Created in COMSOL Multiphysics 5.6

Notch Filter Using a Split Ring Resonator

Introduction

A split ring resonator (SRR) has a band-stop frequency response. In this example, a printed SRR on a dielectric substrate is coupled to a microstrip line. The entire circuit behaves as a notch (band-stop) filter, which can be used to block a specific signal frequency range.

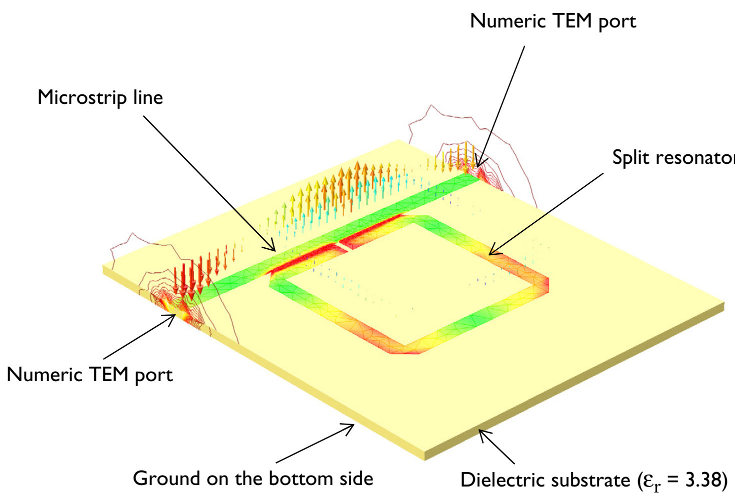


Figure 1: A split ring resonator coupled to a microstrip line.

Model Definition

Using the resonance characteristics of a split ring resonator, either a band-pass or band-stop filter can be realized on a microstrip line type structure. The band-pass or band-stop frequency response depends on the coupling configuration between a microstrip line and a split ring resonator.

In this example, to get a band-stop filter response, the split part of the resonator is adjacent and coupled to the straight microstrip line (Figure 1). On a ground plane, the printed split ring resonator, on a 1.524 mm thick dielectric (ε_r = 3.38) substrate, has multiple resonant modes. Although not included in this example, the resonant modes can be identified using an eigenfrequency analysis. Among those resonant modes, the frequency close to 2.4 GHz is of interest. The split ring resonator's frequency response is studied when it is coupled to the microstrip line.

All metal parts are modeled as perfect electric conductors (PEC). Scattering boundary conditions are assigned on all exterior boundaries of the simulation domain, except for the ground plane. The remaining part is characterized as a vacuum domain.

On the surfaces of each end of the microstrip line, including the air domain, a numeric port is added that calculates the electric mode field on the given structure, with an effective dielectric constant ε_r = sqrt(3.38). This is done through a Boundary Mode analysis. In the numeric port setting, "Analyzed as a TEM field" is selected. To compute the voltage and current of the port, this setting requires defined electric and magnetic field integration lines, respectively. The port characteristic impedance is calculated using the voltage and current extracted from these integration lines. The port mode field is scaled by the ratio of the calculated impedance and the reference impedance, defined in the settings window. The electric fields are guided between two conductors and the field component in the direction of propagation, the normal to the port boundary is negligible. Thus, it is reasonable to analyze the port mode as transverse electromagnetic (TEM) field.

Results and Discussion

The default electric field norm on the *xy*-plane is plotted in Figure 2. The electric fields are confined symmetrically along the split ring resonator at 2.4 GHz. Figure 3 shows the frequency response of the device. Around 2.4 GHz, its S_{11} is almost 0 dB, while its S_{21} is below −10 dB, so it behaves as band-stop (notch) filter.

