Evanescent Mode Cylindrical Cavity Filter

Introduction

An evanescent mode cavity filter is resonant at a frequency lower than the dominant resonant frequency of a metallic cavity. Such evanescent mode resonance can be realized by creating a discontinuity or reactance inside the cavity.

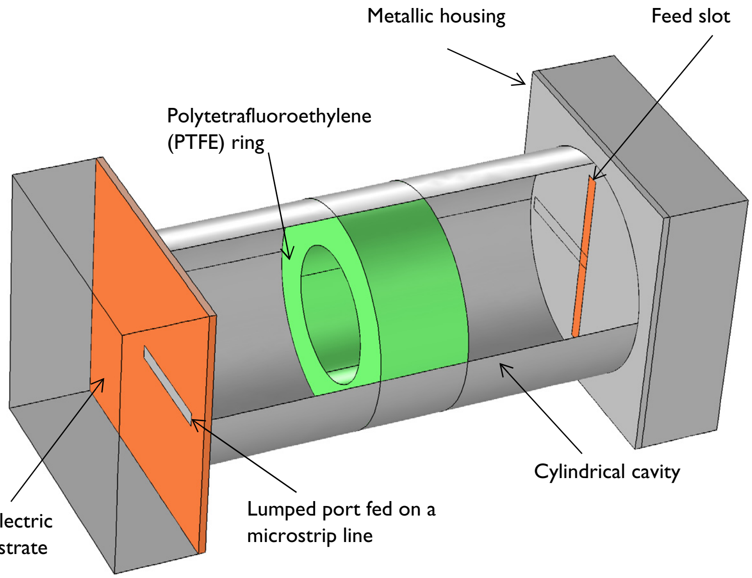


Figure 1: An evanescent mode cavity filter. The signal fed from a microstrip line is slot coupled into the cylindrical cavity loaded with a PTFE ring.

Model Definition

The resonant frequency of the empty cylindrical waveguide cavity TE_{111} mode can be calculated from the equation

$$f_{nml} = \frac{c}{2\pi\sqrt{\varepsilon_r\mu_r}}\sqrt{\left(\frac{p'_{nm}}{a}\right)^2 + \left(\frac{l\pi}{d}\right)^2}$$

where a and d are the radius and length of the cylinder, respectively, and p'_{nm} is the mth root of the Bessel function $J'_n(x)$. The TE_{111} mode is the dominant TE mode of the cylindrical cavity resonator, and for a cavity of 25 mm radius and 100 mm height this resonance is at 3.823 GHz. The starting point of this example was a computation (not presented here) of the TE_{111} mode resonant frequency of an empty cylindrical cavity and a subsequent verification of agreement with the analytic solution.

This basic model was then modified by the addition of a metal box at either end representing a housing. Inside is a dielectric substrate and a microstrip line which is slot coupled into the cavity. This represents the input and output of the device.

The slots are located on the center of the cavity ends to induce symmetric fields and they are also parallel to each other to couple the injected fields maximally. The size of the slots are tuned to provide a better matching to the reference characteristic impedance assigned on ports. The model uses lumped ports to excite the structure. The end of each microstrip line over the slots is shorted to couple the fields from the microstrip lines through the slots and vice versa. The cavity is partially filled with a ring of PTFE, $\varepsilon_r = 2.1$, which causes the resonant frequency to shift down.

Results and Discussion

Figure 2 shows the frequency response of the cavity. The dielectric ring causes the resonant frequency to shift down to 3.53 GHz. This example shows that the center frequency of the device can be lowered without increasing the size, while the insertion loss is still as good as for an air-filled cavity. The electric field distribution in Figure 3 shows a basic resonant mode and the dielectric tube inside the cavity does not distort the distribution significantly.

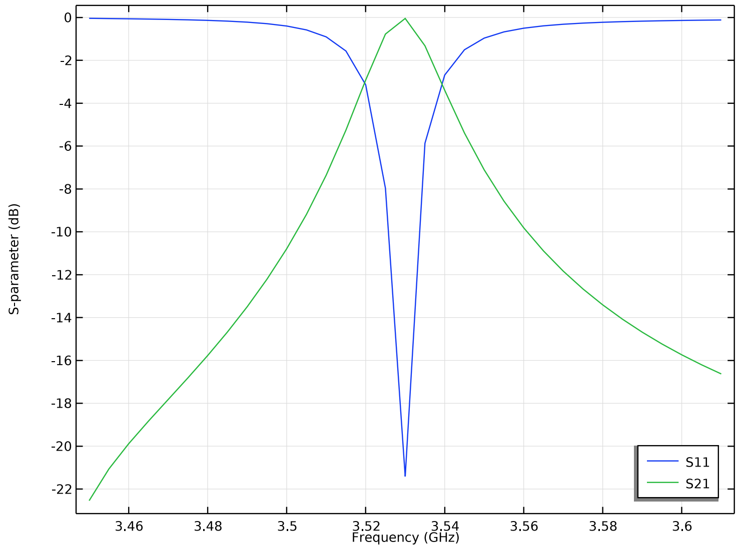


Figure 2: The frequency response of the filter shows bandpass filter characteristics. The center frequency is lower than the dominant mode resonant frequency of the metallic cavity.

