RF Module

Application Library Manual

5.3

COMSOL

RF Module Application Library Manual

Version: COMSOL 5.3

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM021002

Created in COMSOL Multiphysics 5.3

Anechoic Chamber Absorbing Electromagnetic Waves

Introduction

An anechoic chamber is a measurement facility for antenna characterization, electromagnetic interference (EMI), and electromagnetic compatibility (EMC) tests. By absorbing electromagnetic waves inside the chamber and blocking the incoming signals from the outside, it creates a virtually infinite space that has almost zero internal reflection and does not suffer from unwanted external RF noises, so the device-under-test in the chamber can be accurately measured without any interference. This model simulates a biconical antenna, popularly used in EMI and EMC tests, which is located at the center of a small anechoic chamber. The computed far-field radiation pattern and S-parameter (S11) demonstrate that the microwave absorbers reduce reflection from the walls significantly without distorting antenna performance.

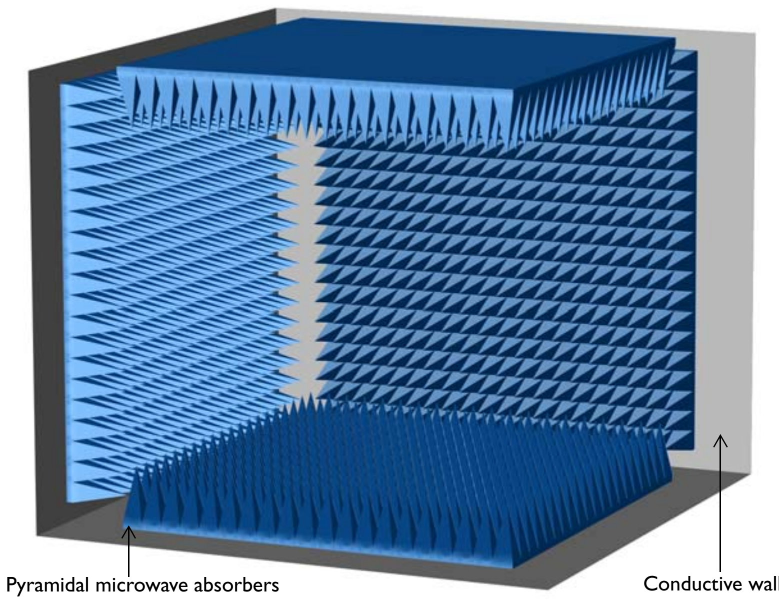


Figure 1: A state-of-the-art anechoic chamber built in a small room (3.9m × 3.9m × 3.3m). It consists of microwave absorbers on thin conductive walls. Two side walls are not included in this figure.

Model Definition

The shape of the absorbers is configured with the array of pyramidal objects to steer the propagation direction of the incident field on the absorbers reflected back not to the radiation source, but toward the surface of the adjacent absorbers. The radiation-absorbent material (RAM), conductive carbon-loaded foam in the pyramidal shaped absorber, is modeled using a low conductive material ($\sigma = 0.5$ S/m). So the electromagnetic waves illuminated on the absorber has the process of partial reflection and partial transmission with subsequent attenuation that is repeated until the wave reaches the base of the pyramid. The amplitude of the field at the base of the pyramid is drastically reduced. Thus, the reflection from the absorbers at this point is marginal.

The exterior of the chamber is finished with a perfect electric conductor (PEC) to model metallic surfaces that insulate the chamber from the outside RF noises.

The imported biconical antenna geometry is identical to the one used in another application library example, Modeling a Biconical Antenna for EMI/EMC Testing (Ref. 1). This reference model is simulating the same antenna geometry but the antenna is enclosed by a numerical version of an anechoic chamber that is a perfectly matched layer (PML).

The metallic surfaces of the antenna are also configured by PEC. A lumped port with a 50 Ω reference impedance is assigned to the gap located at the center of the two structures composed of hexagonal frames. All domains except for the absorbers is filled with the air.

The simulation frequency is set to 240 MHz.

Results and Discussion

The far-field polar plot as a function of azimuth angle is visualized in Figure 2. The plotting plane is perpendicular to the dominant polarization of the antenna so it is the H-plane radiation pattern. Just like the radiation pattern of the biconical antenna surrounded by the PML in Ref. 1, it is isotropic since the reflection from the chamber walls ,that are made of the lossy conductive pyramidal form array, is negligible. The computed S-parameter (S_{11}) is around -10 dB that is very close to the value evaluated at 240 MHz in Ref. 1, which also indicates that the reflection from the chamber walls is marginal.

Figure 3 shows one way to enhance the quality of the results postprocessing by utilizing solution set selections and uniform custom colors. The contour of the norm of electric fields in a dB-scale is plotted in a realistic view of an anechoic chamber. The exterior metallic walls are visualized with the norm of electric fields using the GrayScale color table.

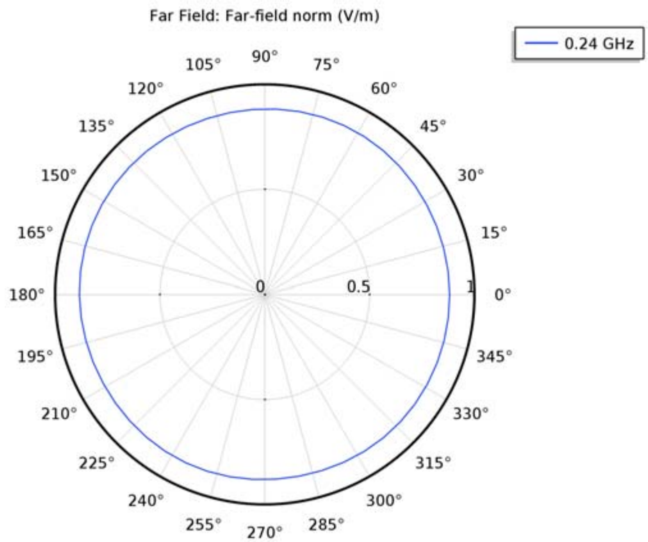


Figure 2: The far-field radiation pattern on the H-plane of the biconical antenna at 240 MHz. It is isotropic as expected.

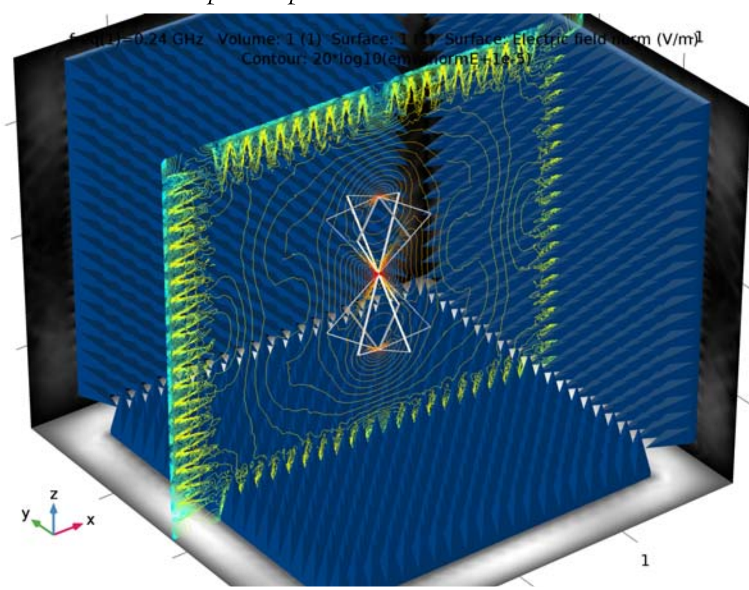


Figure 3: The contour plot of the norm of the electric field (dB-scaled). The strength of the field is gradually decaying inside absorbers.

References

1. *Modeling a Biconical Antenna for EMI/EMC Testing*, COMSOL Application Libraries.

Notes About the COMSOL Implementation

The example model is memory intensive and may require more than 20 GB RAM. The goal of this model is not to simulate an antenna but to design a state-of-the-art anechoic chamber and validate it based on the performance of the antenna. It is recommended to use a PML instead of absorber models to simulate antennas efficiently. The same biconical antenna with the PML (Ref. 1) may need less than 3 GB memory. Note that the anechoic model is not designed in full compliance with well-known standards such as CISPR and ANSI.

Application Library path: `RF_Module/EMI_EMC_Applications/anechoic_chamber`

Model 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 **Preset 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 In the **Frequencies** text field, type `240[MHz]`.

GEOMETRY 1

Block 1 (blk1)

1 On 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 `3.9`.

4 In the **Depth** text field, type `3.9`.

5 In the **Height** text field, type `3.3`.

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

7 Click **Build Selected**.

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

Pyramid 1 (pyr1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Pyramid**.

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

3 In the **Base length 1** text field, type `0.15`.

4 In the **Base length 2** text field, type `0.15`.

5 In the **Height** text field, type `0.4`.

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

7 Locate the **Position** section. In the **x** text field, type `-1.425`.

8 In the **y** text field, type `-1.425`.

9 In the **z** text field, type `-1.6`.

Block 2 (blk2)

1 On 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 `0.15`.

4 In the **Depth** text field, type `0.15`.

5 In the **Height** text field, type `0.05`.

6 Locate the **Position** section. In the **x** text field, type `-1.5`.

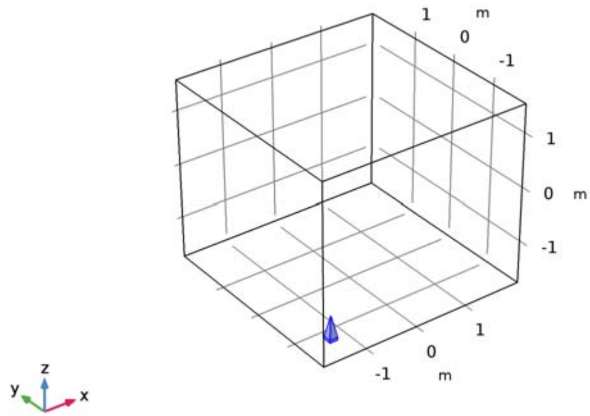
7 In the **y** text field, type `-1.5`.

8 In the **z** text field, type `-1.65`.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **blk2** and **pyr1** only.



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

4 Clear the **Keep interior boundaries** check box.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `20`.

5 In the **y size** text field, type `20`.

6 Locate the **Displacement** section. In the **x** text field, type `0.15`.

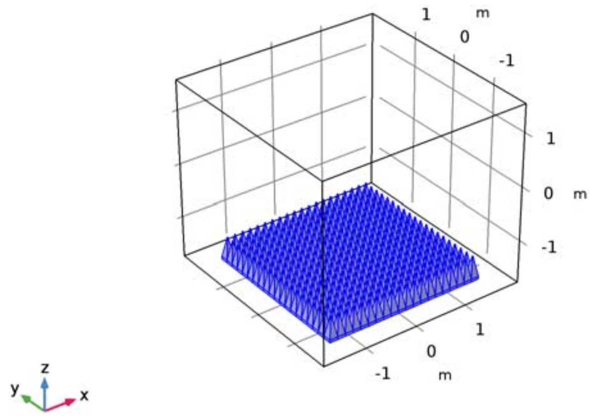
7 In the **y** text field, type `0.15`.

Union 2 (uni2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click the **Select Box** button on the **Graphics** toolbar.

Select all objects in the array as shown in the below figure.



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

4 Clear the **Keep interior boundaries** check box.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **uni2** only.

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

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

Rotate 1 (rot1)

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

2 Select the object **uni2** only.

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

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

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.

6 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `-0.15`.

7 In the **z** text field, type `0.15`.

8 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **y-axis**.

Rotate 2 (rot2)

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

2 Select the object **rot1** only.

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

4 In the **Rotation** text field, type `0 90 180 270`.

Union 3 (uni3)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 Select the objects **rot2(1)**, **rot2(2)**, **mir1**, **uni2**, **rot2(3)**, and **rot2(4)** only.

4 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

5 Click **New**.

Create a set of absorber selections that will make easier to set up the physics and material.

6 In the **New Cumulative Selection** dialog box, type `Absorbers` in the **Name** text field.

7 Click **OK**.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Selections of Resulting Entities** section.

3 Click **New**.

Create a set of far-field selections.

4 In the **New Cumulative Selection** dialog box, type `Far-field` in the **Name** text field.

5 Click **OK**.

Import 1 (imp1)

1 On the **Geometry** toolbar, click **Import**.

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

3 In the **Filename** text field, type `anechoic_chamber_antenna.mphbin`.

4 Locate the **Selections of Resulting Entities** section. Click **New**.

Create a set of antenna geometry selections.

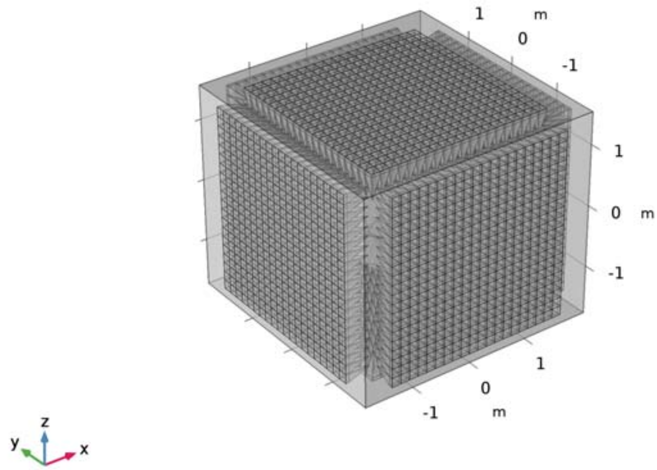
5 In the **New Cumulative Selection** dialog box, type `Antenna` in the **Name** text field.

6 Click **OK**.

7 On the **Geometry** toolbar, click **Build All**.

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

9 Click the **Transparency** button on the **Graphics** toolbar.



Adjust the graphics window settings as you prefer for the remaining modeling steps.

10 Click the **Transparency** button on the **Graphics** toolbar.

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

12 Click the **Zoom In** button on the **Graphics** toolbar.

MATERIALS

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

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

MATERIALS

Air (mat1)

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

Material 2 (mat2)

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 `Absorbers` in the **Label** text field.

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

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

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0.5	S/m	Basic

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

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

4 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

Add a lumped port at the center of the antenna. Zoom in a few of times to get a clear view.

Lumped Port 1

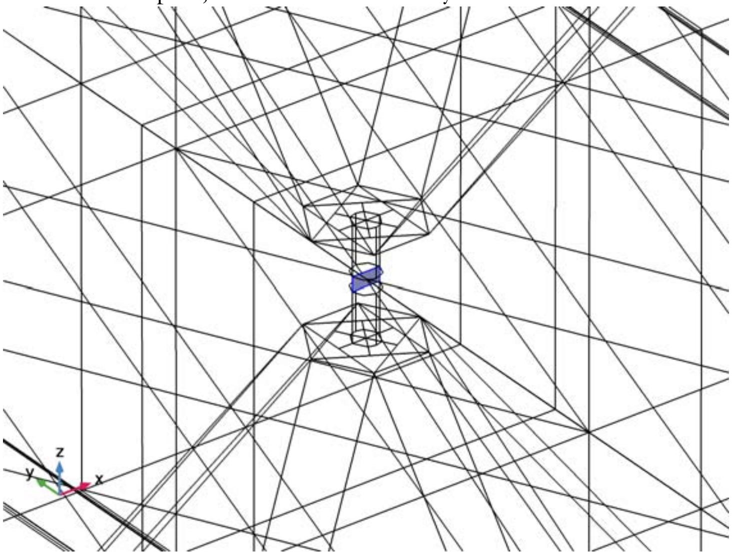
1 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.

2 Click **Paste Selection**.

3 In the **Paste Selection** dialog box, type `6310` in the **Selection** text field.

4 Click **OK**.

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



Far-Field Domain 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

2 In the **Settings** window for **Far-Field Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Far-field**.

Far-Field Calculation 1

1 In the **Model Builder** window, expand the **Far-Field Domain 1** node, then click **Far-Field Calculation 1**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Far-field**.

STUDY 1

Step 1: Frequency Domain

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

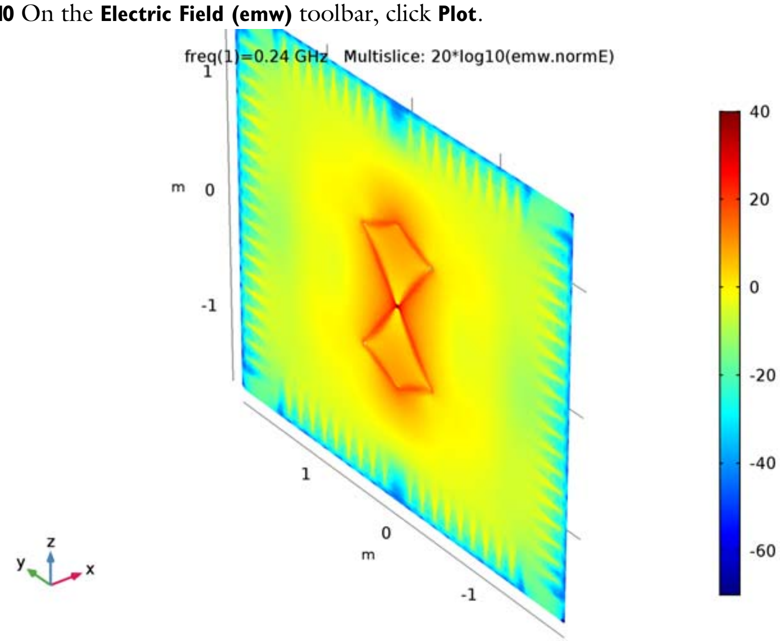
RESULTS

Electric Field (emw)

1. In the **Model Builder** window, under **Results** click **Electric Field (emw)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
3. Clear the **Plot data set edges** check box.

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 **Expression** section.
3. In the **Expression** text field, type `20*log10(emw.normE)`.
4. Locate the **Multiplane Data** section. Find the **X-planes** subsection. In the **Planes** text field, type `1`.
5. Find the **Y-planes** subsection. In the **Planes** text field, type `0`.
6. Find the **Z-planes** subsection. In the **Planes** text field, type `0`.
7. Click to expand the **Range** section. Select the **Manual color range** check box.
8. In the **Minimum** text field, type `-70`.
9. In the **Maximum** text field, type `40`.
10. On the **Electric Field (emw)** toolbar, click **Plot**.



2D Far Field (emw)

1 In the **Model Builder** window, under **Results** click **2D Far Field (emw)**.

2 In the **Settings** window for **Polar Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

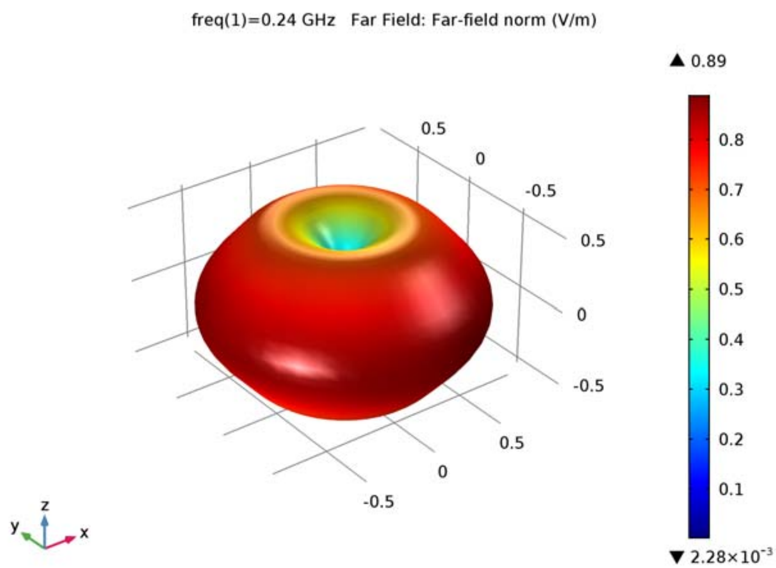
4 In the **r minimum** text field, type 0.

5 In the **r maximum** text field, type 1.

6 On the **2D Far Field (emw)** toolbar, click **Plot**.

Compare the reproduced plot to Figure 2.

3D Far Field (emw)



3D Plot Group 4

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

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

3 Clear the **Show legends** check box.

4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

Volume 1

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

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

3 In the **Expression** text field, type `1`.
4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5 From the **Color** list, choose **Custom**.
6 On Windows, click the colored bar underneath, or—if you are running the cross-platform desktop—the **Color** button.
7 Click **Define custom colors**.
8 Set the RGB values to 0, 64, and 128, respectively.
9 Click **Add to custom colors**.
10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 Right-click **Results>3D Plot Group 4>Volume 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `4, 6, 10` in the **Selection** text field.
5 Click **OK**.

Surface 1

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 4** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `1`.
4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5 From the **Color** list, choose **White**.

Selection 1

1 Right-click **Results>3D Plot Group 4>Surface 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Antenna**.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 4** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 From the **Color table** list, choose **GrayScale**.

Selection 1

1 Right-click **Results>3D Plot Group 4>Surface 2** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `3, 485, 12215` in the **Selection** text field.
5 Click **OK**.

Cut Plane 1

1 On the **Results** toolbar, click **Cut Plane**.
2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
3 From the **Plane** list, choose **ZX-planes**.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 4** and choose **Contour**.
2 In the **Settings** window for **Contour**, locate the **Data** section.
3 From the **Data set** list, choose **Cut Plane 1**.
4 Locate the **Expression** section. In the **Expression** text field, type `20*log10(emw.normE+1e-5)`.

   Adding 1e-5 to the log expression improves the color variation in the contour plot.
5 Locate the **Levels** section. In the **Total levels** text field, type `100`.
6 On the **3D Plot Group 4** toolbar, click **Plot**.

   See Figure 3 to compare the reproduced plot.

S-parameter, S11dB (emw)

1 In the **Model Builder** window, expand the **Derived Values** node, then click **S-parameter, S11dB (emw)**.
2 In the **Settings** window for **Global Evaluation**, click **Evaluate**.

   The computed S_{11} should be around -10 dB.

Created in COMSOL Multiphysics 5.3

Branch-Line Coupler

Introduction

A branch line coupler, also known as a quadrature (90°) hybrid, is a four-port network device with one input port, two output ports, with a 90° phase difference between them, and one isolated port. Due to its symmetry, any port can be used as the input port.

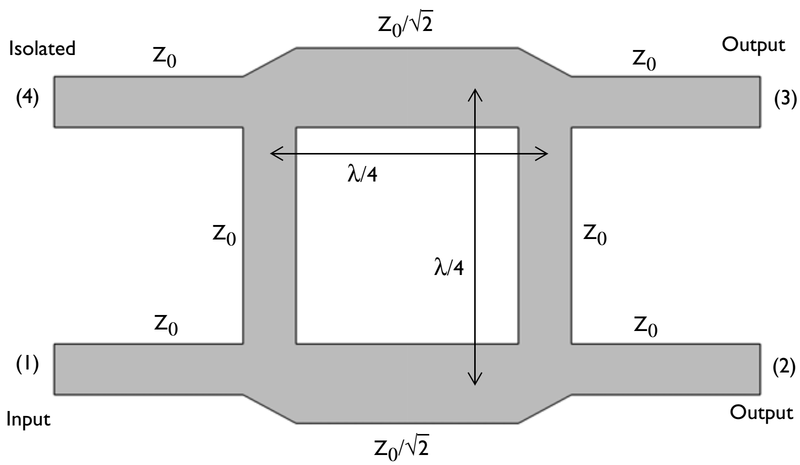


Figure 1: The geometry of a branch line coupler is symmetric.

Model Definition

The form of the branch line coupler is shown schematically in Figure 1. The layout design is based upon Ref. 1, and is tuned to operate at 3 GHz. The design is realized as microstrip lines patterned onto a 0.060 inch dielectric substrate. The microstrip lines are modeled as perfect electric conductor (PEC) surfaces, and another PEC surface on the bottom of the dielectric substrate acts as a ground plane. The entire modeling domain is bounded by PEC boundaries that represent the device packaging. The four ports are modeled as small rectangular faces that bridge the gap between the PEC face that represents the ground plane, and the PEC faces that represent the microstrip line at each port.

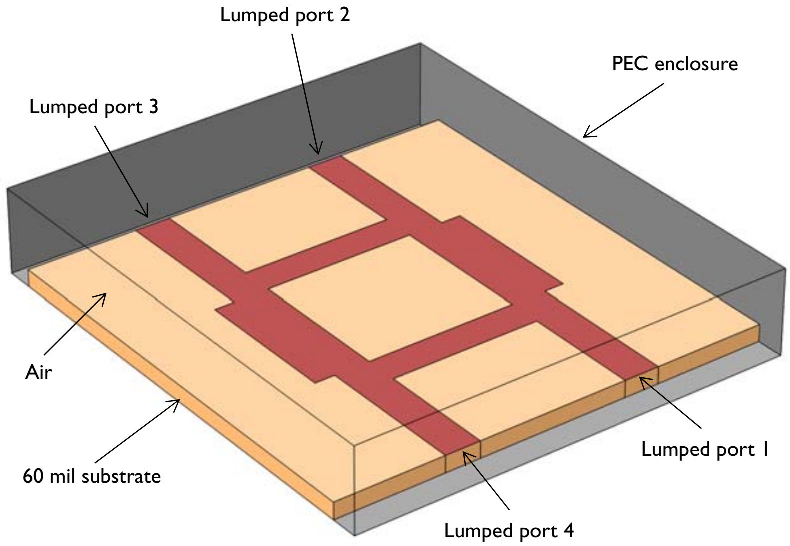


Figure 2: The model of the branch line coupler. Some exterior faces are removed for visualization.

The model is shown in Figure 2. A small air domain bounded by a PEC surface around the device is also modeled. The model is meshed using a tetrahedral mesh. A good rule of thumb is to use approximately five elements per wavelength in each material.

Results and Discussion

The computed S-parameters are plotted in Figure 3. At a frequency of 3 GHz, the signal is evenly split between the two output ports with a very small amount of losses. The input signal is barely coupled to the isolation port where S_{41} is less than –30 dB at 3 GHz. The evaluated phase shift between the two output ports is 89.9°.

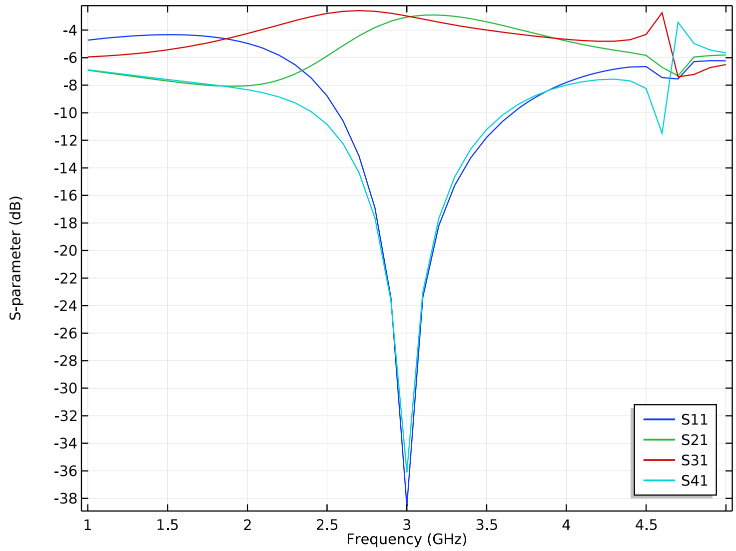


Figure 3: The frequency response of the branch line coupler shows good input matching (S_{11}) and isolation (S_{41}) around 3 GHz. The coupled signal at the two output ports (S_{21} and S_{31}) is about -3 dB at 3 GHz.

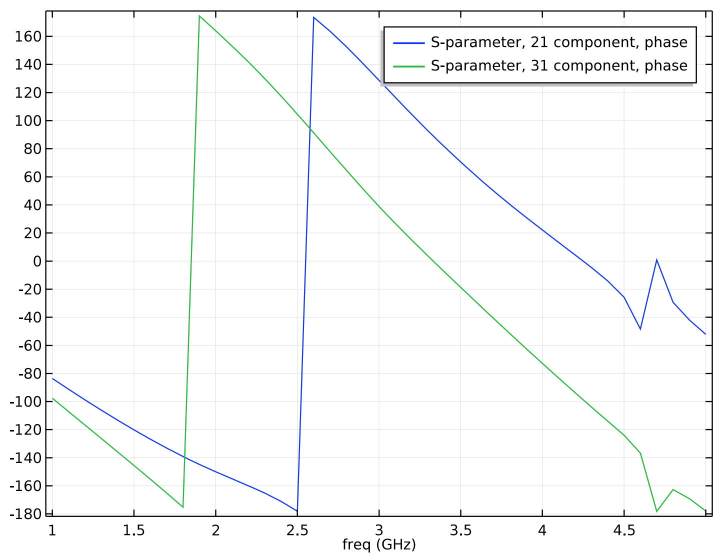


Figure 4: The phases on the two output ports show approximately 90-degree shift at 3 GHz.

Because the metallic housing works as a rectangular cavity, there is a resonance observed around 4.6 GHz. This is the dominant TE_{101} mode of the rectangular cavity resonator partially filled with a dielectric substrate. The resonance can easily be removed in the current frequency sweep range by adding a metallic post in the middle of the cavity.

Reference

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

Application Library path: `RF_Module/Couplers_and_Power_Dividers/branch_line_coupler`

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 **Preset 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. In the **Frequencies** text field, type `range(1[GHz],100[MHz],5[GHz])`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
thickness	60[mil]	0.001524 m	Substrate thickness
l_s	40[mm]	0.04 m	Length, substrate
w_line2	5[mm]	0.005 m	Width, line 2
l_line2	13[mm]	0.013 m	Length, line 2
l_line1	(l_s-l_line2)/2	0.0135 m	Length, line 1
w_line1	3.2[mm]	0.0032 m	Width, line 1
w_line3	3[mm]	0.003 m	Width, line 3
l_line3	13.6[mm]	0.0136 m	Length, line 3

Here, `mil` refers to the unit milliinch.

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)

1 On the **Geometry** toolbar, click **Work Plane**, to add an *xy*-plane for the coupler layout'.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `2*w_line1+l_line3`.

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

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

6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `w_line2*2+l_line3`.
4 In the **Height** text field, type `l_line2`.
5 Locate the **Position** section. From the **Base** list, choose **Center**.
6 On the **Work Plane** toolbar, click **Build All**.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `l_line3`.
4 In the **Height** text field, type `l_line2`.
5 Locate the **Position** section. In the **xw** text field, type `-l_line3/2`.
6 In the **yw** text field, type `l_line2/2+w_line3`.
7 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.
8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Array 1 (arr1)

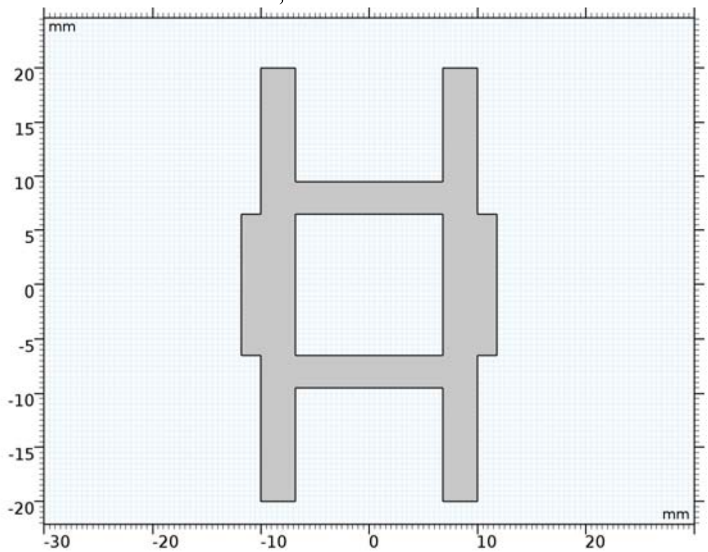
1 On the **Work Plane** toolbar, click **Transforms** and choose **Array**.
2 Select the object **r3** only.
3 In the **Settings** window for **Array**, locate the **Size** section.
4 From the **Array type** list, choose **Linear**.
5 In the **Size** text field, type `3`.
6 Locate the **Displacement** section. In the **yw** text field, type `-l_line2-w_line3`.
7 On the **Work Plane** toolbar, click **Build All**.

Difference 1 (dif1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the objects **r2** and **r1** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5 Select the objects **arr1(1)**, **arr1(2)**, and **arr1(3)** only, the three rectangles belonging to the array object (`arr1`).

6 Clear the **Keep interior boundaries** check box.

7 On the **Work Plane** toolbar, click **Build All**.



Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude the *xy*-plane with the thickness of the substrate. Additional rectangular boundaries at each end of the feed lines are created by this extrusion, too. Use these boundaries to assign lumped ports later.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (mm)
thickness

4 Click **Build All Objects**.

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

Choose wireframe rendering to get a better view of the interior parts.

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

Create a block for the substrate.

Block 1 (blk1)

1. On 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 `l_s`.
4. In the **Depth** text field, type `l_s`.
5. In the **Height** text field, type `thickness`.
6. Locate the **Position** section. From the **Base** list, choose **Center**.
7. In the **z** text field, type `thickness/2`.
8. Click **Build All Objects**.

Union 1 (uni1)

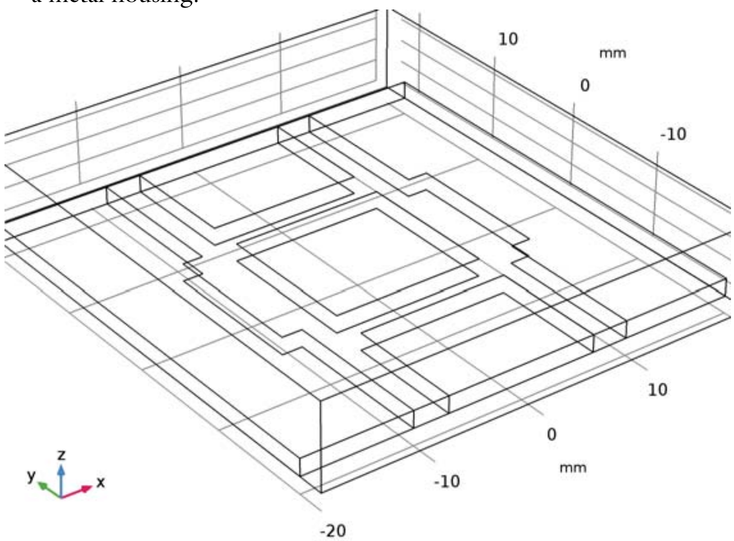
1. On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2. In the **Settings** window for **Union**, type `Substrate` in the **Label** text field.
3. Locate the **Union** section. Clear the **Keep interior boundaries** check box.
4. Click in the **Graphics** window and then press Ctrl+A to select both objects.
5. Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
6. Click **Build All Objects**.

Block 2 (blk2)

1. On the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for **Block**, type `Package` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Width** text field, type `l_s`.
4. In the **Depth** text field, type `l_s+l_s/8`.
5. In the **Height** text field, type `thickness*5`.
6. Locate the **Position** section. From the **Base** list, choose **Center**.
7. In the **z** text field, type `thickness*5/2`.

8 Click **Build All Objects**.

The completed geometry describes the microstrip line device on a substrate enclosed by a metal housing.



DEFINITIONS

Create a selection for the microstrip lines.

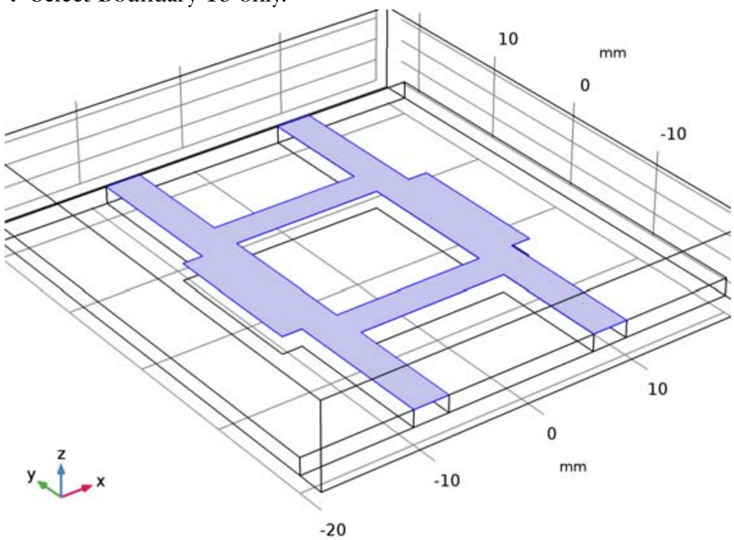
Explicit 1

1 On 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 Boundary 13 only.



View 1

Hide three boundaries to get a better view of the interior parts when reviewing the mesh.

Hide for Physics 1

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. The default boundary condition is perfect electric conductor, which is applied to all exterior boundaries. Apply this condition also to the interior boundaries of the microstrip lines.

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 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 **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 24 only.

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

Lumped Port 2

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

2 Select Boundary 25 only.

Lumped Port 3

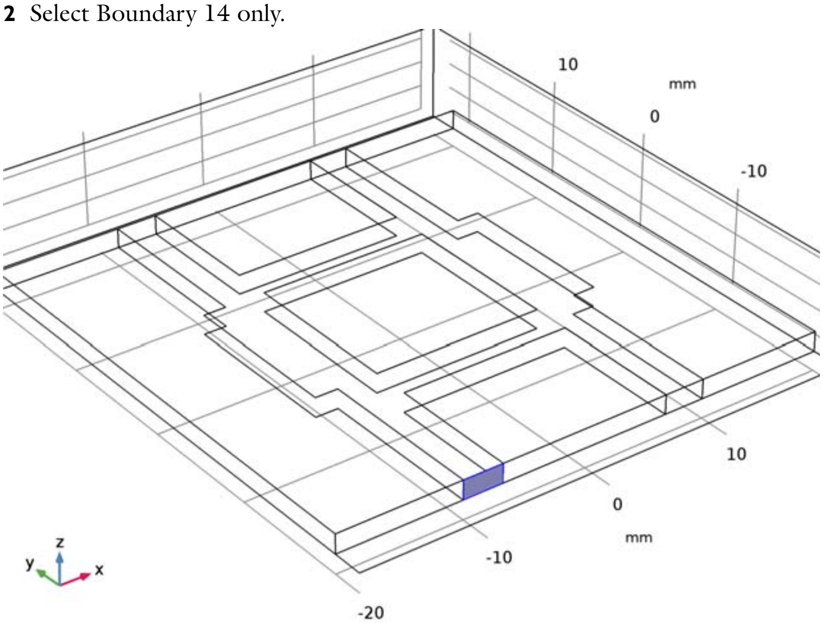
1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 15 only.

Lumped Port 4

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

2 Select Boundary 14 only.



Lumped ports are assigned at each end of the microstrip lines. Wave excitation is on only at the first port.

MATERIALS

Assign material properties to the model. First, apply air to all domains.

ADD MATERIAL

1 On 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.

MATERIALS

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Create a dielectric material of er = 3.38 overriding air in the substrate.

Material 2 (mat2)

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**.

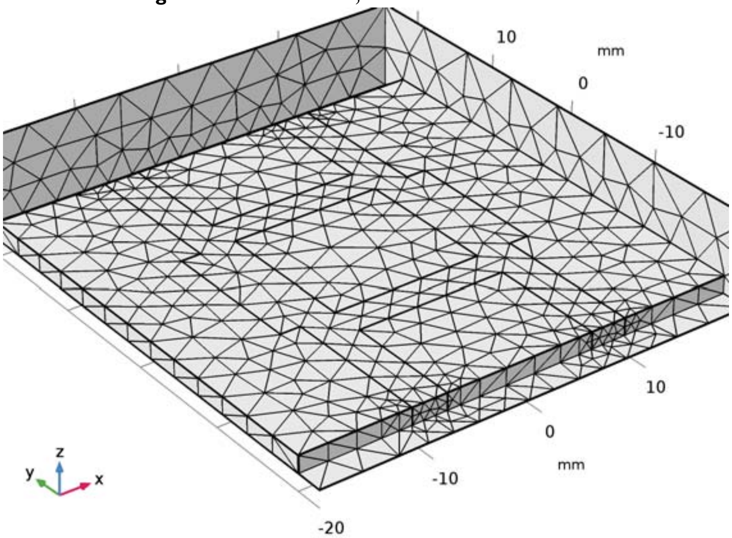
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

MESH 1

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

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



Three exterior boundaries are hidden in this view.

STUDY 1

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

RESULTS

Electric Field (emw)

Begin the results analysis and visualization by modifying the first default plot to show the E-field norm in the middle of the substrate at 3 GHz.

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

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

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

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.

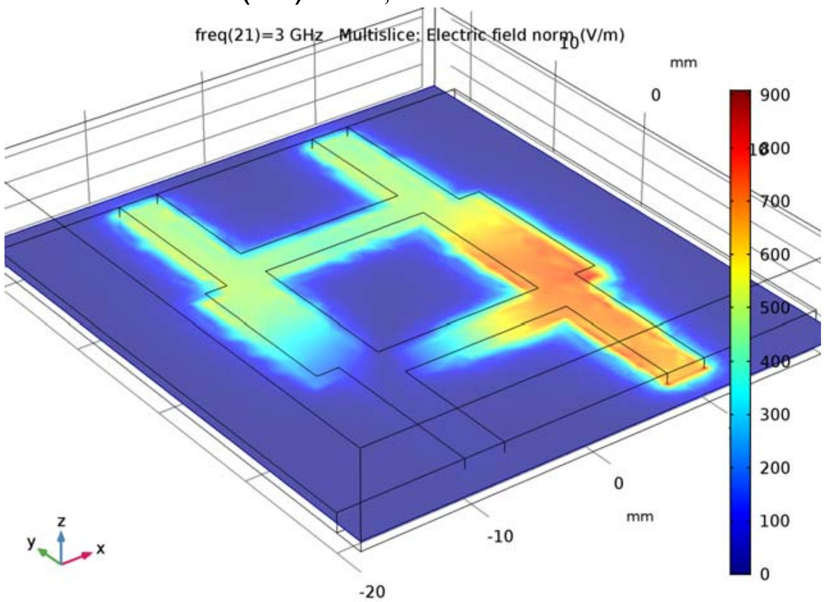
3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

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 `thickness/2`.

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



The input power is evenly split between the two output ports.

S-Parameter (emw)

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

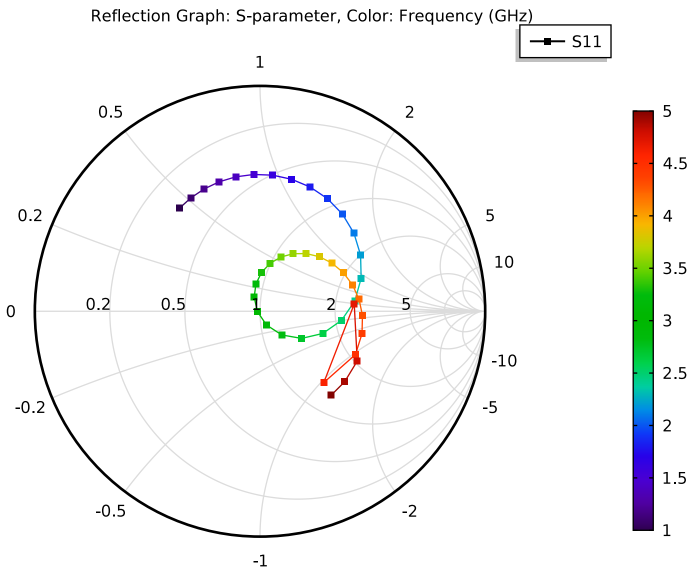
2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

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

Compare the resulting plot with that shown in Figure 3.

Smith Plot (emw)



Plot the phases on two output ports (Figure 4).

1D Plot Group 4

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

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

3 From the **Title type** list, choose **None**.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`arg(emw.S21)`	`deg`	`S-parameter, 21 component, phase`
`arg(emw.S31)`	`deg`	`S-parameter, 31 component, phase`

The unit is degree.

4 On the **1D Plot Group 4** toolbar, click **Plot**.

The phase difference between two output ports is approximately 90 degrees at 3 GHz.

Evaluate the phase difference between two output ports at 3 GHz.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

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

3 From the **Parameter selection (freq)** list, choose **From list**.

4 In the **Parameter values (freq (GHz))** list, select **3**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`arg(emw.S21)-arg(emw.S31)`	°	

6 Click **Evaluate**.

Created in COMSOL Multiphysics 5.3

Thermal Drift in a Microwave Cavity Filter

Introduction

Microwave filters serve to suppress unwanted frequencies in the output of microwave transmitters. Amplifiers are in general nonlinear and produce harmonics that must be suppressed using one or several narrow passband filters on the output. High-frequency stability can be hard to achieve in such filters because microwave systems may be subject to thermal drift caused by high-power loads or harsh environmental conditions like exposure to direct sunlight in the desert. Thus, system engineers need to estimate the drift of the passband frequency that arises due to thermal expansion of a filter.

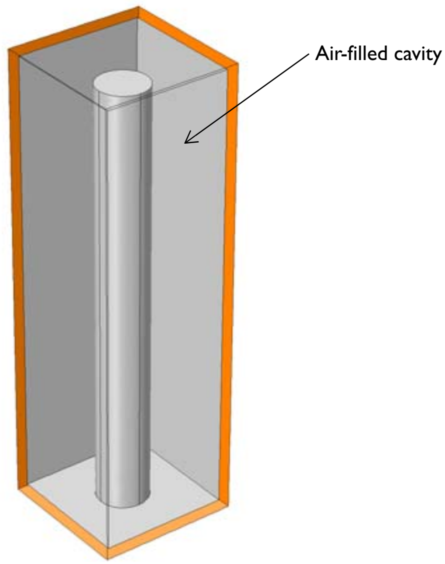


Figure 1: The microwave filter in this example consists of a thin metallic box, made of copper that contains a cylindrical post. This configuration forms a closed air-filled electromagnetic cavity between the box walls and the post.

Note: This example requires the RF Module and the Structural Mechanics Module.

Model Definition

Figure 1 shows the filter geometry. It consists of a box with a cylindrical post centered on one face. It is made of copper covered with a thin layer of silver to minimize losses. The silver layer not modeled is sufficiently thin to have a negligible influence on the device's thermal and mechanical properties. The all-copper design will be compared to a second design using both copper and steel.

Thermal expansion and the associated drift in eigenfrequency are caused by a uniform increase in the temperature of the cavity walls. The thermal expansion is readily computed using the Solid Mechanics physics interface from the Structural Mechanics Module. The eigenfrequency analysis of the cavity structure is easily performed using the 3D Electromagnetic Waves, Frequency Domain physics interface in the RF Module.

The model uses the Deformed Geometry (dg) interface to address the distorted shape of the filter geometry due to the thermal expansion. The deformed shape is used for the electromagnetic analysis.

Results and Discussion

The filter's temperature can rise due to power dissipation in the filter itself, in the surrounding electronics, or due to external heating. Figure 2 shows the thermal expansion results for a filter made entirely of copper.

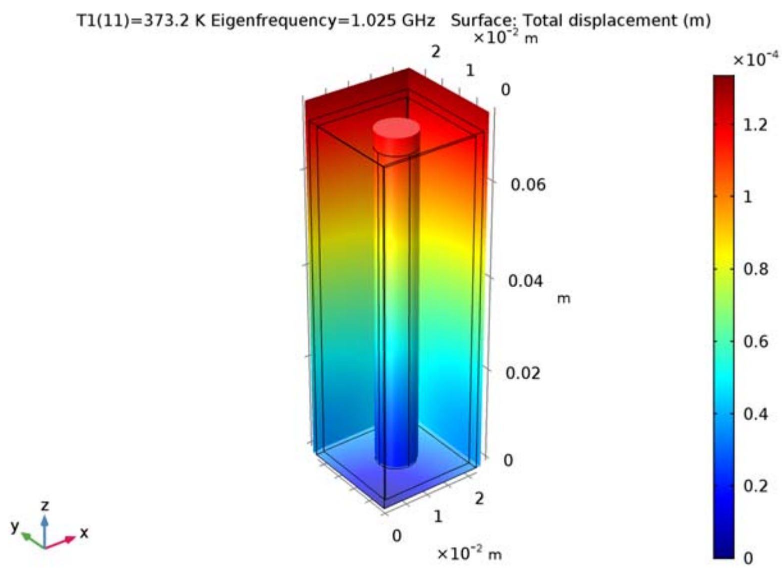


Figure 2: Thermal expansion at 100 °C above the reference temperature.

An actual filter usually consists of multiple cavities cascaded, but this discussion limits the analysis to one cell. Figure 3 shows the filter's lowest eigenfrequency. The typical quarter-wave resonance of the cylindrical post is clearly visible. A strong capacitive coupling between the top of the post and the nearby face of the box is also obvious.

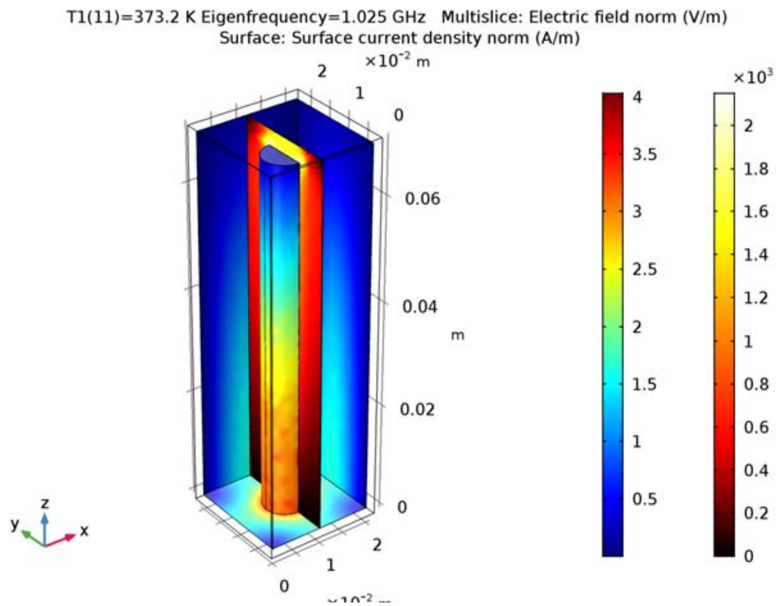


Figure 3: Results from the electromagnetic mode analysis. The plot shows the electric field and surface current patterns of the fundamental mode.

By repeating the structural and electromagnetic analyses for a number of operating temperatures, an eigenfrequency-versus-temperature curve is obtained. The results for two different designs appear in Figure 4.

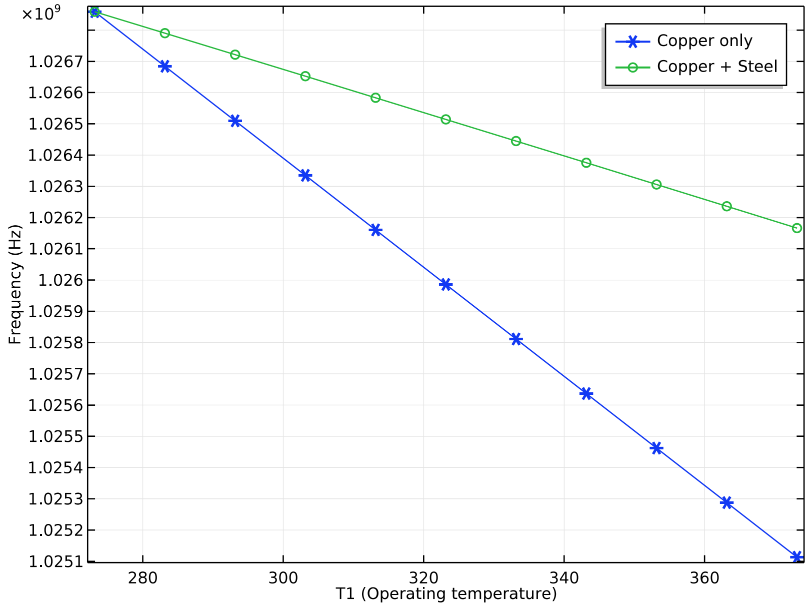


Figure 4: Eigenfrequency (Hz) versus temperature (K) for two different designs.

The first design, where the entire filter is made of copper, has been discussed already. For the second design the post is made of steel. It is obvious that the combination of the steel post and copper box is superior to a design using copper alone. The reason is the reduced capacitive coupling between the top of the post and the nearby face of the box, which results from the different coefficients of thermal expansion for the two materials. This coupling has a strong influence on the resonant frequency and, when reduced, counteracts the effects of an overall increase in cavity size.

Model Library path: `RF_Module/Filters/cavity_filter_thermal_expansion`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
7 Click **Add**.
8 Click **Done**.

In this model you use a parametric sweep to study thermal expansion as a function of the operating temperature. Begin by setting up a solver sequence for the three physics interfaces.

ADD STUDY

1 On 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**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Deformed Geometry (dg)** and **Electromagnetic Waves, Frequency Domain (emw)**.
5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Add Study** in the window toolbar.

STUDY 1

Step 1: Stationary

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

Step 2: Stationary 2

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)** and **Electromagnetic Waves, Frequency Domain (emw)**.

Step 3: Eigenfrequency

1 On the **Study** toolbar, click **Study Steps** and choose **Eigenfrequency>Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)** and **Deformed Geometry (dg)**.

4 Locate the **Study Settings** section. Select the **Desired number of eigenfrequencies** check box.

5 In the associated text field, type `1`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
T0	0[degC]	273.2 K	Reference temperature
T1	100[degC]	373.2 K	Operating temperature

GEOMETRY 1

Block 1 (blk1)

1 On 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 `0.02`.

4 In the **Depth** text field, type `0.02`.

5 In the **Height** text field, type `0.07`.

6 Click **Build Selected**.

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

Block 2 (blk2)

1 On 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 `0.022`.

4 In the **Depth** text field, type `0.022`.

5 In the **Height** text field, type `0.072`.

6 Locate the **Position** section. In the **x** text field, type `-0.001`.

7 In the **y** text field, type `-0.001`.

8 In the **z** text field, type `-0.001`.

Cylinder 1 (cyl1)

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

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

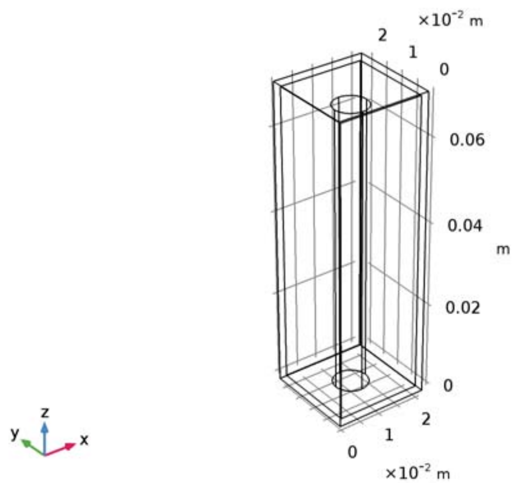
3 In the **Radius** text field, type `0.004`.

4 In the **Height** text field, type `0.067`.

5 Locate the **Position** section. In the **x** text field, type `0.01`.

6 In the **y** text field, type `0.01`.

7 On the **Geometry** toolbar, click **Build All**.



DEFINITIONS

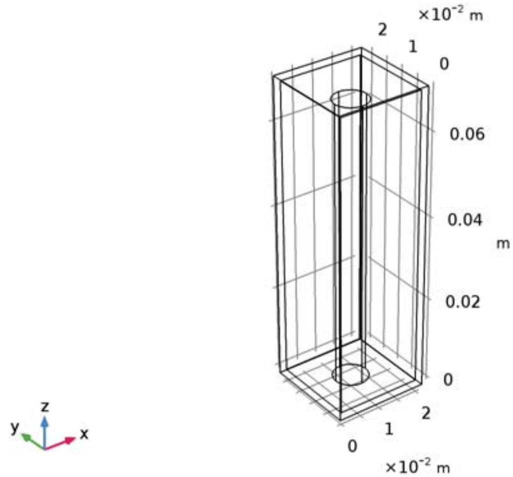
View 1

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

2 Right-click **View 1** and choose **Hide for Geometry**.

3 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

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



5 On the object **fin**, select Boundaries 1, 2, 4, 6, 7, and 9 only.

ADD MATERIAL

1 On 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>Copper**.

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

MATERIALS

Copper (mat1)

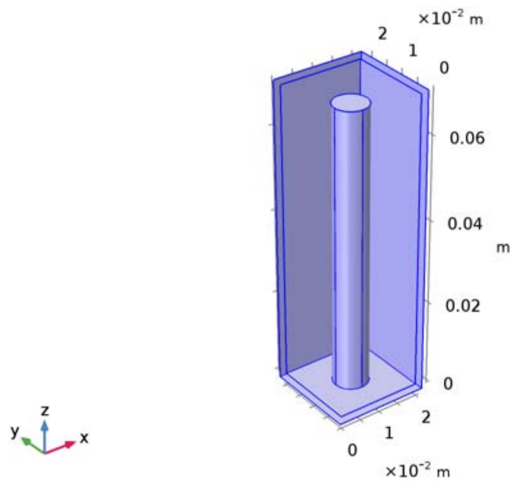
1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat1)**.

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

3 In the list, select **2**.

4 Click **Remove from Selection**.

5 Select Domains 1 and 3 only.



6 Click **Create Selection**.

7 In the **Create Selection** dialog box, type `Metal` in the **Selection name** text field.

8 Click **OK**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

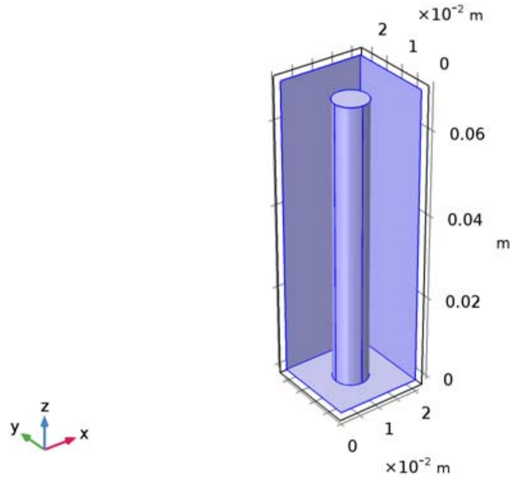
3 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat2)**.

2 Select Domain 2 only.



3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

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

5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

7 Click **OK**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

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

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

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 In the T text field, type `T1`.

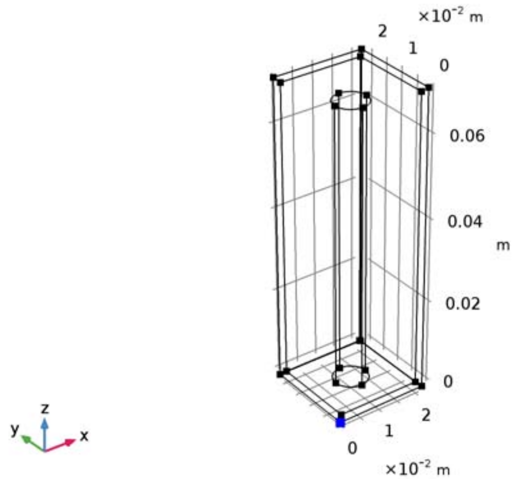
4 Locate the **Thermal Expansion Properties** section. In the T_{ref} text field, type `T0`.

Prescribed Displacement 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Points>Prescribed Displacement**.

Appropriate prescribed displacement on points must be applied to eliminate any translation or rotation of the structure.

2 Select Point 1 only.



3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

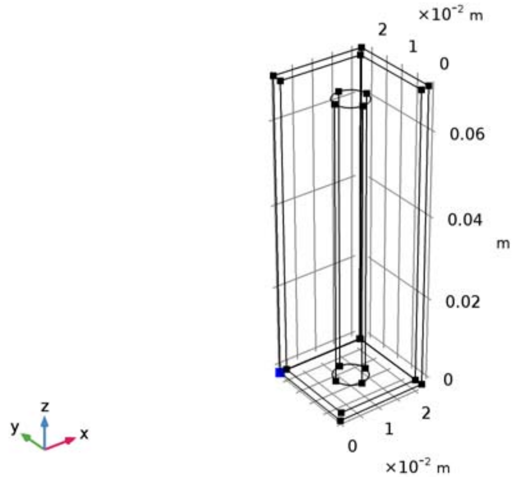
5 Select the **Prescribed in y direction** check box.

6 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1 Right-click **Solid Mechanics (solid)** and choose **Points>Prescribed Displacement**.

2 Select Point 3 only.



3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

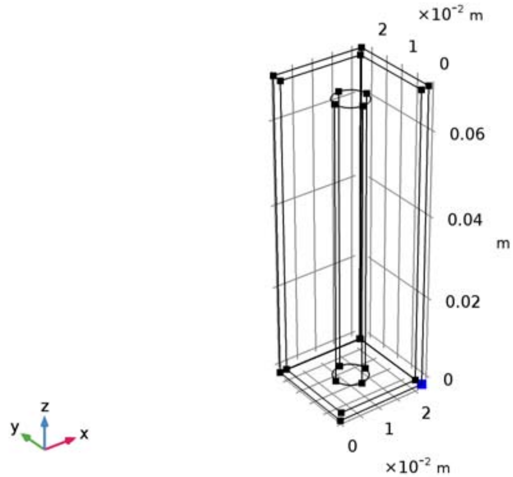
4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in z direction** check box.

Prescribed Displacement 3

1 Right-click **Solid Mechanics (solid)** and choose **Points>Prescribed Displacement**.

2 Select Point 21 only.



3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

5 Select the **Prescribed in z direction** check box.

DEFORMED GEOMETRY (DG)

Free Deformation 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Deformed Geometry (dg)** and choose **Free Deformation**.

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

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

Prescribed Mesh Displacement 2

1 In the **Model Builder** window, right-click **Deformed Geometry (dg)** and choose **Prescribed Mesh Displacement**.

2 Click the **Select All** button on the **Graphics** toolbar.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 In the d_X text field, type u.