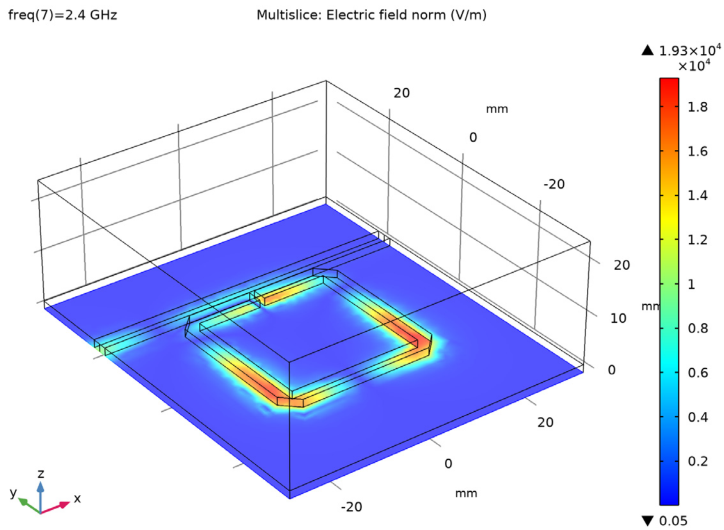


Figure 2: The electric field norm visualized on the xy-plane.

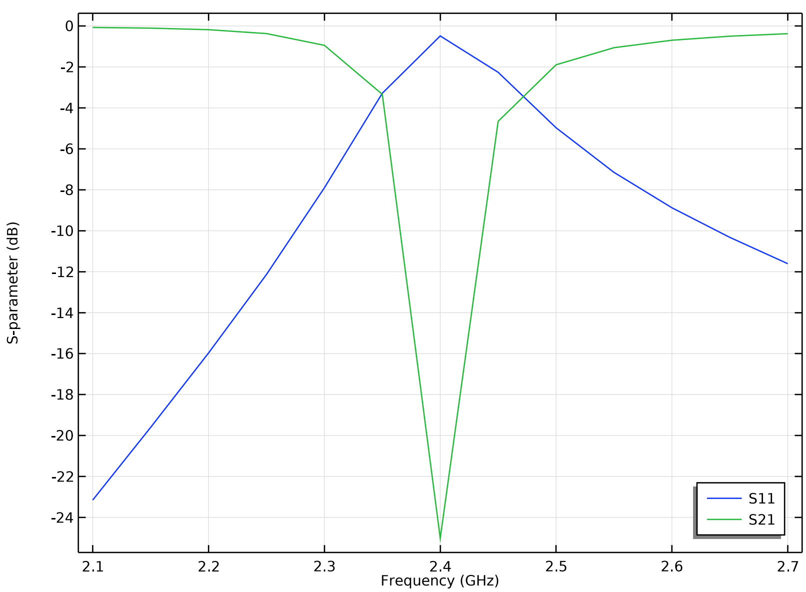


Figure 3: The S-parameter plot showing a band-stop frequency response.

Notes About the COMSOL Implementation

To learn more about how to define integration lines for calculating the voltage and the current of the numeric TEM port, review the following examples in the Application Libraries:

```
RF Module/Verification Models/coaxial_cable_impedance
RF Module/Verification Models/parallel_wires_impedance
```

Application Library path: `RF_Module/Filters/notch_filter_srr`

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 **Done**.

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>Boundary Mode Analysis**.

4 Click **Add Study**.

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

STUDY 1

Step 1: Boundary Mode Analysis

1 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

2 In the **Mode analysis frequency** text field, type `2.4[GHz]`.

3 Select the **Search for modes around** check box.

4 In the associated text field, type `sqrt(3.38)`.

Step 3: Boundary Mode Analysis 1

1 Right-click **Study 1>Step 1: Boundary Mode Analysis** and choose **Duplicate**.

2 Right-click **Step 3: Boundary Mode Analysis 1** and choose **Move Up**.

3 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

4 In the **Port name** text field, type `2`.

Step 3: Frequency Domain

1 In the **Model Builder** window, click **Step 3: Frequency Domain**.

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

3 In the **Frequencies** text field, type `range(2.1[GHz],0.05[GHz],2.7[GHz])`.

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**.

Work Plane 1 (wp1)

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

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 `60`.

4 In the **Height** text field, type `3.2`.

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

6 In the **yw** text field, type `18`.

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 60.

4 In the **Height** text field, type 70.

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

6 Click **Build Selected**.

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

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 32.

