Created in COMSOL Multiphysics 6.1

Coupled-Line Bandpass Filter

Introduction

A narrow-band bandpass filter can be realized using coupled microstrip lines. Each microstrip line piece is approximately a half wavelength long at the bandpass frequency. The bandpass performance of such a filter is much better than that of a single-section coupled-line filter.

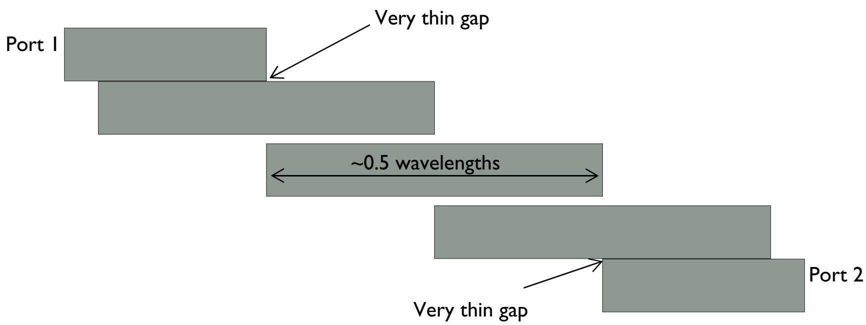


Figure 1: Layout of a coupled-line bandpass filter. It is composed of five sections of microstrip lines.

Model Definition

The form of the coupled microstrip line filter being modeled is shown in Figure 1. The layout was designed based upon Ref. 1 to have a center frequency at 3.6 GHz, and is composed of five sections of microstrip lines. The objective of this design is to have better out-of-band rejection compared to a design with fewer cascading strips. The length of each line section is tuned such that the impedance of the filter is 50 Ω. The ports are assumed to be coupled to a 50 Ω microstrip line that is not modeled.

The microstrip lines are each modeled as perfect electric conductor (PEC) surfaces on a 0.020 inch thick dielectric substrate, with another PEC surface below that acts as the ground plane. The entire modeling domain is bounded by PEC boundaries that represent the packaging.

Two rectangular surfaces are used to model the ports. These small rectangular surfaces that bridge between two PEC surfaces represent a connection to a 50 Ω transmission line. The device is excited at one port and S_{11} and S_{21} are monitored around the bandpass frequency. The model is shown in Figure 2.

Due to the thin gap between the microstrip lines, some care must be taken during the meshing. As a general rule of thumb, at least five elements per wavelength in each material are sufficient, and the elements should have an aspect ratio of approximately unity. However, this design also has a small gap between two of the microstrip lines. This leads to very small elements in this gap region when using the default mesh settings, and the solution time is quite long. An alternative approach is to modify the mesh settings such that the elements in the gap region have a higher aspect ratio. Despite this, the solution agrees well with a finer mesh, and uses less memory.

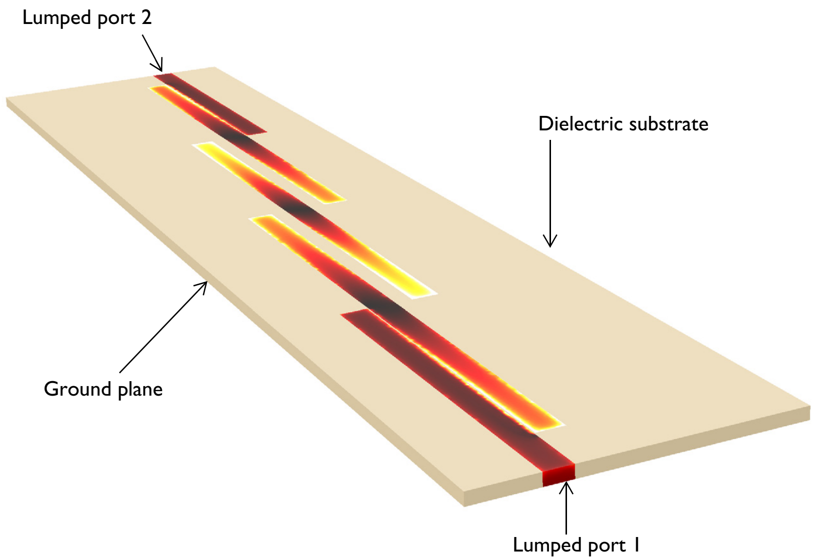


Figure 2: The model of the coupled-line bandpass filter. The color range is scaled to emphasize the resonance on the microstrip lines.