5 In the d_Y text field, type v.

6 In the d_Z text field, type `w`.

Prescribed Deformation 1

1 Right-click **Deformed Geometry (dg)** and choose **Prescribed Deformation**.

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

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

4 Locate the **Prescribed Mesh Displacement** section. In the d_X text-field array, type `u` on the first row.

5 In the d_Y text-field array, type `v` on the second row.

6 In the d_Z text-field array, type `w` on the third row.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

The losses on metallic surfaces are negligible since the analysis frequency is low, so the boundaries are modeled as perfect electric conductors (PEC) by default.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.

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

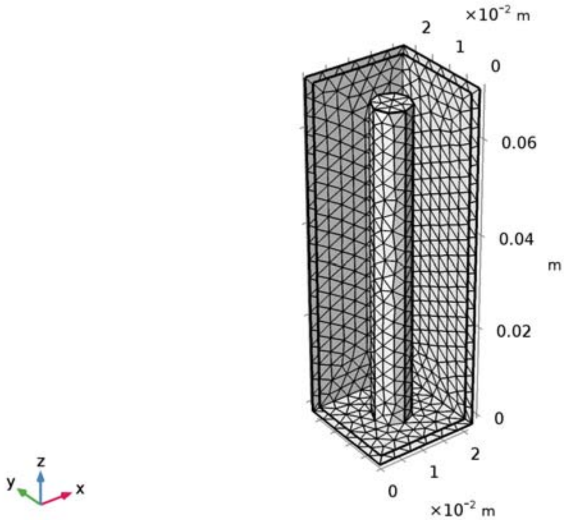
MESH 1

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

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

3 From the **Element size** list, choose **Coarser**.

4 Click **Build All**.



STUDY 1

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
T1		

5 Click **Range**.

6 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

7 In the **Start** text field, type `T0`.

8 In the **Stop** text field, type `T1`.

9 In the **Number of values** text field, type `11`.

10 Click **Replace**.

11 On the **Study** toolbar, click **Compute**.

Follow these instructions to reproduce the plot in Figure 2 that shows the structural deformation.

RESULTS

Stress (solid)

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol4)**.

Surface 1

1. In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.disp`.
4. On the **Stress (solid)** toolbar, click **Plot**.

Next, reproduce the electric field and surface current plot for the filter's lowest eigenmode, shown in Figure 3.

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 **Y-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. From the **Color table** list, choose **Thermal**.

Surface 1

1. In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Surface**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Currents and charge>emw.normJs - Surface current density norm**.
3. On the **Electric Field (emw)** toolbar, click **Plot**.

Having confirmed the correctness of the model setup, compute the thermal drift over the operating temperature range of 0° C to 100° C.

1D Plot Group 3

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol4)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Global 1

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

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.freq`	`Hz`	`Frequency`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

5 Click to expand the **Coloring and style** section. Locate 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 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Copper only`

9 On the **1D Plot Group 3** toolbar, click **Plot**.

Compare the copper-only design with one in which the post is made of steel.

ADD MATERIAL

1 On 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>Steel AISI 4340**.

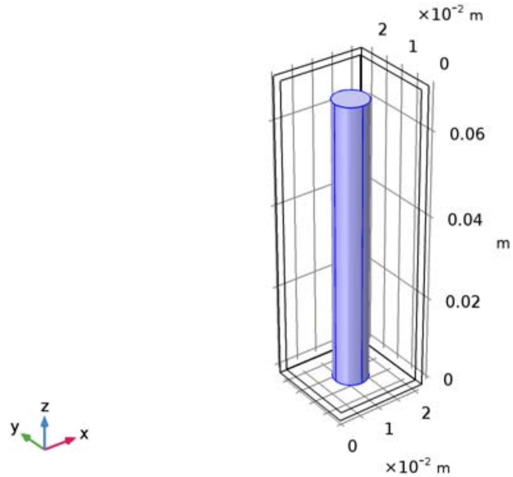
4 Click **Add to Component** in the window toolbar.

MATERIALS

Steel AISI 4340 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat3)**.

2 Select Domain 3 only.



3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

STUDY 1

On the **Study** toolbar, click **Create Solution Copy**.

RESULTS

Global 1

1 In the **Model Builder** window, under **Results>1D Plot Group 3** click **Global 1**.

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 - Copy 1 (sol16)**.

4 On the **Study** toolbar, click **Compute**.

Global 2

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

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

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

4 Click to expand the **Legends** section. In the table, enter the following settings:

Legends
`Copper + Steel`

5 On the **1D Plot Group 3** toolbar, click **Plot**.

This reproduces the plot in Figure 4.

Created in COMSOL Multiphysics 5.3

Computing Q-Factors and Resonant Frequencies of Cavity Resonators

Introduction

A classic benchmark example in computational electromagnetics is to find the resonant frequency and Q-factor of a cavity with lossy walls. Here, models of rectangular, cylindrical, and spherical cavities are shown to be in agreement with analytic solutions.

Model Definition

This example considers three geometries:

- a rectangular cavity of dimensions 0.9 in-by-0.9 in-by-0.4 in;
- a cylindrical cavity of radius 0.48 in and height 0.4 in; and
- a spherical cavity of radius 1.35 cm.

The cavity walls are assumed to be a good conductor, such as copper, with an electric conductivity of $5.7 \cdot 10^7$ S/m, and relative permeability and permittivity of unity. The interior of the cavity is assumed to be vacuum, with zero electric conductivity, and unit permeability and permittivity. The analytic solutions to these three cases are given in Ref. 1.

The lossy walls of the cavity are represented via the impedance boundary condition. This boundary condition accounts for the frequency dependent losses on the walls of a cavity due to the non-zero electric conductivity, which makes the eigenvalue problem nonlinear. When solving any eigenvalue problem, it is necessary to provide a frequency around which to search for modes. In addition, when solving a nonlinear eigenvalue problem, it is also necessary to provide a frequency at which to initially evaluate the frequency-dependent surface losses. Although the guesses for these frequencies do not need to be very close, solution time is less the closer they are.

It is usually possible to estimate the resonant frequency of interest, and to use this as an initial guess. It is also possible to quickly estimate the resonant frequency by building a second model that uses the perfect electrical conductor (PEC) boundary condition instead of the impedance boundary condition. A model that uses only PEC boundaries results in a linear eigenvalue problem, and is less computationally intensive to solve. Such a model only requires a rough guess at the frequency of the mode, and does not require a frequency at which to evaluate the surface losses. Therefore, it is often convenient to also solve a version of a model without losses.

Q-FACTOR AND RESONANT FREQUENCY IN CAVITY STRUCTURES

Q-factor is one of important parameters characterizing a resonant structure and defined as $Q = \omega$ (average energy stored/dissipated power). The average energy stored can be evaluated as a volume integral of Energy density time average (`emw.Wav`) and the dissipated power can be evaluated as a surface integral of Surface losses (`emw.Qsh`).

Another way to calculate Q-factor at the dominant mode is via equations in Ref. 1. For a rectangular cavity, the dominant mode is TE_{101}, at which the cavity provides the lowest resonant frequency. The Q-factor and resonant frequency at this mode is

$$Q_{\mathrm{TE}_{101}} = \frac{1.1107\eta}{R_s\left(1+\frac{a}{2b}\right)}, f_{\mathrm{TE}_{101}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}}\sqrt{\left(\frac{\pi}{a}\right)^2+\left(\frac{\pi}{c}\right)^2}$$

There are two dominant modes for a cylindrical cavity. One dominant mode of the cylindrical cavity is TE_{111} when the ratio between the height and radius is more than 2.03. The other dominant mode is TM_{010} when the ratio is less than 2.03. For this case, the Q-factor and resonant frequency are given as

$$Q_{\mathrm{TM}_{010}} = \frac{1.2025\eta}{R_s\left(1+\frac{a}{h}\right)}, f_{\mathrm{TM}_{010}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}}\sqrt{\left(\frac{2.40492}{a}\right)^2}$$

For a spherical cavity, TM mode provides the lowest resonant frequency.

$$Q_{\mathrm{TM}_{011}} = \frac{1.0041\eta}{R_s}, f_{\mathrm{TM}_{011}} = \frac{2.744}{2\pi a\sqrt{\mu\varepsilon}}$$

In the above equations, R_s is surface resistance defined as

$$R_s = \sqrt{\frac{\omega_r\mu}{2\sigma}}$$

and η is the characteristic impedance of free space, $\sqrt{\mu_0/\varepsilon_0}$.

These two analytical approaches are compared with the Q-factor obtained from Eigenfrequency analysis.

Results and Discussion

The analytic resonant frequencies and Q-factors for these three cases, and the results of the COMSOL model for various levels of mesh refinement, are shown below. These show that the solutions agree. As the mesh is refined, the polynomial basis functions used by the finite element method better approximate the analytic solutions, which are described by sinusoidal functions for the rectangular cavity and Bessel functions for the cylindrical and spherical cavities. This difference between the numerical results and the analytic solution is discretization error, and is always reduced with mesh refinement.

TABLE 1: RESULTS FOR THE TE101 MODE OF A RECTANGULAR CAVITY

MAXIMUM MESH SIZE	RESONANT FREQUENCY, GHZ (ANALYTIC=9.273)	Q-FACTOR (ANALYTIC=7770)
h_max	9.706	7039
h_max/2	9.283	7687
h_max/4	9.273	7765
h_max/8	9.273	7770

TABLE 2: RESULTS FOR THE TM010 MODE OF A CYLINDRICAL CAVITY

MAXIMUM MESH SIZE	RESONANT FREQUENCY, GHZ (ANALYTIC=9.412)	Q-FACTOR (ANALYTIC=8065)
h_max	9.458	7891
h_max/2	9.419	8004
h_max/4	9.411	8056
h_max/8	9.411	8065

TABLE 3: RESULTS FOR THE TM011 MODE OF A SPHERICAL CAVITY

MAXIMUM MESH SIZE	RESONANT FREQUENCY, GHZ (ANALYTIC=9.698)	Q-FACTOR (ANALYTIC=14594)
h_max	9.752	14121
h_max/2	9.723	14430
h_max/4	9.701	14616
h_max/8	9.697	14641

Note that convergence with respect to the mesh is fastest for the rectangular cavity and slowest for the spherical cavity. This is because the isoparametric finite-element mesh represents curved surfaces approximately, via second order polynomials by default. This introduces some small geometric discretization error that is always reduced with mesh refinement. Although it is possible to use different element orders, the default second-order curl element (also known as a vector or Nedelec element) is the best compromise between accuracy and memory requirements. Because memory requirements for three-dimensional models increase exponentially with increasing element order, and increasing number of elements, there is strong motivation to use as coarse a mesh as reasonable. Figure 1 shows the fields within the cavities, as well as the surface currents and surface losses.

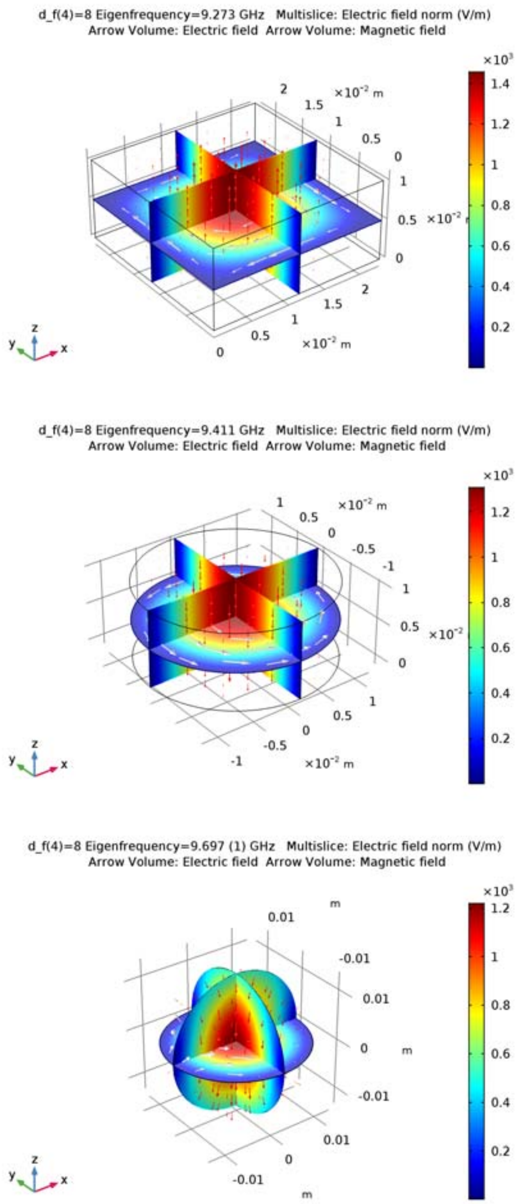


Figure 1: Arrow plots of electric and magnetic fields. Slice plot of electric field.

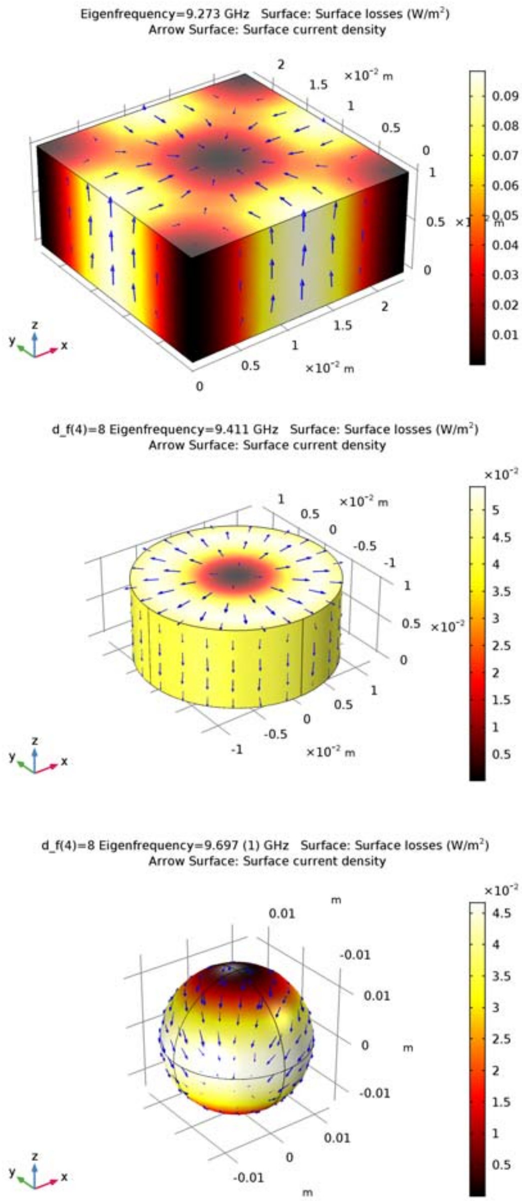


Figure 2: Arrow plots of surface currents. Surface plot of surface losses.

Notes About the COMSOL Implementation

Solve this example using an Eigenfrequency study. Search for a single eigenfrequency around $9 \cdot 10^9$ Hz. Because of the impedance boundary condition with a finite conductivity value, the model becomes a nonlinear eigenvalue problem and it is necessary to provide a frequency at which to initially evaluate the frequency-dependent surface losses. In the Eigenvalue Solver settings window you can see the linearization point is automatically specified to the value in “Search for eigenfrequencies around” in the study settings.

Reference

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, John Wiley & Sons, 1989.

Application Library path: `RF_Module/Verification_Examples/cavity_resonators`

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 **Preset Studies>Eigenfrequency**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_parameters.txt`.

   Here, `mu0_const` and `epsilon0_const` in the imported table are predefined COMSOL constants for the permeability and permittivity in free space. From the Value column you can read off the values `f_TE101_analytic_r` = 9.273 GHz, `Q_TE101_analytic_r` = 7770 for the rectangular cavity, `f_TM010_analytic_c` = 9.412 GHz, `Q_TM010_analytic_c` = 8065 for the cylindrical cavity, `f_TM011_analytic_s` = 9.698 GHz, and `Q_TM011_analytic_s` = 14594 for the spherical cavity.

Since air and lossy wall materials will be used on multiple components, add them on the global material node. They will be linked to each individual component later on.

ADD MATERIAL

1 On 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 Global Materials**.

GLOBAL DEFINITIONS

Air (mat1)

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

Material 2 (mat2)

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

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

3 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Relative Permittivity**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties>Relative Permeability**.

6 Click **Add to Material**.

7 In the **Material properties** tree, select **Basic Properties>Electrical Conductivity**.

8 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	`sigma_wall`	S/m	Basic

GEOMETRY 1

Create a block for the rectangular cavity.

Block 1 (blk1)

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

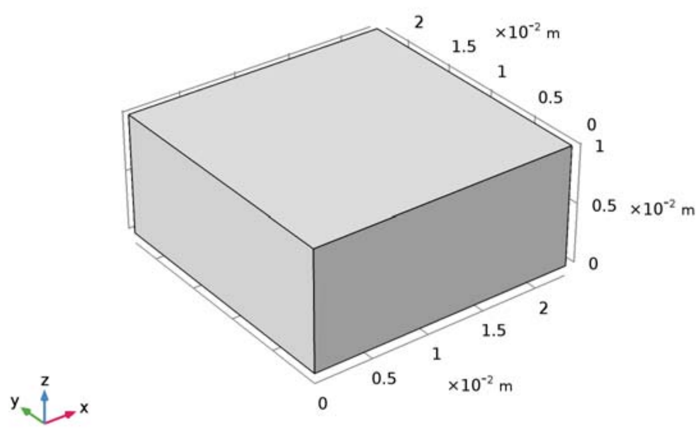
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

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

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

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

6 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. Override the default perfect electric conductor condition on the exterior boundaries by an impedance condition.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Physics-Controlled Mesh** section.
3. Clear the **Enable** check box.

Impedance Boundary Condition 1

1. Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** and choose **Impedance Boundary Condition**.
2. In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the model by linking the global material already created. First, apply air to all domains.

Material Link 2 (matlnk2)

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

   Define a lossy conductive material for all exterior boundaries.
3. In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.
4. From the **Geometric entity level** list, choose **Boundary**.
5. From the **Selection** list, choose **All boundaries**.
6. Locate the **Link Settings** section. From the **Material** list, choose **Lossy Wall (mat2)**.

DEFINITIONS

Add variables for Q-factor calculation and visualization. For this Q-factor calculation, add two integration coupling operators: one for volume and the other for surface integration.

Integration 1 (intop1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `int_v` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_s` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_model1_variables.txt`.

The `emw.` prefix is for the Electromagnetic Waves, Frequency Domain interface in the first model. `Wav` and `Qsh` are Energy density time average and Surface losses, respectively. `Qfactor` included in this text file shows up in orange indicating an unknown variable. It will be known after solving the model.

MESH 1

The maximum mesh size is one dimension of the cavity scaled inversely by `d_f`, a discretization factor defined in Parameters. The discretization factor is also used as a parametric sweep variable to see the effect of the mesh refinement.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max_r/d_f`.

5 In the **Maximum element growth rate** text field, type `2`.

6 In the **Curvature factor** text field, type `1`.

7 In the **Resolution of narrow regions** text field, type `0.1`.

8 Click **Build All**.

STUDY 1

Provide the number of modes and a frequency around which to search for modes.

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `1`.

5 Select the **Search for eigenfrequencies around** check box.

6 In the associated text field, type `9[GHz]`.

Add a Parametric Sweep over the discretization factor, d_f.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
d_f	1 2 4 8	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows the distribution of the norm of the electric field. Add arrow plots of the electric and magnetic fields.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Electric>emw.Ex,emw.Ey,emw.Ez - Electric field**.

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

Arrow Volume 2

1 Right-click **Electric Field (emw)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Magnetic>emw.Hx,emw.Hy,emw.Hz - Magnetic field**.

3 Locate the **Arrow Positioning** section. Find the **Z grid points** subsection. In the **Points** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

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

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

Compare the resulting plot with that shown in Figure 1, top. The exact numbers that you get may differ slightly.

Add a surface plot of the surface losses and an arrow plot of the surface current (Figure 2, top).

Surface 1

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

2 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Surface**.

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qsh - Surface losses**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

3D Plot Group 2

1 In the **Model Builder** window, under **Results** click **3D Plot Group 2**.

2 In the **Settings** window for **3D Plot Group**, type `Surface Losses (emw)` in the **Label** text field.

Arrow Surface 1

1 Right-click **Results>Surface Losses (emw)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Currents and charge>emw.Jsx,...,emw.Jsz - Surface current density**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

4 On the **Surface Losses (emw)** toolbar, click **Plot**.

ROOT

Next, set up a model for the cylindrical cavity.

1 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

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

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

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On 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**.

You will copy the settings from the existing study later on.

4 Click **Add Study** in the window toolbar.

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

GEOMETRY 2

Cylinder 1 (cyl1)

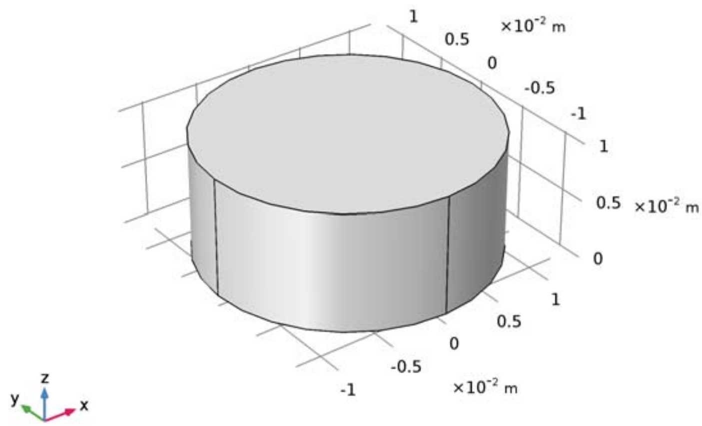
1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `a_c`.

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

5 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EMW2)

Set up the second physics interface. The steps are same as for the first model.

Impedance Boundary Condition 1

1. In the **Model Builder** window, under **Component 2 (comp2)** right-click **Electromagnetic Waves, Frequency Domain 2 (emw2)** and choose **Impedance Boundary Condition**.
2. In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the second model. Apply air to all domains.

Material Link 4 (matlnk4)

1. In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Material Link**.
2. Right-click **Component 2 (comp2)>Materials** and choose **Material Link**.

   Define a lossy conductive material for all exterior boundaries.
3. In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.

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

5 From the **Selection** list, choose **All boundaries**.

6 Locate the **Link Settings** section. From the **Material** list, choose **Lossy Wall (mat2)**.

DEFINITIONS

Add variables and two integration coupling operators. The purpose of these is same as in the first model.

Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_v` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 4 (intop4)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_s` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

Variables 2

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_model2_variables.txt`.

   The `emw2.` prefix refers to the Electromagnetic Waves, Frequency Domain interface for the second model.

MESH 2

Apply the same logic in the mesh set up as you have done in the first model.

Size

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **Free Tetrahedral**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max_c/d_f`.

5 In the **Maximum element growth rate** text field, type `2`.

6 In the **Curvature factor** text field, type `1`.

7 In the **Resolution of narrow regions** text field, type `0.1`.

8 Click **Build All**.

STUDY 1

Parametric Sweep

Select both **Study 1> Step 1: Eigenfrequency 1** and **Study 1> Parametric Sweep 1** using shift-key. **Copy** them and **Paste** on Study 2.

STUDY 2

Step 1: Eigenfrequency 1

1 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency 1**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Electromagnetic Waves, Frequency Domain		physics
Electromagnetic Waves, Frequency Domain 2	√	physics

4 On the **Home** toolbar, click **Compute**.

RESULTS

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw2)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2> Electromagnetic Waves, Frequency Domain 2>Electric>emw2.Ex,emw2.Ey,emw2.Ez - Electric field**.

3 On the **Electric Field (emw2)** toolbar, click **Plot**.

Arrow Volume 2

1 Right-click **Electric Field (emw2)** and choose **Arrow Volume**.
2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2>Electromagnetic Waves, Frequency Domain 2>Magnetic>emw2.Hx,emw2.Hy,emw2.Hz - Magnetic field**.
3 Locate the **Arrow Positioning** section. Find the **Z grid points** subsection. In the **Points** text field, type `1`.
4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.
5 On the **Electric Field (emw2)** toolbar, click **Plot**.
6 Click the **Zoom Extents** button on the **Graphics** toolbar.

   The plot should now look like that in Figure 1, middle.

Again, add a surface plot of the surface losses and an arrow plot of the surface current (Figure 2, middle).

3D Plot Group 4

1 On 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 **Data set** list, choose **Study 2/Parametric Solutions 2 (6) (sol8)**.

Surface 1

1 Right-click **3D Plot Group 4** and choose **Surface**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2>Electromagnetic Waves, Frequency Domain 2>Heating and losses>emw2.Qsh - Surface losses**.
3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

3D Plot Group 4

1 In the **Model Builder** window, under **Results** click **3D Plot Group 4**.
2 In the **Settings** window for **3D Plot Group**, type `Surface Losses (emw2)` in the **Label** text field.

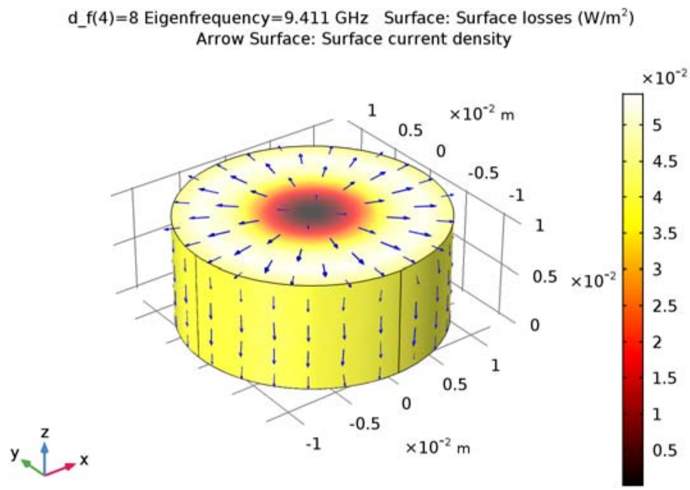
Arrow Surface 1

1 Right-click **Results>Surface Losses (emw2)** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2>**

Electromagnetic Waves, Frequency Domain 2>Currents and charge>emw2.Jsx,...,emw2.Jsz - Surface current density.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

4 On the **Surface Losses (emw2)** toolbar, click **Plot**.



ROOT

Now add a model for the spherical cavity.

1 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

GEOMETRY 3

In the **Model Builder** window, under **Component 3 (comp3)** click **Geometry 3**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

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

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

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On 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 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 3

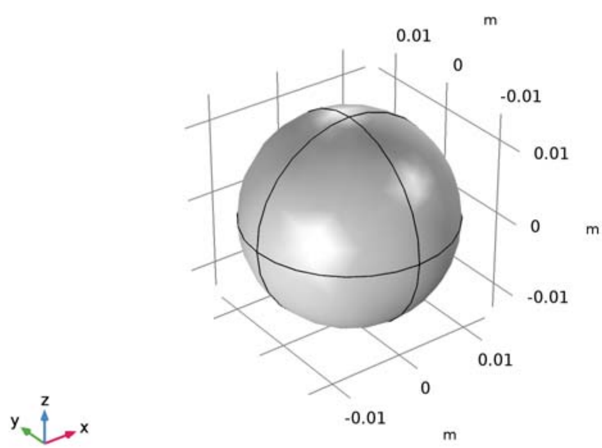
Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `a_s`.

4 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 3 (EMW3)

Set up the third physics interface.

Impedance Boundary Condition 1

1 In the **Model Builder** window, under **Component 3 (comp3)** right-click **Electromagnetic Waves, Frequency Domain 3 (emw3)** and choose **Impedance Boundary Condition**.

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

3 From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the third model. Apply air to all domains.

Material Link 6 (matlnk6)

1 In the **Model Builder** window, under **Component 3 (comp3)** right-click **Materials** and choose **Material Link**.

2 Right-click **Component 3 (comp3)>Materials** and choose **Material Link**.

Define a lossy conductive material for all exterior boundaries.

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

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

5 From the **Selection** list, choose **All boundaries**.

6 Locate the **Link Settings** section. From the **Material** list, choose **Lossy Wall (mat2)**.

DEFINITIONS

Add variables and two integration coupling operators.

Integration 5 (intop5)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_v` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 6 (intop6)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_s` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

Variables 3

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_model3_variables.txt`.

The `emw3.` prefix in the imported table is for the physics interface, **Electromagnetic Waves, Frequency Domain**, in the third model.

MESH 3

Size

1 In the **Model Builder** window, under **Component 3 (comp3)** right-click **Mesh 3** and choose **Free Tetrahedral**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max_s/d_f`.

5 In the **Maximum element growth rate** text field, type `2`.

6 In the **Curvature factor** text field, type `1`.

7 In the **Resolution of narrow regions** text field, type `0.1`.

8 Click **Build All**.

STUDY 2

Parametric Sweep 1

Select both **Study 2> Step 1: Eigenfrequency 1** and **Study 2> Parametric Sweep 1** using shift-key. **Copy** them and **Paste** on Study 3.

STUDY 3

Step 1: Eigenfrequency 1

1 In the **Model Builder** window, under **Study 3** click **Step 1: Eigenfrequency 1**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Electromagnetic Waves, Frequency Domain		physics
Electromagnetic Waves, Frequency Domain 2		physics
Electromagnetic Waves, Frequency Domain 3	√	physics

Solution 13 (sol13)

1 On the **Study** toolbar, click **Show Default Solver**.

2 Click **Compute**.

RESULTS

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw3)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 3>Electromagnetic Waves, Frequency Domain 3>Electric>emw3.Ex,emw3.Ey,emw3.Ez - Electric field**.

Arrow Volume 2

1 Right-click **Electric Field (emw3)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 3>Electromagnetic Waves, Frequency Domain 3>Magnetic>emw3.Hx,emw3.Hy,emw3.Hz - Magnetic field**.

3 Locate the **Arrow Positioning** section. Find the **Z grid points** subsection. In the **Points** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

5 On the **Electric Field (emw3)** toolbar, click **Plot**.

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

Compare the resulting plot with that shown in Figure 1, bottom.

Again, add a surface plot of the surface losses and an arrow plot of the surface current (Figure 2, bottom).

3D Plot Group 6

1 On 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 **Data set** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

Surface 1

1 Right-click **3D Plot Group 6** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 3>Electromagnetic Waves, Frequency Domain 3>Heating and losses>emw3.Qsh - Surface losses**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

3D Plot Group 6

1 In the **Model Builder** window, under **Results** click **3D Plot Group 6**.

2 In the **Settings** window for **3D Plot Group**, type `Surface Losses (emw3)` in the **Label** text field.

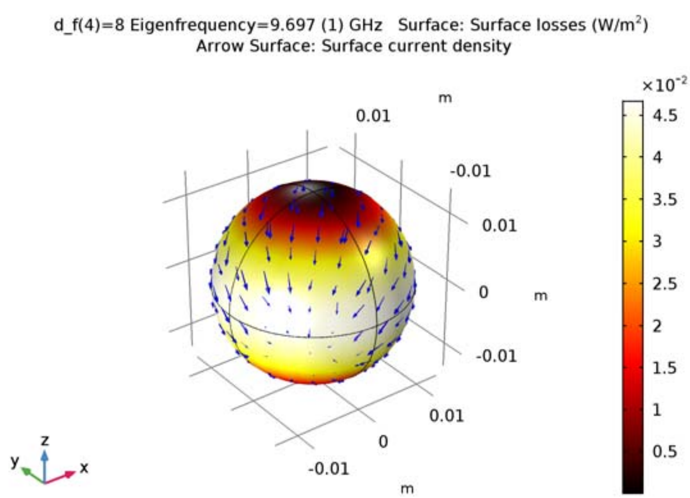
Arrow Surface 1

1 Right-click **Results>Surface Losses (emw3)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 3> Electromagnetic Waves, Frequency Domain 3>Currents and charge>emw3.Jsx,...,emw3.Jsz - Surface current density**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

4 On the **Surface Losses (emw3)** toolbar, click **Plot**.



Derived Values

Finish by evaluating the Q-factor and resonant frequency. Compare them with those values in Table 1, Table 2 and Table 3.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Eigenfrequency selection** list, choose **First**.

5 From the **Table columns** list, choose **Inner solutions**.

6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>Q_computed - Q-factor, computed from eigenvalue**.

7 Click **Evaluate**.

8 Locate the **Data** section. From the **Data set** list, choose **Study 2/ Parametric Solutions 2 (6) (sol8)**.

9 Click **Evaluate**.

10 From the **Data set** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

11 Click **Evaluate**.

Global Evaluation 2

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>Q_definition - Q-factor, definition**.

5 Click **New Table**.

6 Locate the **Data** section. From the **Data set** list, choose **Study 2/ Parametric Solutions 2 (6) (sol8)**.

7 Click **Evaluate**.

8 From the **Data set** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

9 Click **Evaluate**.

Global Evaluation 3

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>frequency - Frequency, simulated**.

5 Click **New Table**.

6 Locate the **Data** section. From the **Data set** list, choose **Study 2/ Parametric Solutions 2 (6) (sol8)**.

7 Click **Evaluate**.

8 From the **Data set** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

9 Click **Evaluate**.

Created in COMSOL Multiphysics 5.3

Connecting a 3D Electromagnetic Wave Model to an Electrical Circuit

Introduction

An application built with the RF Module can be connected to an electrical circuit equivalent, if there is some structure outside of the model space that you wish to approximate as a circuit equivalent. An example is shown in Figure 1, the 3D model of a coaxial cable is connected to a voltage source, in series with a matched impedance, and sees a load, also of matched impedance.

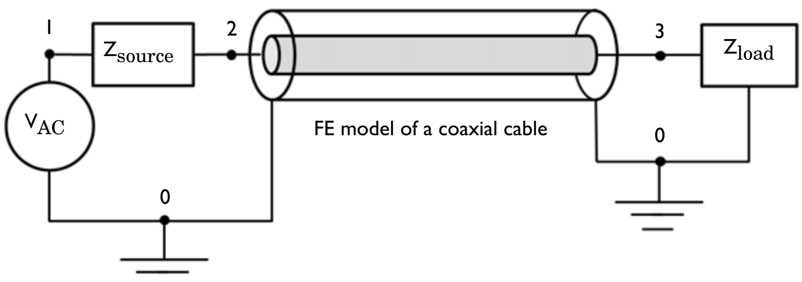


Figure 1: Schematic of a section of a coaxial transmission line connected to a voltage source, source impedance, and load.

Model Definition

The geometry in this example is a short section of a air-filled coaxial transmission line, shown schematically in Figure 1. A 3D modeling space is used to model the coaxial cable. The walls of the coax are treated as perfect electric conductors. This is appropriate when the skin depth, and the losses in the conductors, are insignificant.

At one end of the coaxial cable, Lumped Port boundary condition is used to connect the model to nodes 0 and 2 of the Electrical Circuit. A Voltage Source between circuit nodes 0 and 1 excites the system, and a Resistor representing the source impedance is added between nodes 1 and 2. Node 0 is specified as the Ground Node by default, which fixes the absolute voltage. The connection from the Electrical Circuit model to the Electromagnetic Waves interface is via the External I Vs. U features.

At the other end of the coaxial cable, another Lumped Port boundary condition is used to connect the model to nodes 3 and 0 of the Electrical Circuit. A Resistor which works as a matched load is added between nodes 3 and 0. At any non-zero frequency, the absolute voltage has no well-defined meaning, voltage only has a meaning as the path integral of electric field between two points, so any arbitrary point in the model can be chosen to have zero voltage. If you are working with a purely RF model, without an electrical circuit, it is not even possible to fix the absolute voltage. However, when using the Electrical Circuit interface, it requires that the absolute voltage be fixed at one node (Node 0) in the model.

Results and Discussion

Figure 2 is a combined plot of the default electric field norm, magnetic field, and power flow.

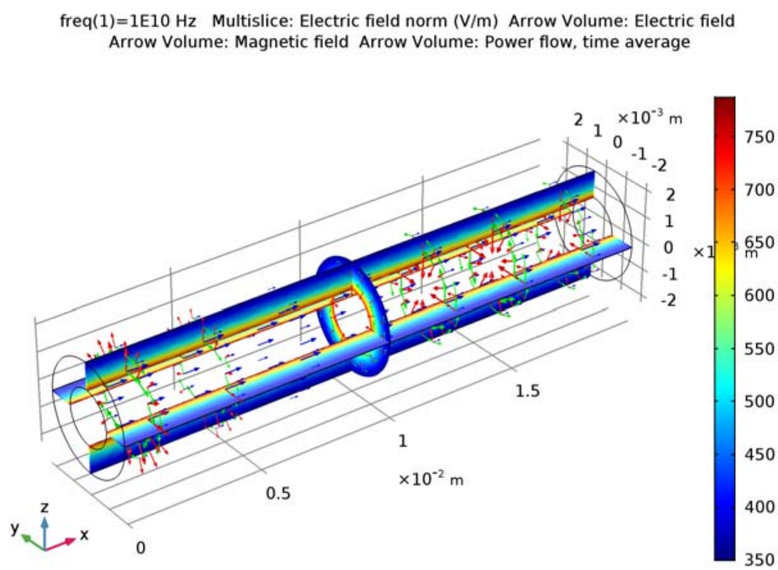


Figure 2: Electric field norm (multislices) and magnetic field, and power flow (green, blue arrows) inside the coaxial cable.

The fields and power flow plot shows the TEM wave propagation inside the coaxial cable, which is excited by the Electrical Circuit interface.

Notes About the COMSOL Implementation

The Electrical Circuit interface is located under the AD/DC Module branch, but it is included with the RF Module.

Application Library path: `RF_Module/Transmission_Lines_and_Waveguides/coaxial_cable_circuit`

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 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Frequency Domain**.

8 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 In the **Frequencies** text field, type `10[GHz]`.

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 In the table, enter the following settings:

Name	Expression	Value	Description
r_coax	1[mm]	0.001 m	Coax inner radius
R_coax	2[mm]	0.002 m	Coax outer radius
L_coax	20[mm]	0.02 m	Length of coax core into cavity
Z_coax	Z0_const/(2*pi)* log(R_coax/r_coax)	41.56 Ω	Analytical impedance

Here, Z0_const is a predefined COMSOL constant for the speed of the light and the wave impedance in vacuum, respectively.

GEOMETRY 1

Create the geometry of the coaxial cable using two cylinders.

Cylinder 1 (cyl1)

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

2 In the **Settings** window for **Cylinder**, type Coax outer in the **Label** text field.

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

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

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 2 (cyl2)

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

2 In the **Settings** window for **Cylinder**, type Coax inner in the **Label** text field.

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

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

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

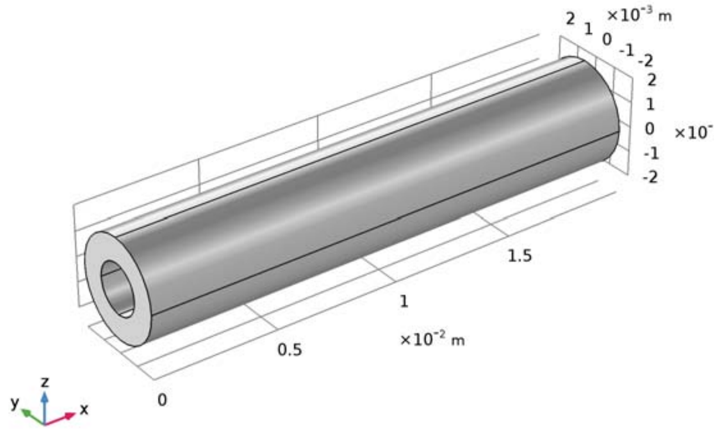
2 Select the object **cyl1** only.

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

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

5 Select the object **cyl2** only.

6 Click **Build All Objects**.



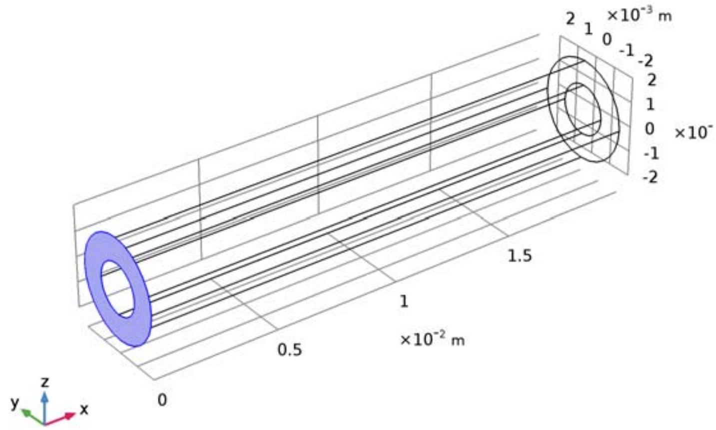
7 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Lumped Port 1

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

2 Select Boundary 1 only.



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

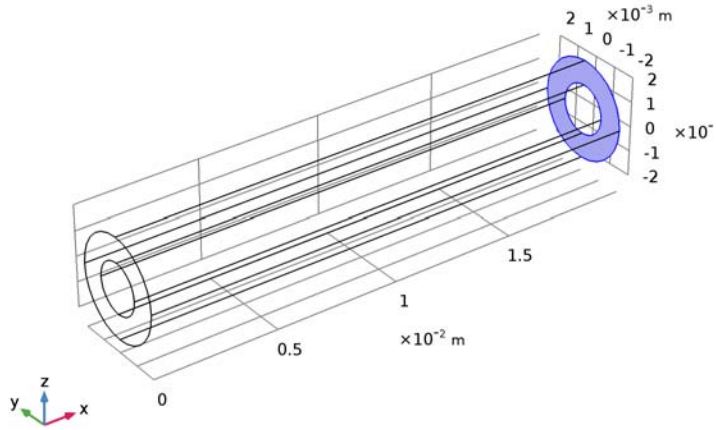
4 From the **Type of lumped port** list, choose **Coaxial**.

5 From the **Terminal type** list, choose **Circuit**.

Lumped Port 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 10 only.



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

4 From the **Type of lumped port** list, choose **Coaxial**.

5 From the **Terminal type** list, choose **Circuit**.

ELECTRICAL CIRCUIT (CIR)

Voltage Source V1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrical Circuit (cir)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. From the **Source type** list, choose **AC-source**.

Resistor R1

1 In the **Model Builder** window, right-click **Electrical Circuit (cir)** and choose **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `Z_coax`.

Resistor R2

1 Right-click **Electrical Circuit (cir)** and choose **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	0

4 Locate the **Device Parameters** section. In the R text field, type `Z_coax`.

External I Vs. U 1

1 Right-click **Electrical Circuit (cir)** and choose **External Couplings>External I Vs. U**.

2 In the **Settings** window for **External I Vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **External Device** section. From the V list, choose **Lumped port voltage (emw/lport1)**.

External I Vs. U 2

1 Right-click **Electrical Circuit (cir)** and choose **External Couplings>External I Vs. U**.

2 In the **Settings** window for **External I Vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	0

4 Locate the **External Device** section. From the *V* list, choose **Lumped port voltage (emw/lport2)**.

MATERIALS

Next, assign material properties on the model. Specify air for the coaxial cable.

ADD MATERIAL

1 On 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.

MATERIALS

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

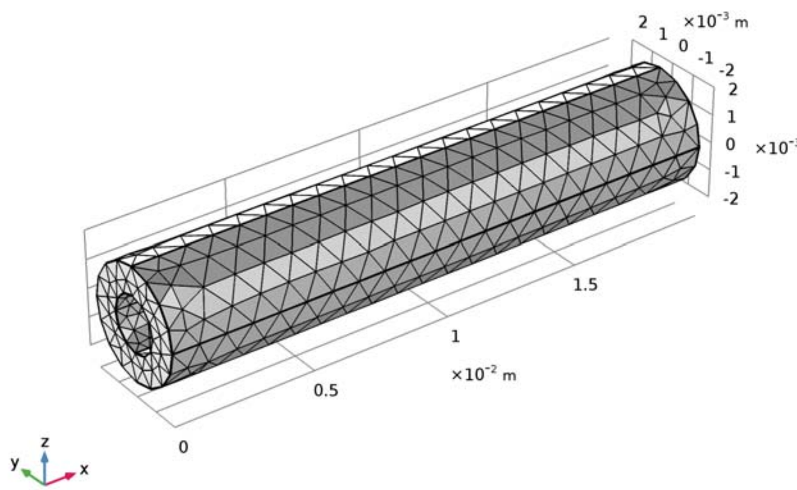
MESH 1

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

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

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Electric Field (emw)

The default plot shows the E-field norm inside the coaxial cable. Add arrow plots for the electric field, magnetic field, and power flow.

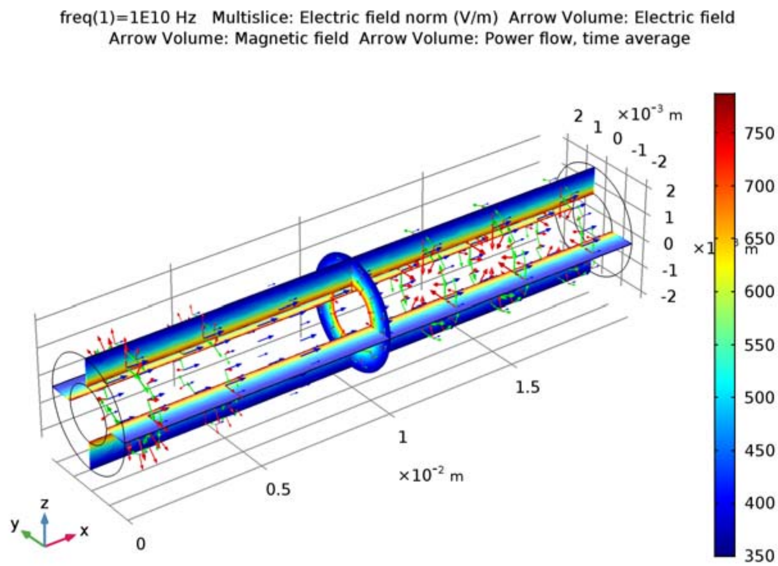
Arrow Volume 2

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Volume**.
2 Right-click **Electric Field (emw)** and choose **Arrow Volume**.
3 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Magnetic>emw.Hx,emw.Hy,emw.Hz - Magnetic field**.
4 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

Arrow Volume 3

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Volume**.
2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Energy and power>emw.Poavx,...,emw.Poavz - Power flow, time average**.
3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

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



Compare the resulting plot with that shown in Figure 2.

Created in COMSOL Multiphysics 5.3

Finding the Impedance of a Coaxial Cable

Introduction

The coaxial cable (coax) is one of the most ubiquitous transmission line structures. It is composed of a central circular conductor, surrounded by an annular dielectric and shielded by an outer conductor; see Figure 1. In this example, you compute the electric and magnetic field distributions inside the coax. Using these fields, you then compute the characteristic impedance and compare the result with the known analytic expression.

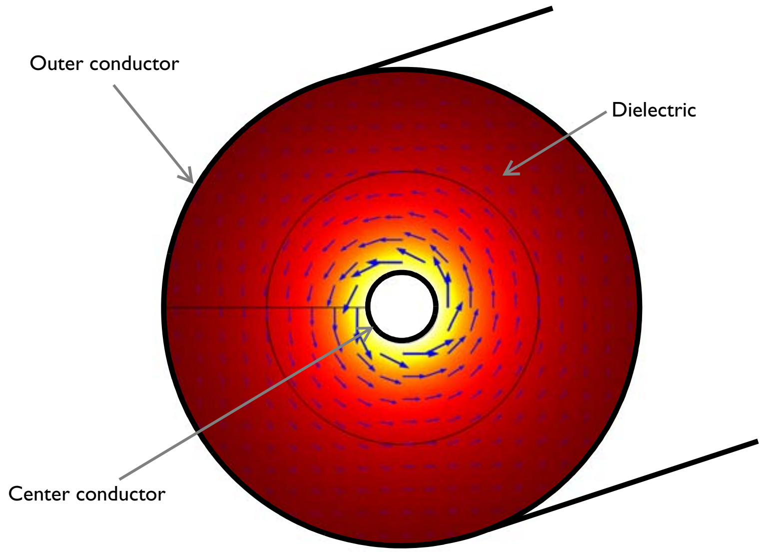


Figure 1: Cross section of a coaxial cable. The arrows visualize the magnetic field.

Model Definition

Because a coax operates in TEM mode—with the electric and magnetic fields normal to the direction of propagation along the cable—modeling a 2D cross section suffices to compute the fields and the impedance. For this example, assume perfect conductors and a lossless dielectric with relative permittivity $\varepsilon_r = 2.4$. The inner and outer radii are 0.5 mm and 3.43 mm, respectively.

The characteristic impedance, $Z_0 = V/I$, of a transmission line relates the voltage between the conductors to the current through the line. Although the model does not involve computing the potential field, the voltage of the TEM waveguide can be evaluated as a line integral of the electric field between the conductors:

$$V = V_{\mathrm{i}} - V_{\mathrm{o}} = -\int_{r_o}^{r_i} \mathbf{E} \cdot d\mathbf{r} \tag{1}$$

Similarly, the current is obtained as a line integral of the magnetic field along the boundary of either conductor or any closed contour, C, bisecting the space between the conductors:

$$I = \oint_C \mathbf{H} \cdot d\mathbf{r}$$

The voltage and current in the direction out of the plane are positive for integration paths oriented as in Figure 2.

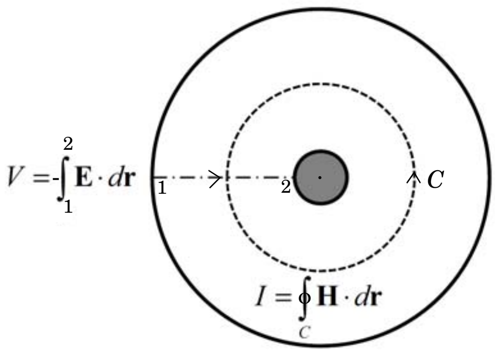


Figure 2: The impedance of a coaxial cable can be found from the voltage, V, and current, I, which are computed via line integrals as shown.

The value of Z_0 obtained in this way, should be compared with the analytic result

$$Z_{0,\mathrm{analytic}} = \frac{1}{2\pi}\sqrt{\frac{\mu_0}{\varepsilon_r \varepsilon_0}} \log\left(\frac{r_{\mathrm{o}}}{r_{\mathrm{i}}}\right) \approx 74.5\ \Omega$$

Results and Discussion

Figure 3 is a combined plot of the electric field magnitude and the magnetic field visualized as an arrow plot.

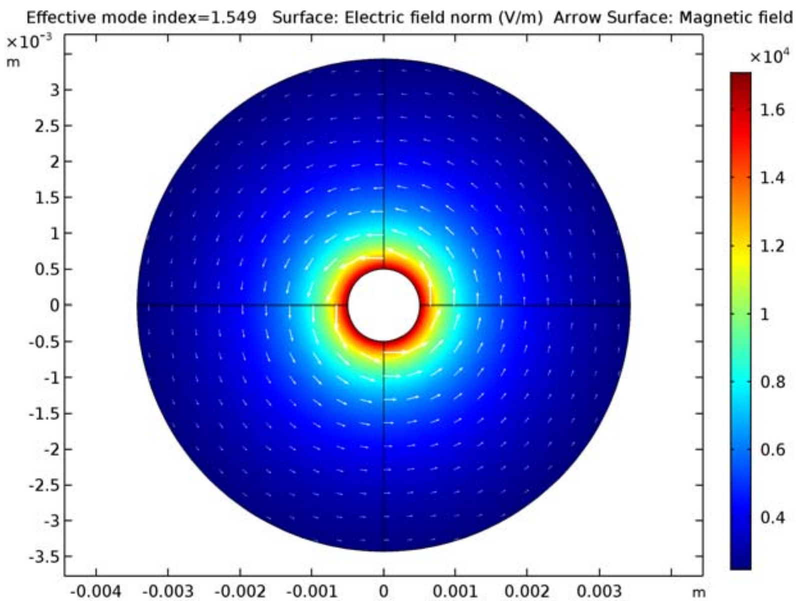


Figure 3: Electric field magnitude (surface) and magnetic field (arrows) inside the coaxial cable.

The impedance computed with the default mesh is $Z_0 = 74.65\ \Omega$. When the mesh is refined, the result approaches the analytic value of $74.5\ \Omega$.

Notes About the COMSOL Implementation

Solve this example using a Mode Analysis study. The effective mode index for the propagating TEM mode is $n_{\text{eff}} = \sqrt{\varepsilon_r} \approx 1.5$. Use the default frequency, $f = 1$ GHz, which is well below the cut-off frequency for TE modes and TM modes for the chosen cable diameter.

Application Library path: `RF_Module/Verification_Examples/coaxial_cable_impedance`

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 **2D**.
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 **Preset Studies>Mode Analysis**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
r_i	0.5[mm]	5E-4 m	Coax inner radius
r_o	3.43[mm]	0.00343 m	Coax outer radius
eps_r	2.4	2.4	Relative dielectric constant
Z0_analytic	(Z0_const/(2*pi* sqrt(eps_r)))* log(r_o/r_i)	74.53 Ω	Characteristic impedance, analytic

Here `Z0_const` is a predefined COMSOL constant for the characteristic impedance of vacuum, $Z_0 = \mathrm{sqrt}(\mu_0/\varepsilon_0)$. From the Value column you can read off the value `Z0, analytic` = 74.53 Ω.

GEOMETRY 1

Create the geometry using a single circle node with the radius of the outer conductor and an extra layer representing the inner conductor.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `r_o`.

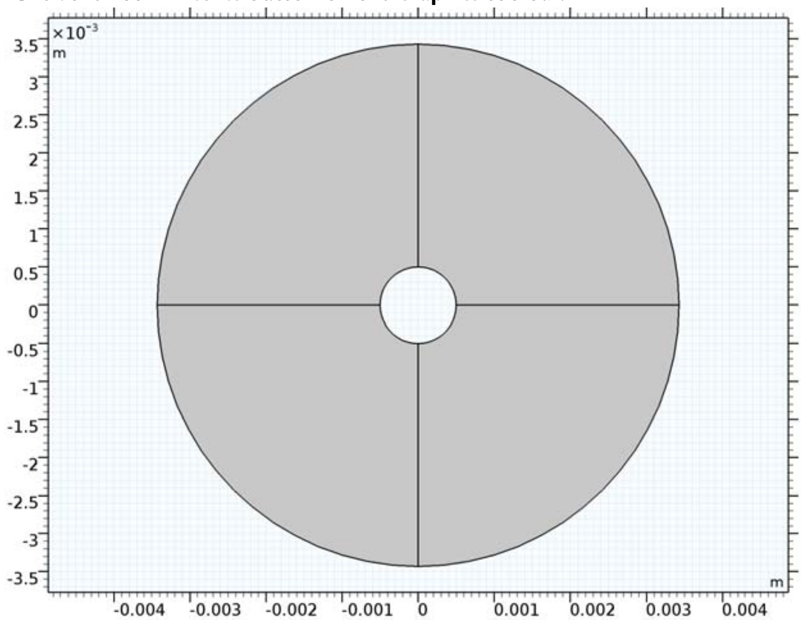
5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`r_o-r_i`

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

An advantage of using layers is that you automatically get a radial line to use for computing the voltage as a line integral of the electric field.

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



MATERIALS

Define a dielectric material for the region between the conductors.

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**, type `Insulator` in the **Label** text field.

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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	`eps_r`	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

DEFINITIONS

Add a variable for the characteristic impedance computed as the voltage between the conductors divided by the current through the cable. For this purpose, you need two integration coupling operators.

Integration 1 (intop1)

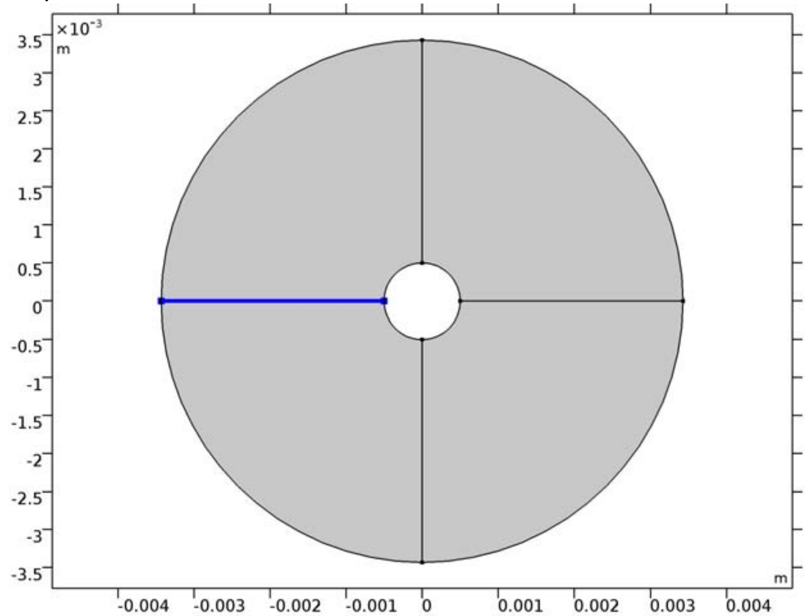
1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_rad` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

Any of the four interior boundaries that connect the two conductors would do.



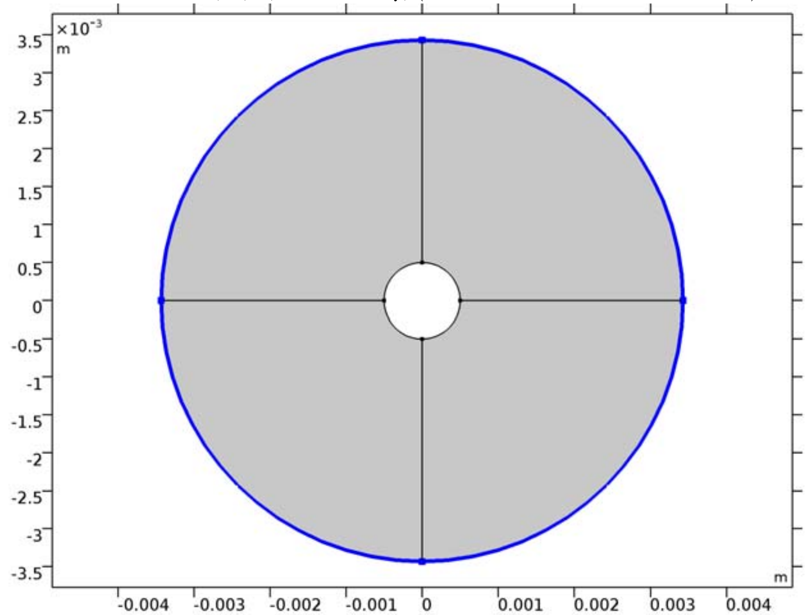
Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_circ` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 5, 6, 9, and 12 only, (the outer conductor boundaries).



Now define the variable for the characteristic impedance computed from the simulation.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`V`	`int_rad(-emw.Ex*t1x-emw.Ey*t1y)`	V	`Voltage`
`I`	`-int_circ(emw.Hx*t1x+emw.Hy*t1y)`	A	`Current`
`Z0_model`	`V/I`	Ω	`Characteristic impedance`

Here, `t1x` and `t1y` are the tangential vector components along the integration boundaries (`1` refers to the boundary dimension). Shortly, you will determine the tangential vector directions along the boundaries using an arrow plot of `t1`. The signs in the definitions above are chosen such that $V = V_i - V_o$ (see Equation 1) and to have a positive current value correspond to a current in the positive z direction. The `emw.`

prefix gives the correct physics-interface scope for the electric and magnetic field vector components.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

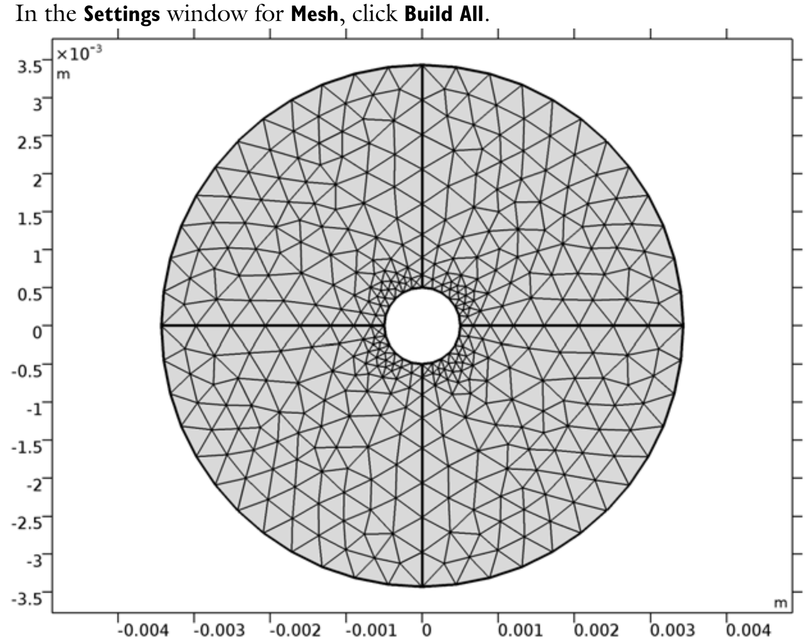
Keep the default physics settings, which include perfect electric conductor conditions for the outer boundaries.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Physics-Controlled Mesh** section.
3 Clear the **Enable** check box.

MESH 1

Use the default mesh.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

Step 1: Mode Analysis

1. In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.
2. Select the **Desired number of modes** check box.
3. In the associated text field, type `1`.
4. Select the **Search for modes around** check box.
5. In the associated text field, type `sqrt(eps_r)`.
6. On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows the distribution of the norm of the electric field. Add an arrow plot of the magnetic field.

Arrow Surface 1

1. In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.
3. Find the **X grid points** subsection. In the **Points** text field, type `21`.
4. Find the **Y grid points** subsection. In the **Points** text field, type `21`.
5. Locate the **Coloring and Style** section. Select the **Scale factor** check box.
6. In the associated text field, type `7e-6`.

   You can use the slider to adjust the arrow lengths.
7. From the **Color** list, choose **White**.
8. On the **Electric Field (emw)** toolbar, click **Plot**.
9. Click the **Zoom Extents** button on the **Graphics** toolbar.

   Compare the resulting plot with that shown in Figure 2.

