material Browser

1. In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
2. In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

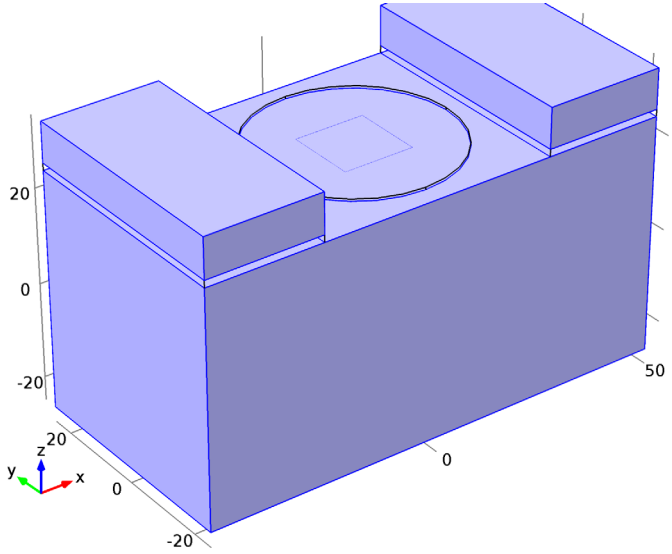
 ©2013 COMSOL

3 Click **Add Material to Model**.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 Select Domains 1, 3, 4, 7, and 8 only.



Material 3

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

4 In the **Material** settings window, locate the **Material Contents** section.

5 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilonr	3.38
Relative permeability	mur	1
Electrical conductivity	sigma	0

6 Right-click **Model 1>Materials>Material 3** and choose **Rename**.

7 Go to the **Rename Material** dialog box and type `Substrate` in the **New name** edit field.

8 Click **OK**.

MESH 1

Choose the maximum mesh size in each material domain smaller than 0.2 wavelengths using the parameter `h_max` that you defined.

Size 1

1. In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Size**.
2. In the **Size** settings window, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
5. In the associated edit field, type `h_max/2`.

Size 2

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
2. In the **Size** settings window, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Substrate**.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7. In the associated edit field, type `h_max/sqrt(3.38)`.

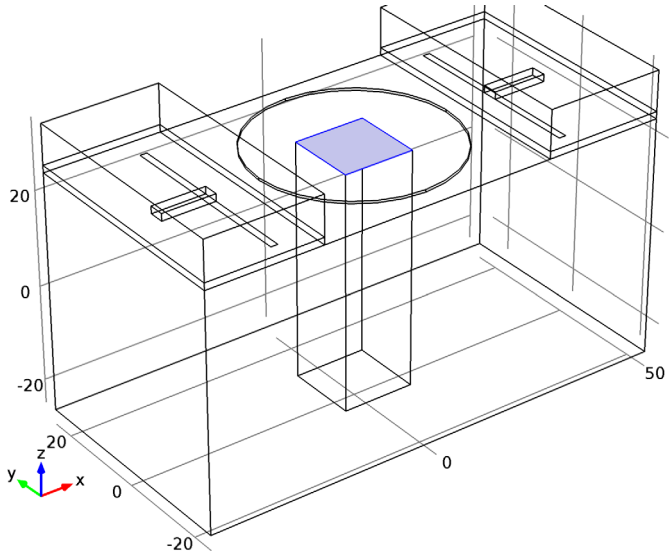
Add a finer mesh size on the top of the metallic post inside the cavity where strong electric fields are confined.

Size 3

1. Right-click **Mesh 1** and choose **Size**.
2. In the **Size** settings window, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.