Results and Discussion

Figure 3 shows S_{11} and S_{21} around the bandpass frequency. Excellent out of band rejection is observed. The coupling gaps can further be adjusted for frequency response optimization.

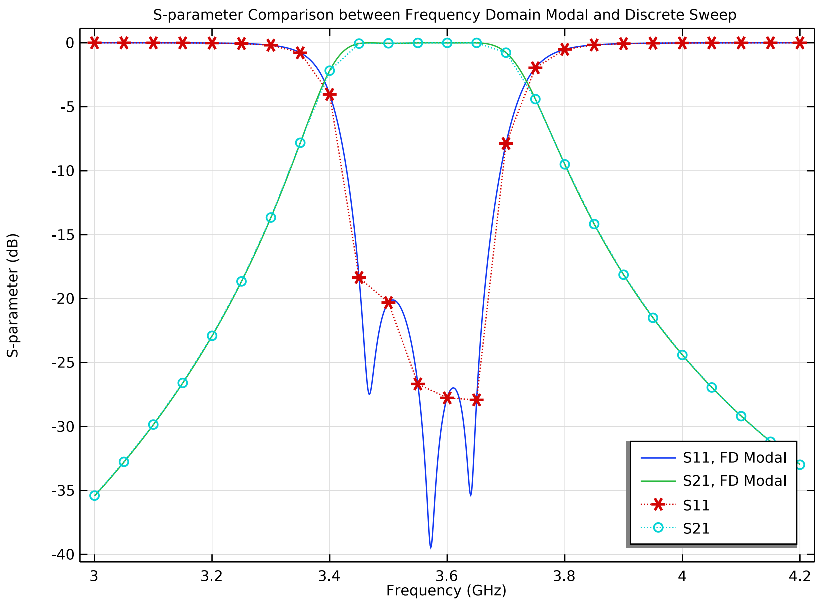


Figure 3: Frequency response of the coupled-line bandpass filter. The bandpass performance of the filter is much better than that of a single-section coupled-line filter.

Reference

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

Application Library path: `RF_Module/Filters/coupled_line_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 **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

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 Click **Range**.

4 In the **Range** dialog box, type `3[GHz]` in the **Start** text field.

5 In the **Step** text field, type `50[MHz]`.

6 In the **Stop** text field, type `4.2[GHz]`.

7 Click **Replace**.

GLOBAL DEFINITIONS

Define some parameters that are useful when setting up the mesh and the study.

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
thickness	20[mil]	5.08E-4 m	Substrate thickness
w_line	1.13[mm]	0.00113 m	Line width
l_line	25[mm]	0.025 m	Line length
gap	0.7[mm]	7E-4 m	Gap between lines

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

GEOMETRY 1

First, create a block for the substrate.

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 l_line/2.

4 In the **Depth** text field, type l_line*2.2.

5 In the **Height** text field, type thickness.

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

Then, add a block for the metal package enclosing the substrate.

Enclosure

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

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

3 Locate the **Size and Shape** section. In the **Width** text field, type l_line/2.

4 In the **Depth** text field, type l_line*2.4.

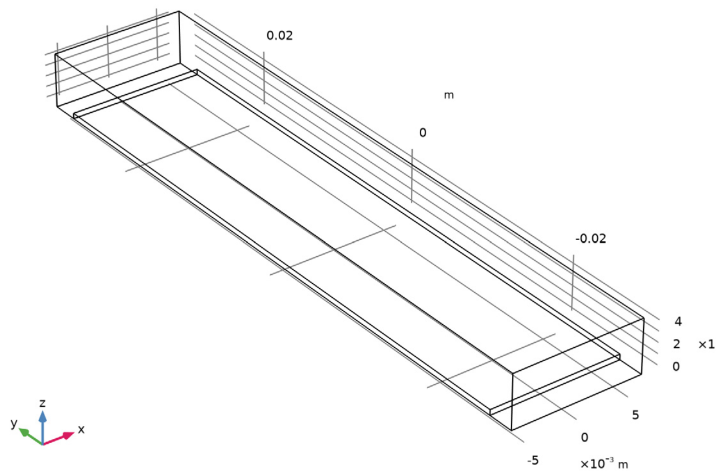
5 In the **Height** text field, type l_line/5.