To find out the integration contour orientations, plot the tangent vector, `t1`, along the boundaries as follows:

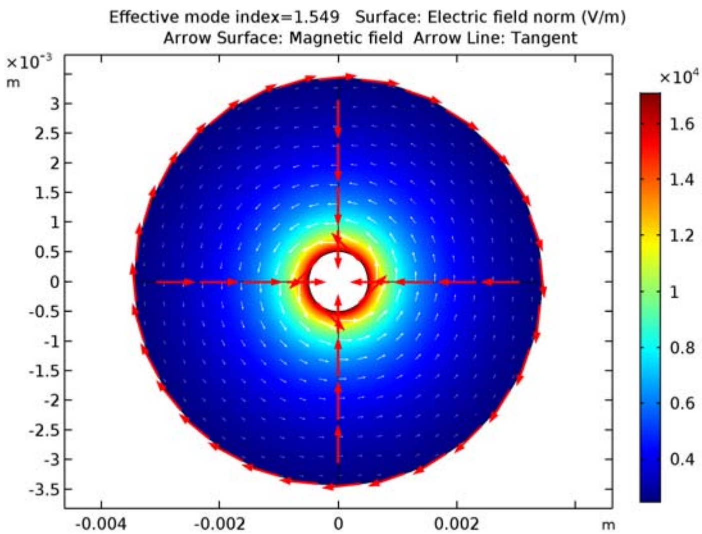
Arrow Line 1

1. Right-click **Electric Field (emw)** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Geometry>tx,ty - Tangent**.

3 Locate the **Coloring and Style** section. In the **Number of arrows** text field, type `50`.

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



A comparison with Equation 1 reveals that the line integral for the voltage computes the potential difference V_i - V_o. When computing the line integral for the current, the clockwise orientation of the integration contour would mean that a positive current is directed in the negative *z* direction, that is, into the modeling plane. The minus sign added in the definition of I reverses this direction.

5 Right-click **Results>Electric Field (emw)>Arrow Line 1** and choose **Disable**, to retrieve the result plot.

Finish by computing the characteristic impedance.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>Z0_model - Characteristic impedance**.

3 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result, roughly 74.65 Ω, is within 0.2% of the analytic value, 74.53 Ω.

Transient Modeling of a Coaxial Cable

Introduction

Time-domain simulations of Maxwell's equations are useful for

- observing transient phenomena,
- finding the time it takes for a signal to propagate, or
- modeling materials that are nonlinear with respect to the electric or magnetic field strength.

This example considers a pulse propagating down a coaxial transmission line for three different termination types: short, open, and matched. The signal propagation time is deduced from the reflected waves detected at the input port.

Model Definition

The model setup, schematically shown in Figure 1, is a short section of an air-filled coaxial transmission line. The symmetry of the structure allows for a 2D axisymmetric model geometry.

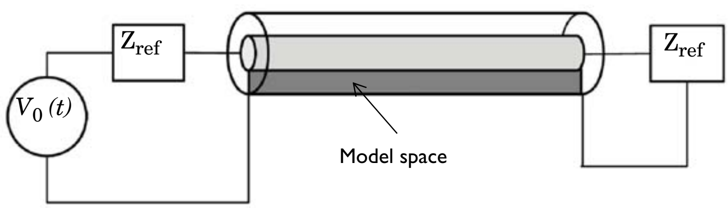


Figure 1: Schematic of a section of a coaxial transmission line connected to a transient voltage source and a load.

At one end of the coaxial cable, or coax for short, a *lumped port* boundary condition excites the structure; specify a transient excitation pulse, $V_0(t)$, by using a Gaussian pulse-windowed sine function. Apply the excitation as a current of magnitude $I(t) = V_0(t) / Z_{\mathrm{ref}}$ flowing tangentially to the excitation boundary. Here Z_{ref} refers to the specified characteristic impedance between the voltage generator and the model.

At the other end of the coax, consider, in turn, three different boundary conditions:

1. *perfect electric conductor* (PEC)—to simulate the short condition;
2. *perfect magnetic conductor* (PMC)—to simulate an open condition; and
3. *lumped port*—to simulate a matched load.

On the walls of the coax, apply a PEC boundary condition; this condition is appropriate when both skin depth and losses in the conductors are very small.

Use a triangular mesh with the maximum element size chosen such that there are at least two elements in the radial direction and at least eight elements per wavelength.

The only changes required to the default solver settings are to tighten the relative tolerance from the default value, and to adjust the timespan and output time steps. The internal time steps taken by the solver are auto-selected based on the specified relative tolerance.

Results and Discussion

Figure 2 shows the results of the transient simulation for the three different termination types. The figure plots the radial component of the electric field at the input port as a function of time for the three different termination conditions. The short (PEC) and open (PMC) terminations reflect waves that are 180° out of phase, and the matched load produces almost no reflections. From the reflected waves in the plot, you can read off an approximate signal propagation time through the air-filled transmission line of (0.37 – 0.10) / 2 ns = 0.135 ns. This matches the expected value of L_{coax} / c, where L_{coax} = 40 mm is the length of the line and c is the speed of light in air.

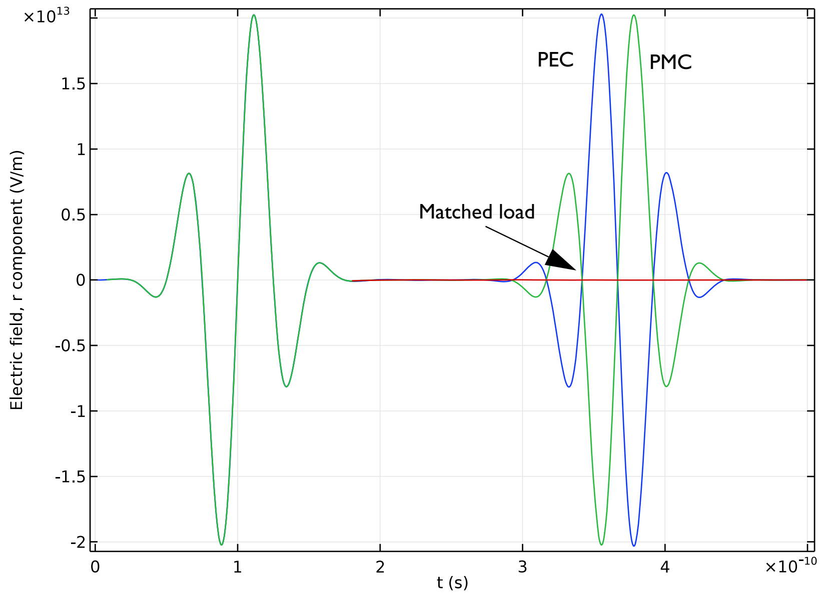


Figure 2: Radial component of electric field at the input port versus time for three different termination conditions: short (blue), open (green), and matched load (red).

Application Library path: `RF_Module/Verification_Examples/coaxial_cable_transient`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Transient (temw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`r_coax`	`1[mm]`	0.001 m	`Coax inner radius`
`R_coax`	`2[mm]`	0.002 m	`Coax outer radius`
`L_coax`	`40[mm]`	0.04 m	`Length of coax core into cavity`
`f`	`20[GHz]`	2E10 Hz	`Pulse frequency`
`L`	`c_const/f`	0.01499 m	`Wavelength, free space`
`T`	`1/f`	5E-11 s	`Period`
`h_max`	`min(L/8,(R_coax-r_coax)/2)`	5E-4 m	`Maximum element size`

Next, define the excitation, V0(t), in terms of a Gaussian pulse and a sine function.

Define a Gaussian pulse.

Gaussian Pulse 1 (gp1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Gaussian Pulse**.

2 In the **Settings** window for **Gaussian Pulse**, type `gauss_pulse` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type `2*T`.

4 In the **Standard deviation** text field, type `T/2`.

Now use this pulse in an analytic function for V0(t):

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type V0 in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type gauss_pulse(t)*sin(2*pi*f*t).

4 In the **Arguments** text field, type t.

5 Locate the **Units** section. In the **Arguments** text field, type s.

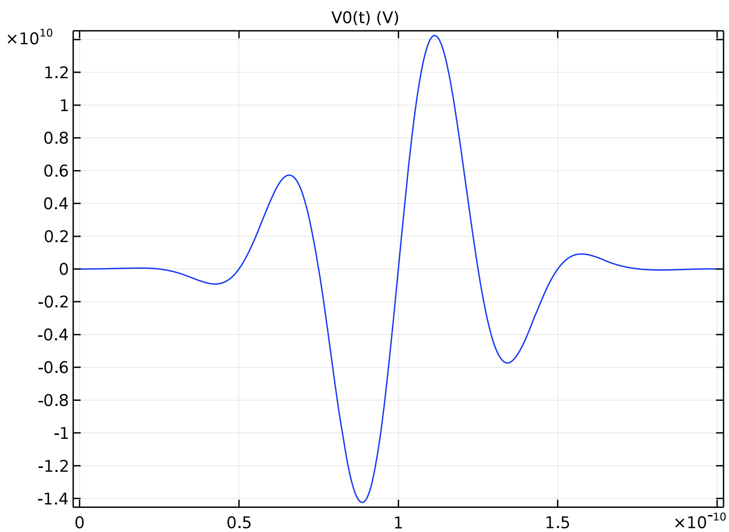
6 In the **Function** text field, type V.

To plot the function, you need to specify a suitable time interval.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
t	0	0.2[ns]

8 Click to collapse the **Plot parameters** section. Click **Plot**.



GEOMETRY 1

An elongated rectangle offset from the symmetry axis represents the straight coaxial cable.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type R_coax-r_coax.

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

5 Locate the **Position** section. In the **r** text field, type `r_coax`.

6 Click **Build All Objects**.

DEFINITIONS

Set up a point probe for plotting the electric field component Er while solving. You will also use this plot to reproduce Figure 2.

Domain Point Probe 1

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

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

3 In row **Coordinates**, set **r** to `r_coax`.

4 Select the **Snap to closest boundary** check box.

5 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

6 In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.

7 Click **temw.Er - Electric field, r component** in the upper-right corner of the section.

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Now set up the physics. Begin by defining the Lumped port input condition.

Lumped Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 2 only, (the bottom boundary).

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

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

4 In the V_0 text field, type `V0(t)`.

5 Locate the **Settings** section. In the Z_{ref} text field, type `(Z0_const/2/pi)*log(R_coax/r_coax)`.

The open case uses a PMC condition at the termination.

Perfect Magnetic Conductor 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Transient (temw)** and choose **Perfect Magnetic Conductor**.

2 Select Boundary 3 only, (the top boundary).

Finally, define a lumped port condition to use for the matched load case.

Lumped Port 2

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 3 only.

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

4 In the Z_{ref} text field, type `(Z0_const/2/pi)*log(R_coax/r_coax)`.

ADD MATERIAL

1 On 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.

MATERIALS

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

MESH 1

Size

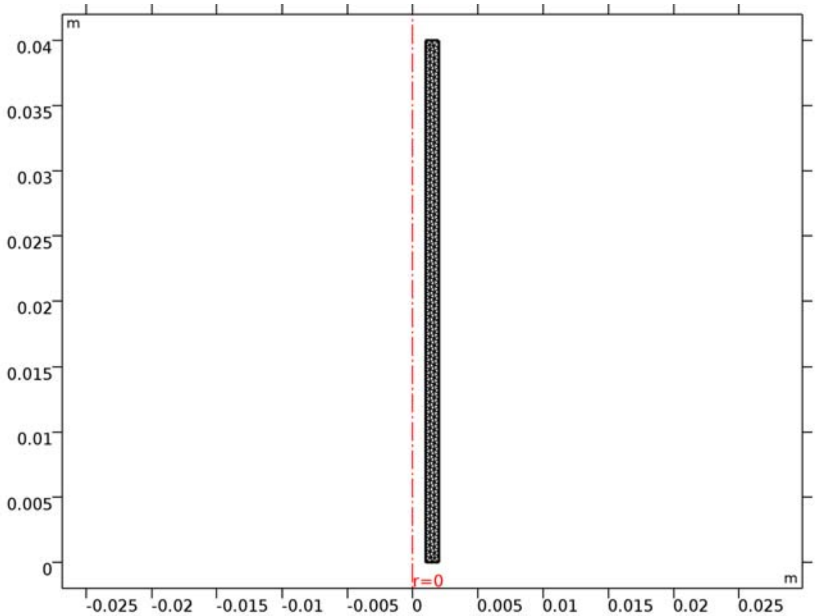
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max`.

5 Click **Build All**.



STUDY 1

Step 1: Time Dependent

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

2 In the **Times** text field, type `range(0,T/24,10*T)`.

3 From the **Tolerance** list, choose **User controlled**.

4 In the **Relative tolerance** text field, type `0.0001`.

To study the short termination case first, disable the PMC and lumped port conditions so that the default PEC condition is activated on the termination boundary.

5 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box, disable **Perfect Magnetic Conductor 1** and **Lumped Port 2**.

6 On the **Home** toolbar, click **Compute**.

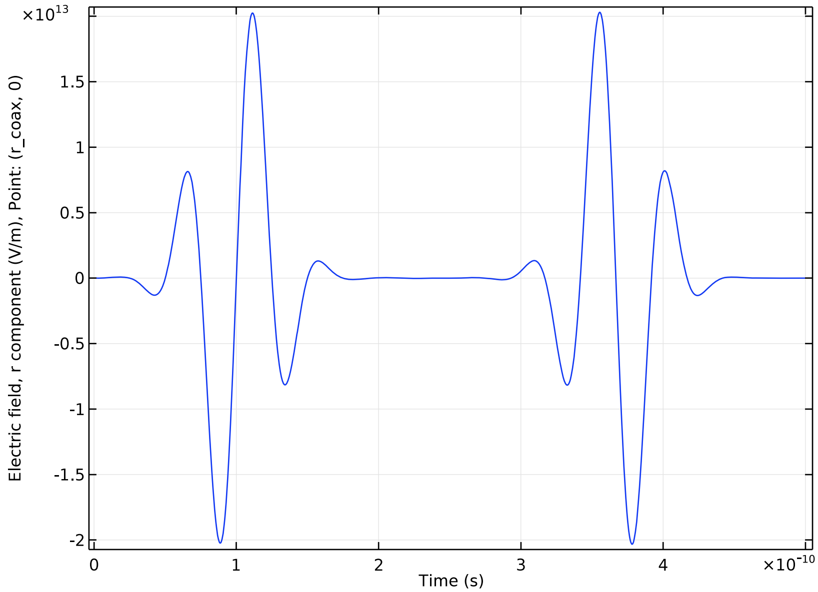
RESULTS

2D Plot Group 1

Click on the **Probe Plot 1** tab to place it in focus.

Probe Plot Group 2

When the solver finishes the plot should look like that in the figure below.



2D Plot Group 1

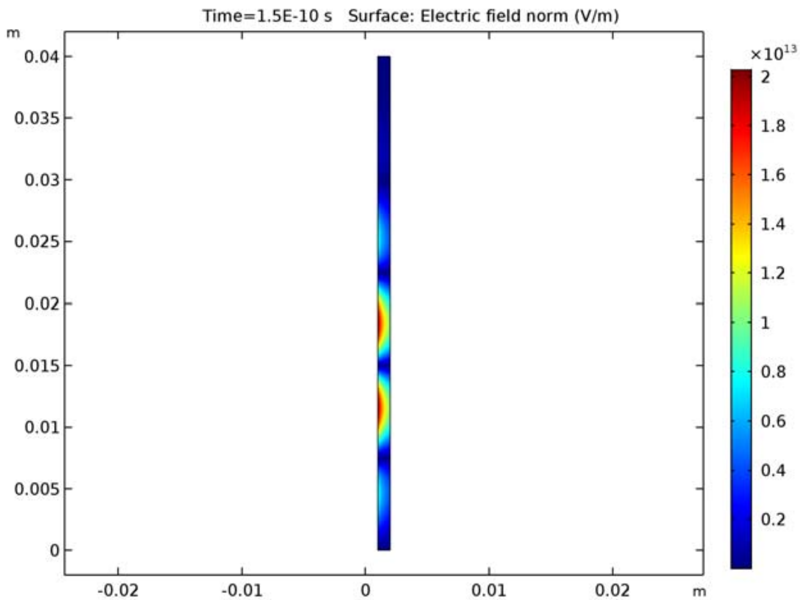
The default surface plot shows the electric field in the coax at the end of the simulation interval. Because the transient has died out, the solution you see is only noise. Modify the time to get a more interesting plot.

1 In the **Model Builder** window, under **Results** click **2D Plot Group 1**.

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

3 From the **Time (s)** list, choose **1.5E-10**.

4 On the **2D Plot Group 1** toolbar, click **Plot**.



Now turn to the open termination case.

DEFINITIONS

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions> Domain Point Probe 1** click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, click to expand the **Table and window settings** section.

3 Locate the **Table and Window Settings** section. From the **Output table** list, choose **New table**.

With these settings you get a plot for the short and open termination cases in the same plot window.

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Perfect Magnetic Conductor 1

In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)** right-click **Perfect Magnetic Conductor 1** and choose **Enable**.

STUDY 1

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

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

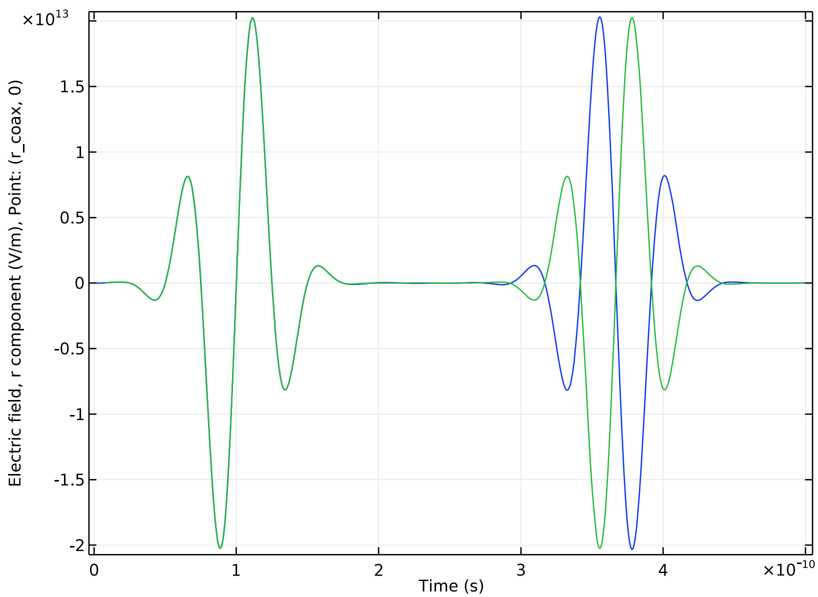
3 Clear the **Generate default plots** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Probe Plot Group 2

The reflected waves for the short and open terminations are 180 degrees out of phase.



Finally, activate the matched load case.

DEFINITIONS

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions> Domain Point Probe 1** click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, click to expand the **Table and window settings** section.

3 Locate the **Table and Window Settings** section. From the **Output table** list, choose **New table**.

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Lumped Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)** right-click **Lumped Port 2** and choose **Enable**.

   Note that you do not need to disable the PMC condition because it is overridden by the lumped port.

STUDY 1

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

RESULTS

Probe Plot Group 2

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **x-axis label** check box.
4 In the associated text field, type `t (s)`.
5 Select the **y-axis label** check box.
6 In the associated text field, type `Electric field, r component (V/m)`.
7 On the **Probe Plot Group 2** toolbar, click **Plot**.

   The plot should now look like that in Figure 2, with the red graph corresponding to the matched case.

Created in COMSOL Multiphysics 5.3

Conical Antenna

Introduction

Conical antennas are useful for many applications due to their broadband characteristics and relative simplicity. This example includes an analysis of the antenna impedance and the radiation pattern as functions of the frequency for a monoconical antenna with a finite ground plane and a 50 Ω coaxial feed. The rotational symmetry makes it possible to model this in axially symmetric 2D. When modeling in 2D, you can use a dense mesh, giving an excellent accuracy for a wide range of frequencies.

Model Definition

The antenna geometry consists of a 0.2 m tall metallic cone with a top angle of 90 degrees on a finite ground plane of a 0.282 m radius. The coaxial feed has a central conductor of 1.5 mm radius and an outer conductor (screen) of 4.916 mm radius separated by a Teflon dielectric of relative permittivity of 2.07. The central conductor of the coaxial cable is connected to the cone, and the screen is connected to the ground plane.

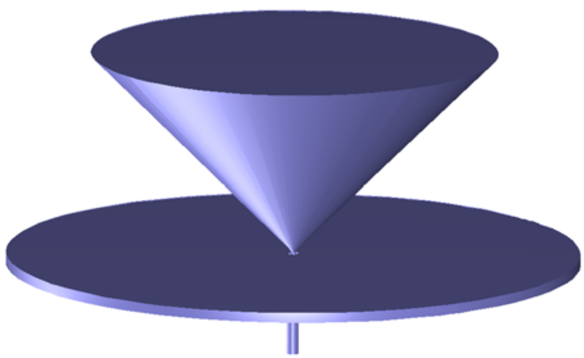


Figure 1: The geometry of the antenna. The central conductor of the coaxial cable is connected to the metallic cone, and the cable screen is connected to the finite ground plane.

The model takes advantage of the rotational symmetry of the problem, which allows modeling in 2D using cylindrical coordinates. You can then use a very fine mesh to achieve an excellent accuracy.

DOMAIN EQUATIONS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic (TEM) fields. Assuming time-harmonic fields with complex amplitudes containing the phase information, you have:

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$

$$\mathbf{H} = \mathbf{e}_\varphi \frac{C}{rZ} e^{j(\omega t - kz)}$$

where z is the direction of propagation and r, φ, and z are cylindrical coordinates centered on axis of the coaxial cable. Z is the wave impedance in the dielectric of the cable, and C is an arbitrary constant. The angular frequency is denoted by ω. The propagation constant, k, relates to the wavelength in the medium λ as

$$k = \frac{2\pi}{\lambda}$$

In the air, the electric field also has a finite axial component whereas the magnetic field is purely azimuthal. Thus it is possible to model the antenna using an axisymmetric transverse magnetic (TM) formulation, and the wave equation becomes scalar in H_φ:

$$\nabla\times\left(\frac{1}{\varepsilon}\nabla\times H_\varphi\right) - \mu\omega^2 H_\varphi = 0$$

BOUNDARY CONDITIONS

The boundary conditions for the metallic surfaces are:

$$\mathbf{n}\times\mathbf{E} = 0$$

At the feed point, a matched coaxial port boundary condition is used to make the boundary transparent to the wave. The antenna is radiating into free space, but you can only discretize a finite region. Therefore, truncate the geometry some distance from the antenna using a scattering boundary condition allowing for outgoing spherical waves to pass with very little reflections. A symmetry boundary condition is automatically applied on boundaries at $r = 0$.

Results and Discussion

Figure 2 shows the antenna impedance as a function of frequency. Ideally, the antenna impedance should be matched to the characteristic impedance of the feed, 50 Ω, to obtain maximum transmission into free space. This is quite well fulfilled in the high frequency range.

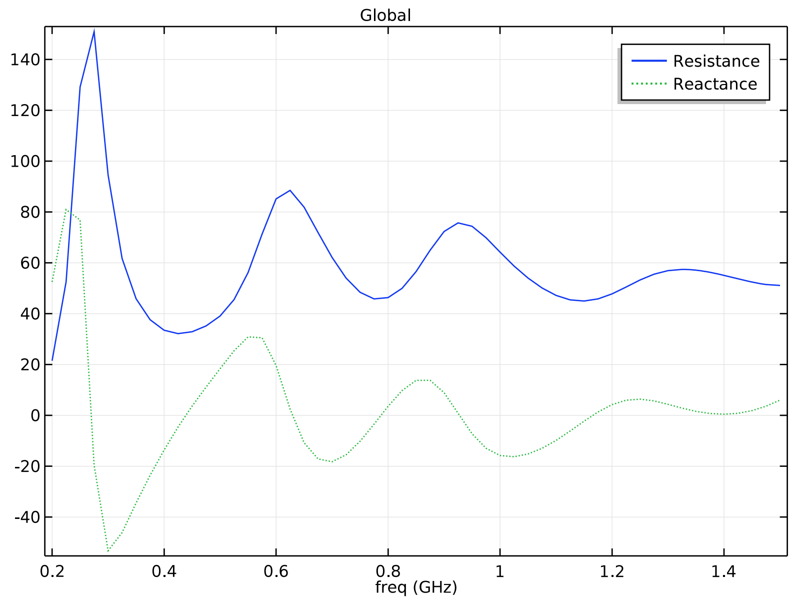


Figure 2: The antenna impedance in Ω as a function of frequency from 200 MHz to 1.5 GHz. The solid line shows the radiation resistance and the dotted line represents the reactance.

Figure 3 shows the antenna radiation pattern in the near-field for three different frequencies. The effect of the finite diameter of the ground plane is to lift the main lobe from the horizontal plane. For an infinite ground plane or in the high-frequency limit, the radiation pattern is symmetric around zero elevation. This is easy to understand, as an infinite ground plane can be replaced by a mirror image of the monocone below the plane. Such a biconical antenna is symmetric around zero elevation and has its main lobe in the horizontal direction. The decreased lobe lifting at higher frequencies is just about visible in Figure 3.

Figure 4 shows the antenna radiation pattern in the far field for the same frequencies as the radiation pattern at the boundary in Figure 3.

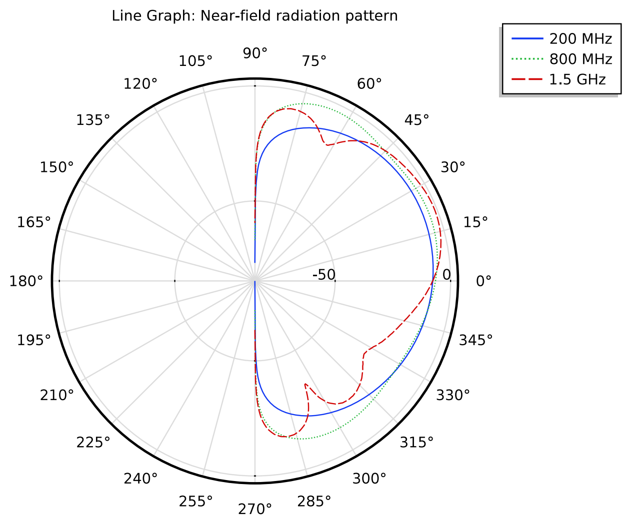


Figure 3: Polar plot of the antenna radiation pattern in the near field versus the elevation angle for 200 MHz, 800 MHz, and 1.5 GHz. The scale is logarithmic.

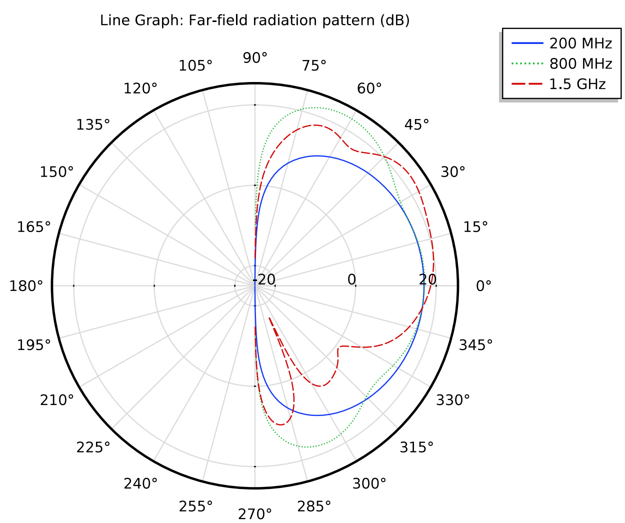


Figure 4: Polar plot of the antenna radiation pattern for the far field versus the elevation angle for 200 MHz, 800 MHz, and 1.5 GHz. This plot is normalized differently but has a shape resembling the near field.

As the frequency increases the antenna impedance gets closer to 50 Ω, which means that a voltage generator connected to the input of the antenna should have an output impedance of 50 Ω.

Application Library path: `RF_Module/Antennas/conical_antenna`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

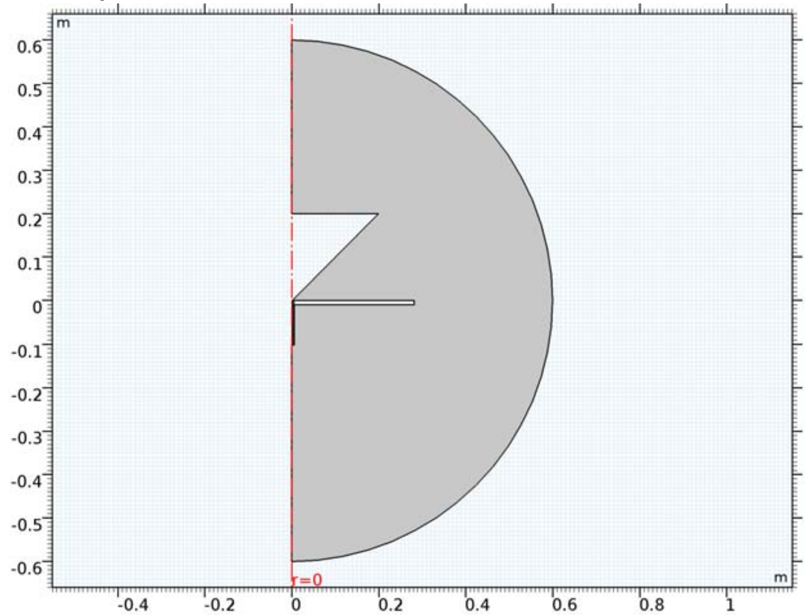
1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `conical_antenna.mphbin`.

5 Click **Import**.



The imported geometry is effectively a half circle with the metal areas removed. You model the electromagnetic waves in the air and the dielectric material inside the coaxial cable. There is no need to include the metal as a domain in the model because the fields in it are essentially zero except on its surface.

GLOBAL DEFINITIONS

Prepare for the impedance computation by making a few definitions.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`Z_tl`	`50[ohm]`	50 Ω	`Characteristic transmission line impedance`

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`Z`	`Z_tl*(1+emw.S11)/(1-emw.S11)`		`Antenna impedance`

`emw.S11` is the name of the automatically computed reflection S-parameter.

Define the following selections in order to get easy access to some frequently used domains and boundaries.

Explicit 1

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

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

3 Select Domain 1 only.

Explicit 2

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

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

The dielectric domain is inside the coaxial cable just below the cone. It is easier to select it if you zoom in a little.

3 Select Domain 2 only.

Explicit 3

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

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

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

4 Select Boundaries 14 and 15 only.

With all selections and expressions now defined, it is time to set up the materials and the physics of the model.

ADD MATERIAL

1 On 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.

MATERIALS

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

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

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

Material 2 (mat2)

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 **Geometric Entity Selection** section.

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

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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	2.07	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Port 1

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

Set up Boundary 6, at the bottom of the coaxial cable, to be a Port. You can zoom in on this part of the geometry to easier find and select this boundary.

2 Select Boundary 6 only.

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

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

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

Scattering Boundary Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Scattering Boundary Condition**.

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

3 From the **Selection** list, choose **Outer Air Boundaries**.

The Scattering boundary condition is a simple way of letting the waves undergo only minor artificial reflections as they leave the computational domain through the exterior boundaries. To minimize these reflections, but at a greater computational cost, you can use Perfectly Matched Layers).

Perfect Electric Conductor 1

As you can see if you click the Perfect Electric Conductor node under Electromagnetic Waves, the physical boundaries to which you have not assigned any boundary condition will by default be considered perfect electric conductors. This is a good approximation for most metals throughout the frequency range considered in this model.

Far-Field Domain 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

2 Select Domain 1 only.

Far-Field Calculation 1

1 In the **Model Builder** window, expand the **Far-Field Domain 1** node, then click **Far-Field Calculation 1**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Boundary Selection** section.

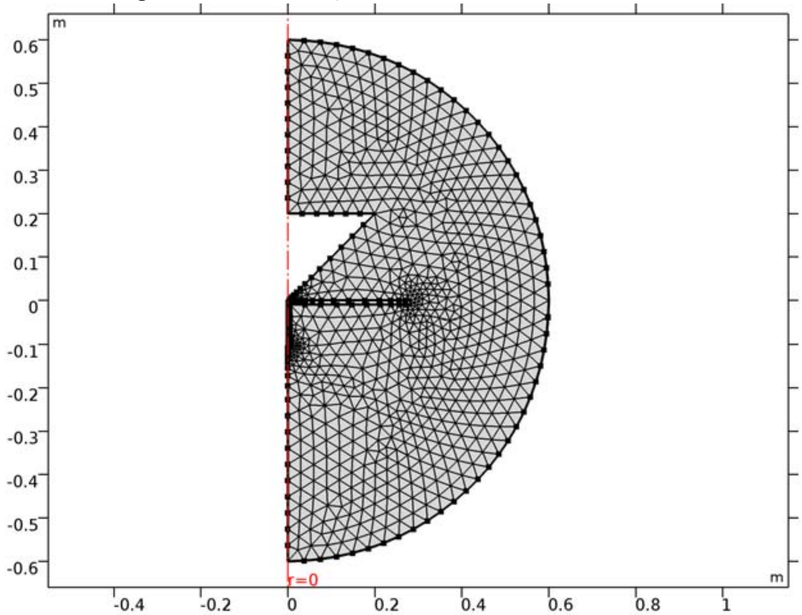
3 From the **Selection** list, choose **Outer Air Boundaries**.

Adding a Far Field Calculation feature does not affect the physics of the model, but makes it possible to study the far field generated by the antenna. Select the boundaries to use for this computation so that, in the physical (3D) geometry, they surround all sources and reflecting objects. The outer air boundaries are a convenient choice.

MESH 1

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

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



STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `range(200[MHz],25[MHz],1.5[GHz])`.

The frequency range you just entered runs from 200 MHz to 1.5 GHz in steps of 25 MHz.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The plot that appears once the solution process is finished shows the norm of the electric field at 1.5 GHz. The reason it is mostly dark blue is because the range is dominated by the high values in and near the coaxial cable. To better see how the wave propagates, try plotting the instantaneous value of the H-field using a manual range.

Surface

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>emw.Hphi - Magnetic field, phi component**.

3 Click to expand the **Range** section. Select the **Manual color range** check box.

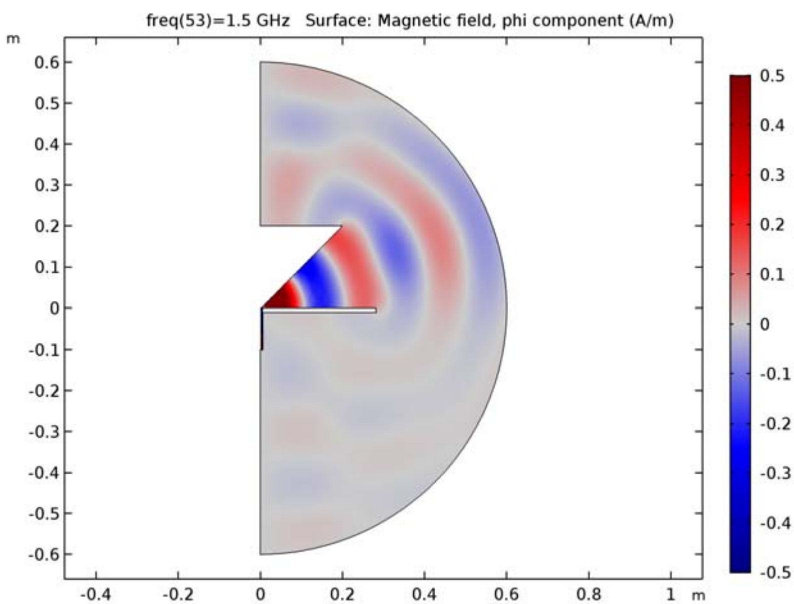
4 In the **Minimum** text field, type `-0.5`.

5 In the **Maximum** text field, type `0.5`.

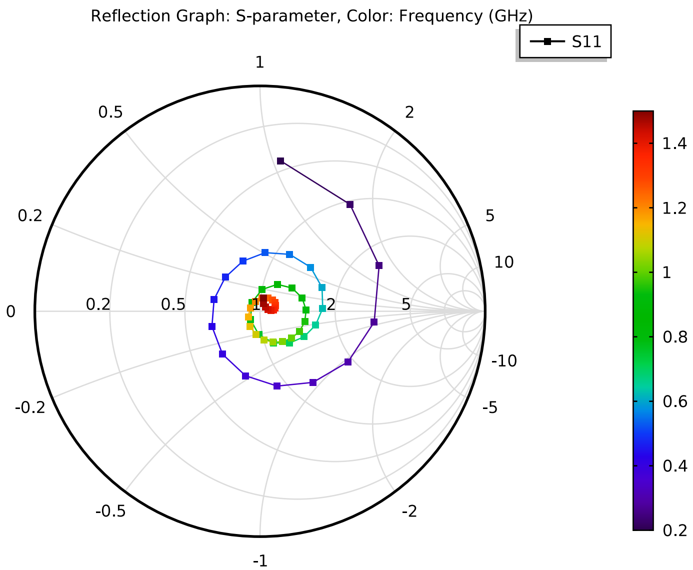
6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

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

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



Smith Plot (emw)



Far Field 1

1 In the **Model Builder** window, expand the **2D Far Field (emw)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `100`.

4 Find the **Reference direction** subsection. In the **x** text field, type `-1`.

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

6 On the **2D Far Field (emw)** toolbar, click **Plot**.

To plot the impedance as a function of the frequency, set up a 1D plot.

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Global**.

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

4 In the table, enter the following settings:

Expression	**Unit**	**Description**
`real(Z)`	Ω	`Resistance`
`imag(Z)`	Ω	`Reactance`

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

6 On the **1D Plot Group 6** toolbar, click **Plot**.

You have now reproduced Figure 2. Next, visualize the near-field and far-field radiation patterns using polar plots.

Polar Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **Polar Plot Group**.

Select a few of the frequencies from the list of parameter values. Showing the radiation pattern for all of them would take a bit of time and lead to a cluttered plot.

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

3 From the **Parameter selection (freq)** list, choose **From list**.

4 In the **Parameter values (freq (GHz))** list, choose **0.2**, **0.8**, and **1.5**.

Use Ctrl-key to select multiple frequencies.

Line Graph 1

1 Right-click **Polar Plot Group 7** and choose **Line Graph**.

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

3 From the **Selection** list, choose **Outer Air Boundaries**.

4 Click **Replace Expression** in the upper-right corner of the **r-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain> Energy and power>emw.nPoav - Power outflow, time average**.

5 Locate the **r-Axis Data** section. In the **Expression** text field, type `10*log10(emw.nPoav)`.

The variable `emw.nPoav` represents the outgoing power flow through the boundaries where it is evaluated. The expression you just entered gives you the same in a logarithmic scale.

6 Select the **Description** check box.

7 In the associated text field, type `Near-field radiation pattern`.

8 Locate the θ **Angle Data** section. From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `atan2(z,r)`.

10 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 Click to collapse the **Coloring and style** section. Click to expand the **Legends** section. Select the **Show legends** check box.

12 From the **Legends** list, choose **Manual**.

13 In the table, enter the following settings:

Legends
200 MHz
800 MHz
1.5 GHz

14 Click to collapse the **Legends** section. On the **Polar Plot Group 7** toolbar, click **Plot**.

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

Your near-field radiation plot should look like that in Figure 3.

Finally, visualize the far-field radiation pattern.

Line Graph 1

1 In the **Model Builder** window, under **Results** right-click **Polar Plot Group 7** and choose **Duplicate**.

2 In the **Model Builder** window, expand the **Polar Plot Group 8** node, then click **Line Graph 1**.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **r-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Far field>emw.normdBEfar - Far-field norm, dB**.

4 Locate the **r-Axis Data** section. In the **Description** text field, type `Far-field radiation pattern`.

5 On the **Polar Plot Group 8** toolbar, click **Plot**.

The plot should look like that in Figure 4.

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.

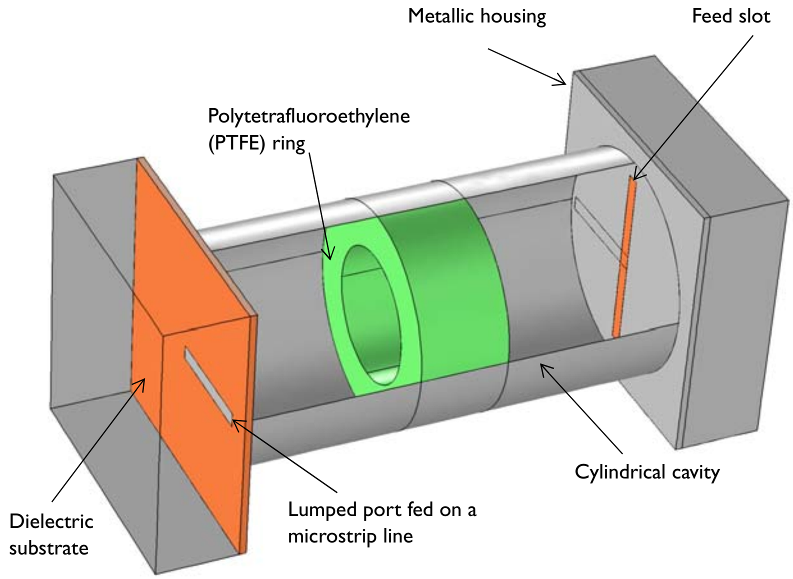


Figure 1: An evanescent mode cavity filter. The signal fed from a microstrip line is slot coupled into the cylindrical cavity loaded with a PTFE ring.

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.

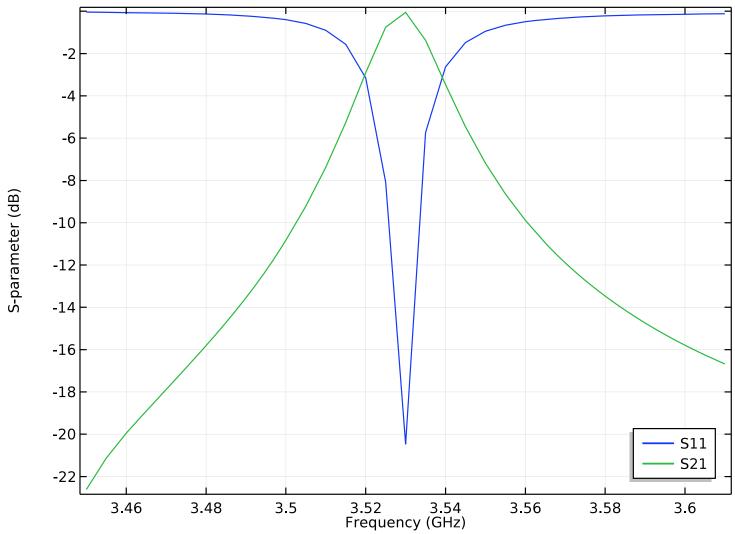


Figure 2: The frequency response of the filter shows bandpass filter characteristics. The center frequency is lower than the dominant mode resonant frequency of the metallic cavity.

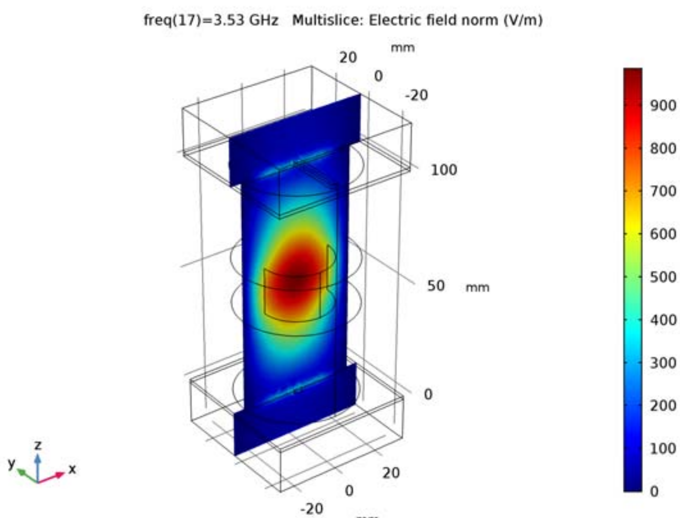


Figure 3: The dielectric tube inside the cavity does not distort the electric field distribution at resonance significantly.

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

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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.

Cylinder 1 (cyl1)

1 On 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 Right-click **Cavity** and choose **Build Selected**.

Create a coupling slot.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **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 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

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

Create a substrate.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **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 Right-click **Bottom_plate** and choose **Build Selected**.

Create a 50Ω microstrip line.

Block 2 (blk2)

1 On 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 Right-click **Bottom_feed** and choose **Build Selected**.

Create a metallic housing.

Block 3 (blk3)

1 On 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`.

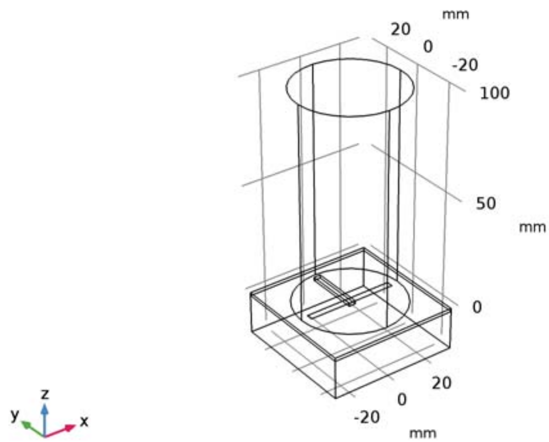
6 Locate the **Position** section. From the **Base** list, choose **Center**.

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

8 Right-click **Housing** and choose **Build Selected**.

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

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



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

Rotate 1 (rot1)

1. On 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 Angle** section.
4. In the **Rotation** text field, type `0,180`.
5. Locate the **Point on Axis of Rotation** section. In the **z** text field, type `50`.
6. Locate the **Axis of 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. Right-click **Rotate 1 (rot1)** and choose **Build Selected**.
10. Click the **Zoom Extents** button on the **Graphics** toolbar.

Create a dielectric ring.

Cylinder 2 (cyl2)

1. On 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 On 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 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

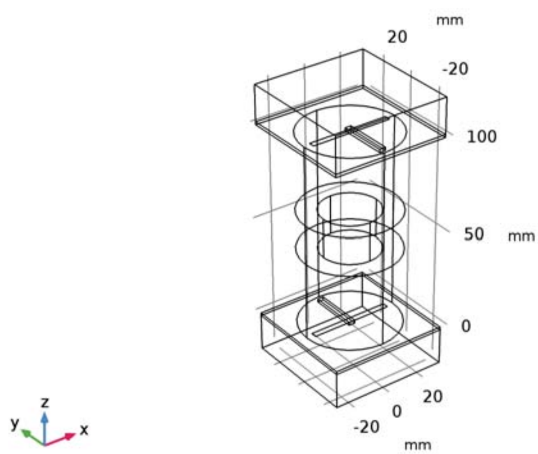
2 Select the object **cyl2** only.

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

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

5 Select the object **cyl3** only.

6 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

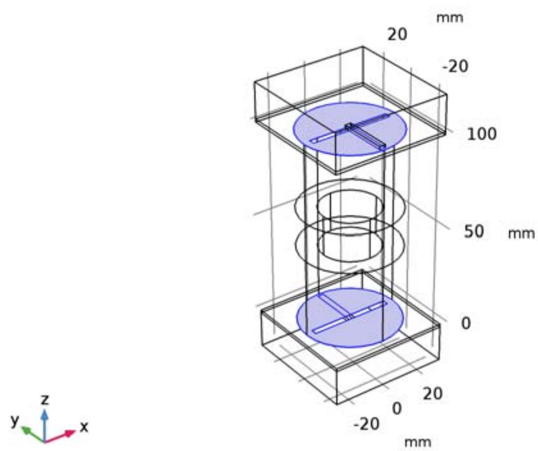
The default boundary condition is perfect electric conductor, which applies to all exterior boundaries. Assign a perfect electric conductor condition to the remaining boundaries of the cavity.

Perfect Electric Conductor 2

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

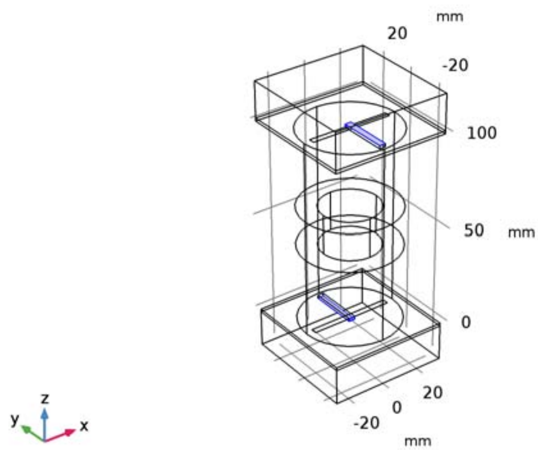


Proceed to define boundary condition for the shorted microstrip lines.

Perfect Electric Conductor 3

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

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



Lumped Port 1

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

2 Select Boundary 44 only.

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

Lumped Port 2

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

2 Select Boundary 34 only.

ADD MATERIAL

1 On 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.

MATERIALS

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Create a substrate material.

Material 2 (mat2)

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	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

Create a dielectric ring material.

Material 3 (mat3)

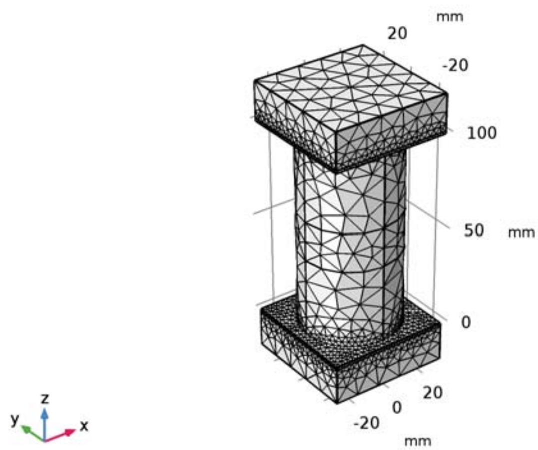
1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `PTFE` in the **Label** text field.
3 Select Domain 6 only.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	2.1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

MESH 1

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

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



STUDY 1

On 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 **Model Builder** window, under **Results** click **Electric Field (emw)**.

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

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

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

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

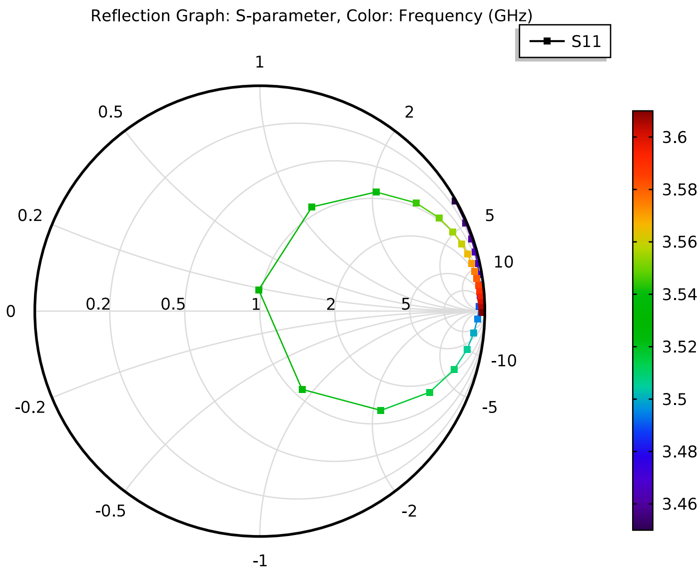
S-Parameter (emw)

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

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

Smith Plot (emw)



Analyze the same model with a much finer frequency resolution using asymptotic waveform evaluation (AWE). When a device presents a bandpass 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.

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, click **OK**.

Lumped Port 2

1. In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** 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, click **OK**.

ROOT

On the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

1. Go to the **Add Study** window.
2. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.
3. Click **Add Study** in the window toolbar.
4. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 2** 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],(3.61[GHz]-3.45[GHz])/32/50,3.6[GHz])`.

   Use a 50 times finer frequency resolution.
4. Click to expand the **Values of dependent variables** section. Locate 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 **Explicit 1** and **Explicit 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 fields in output** settings, it is possible to reduce the size of a model file a lot.
8. In the **Settings** window for **Frequency Domain**, click to expand the **Study extensions** section.

9 Locate the **Study Extensions** section. Select the **Use asymptotic waveform evaluation** check box.

10 In the **AWE expressions** text field, type `abs(comp1.emw.S21)`.

For two port bandpass-type devices, use `abs(emw.S21)` for AWE expression.

11 On the **Home** toolbar, click **Compute**.

RESULTS

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	dB	S-parameter, dB, 11 AWE
emw.S21dB	dB	S-parameter, dB, 21 AWE

Global 2

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	dB	S-parameter, dB, 11 regular sweep
emw.S21dB	dB	S-parameter, dB, 21 regular sweep

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

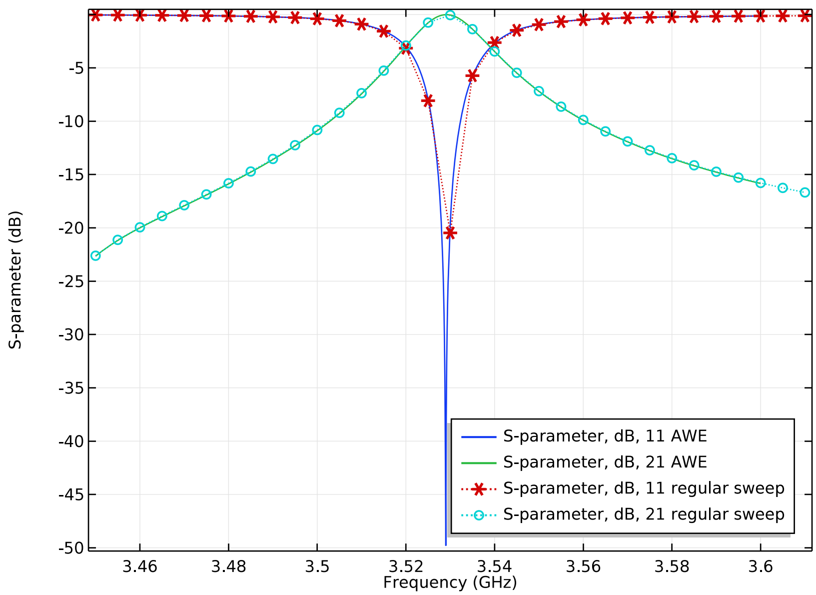
5 From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.

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

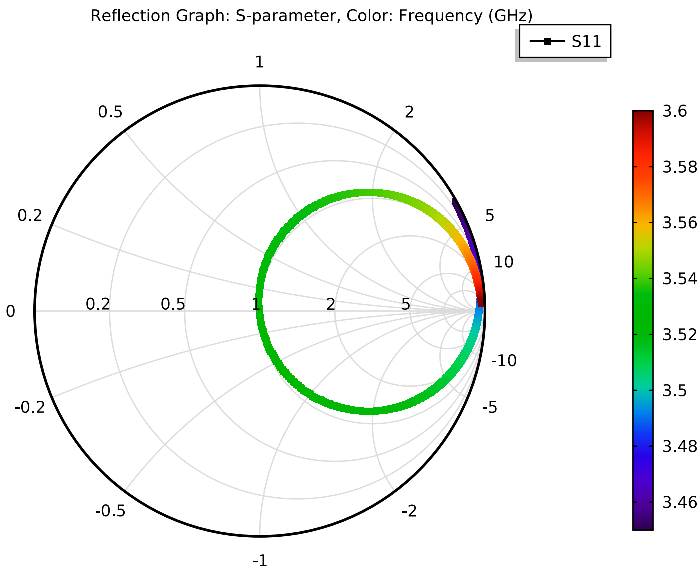
7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

9 On the **S-Parameter (emw) 1** toolbar, click **Plot**.



Smith Plot (emw) 1



Dipole Antenna

Introduction

The dipole antenna is one of the most straightforward antenna configurations. It can be realized with two thin metallic rods that have a sinusoidal voltage difference applied between them. The length of the rods is chosen such that they are quarter wavelength elements at the operating frequency. Such an antenna has a well-known torus-like radiation pattern.

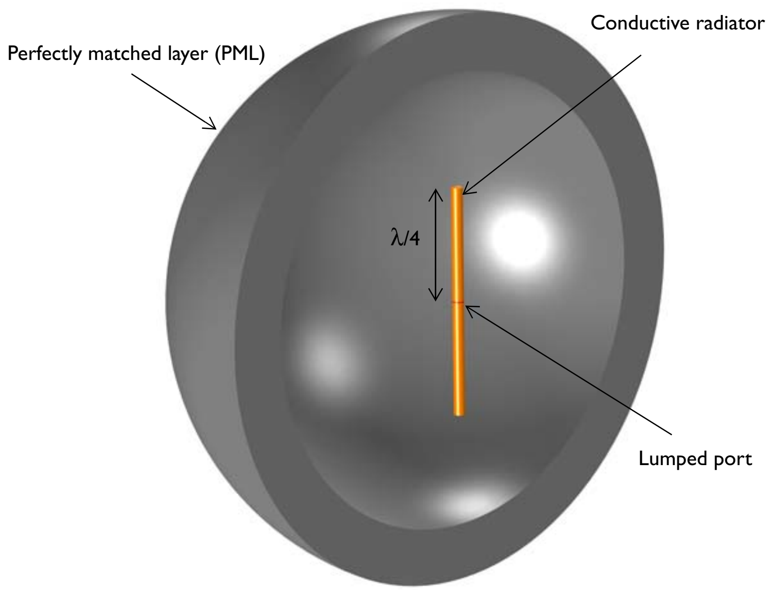


Figure 1: A dipole antenna. The model consists of two cylindrical arms of conductive material with a voltage source in between. A region of free space bounded by a perfectly matched layer (PML) surrounds the antenna.

Model Definition

The model of the antenna consists of two cylinders representing each of the dipole arms. The free space wavelength at the antenna's operating frequency is 4 m. Thus, each of the antenna arms is 1 m long and aligned with the z-axis. The arm radius is chosen to be 0.05 m. In the limit as the radius approaches zero, this antenna approaches the analytic solution.

A small cylindrical gap of size 0.01 m between the antenna arms represents the voltage source. The power supply and feed structure are not modeled explicitly, and it is assumed that a uniform voltage difference is applied across these faces. This source induces electromagnetic fields and surface currents on the adjacent conductive faces.

The dipole arm surfaces are modeled using the Impedance Boundary Condition, which is appropriate for conductive surfaces that have dimensions much larger than the skin depth. This boundary condition introduces a finite conductivity at the surface as well as resistive losses.

The air domain around the antenna is modeled as sphere of free space of radius 2 m, which is approximately the boundary between the near-field and the far-field. This sphere of air is truncated with a perfectly matched layer (PML) that acts as an absorber of outgoing radiation. The far-field pattern is computed on the boundary between the air and the PML domains.

The mesh is manually adjusted such that there are five elements per free space wavelength and that the boundaries of the antenna are meshed more finely. The PML is swept with a total of five elements along the radial direction.

Results and Discussion

The magnitude of the electric field around the antenna is shown in Figure 2. The fields appear artificially high near the excitation, as well as at the ends of the arms. These peaks in the intensity are due to local singularities; the fields at sharp transitions in the model are locally artificially high, but they do not affect the results some distance (1~2 elements) away from these regions.

The polar plot in Figure 3 of the far-field pattern in the xy-plane shows the expected isotropic radiation pattern. The 3D visualization of the far-field intensity in Figure 4 shows the expected torus-shaped pattern.

The impedance as seen by the port is evaluated to be 121 + 28i Ω, which agrees reasonably with expectations. In the limit as the antenna radius and gap height go to zero and in the limit of mesh refinement, the model approaches the analytic solution for a dipole antenna.

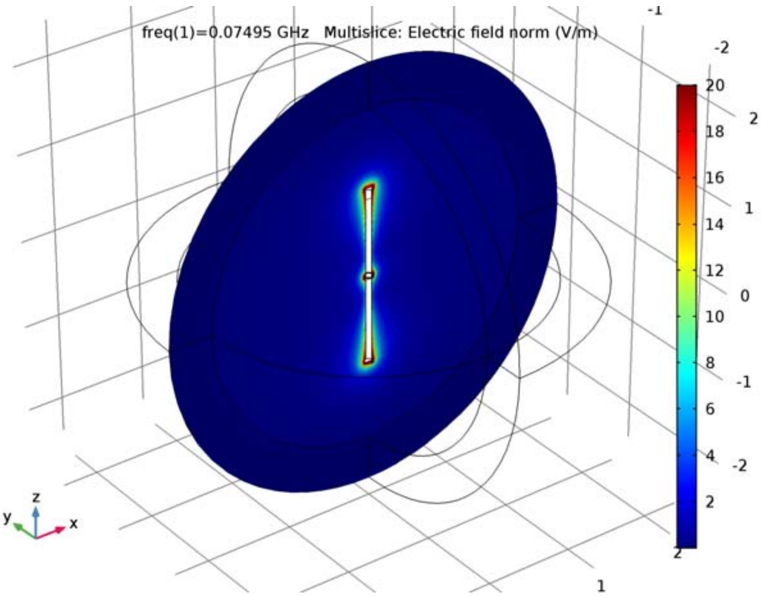


Figure 2: A slice plot of the electric field magnitude around the antenna.

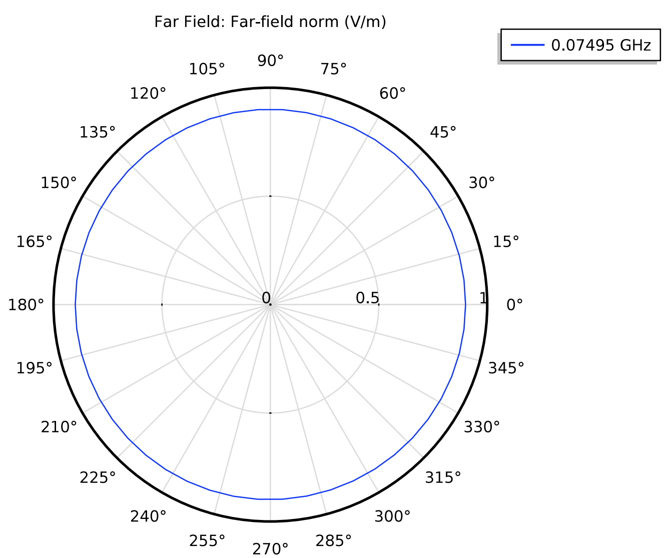


Figure 3: The polar plot of the far field pattern in the xy-plane is isotropic.