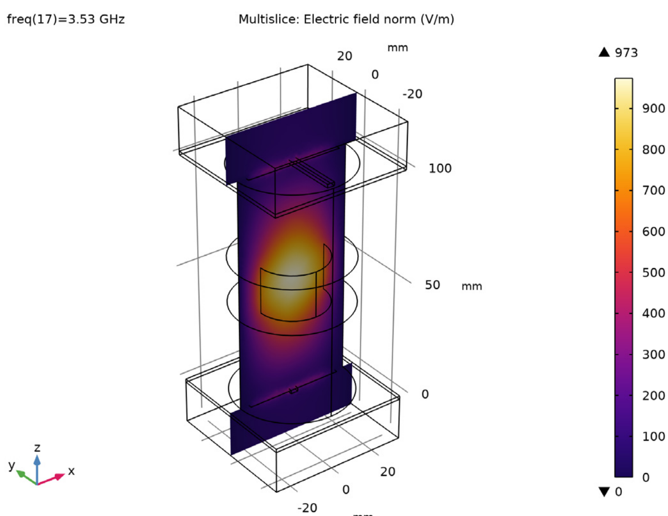


Figure 3: The dielectric tube inside the cavity does not distort the electric field distribution at resonance significantly.

Reference

1. D.M. Pozar, *Microwave Engineering*, John Wiley & Sons, 1998.

Application Library path: `RF_Module/Filters/cylindrical_cavity_filter_evanescent`

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 **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

STUDY 1

Step 1: Frequency Domain

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

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

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

3 In the **Frequencies** text field, type `range(3.45[GHz],5[MHz],3.61[GHz])`.

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
d	60[mil]	0.001524 m	Substrate thickness
l_slot	42[mm]	0.042 m	Slot length
w_slot	3[mm]	0.003 m	Slot width

Here `mil` refers to the unit milliinch, that is 1 mil = 0.0254 mm.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Create a cylindrical cavity.

Cavity

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

2 In the **Settings** window for **Cylinder**, type `Cavity` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `25`.

4 In the **Height** text field, type `100`.

5 Click **Build Selected**.

Create a coupling slot.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Plane Geometry

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

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 `l_slot`.

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

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

6 Click **Build Selected**.

Create a substrate.

Bottom_plate

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

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

3 Locate the **Size and Shape** section. In the **Width** text field, type `60`.

4 In the **Depth** text field, type `60`.

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

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

7 In the **z** text field, type `-d/2`.

8 Click **Build Selected**.

Create a 50Ω microstrip line.

Bottom_feed

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

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

3 Locate the **Size and Shape** section. In the **Width** text field, type `3.2`.

4 In the **Depth** text field, type `25`.

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

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

7 In the **y** text field, type `25/2-w_slot/2`.

8 In the **z** text field, type `-d/2`.

9 Click **Build Selected**.

Create a metallic housing.

Housing

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

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

3 Locate the **Size and Shape** section. In the **Width** text field, type `60`.

4 In the **Depth** text field, type `60`.

5 In the **Height** text field, type `20`.

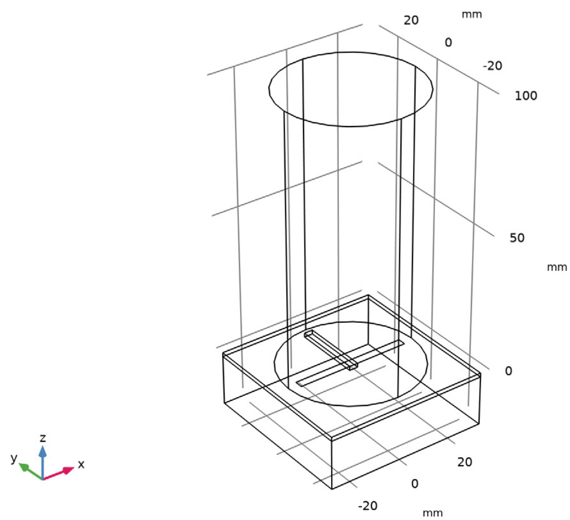
6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **z** text field, type `-10`.