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

7 In the **z** text field, type l_line/10-thickness/2.

8 Click **Build All Objects**.

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



Add a work plane on the substrate to create the microstrip lines.

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 `thickness/2`.

4 Click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

Create a rectangle for the microstrip line connected to Lumped port 1.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

4 In the **Width** text field, type `w_line`.

5 In the **Height** text field, type `l_line*0.6`.

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

7 In the **xw** text field, type `-w_line*2-gap*1.07`.

8 In the **yw** text field, type `-l_line*0.8`.

Work Plane 1 (wp1)>Plane Geometry

Then, add a rectangle next to the first microstrip line. There is a very thin gap between two microstrip lines.

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

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `w_line`.

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

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

6 In the **xw** text field, type `-w_line-gap`.

7 In the **yw** text field, type `-l_line/2`.

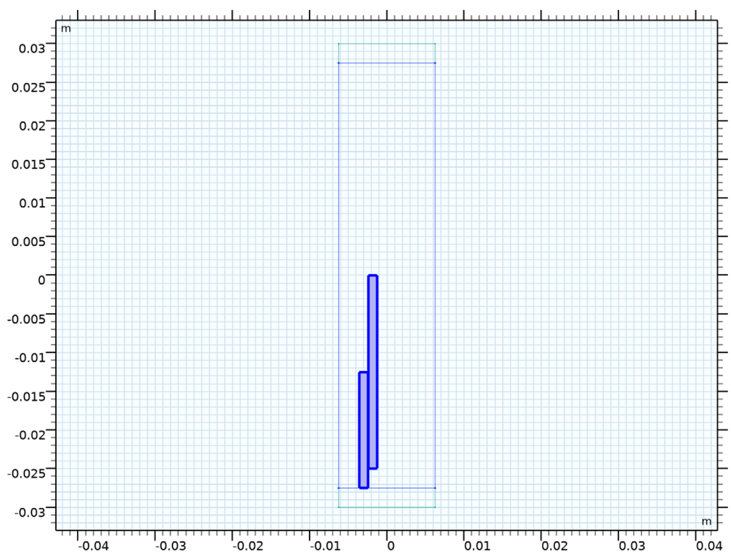
Work Plane 1 (wp1)>Plane Geometry

Generate another pair of microstrip lines with a thin gap at the other side of the substrate by rotating the above two rectangles 180 degrees. Keep the input objects.

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

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.



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

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

5 Locate the **Rotation** section. In the **Angle** text field, type `180`.

Work Plane 1 (wp1)>Plane Geometry

Finish the layout on the substrate by adding the last piece of the microstrip lines on the center of the substrate.

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

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

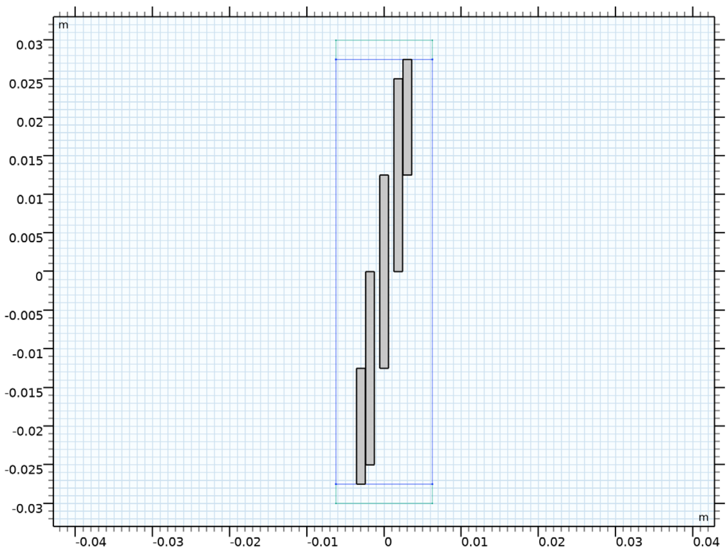
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `w_line`.