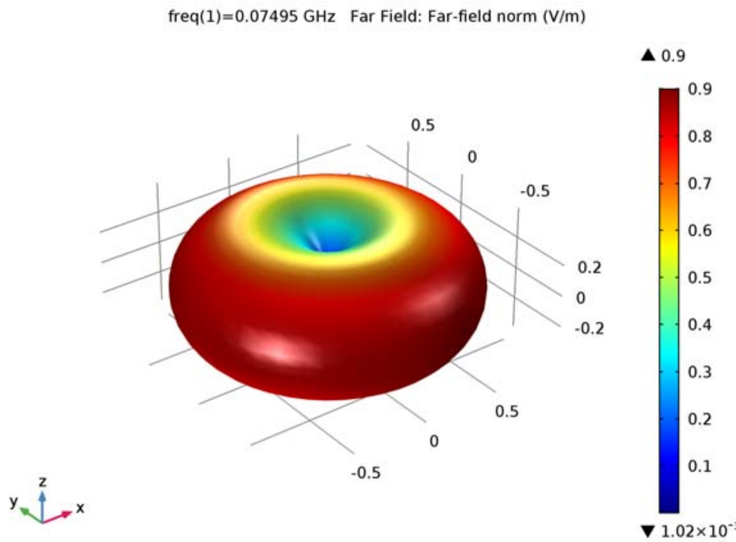


Figure 4: A 3D visualization of the far-field pattern of the dipole shows the expected torus-shaped pattern.

Application Library path: `RF_Module/Antennas/dipole_antenna`

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 **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
lambda0	4[m]	4 m	Operating wavelength
arm_length	lambda0/4	1 m	Dipole antenna arm length
r_antenna	arm_length/20	0.05 m	Dipole antenna arm radius
gap_size	arm_length/100	0.01 m	Gap between arms

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 In the **Frequencies** text field, type `c_const/lambda0`.

GEOMETRY 1

Create a sphere with a layer. The outer layer presents the PML.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `2.4*arm_length`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.5*arm_length

5 Right-click **Sphere 1 (sph1)** and choose **Build Selected**.

Choose wireframe rendering to get a better view of the interior parts.

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

Then, add a cylinder with layers. The top and bottom parts are the antenna radiators. A small gap between the antenna radiators is for the voltage source.

Cylinder 1 (cyl1)

1 On 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 `r_antenna`.
4 In the **Height** text field, type `2*arm_length+gap_size`.
5 Locate the **Position** section. In the **z** text field, type `-(arm_length+gap_size/2)`.
6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	arm_length

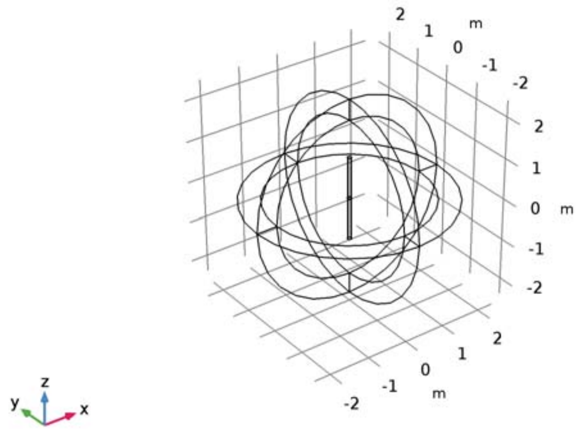
7 Clear the **Layers on side** check box.
8 Select the **Layers on bottom** check box.
9 Select the **Layers on top** check box.
10 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

The domain inside the antenna radiators is not part of the model analysis.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **sph1** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5 Select the object **cyl1** only.

6 Click **Build All Objects**.



DEFINITIONS

Create a set of selections to be used when setting up the physics. First, create a selection for the antenna radiator surface.

Explicit 1

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

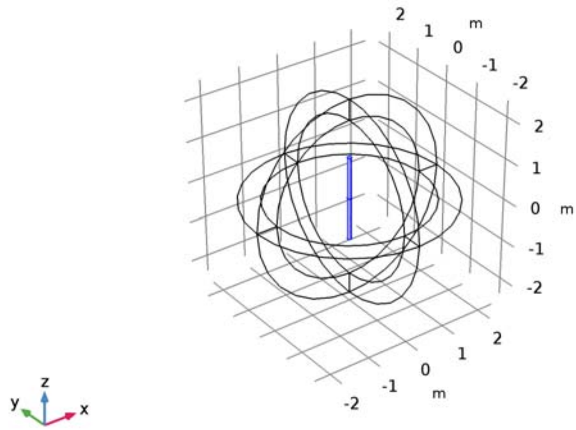
2 In the **Settings** window for **Explicit**, type `Antenna` in the **Label** text field.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `13-15, 18-20, 28, 30, 39, 41` in the **Selection** text field.

6 Click **OK**.

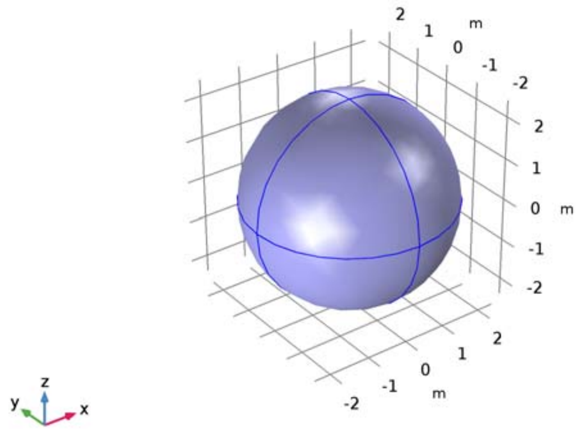


Add a perfectly matched layer on the outermost domain of the sphere.

Perfectly Matched Layer 1 (pml1)

1. On the **Definitions** toolbar, click **Perfectly Matched Layer**.
2. Select Domains 1–4 and 6–9 only.
3. In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.



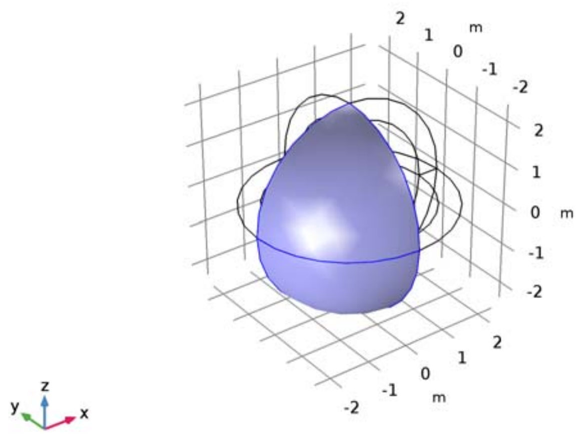
View 1

Suppress some domains and boundaries. This helps to see the interior parts when setting up the physics and reviewing the mesh.

Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **View 1** and choose **Hide for Physics**.

2 Select Domains 1 and 2 only.



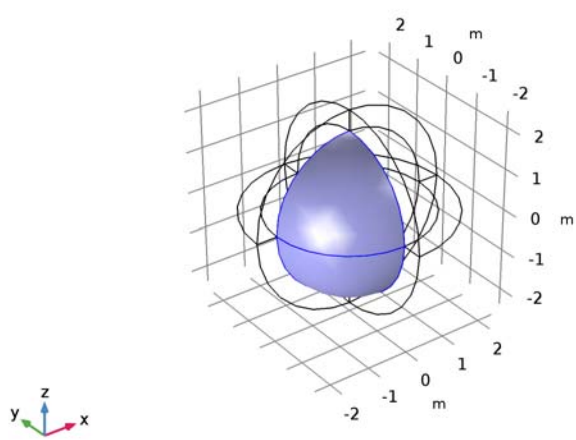
Hide for Physics 2

1 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 9 and 10 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Set up the physics for the model. Add an Impedance Boundary Condition that overrides the default PEC boundary condition on the antenna radiator surface.

Impedance Boundary Condition 1

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

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

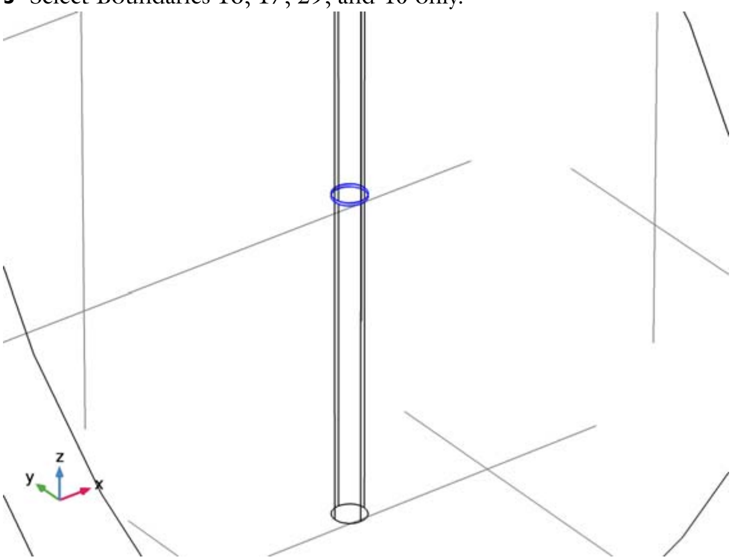
3 From the **Selection** list, choose **Antenna**.

Lumped Port 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Click the **Zoom In** button on the **Graphics** toolbar, a couple of times to see the small gap between antenna radiators clearly.

3 Select Boundaries 16, 17, 29, and 40 only.



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

5 From the **Type of lumped port** list, choose **User defined**.

6 In the h_{port} text field, type `gap_size`.

7 In the w_{port} text field, type `2*pi*r_antenna`.

8 Specify the $\mathbf{a}_\mathrm{h}$ vector as

0	x
0	y
1	z

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

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

10 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

MATERIALS

Assign air as the material for all domains and override the antenna radiator surface with copper.

ADD MATERIAL

1 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Copper**.

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

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

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

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

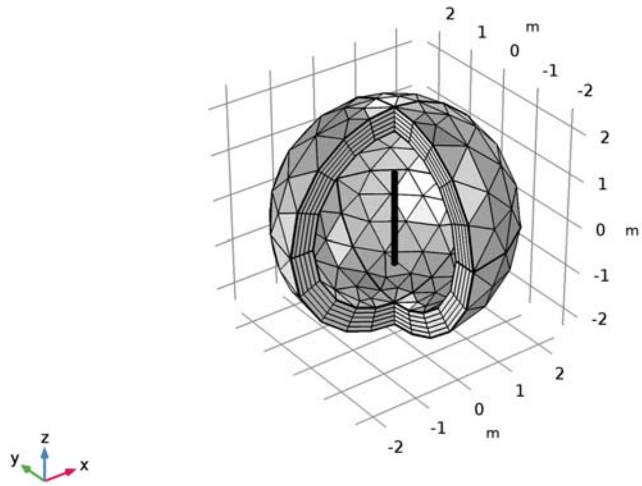
4 From the **Selection** list, choose **Antenna**.

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

MESH 1

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

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



STUDY 1

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

RESULTS

Electric Field (emw)

The default plot shows the E-field norm, 2D far-field polar plot, and 3D far-field radiation pattern.

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`.

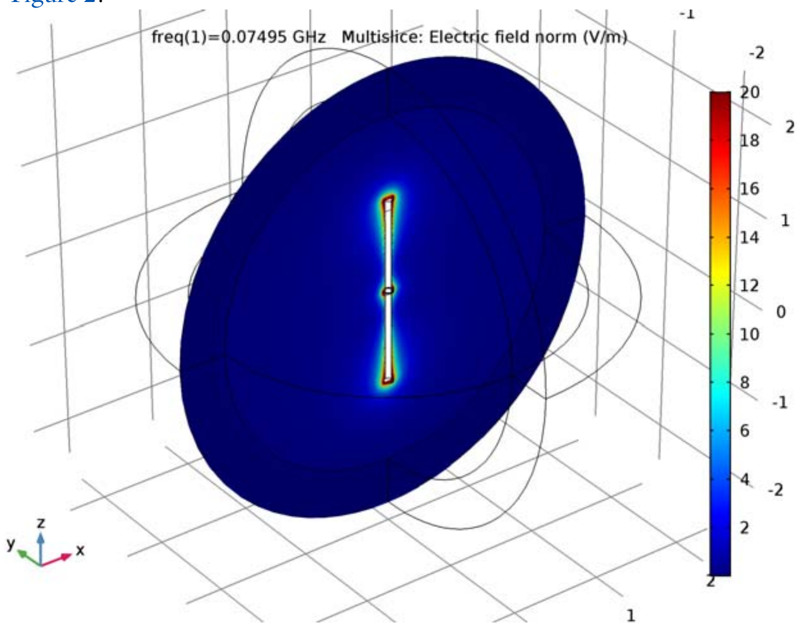
5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `20`.

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

8 Click the **Zoom In** button on the **Graphics** toolbar.

The results show the E-field norm distribution on the antenna radiators. It is plotted in Figure 2.



2D Far Field (emw)

Adjust the axis range.

1 In the **Model Builder** window, under **Results** click **2D Far Field (emw)**.

2 In the **Settings** window for **Polar Plot Group**, click to expand the **Axis** section.

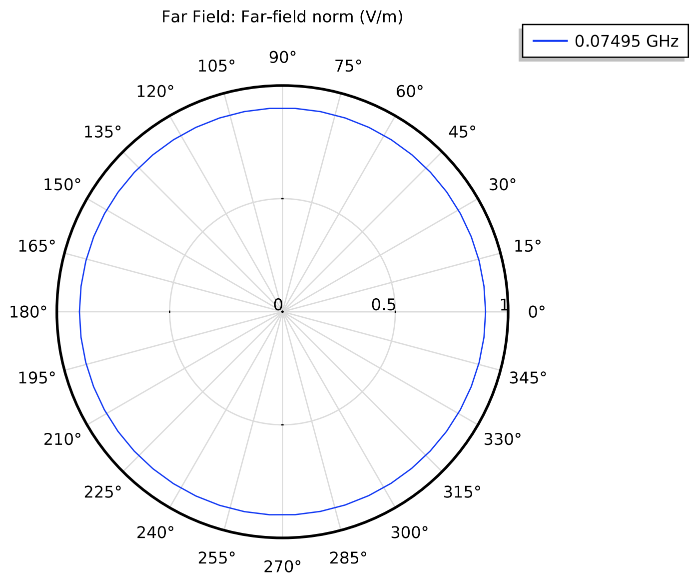
3 Select the **Manual axis limits** check box.

4 In the **r minimum** text field, type `0`.

5 In the **r maximum** text field, type `1`.

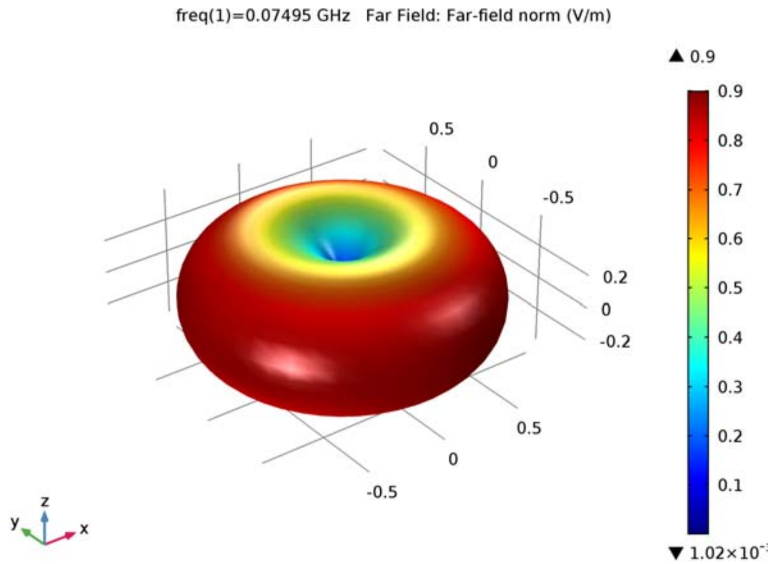
6 On the **2D Far Field (emw)** toolbar, click **Plot**.

The plotted H-plane pattern is omni-directional (isotropic) on the *xy*-plane as shown in Figure 3. The E- and H-plane of a linearly polarized antenna are defined by the antenna main polarization. The E-plane includes the main polarization that is E_z in this model while the H-plane is perpendicular to the main polarization.



3D Far Field (emw)

Compare the reproduced plot with Figure 4.



Derived Values

Finish the result analysis by evaluating the port impedance.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Ports>emw.Zport_1 - Lumped port impedance**.

3 Click **Evaluate**.

Created in COMSOL Multiphysics 5.3

Time-Domain Modeling of Dispersive Drude-Lorentz Media

Introduction

Plasmonic hole arrays have attracted a lot of scientific interest, since the discovery of extraordinary transmission through sub-wavelength hole arrays (c.f. Ref. 1). The classical Bethe theory predicts that transmittance through a sub-wavelength circular hole of diameter d in a PEC screen scales as $(d/\lambda)^4$, where λ is the wavelength. Yet, transmission through holes in realistic metallic films can exceed 50% and even approach 100%. This phenomenon was attributed to surface plasmon polaritons that can tunnel electromagnetic energy through the hole even if it is very much smaller than the wavelength.

This particular model is intended as a tutorial that shows how to model the full time-dependent wave equation in dispersive media, such as plasmas and semiconductors (and any linear medium describable by a sum of Drude-Lorentz resonant terms). The dispersion of the medium in the frequency domain is assumed to be of the form

$$\varepsilon_r(\omega) = \varepsilon_\infty - \frac{\omega_p^2}{\omega^2 - j\Gamma_i\omega - \omega_i^2}, \tag{1}$$

where the constant $\varepsilon_\infty > 1$ absorbs contributions from high-frequency contributions that are not modeled explicitly, ω_p is the plasma frequency, Γ_i is a damping coefficient, and ω_i is a resonance frequency, The particular case when the resonance frequency ω_i is zero is known as plasma (or Drude medium), and it covers most metals in the optical frequency range, from mid-IR to visible. For lossless plasmas, when the damping coefficient also is zero ($\omega_i = \Gamma_i = 0$), modeling simplifies significantly since then the polarization density is linearly related to the magnetic vector potential.

In this model, the wave equation for the magnetic vector potential

$$\nabla \times \mu_r^{-1}(\nabla \times \mathbf{A}) + \mu_0\sigma\frac{\partial \mathbf{A}}{\partial t} = \mu_0\frac{\partial \mathbf{D}}{\partial t}, \tag{2}$$

where the electric displacement field is defined by

$$\mathbf{D} = \varepsilon_0\varepsilon_\infty\mathbf{E} + \mathbf{P}, \tag{3}$$

is solved together with an ordinary differential equation for the polarization field, obtained by a Fourier transformation of Equation 1,

$$\left(\frac{\partial^2}{\partial t^2} + \Gamma_i\frac{\partial}{\partial t} + \omega_i^2\right)\mathbf{P} = \varepsilon_0 f\omega_p^2\mathbf{E}. \tag{4}$$

Here f is an oscillator strength (normally set to 1).

Notice that this model is not primarily intended to demonstrate the anomalously high transmission through hole arrays, but rather to demonstrate temporal dispersion modeling.

Model Definition

The geometry consists of a single dispersive slab of thickness 1 µm with a slit of width 0.5 µm in it. The wavelength used is 1 µm. Periodic boundary conditions are applied to make the structure physically appear as an array of slits. The source of electromagnetic radiation is a plane wave pulse with flat front and Gaussian temporal shape.

Results and Discussion

Figure 1 shows the probe plot of the y-component of the electric field at the input boundary. The left part of the curve represents the incoming wave, whereas the right part shows the reflected wave returning to the input boundary.

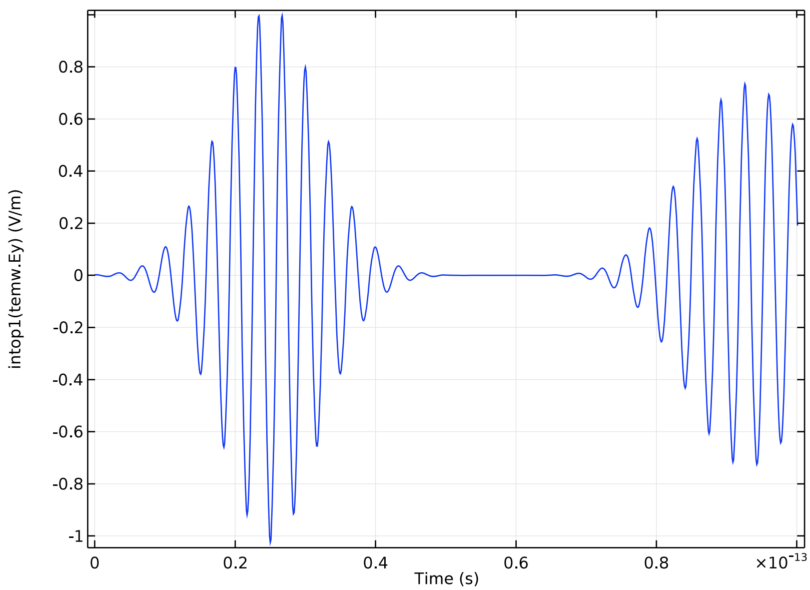


Figure 1: The y-component of the electric field at the input boundary. The left part shows the incident pulse and the right part shows the reflected pulse.

Figure 2 shows the probe plot of the y-component of the polarization at a point in the entrance of the slit. Notice the propagation delay between the incident field, shown in

Figure 1, and the onset of the polarization oscillations at this point.

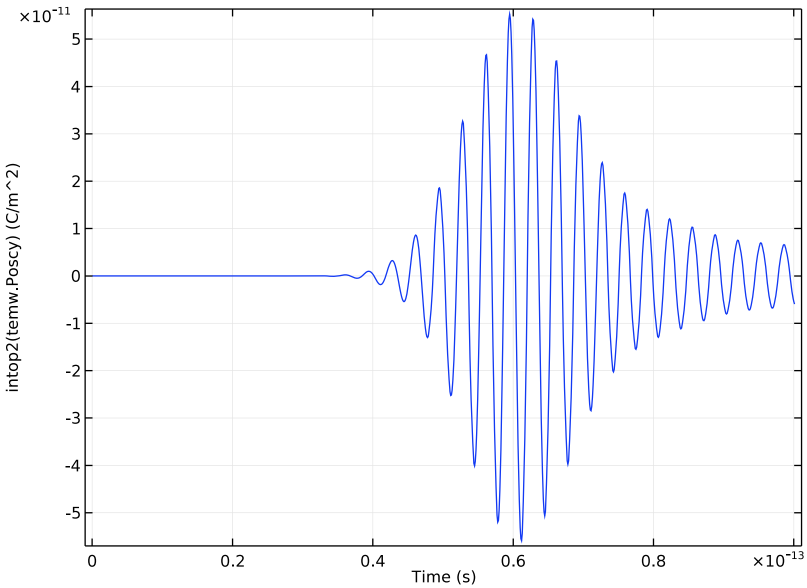


Figure 2: The y-component of the polarization at a point at the entrance of the slit.

Figure 3 shows the probe plot of the y-component of the polarization field at a point at the rear end of the slit.

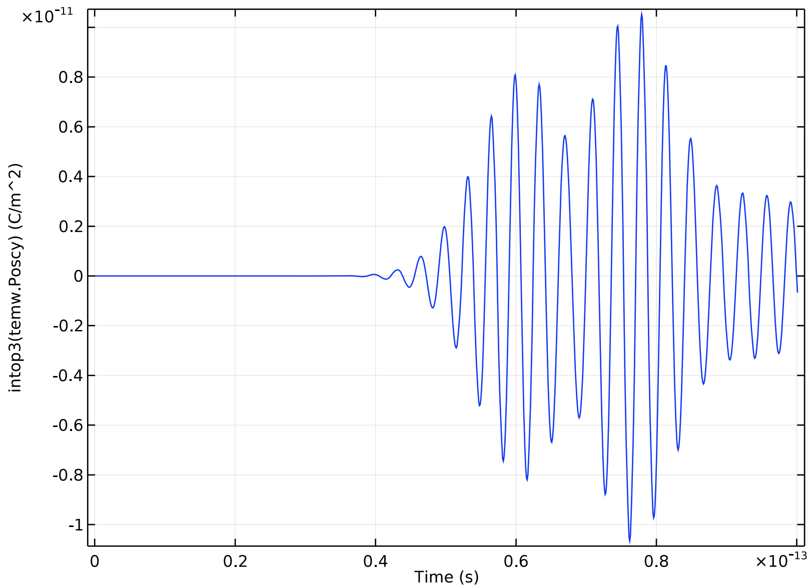


Figure 3: The y-component of the polarization at a point at the exit of the slit.

Figure 4 shows a field plot of the y-component of the polarization field after the last time step (100 fs).

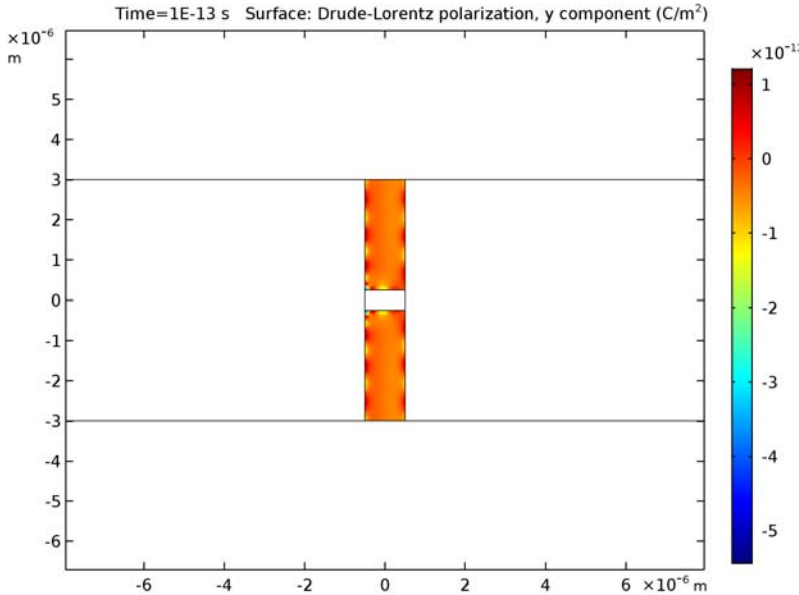


Figure 4: The y-component of the polarization field after 100 fs.

Finally, the out-of-plane component of the magnetic field and, as an overlaid contour plot, the y-component of the polarization field are shown in Figure 5, after 100 fs.

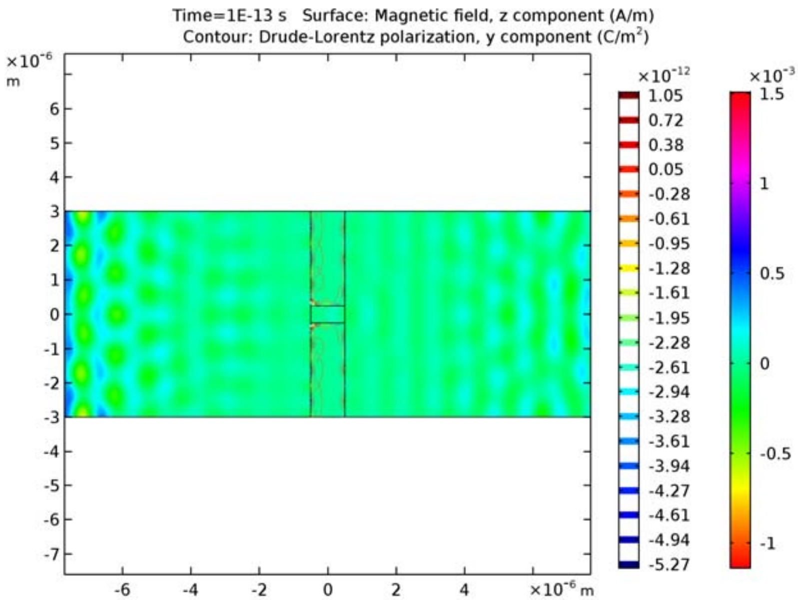


Figure 5: The out-of-plane component of the magnetic field and the y-component of the polarization field (contours) after 100 fs.

Reference

1. T. W. Ebbesen H. J. Lezec, H. F. Ghaemi, T. Thio, and P. A. Wolff, "Extraordinary Optical Transmission Through Sub-wavelength Hole Arrays," *Nature*, vol. 391, pp. 667-9, 1998.

Application Library path: `RF_Module/Tutorials/drude_lorentz_media`

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 **2D**.
2. In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Transient (temw)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.

   Add some parameters that will define the geometry and the properties of the incident field.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
lambda0	1[um]	1E-6 m	Wavelength
E0	1[V/m]	1 V/m	Electric field amplitude
k0	2*pi/lambda0	6.283E6 1/m	Wave number in vacuum
t0	25[fs]	2.5E-14 s	Time delay
dt	10[fs]	1E-14 s	Pulse duration

DEFINITIONS

Variables 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

   Now add some variables that defines the incident field and the material properties.
2. In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
omega0	2*pi[rad]*c_const/ lambda0	rad/s	Angular frequency
E_bnd	E0*cos(omega0*t-k0*x)	V/m	Plane-wave factor for electric field
E_pulse	exp(-(t-t0)^2/dt^2)		Temporal factor for electric field
omega_p	1.5*omega0	rad/s	Plasma frequency
omega_1	0.5*omega_p	rad/s	Resonance frequency
gamma_1	0.1*omega_1	rad/s	Damping coefficient

GEOMETRY 1

The geometry is simple, consisting of only three centered rectangles.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `lambda0`.

4 In the **Height** text field, type `6*lambda0`.

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

Rectangle 2 (r2)

1 In the **Model Builder** window, right-click **Rectangle 1 (r1)** and choose **Duplicate**.

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

3 In the **Width** text field, type `20*lambda0`.

Rectangle 3 (r3)

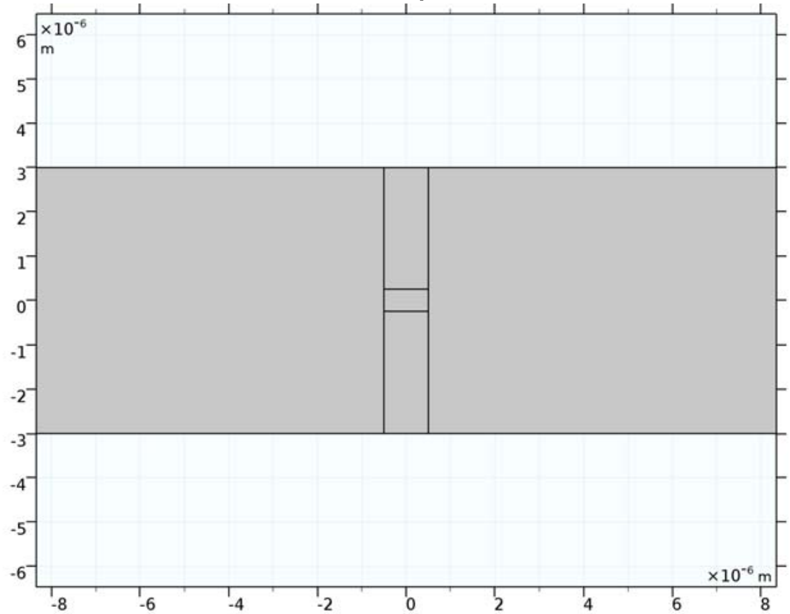
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rectangle 1 (r1)** and choose **Duplicate**.

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

3 In the **Height** text field, type `0.5*lambda0`.

4 Click **Build All Objects**.

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



DEFINITIONS

Now, add three integration operator that will be used for probing the field and the polarization in three different points.

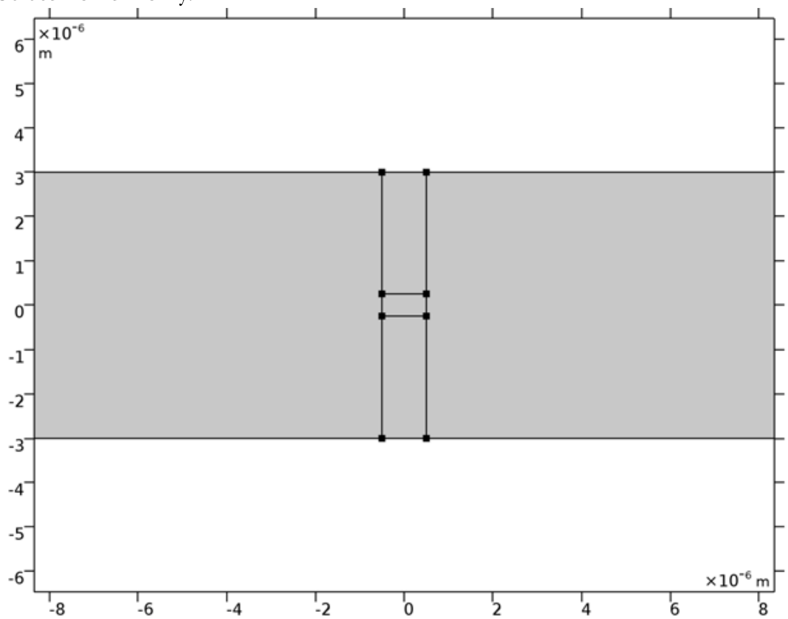
Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

4 Select Point 2 only.



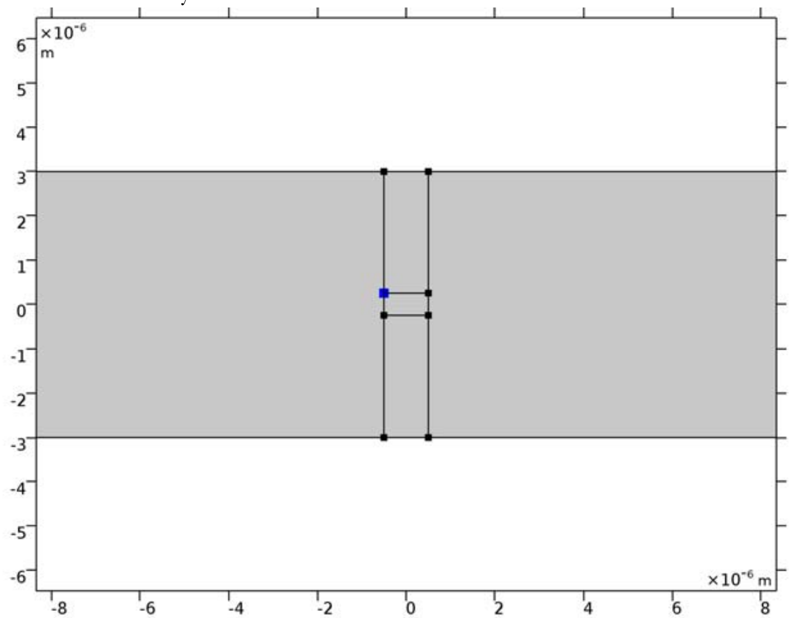
Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

4 Select Point 5 only.



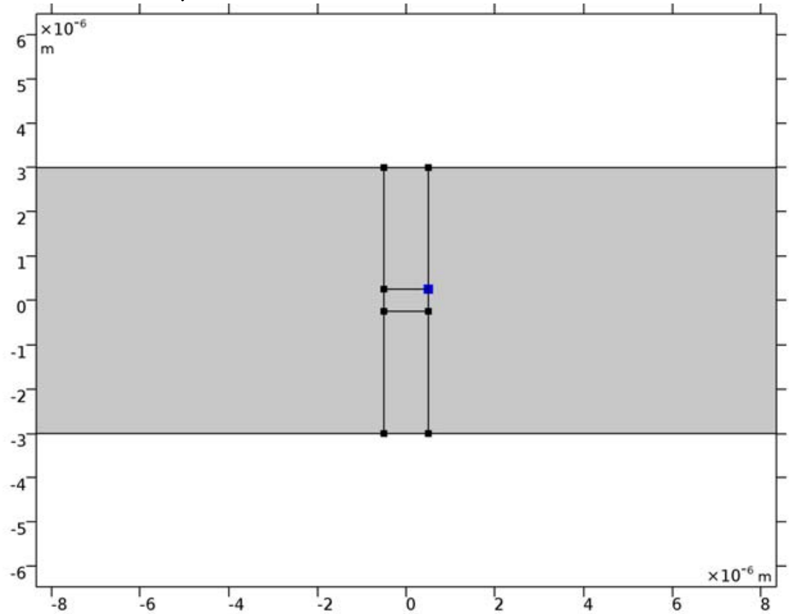
Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

4 Select Point 9 only.



ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Transient (temw)**.

2 In the **Settings** window for **Electromagnetic Waves, Transient**, locate the **Components** section.

3 From the **Electric field components solved for** list, choose **In-plane vector**, to solve only for the in-plane components of the field.

Wave Equation, Electric 1

Define the first wave equation feature to use the Drude-Lorentz dispersion model. Later you will add another wave equation feature for the air domain.

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)** click **Wave Equation, Electric 1**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Drude-Lorentz dispersion model**.

4 From the **Relative permittivity, high frequency** list, choose **User defined**. From the list, choose **Diagonal**.

5 In the **Relative permittivity, high frequency** table, enter 4 for the two first diagonal elements.

6 In the ω_P text field, type `omega_p`.

7 Locate the **Magnetic Field** section. From the μ_r list, choose **User defined**. Accept the default value 1.

8 Locate the **Conduction Current** section. From the σ list, choose **User defined**. Accept the default value 0.

Next, you add a Drude-Lorentz Polarization feature, as a subfeature to the wave equation. There, more material parameters will be defined for the polarization field.

Drude-Lorentz Polarization 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)> Wave Equation, Electric 1** and choose **Drude-Lorentz Polarization**.

2 In the **Settings** window for **Drude-Lorentz Polarization**, locate the **Drude-Lorentz Dispersion Model** section.

3 In the f_n text field, type 1.

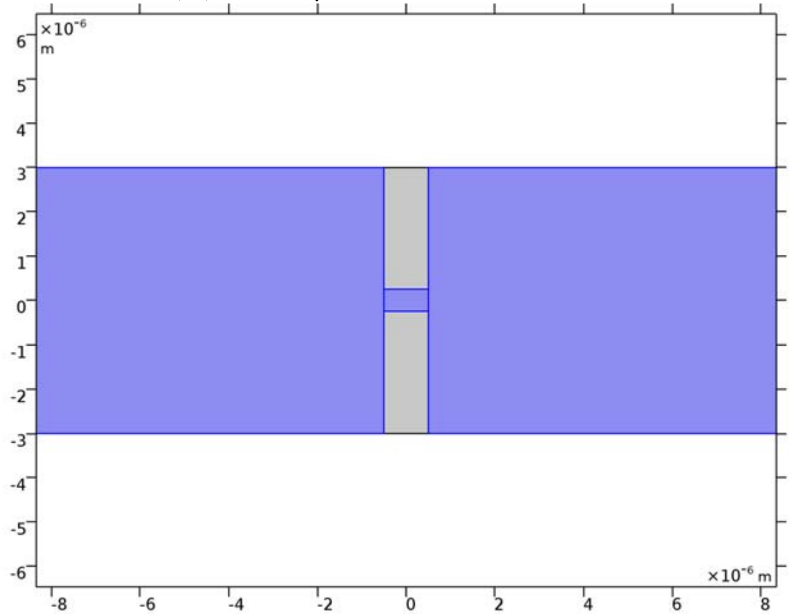
4 In the ω_n text field, type `omega_1`.

5 In the Γ_n text field, type `gamma_1`.

Wave Equation, Electric 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Transient (temw)** and choose **Wave Equation, Electric**.

2 Select Domains 1, 3, and 5 only.



3 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

4 From the ε_r list, choose **User defined**. Locate the **Magnetic Field** section. From the μ_r list, choose **User defined**. Locate the **Conduction Current** section. From the σ list, choose **User defined**. Use scattering boundary conditions to excite the wave and to absorb it.

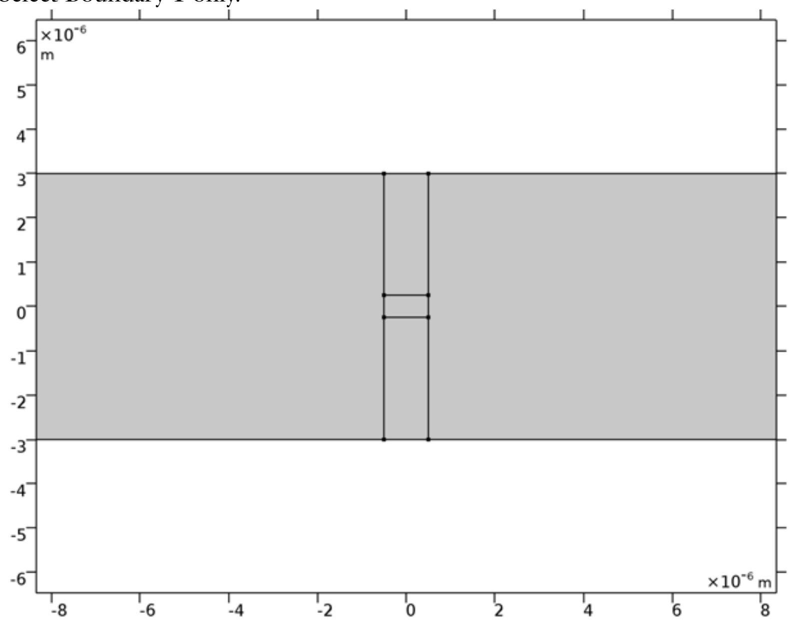
Scattering Boundary Condition 1

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Scattering Boundary Condition**.

2 In the **Settings** window for **Scattering Boundary Condition**, locate the **Scattering Boundary Condition** section.

3 From the **Incident field** list, choose **Wave given by E field**.

4 Select Boundary 1 only.



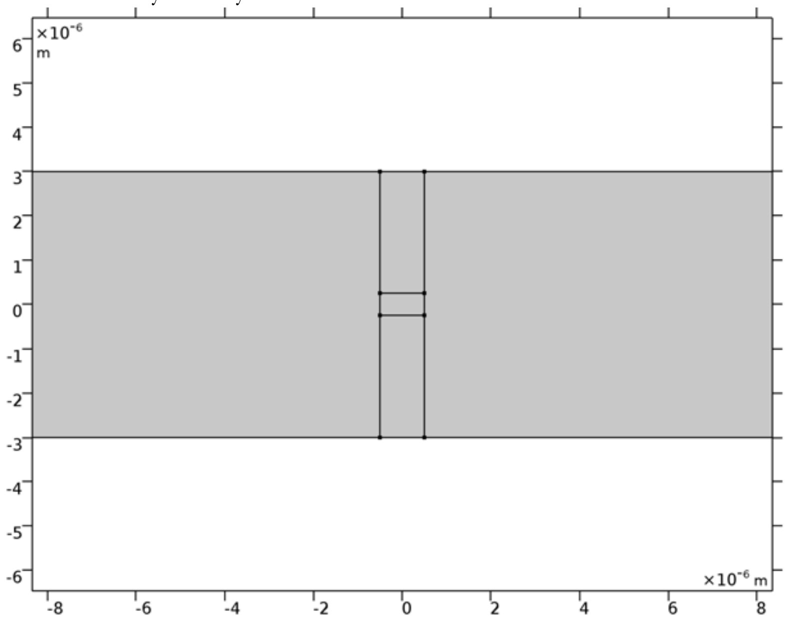
5 Specify the $\mathbf{E}_0$ vector as

0	x
E_pulse*E_bnd	y
0	z

Scattering Boundary Condition 2

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Scattering Boundary Condition**.

2 Select Boundary 16 only.

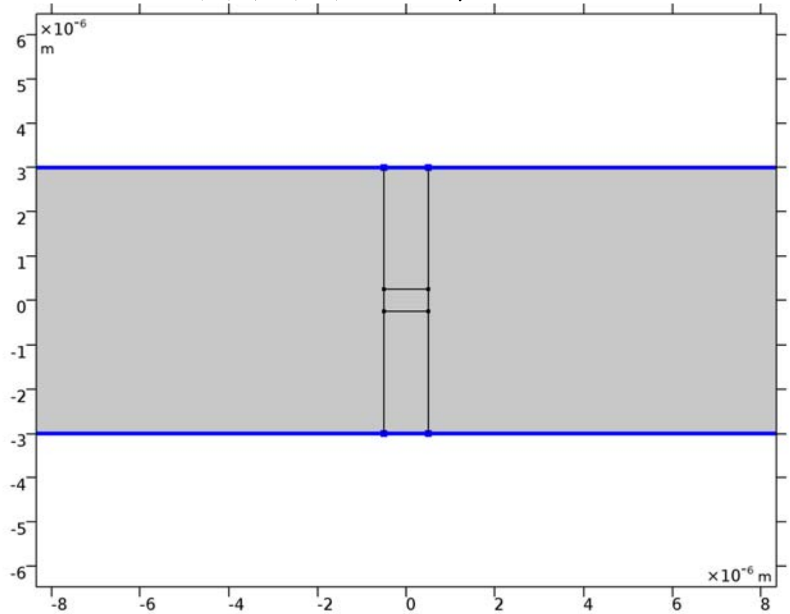


To model a hole array, periodic boundary conditions will be used.

Periodic Condition 1

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Periodic Condition**.

2 Select Boundaries 2, 3, 5, 10, 12, and 15 only.



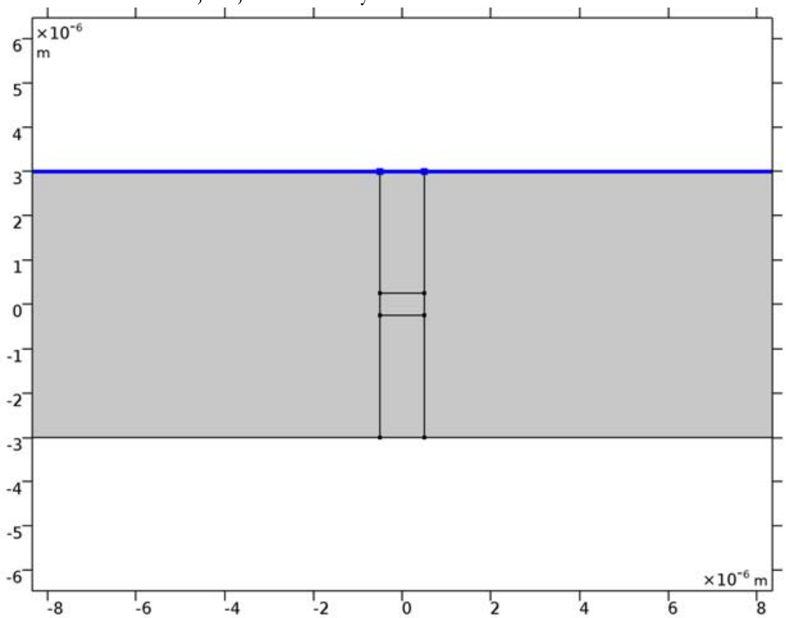
MESH 1

Since a periodic boundary condition is used, the mesh should also be the same on the top and bottom edge. Thus, add first an edge mesh and copy the mesh points to the opposite edge. Then add a triangular mesh.

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 3, 10, and 15 only.



Copy Edge 1

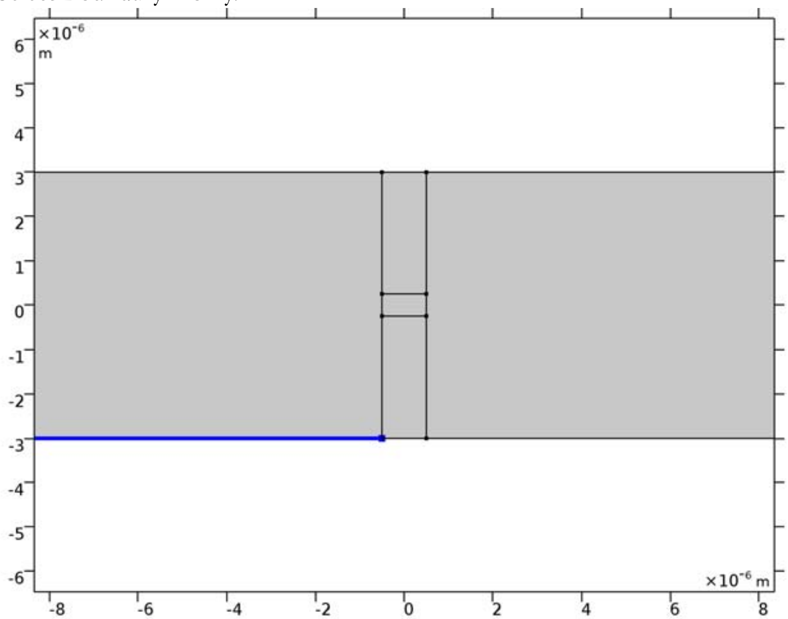
1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Edge**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundary 2 only.



Copy Edge 2

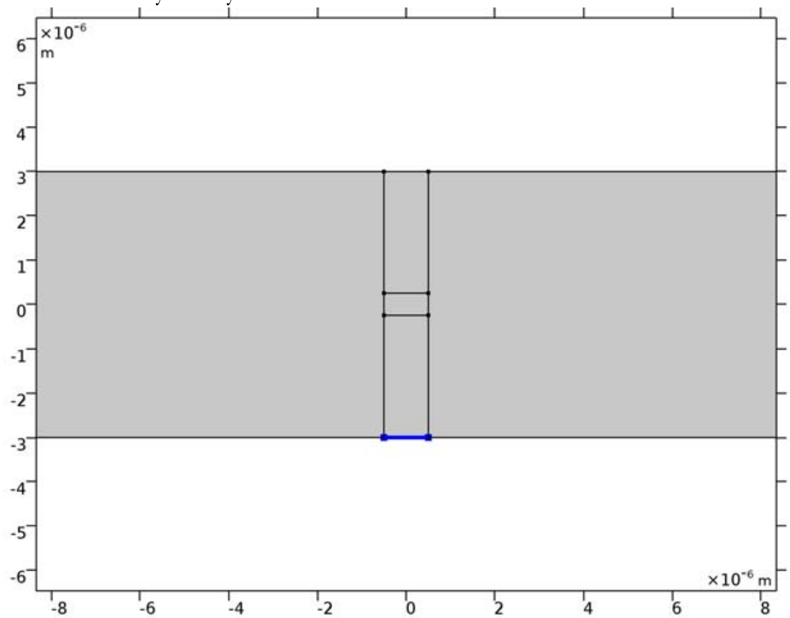
1 Right-click **Mesh 1** and choose **More Operations>Copy Edge**.

2 Select Boundary 10 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundary 5 only.



Copy Edge 3

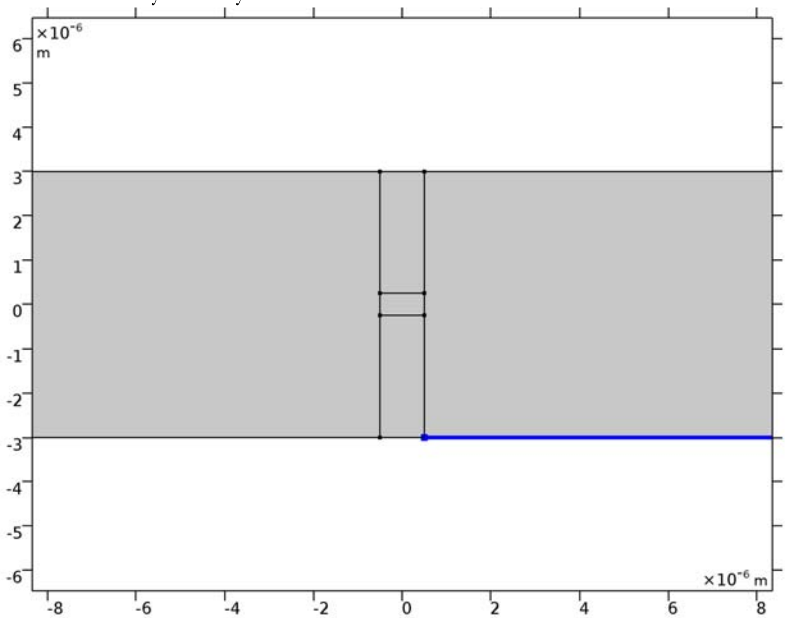
1 Right-click **Mesh 1** and choose **More Operations>Copy Edge**.

2 Select Boundary 15 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundary 12 only.



Size

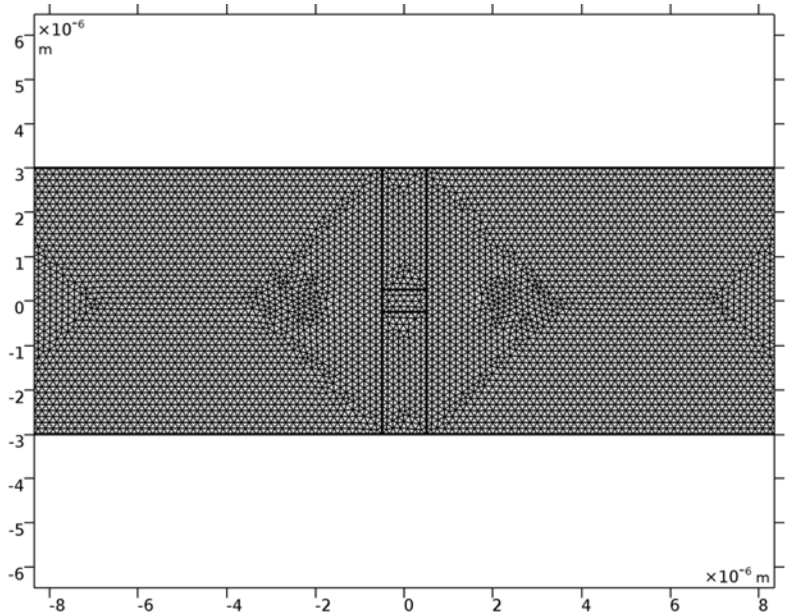
1 Right-click **Mesh 1** and choose **Free Triangular**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda0/6`.

5 Click **Build All**.



STUDY 1

Step 1: Time Dependent

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

2 In the **Times** text field, type `range(0,10[fs],100[fs])`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**, to be able to make some modifications of the solver settings.

Force the solver to use a fixed small step size that resolves the temporal field oscillations.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `0.1[fs]`.

DEFINITIONS

Before computing the solution, define the three Global Variable Probes that can be used for monitoring the computation progress.

Global Variable Probe 1 (var1)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.
3 In the **Expression** text field, type `intop1(temw.Ey)`.
4 Click to expand the **Table and window settings** section.

Global Variable Probe 2 (var2)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.
3 In the **Expression** text field, type `intop2(temw.Poscy)`.
4 Locate the **Table and Window Settings** section. From the **Plot window** list, choose **New window**.

Global Variable Probe 3 (var3)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.
3 In the **Expression** text field, type `intop3(temw.Poscy)`.
4 Locate the **Table and Window Settings** section. From the **Plot window** list, choose **New window**.
5 On the **Home** toolbar, click **Compute**.

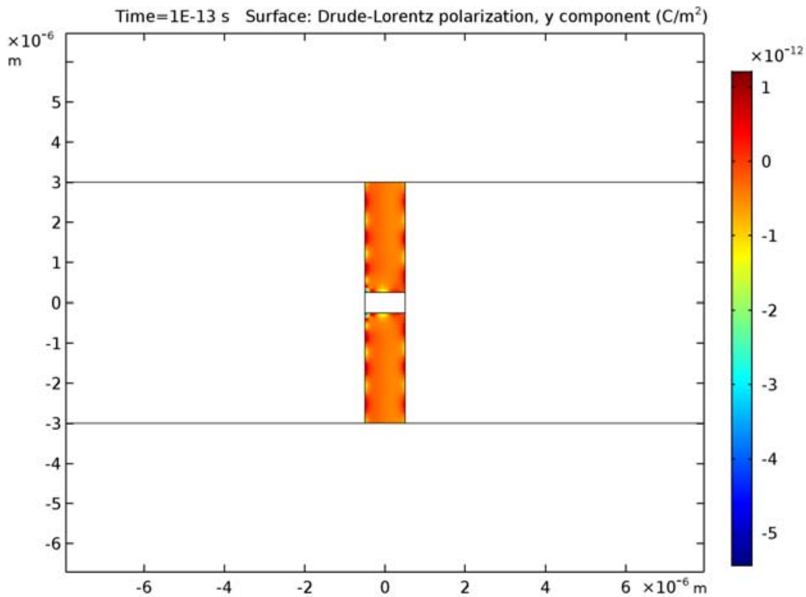
RESULTS

Surface 1

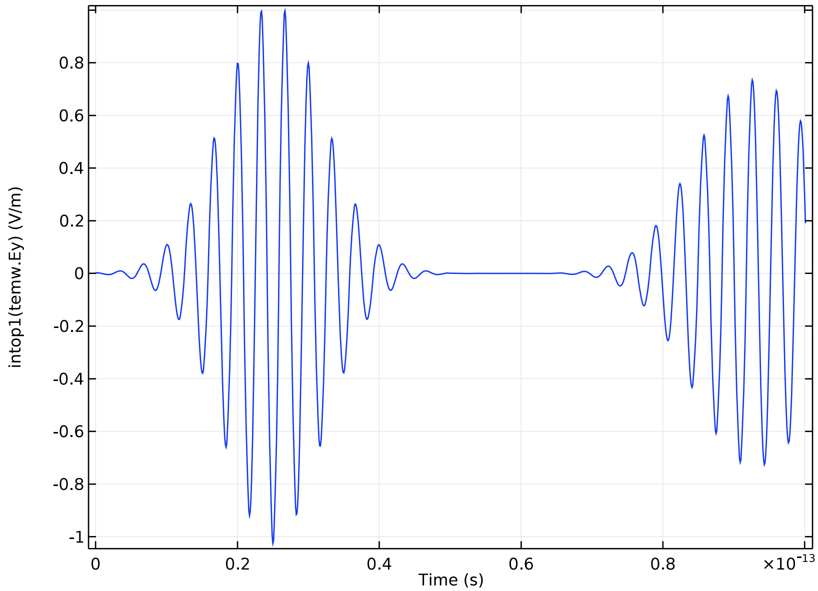
Modify this surface plot to show the *y* component of the Drude-Lorentz polarization.

1 In the **Model Builder** window, expand the **2D Plot Group 1** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `temw.Poscy`.
4 On the **2D Plot Group 1** toolbar, click **Plot**.

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

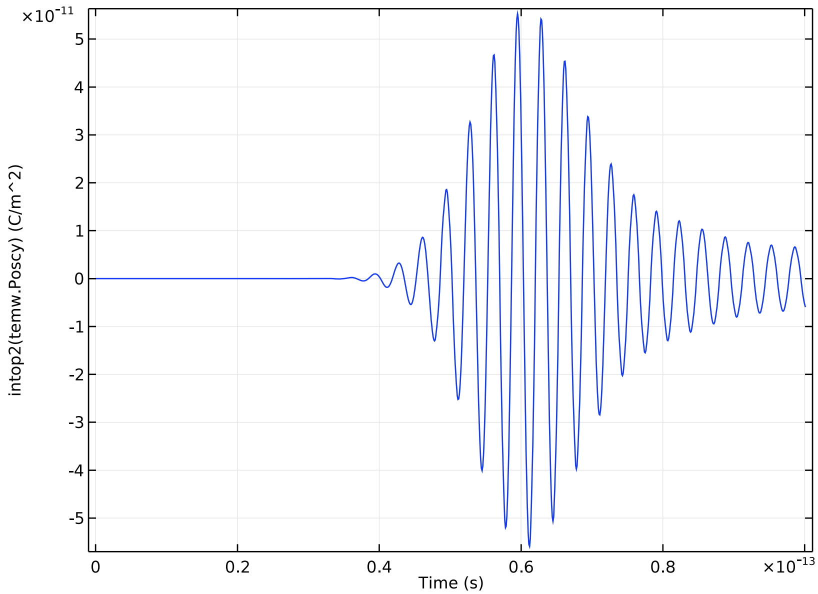


Probe Plot Group 2



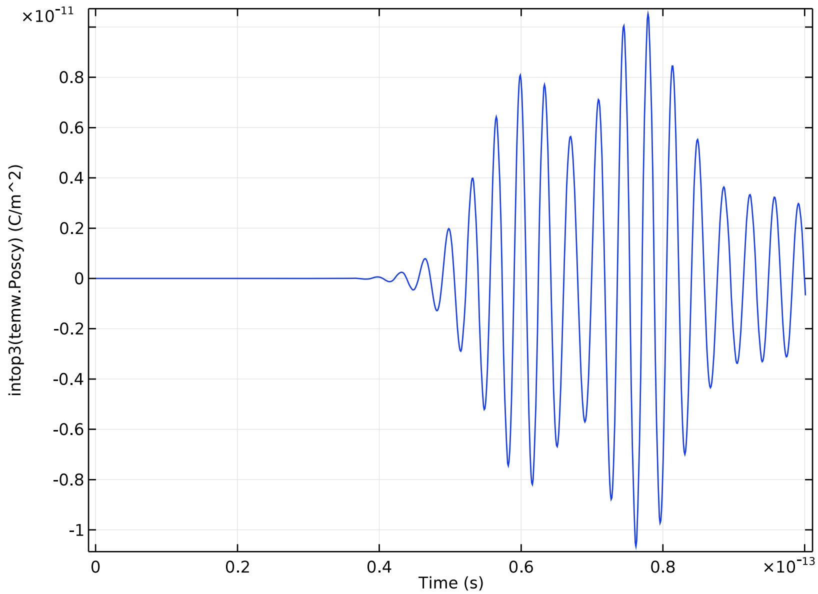
The first probe plot should look like the one above.

Probe Plot Group 3



The second probe plot should look like the one above.

Probe Plot Group 4



Finally, the third probe plot should look like the one above.

Now, add a surface plot of the *z* component of the magnetic field and overlay a contour plot of the *y* component of the Drude-Lorentz polarization.

Surface 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 5** and choose **Surface**.