4 In the **Height** text field, type 32.

5 Locate the **Position** section. In the **xw** text field, type -16.

6 In the **yw** text field, type -15.7.

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

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

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

3 In the **Width** text field, type 26.

4 In the **Height** text field, type 26.

5 Locate the **Position** section. In the **xw** text field, type -13.

6 In the **yw** text field, type -12.7.

Work Plane 1 (wp1)>Rectangle 5 (r5)

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

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

3 In the **Height** text field, type 6.

4 Locate the **Position** section. In the **xw** text field, type -0.5.

5 In the **yw** text field, type 12.

Work Plane 1 (wp1)>Difference 1 (dif1)

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

2 Select the object **r3** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the objects **r4** and **r5** only.

Work Plane 1 (wp1)>Chamfer 1 (cha1)

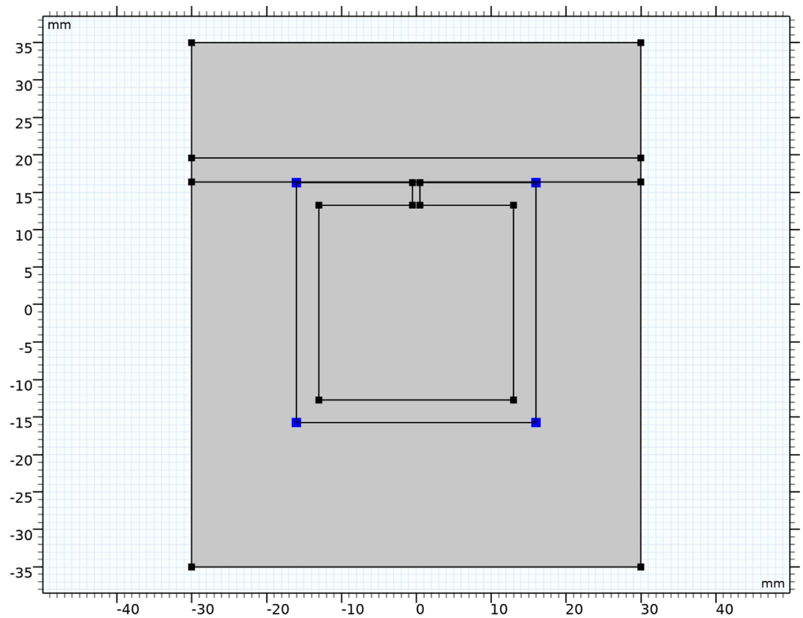
1 In the **Work Plane** toolbar, click **Chamfer**.

2 In the **Settings** window for **Chamfer**, locate the **Distance** section.

3 In the **Distance from vertex** text field, type 3.

4 On the object **dif1**, select Points 1, 2, 11, and 12 only.

It might be easier to select the points by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (mm)
1.524

4 Click **Build Selected**.

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

Block 1 (blk1)

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

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

3 In the **Width** text field, type 60.

4 In the **Depth** text field, type 70.

5 In the **Height** text field, type 25.

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

7 In the **z** text field, type 12.5.

Work Plane 2 (wp2)

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

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

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type -30.

5 Click **Show Work Plane**.

Work Plane 2 (wp2)>Polygon 1 (pol1)

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

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

3 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

4 In the table, enter the following settings:

xw (mm)	yw (mm)
18	0
18	1.524

Mirror 1 (mir1)

1 Right-click **Geometry 1** and choose **Transforms>Mirror**.

2 Select the object **wp2** only.

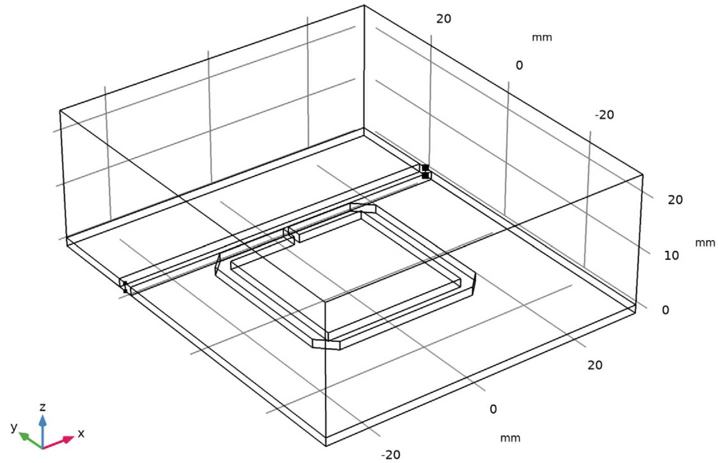
3 In the **Settings** window for **Mirror**, locate the **Input** section.