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

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

6 Click **Build Selected**.



Add a work plane to define boundaries for the lumped ports.

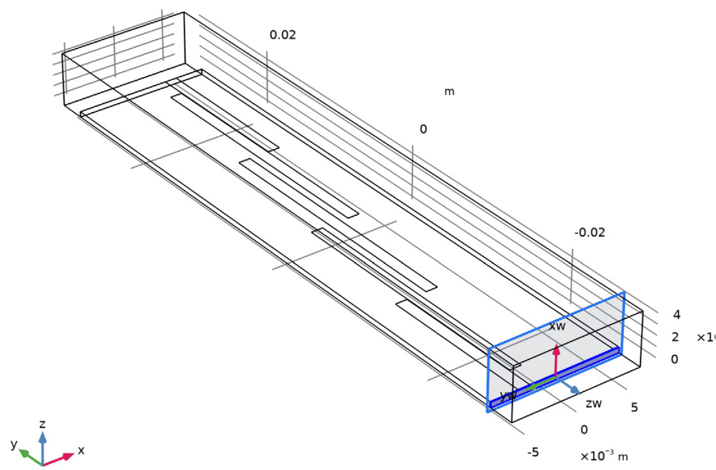
Work Plane 2 (wp2)

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

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

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **blk1**, select Boundary 3 only.



Work Plane 2 (wp2)>Plane Geometry

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

Create a rectangle for the first lumped port.

Work Plane 2 (wp2)>Rectangle 1 (r1)

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

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

3 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

4 In the **Width** text field, type `thickness`.

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

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

7 In the **yw** text field, type `w_line*2+gap*1.07`.

Then, rotate the above rectangle to create the second lumped port.

Rotate 1 (rot1)

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

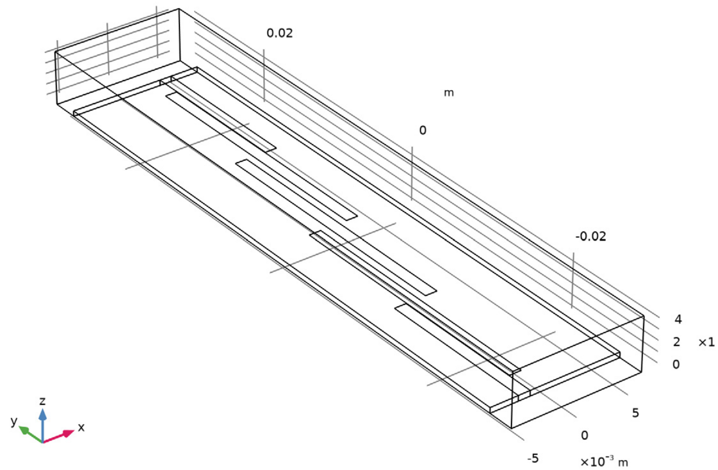
2 Select the object **wp2** only.

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

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

5 Locate the **Rotation** section. In the **Angle** text field, type `180`.

6 Click **Build All Objects**.



DEFINITIONS

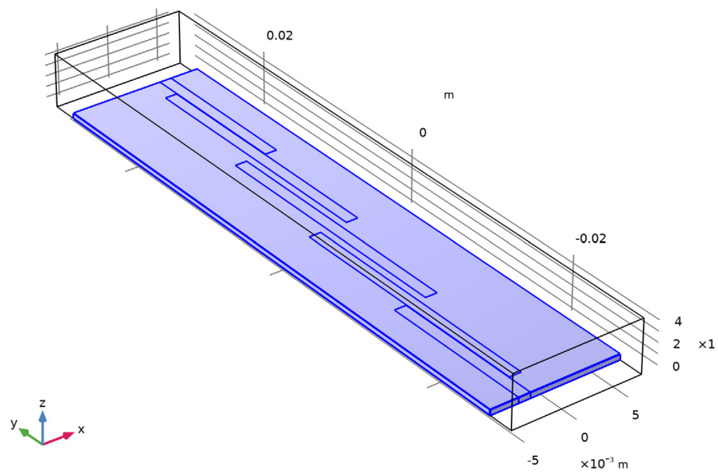
Create a set of selections for use before setting up the physics.