8 Click **Build Selected**.

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

10 Click the **Wireframe Rendering** button in the **Graphics** toolbar, to see the interior.



Create a pair of slots, substrates, microstrip lines, and metallic housings.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **blk1**, **blk2**, **blk3**, and **wp1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `0,180`.

5 Locate the **Point on Axis of Rotation** section. In the **z** text field, type `50`.

6 Locate the **Rotation** section. From the **Axis type** list, choose **Cartesian**.

7 In the **x** text field, type `1`.

8 In the **z** text field, type `0`.

9 Click **Build Selected**.

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

Create a dielectric ring.

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 25.
4 In the **Height** text field, type 20.
5 Locate the **Position** section. In the **z** text field, type 40.

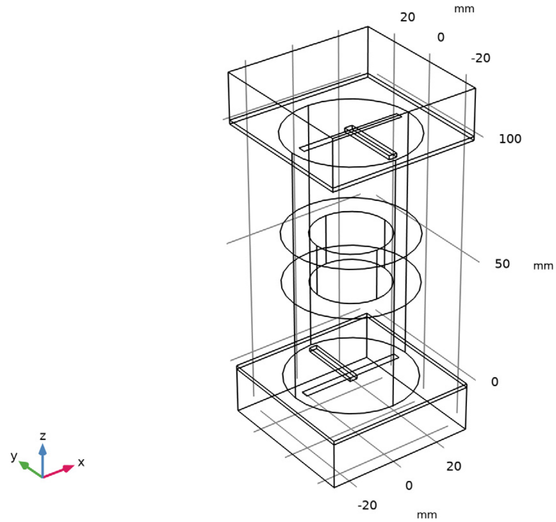
Cylinder 3 (cyl3)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 15.
4 In the **Height** text field, type 20.
5 Locate the **Position** section. In the **z** text field, type 40.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **cyl2** 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 **cyl3** only.

6 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

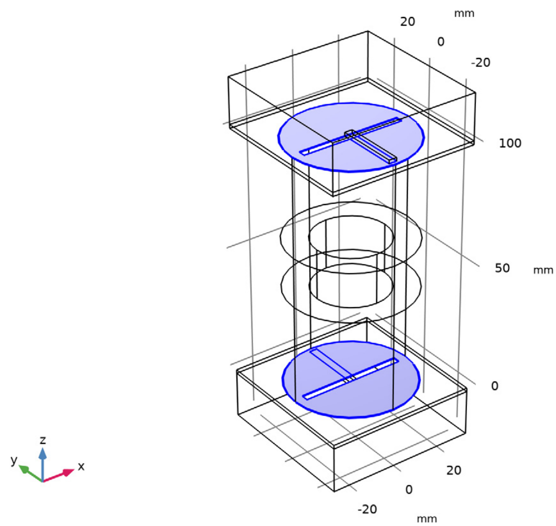
The default boundary condition is perfect electric conductor, which applies to all exterior boundaries. Assign a perfect electric conductor condition to the remaining boundaries of the cavity.

Perfect Electric Conductor 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose the boundary condition **Perfect Electric Conductor**.

2 Select Boundaries 21, 28, 35, and 42 only.

You can do this most easily by copying the text '21, 28, 35, and 42', clicking in the selection box, and then pressing Ctrl+V, or by using the Paste Selection dialog box.

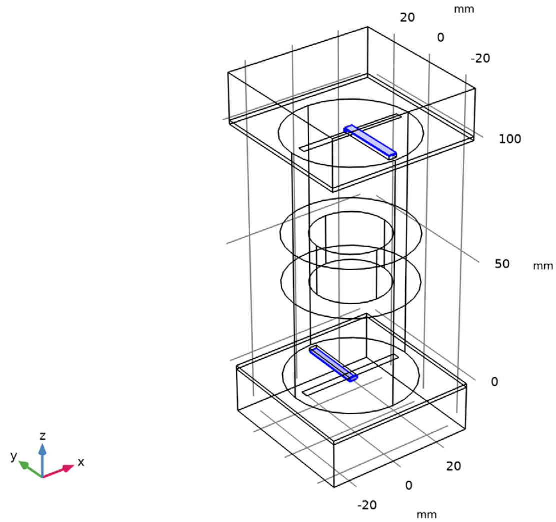


Proceed to define boundary condition for the shorted microstrip lines.

Perfect Electric Conductor 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.

2 Select Boundaries 36, 38, 39, and 43 only.



Lumped Port 1

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

2 Select Boundary 44 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 34 only.

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 **Built-in>Air**.