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

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

6 In the **z** text field, type 0.

7 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

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

2 Select Boundaries 11 and 22 only.

Scattering Boundary Condition 1

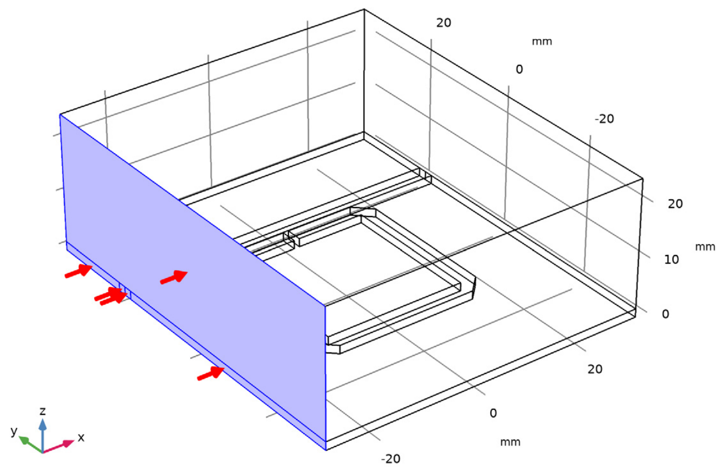
1 In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 Select Boundaries 2, 5, 7, 17, and 18 only.

Port 1

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

2 Select Boundaries 1, 4, 8, 12, and 13 only.



3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Numeric**.

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

5 Select the **Analyze as a TEM field** check box.

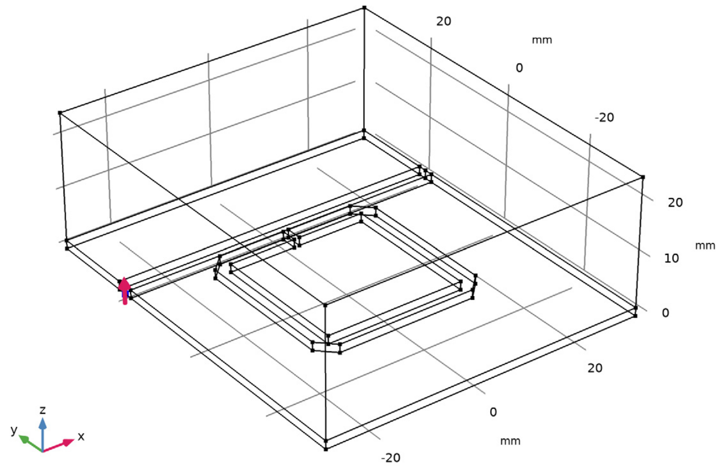
Integration Line for Voltage 1

1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Voltage**.

2 In the **Settings** window for **Integration Line for Voltage**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Select Edge 14 only.



Port 1

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

Integration Line for Current 1

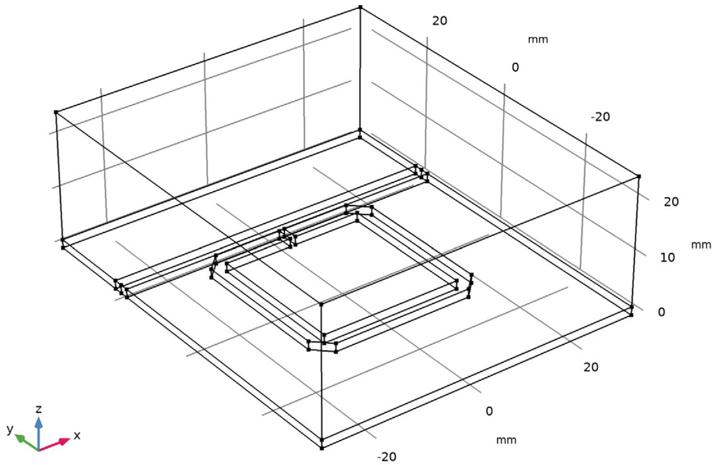
1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Current**.