Substrate

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

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

3 Select Domain 2 only.



Add a selection for the microstrip line traces.

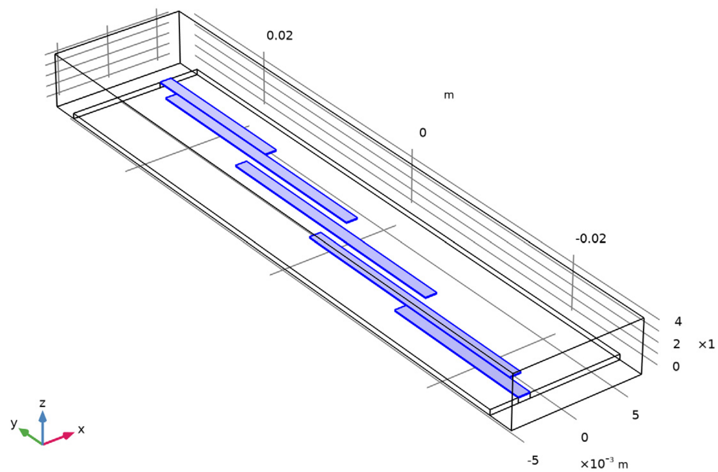
Microstrip line

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

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

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

4 Select Boundaries 13 and 15–18 only.



View 1

Hide three boundaries to get a better view of the interior parts when setting up the physics.

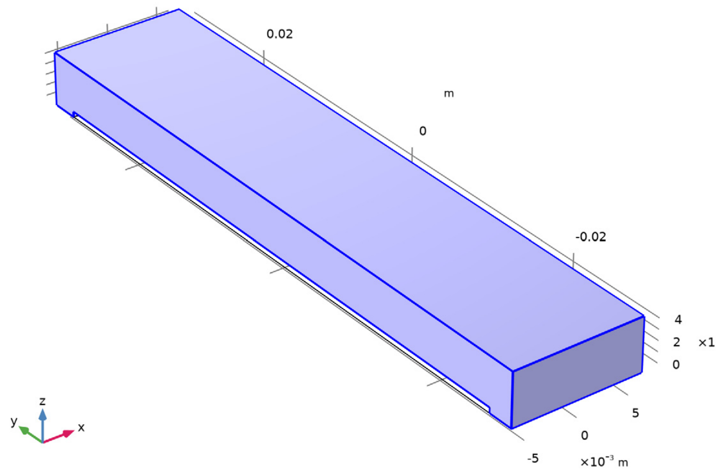
Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 1, 2, and 4 only.



To see the objects in selections, make sure that **Show Objects in Selection** is turned on. The button is located at the upper-left corner in the **Hide Geometric Entities** settings window.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. The default boundary condition is **Perfect Electric Conductor**, which applies to all exterior boundaries. Also, assign a perfect electric conductor condition on the interior boundary of the microstrip lines. Use the selection for the microstrip line boundaries that you defined earlier.

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 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

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

Lumped Port 1

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

2 Select Boundary 12 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 19 only.

MATERIALS

Next, assign material properties on the model. Begin by specifying air for all domains.

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

Override the substrate with a dielectric material of $\varepsilon_r = 3.38$.

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 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Substrate**.

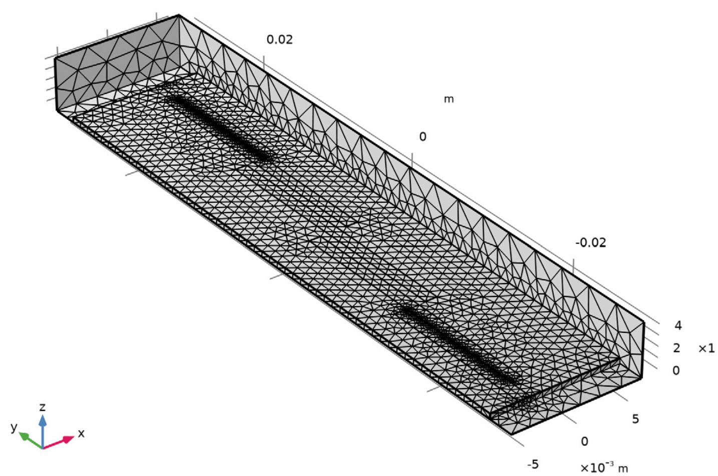
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	3.38	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 distribution of the norm of the electric field. Show that only on the *xy*-plane.