 ©2013 COMSOL

4 Select Boundary 32 only.



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

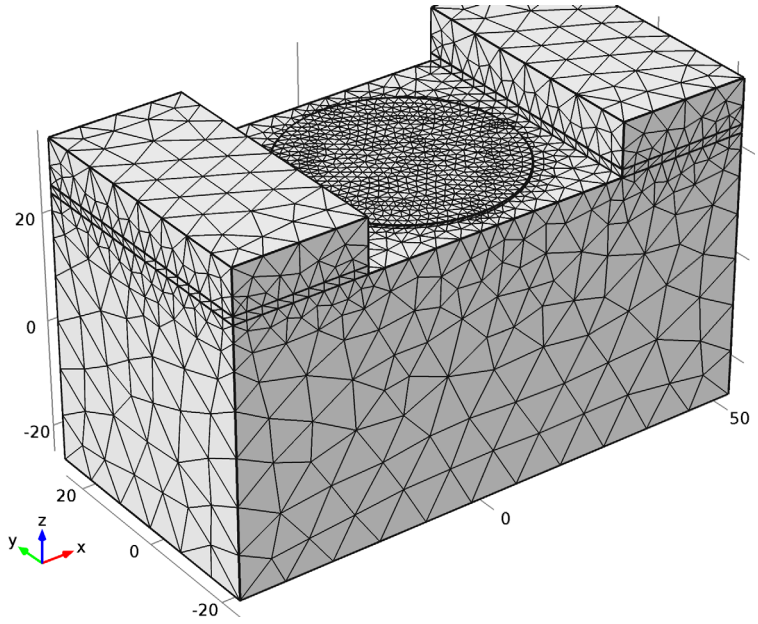
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `h_max/10`.

Free Tetrahedral 1

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

2 In the **Settings** window, click **Build All**.



STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Stationary**.

2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	Solve for
Electromagnetic Waves, Frequency Domain	×

Step 2: Frequency Domain

1 Right-click **Study 1** and choose **Study Steps>Frequency Domain**.

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

3 In the **Frequencies** edit field, type `range(f_min,0.0025[GHz],f_max)`.

 ©2013 COMSOL

4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics	Solve for
Piezoelectric Devices	×
Moving Mesh	×

Parametric Sweep

1 Right-click **Study 1** and choose **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter names	Parameter value list
V0	-300,300

5 Right-click **Study 1** and choose **Compute**.

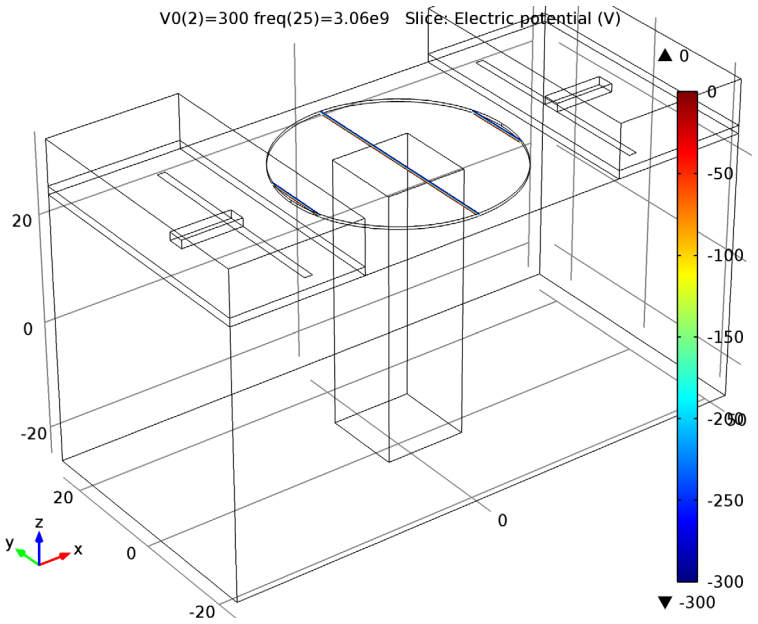
RESULTS

There are three default plots: the displacement, the piezo actuator potential, and the electric field norm in the cavity filter.

Displacement (pzd)

This plot shows the deflected piezo actuator disk; compare with Figure 2.

Potential (pzd)



The electric potential inside the piezo actuator disk.

Electric Field (emw)

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

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

3 From the **Parameter value (freq)** list, choose **3.0075e9**.

4 In the **Model Builder** window, under **Results>Electric Field (emw)** click **Multislice**.

5 In the **Multislice** settings window, click to expand the **Range** section.

6 Select the **Manual color range** check box.

7 Set the **Maximum** value to **3**.

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.

1D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 3**.

 ©2013 COMSOL

4 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

5 Right-click **Results>1D Plot Group 4** and choose **Global**.

6 In the **Global** settings window, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 11 component (emw.S11dB)**.

7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 21 component (emw.S21dB)**.

8 Click the **Plot** button.

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

 ©2013 COMSOL