2 In the **Settings** window for **Integration Line for Current**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1-2, 4, 7, 10, 15, 18, 22, 24` in the **Selection** text field.



6 Click **OK**.

Port 2

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

2 Select Boundaries 37–41 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Numeric**.

5 Select the **Analyze as a TEM field** check box.

Integration Line for Voltage 1

1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Voltage**.

2 In the **Settings** window for **Integration Line for Voltage**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Select Edge 83 only.

Port 2

In the **Model Builder** window, click **Port 2**.

Integration Line for Current 1

1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Current**.

2 In the **Settings** window for **Integration Line for Current**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `75-77, 79, 81, 84, 87, 89-90` in the **Selection** text field.

6 Click **OK**.

MATERIALS

Material 1 (mat1)

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

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

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

Material 2 (mat2)

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1, 3-5` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material**, locate the **Material Contents** section.

7 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

MESH 1

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

2 Right-click **Component 1 (comp1)>Mesh 1** and choose **Build All**.

DEFINITIONS

Hide for Physics 1

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

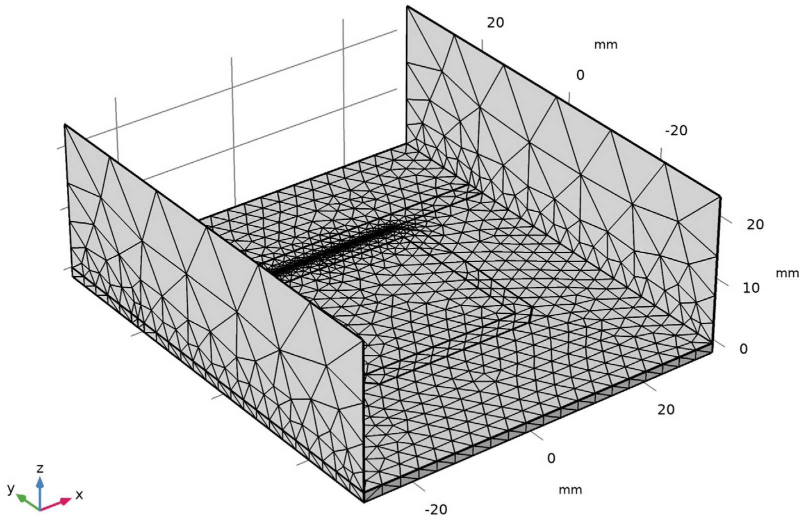
2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 5, 7, and 18 only.

MESH 1

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



STUDY 1

Step 3: Frequency Domain

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

RESULTS

Electric Field (emw)

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

2 From the **Parameter value (freq (GHz))** list, choose **2.4**.

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**.

Reproduce Figure 2.

S-parameter (emw)

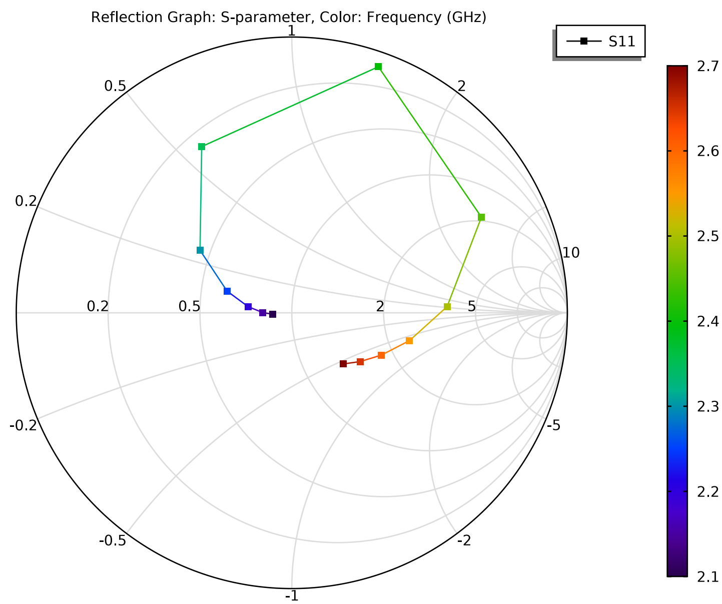
1 In the **Model Builder** window, click **S-parameter (emw)**.