3 In the **Settings** window for **Surface**, locate the **Expression** section.

4 In the **Expression** text field, type `temw.Hz`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Cyclic**.

Contour 1

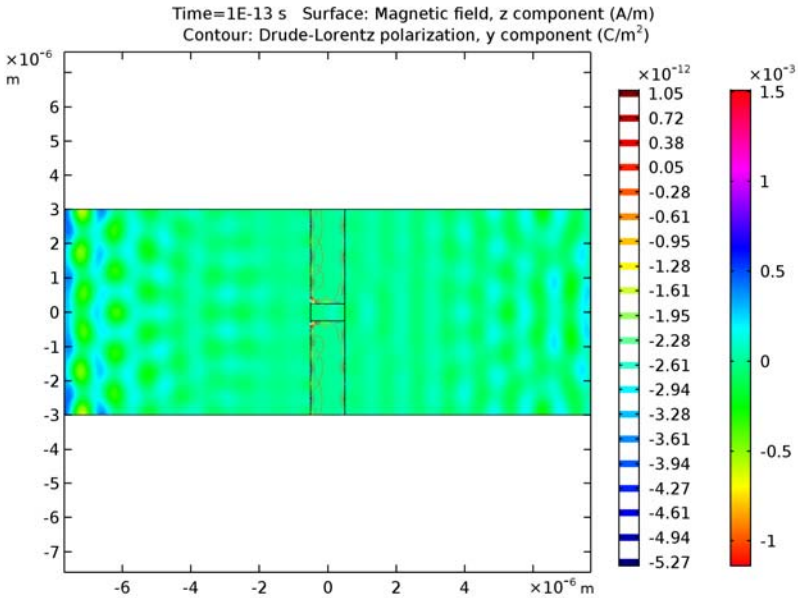
1 In the **Model Builder** window, under **Results** right-click **2D Plot Group 5** and choose **Contour**.

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

3 In the **Expression** text field, type `temw.Poscy`.

4 On the **2D Plot Group 5** toolbar, click **Plot**.

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



Created in COMSOL Multiphysics 5.3

Frequency Selective Surface, Complementary Split Ring Resonator

Introduction

Frequency selective surfaces (FSS) are periodic structures with a bandpass or a bandstop frequency response. This example shows that only signals around the center frequency can pass through the periodic complementary split ring resonator layer.

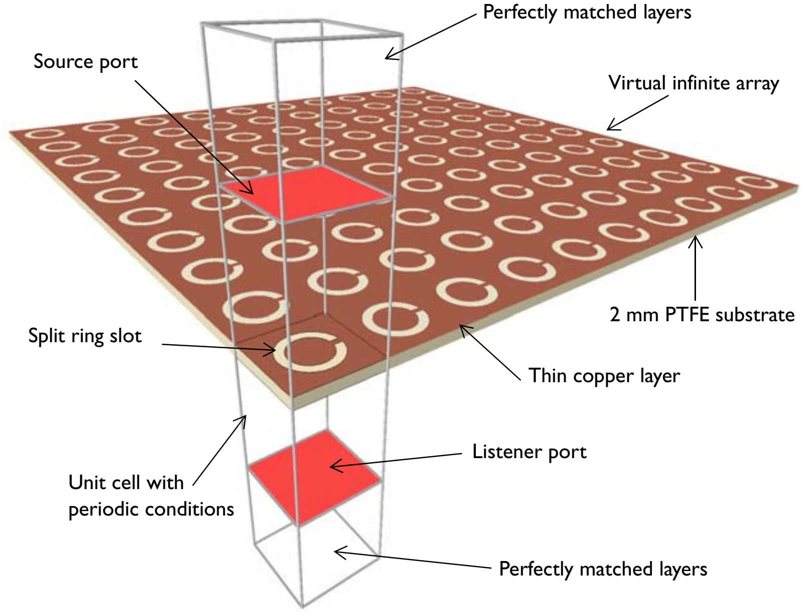


Figure 1: One unit cell of the complementary split ring resonator is modeled with periodic boundary conditions to simulate an infinite 2D array. Perfectly matched layers at the top and bottom of the unit cell absorb the excited and higher order modes.

Model Definition

A split ring slot is patterned on a geometrically thin copper layer that sits on a 2 mm PTFE substrate (Figure 1). The copper layer is much thicker than the skin depth in the simulated frequency range, so it is modeled as a perfect electric conductor (PEC). The rest of the simulation domain is filled with air.

Floquet-periodic boundary conditions are used on four sides of the unit cell to simulate the infinite 2D array. Perfectly matched layers (PMLs) on the top and bottom of the unit cell absorb the excited mode from the source port and any higher order modes generated by the periodic structure. The PMLs attenuate the wave as it propagates in the direction perpendicular to the PML boundary. Since the model is solved for a range of incident angles, the wavelength in the PMLs is set to $2\pi/|k_0\cos\theta|$. This accounts for the angular dependence of the normal component of the wave vector inside the PMLs.

Port boundary conditions are placed on the interior boundaries of the PMLs, adjacent to the air domains. The Port boundary conditions automatically determine the reflection and transmission characteristics in terms of S-parameters. The interior port boundaries with PML backing require the slit condition. The port orientation is specified to define the inward direction for the S-parameter calculation. Since higher order diffraction modes are not of particular interest in this example, the combination of Domain-backed type slit port and PMLs is used instead of adding a Diffraction order port for each diffraction order and polarization.

The periodic boundary condition requires identical surface meshes on paired boundaries. This is accomplished in two steps: first by creating a mesh on only one of the boundaries and then using the Copy Face operation for the mesh on the other boundary. This mesh configuration is automatically set when using the physics-controlled mesh as shown in the step-by-step instructions. If you are interested in seeing more details about the mesh, build the physics-controlled mesh once and then change the mesh sequence type to the user-controlled mesh in the mesh settings. Then you can inspect the generated mesh sequence.

Results and Discussion

The modified multislice default plot (Figure 2) shows the electric field norm on the complimentary split ring resonator. Strong fields are observed inside the slot. The S-parameter plot in Figure 3 shows that this periodic structure functions as a bandpass filter near 4.6 GHz. In Figure 4, the S-parameters appear as a function of incident angle and show that the periodic structure is penetrable at 4.6 GHz over the simulated range, except for grazing angles.

The resonance frequency of this periodic structure can be quickly evaluated as 4.59 GHz using an Eigenfrequency study, which is not included in this example.

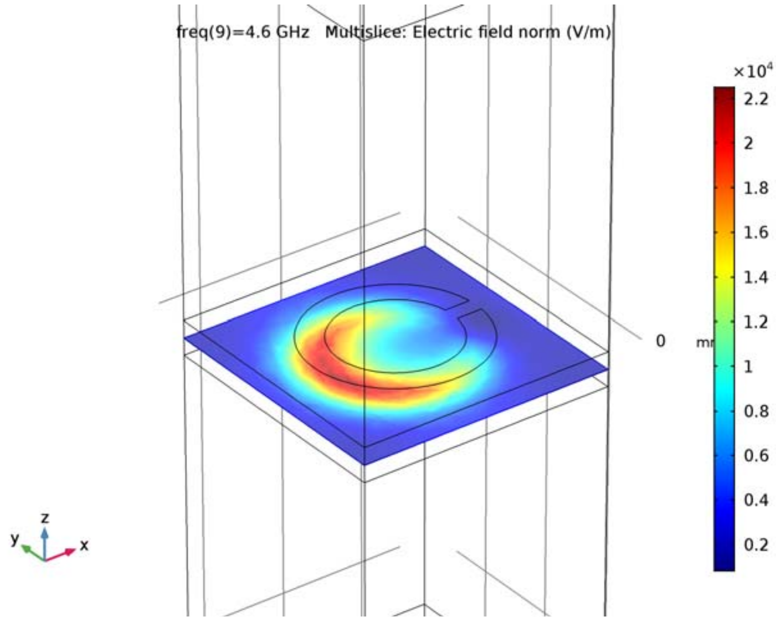


Figure 2: The fields are confined in the split ring slot.

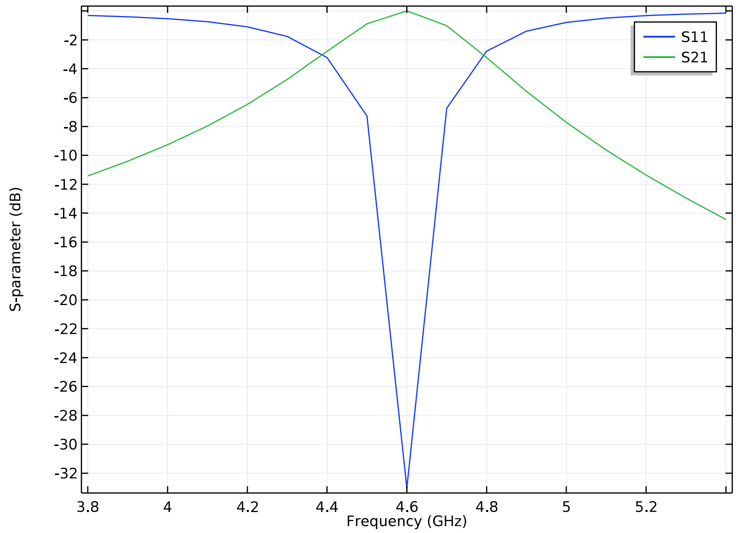


Figure 3: The S-parameter plot shows a bandpass resonance near 4.6 GHz

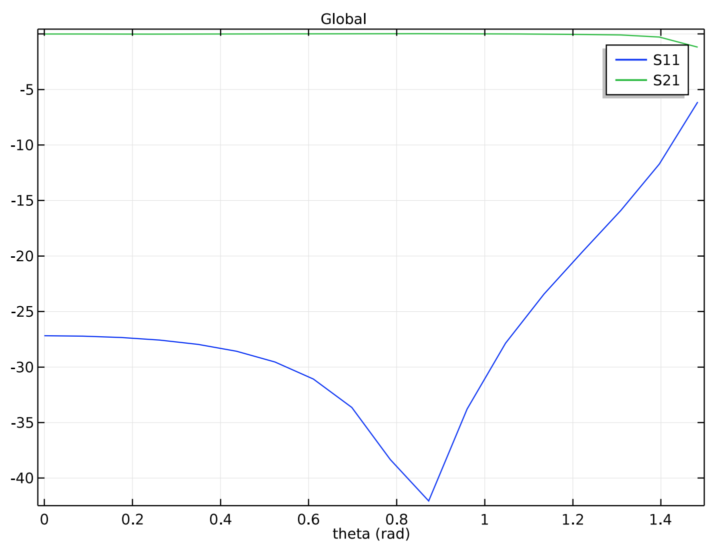


Figure 4: The S-parameter plot is shown as a function of incident angle.

Application Library path: `RF_Module/EMI_EMC_Applications/frequency_selective_surface_csrr`

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 **Preset 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 In the **Frequencies** text field, type `range(3.8[GHz],0.1[GHz],5.4[GHz])`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
theta	0[deg]	0 rad	Elevation angle

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**.

Block 1 (blk1)

1 On 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 15.

4 In the **Depth** text field, type 15.

5 In the **Height** text field, type 45.

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

7 Right-click **Block 1 (blk1)** and choose **Build Selected**.

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

Circle 1 (c1)

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

2 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

4 In the **Radius** text field, type 5.

5 Right-click **Circle 1 (c1)** and choose **Build Selected**.

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

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type 3.5.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 4.

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

5 In the **xw** text field, type 4.

Difference 1 (dif1)

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

2 Select the object **c1** only.

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

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

5 Click the **Select Box** button on the **Graphics** toolbar.

6 Select the objects **r1** and **c2** only.

7 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

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

Block 2 (blk2)

1 On 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 15.

4 In the **Depth** text field, type 15.

5 In the **Height** text field, type 2.

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

7 In the **z** text field, type -1.

Block 3 (blk3)

1 On 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 15.

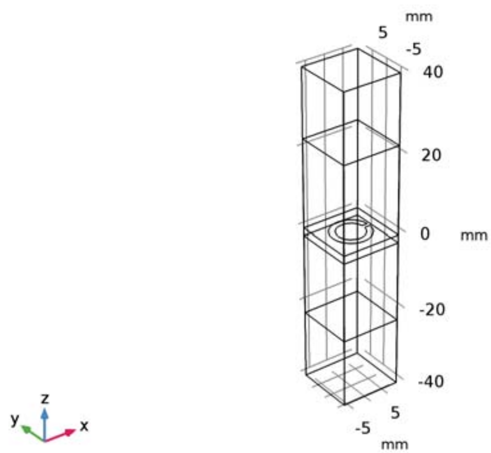
4 In the **Depth** text field, type 15.

5 In the **Height** text field, type 80.

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

7 Click **Build All Objects**.

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



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Set up the physics. First, define physics-controlled mesh maximum element size. Use Floquet-periodic conditions on all side boundaries.

Perfect Electric Conductor 2

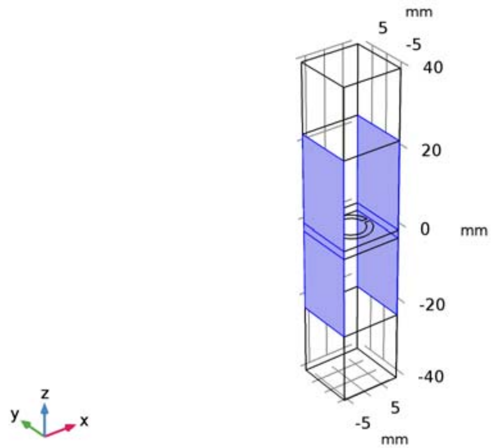
1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Perfect Electric Conductor**.

2 Select Boundary 12 only.

Periodic Condition 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Periodic Condition**.

2 Select Boundaries 4, 7, 10, and 24–26 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

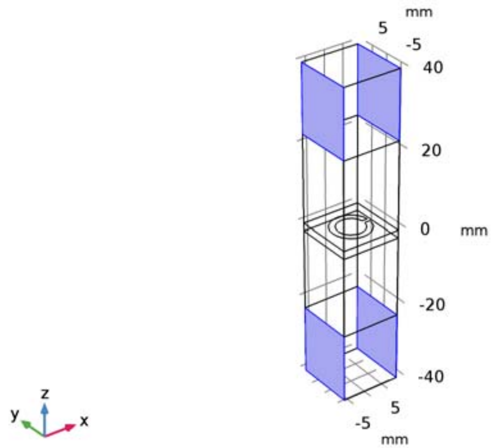
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

Periodic Condition 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Periodic Condition**.

2 Select Boundaries 1, 13, 23, and 27 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

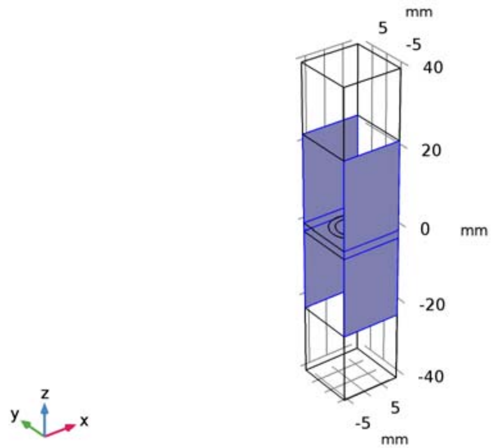
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

Periodic Condition 3

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Periodic Condition**.

2 Select Boundaries 5, 8, 11, and 18–20 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

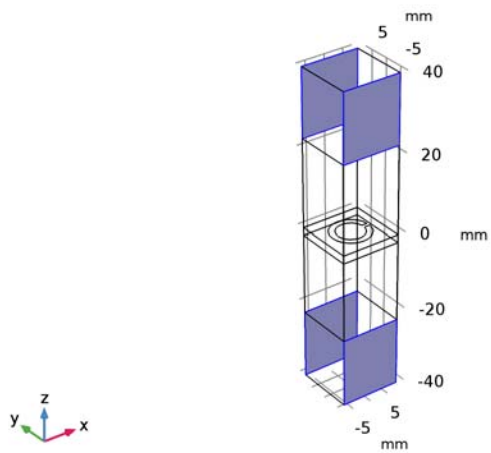
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

Periodic Condition 4

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Periodic Condition**.

2 Select Boundaries 2, 14, 17, and 21 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

The wave is excited from the port on the top.

Port 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 15 only.

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

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

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

5 Select the **Activate slit condition on interior port** check box.

6 From the **Slit type** list, choose **Domain-backed**.

7 From the **Port orientation** list, choose **Reverse**.

8 Locate the **Port Mode Settings** section. From the **Input quantity** list, choose **Magnetic field**.

9 Specify the $\mathbf{H}_0$ vector as

0	x
1	y
0	z

10 In the α_1 text field, type `theta`.

The maximum frequency in the setting window will be used only when **Compute Diffraction Order** button is clicked to generate Diffraction Order features handling higher order mode individually. In this model, PML absorbs all higher order modes, so this setting is ineffective.

Port 2

1. Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.
2. Select Boundary 6 only.
3. In the **Settings** window for **Port**, locate the **Port Properties** section.
4. From the **Type of port** list, choose **Periodic**.
5. Select the **Activate slit condition on interior port** check box.
6. From the **Slit type** list, choose **Domain-backed**.
7. From the **Port orientation** list, choose **Reverse**.
8. Locate the **Port Mode Settings** section. From the **Input quantity** list, choose **Magnetic field**.
9. Specify the $\mathbf{H}_0$ vector as

0	x
1	y
0	z

Scattering Boundary Condition 1

1. Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Scattering Boundary Condition**.
2. Select Boundary 3 only.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1. On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 5 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Typical wavelength from** list, choose **User defined**.

5 In the **Typical wavelength** text field, type `2*pi/abs(emw.k0*cos(theta))`.

Perfectly Matched Layer 2 (pml2)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 1 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Typical wavelength from** list, choose **User defined**.

5 In the **Typical wavelength** text field, type `2*pi/abs(emw.k0*cos(theta))`.

ADD MATERIAL

1 On 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.

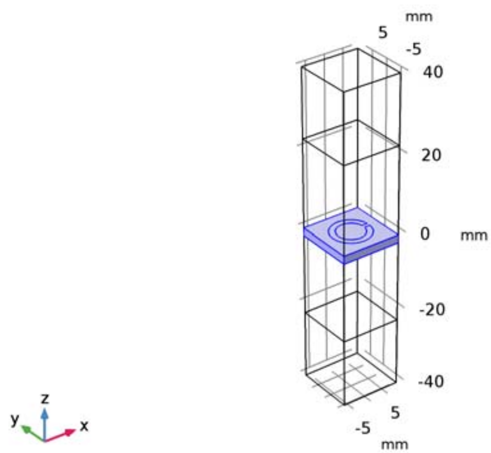
MATERIALS

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

Material 2 (mat2)

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

2 Select Domain 3 only.



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

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	2.1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

MESH 1

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

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

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

4 Locate the **Mesh Settings** section. From the **Element size** list, choose **Extremely fine**.

5 Click **Build All**.

DEFINITIONS

Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **View 1** and choose **Hide for Physics**.

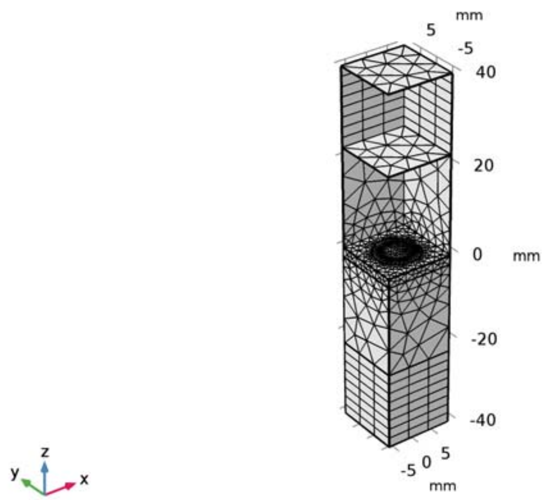
2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 10, 11, 13, and 14 only.

MESH 1

In the Model Builder window, click Mesh 1.



STUDY 1

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

RESULTS

Electric Field (emw)

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

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

3 From the **Parameter value (freq (GHz))** list, choose **4.6**.

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`.

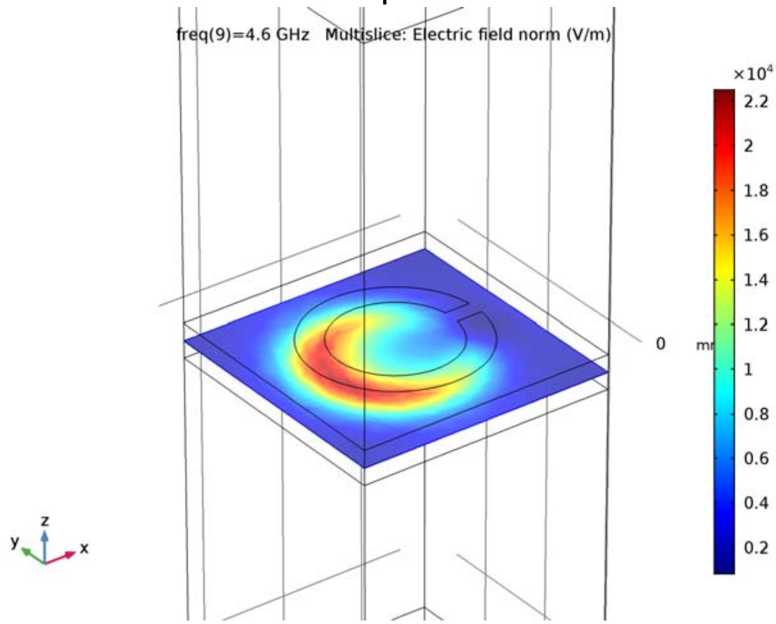
5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `-1`.

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

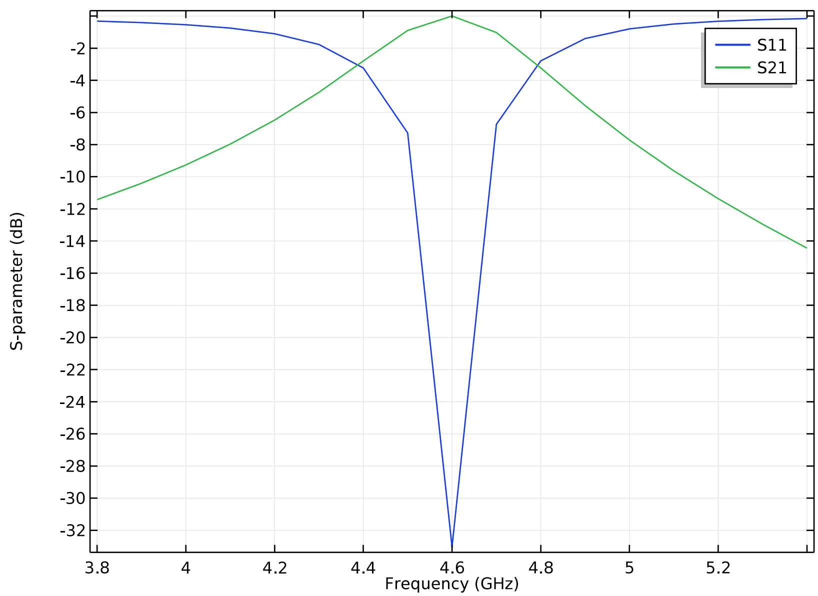
8 Click the **Zoom In** button on the **Graphics** toolbar.

9 Click the **Zoom In** button on the **Graphics** toolbar.

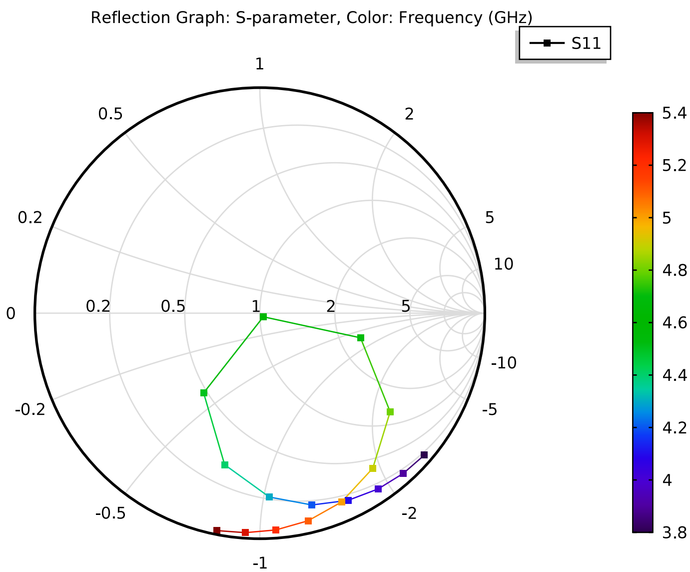


This reproduces Figure 2.

S-Parameter (emw)



Smith Plot (emw)



Identify the resonant frequency of the periodic structure from the S-parameter plot Figure 3.

Next, evaluate the reflectivity and transmittivity performance of the model with different incident angles.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

STUDY 1

1 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

2 Click **Add Study** in the window toolbar.

If you want to clear the Add Study window after adding, click Add Study again on the Home toolbar.

ROOT

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

STUDY 2

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta	`range(0[deg],5[deg],85[deg])`	

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `4.6[GHz]`.

4 On the **Study** toolbar, click **Compute**.

RESULTS

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw) 1** 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 -1.
7 On the **Electric Field (emw) 1** toolbar, click **Plot**.

Add a 1D plot.

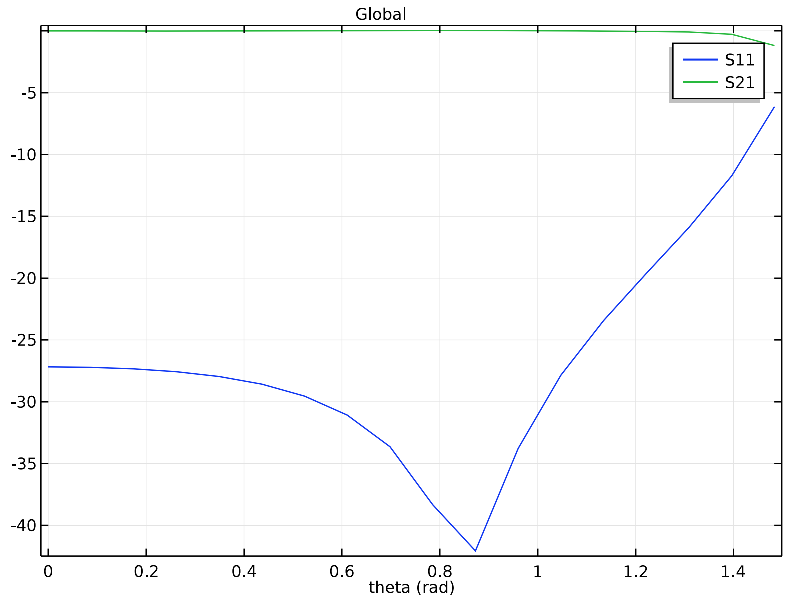
1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

Global 1

1 Right-click **1D Plot Group 5** 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>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S11dB - S11**.
3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S21dB - S21**.

4 On the **1D Plot Group 5** toolbar, click **Plot**.



This is the S-parameter plot as a function of incident angle shown in Figure 4.

Created in COMSOL Multiphysics 5.3

Fresnel Equations

Introduction

A plane electromagnetic wave propagating through free space is incident at an angle upon an infinite dielectric medium. This model computes the reflection and transmission coefficients and compares the results to the Fresnel equations.

Model Definition

A plane wave propagating through free space ($n = 1$) as shown in Figure 1 is incident upon an infinite dielectric medium ($n = 1.5$) and is partially reflected and partially transmitted. If the electric field is *p-polarized*—that is, if the electric field vector is in the same plane as the Poynting vector and the surface normal—then there are no reflections at an incident angle of roughly 56°, known as the *Brewster angle*.

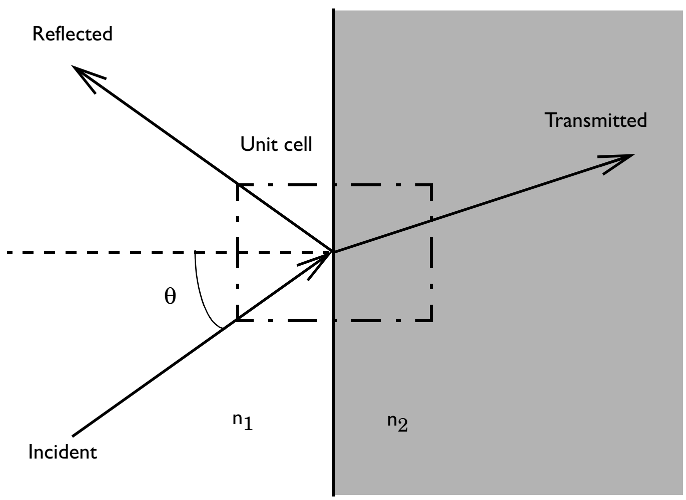


Figure 1: A plane wave propagating through free space incident upon an infinite dielectric medium.

Although, by assumption, space extends to infinity in all directions, it is sufficient to model a small unit cell, as shown in Figure 1; a Floquet-periodic boundary condition applies on the top and bottom unit-cell boundaries because the solution is periodic along the interface. This model uses a 3D unit cell, and applies perfect electric conductor and perfect magnetic conductor boundary conditions as appropriate to model out-of-plane symmetry. The angle of incidence ranges between 0–90° for both polarizations.

For comparison, Ref. 1 and Ref. 2 provide analytic expressions for the reflectance and transmittance[1]. Reflection and transmission coefficients for s-polarization and p-polarization are defined respectively as

$$r_s = \frac{n_1 \cos\theta_{\text{incident}} - n_2 \cos\theta_{\text{transmitted}}}{n_1 \cos\theta_{\text{incident}} + n_2 \cos\theta_{\text{transmitted}}}$$

$$t_s = \frac{2n_1 \cos\theta_{\text{incident}}}{n_1 \cos\theta_{\text{incident}} + n_2 \cos\theta_{\text{transmitted}}}$$

$$r_p = \frac{n_2 \cos\theta_{\text{incident}} - n_1 \cos\theta_{\text{transmitted}}}{n_1 \cos\theta_{\text{transmitted}} + n_2 \cos\theta_{\text{incident}}}$$

$$t_p = \frac{2n_1 \cos\theta_{\text{incident}}}{n_1 \cos\theta_{\text{transmitted}} + n_2 \cos\theta_{\text{incident}}}$$

Reflectance and transmittance are defined as

$$R = |r|^2$$

$$T = \frac{n_2 \cos\theta_{\text{transmitted}}}{n_1 \cos\theta_{\text{incident}}} |t|^2$$

The Brewster angle at which $r_{\text{p}} = 0$ is defined as

$$\theta_B = \operatorname{atan}\frac{n_2}{n_1}$$

1. Note that depending on the sign convention for what is defined as a positive polarization for the reflected wave with in-plane (p-polarization), the sign for the reflection coefficient r_p can differ for different authors. However, the total field solution will always be the same.

Results and Discussion

Figure 2 is a combined plot of the y component of the electric-field distribution and the power flow visualized as an arrow plot for the TE case.

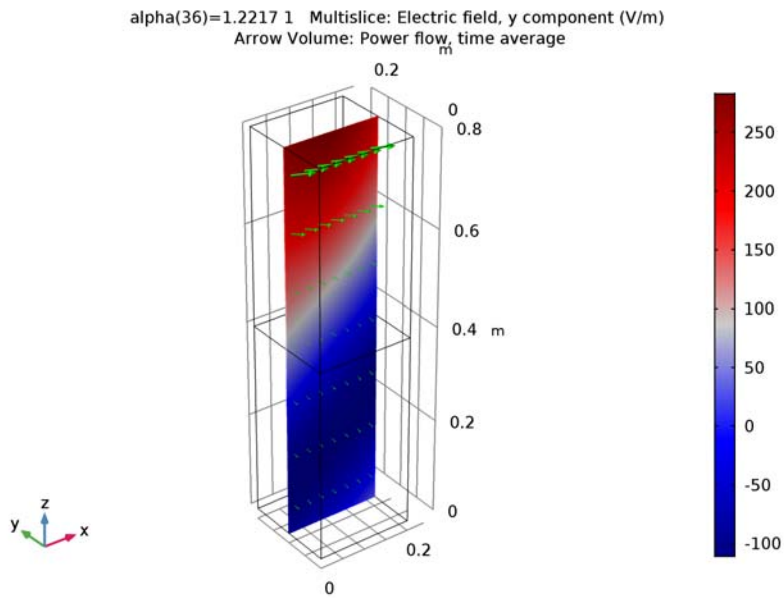


Figure 2: Electric field, E_y (slice) and power flow (arrows) for TE incidence at 70° inside the unit cell.

For the TM case, Figure 3 visualizes the y component of the magnetic-field distribution instead, again in combination with the power flow.

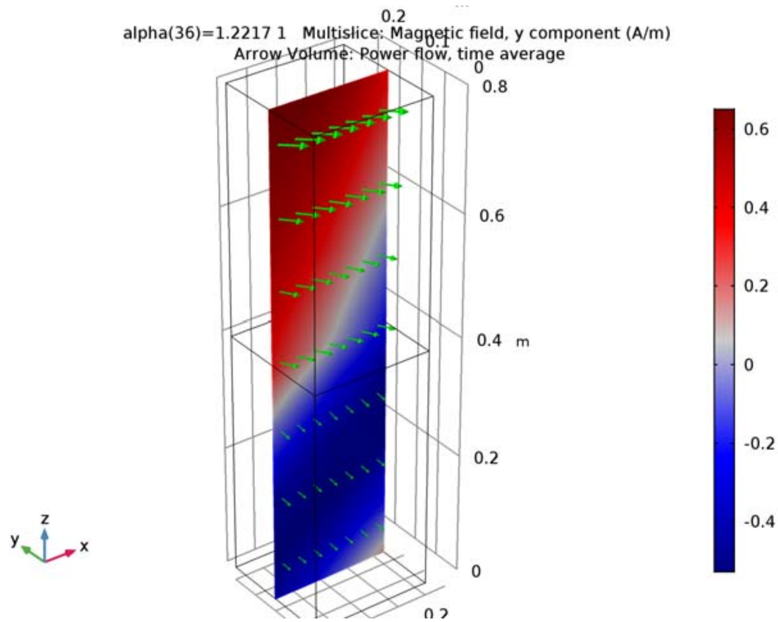


Figure 3: Magnetic field, H_y (slice) and power flow (arrows) for TM incidence at 70° inside the unit cell.

Note that the sum of reflectance and transmittance in Figure 4 and Figure 5 equals 1, showing conservation of power. Figure 5 also shows that the reflectance around 56°—the Brewster angle in the TM case—is close to zero.

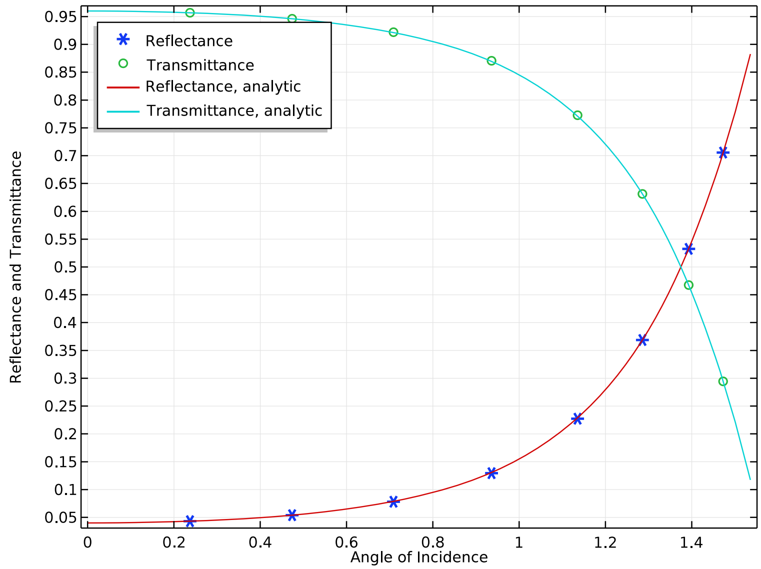


Figure 4: The reflectance and transmittance for TE incidence agree well with the analytic solutions.

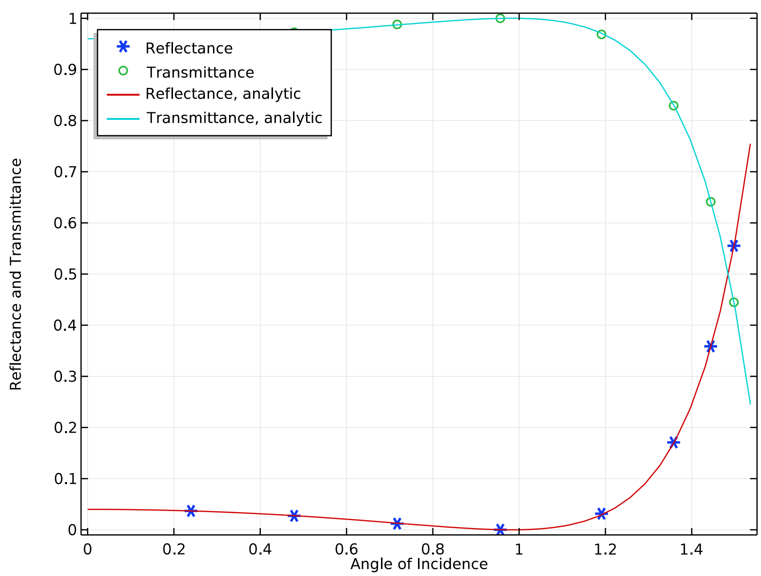


Figure 5: The reflectance and transmittance for TM incidence agree well with the analytic solutions. The Brewster angle is also observed at the expected location.

References

1. J.D. Jackson, *Classical Electrodynamics*, 3rd Ed., Wiley, 1999.

2. B.E.A. Saleh and M.C. Teich, *Fundamentals of Photonics*, Wiley, 1991.

Application Library path: `RF_Module/Verification_Examples/fresnel_equations`

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 **Preset Studies>Frequency Domain**.
6 Click **Done**.

GLOBAL DEFINITIONS

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

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
n_air	1	1	Refractive index, air
n_slab	1.5	1.5	Refractive index, slab
lda0	1[m]	1 m	Wavelength
f0	c_const/lda0	2.998E8 1/s	Frequency
alpha	70[deg]	1.222 rad	Angle of incidence
beta	asin(n_air*sin(alpha)/ n_slab)	0.677 rad	Refraction angle
alpha_brewster	atan(n_slab/n_air)	0.9828 rad	Brewster angle, TM only
r_s	(n_air*cos(alpha)- n_slab*cos(beta))/ (n_air*cos(alpha)+ n_slab*cos(beta))	-0.5474	Reflection coefficient, TE
r_p	(n_slab*cos(alpha)- n_air*cos(beta))/ (n_air*cos(beta)+ n_slab*cos(alpha))	-0.2061	Reflection coefficient, TM
t_s	(2*n_air*cos(alpha))/ (n_air*cos(alpha)+ n_slab*cos(beta))	0.4526	Transmission coefficient, TE
t_p	(2*n_air*cos(alpha))/ (n_air*cos(beta)+ n_slab*cos(alpha))	0.5292	Transmission coefficient, TM

The angle of incidence is updated while running the parametric sweep. The refraction (transmitted) angle is defined by Snell's law with the updated angle of incidence. The Brewster angle exists only for TM incidence, p-polarization, and parallel polarization.

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 In the **Frequencies** text field, type `f0`.

GEOMETRY 1

First, create a block composed of two domains. Use layers to split the block.

Block 1 (blk1)

1 On 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 `0.2`.

4 In the **Depth** text field, type `0.2`.

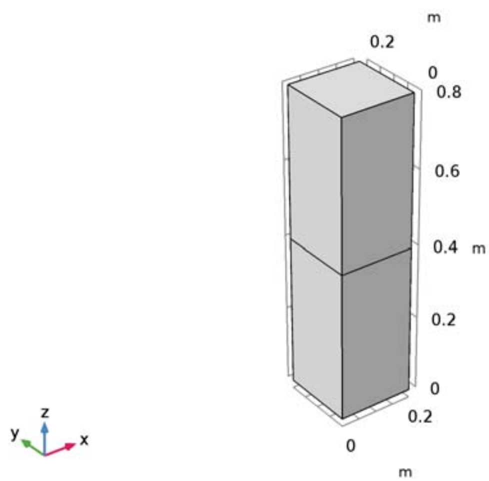
5 In the **Height** text field, type `0.8`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.4`

7 Click **Build All Objects**.

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



Choose wireframe rendering to get a better view of each boundary.

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Set up the physics based on the direction of propagation and the E-field polarization. First, assume a TE-polarized wave which is equivalent to s-polarization and perpendicular polarization. E_x and E_z are zero while E_y is dominant.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** node.

Port 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.
2 Select Boundary 7 only.
3 In the **Settings** window for **Port**, locate the **Port Properties** section.
4 From the **Type of port** list, choose **Periodic**.

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

5 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

0	x
1	y
0	z

6 Locate the **Automatic Diffraction Order Calculation** section. In the n text field, type `n_air`.
7 Locate the **Port Mode Settings** section. In the α_1 text field, type `alpha`The maximum frequency in the setting window will be used only when **Compute Diffraction Order** button is clicked to generate **Diffraction Order** features handling higher order mode individually. In this model, no diffraction is expected from the given geometry, so this setting is ineffective.

Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.
2 Select Boundary 3 only.
3 In the **Settings** window for **Port**, locate the **Port Properties** section.
4 From the **Type of port** list, choose **Periodic**.
5 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

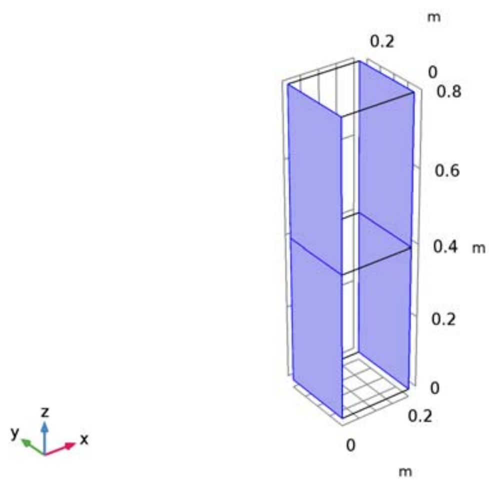
0	x
1	y
0	z

6 Locate the **Automatic Diffraction Order Calculation** section. In the n text field, type `n_slab`.

The bottom surface is an observation port. The S_{21}-parameter from Port 1 and Port 2 provides the transmission characteristics.

Periodic Condition 1

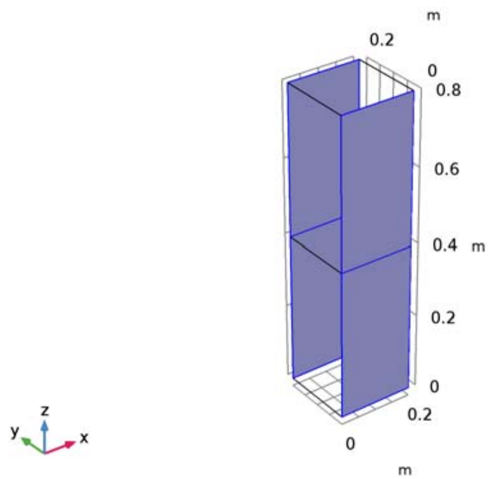
1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Periodic Condition**.
2 Select Boundaries 1, 4, 10, and 11 only.
3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.
4 From the **Type of periodicity** list, choose **Floquet periodicity**.
5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.



Periodic Condition 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Periodic Condition**.
2 Select Boundaries 2, 5, 8, and 9 only.
3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.



Wave Equation, Electric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** click **Wave Equation, Electric 1**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Refractive index**.

MATERIALS

Now set up the material properties based on refractive index. The top half is filled with air.

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**, type `Air` in the **Label** text field.

3 Select Domain 2 only.

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

Property	Name	Value	Unit	Property group
Refractive index	n	n_air	1	Refractive index

The bottom half is glass.

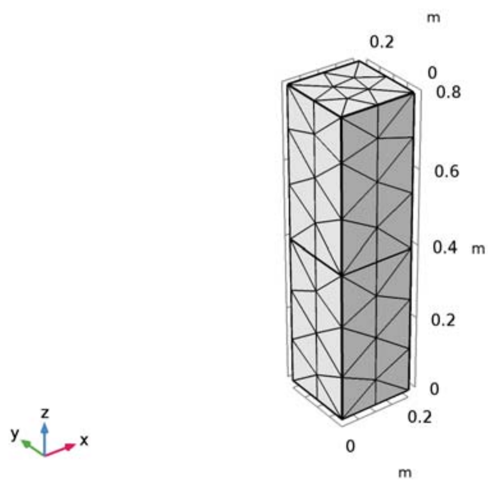
Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Glass` in the **Label** text field.
3 Select Domain 1 only.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Refractive index	n	n_slab	1	Refractive index

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alpha	range(0,2[deg],88[deg])	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot is the E-field norm for the last solution, which corresponds to tangential incidence. Replace the expression with E_y, add an arrow plot of the power flow (Poynting vector), and choose a more interesting angle of incidence for the plot.

Multislice

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

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Electric>Electric field>emw.Ey - Electric field, y component**.

3 Locate the **Multiplane Data** section. 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. From the **Color table** list, choose **Wave**.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Energy and power>emw.Poavx,...,emw.Poavz - Power flow, time average**.

3 Locate the **Arrow Positioning** section. Find the **Y grid points** subsection. In the **Points** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

Electric Field (emw)

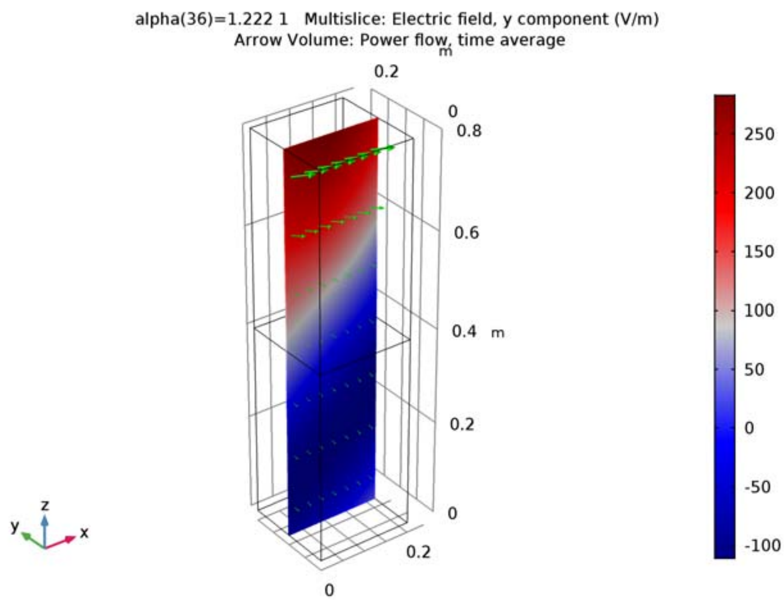
1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

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

3 From the **Parameter value (alpha (1))** list, choose **1.222**.

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

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



The plot should look like that in Figure 2.

1D Plot Group 2

Add a 1D plot to see the reflection and transmission versus the angle of incidence.

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

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

3 Select the **x-axis label** check box.

4 In the associated text field, type `Angle of Incidence`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Reflectance and Transmittance`.

7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

8 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 Right-click **1D Plot Group 2** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(emw.S11)^2`	1	`Reflectance`
`abs(emw.S21)^2`	1	`Transmittance`

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

1D Plot Group 2

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, type `Reflection and Transmission` in the **Label** text field.

Global 2

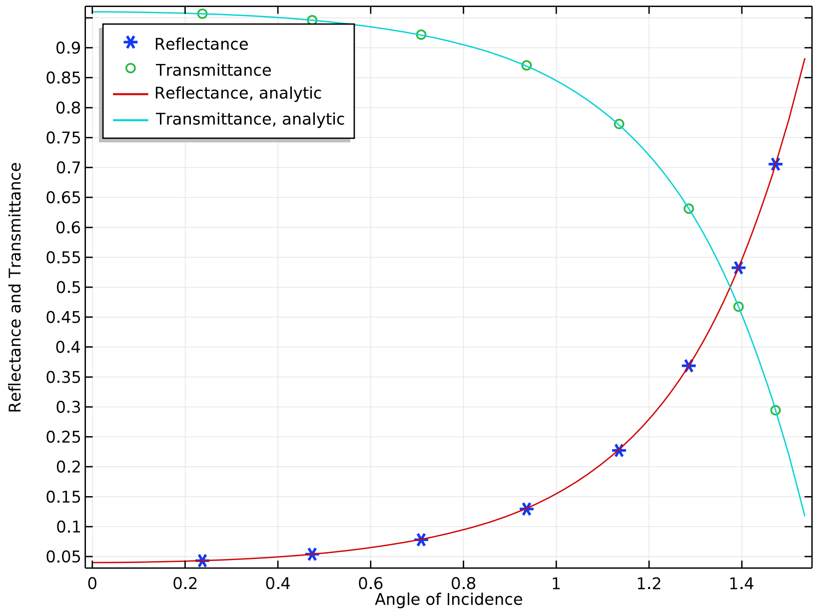
1 Right-click **Results>Reflection and Transmission** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(r_s)^2`		`Reflectance, analytic`
`n_slab*cos(beta)/(n_air* cos(alpha))*abs(t_s)^2`		`Transmittance, analytic`

4 On the **Reflection and Transmission** toolbar, click **Plot**.



Compare the resulting plots with Figure 4.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

The remaining instructions are for the case of TM-polarized wave, p-polarization, and parallel polarization. In this case, E_y is zero while E_x and E_z characterize the wave. In other words, H_y is dominant while H_x and H_z have no effect. Thus, the H-field is perpendicular to the plane of incidence and it is convenient to specify the port mode fields as the H-field.

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 **Port Mode Settings** section.

3 From the **Input quantity** list, choose **Magnetic field**.

4 Specify the $\mathbf{H}_0$ vector as

0	x
1	y
0	z

Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** click **Port 2**.

2 In the **Settings** window for **Port**, locate the **Port Mode Settings** section.

3 From the **Input quantity** list, choose **Magnetic field**.

4 Specify the $\mathbf{H}_0$ vector as

0	x
1	y
0	z

STUDY 1

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

2 Right-click **Solution 1 (sol1)** and choose **Solution>Copy**.

RESULTS

Electric Field (emw)

Ctrl-click to select both **Results>Electric Field (emw)** and **Results>Reflection and Transmission**, then right-click and choose **Duplicate**.

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

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

3 From the **Data set** list, choose **Study 1/Solution 1 - Copy 1 (sol2)**.

Reflection and Transmission

1 In the **Model Builder** window, under **Results** click **Reflection and Transmission**.

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

3 From the **Data set** list, choose **Study 1/Solution 1 - Copy 1 (sol2)**.

STUDY 1

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

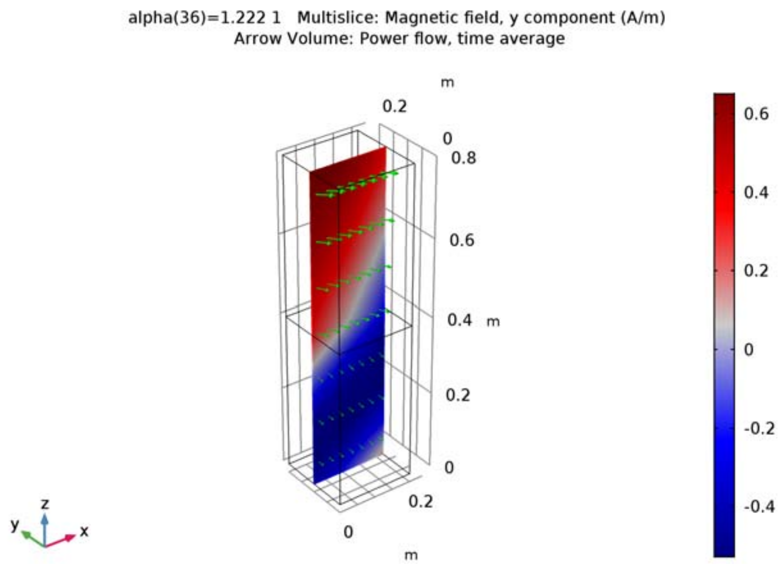
RESULTS

Multislice 1

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

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>emw.Hy - Magnetic field, y component**.

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



This reproduces Figure 3.

Global 2

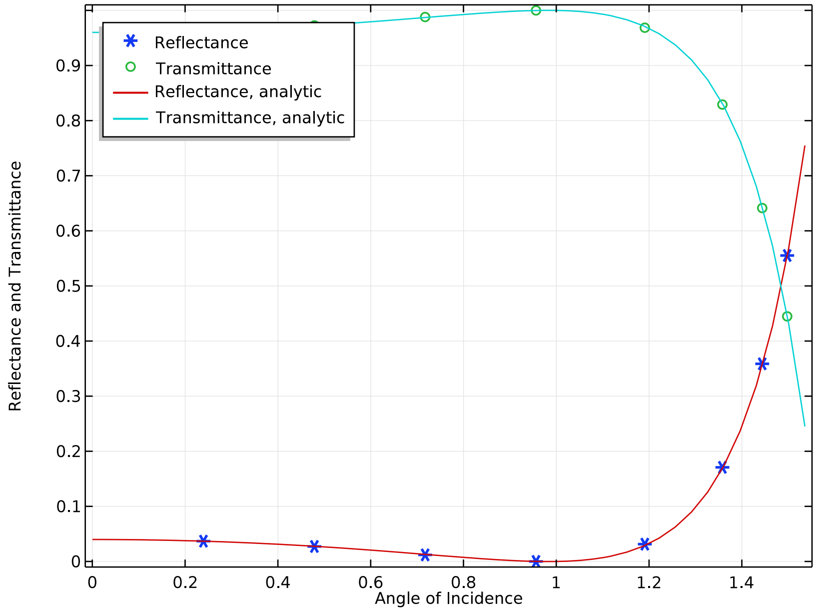
1 In the **Model Builder** window, expand the **Results>Reflection and Transmission 1** node, then click **Global 2**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(r_p)^2`		`Reflectance, analytic`
`n_slab*cos(beta)/(n_air* cos(alpha))*abs(t_p)^2`		`Transmittance, analytic`

4 On the **Reflection and Transmission 1** toolbar, click **Plot**.



The plot should look like Figure 5. The Brewster angle is observed around 56 degrees, which is close to the analytic value.

H-Bend Waveguide 2D

Introduction

This example is a 2D version of H-Bend Waveguide 3D, which shows how to model a bending rectangular waveguide for microwaves. For a general introduction, see the model H-Bend Waveguide 3D.

The dimensions of the waveguide and the frequency range used in this example are such that TE_{10} is the single propagating mode. In this mode, if the bend is in the xy-plane, the electric field only has a z-component, which furthermore is independent of the z-coordinate. This makes it possible to set up and solve the model in a 2D geometry.

Model Definition

The considered geometry is an xy-plane cross-section of the 3D geometry, as seen in Figure 1. This figure also sums up the material and boundary settings, which are the same as in the 3D model. The main advantage with setting up the model in 2D is that it solves much faster and uses less memory. As a consequence, this version of the example does not stress the need to adapt the mesh to the wavelength, but simply lets you apply a mesh that is more than fine enough.

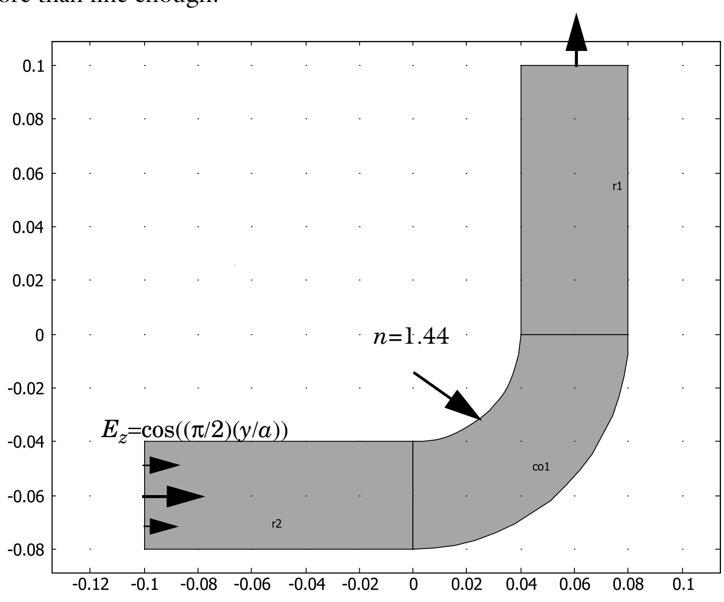


Figure 1: The geometry of the example.

Results and Discussion

Figure 2 shows the norm of the electric field at one of the frequencies where the bend has a resonance. The absence of a wave pattern in the input section indicates that the transmission is nearly perfect.

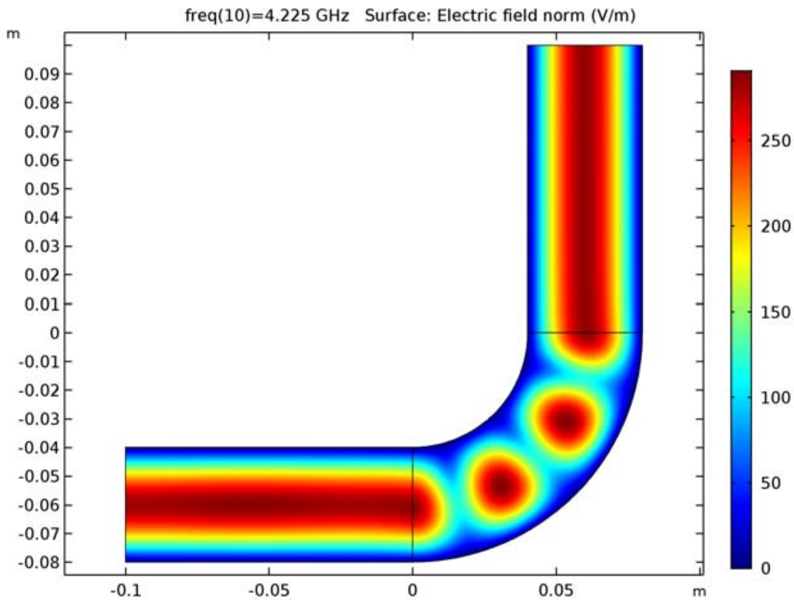


Figure 2: The electric field norm at a frequency of 4.225 GHz.

Figure 3 shows the S-parameters in a dB scale. The result agrees very well with that of the 3D model.

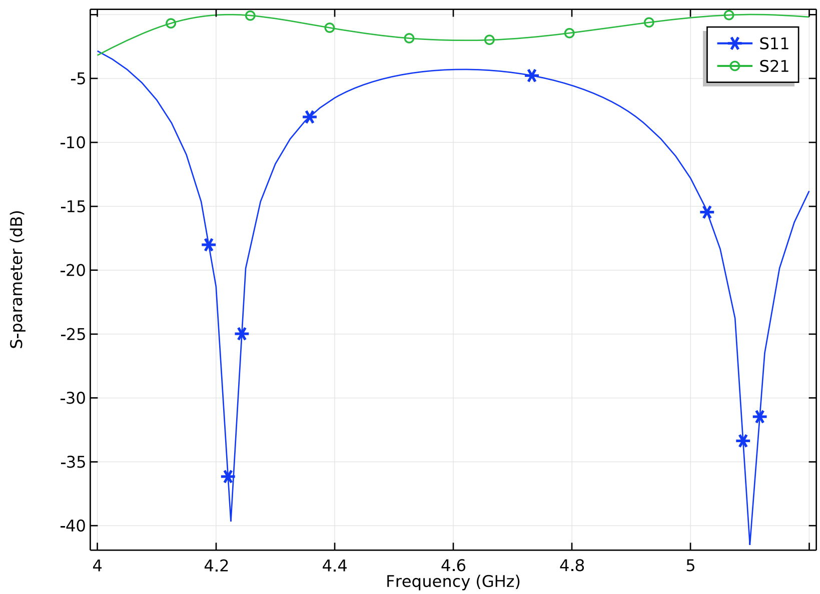


Figure 3: The S-parameters, in a dB scale, as functions of the frequency.

Application Library path: `RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_2d`

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 **2D**.

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 **Preset 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 In the **Frequencies** text field, type `range(4[GHz],25[MHz],5.2[GHz])`.

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.08`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.04`.

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `0.08`.

4 Locate the **Position** section. In the **y** text field, type `-0.08`.

Compose 1 (co1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Compose**.

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

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

4 In the **Set formula** text field, type `sq1*(c1-c2)`.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.04`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **x** text field, type `0.04`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.1`.

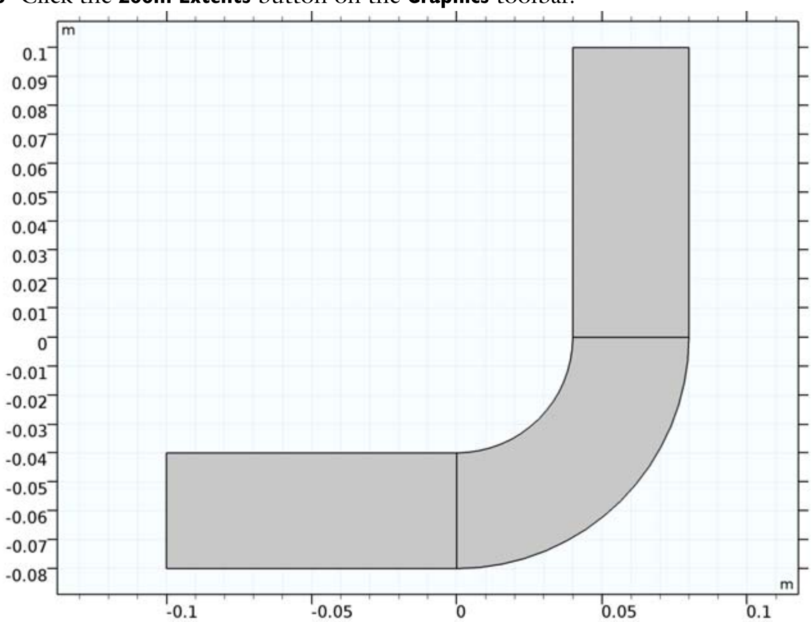
4 In the **Height** text field, type `0.04`.