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

2 From the **Parameter value (freq (GHz))** list, choose **3.55**.

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`.

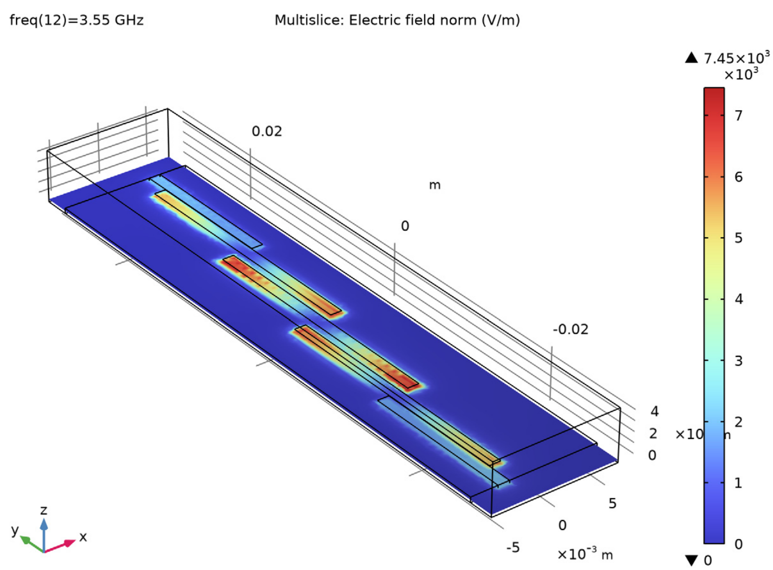
4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `0`.

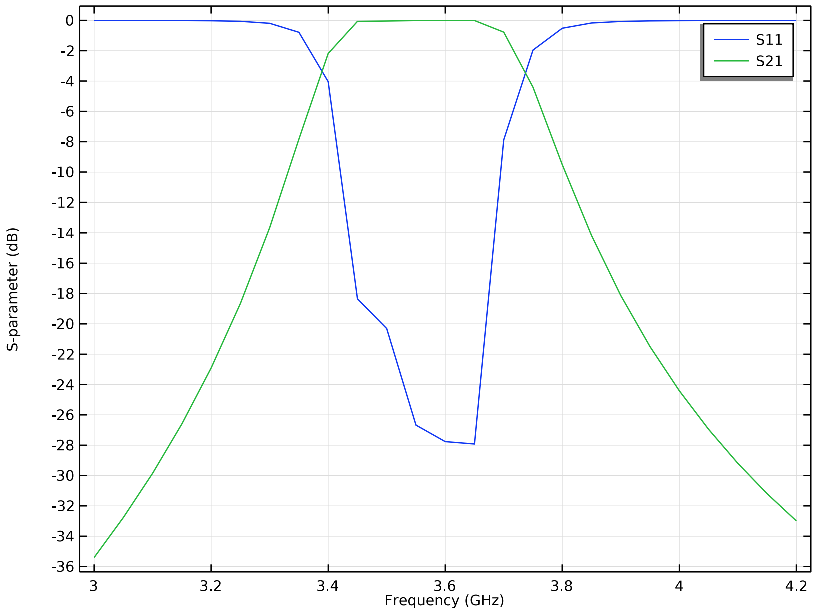
7 In the **Electric Field (emw)** toolbar, click **Plot**.