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

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

Compare the reproduced plot with Figure 3.

Smith Plot (emw)



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 25 times faster than regular Frequency Domain sweeps with a same finer frequency resolution.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Port 1

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

Port 2

1. In the **Model Builder** window, click **Port 2**.
2. In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3. Click **Create Selection**.
4. In the **Create Selection** dialog box, type `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 **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Boundary Mode Analysis, Step 2: Boundary Mode Analysis 1

1. In the **Model Builder** window, under **Study 1**, Ctrl-click to select **Step 1: Boundary Mode Analysis** and **Step 2: Boundary Mode Analysis 1**.
2. Right-click and choose **Copy**.

STUDY 2

Step 1: Boundary Mode Analysis

In the **Model Builder** window, right-click **Study 2** and choose **Paste Multiple Items**.

Adaptive Frequency Sweep

1 In the **Study** toolbar, click **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(2.1[GHz],1[MHz],2.7[GHz])`.

   Use a 50 times finer frequency resolution.

4 From the **AWE expression type** list, choose **User controlled**.

5 In the table, enter the following settings:

Asymptotic waveform evaluation (AWE) expressions
`abs(comp1.emw.S11)`

   A slowly varying scalar value curve works well for AWE expressions. For two-port bandstop-type devices, use `abs(comp1.emw.S11)`.

   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 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 port variables are calculated. The port size is relatively 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 port boundaries are stored.

6 Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

7 Under **Selections**, click **Add**.

8 In the **Add** dialog box, in the **Selections** list, choose **Port 1** and **Port 2**.

9 Click **OK**.

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

10 In the **Study** 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

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

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

Selection 1

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

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

3 From the **Selection** list, choose **Port 1**.

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

S-parameter (emw) 1

1 In the **Model Builder** window, 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

Global 2

1 In the **Model Builder** window, right-click **S-parameter (emw) 1** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

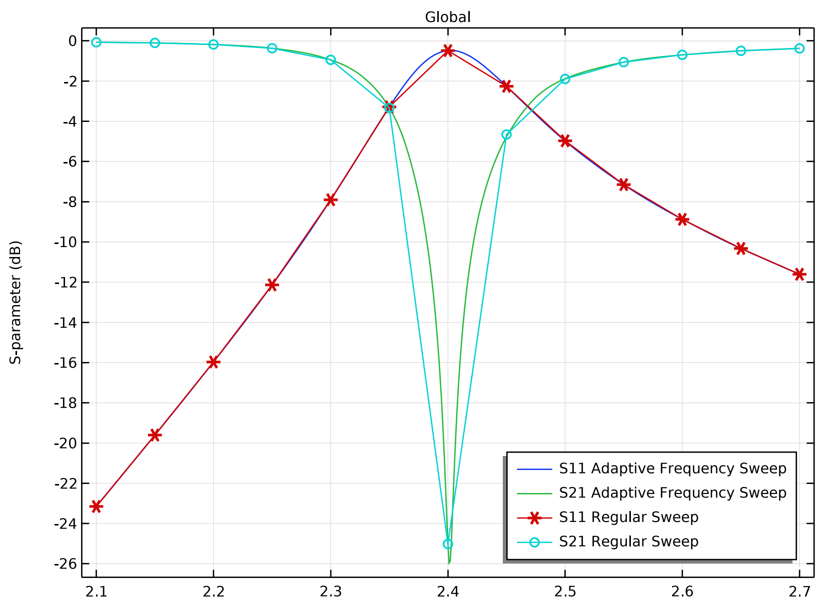
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

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

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

6 From the **Positioning** list, choose **In data points**.

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



Smith Plot (emw) 1