5 Locate the **Position** section. In the **x** text field, type `-0.1`.

6 In the **y** text field, type `-0.08`.

7 Click **Build All Objects**.

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



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**, type `Air` in the **Label** text field.

3 Select Domains 1 and 3 only.

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Refractive Index> Refractive index, real part (n)**.

5 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Refractive index	n	1	1	Refractive index

Material 2 (mat2)

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

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

3 Select Domain 2 only.

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Refractive Index> Refractive index, real part (n)**.

5 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Refractive index	n	1.44	1	Refractive index

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

With TE waves, only the z component of the electric field needs to be solved for

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Components** section.

3 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Wave Equation, Electric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** click **Wave Equation, Electric 1**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Refractive index**.

The default boundary condition is perfect electric conductor, which is fine for all exterior boundaries except the ports. The software automatically imposes continuity on interior boundaries.

Port 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 1 only.

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

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

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

Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 7 only.

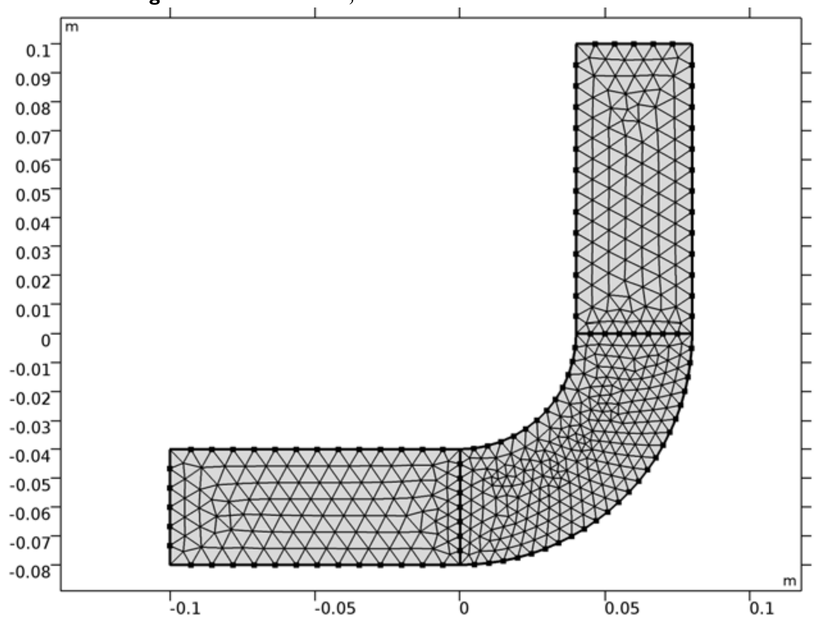
3 In the **Settings** window for **Port**, locate the **Port Properties** section.

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

MESH 1

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

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

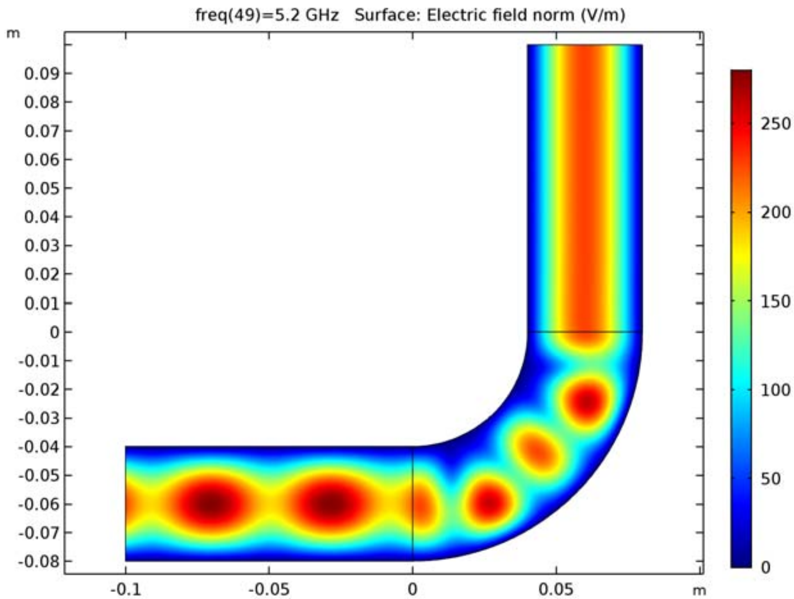


STUDY 1

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

RESULTS

Electric Field (emw)



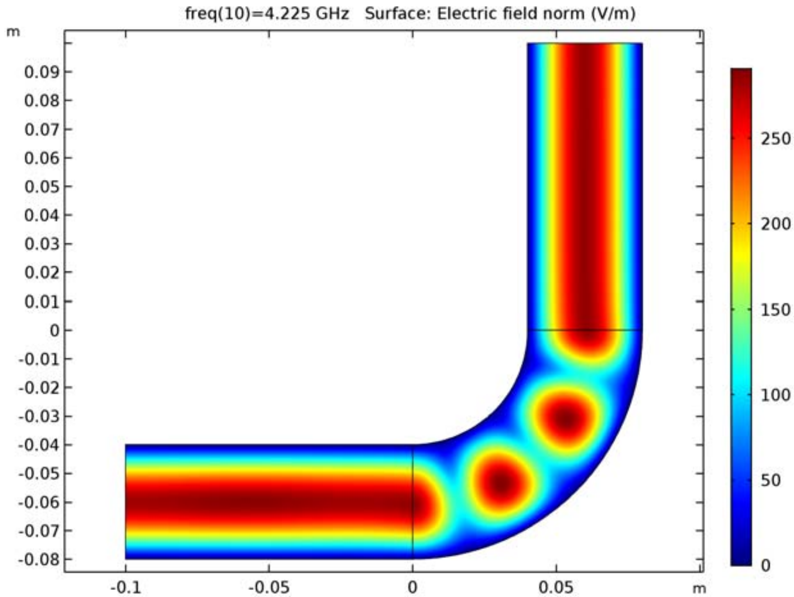
The default plot shows the norm of the electric field for the highest frequency, 5.2 GHz. To verify that the solution resembles the 3D version, try plotting a frequency where you expect a transmission peak.

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

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

3 From the **Parameter value (freq (GHz))** list, choose **4.225**.

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



Finally, plot the S-parameters.

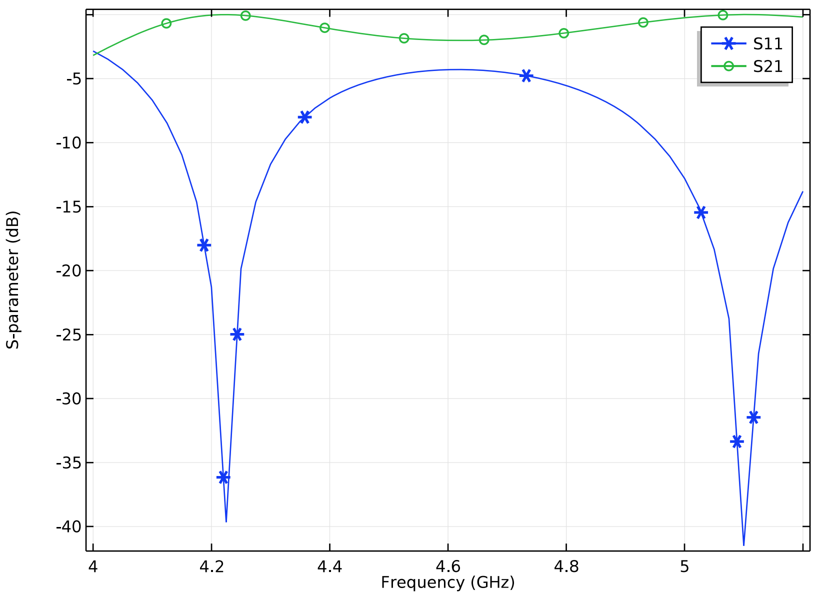
Global 1

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

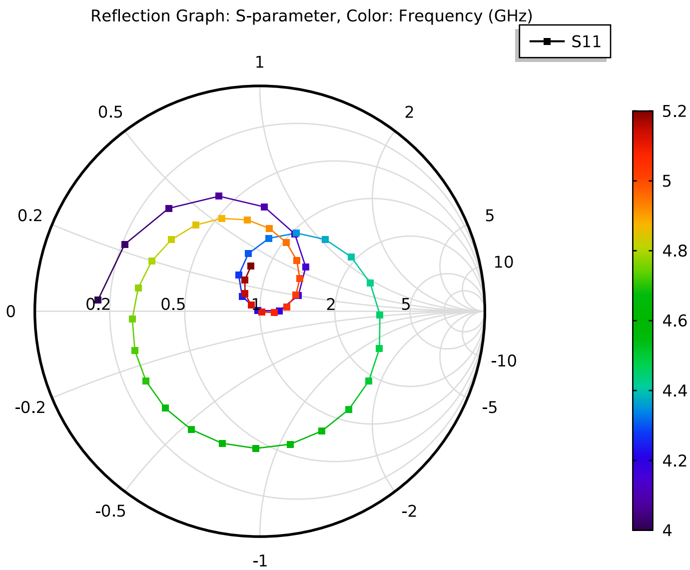
2 In the **Settings** window for **Global**, click to expand the **Coloring and style** section.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 On the **S-Parameter (emw)** toolbar, click **Plot**.



Smith Plot (emw)



Created in COMSOL Multiphysics 5.3

H-Bend Waveguide 3D

Introduction

This example shows how to model a rectangular waveguide for microwaves. A single hollow waveguide can conduct two kinds of electromagnetic waves: transversal magnetic (TM) or transversal electric (TE) waves. This example examines a TE wave, one that has no electric field component in the direction of propagation. More specifically, for this example you select the frequency and waveguide dimension so that TE_{10} is the single propagating mode. In that mode the electric field has only one nonzero component—a sinusoidal with two nodes, one at each of the walls of the waveguide. This makes it possible to set up and solve the model in 2D, which is done in a separate version; see H-Bend Waveguide 2D.

One important design aspect is how to shape a waveguide to go around a corner without incurring unnecessary losses in signal power. Unlike in wires, these losses usually do not result from ohmic resistance but instead arise from unwanted reflections. You can minimize these reflections by keeping the bend smooth with a large enough radius. In the range of operation the transmission characteristics (the ability of the waveguide to transmit the signal) must be reasonably uniform for avoiding signal distortions.

With air as the inside medium of the waveguide, the transmission is nearly perfect throughout the range of operation. In this example, to make the simulation and the results more interesting, the bend is filled with silica glass, a dielectric medium.

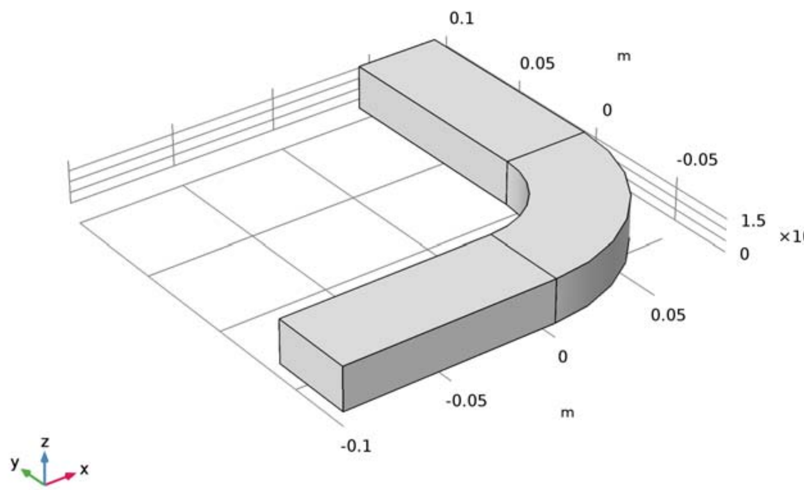
The model also shows how to systematically compute and export all S-parameters to a Touchstone file.

Model Definition

This example illustrates how to create a model that computes the electromagnetic fields and transmission characteristics of a 90° bend for a given radius. This type of waveguide bends changes the direction of the **H** field components and leaves the direction of the **E** field unchanged. The waveguide is therefore called an *H-bend*. The H-bend design used in this example is well-proven in real-world applications and you can buy similar waveguide bends online from a number of manufacturers. This particular bend performs optimally in the ideal case of perfectly conducting walls.

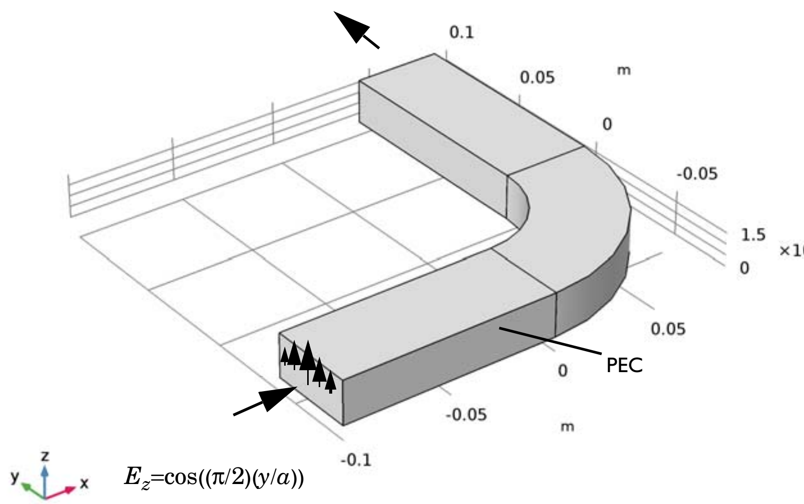
The waveguide walls are typically plated with a very good conductor, such as silver. In this example the walls are considered to be made of a perfect conductor, which means that the tangential component of the electric field is zero, or that $\mathbf{n} \times \mathbf{E} = \mathbf{0}$ on the boundaries. This boundary condition is referred to as a *perfect electric conductor* (PEC) boundary condition.

The geometry is as follows:



The waveguide is considered to continue indefinitely before and after the bend. This means that the input wave needs to have the form of a wave that has been traveling through a straight waveguide. The shape of such a wave is determined by the boundary conditions of Maxwell's equations on the sides of the metallic boundaries, that is, the PEC boundary condition. If polarized according to a TE_{10} mode, the shape is known analytically to be $\mathbf{E} = (0, 0, \sin(\pi(a-y)/(2a)))\cos(\omega t)$ given that the entrance boundary is centered around the $y = 0$ axis, and that the width of the waveguide, in the y direction, is $2a$.

The model is set up using the time-harmonic Electromagnetic Waves interface. This means that only the phasor component of the field is modeled. The incident field then has the form $\mathbf{E} = (0, 0, E_{0z}) = (0, 0, \sin(\pi(a-y)/(2a)))$, and is considered as part of the expression $\mathbf{E} = \mathrm{Re}\{(0, 0, \sin(\pi(a-y)/(2a))e^{j\omega t})\} = \mathrm{Re}\{\mathbf{E}e^{j\omega t}\}$, where complex-valued arithmetic has been used (also referred to as the $j\omega$ method).



The width of the waveguide is chosen so that it has a cutoff frequency of 3.7 GHz. This makes the waveguide operational up to 7.5 GHz. At higher frequencies other modes than the TE_{10} appear, causing a "dirty" signal. The input wave then splits into several modes that are hard to control without having large power losses. Below the cutoff frequency, no waves can propagate through the waveguide. This is an intrinsic property of microwave waveguides.

The cutoff frequency of different modes in a straight waveguide is given by the relation

$$(\nu_c)_{mn} = \frac{c}{2}\sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers ($m = 1, n = 0$ for the TE_{10} mode), a and b are the lengths of the sides of the waveguide cross-section, and c is the speed of light.

For this waveguide, $a = 2b$ and $b = 2$ cm.

The first few cutoff frequencies are $(\nu_c)_{10} = 3.7$ GHz, $(\nu_c)_{01} = 7.5$ GHz, $(\nu_c)_{11} = 8.4$ GHz. The frequencies used in this example are from 4.0 GHz to 5.2 GHz, and hence entirely within the single-mode range.

On the input boundary, the Port boundary condition lets you choose which mode to send in. Any reflected waves having the same shape are transmitted back through this same boundary. The output boundary also uses a Port condition, but without field excitation, to specify the shape of the wave that it lets pass through. Using port boundary conditions means that you automatically gain access to postprocessing variables for the S-parameters.

Results and Discussion

The wave is found to propagate through the bend with a varying amount of reflection depending on the frequency.

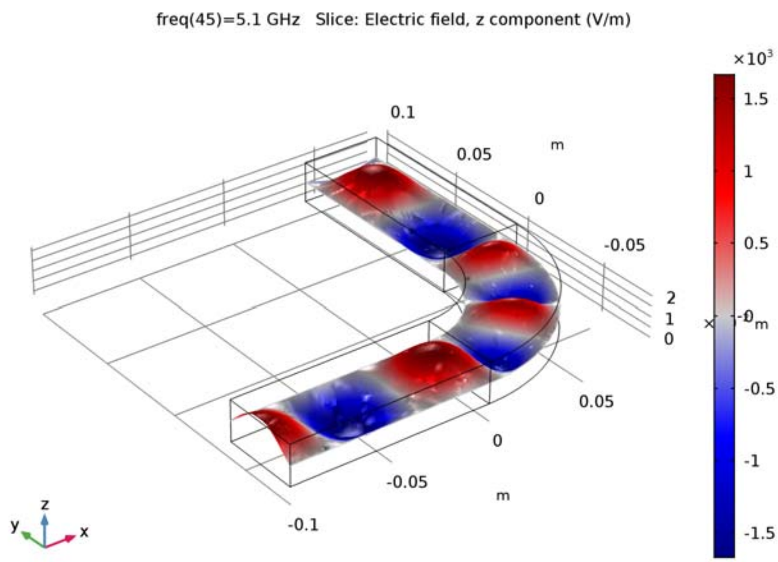


Figure 1: The z-component of the electric field for a frequency of 5.1 GHz.

The S-parameters are shown as functions of the frequency in Figure 2.

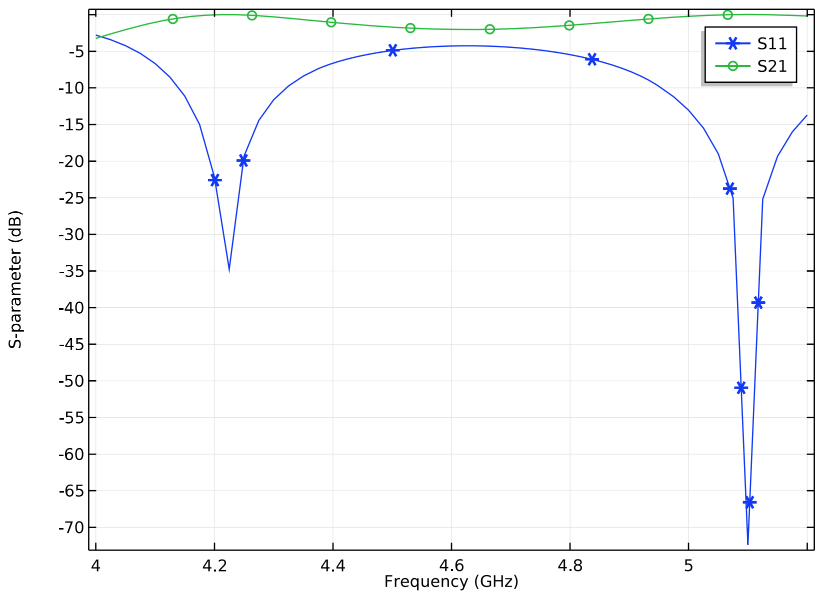


Figure 2: The S-parameters, on a dB scale, as a function of the frequency.

The two dips in S_{21} closely correspond to cavity resonances of the dielectric region in the bend. At these frequencies, the transmission is almost perfect. (Without the dielectric, the transmission would be nearly as good throughout the frequency range.)

Application Library path: `RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_3d`

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 **Preset 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 In the **Frequencies** text field, type `range(4[GHz],25[MHz],5.2[GHz])`.

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.08`.

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.04`.

Square 1 (sq1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `0.08`.

4 Locate the **Position** section. In the **yw** text field, type `-0.08`.

Compose 1 (co1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Compose**.

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

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

4 In the **Set formula** text field, type `sq1*(c1-c2)`.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.04`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **xw** text field, type `0.04`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.04`.

5 Locate the **Position** section. In the **xw** text field, type `-0.1`.

6 In the **yw** text field, type `-0.08`.

7 On the **Work Plane** toolbar, click **Build All**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

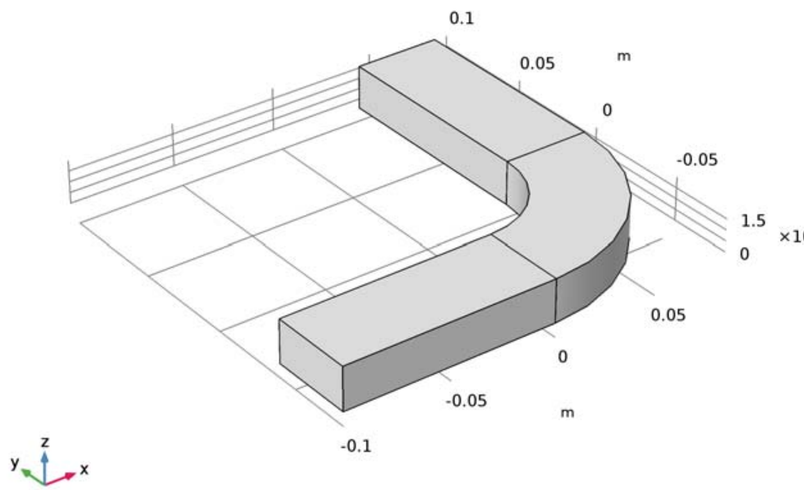
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
`0.02`

4 Click **Build All Objects**.

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



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**, type `Air` in the **Label** text field.

3 Select Domains 1 and 3 only.

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Refractive Index> Refractive index, real part (n)**.

5 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Refractive index	n	1	1	Refractive index

Material 2 (mat2)

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

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

3 Select Domain 2 only.

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index, real part (n)**.

5 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Refractive index	n	`1.44`	1	Refractive index

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Wave Equation, Electric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** click **Wave Equation, Electric 1**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Refractive index**.

Port 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 1 only.

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

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

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

Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 15 only.

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

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

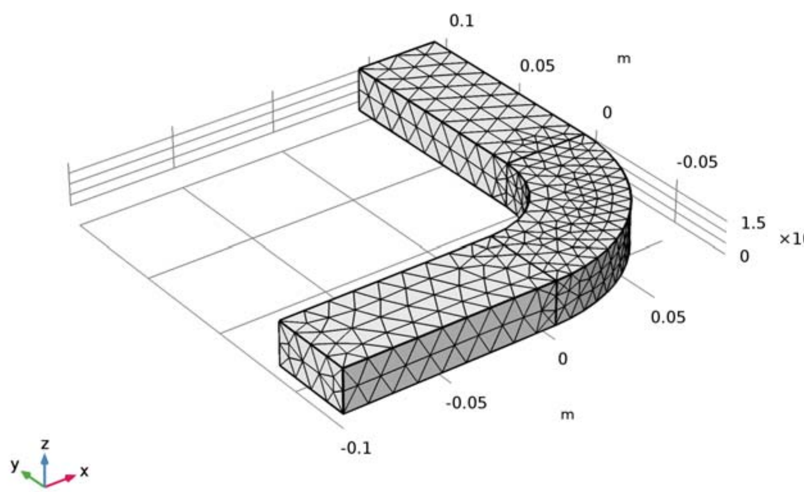
5 From the **Type of port** list, choose **Rectangular**.

The default boundary condition is perfect electric conductor, which is fine for all exterior boundaries except the ports. The software automatically imposes continuity on interior boundaries.

MESH 1

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

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



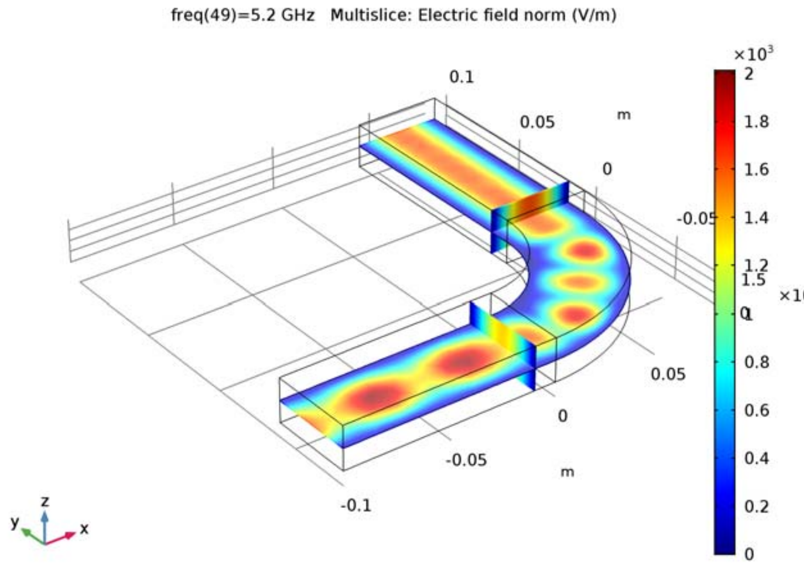
If you look closely at the mesh, you can see that it is indeed a bit finer in the bend than elsewhere.

STUDY 1

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

RESULTS

Electric Field (emw)



The default plot shows the distribution of the electric field norm on slices of the waveguide, for the highest frequency in the sweep. Note the wave pattern in the bend and the rectangular input section. This indicates standing waves caused by reflections in the bend. In contrast, the pattern beyond the bend is independent of the y-coordinate, showing that the output port does a good job of transmitting the wave.

An S-parameter plot gives you a quantitative measure of how much of the wave is transmitted and reflected at different frequencies.

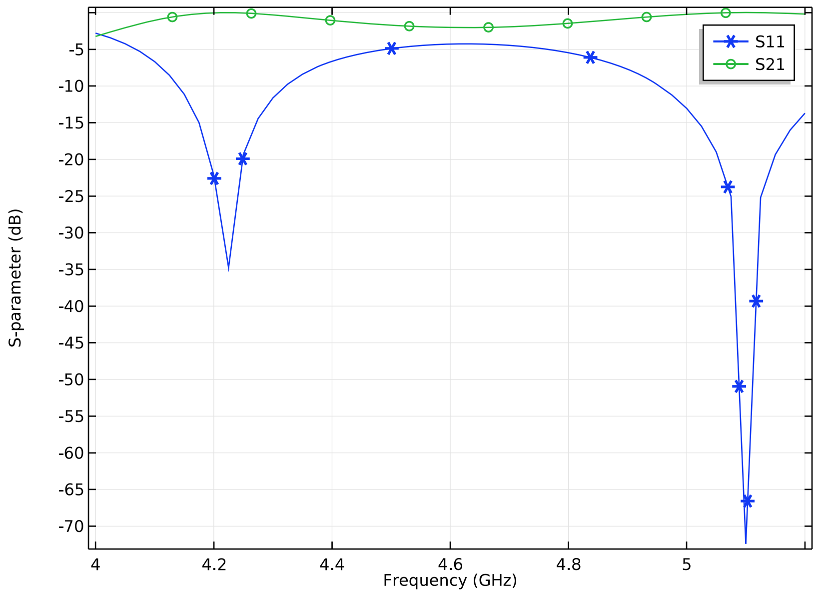
Global 1

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

2 In the **Settings** window for **Global**, click to expand the **Coloring and style** section.

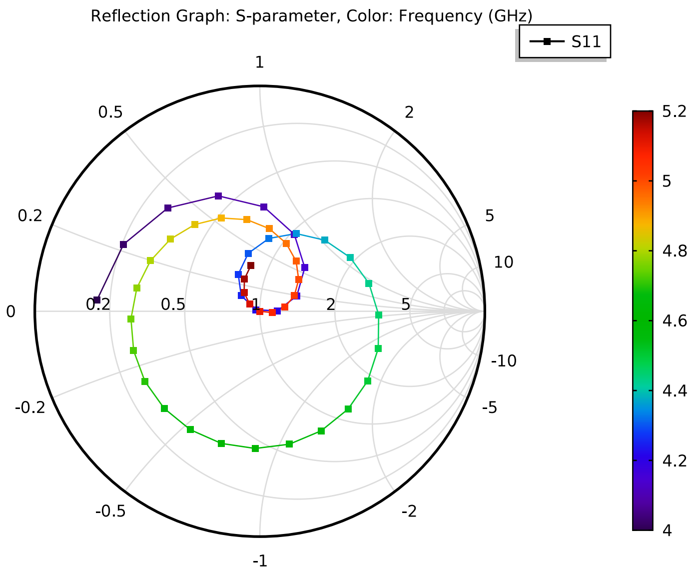
3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 On the **S-Parameter (emw)** toolbar, click **Plot**.



The result, which should look like Figure 2, shows that the transmission varies throughout the frequency range. Note in particular that S_{21} has two deep dips, corresponding to almost perfect transmission. This is the result of resonances in the bend. To confirm this, try looking at the field distribution for the frequency where the upper peak is located, 5.1 GHz.

Smith Plot (emw)



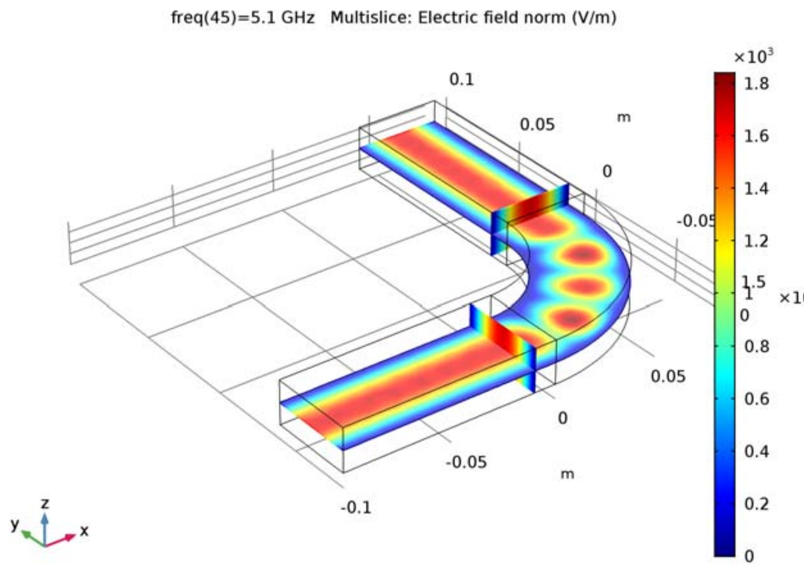
Electric Field (emw)

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

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

3 From the **Parameter value (freq (GHz))** list, choose **5.1**.

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



The standing wave pattern still remains in the bend, but at this frequency it is almost completely gone in the input section.

For an alternative view, you can plot the instantaneous value of the electric field inside the waveguide. Only the z component will be substantially non-zero. For a better view, add also deformation. Replace the Multislice with a single horizontal slice plot.

Multislice

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

Slice 1

1. Right-click **Multislice** and choose **Delete**.
2. In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Slice**.
3. In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Electric>Electric field>emw.Ez - Electric field, z component**.
4. Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.
5. In the **Planes** text field, type 1.
6. Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

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

The Wave color table looks its best using a symmetric range. You can also play with a deformed shape plot to make the waves appear more clearly.

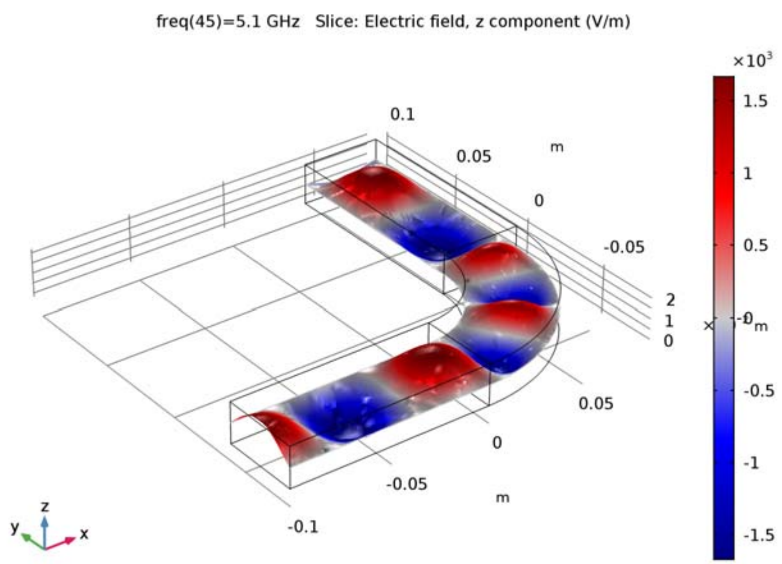
8 Click to expand the **Range** section. Locate the **Coloring and Style** section. Select the **Symmetrize color range** check box.

Deformation 1

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

2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Electric>emw.Ex,emw.Ey,emw.Ez - Electric field**.

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



The remaining instructions show you how to systematically solve with one port active at a time, and save the results in the Touchstone format.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Port Sweep Settings** section.

3 Select the **Activate port sweep** check box.

Click the Browse button and select a file to which you want to export the results. If the file does not exist, it will be created.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
PortName	1	1	Port name

STUDY 1

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
PortName	1 2	

The parameter is the same as the name suggested by the port sweep. The parameter values should be the same as your port numbers.

5 On the **Study** toolbar, click **Compute**.

RESULTS

The Touchstone file should now contain the complete output from the model. The new solution data set contains two frequency sweeps, one for each port.

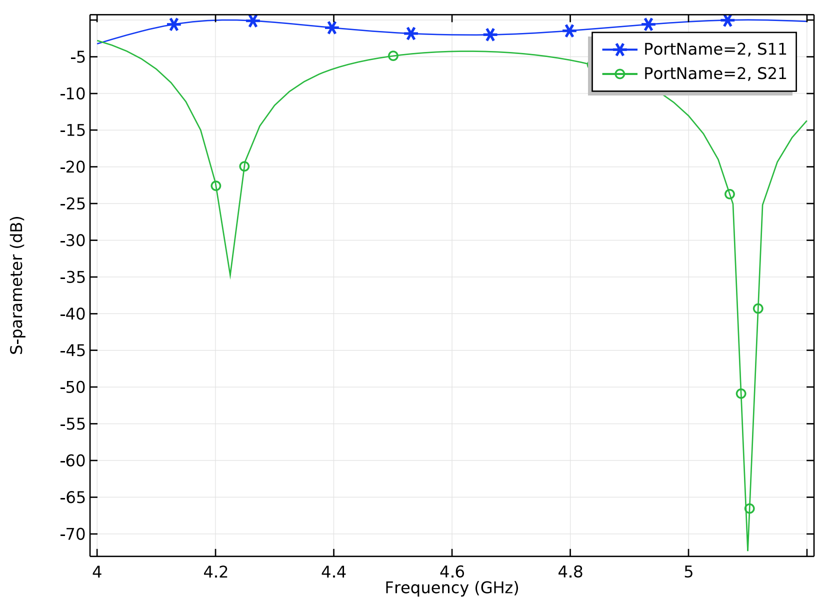
Global 1

As you can see, after performing the parametric sweep over the ports, the S-parameter plot you created previously is empty. To restore the plot, you need to change the data set and specify the inner parameter - that is, the frequency - as the quantity to display along the horizontal axis.

1 In the **Model Builder** window, under **Results>S-Parameter (emw)** click **Global 1**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

To verify the reciprocity of the waveguide, you can add the S-parameters S12dB and S22dB to the Expressions table and change the parameter selection for PortName:

4 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S12dB - S12**.
5 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S22dB - S22**.
6 Locate the **Data** section. From the **Parameter selection (PortName)** list, choose **Last**.
7 On the **S-Parameter (emw)** toolbar, click **Plot**.



Smith Plot (emw)

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

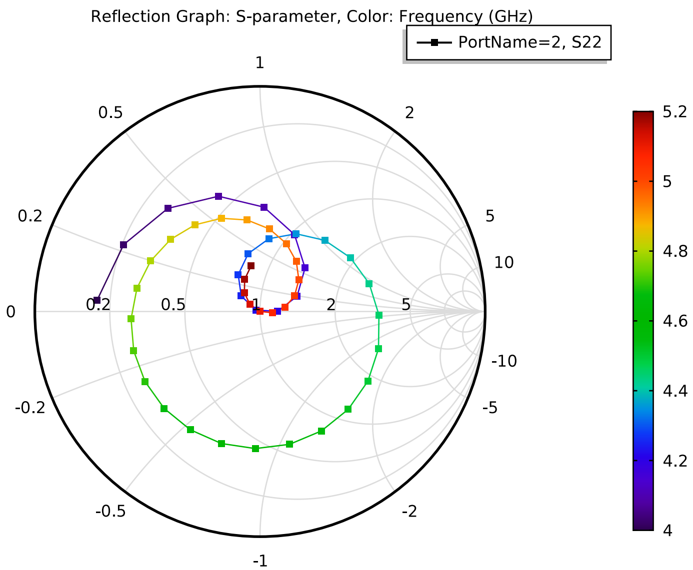
4 From the **Parameter selection (PortName)** list, choose **Last**.

Reflection Graph 1

1 In the **Model Builder** window, expand the **Smith Plot (emw)** node, then click **Reflection Graph 1**.

2 In the **Settings** window for **Reflection Graph**, locate the **Expressions** section.

3 Click **emw.S22 - S22** in the upper-right corner of the section. On the **Smith Plot (emw)** toolbar, click **Plot**.



Reflection Graph 1

1 In the **Model Builder** window, expand the **Smith Plot (emw) 1** node, then click **Reflection Graph 1**.

2 In the **Settings** window for **Reflection Graph**, click to expand the **Coloring and style** section.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

   Finish by verifying the reciprocity of the waveguide on the Smith plot.

Created in COMSOL Multiphysics 5.3

Impedance Matching of a Lossy Ferrite 3-Port Circulator

Introduction

A microwave circulator is a nonreciprocal multiport device. It has the property that a wave incident on port 1 is routed into port 3 yet a wave incident on port 3 is not routed back into port 1 but is instead routed into port 2, and so on. This property of a circulator is used to isolate microwave components from each other, for example, when connecting a transmitter and a receiver to a common antenna. By connecting the transmitter, receiver, and antenna to different ports of a circulator, the transmitted power is routed to the antenna whereas any power received by the antenna goes into the receiver. Circulators typically rely on the use of ferrites, a special type of highly permeable and low-loss magnetic material that is anisotropic for a small RF signal when biased by a much larger static magnetic field. In the example, a three-port circulator is constructed from three rectangular waveguide sections joining at 120° and with a ferrite post inserted at the center of the joint.

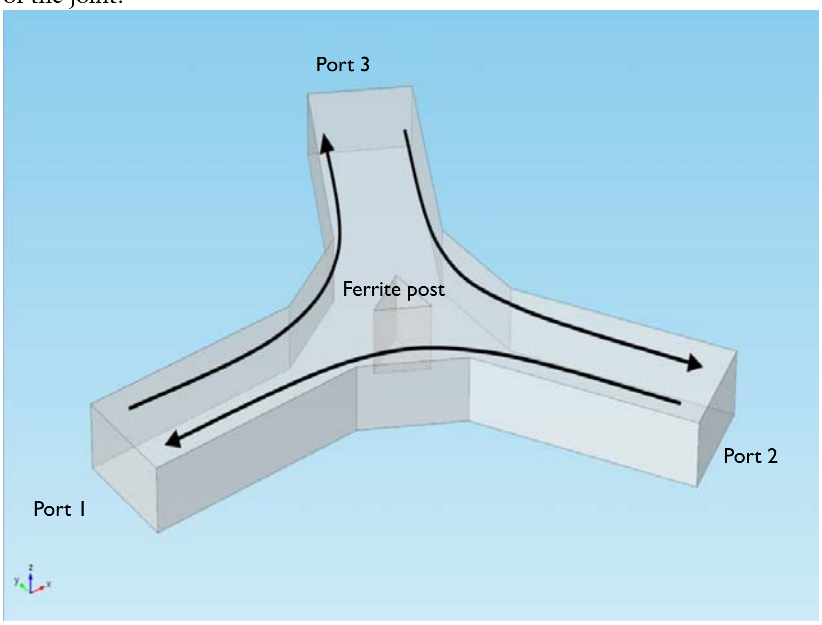


Figure 1: The post is magnetized by a static H_0 bias field along its axis. The bias field is supplied by external permanent magnets which are not explicitly modeled in this example.

IMPEDANCE MATCHING

An important step in the design of any microwave device is to match its input impedance for a given operating frequency. Impedance matching is equivalent to minimizing the reflections back to the inport. The parameters that need to be determined are the size of the ferrite post and the width of the wider waveguide section surrounding the ferrite. In this tutorial, these are varied in order to minimize the reflectance. The scattering parameters (S-parameters) used as measures of the reflectance and transmittance of the circulator are automatically computed.

The nominal frequency for the design of the device is chosen as 3 GHz. The circulator can be expected to perform reasonably well in a narrow frequency band around 3 GHz, and so a frequency range of 2.8 – 3.2 GHz is studied. It is desired that the device operates in single mode. Thus a rectangular waveguide cross section of 6.67 cm by 3.33 cm is selected to set the cut-off frequency for the fundamental TE_{10} mode to 2.25 GHz. The cut-off frequencies for the two nearest higher modes, the TE_{20} and TE_{01} modes, are both at 4.5 GHz, leaving a reasonable safety margin.

Model Definition

One of the rectangular ports is excited by the fundamental TE_{10} mode. At the ports, the boundaries are transparent to the TE_{10} mode. The following equation applies to the electric field vector $\mathbf{E}$ inside the circulator:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2\left(\varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0}\right)\mathbf{E} = 0$$

where μ_r denotes the relative permeability tensor, ω is the angular frequency, σ is the conductivity tensor, ε_0 is the permittivity of vacuum, ε_r is the relative permittivity tensor, and k_0 is the free space wave number. In this particular model, the conductivity is zero everywhere. Losses in the ferrite are introduced as complex-valued permittivity and permeability tensors. The magnetic permeability is of key importance as it is the anisotropy of this parameter that is responsible for the nonreciprocal behavior of the circulator. For simplicity, the rather complicated material expressions are predefined in a text file that is imported into the model. The expressions are also included in the next section for reference.

THE LOSSY FERRITE MATERIAL MODEL

Complete treatises on the theory of magnetic properties of ferrites can be found in Ref. 1 and Ref. 2. The model assumes that the static magnetic bias field, H_0, is much stronger than the alternating magnetic field of the microwaves, so the quoted expressions are a linearization for a small-signal analysis around this operating point. Under these assumptions, and including losses, the anisotropic permeability of a ferrite magnetized in the positive z direction is given by:

$$[\mu] = \begin{bmatrix} \mu & j\kappa & 0 \\ -j\kappa & \mu & 0 \\ 0 & 0 & \mu_0 \end{bmatrix}$$

where

$$\kappa = -j\mu_0\chi_{xy}$$

$$\mu = \mu_0(1+\chi_{xx})$$

and the unique elements of the magnetic susceptibility tensor χ are given by:

$$\chi_{xx} = \frac{\omega_0\omega_m(\omega_0^2 - \omega^2) + \omega_0\omega_m\omega^2\alpha^2}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2} - j\frac{\alpha\omega\omega_m(\omega_0^2 + \omega^2(1+\alpha^2))}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2}$$

$$\chi_{xy} = \frac{2\omega_0\omega_m\omega^2\alpha}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2} + j\frac{\omega\omega_m(\omega_0^2 - \omega^2(1+\alpha^2))}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2}$$

where

$$\omega_0 = \mu_0\gamma H_0$$

$$\omega_m = \mu_0\gamma M_s$$

$$\alpha = \frac{\mu_0\gamma\Delta H}{2\omega}$$

Here μ_0 denotes the permeability of free space; ω is the angular frequency of the microwave field; ω_0 is the precession resonance frequency (Larmor frequency) of a spinning electron in the applied magnetic bias field, H_0; ω_m is the electron Larmor frequency at the saturation magnetization of the ferrite, M_s; and γ is the gyromagnetic ratio of the electron. For a lossless ferrite ($\alpha = 0$), the permeability becomes infinite at $\omega = \omega_0$. In a lossy ferrite ($\alpha \neq 0$), this resonance becomes finite and is broadened. The loss factor, α, is related to the line width, ΔH, of the susceptibility curve near the resonance as given by the last expression above. The material data,

$$M_s = 5.41\cdot 10^4 \text{ A/m}, \varepsilon_r = 14.5$$

with an effective loss tangent of $2\cdot 10^{-4}$ and $\Delta H = 3.18\cdot 10^3$ A/m, are taken for aluminum garnet from Ref. 2. The applied bias field is set to $H_0 = 7.96\cdot 103$ A/m. The electron gyromagnetic ratio taken from Ref. 2 is $1.759 \cdot 10^{11}$ C/kg.

Results and Discussion

The default multislice plot shows the electric field norm. The electric field norm gives a good indication of where the main power is flowing and where there are standing waves due to reflections from the impedance mismatch at the center.

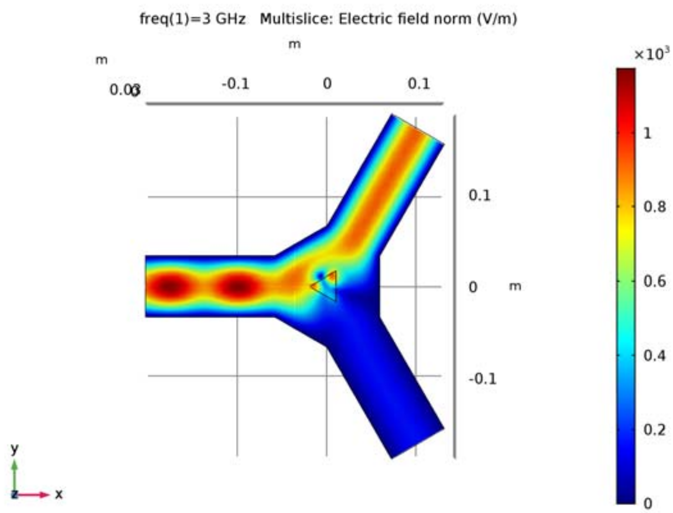


Figure 2: The default electric field norm plot shown on xy-plane.

The plot of the S-parameter from the parametric sweep of `sc_ferrite` indicates a minimum for a scale factor of 0.518.

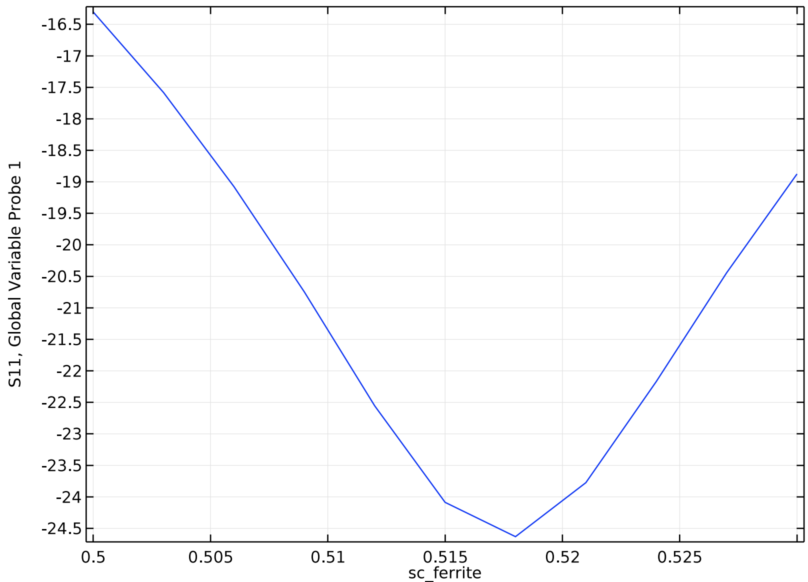


Figure 3: S-parameter as a function of sc_ferrite parameter

The plot of the S-parameter from the parametric sweep of `sc_chamfer` indicates a minimum for a scale factor of about 3.0.

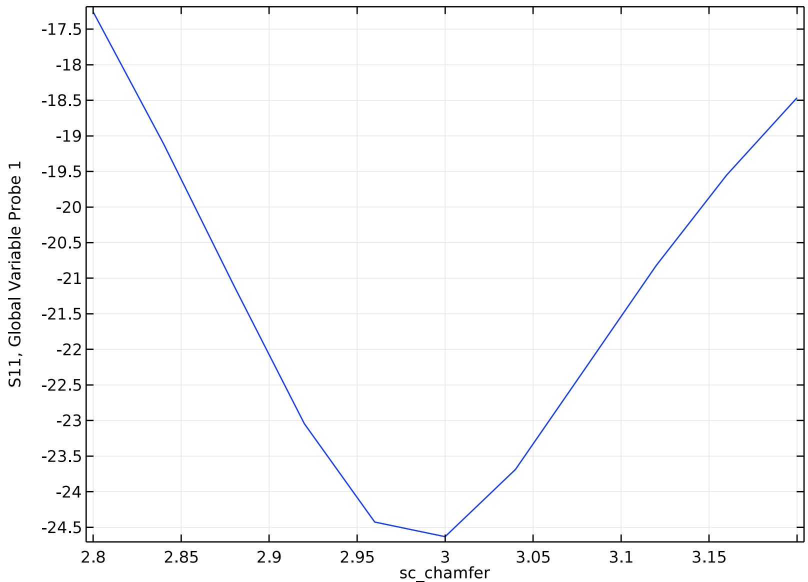


Figure 4: S-parameter as a function of sc_chamfer parameter

At the center frequency most of the standing waves are gone with the optimized values of sc_ferrite and sc_chamfer.

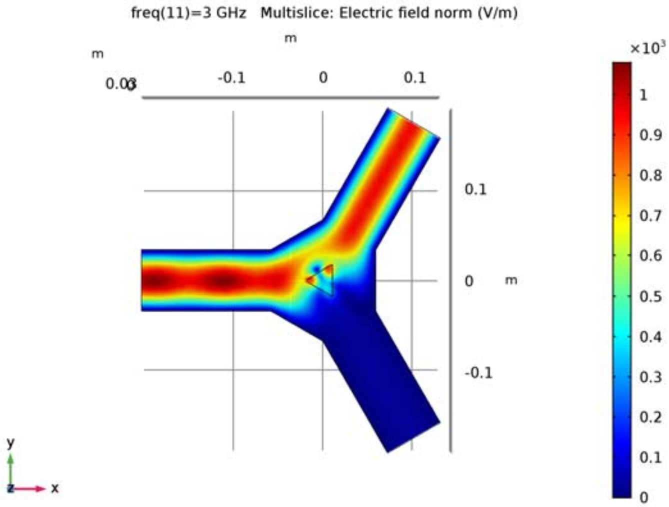


Figure 5: Electric field norm plot with the optimized sc_ferrite and sc_chamfer values.

This is the frequency response of the final design.

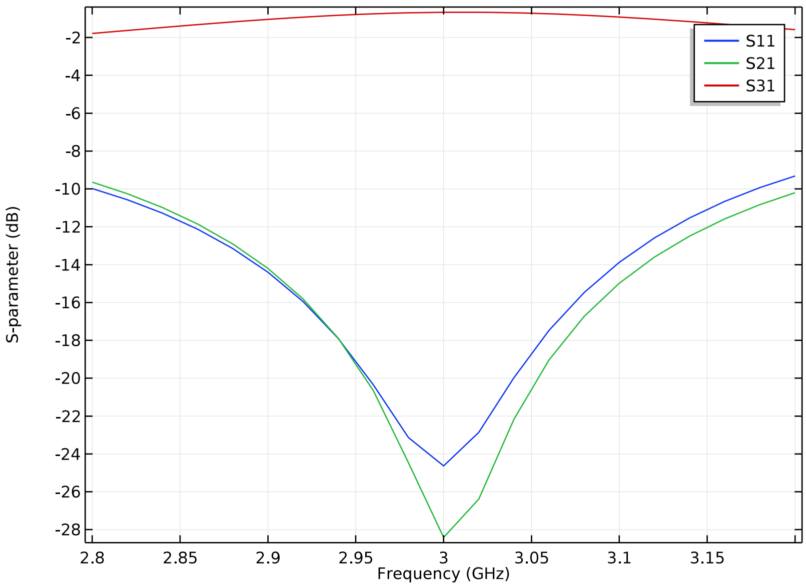


Figure 6: S-parameter as a function of frequency with the optimized sc_ferrite and sc_chamfer values.

From the below plot, it should be possible to identify the model at first glance so it has to display the geometry and some characteristic simulation results.

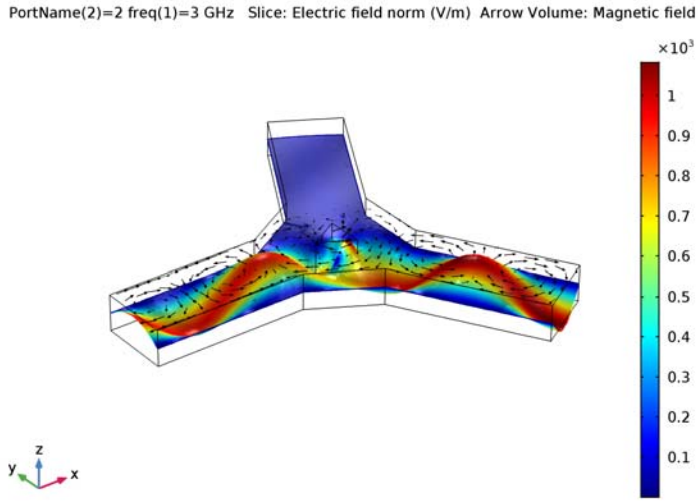


Figure 7: 3D plot used for model thumbnail generation

Reference

1. R.E. Collin, *Foundations for Microwave Engineering*, 2nd ed., IEEE Press/Wiley-Interscience, 2000.

2. D.M. Pozar, *Microwave Engineering*, 3rd ed., John Wiley & Sons Inc, 2004.

Application Library path: `RF_Module/Ferrimagnetic_Devices/lossy_circulator_3d`

Modeling Instructions

From the **File** menu, choose **Open**.

Browse to the model's Application Libraries folder and double-click the file `lossy_circulator_3d_geom.mph`.

GEOMETRY 1

Form Union (fin)

Next add material settings to the model. The lossy ferrite does not fit easily into the material settings so it will be taken care of later. Air is the only material to enter here.

ADD MATERIAL

1 On 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.

MATERIALS

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

Air (mat1)

In the Electromagnetic Waves interface, the ferrite is entered as a separate, user-defined equation model referring to the global variables defined above.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

In the **Model Builder** window, expand the **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** node.

Wave Equation, Electric 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Wave Equation, Electric**.

2 Select Domain 2 only.

3 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

4 From the **Electric displacement field model** list, choose **Dielectric loss**.

5 From the ε' list, choose **User defined**. In the associated text field, type `eps_r_p`.

6 From the ε'' list, choose **User defined**. In the associated text field, type `eps_r_b`.

7 Locate the **Magnetic Field** section. From the μ_r list, choose **User defined**. From the list, choose **Anisotropic**.

8 In the μ_r table, enter the following settings:

murxx	murxy	murxz

`muryx`	`muryy`	`muryz`
`murzx`	`murzy`	`murzz`

9 Locate the **Conduction Current** section. From the σ list, choose **User defined**. In the **Model Builder** window, click **Electromagnetic Waves, Frequency Domain (emw)**.

10 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Physics-Controlled Mesh** section.

11 Clear the **Enable** check box.

One inport for excitation and two outports need to be added next.

Port 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 1 only.

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

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

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

Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 18 only.

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

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

Port 3

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 19 only.

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

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

The mesh needs to resolve the local wavelength and, for lossy domains, the skin depth. The skin depth in the ferrite is large so the main concern is to resolve the local wavelength. This is done by providing maximum mesh sizes per domain. The rule of thumb is to use a maximum element size that is one fifth of the local wavelength (at the maximum frequency) or smaller.

MESH 1

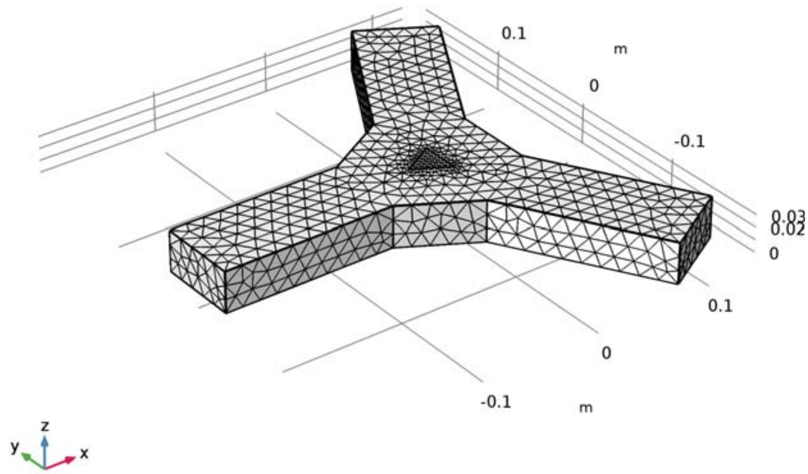
Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.
2. Right-click **Free Tetrahedral 1** and choose **Size**.
3. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
4. From the **Geometric entity level** list, choose **Domain**.
5. Select Domain 1 only.
6. Locate the **Element Size** section. Click the **Custom** button.
7. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
8. In the associated text field, type `1.5e-2`.

Size 2

1. Right-click **Free Tetrahedral 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
5. Select Domain 2 only.
6. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7. In the associated text field, type `4.5e-3`.
8. In the **Model Builder** window, click **Mesh 1**.

9 In the **Settings** window for **Mesh**, click **Build All**.



The mesh should now look as in the above figure.

The final step in the model set up is to solve it for the nominal frequency and inspect the results for possible modeling errors.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then 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 `3[GHz]`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows a slice plot of the electric field norm. It is best viewed from above.

1 Click the **Go to XY View** button on the **Graphics** toolbar.

The electric field norm gives a good indication on where the main power is flowing and where there are standing waves due to reflections from the impedance mismatch at the center. See Figure 2.

The remaining work is to vary the two design parameters in order to minimize reflections at the nominal frequency. To do this, perform parametric sweeps over the design parameters (scale factors). To avoid accumulating a lot of data while solving, throw away the solution and log only the S-parameter representing reflection in a table. For this purpose, add a global probe to the model.

DEFINITIONS

Global Variable Probe 1 (var1)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

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

STUDY 1

Modify the study in order to vary the scale factor determining the size of the ferrite post. The study type is still Frequency Domain.

The parametric sweep over the scale factor is added as an extension to the frequency domain study.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sc_ferrite	`range(0.5,3e-3,0.53)`	

5 Locate the **Output While Solving** section. Select the **Accumulated probe table** check box.

6 Find the **Memory settings for jobs** subsection. From the **Keep solutions in memory** list, choose **Only last**.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Probe Plot Group 2

The probe with the reflection coefficient versus the scale parameter is automatically logged to a table and plotted while solving. A dedicated 1D plot group is also created, but it plots the S-parameter versus frequency. To plot versus the geometry parameter, proceed as follows.

TABLE

1 Go to the **Table** window.

   In Accumulated Probe Table 1, delete column number 2 from the left with heading freq.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

Compare with the plot shown Figure 3. The plot of the S-parameter indicates a minimum for a scale factor of 0.518, so freeze the parameter at this value and add a new study for varying the next scale factor.

1 In the **Model Builder** window, expand the **Results>Tables** node.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
sc_ferrite	0.518	0.518	Geometry scale factor

STUDY 1

Parametric Sweep

1 In the **Model Builder** window, under **Study 1** click **Parametric Sweep**.

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

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sc_chamfer	`range(2.8,0.04,3.2)`	

4 On the **Study** toolbar, click **Compute**.

RESULTS

Probe Plot Group 2

Again, the probe with the reflection coefficient versus the frequency is automatically logged to a table and plotted while solving. To get the desired plot versus the geometry parameter, proceed as follows.

TABLE

1 Go to the **Table** window.

In Accumulated Probe Table 1, delete column number 2 from the left with heading freq.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

See Figure 4. The plot of the S-parameter indicates a minimum for a scale factor of about 3.0, so leave the parameter at this value and add a study for the frequency response.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Port Sweep Settings** section.

3 Select the **Activate port sweep** check box.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
PortName	1	1	Port name

ADD STUDY

1. On the **Study** 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> Frequency Domain**.
4. Click **Add Study** in the window toolbar.
5. On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 2** 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(2.8[GHz],20[MHz],3.2[GHz])`.
4. On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw) 1

At the final frequency, there are pronounced standing waves. Change to the center frequency.

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

   In the reproduced Figure 5 most of the standing waves are gone at the center frequency.

Finally plot all the S-parameters as a function of frequency.

Global 1

1. In the **Model Builder** window, expand the **S-Parameter (emw)** node, then click **Global 1**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 Click to select row number 2 in the table.

4 Click **Delete**.

5 Click **Delete**.

6 Click to select row number 3 in the table.

7 Click **Delete**.

8 Click **Delete**.

9 Click to select row number 4 in the table.

10 Click **Delete**.

11 Click **Delete**.

The table should now only contain `emw.S11dB`, `emw.S21dB` and `emw.S31dB`.

12 On the **S-Parameter (emw)** toolbar, click **Plot**.

Reproduce Figure 6. This is the frequency response of the final design.

Now, let the solver excite one port at a time in order to get the full S-parameter matrix exported to a Touchstone file for potential use in a system simulation tool. The necessary steps are as follows:

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

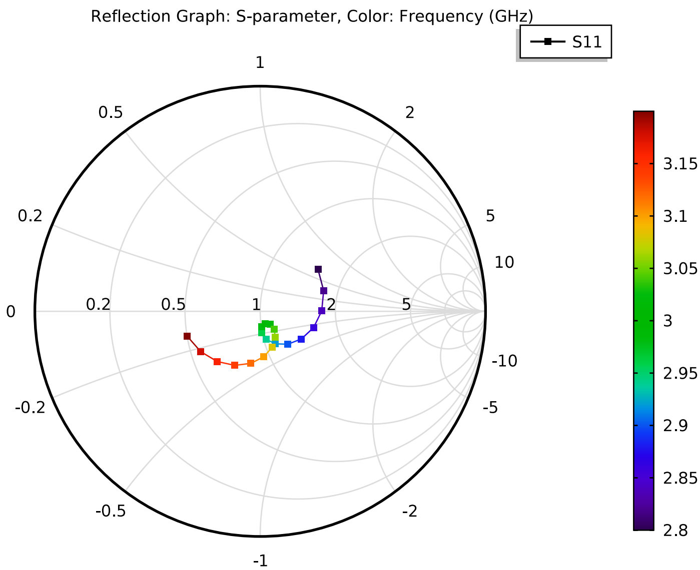
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Port Sweep Settings** section.