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

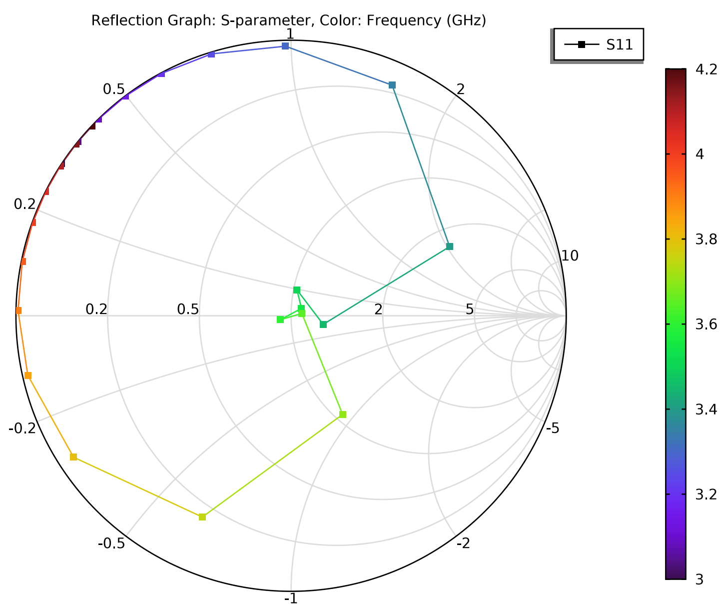


The microstrip line in the middle of the substrate shows the resonance at the selected frequency.

S-parameter (emw)



Smith Plot (emw)



Analyze the same model with a Frequency Domain Modal method. When a device presents resonances, the **Frequency Domain Modal** method combined with an **Eigenfrequency** analysis provides a faster solution time.

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, Create a set of selections for use in the study settings.

5 type `Lumped port 1` in the **Selection name** text field.

6 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>Frequency Domain, Modal**.

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: Eigenfrequency

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

2 In the **Search for eigenfrequencies around** text field, type `3[GHz]`.

Step 2: Frequency Domain, Modal

1 In the **Model Builder** window, click **Step 2: Frequency Domain, Modal**.

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

3 In the **Frequencies** text field, type `range(3[GHz],50[MHz]/50,4.2[GHz])`.

With a 50 times finer frequency step, the solutions will increase the model file size tremendously when it is saved. When only S-parameters are of interest, a common theme in most passive RF and microwave device designs, it is not necessary to store all of the field solutions. By selecting the **Store fields 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 S-parameters are calculated. The lumped port size is typically very small compared to the entire modeling domain, and the saved file size with the finer frequency step is more or less that of the regular discrete frequency sweep model when only the solutions on the port boundaries are stored.

4 Click to expand the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

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**.

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

RESULTS

Electric Field (emw) 1

Since the results are stored only on the lumped port boundaries, this default E-field norm plot does not provide useful information.

1 Right-click **Results>Electric Field (emw) 1** and choose **Delete**.

Generate all S-parameters from each analysis on the same plot and compare them to each other.

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**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `S-parameter Comparison between Frequency Domain Modal and Discrete Sweep`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 2

1 In the **Model Builder** window, expand the **S-parameter (emw) 1** node.

2 Right-click **Results>S-parameter (emw) 1>Global 1** and choose **Duplicate**.

3 In the **Settings** window for **Global**, locate the **Data** section.

4 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**.

Global 1

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

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

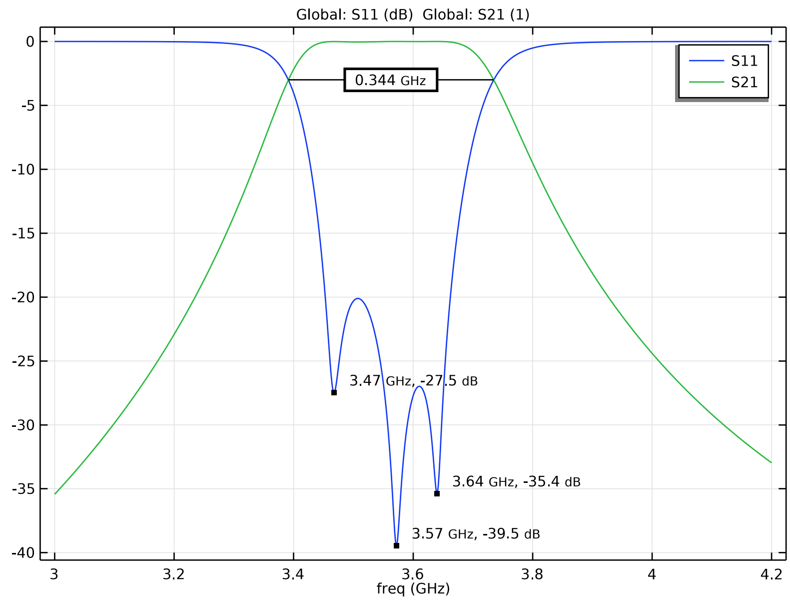
3 In the table, enter the following settings:

Expression	Unit	Description
`emw.S11dB`	1	`S11, FD Modal`
`emw.S21dB`	1	`S21, FD Modal`

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

Compare the resulting plots with that shown in Figure 3.