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

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

MATERIALS

Create a substrate material.

Substrate

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Substrate` in the **Label** text field.

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

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	3.38	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Create a dielectric ring material.

PTFE

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `PTFE` in the **Label** text field.

3 Select Domain 6 only.

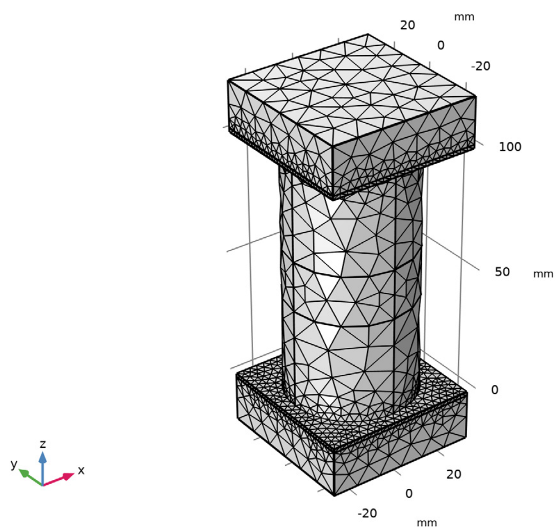
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	2.1	1	Basic

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



STUDY 1

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

RESULTS

Electric Field (emw)

The default plot shows the norm of the electric field for the highest frequency. Follow the instructions to reproduce Figure 3.

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

2 From the **Parameter value (freq (GHz))** list, choose **3.53**.

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Z-planes** subsection. In the **Planes** text field, type `0`.

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

6 In the **Color Table** dialog box, select **Thermal>HeatCamera** in the tree.

7 Click **OK**.

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

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

S-parameter (emw)

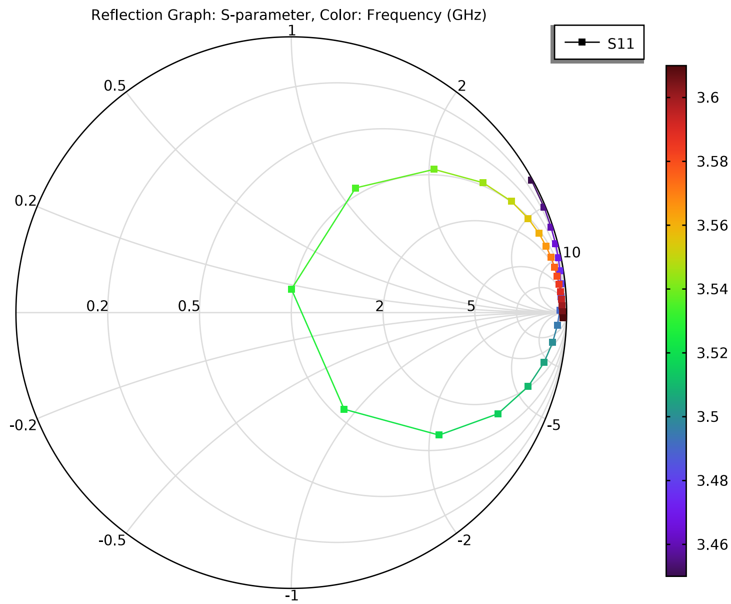
1 In the **Model Builder** window, under **Results** click **S-parameter (emw)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

Smith Plot (emw)

In the **Model Builder** window, click **Smith Plot (emw)**.



3D Plot Group 4

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

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

3 From the **Parameter value (freq (GHz))** list, choose **3.53**.

Isosurface 1

1 Right-click **3D Plot Group 4** and choose **Isosurface**.

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

3 In the **Expression** text field, type `20*log(emw.normE+1)`.

4 Locate the **Levels** section. In the **Total levels** text field, type `20`.

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

6 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Isosurface**, locate the **Coloring and Style** section.