3 In the **Touchstone file export** text field, type `lossy_circulator_3d.s3p`.

RESULTS

Smith Plot (emw)



Reuse the first study for the port sweep. The study is solved for a single frequency to keep down simulation time though it is possible to solve for a range of frequencies.

STUDY 1

Parametric Sweep

The parametric sweep is used to control which port is excited. It overrides the settings on individual port features and drives one port at a time using 1 W of input power.

1 In the **Model Builder** window, under **Study 1** click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
PortName	1 2 3	

4 Locate the **Output While Solving** section. Find the **Memory settings for jobs** subsection. From the **Keep solutions in memory** list, choose **All**.
5 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw) 2

Display the S-parameter matrix in a table.

Global Matrix Evaluation 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Other> Global Matrix Evaluation**.

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.SdB - S-parameter, dB**.

5 Click **Evaluate**.

Electric Field (emw) 2

As a final step, create a nice plot to use as a thumbnail. First change to the default 3D view and switch off grid.

1 In the **Model Builder** window, under **Results** click **Electric Field (emw) 2**.

2 In the **Settings** window for **3D Plot Group**, click **Go to Default View**.

DEFINITIONS

View 3

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

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

3 Clear the **Show grid** check box.

RESULTS

Electric Field (emw) 2

1 In the **Model Builder** window, under **Results** click **Electric Field (emw) 2**.

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

3 From the **View** list, choose **View 3**.

4 On the **Electric Field (emw) 2** toolbar, click **Plot**.

Next, delete the Multislice plot and add a single slice.

Multislice

In the **Model Builder** window, expand the **Electric Field (emw) 2** node.

Electric Field (emw) 2

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

Slice 1

In the **Model Builder** window, under **Results** right-click **Electric Field (emw) 2** and choose **Slice**.

Add deformation proportional to the electric field to the remaining slice.

Slice 1

1 In the **Model Builder** window, expand the **Electric Field (emw) 2** node, then click **Slice 1**.
2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
3 From the **Plane** list, choose **XY-planes**.
4 In the **Planes** text field, type 1.

Deformation 1

1 Right-click **Results>Electric Field (emw) 2>Slice 1** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Expression** section.
3 Click **emw.Ex,emw.Ey,emw.Ez - Electric field** in the upper-right corner of the section. Select the **Description** check box.

Electric Field (emw) 2

Display the magnetic field as arrows. Use logarithmic length scaling to make sure that the arrows are clearly visible everywhere. Place the arrows well above the slice.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw) 2** and choose **Arrow Volume**.
2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Magnetic>emw.Hx,emw.Hy,emw.Hz - Magnetic field**.
3 Locate the **Expression** section. Select the **Description** check box.
4 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type 30.
5 Find the **Y grid points** subsection. In the **Points** text field, type 30.

6 Find the **Z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

7 In the **Coordinates** text field, type `0.1/3`.

8 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

9 From the **Color** list, choose **Black**.

The port excitation can now be selected on the plot group. For the model thumbnail, select the second port.

Electric Field (emw) 2

1 In the **Model Builder** window, under **Results** click **Electric Field (emw) 2**.

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

3 From the **Parameter value (PortName)** list, choose **2**.

4 On the **Electric Field (emw) 2** toolbar, click **Plot**.

   By plotting Figure 7, conclude this modeling session.

Created in COMSOL Multiphysics 5.3

Parameterized Circulator Geometry

Introduction

This is a template MPH-file containing the physics interfaces and the parameterized geometry for the model Impedance Matching of a Lossy Ferrite 3-port Circulator. For a description of that application, see the book *Introduction to the RF Module* or the application documentation *Impedance Matching of a Lossy Ferrite 3-Port Circulator*.

Application Library path: `RF_Module/Ferrimagnetic_Devices/lossy_circulator_3d_geom`

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 **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

The geometry is set up using a parameterized approach. This allows you to match the input impedance to that of the connecting waveguide sections by variation of two geometric design parameters. Before starting to build the geometry the geometric design parameters need to be entered. These are two dimensionless numbers used to scale selected geometric building blocks.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`sc_chamfer`	`3`	3	`Geometry scale factor`
`sc_ferrite`	`0.5`	0.5	`Geometry scale factor`

The lossy ferrite material model is set up by referring to global variables. For convenience the definitions are stored in an external text file that is imported into the model. The external text file also contains comments.

Variables 1

1. On the **Home** toolbar, click **Variables** and choose **Global Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `lossy_circulator_3d_parameters.txt`.

The geometry is built by first defining a 2D cross section of the 3D geometry in a work plane. The 2D geometry is then extruded into 3D.

GEOMETRY 1

Work Plane 1 (wp1)

1. On the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Plane Geometry

Start by defining one arm of the circulator, then twice copy and rotate it to build all three arms.

Rectangle 1 (r1)

1. On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `0.2-0.1/(3*sqrt(3))`.
4. In the **Height** text field, type `0.2/3`.
5. Locate the **Position** section. In the **xw** text field, type `-0.2`.
6. In the **yw** text field, type `-0.1/3`.
7. Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Copy 1 (copy1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r1** only.

Rotate 1 (rot1)

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

2 Select the object **copy1** only.

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

4 In the **Rotation** text field, type `120`.

5 Right-click **Rotate 1 (rot1)** and choose **Build Selected**.

Copy 2 (copy2)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r1** only.

Rotate 2 (rot2)

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

2 Select the object **copy2** only.

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

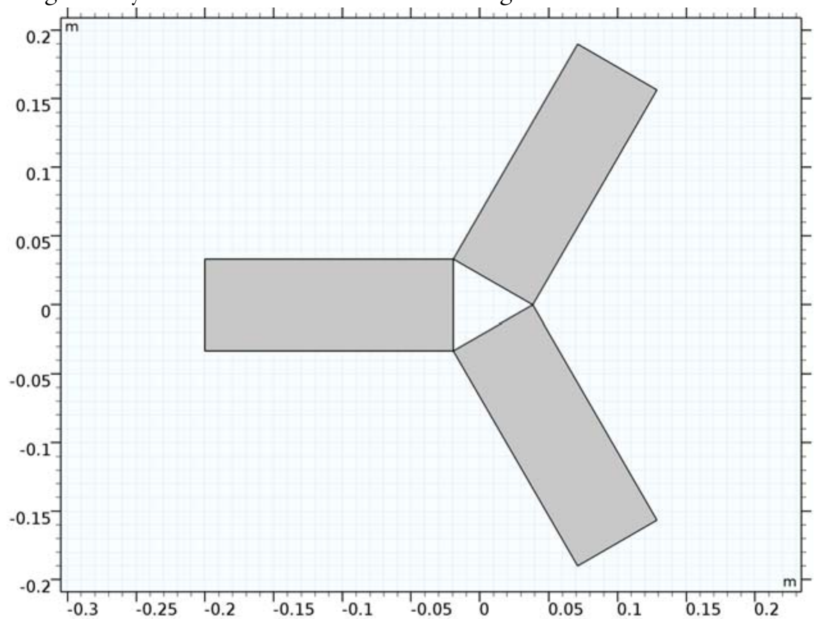
4 In the **Rotation** text field, type `-120`.

5 Right-click **Rotate 2 (rot2)** and choose **Build Selected**.

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

Plane Geometry

The geometry should now look as in the below figure.



Unite the three arms to one object.

Union 1 (uni1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 Right-click **Union 1 (uni1)** and choose **Build Selected**.

Plane Geometry

Next build the central connecting region and add the ferrite domain. During these stages, the geometric design parameters will be used. First build/add a triangle connecting the arms by subtracting a copy of what has already been drawn from a circle of proper radius.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.2/(3*sqrt(3))`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Copy 3 (copy3)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **uni1** only.

Difference 1 (dif1)

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

2 Select the object **c1** only.

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

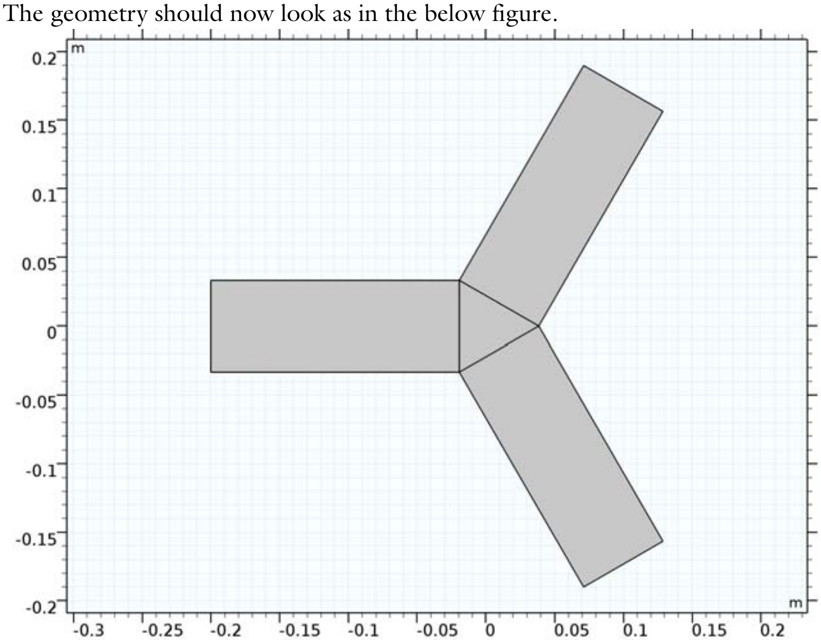
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **copy3** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Plane Geometry

The geometry should now look as in the below figure.



Rotate the newly created triangle 180 degrees and use one scaled copy of it to create linear fillets for impedance matching. Use another scaled copy to define the ferrite.

Rotate 3 (rot3)

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

2 Select the object **dif1** only.

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

4 In the **Rotation** text field, type `180`.

5 Right-click **Rotate 3 (rot3)** and choose **Build Selected**.

Copy 4 (copy4)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **rot3** only.

Plane Geometry

Apply the scaling for the impedance matching.

Scale 1 (sca1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Scale**.

2 In the **Settings** window for **Scale**, locate the **Scale Factor** section.

3 In the **Factor** text field, type `sc_chamfer`.

4 Select the object **copy4** only.

5 Right-click **Scale 1 (sca1)** and choose **Build Selected**.

Union 2 (uni2)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **sca1** and **uni1** only.

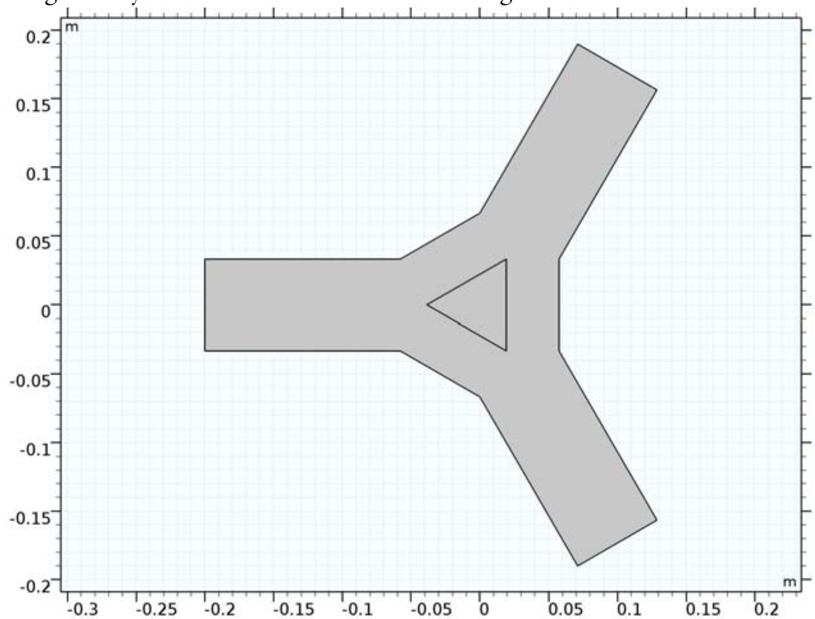
3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Right-click **Union 2 (uni2)** and choose **Build Selected**.

Plane Geometry

The geometry should now look as in the below figure.



Apply the scaling for the ferrite region.

Scale 2 (sca2)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Scale**.

2 Select the object **rot3** only.

3 In the **Settings** window for **Scale**, locate the **Scale Factor** section.

4 In the **Factor** text field, type `sc_ferrite`.

5 Right-click **Scale 2 (sca2)** and choose **Build Selected**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extruding the 2D cross-section into a 3D solid geometry finalizes the geometry definition.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
0.1/3

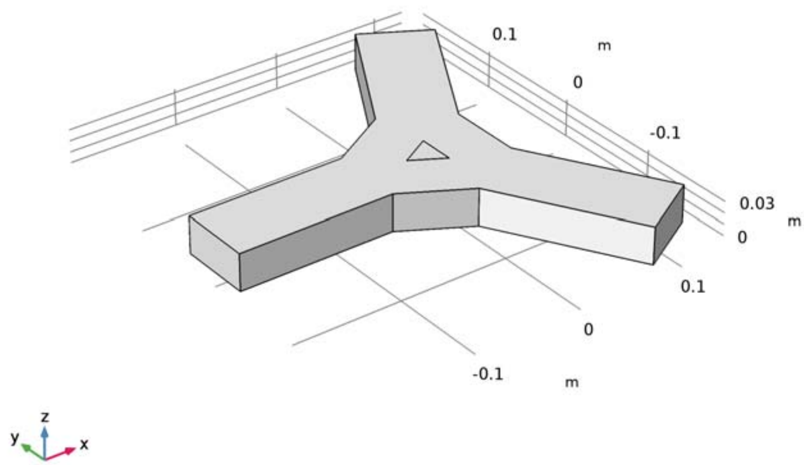
4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

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

Form Union (fin)

1 On the **Geometry** toolbar, click **Build All**.

The geometry should now look as in the below figure.



Created in COMSOL Multiphysics 5.3

Defining a Mapped Dielectric Distribution of a Material

Introduction

This example demonstrates how to set up a spatially varying dielectric distribution, such as might be engineered with a metamaterial. Here, a convex lens shape is defined via a known deformation of a rectangular domain. The dielectric distribution is defined on the undeformed, original rectangular domain and is mapped onto the deformed shape of the lens. Although the lens shape defined here is convex, the dielectric distribution causes the incident beam to diverge.

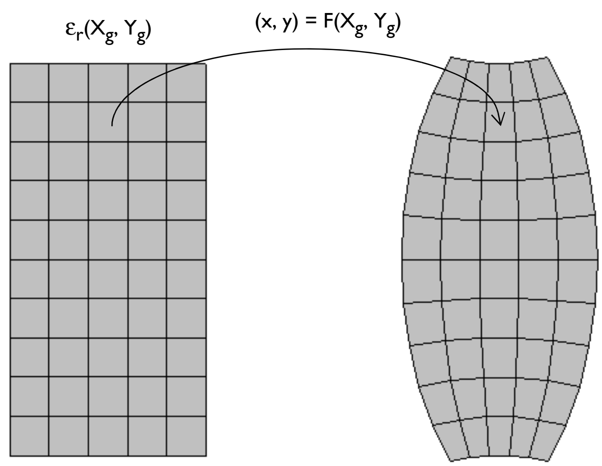


Figure 1: A convex metamaterial lens. Both the shape and the dielectric distribution are defined on a rectangular domain, and mapped into the deformed state.

Model Definition

Consider a 2D model geometry as shown in Figure 2. A square air domain, bounded by a perfectly matched layer (PML) on all sides, encloses a rectangular region in which the metamaterial lens is defined.

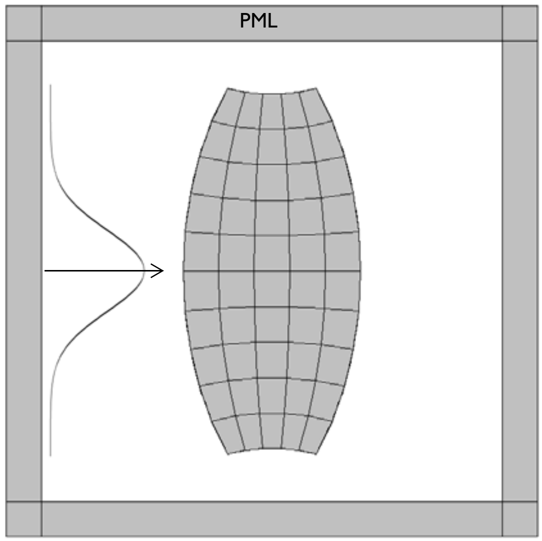


Figure 2: The modeling domain consists of the metamaterial lens in an air domain, and a surrounding PML. A Gaussian beam is incident from the left.

Model a Gaussian beam entering the domain from the left side, via a surface current excitation at an interior boundary. The surface current, J_{s0},can also be thought of as a displacement current excitation. The waist of the beam is at the boundary, so the excitation at this boundary can be specified as

$$J_{s0} = \exp\left(-\left(\frac{y}{w_0}\right)^2\right)$$

where w_0 is the waist size. The excitation is at the boundary between a domain of free space and the PML, and excites a wave propagating in both directions—into the PML and into the modeling domain. The wave propagating into the PML is completely absorbed, and the wave propagating into the domain is diffracted by the lens.

Both the shape and the dielectric distribution of the metamaterial lens are defined with respect to the original Cartesian coordinate system, as shown in Figure 1. The true shape of the lens is described by the relationship

$$\begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} F_x(X_g, Y_g) \\ F_y(X_g, Y_g) \end{bmatrix} = \begin{bmatrix} \frac{1}{2}X_g(2 - Y_g{}^2) \\ Y_g\left(1 + \frac{1}{2}x^2\right) \end{bmatrix}$$

where X_g, Y_g are the Cartesian coordinates of the undeformed frame.

The dielectric distribution is defined on the original Cartesian domain as:

$$\varepsilon_r = \left(1 + \frac{1}{2}Y_g{}^2\right)^2$$

The above expression introduces a variation in the dielectric in the y-coordinate of the undeformed lens. On the deformed lens, the dielectric varies in both directions.

The Deformed Geometry uses the above expressions to define the shape of the lens and maps the Cartesian coordinates of the undeformed frame onto the deformed frame. The dielectric distribution is defined with respect to the undeformed frame, and then mapped onto the deformed shape using the above expressions.

Results and Discussion

The model is solved for the out-of-plane electric field. Figure 3 plots the electric field norm, showing a Gaussian beam with minimal divergence incident upon the lens from the left. The beam is diffracted by the convex lens and spreads out.

Figure 4 displays the dielectric distribution, and shows variation in both directions defined via the mapping described above.

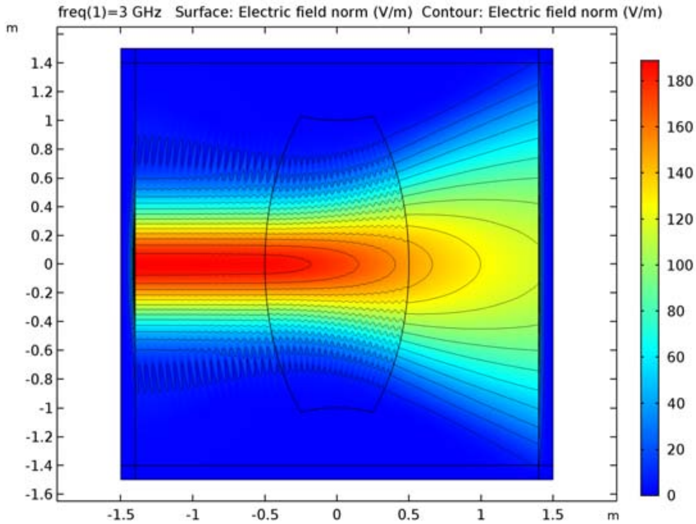


Figure 3: The norm of the electric field shows the Gaussian beam diffracted by the metamaterial lens.

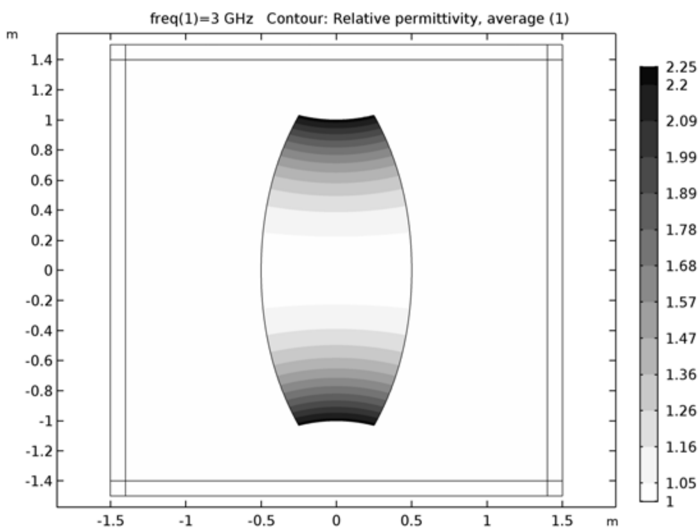


Figure 4: Contour plot of the dielectric distribution.

Application Library path: `RF_Module/Tutorials/mapped_dielectric_distribution`

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 **2D**.
2 In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Stationary**.
8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
f0	3[GHz]	3E9 Hz	Operating frequency
lda0	c_const/f0	0.09993 m	Free space wavelength
w0	lda0*4	0.3997 m	Gaussian beam waist size

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

GEOMETRY 1

First, create a square for the entire model domain. Add a layer on each side of the square.

Square 1 (sq1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.
3. In the **Side length** text field, type `3`.
4. Locate the **Position** section. From the **Base** list, choose **Center**.
5. Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	**Thickness (m)**
`Layer 1`	`lda0`

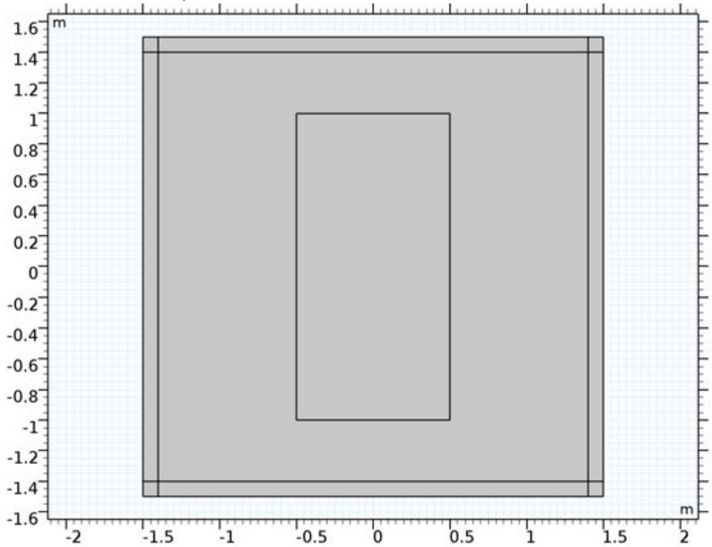
6. Select the **Layers to the left** check box.
7. Select the **Layers to the right** check box.
8. Select the **Layers on top** check box.
9. Right-click **Square 1 (sq1)** and choose **Build Selected**.

Add a rectangle for the lens.

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Height** text field, type `2`.
4. Locate the **Position** section. From the **Base** list, choose **Center**.

5 Click **Build All Objects**.



DEFINITIONS

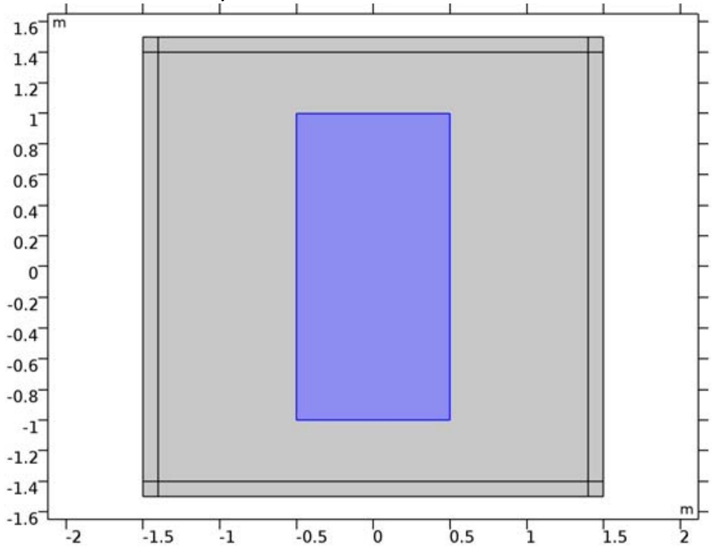
Add a selection for the lens domain which will be recalled frequently while setting up the model properties.

Explicit 1

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

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

3 Select Domain 7 only.



Next, add a set of variables for the shape and the dielectric distribution of the lens.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

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

4 From the **Selection** list, choose **Lens**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`xp`	`0.5[m]*Xg[1/m]*(2-(Yg[1/m])^2)`	m	`Mapping of Xg -> x`
`yp`	`Yg*(1+(0.5*(xp[1/m])^2))`	m	`Mapping of Yg -> y`
`erp`	`(1+0.5*(Yg[1/m])^2)^2`		`Dielectric distribution`

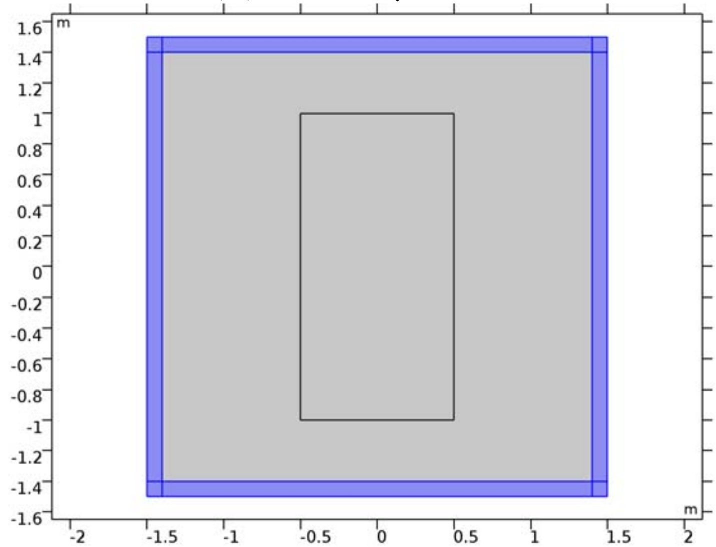
Here, `Xg` and `Yg` are predefined Deformed Geometry physics variables representing the Cartesian coordinates of the undeformed frame.

Add a perfectly matched layer (PML).

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1–4, 6, and 8–10 only.



DEFORMED GEOMETRY (DG)

Set up Deformed Geometry. You need to specify Free Deformation, Prescribed Mesh Displacement and Prescribed Deformation.

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

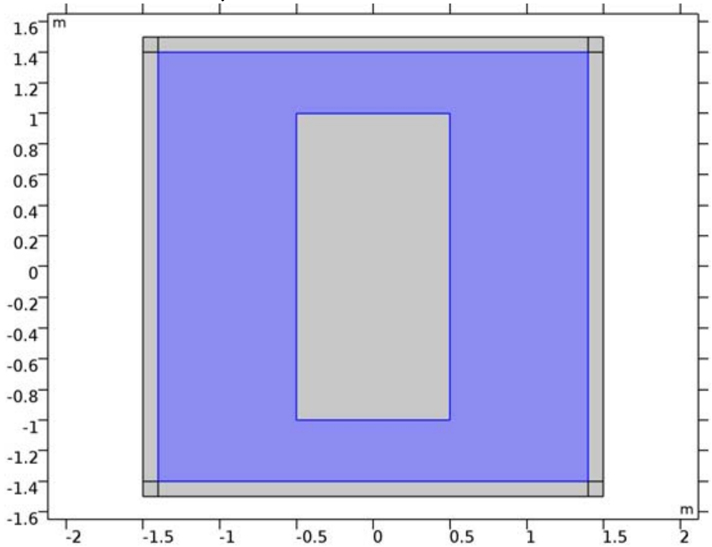
2 In the **Settings** window for **Deformed Geometry**, locate the **Frame Settings** section.

3 From the **Geometry shape order** list, choose **1**.

Free Deformation 1

1 Right-click **Component 1 (comp1)>Deformed Geometry (dg)** and choose **Free Deformation**.

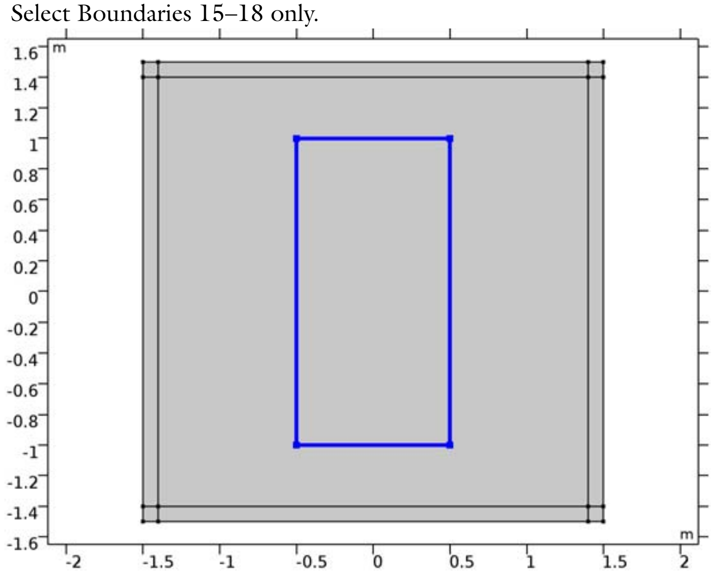
2 Select Domain 5 only.



Prescribed Mesh Displacement 2

1 In the **Model Builder** window, right-click **Deformed Geometry (dg)** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 15–18 only.



3. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.
4. In the d_X text field, type `xp-Xg`.
5. In the d_Y text field, type `yp-Yg`.

Prescribed Deformation 1

1. Right-click **Deformed Geometry (dg)** and choose **Prescribed Deformation**.
2. In the **Settings** window for **Prescribed Deformation**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Lens**.
4. Locate the **Prescribed Mesh Displacement** section. In the d_X text-field array, type `xp-Xg` on the first row.
5. In the d_Y text-field array, type `yp-Yg` on the second row.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

In **Electromagnetic Waves, Frequency Domain**, the dielectric distribution is configured via the user-defined variable erp and the Gaussian beam is modeled as entering the domain from the left side, via a surface current excitation.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Components** section.
3. From the **Electric field components solved for** list, choose **Out-of-plane vector**, to only perform the calculation for the out-of-plane component. The in-plane components are both zero.
4. Locate the **Physics-Controlled Mesh** section. Clear the **Enable** check box.

Wave Equation, Electric 2

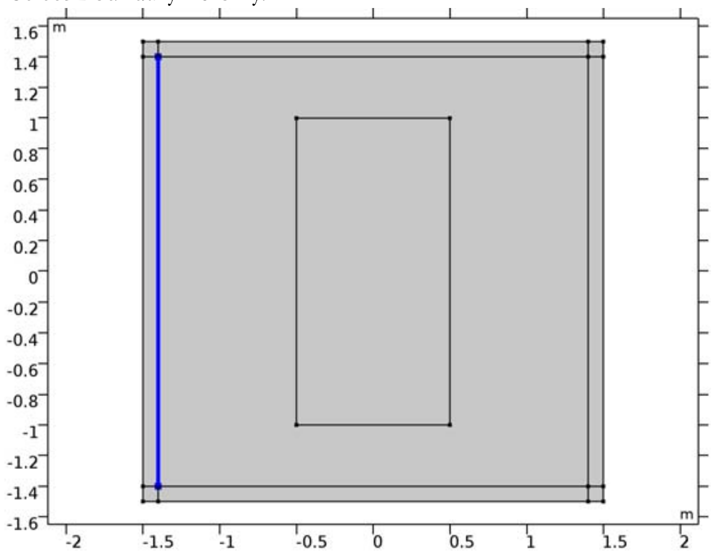
1. Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** and choose **Wave Equation, Electric**.
2. In the **Settings** window for **Wave Equation, Electric**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Lens**.
4. Locate the **Electric Displacement Field** section. From the ε_r list, choose **User defined**. In the associated text field, type `erp`.
5. Locate the **Magnetic Field** section. From the μ_r list, choose **User defined**, (Leave the default value `1`).

6 Locate the **Conduction Current** section. From the σ list, choose **User defined**, (Leave the default value 0).

Surface Current Density 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose the boundary condition **More> Surface Current Density**.

2 Select Boundary 10 only.



3 In the **Settings** window for **Surface Current Density**, locate the **Surface Current Density** section.

4 Specify the $\mathbf{J}_{s0}$ vector as

0	x
0	y
`exp(-(y/w0)^2)`	z

MATERIALS

Set all domain with vacuum. The lens domain material properties are explicitly configured by **Wave Equation, Electric 2** in **Electromagnetic Waves, Frequency Domain**.

ADD MATERIAL

1 On 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.

MATERIALS

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

MESH 1

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lda0/10`.

5 In the **Minimum element size** text field, type `0.0012`.

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

7 In the **Settings** window for **Mesh**, click **Build All**.

You may zoom in a few times to check the quality of the mesh.

STUDY 1

The model is analyzed with two study steps. First, make sure that Stationary study step is solved only for Deformed Geometry.

Step 1: Stationary

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

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

3 In the table, clear the **Solve for** check box for the **Electromagnetic Waves, Frequency Domain** interface.

Add a **Frequency Domain** study step and set as solved only for **Electromagnetic Waves, Frequency Domain**.

Step 2: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain**.

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

3 In the **Frequencies** text field, type `f0`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Deformed Geometry** interface.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows the magnitude of electric fields. Change the default color pattern and add a contour plot for the magnitude.

Surface

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **RainbowLight**.

Contour 1

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

2 In the **Settings** window for **Contour**, locate the **Levels** section.

3 In the **Total levels** text field, type `14`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

6 Clear the **Color legend** check box.

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

See Figure 3 to compare the reproduced plot.

Add a filled contour plot describing the dielectric distribution over the lens.

Contour 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 2** and choose **Contour**.

3 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Material properties>emw.epsrAv - Relative permittivity, average**.

4 Locate the **Levels** section. In the **Total levels** text field, type `12`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

6 From the **Color table** list, choose **GrayScale**.

7 Select the **Reverse color table** check box.

8 On the **2D Plot Group 2** toolbar, click **Plot**.

The plot for the dielectric distribution is shown in Figure 4.

Created in COMSOL Multiphysics 5.3

Signal Integrity and TDR Analysis of Adjacent Microstrip Lines

Introduction

The signal integrity (SI) analysis gives an overview of the quality of an electrical signal transmitted through electrical circuits such as high-speed interconnects, cables, and printed circuit boards. The quality of the received signal can be distorted by noise from outside the circuit, and can be degraded by impedance mismatch, insertion loss, and crosstalk; in practice, EMC/EMI analyses are run to estimate the susceptibility of a device or a network to an undesired coupling. In this example model, we examine the crosstalk effect between two adjacent microstrip lines on a microwave substrate. The simulated results provide the time-domain reflectometry (TDR) response at the coupled ports and show increased distortion of a signal at higher data rates.

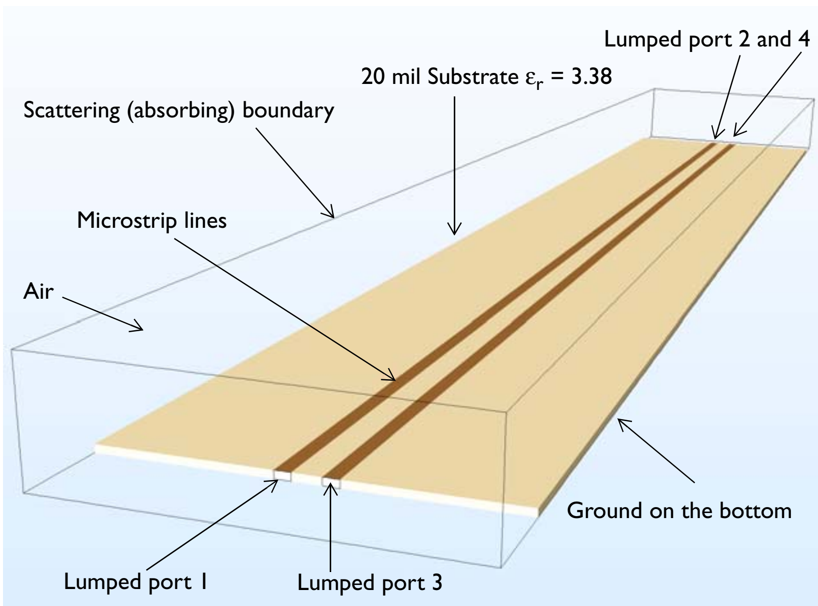


Figure 1: A microstrip line crosstalk model is composed of 20 mil microwave substrate with a ground plane and two adjacent microstrip lines 1.8 mm apart.

Model Definition

Two parallel 50 Ω microstrip lines are patterned on 20 mil substrate with a dielectric constant $\varepsilon_r = 3.38$. All metallic parts, including the patterned lines and bottom ground plane, are configured using perfect electric conductor (PEC) boundary conditions. The small rectangular surfaces, bridging between two parallel lines and the ground plane, are used to model lumped ports with which the microstrip lines are excited or terminated by

50 Ω. The air domain on top of the circuit board is defined using vacuum material properties. The exterior surfaces of the air are finished by a scattering boundary condition that is an absorbing boundary to describe an open space.

One bit of a single rectangular pulse is used to excite the circuit board. The widths of the two pulses are set to half of the 300 MHz and 600 MHz signals. The corresponding data rates for each frequency are 600 Mbit/s and 1.2 Gbit/s, respectively. A parametric sweep switches the frequency of the pulse during the simulation. It is necessary to apply smoothing to the transition zone of the pulse to remove undesirable high-frequency components from the signal.

The maximum simulation time is calculated using an approximated traveling time of a wave through a microstrip line based on the phase velocity. The effective dielectric constant for the phase velocity calculation is obtained using an equation in Ref. 1

$$\frac{\varepsilon_r + 1}{2} + \frac{\varepsilon_r - 1}{2\sqrt{1 + 12\frac{d}{W}}} \tag{1}$$

where d is the thickness of the substrate and W is the width of the line.

It is assumed that a frequency about ten times greater than the input pulse signal frequency is enough to describe the highest frequency component in the smoothed rectangular pulse. The maximum mesh element size is set to 0.2 wavelengths in the dielectric substrate.

It is also important to define a time step that resolves the wave well in time as the mesh does in space. Any longer time steps would not optimally utilize the fine mesh, and any shorter time steps would unnecessarily lead to a longer simulation time without gaining significant accurate results. While running a simulation, the time step is continuously adjusted to meet the specified tolerances by the time-dependent solver. If there is an exact time step the solver needs to take, it can be manually set. In the Settings window of the Time-Dependent Solver node, the time step can be specified manually. See the step by step instructions to learn how to access this setting.

Results and Discussion

Figure 2 shows the input pulse signal as well as the voltage at lumped port 1 with a data rate of 600 Mbit/s (300 MHz) and 1.2 Gbit/s (600 MHz), respectively. Since the input signal is flowing through a straight 50 Ω line terminated with a 50 Ω resistor without discontinuity on the line, no distortion is evident on the port voltage.

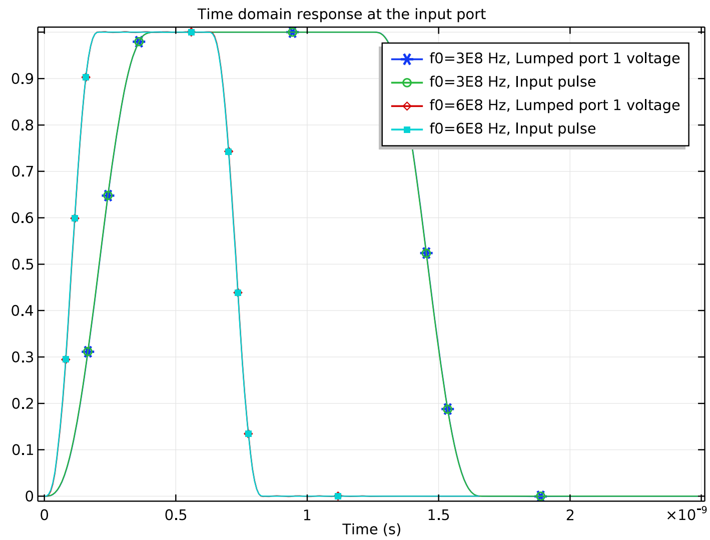


Figure 2: The input pulse and the voltage at lumped port 1 (the excitation port) with a data rate of 600 Mbit/s and 1.2 Gbit/s.

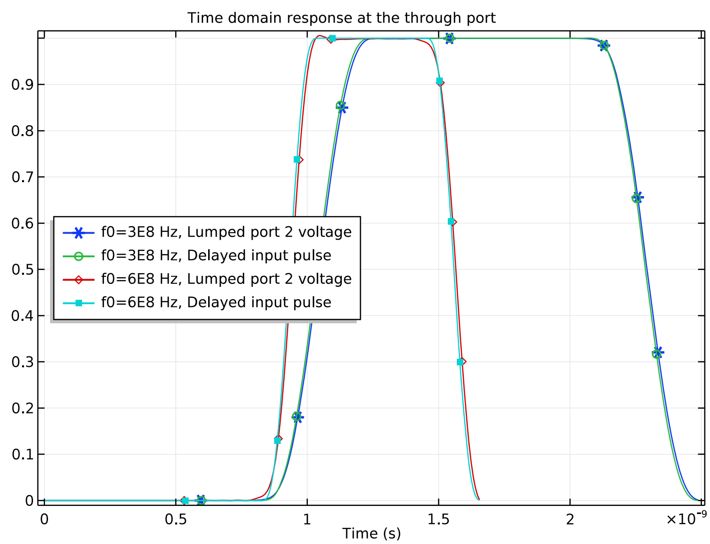


Figure 3: The delayed input pulse and the voltage at lumped port 2 (the through port) with a data rate of 600 Mbit/s and 1.2 Gbit/s.

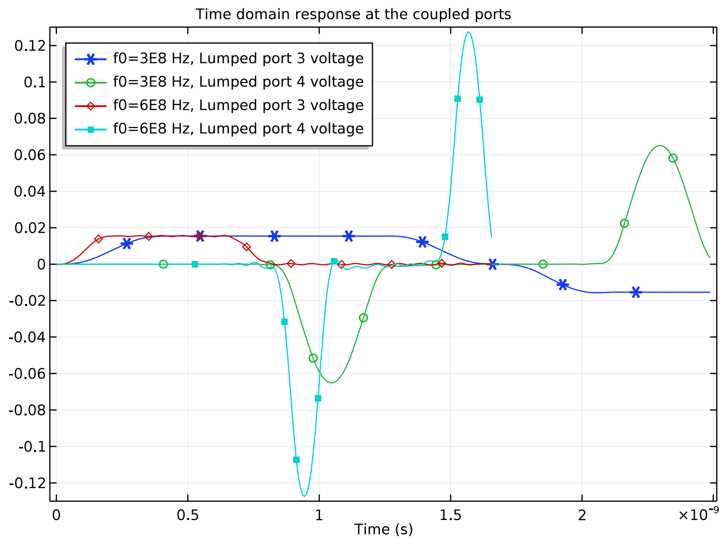


Figure 4: Voltage for the coupled signals at lumped ports 3 and 4. The voltage of a coupled signal increases at a higher data rate.

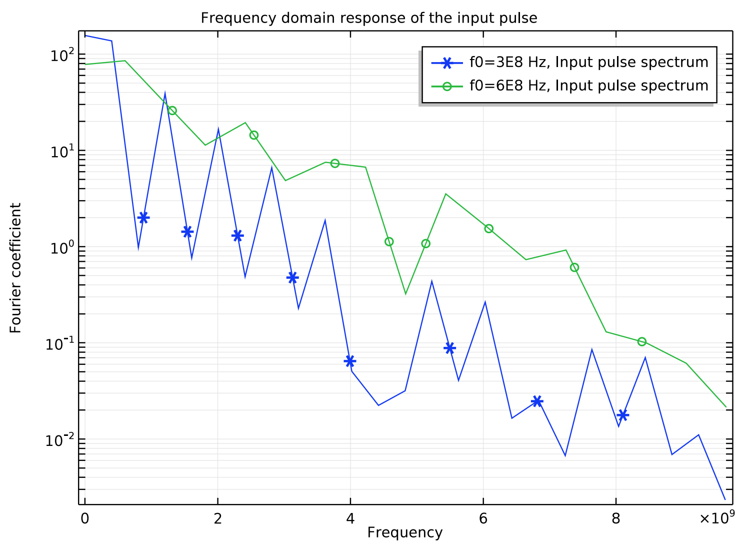


Figure 5: The spectrum of input pulses up to 10 GHz. The signal strength decreases as frequency increases.

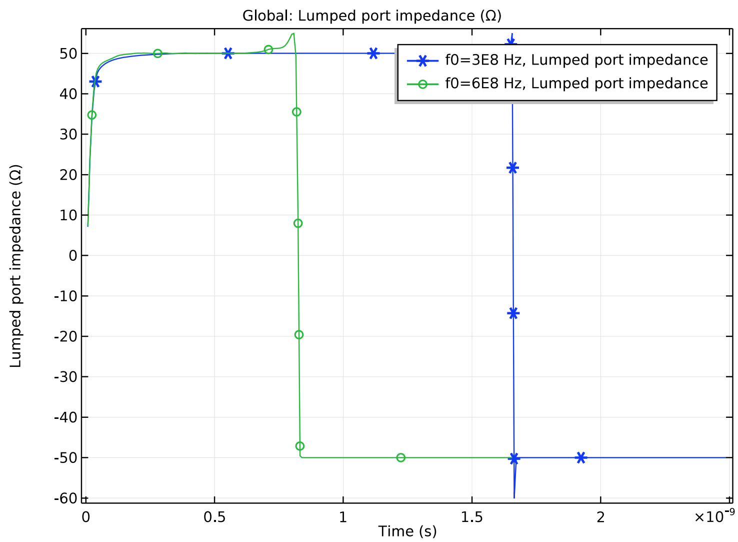


Figure 6: The impedance of lumped port 1 with data rates of 600 Mbit/s and 1.2 Gbit/s

Figure 3 shows the delayed input pulse and the received signals with two data rates at lumped port 2. The time domain response of the 1.2 Gbit/s signal is slightly distorted in the beginning when it reaches 1 V while that of the 600 Mbit/s signal seems to remain undistorted.

The crosstalk between two microstrip lines is observed in Figure 4. The coupled signal level between two data rates is quite similar at lumped port 3, which is next to the excitation port. The time domain response at lumped port 4 next to the through port, shows that the higher data rate signal causes the stronger crosstalk on another signal path.

Figure 5 works as a reference to define the effective highest frequency component in the smoothed rectangular pulse since it provides the spectrum of results for 600 Mbit/s and 1.2 Gbit/s. A periodic rectangular pulse can be decomposed into a sum of sinusoidal functions. By estimating the level of a particular frequency, a proper frequency range can be defined for efficient simulations. The estimated highest frequency is used to choose the mesh size. With a finer mesh size, higher frequency components can be analyzed more accurately but it will increase the computation time. In this model, we set the maximum frequency component to 5 GHz that is two orders of magnitude smaller than the level of the DC component of each rectangular pulse.

In Figure 6, the TDR at lumped port 1 is presented in terms of impedance. The computed port impedance is around 50 Ω while the signal level is 1 V.

Notes About the COMSOL Implementation

Changing the number of output times in the **Step 1: Time Dependent** node configures the output times for the results analysis but has a minimal effect on the time steps taken by the solver.

Reference

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

Application Library path: `RF_Module/EMI_EMC_Applications/microstrip_line_crosstalk`

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, Transient (temw)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 Click **Load from File**.

3 Browse to the model's Application Libraries folder and double-click the file `microstrip_line_crosstalk_parameters.txt`.

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 **in**.

Block 1 (blk1)

1 On 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 `blength+0.5`.

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

5 In the **Height** text field, type `tsub*15`.

6 Locate the **Position** section. In the **x** text field, type `-0.25`.

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

8 Click to expand the **Layers** section. Find the **Layer position** subsection. Select the **Left** check box.

9 Select the **Right** check box.

10 Clear the **Bottom** check box.

11 In the table, enter the following settings:

Layer name	Thickness (in)
Layer 1	0.25

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

Block 2 (blk2)

1 On 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 `blength`.

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

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

6 Locate the **Position** section. In the **y** text field, type `-bwidth/2`.

Block 3 (blk3)

1 On 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 `blength`.

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

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

6 Locate the **Position** section. In the **y** text field, type `-spacing/2-lwidth`.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **blk3** only.

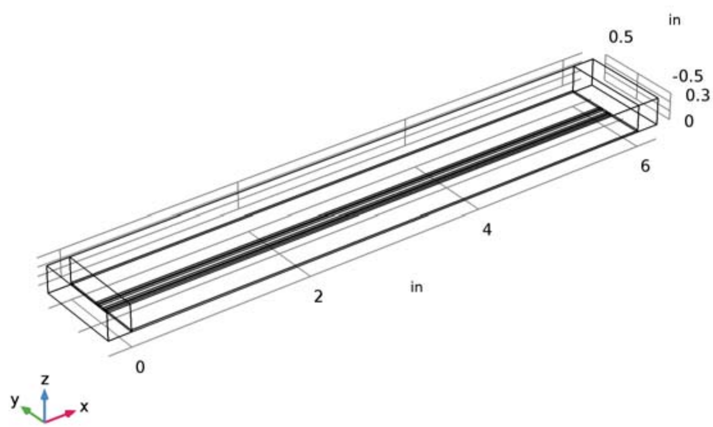
3 In the **Settings** window for **Mirror**, locate the **Input** section.

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

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

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

7 Click **Build All Objects**.



Rectangle 1 (rect1)

On the **Home** toolbar, click **Functions** and choose **Global>Rectangle**.

GLOBAL DEFINITIONS

Rectangle 1 (rect1)

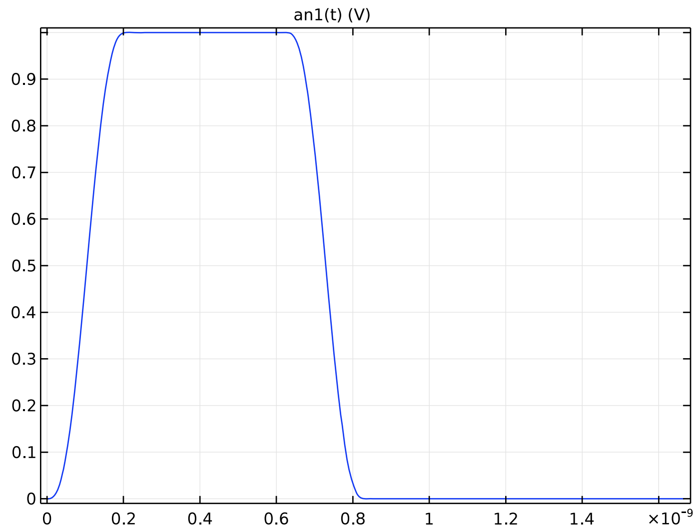
1 In the **Settings** window for **Rectangle**, locate the **Parameters** section.
2 In the **Lower limit** text field, type `0`.
3 In the **Upper limit** text field, type `Tb-Tb/4`.
4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `Tb/4`.

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.
2 In the **Settings** window for **Analytic**, locate the **Definition** section.
3 In the **Expression** text field, type `rect1((t-Tb/8)/1[s])`.
4 In the **Arguments** text field, type `t`.
5 Locate the **Units** section. In the **Arguments** text field, type `s`.
6 In the **Function** text field, type `V`.
7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
t	0	2*Tb

8 Click **Plot**.



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.

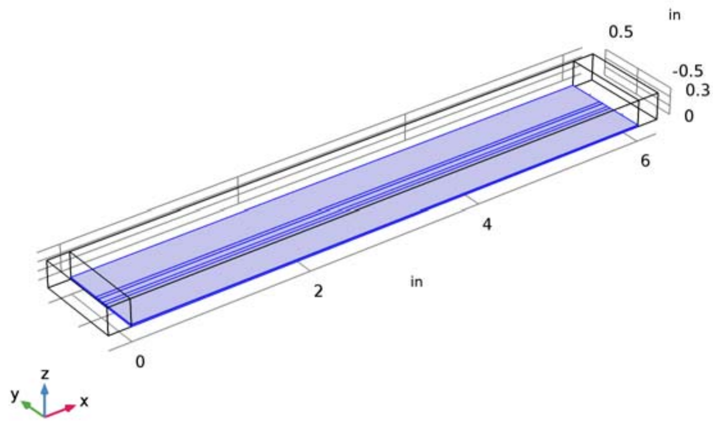
3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

Material 2 (mat2)

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

2 Select Domains 2 and 4–7 only.



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

4 In the table, enter the following settings:

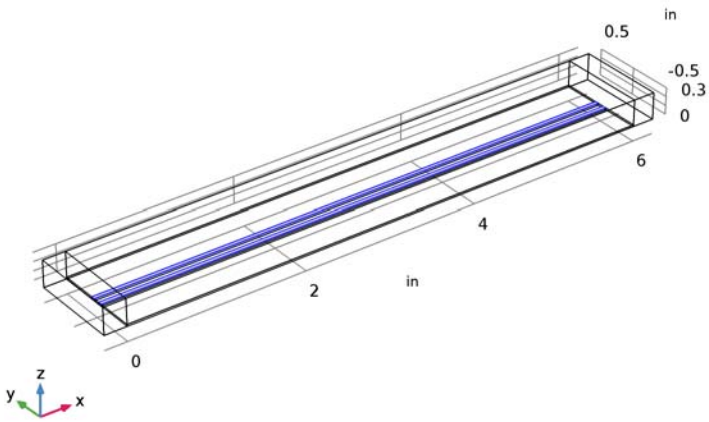
Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	`er_sub`	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Perfect Electric Conductor 2

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

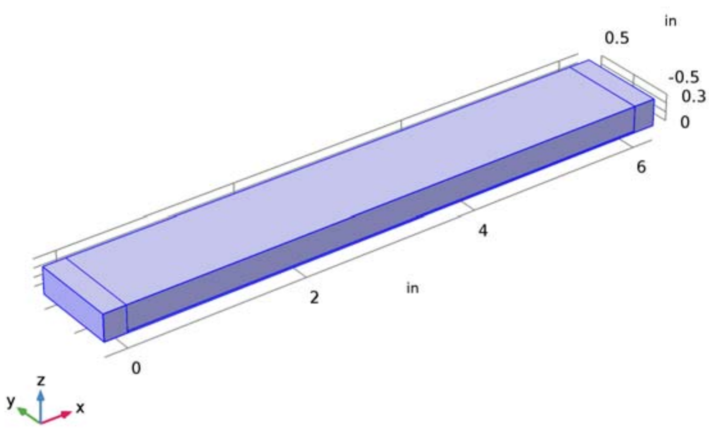
2 Select Boundaries 16 and 24 only.



Scattering Boundary Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Transient (temw)** and choose **Scattering Boundary Condition**.

2 Select Boundaries 1, 2, 4, 5, 7, 10, 12, 30, 32, 35, 40, and 41 only.



Lumped Port 1

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 21 only.

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

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

4 In the V_0 text field, type `an1(t)`.

Lumped Port 2

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 38 only.

Lumped Port 3

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 13 only.

Lumped Port 4

1 Right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 36 only.

MESH 1

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

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

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

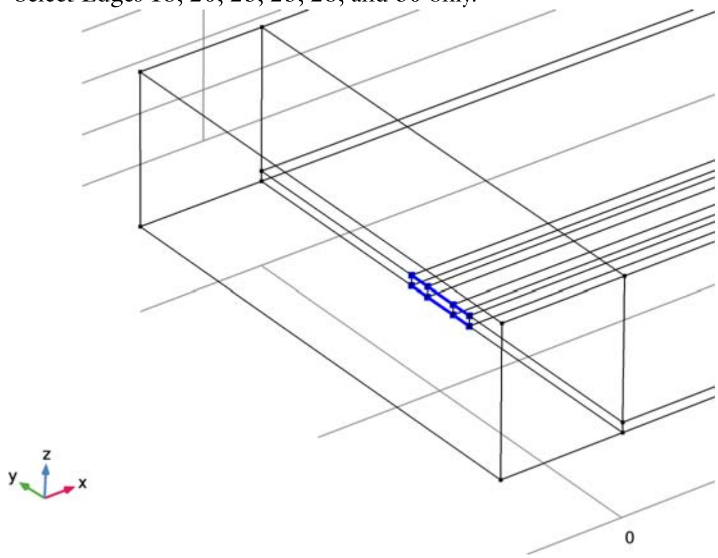
3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `hm`.

Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Edge**.

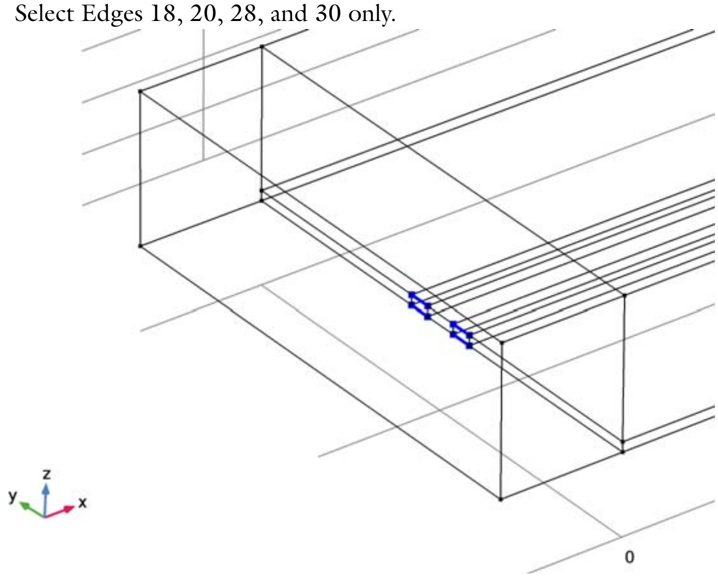
2 Select Edges 18, 20, 23, 25, 28, and 30 only.



Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Distribution**.

2 Select Edges 18, 20, 28, and 30 only.



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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

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

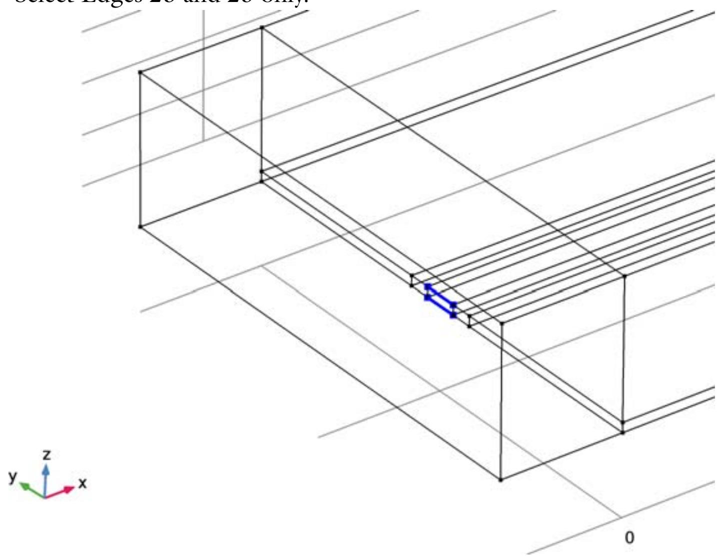
6 In the **Element ratio** text field, type 4.

7 Select the **Symmetric distribution** check box.

Distribution 2

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

2 Select Edges 23 and 25 only.



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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 4.

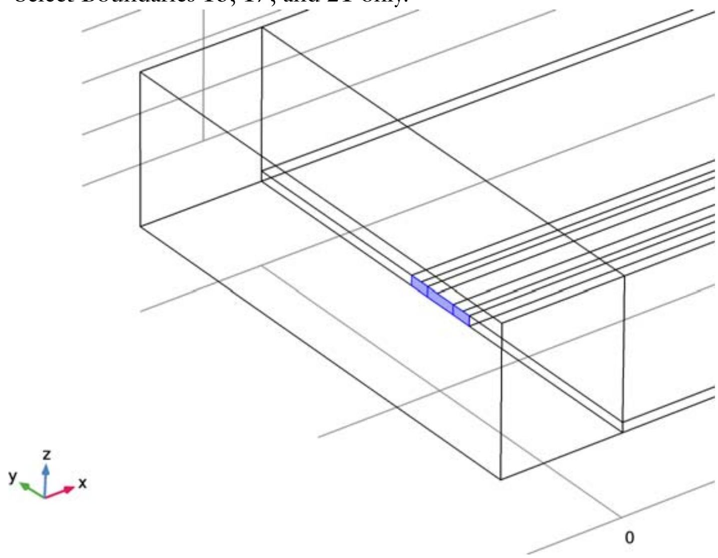
6 In the **Element ratio** text field, type 4.