Smith Plot (emw) 1



Compare the solution time between two studies.

The following instruction shows how to use the **Graph Marker** subfeature to analyze 1D plots. When plotting S11 of a bandpass filter, poles are of interest and a graph marker captures the local minima. For analyzing the insertion loss such as S21, the -3dB bandwidth of the passband can be computed through an additional graph marker.

S-parameter with Graph Markers

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

2 In the **Settings** window for **1D Plot Group**, type `S-parameter with Graph Markers` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Global 1

1 Right-click **S-parameter with Graph Markers** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S11dB - S11**.

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

Expression	Unit	Description
emw.S11dB	dB	S11

Graph Marker 1

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

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

3 From the **Display** list, choose **Min**.

4 From the **Scope** list, choose **Local**.

5 Locate the **Text Format** section. In the **Display precision** text field, type `3`.

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

7 Select the **Include unit** check box.

8 Click to expand the **Coloring and Style** section. From the **Anchor point** list, choose **Lower left**.

Global 2

1 In the **Model Builder** window, right-click **S-parameter with Graph Markers** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S21dB - S21**.

Graph Marker 1

1 Right-click **Global 2** and choose **Graph Marker**.

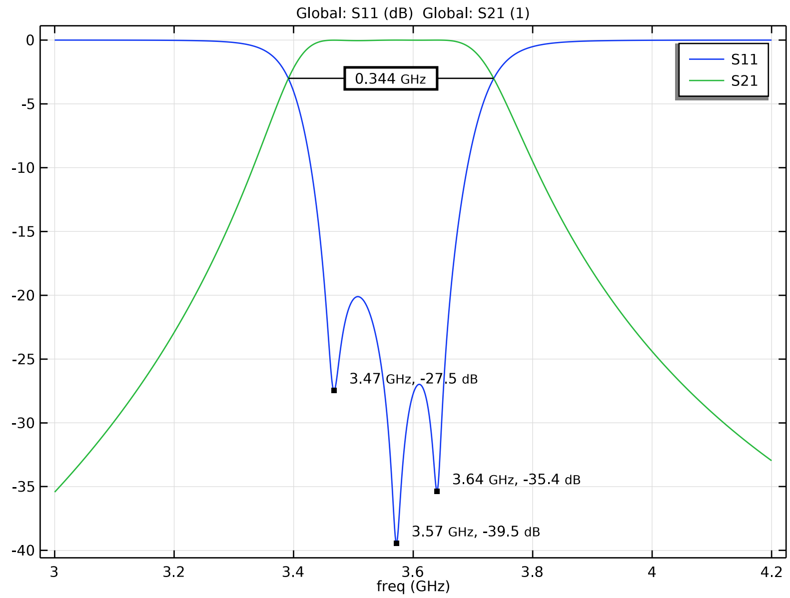
2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Display mode** list, choose **Bandwidth**.

4 Locate the **Text Format** section. In the **Display precision** text field, type `3`.

5 Select the **Include unit** check box.

6 Locate the **Coloring and Style** section. Select the **Show frame** check box.



Touchstone 1

1 In the **Results** toolbar, click **Data** and choose **Touchstone**.

2 In the **Settings** window for **Touchstone**, locate the **Output** section.

3 In the **Filename** text field, type `coupled_line_filter_touchstone.s2p`.

4 From the **Parameter format** list, choose **Magnitude in dB and angle**.

5 Click **Export**.

Warning 1

Review the warning. Since a port sweep is not performed, the solution is incomplete to generate a full S-parameter matrix. The Touchstone export assumes the simulation model is a reciprocal network and creates a symmetric matrix.