9 From the **Color table transformation** list, choose **Reverse**.

Filter 1

1 Right-click **Isosurface 1** and choose **Filter**.

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

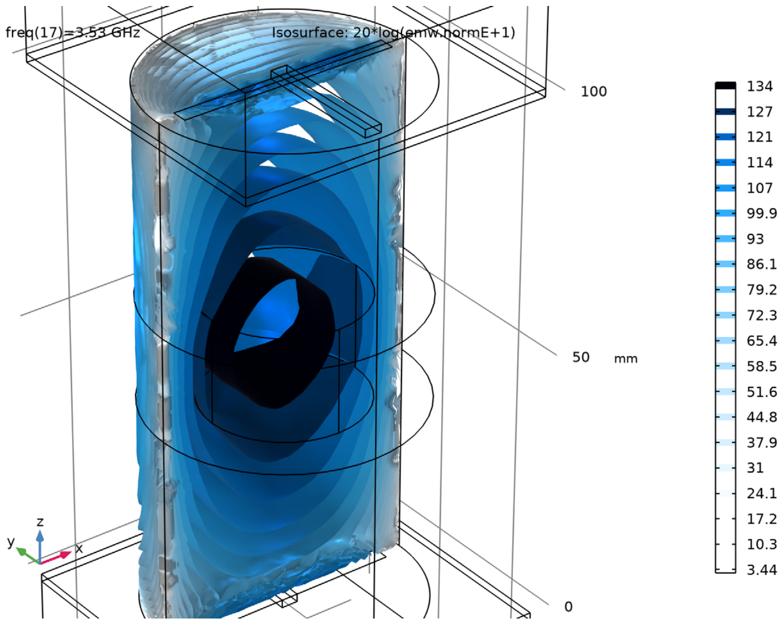
3 In the **Logical expression for inclusion** text field, type `y>0`.

Selection 1

1 In the **Model Builder** window, right-click **Isosurface 1** and choose **Selection**.

2 Select Domains 5 and 6 only.

3 In the **3D Plot Group 4** toolbar, click **Plot**.



Analyze the same model with a much finer frequency resolution using **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 50 times faster than regular Frequency Domain sweeps with a same finer 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 box, 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 box, 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 **Preset Studies for Selected Physics Interfaces>Adaptive Frequency Sweep**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Adaptive Frequency Sweep

1. In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Study Settings** section.
2. In the **Frequencies** text field, type `range(3.45[GHz],5[MHz]/50,3.61[GHz])`.

   Use a 50 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.

   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** check box 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.

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

Interface	Output
Electromagnetic Waves, Frequency Domain (emw)	Selection

4 Click to select row number 1 in the table.

5 Under **Selections**, click **Add**.

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

7 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.

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

RESULTS

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw) 1** node.

2 Right-click **Multislice** and choose **Delete**.

Surface 1

In the **Model Builder** window, right-click **Electric Field (emw) 1** and choose **Surface**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 34 and 44 only.

3 In the **Electric Field (emw) 1** toolbar, click **Plot**.

S-parameter (emw) 1

1 In the **Model Builder** window, under **Results** click **S-parameter (emw) 1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

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`	1	`S11 Adaptive Frequency Sweep`
`emw.S21dB`	1	`S21 Adaptive Frequency Sweep`

4 Right-click **Global 1** and choose **Duplicate**.

Global 2

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

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

3 In the table, enter the following settings:

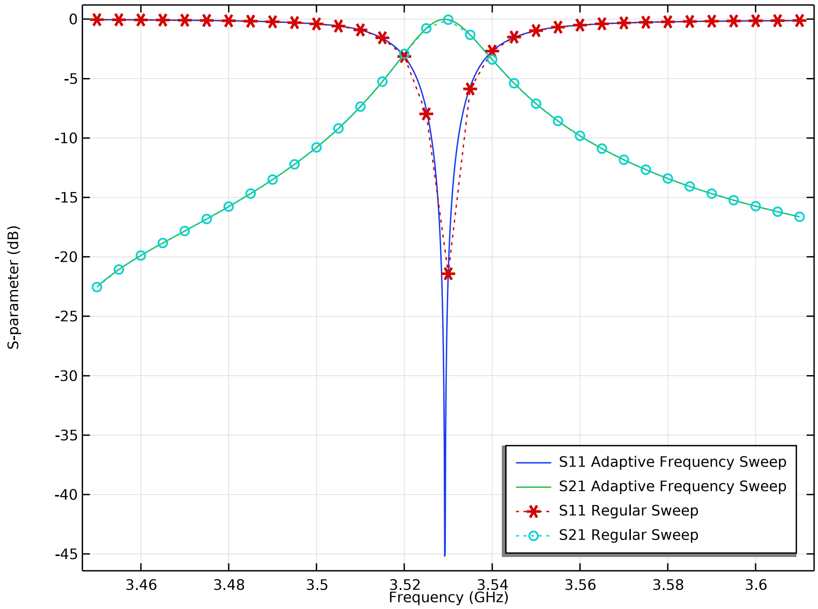
Expression	Unit	Description
`emw.S11dB`	1	`S11 Regular Sweep`
`emw.S21dB`	1	`S21 Regular Sweep`

4 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 In the **S-parameter (emw) 1** toolbar, click **Plot**.



Smith Plot (emw) 1

In the **Model Builder** window, under **Results** click **Smith Plot (emw) 1**.