7 Select the **Symmetric distribution** check box.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.

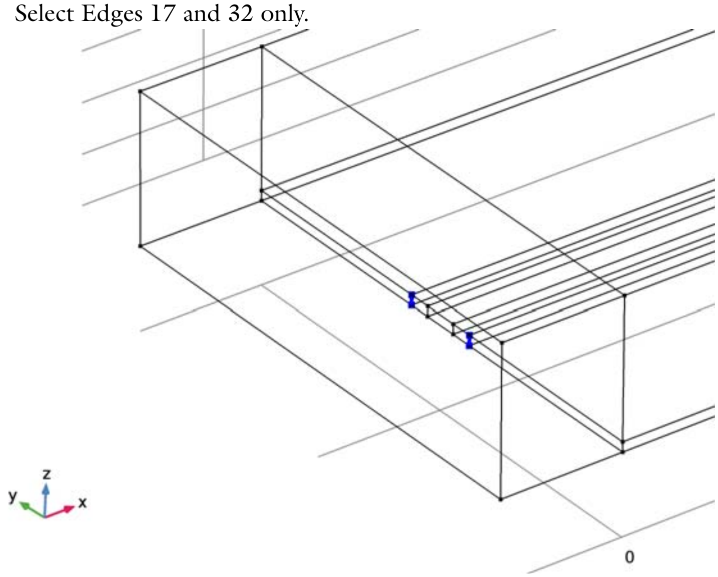
2 Select Boundaries 13, 17, and 21 only.



Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 17 and 32 only.



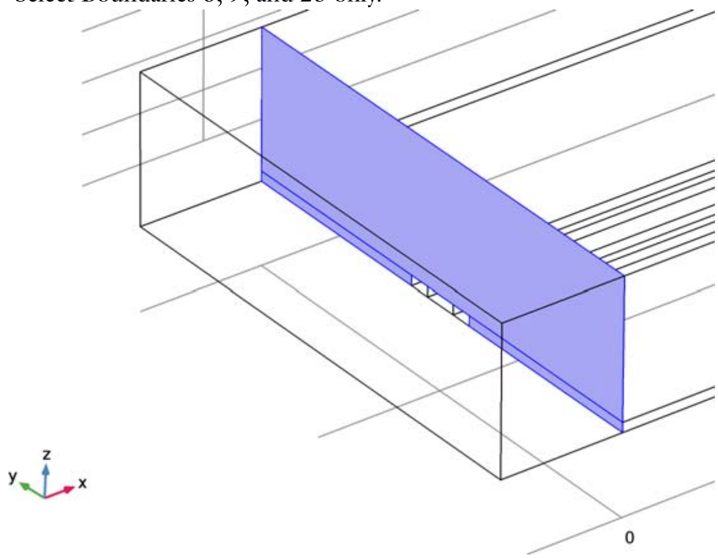
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations> Free Triangular**.

2 Select Boundaries 6, 9, and 25 only.



3 Right-click **Mesh 1** and choose **Swept**.

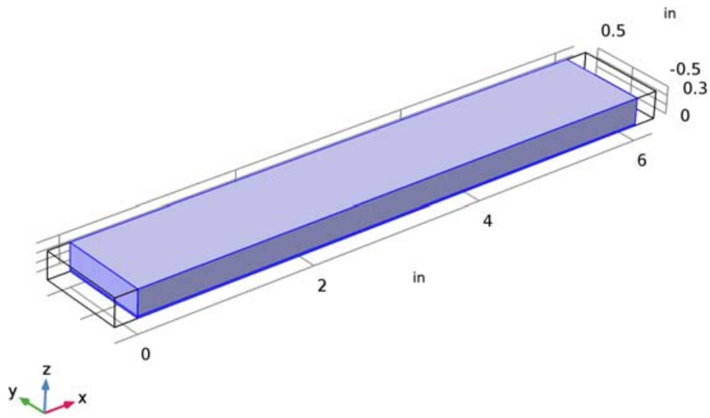
4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Swept 1

1 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 Select Domains 2–7 only.



Free Tetrahedral 1

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

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Delete**.

MESH 1

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

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

3 Click the **Click and Hide** button on the **Graphics** toolbar.

4 Click the **Select Boundaries** button on the **Graphics** toolbar.

5 Select Boundary 12 only.

6 Select Boundary 10 only.

7 Select Boundary 4 only.

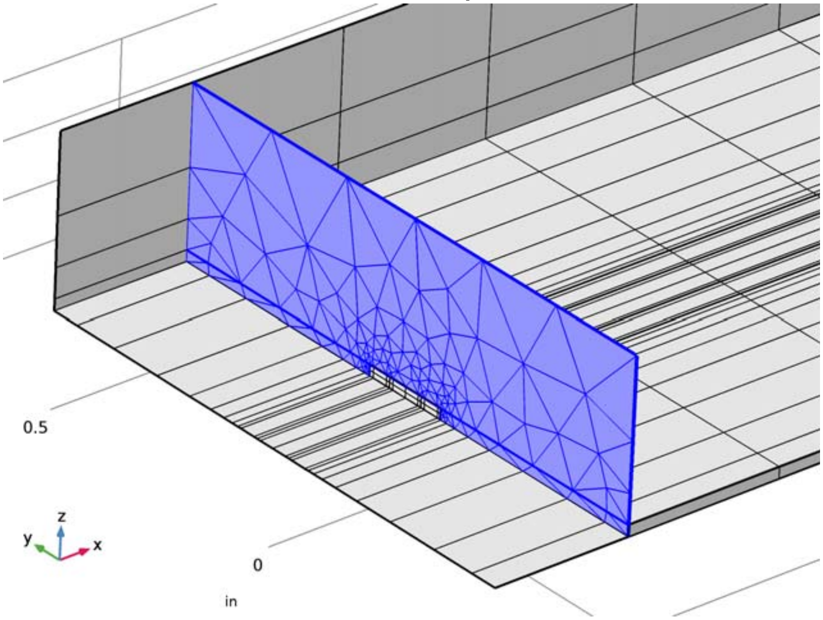
8 Select Boundary 1 only.

9 Select Boundary 2 only.

10 Click the **Click and Hide** button on the **Graphics** toolbar.

Free Triangular 1

Click the **Zoom to Selection** button on the **Graphics** toolbar.



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,sim_time_step,sim_time_max)`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `sim_time_step`.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Direct**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

Parametric Sweep

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

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

3 Click **Add**.

4 Click to select row number 1 in the table.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0	300[MHz] 600[MHz]	

6 On the **Study** toolbar, click **Compute**.

RESULTS

3D Plot Group 1

1 In the **Model Builder** window, under **Results** click **3D Plot Group 1**.

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

3 From the **Time (s)** list, choose **6E-10**.

Multislice 1

1 In the **Model Builder** window, expand the **3D Plot Group 1** node, then click **Multislice 1**.

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

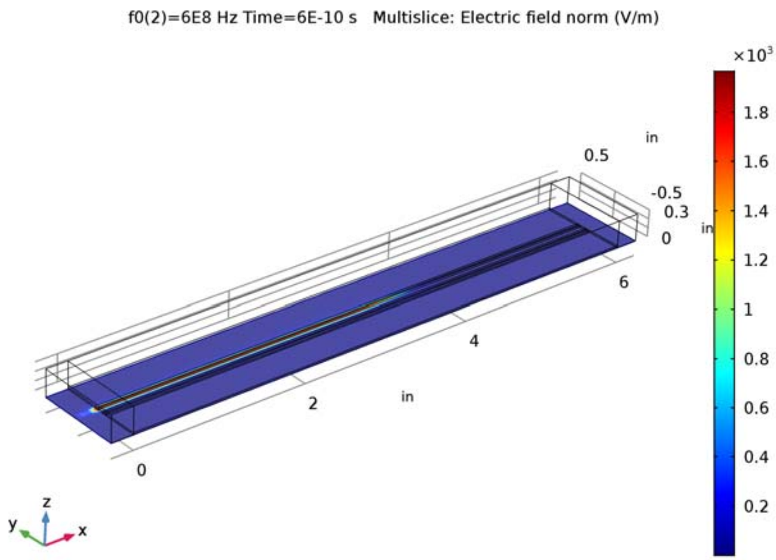
3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

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 On the **3D Plot Group 1** toolbar, click **Plot**.



1D Plot Group 2

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Time domain response at the input port`.

Global 1

1 Right-click **1D Plot Group 2** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`temw.Vport_1`	V	`Lumped port 1 voltage`
`an1(t)`		`Input pulse`

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

5 On the **1D Plot Group 2** toolbar, click **Plot**.

Figure 2 shows the input pulse and the voltage at lumped port 1.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 2** and choose **Duplicate**.

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

3 In the **Title** text area, type `Time domain response at the through port`.

4 Locate the **Legend** section. From the **Position** list, choose **Middle left**.

Global 1

1 In the **Model Builder** window, expand the **1D Plot Group 3** 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**
`temw.Vport_2`	`V`	`Lumped port 2 voltage`
`an1(t-delay)`	`V`	`Delayed input pulse`

4 On the **1D Plot Group 3** toolbar, click **Plot**.

Figure 3 shows the delayed input pulse and the voltage at lumped port 2.

1D Plot Group 4

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Time domain response at the coupled ports`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`temw.Vport_3`	V	`Lumped port 3 voltage`
`temw.Vport_4`	V	`Lumped port 4 voltage`

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 On the **1D Plot Group 4** toolbar, click **Plot**.

The coupled signals at lumped port are shown in Figure 4.

1D Plot Group 5

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Frequency domain response of the input pulse`.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`an1(t)`	V	`Input pulse spectrum`

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Frequency spectrum**.

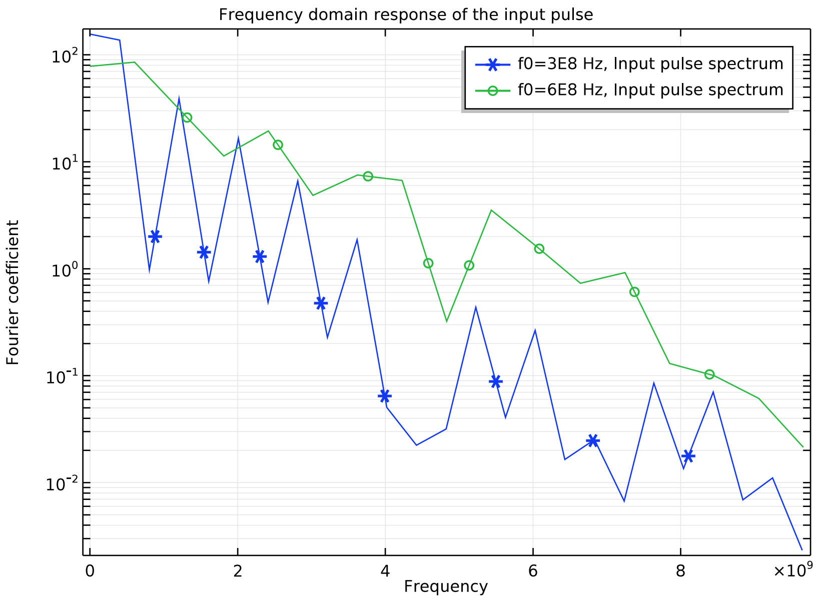
5 Select the **Frequency range** check box.

6 In the **Maximum** text field, type `10[GHz]`.

7 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 On the **1D Plot Group 5** toolbar, click **Plot**.

9 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.



Compare to the spectra of input pulses in Figure 5.

1D Plot Group 6

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Right-click **1D Plot Group 6** and choose **Global**.

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

6 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

Global 1

1 In the **Model Builder** window, under **Results>1D Plot Group 6** click **Global 1**.

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>Electromagnetic Waves, Transient>Ports>temw.Zport_1 - Lumped port impedance**.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 On the **1D Plot Group 6** toolbar, click **Plot**.

Figure 6 describes the impedance of lumped port 1 with two data rates as a function of time.

Created in COMSOL Multiphysics 5.3

Microstrip Patch Antenna

Introduction

The microstrip patch antenna is used in a wide range of applications because it is easy to design and fabricate. The antenna is attractive due to its low-profile conformal design, relatively low cost, and very narrow bandwidth. This example uses an inset feeding strategy that does not need any additional matching parts.

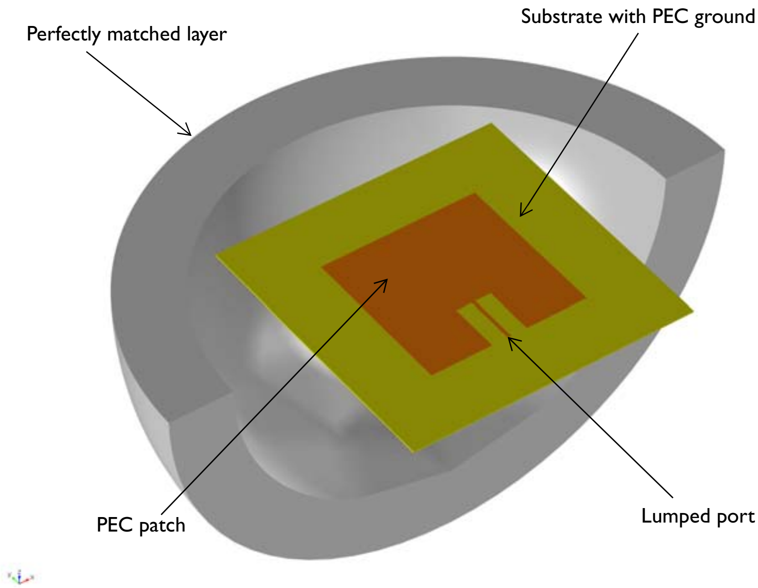


Figure 1: Microstrip patch antenna. The model consists of a PEC ground plane, a 50 Ω microstrip line fed by a lumped port, a region of free space, and a perfectly matched layer (PML) domain.

Model Definition

Feeding a patch antenna from the edge leads to a very high input impedance, causing an undesirable impedance mismatch if a conventional 50 Ω line is directly connected. One solution to this problem is to use a matching network of quarter-wave transformers between the feed point of the 50 Ω line and the patch. However, this approach has two drawbacks. First, the quarter-wave transformers would be realized as microstrip lines that would have to extend beyond the patch antenna, significantly increasing the overall structure size. Second, these microstrip lines should have a high characteristic impedance and thus would have to be narrower than practical for fabrication. Therefore, a better approach is desired.

This example uses a different feed point for the patch antenna to improve matching between the 50 Ω feed and the antenna. It is known that the antenna impedance is higher than 50 Ω if fed from the edge, and lower if fed from the center. Therefore, an optimum feed point exists between the center and the edge. The matching strategy is shown in Figure 2. A 50 Ω microstrip line, fed from the end, extends into the patch antenna structure. The width of the cutout region, W, is chosen to be large enough so that there is minimal coupling between the antenna and the microstrip, but not so large as to significantly affect the antenna characteristics. The length of the microstrip line, L, is chosen to minimize the reflected power, S_{11}. These optimal dimensions can be found via a parametric sweep; this example only treats the final design.

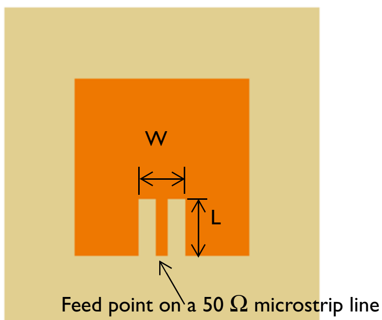


Figure 2: The matching strategy between a 50 Ω line and a patch antenna. A microstrip line of length L extends into a slot of with W cut into the patch antenna.

Results and Discussion

Figure 3 shows the radiation pattern in the E-plane; the E-plane is defined by the direction of the antenna polarization and may include the direction of maximum radiation. 3D far-field radiation pattern is visualized in Figure 4 showing the directive beam pattern due to the ground plane that blocks the radiation toward the bottom side. With the choice of feed point used in this example, the S_{11} parameter is better than −10 dB, and the front-to-back ratio in the radiation pattern is more than 15 dB.

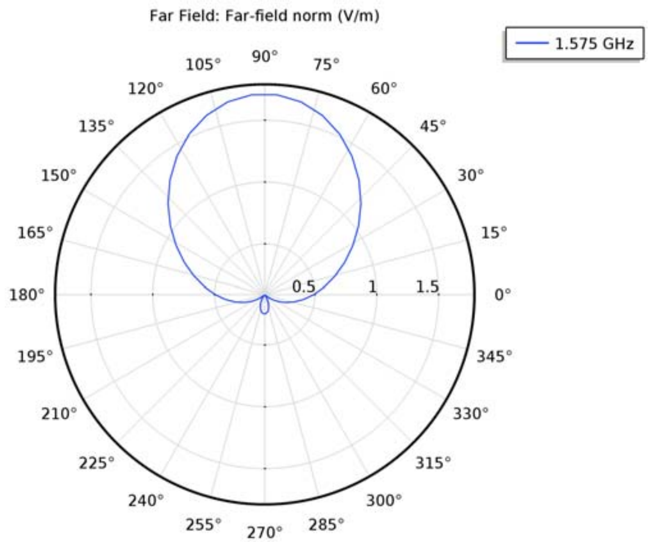


Figure 3: Far-field radiation pattern at E-plane. Because of the bottom ground plane, the radiation pattern is directed toward the top.

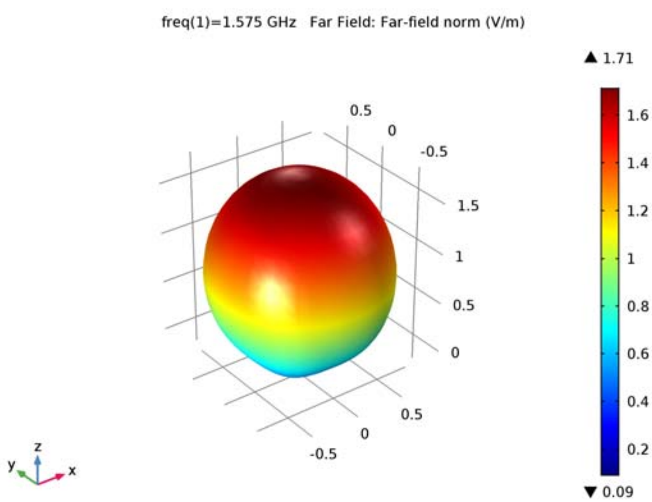


Figure 4: 3D Far-field radiation pattern is directed toward the top.

References

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

2. C.A. Balanis, *Antenna Theory*, John Wiley & Sons, 1997.

3. R.E. Collin, *Antennas and Radiowave Propagation*, McGraw-Hill, 1985.

Application Library path: `RF_Module/Antennas/microstrip_patch_antenna_inset`

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 **Preset 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 In the **Frequencies** text field, type `1.575[GHz]`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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
w_line	3.2[mm]	0.0032 m	50 ohm line width
w_patch	53[mm]	0.053 m	Patch width
l_patch	52[mm]	0.052 m	Patch length
w_stub	7[mm]	0.007 m	Tuning stub width
l_stub	16[mm]	0.016 m	Tuning stub length
w_sub	100[mm]	0.1 m	Substrate width
l_sub	100[mm]	0.1 m	Substrate length

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**.

First, create the substrate block.

Block 1 (blk1)

1 On 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 `w_sub`.

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

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

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

7 Right-click **Substrate** and choose **Build Selected**.

Now add the patch antenna.

Block 2 (blk2)

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

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

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

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

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

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

7 Right-click **Patch** and choose **Build Selected**.

Create impedance matching parts and a 50Ω feed line.

Block 3 (blk3)

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

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

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

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

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

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

7 In the **x** text field, type `w_stub/2+w_line/2`.

8 In the **y** text field, type `l_stub/2-w_patch/2`.

9 Right-click **Stub** and choose **Build Selected**.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **blk3** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `-w_stub-w_line`.

5 Right-click **Copy 1 (copy1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **blk2** only.

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

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

5 Select the objects **blk3** and **copy1** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Choose wireframe rendering to get a better view of the interior parts.

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

Continue with the surrounding air and the PML regions.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

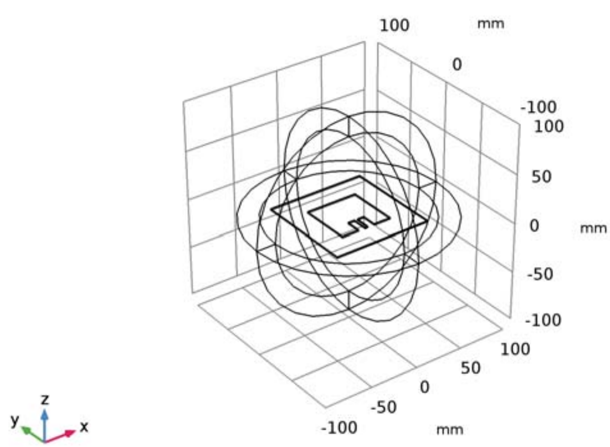
3 In the **Radius** text field, type `l_sub`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	l_sub/5

5 Click **Build All Objects**.

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



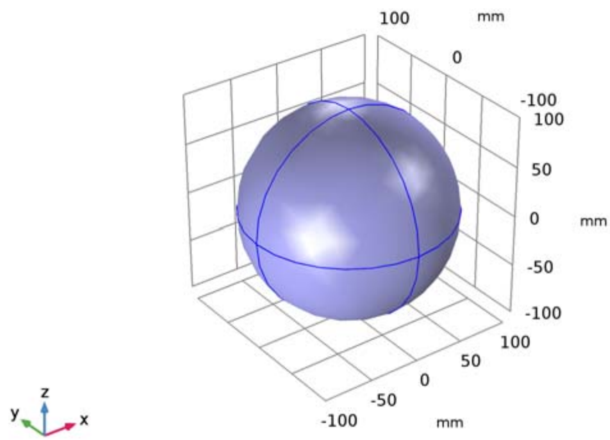
DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1–4 and 8–11 only.

These are all of the outermost domains of the sphere.



3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

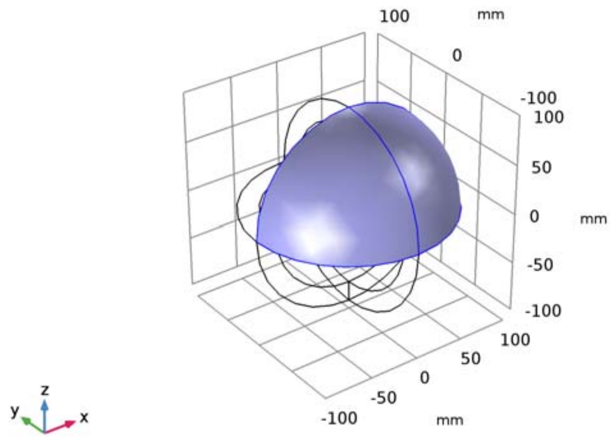
View 1

Suppress some domains and boundaries. This helps to see the interior parts when setting up the physics and reviewing the mesh.

Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **View 1** and choose **Hide for Physics**.

2 Select Domains 2 and 9 only.



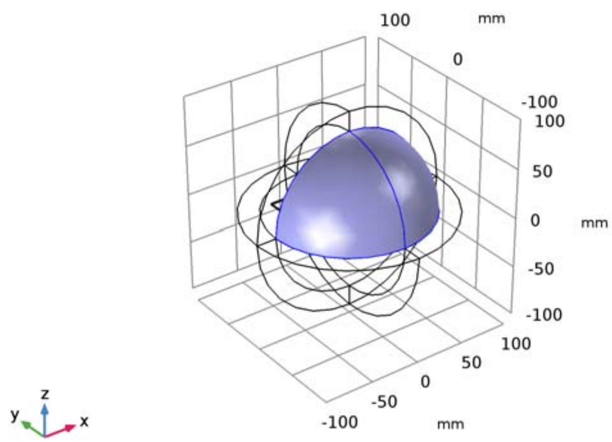
Hide for Physics 2

1 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 10 and 33 only.



Hidden domains and boundaries can be shown by pressing the **Reset Hiding** button in the Graphic Window toolbar.

Before creating the materials for the model, specify the physics. Using this information, the software can detect which material properties are needed.

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**.

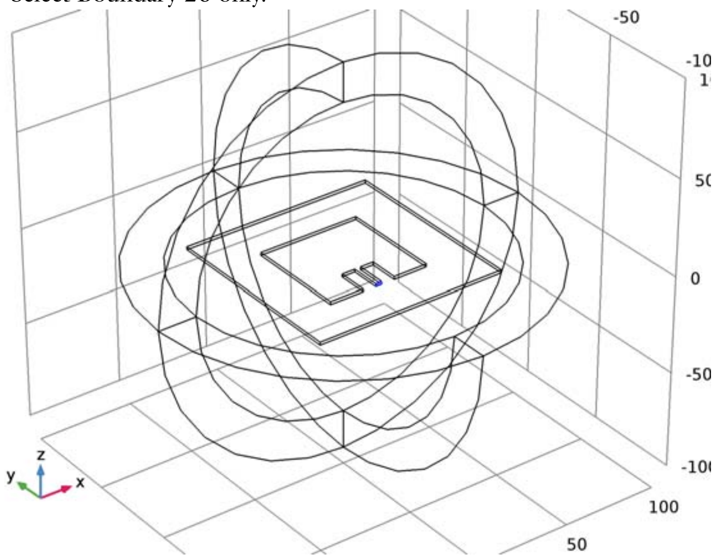
2 Select Boundaries 15, 20, and 21 only.

Lumped Port 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

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

3 Select Boundary 26 only.



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

4 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

ADD MATERIAL

1 On 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.

MATERIALS

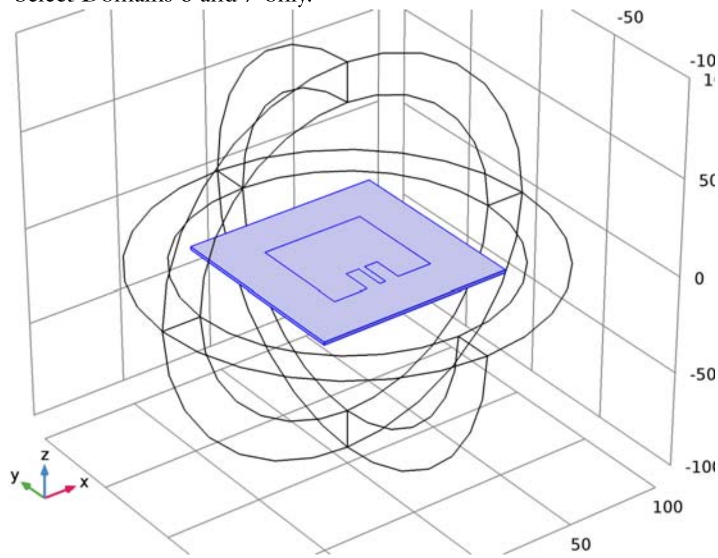
On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 2 (mat2)

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 6 and 7 only.



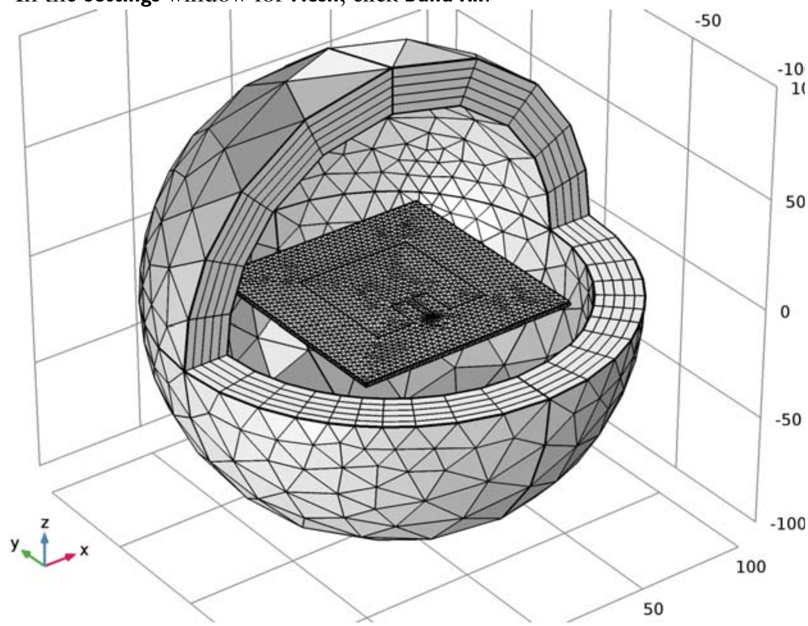
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

MESH 1

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

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



STUDY 1

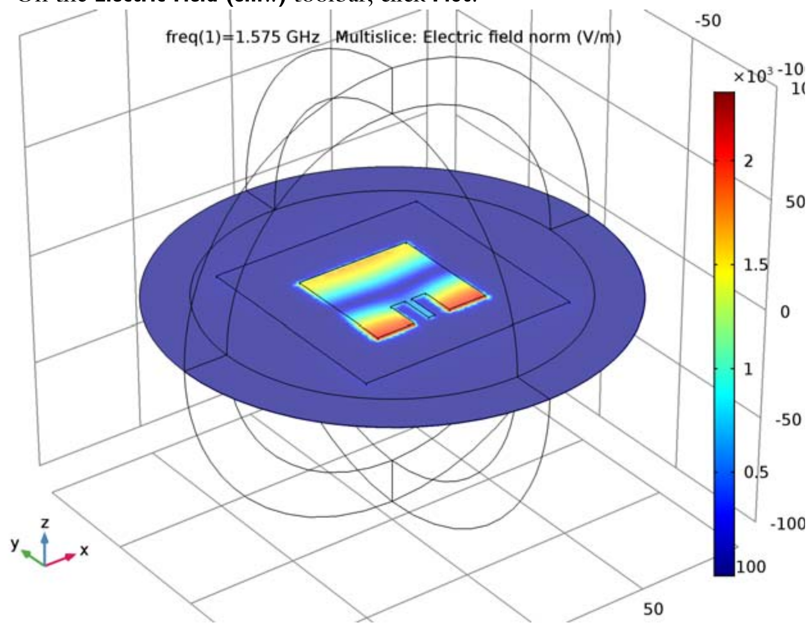
On the **Home** toolbar, click **Compute**.

RESULTS

Multislice

1. In the **Model Builder** window, expand the **Results>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 On the **Electric Field (emw)** toolbar, click **Plot**.



Strong electric fields are observed on the radiating edges.

Far Field 1

1 In the **Model Builder** window, expand the **Results>2D Far Field (emw)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Reference direction** subsection. In the **x** text field, type `0`.

4 In the **y** text field, type `1`.

5 Find the **Normal** subsection. In the **x** text field, type `1`.

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

7 On the **2D Far Field (emw)** toolbar, click **Plot**.

This is the far-field radiation patterns on the E-plane (Figure 3).

3D Far Field (emw)

Compare the 3D far-field radiation pattern plot with Figure 4.

Derived Values

Evaluate the input matching property (S_{11}) at the simulated frequency.

S-parameter, S11dB (emw)

1 In the **Model Builder** window, expand the **Derived Values** node, then click **S-parameter, S11dB (emw)**.

2 In the **Settings** window for **Global Evaluation**, click **Evaluate**.

Created in COMSOL Multiphysics 5.3

Microwave Oven

Introduction

This is a model of the heating process in a microwave oven. The distributed heat source is computed in a stationary, frequency-domain electromagnetic analysis followed by a transient heat transfer simulation showing how the heat redistributes in the food.

Model Definition

The microwave oven is a metallic box connected to a 2.45 GHz microwave source via a rectangular waveguide operating in the TE_{10} mode. Near the bottom of the oven there is a cylindrical glass plate with a spherical potato placed on top of it. The microwave operates at 1 kW, but when we use symmetry to reduce the model size by one half, we only input 500 W in simulation. The symmetry cut is applied vertically through the oven, waveguide, potato, and plate. Figure 1 below shows both the full and reduced size geometry.

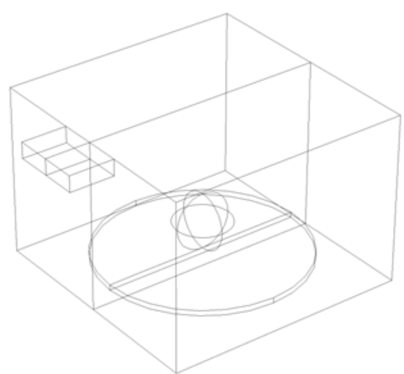


Figure 1: Geometry of the microwave oven, potato, and waveguide feed. Full size (left) and half size (right).

The model uses copper for the walls of the oven and the waveguide. Although resistive metals losses are expected to be small, the *impedance boundary condition* on these walls ensures that they get accounted for. For more information on this boundary condition, see the section Impedance Boundary Condition in the *RF Module User's Guide*. The symmetry cut has mirror symmetry for the electric field and is represented by the boundary condition $\mathbf{n} \times \mathbf{H} = \mathbf{0}$.

The rectangular port is excited by a transverse electric (TE) wave, which is a wave that has no electric field component in the direction of propagation. At an excitation frequency of

2.45 GHz, the TE_{10} mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes are given analytically from the relation

$$(\nu_c)_{mn} = \frac{c}{2}\sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers and c denotes the speed of light. For the TE_{10} mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 7.8$ cm and $b = 1.8$ cm), the TE_{10} mode is the only propagating mode for frequencies between 1.92 GHz and 3.84 GHz.

The port condition requires a propagation constant β, which at the frequency ν is given by the expression

$$\beta = \frac{2\pi}{c}\sqrt{\nu^2 - \nu_c^2}$$

With the stipulated excitation at the rectangular port, the following equation is solved for the electric field vector $\mathbf{E}$ inside the waveguide and oven:

$$\nabla\times(\mu_r^{-1}\nabla\times\mathbf{E}) - k_0^2\left(\varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0}\right)\mathbf{E} = 0$$

where μ_r denotes the relative permeability, j the imaginary unit, σ the conductivity, ω the angular frequency, ε_r the relative permittivity, and ε_0 the permittivity of free space. The model uses material parameters for air: $\sigma = 0$ and $\mu_r = \varepsilon_r = 1$. In the potato the same parameters are used except for the permittivity which is set to $\varepsilon_r = 65 - 20j$ where the imaginary part accounts for dielectric losses. The glass plate has $\sigma = 0$, $\mu_r = 1$ and $\varepsilon_r = 2.55$.

Results and Discussion

Figure 2 below shows the distributed microwave heat source as a slice plot through the center of the potato. The rather complicated oscillating pattern, which has a strong peak in the center, shows that the potato acts as a resonant cavity for the microwave field. The power absorbed in the potato is evaluated and amounts to about 60% of the input microwave power. Most of the remaining power is reflected back through the port.

Figure 3 shows the temperature in the center of the potato as a function of time for the first 5 seconds. Due to the low thermal conductivity of the potato, the heat distributes rather slowly, and the temperature profile after 5 seconds has a strong peak in the center (see Figure 4). When heating the potato further, the temperature in the center eventually reaches 100 °C and the water contents start boiling, drying out the center and transporting heat as steam to outer layers. This also affects the electromagnetic properties of the potato. The simple microwave absorption and heat conduction model used here does not capture these nonlinear effects. However, the model can serve as a starting point for a more advanced analysis.

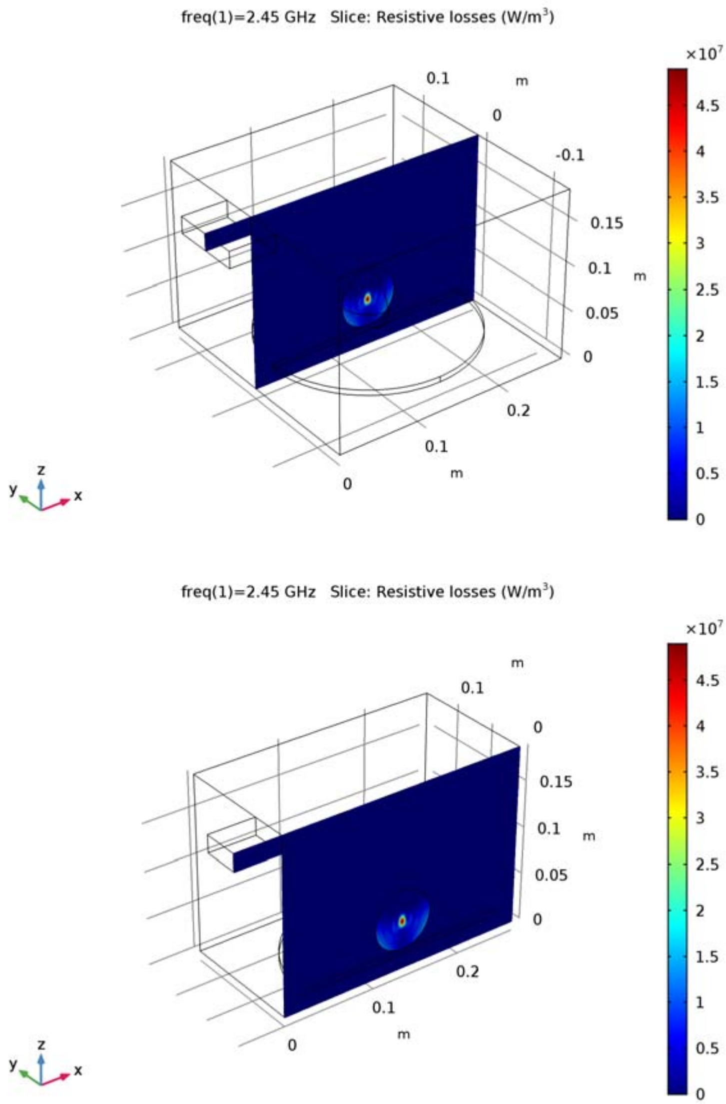
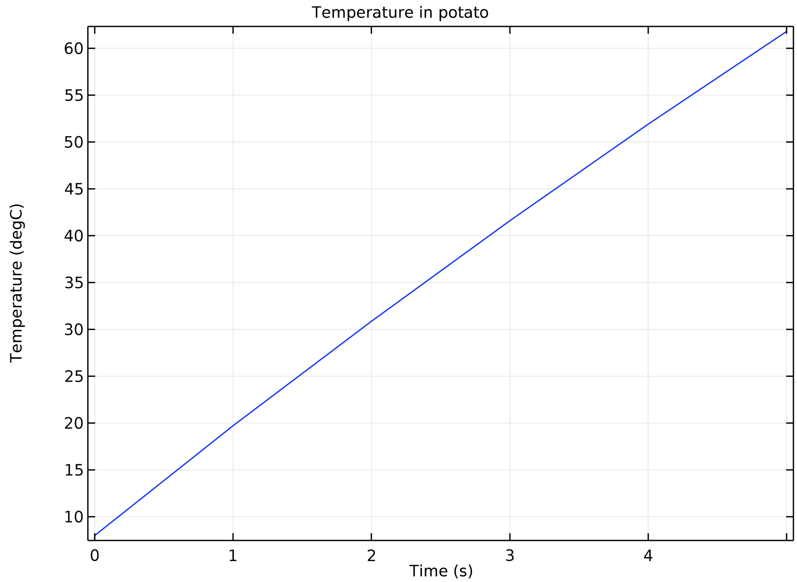


Figure 2: Dissipated microwave power distribution (W/m^3). Full size (top) and half size (bottom).



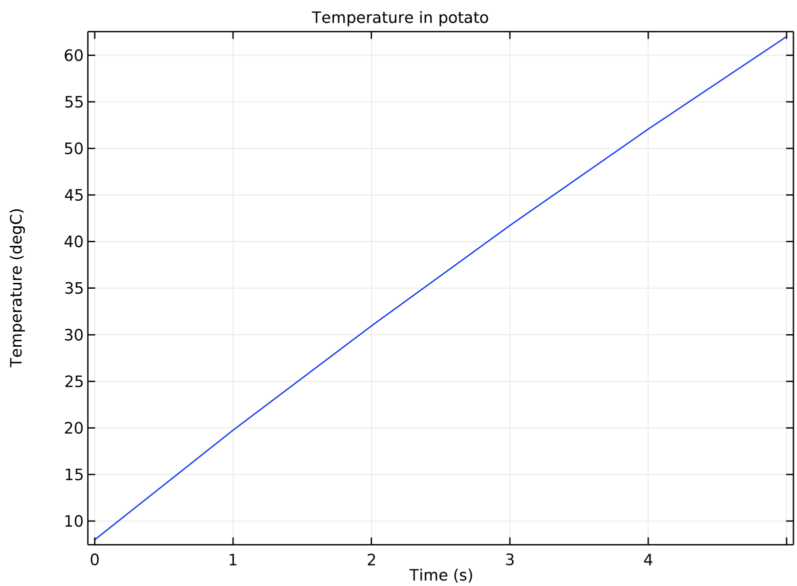


Figure 3: Temperature in the center of the potato during the first 5 seconds of heating. Full size (top) and half size (bottom).

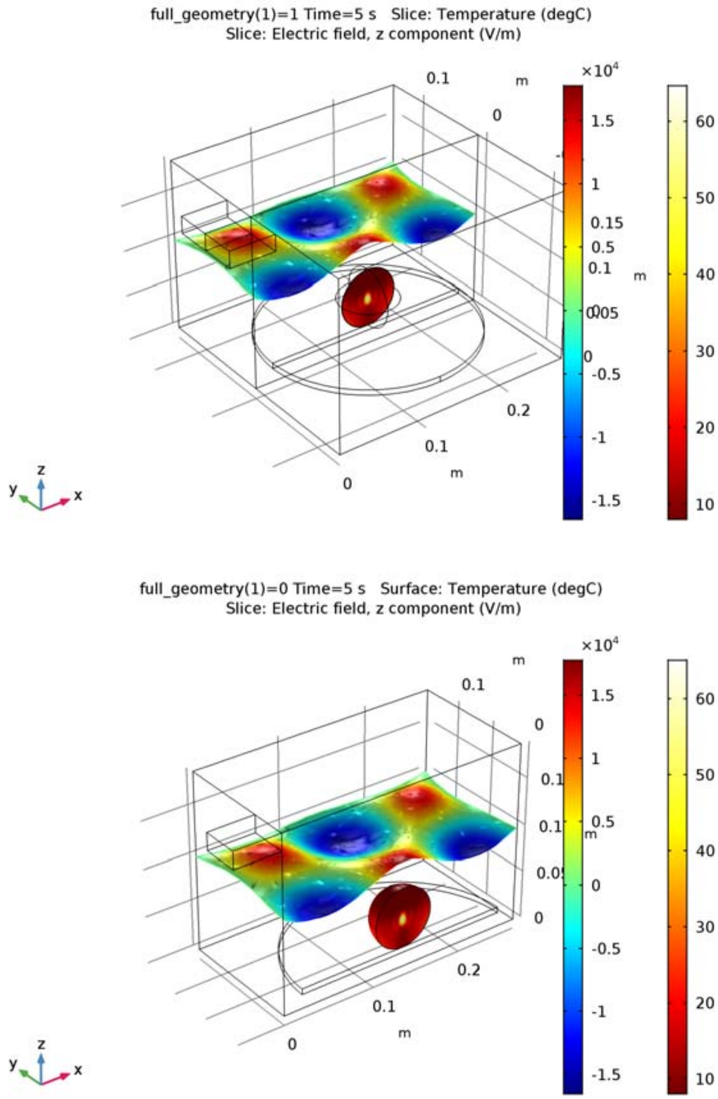


Figure 4: Deformed electric field and Temperature distribution after 5 seconds of heating. Full size (top) and half size (bottom).

Notes About the COMSOL Implementation

In this example model, the material properties of the potato are assumed to be constant as temperature rises, for a simpler and faster numerical modeling. It uses manually configured multiple study steps to perform one-way physics coupling from electromagnetics in the frequency domain to heat transfer in the time domain. Two-way bidirectional physics coupling between electromagnetics and heat transfer, using a predefined multiphysics study step, is addressed in another Application Libraries example, *RF Heating*.

Application Library path: `RF_Module/Microwave_Heating/microwave_oven`

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 **Heat Transfer>Electromagnetic Heating>Microwave Heating**.

3 Click **Add**.

4 In the **Added physics interfaces** tree, select **Electromagnetic Waves, Frequency Domain (emw)**.

5 Click **Study**.

Add a **Frequency Domain** study type for the **Electromagnetic Waves, Frequency Domain** interface.

6 In the **Select Study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Frequency Domain**.

7 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 In the **Frequencies** text field, type `2.45[GHz]`.

4 In the **Model Builder** window, click **Study 1**.

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Select the **Store solution for all intermediate study steps** check box.

Add a **Time Dependent** study for the **Heat Transfer in Solids** interface.

Step 2: Time Dependent

1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

The **Frequency Domain** study is only used for the **Electromagnetic Waves, Frequency Domain** interface, whereas the **Time Dependent** study is only applicable for the**Heat Transfer in Solids** interface in this model. Notice that the electromagnetic heat source will be computed first, and then used in the time-dependent heat transfer study step.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Electromagnetic Waves, Frequency Domain (emw)** interface.

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 **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Heat Transfer in Solids (ht)** interface.

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 Click **Load from File**.

3 Browse to the model's Application Libraries folder and double-click the file `microwave_oven_parameters.txt`.

Block 1 (blk1)

On the **Geometry** toolbar, click **Block**.

GEOMETRY 1

Block 1 (blk1)

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

2 In the **Width** text field, type `wo`.

3 In the **Depth** text field, type `do`.

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

5 Locate the **Position** section. In the **y** text field, type `-do/2`.

Block 2 (blk2)

1 On 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 `wg`.

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

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

6 Locate the **Position** section. In the **x** text field, type `-wg`.

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

8 In the **z** text field, type `ho-hg`.

Cylinder 1 (cyl1)

1 On 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 `rp`.

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

5 Locate the **Position** section. In the **x** text field, type `wo/2`.

6 In the **z** text field, type `bp`.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `rpot`.

4 Locate the **Position** section. In the **x** text field, type `wo/2`.

5 In the **z** text field, type `rpot+bp+hp`.

6 Click **Build All Objects**.

Now, it is possible exploit the mirror symmetry of the model by chopping the geometry and only simulating one half of the model. For this purpose, form a union of all geometric and build an intersection with a block that includes only half of the model.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click the **Select All** button on the **Graphics** toolbar.

Block 3 (blk3)

1 On 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 `0.4`.

4 In the **Depth** text field, type `0.4`.

5 In the **Height** text field, type `0.4`.

6 Locate the **Position** section. In the **x** text field, type `-0.1`.

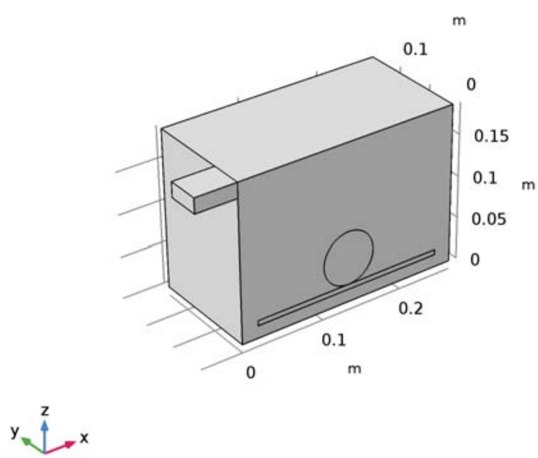
7 Click **Build Selected**.

Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click the **Select All** button on the **Graphics** toolbar.

3 In the **Settings** window for **Intersection**, click **Build All Objects**.



If 1 (if1)

1 On the **Geometry** toolbar, click **Programming** and choose **If + End If**.

2 In the **Settings** window for **If**, type `If Full Geometry` in the **Label** text field.

3 Locate the **If** section. In the **Condition** text field, type `full_geometry`.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **int1** 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 **y** text field, type `1`.

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

7 Click **Build All Objects**.

8 Click the

Created in COMSOL Multiphysics 5.3

MRI Birdcage Coil

Introduction

This example involves designing and optimizing a birdcage coil so it can provide a homogeneous magnetic field distribution for a Magnetic Resonance Imaging (MRI) system. This is usually done to MRI systems so they can generate higher resolution images.

The homogeneous magnetic field is obtained by quadrature excitation and an optimal value of lumped elements in the coil. To find the optimal value at the desired Larmor frequency with an air phantom, a parametric sweep is carried out for the capacitance of the coil's lumped elements. The performance of the coil, loaded with a human head phantom, is also studied.

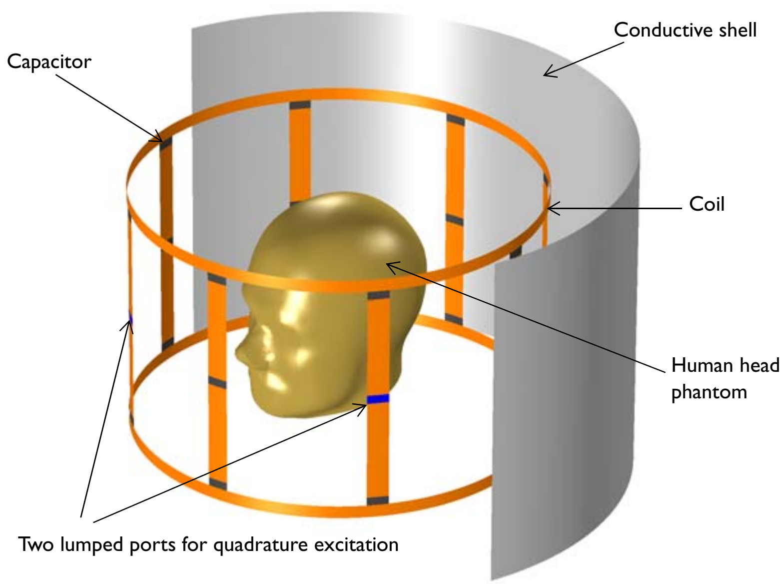


Figure 1: The birdcage coil is shielded by a cylindrical conductive shell. The front part of the shield is removed for visualization purposes. Absorbing boundaries are not included in this figure.

Model Definition

Figure 1 shows the geometry of the example, which consists of a bird cage coil placed around a human head phantom. Noticeably, there are a number of capacitors on the coil. These determine the resonant frequency of the coil and the homogeneity of the field it produces. The coil is placed inside an RF shield. The coil surfaces and the shield around the coil are assigned a Perfect Electric Conductor (PEC) condition. Lumped ports are used to provide quadrature excitation for the coil, while the coil's capacitors are defined using lumped elements.

The air domain around the coil is modeled using an air sphere. Scattering boundary conditions are used along the sphere's boundaries to prevent any reflections into the modeling domain from the outermost boundaries.

To obtain a homogeneous field at the Larmor frequency for an air phantom, the capacitance of the lumped elements in the coil are tuned using a parametric sweep. The circularity of the field is evaluated by estimating the axial ratio of the magnetic field around the phantom. The sum of the axial ratio in dB is evaluated by the line integration of the following quantity:

$$20\log_{10}((B_{\mathrm{right}}+B_{\mathrm{left}})/(B_{\mathrm{right}}-B_{\mathrm{left}}))$$

Here, B_{right} and B_{left} are the right- and left-hand rotating components of the magnetic field, respectively. The homogeneity of the field is quantified by evaluating the standard deviation of the electric field around the phantom.

The automatic mesh control option in the Electromagnetic Waves, Frequency Domain interface is used with the maximum mesh element size as 1/6 of free space wavelength. This example also estimates the homogeneity and circularity of the field in the coil when loaded with a human head phantom.

Results and Discussion

Figure 2 shows the magnetic field around the air phantom with an optimum value for the capacitance of the lumped elements at the Larmor frequency. The real part of the magnetic flux density is almost orthogonal to the imaginary part of it, which indicates the flux is rotating circularly.

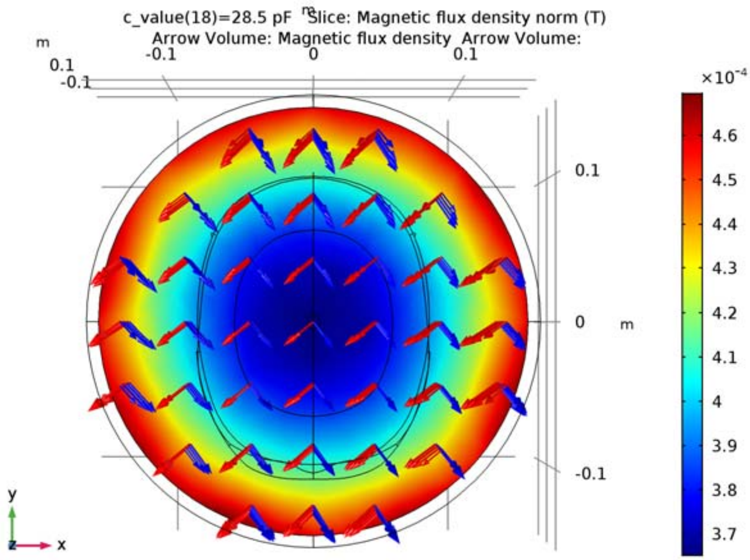


Figure 2: Magnetic density norm distribution with an arrow plot of the real (red) and imaginary (blue) part of the magnetic flux density for the coil only.

Figure 3 shows the field for the coil loaded with the human head phantom. Compared to the case with an air phantom, the uniformity and circularity of the field is distorted, due to the high dielectric loading in the middle of the coil. The coil's capacitors can be tuned further for this loaded case.

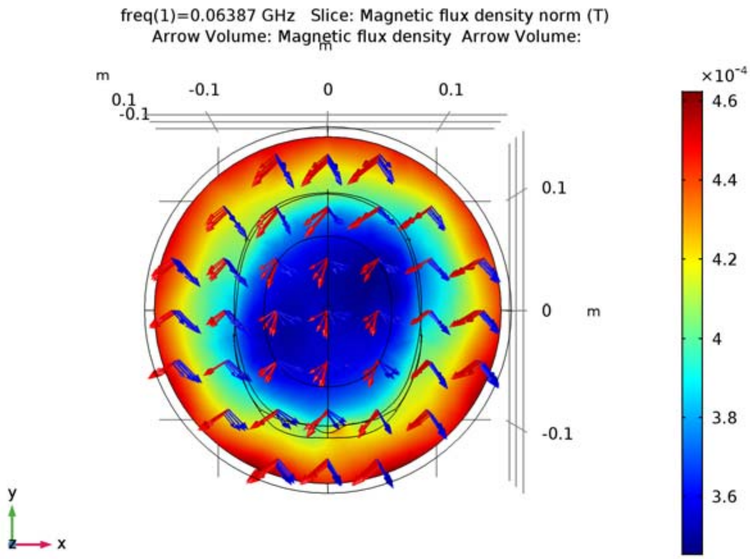


Figure 3: Magnetic density norm distribution with an arrow plot that shows the real (red) and imaginary (blue) part of the magnetic flux density when loaded with the human head phantom

Figure 4 and Figure 5 show the integration of the axial ratio of the magnetic flux density and the standard deviation of the electric field norm around the head model for different values of the lumped elements' capacitances. To achieve a homogeneous circularly-polarized magnetic field, it can be seen that the optimal value of the capacitance is around 28.5 pF.

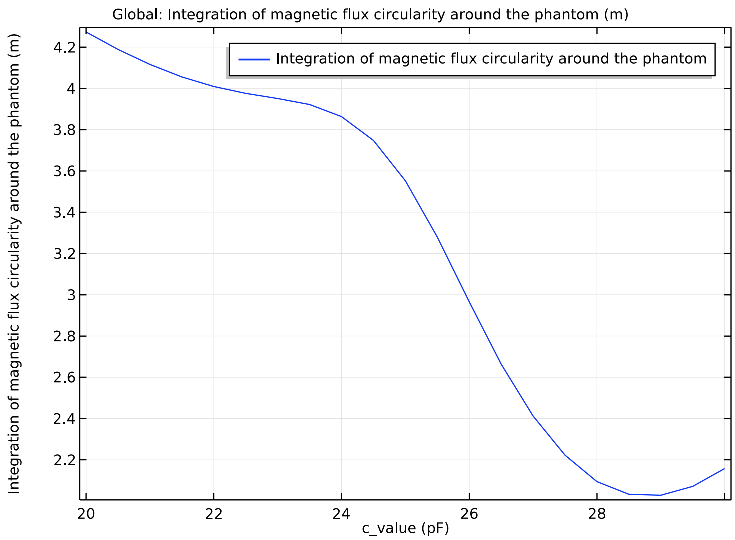


Figure 4: The integration of the axial ratio of the magnetic flux density around the head model

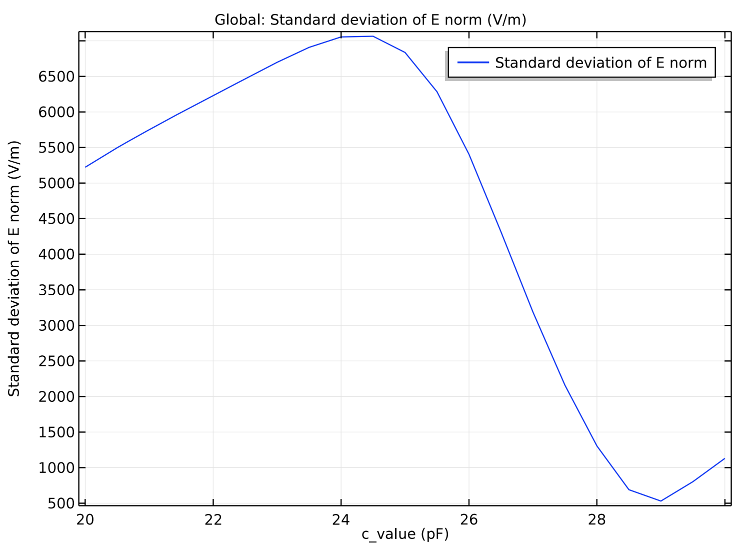


Figure 5: The standard deviation of the electric field norm around the head model

Application Library path: `RF_Module/Passive_Devices/mri_coil`

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 **Preset 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 In the **Frequencies** text field, type `63.87[MHz]`.

GEOMETRY 1

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 In the table, enter the following settings:

Name	Expression	Value	Description
c_value	10[pF]	1E-11 F	Capacitance used on the rungs
r_coil	0.24[m]	0.24 m	Radius of the coil
h_coil	0.3[m]	0.3 m	Height of the coil
l_element	0.01[m]	0.01 m	Length of the capacitive elements

GEOMETRY 1

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

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

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

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

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

Import 1 (imp1)

1 On the **Geometry** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `mri_coil.mphbin`.

Form Union (fin)

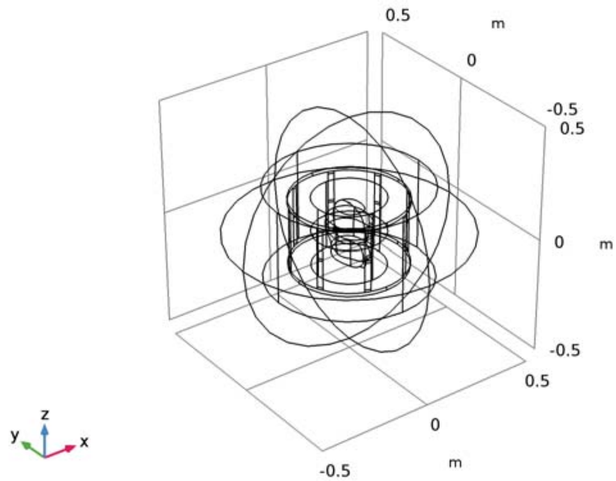
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 In the **Relative repair tolerance** text field, type `2.0E-9`.

Adjusting the tolerance is required only if you build the geometry with CAD kernel. The default tolerance is fine if you use COMSOL kernel.

4 On the **Geometry** toolbar, click **Build All**.



5 Click **Build All**.

Create a set of selections to be used when setting up the physics. First, create a selection for the coil surfaces.

Explicit 1

On the **Definitions** toolbar, click **Explicit**.

DEFINITIONS

Explicit 1

1 Click the **Zoom In** button on the **Graphics** toolbar.

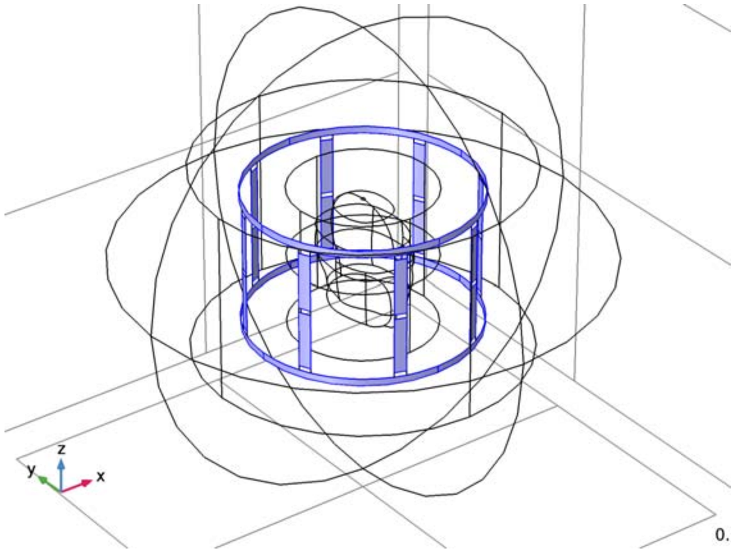
2 In the **Settings** window for **Explicit**, type `Coil` in the **Label** text field.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `9 10 11 12 14 16 19 21 31 33 36 38 66 67 76 77 85 87 90 92 95 97 100 102` in the **Selection** text field.

6 Click **OK**.



Add a selection for the edges around the coil to evaluate the average field.

Explicit 2

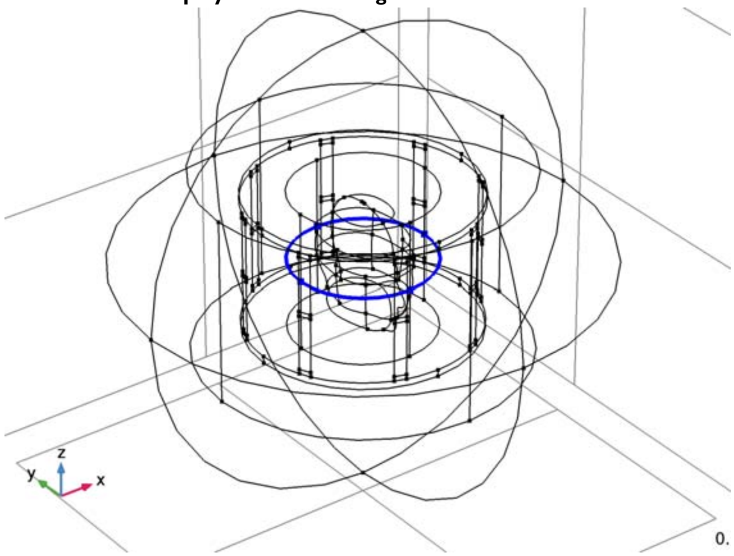
1 On the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Edge 61 only.

5 Select the **Group by continuous tangent** check box.



Add a selection for the absorbing boundaries surrounding the model domain.

Explicit 3

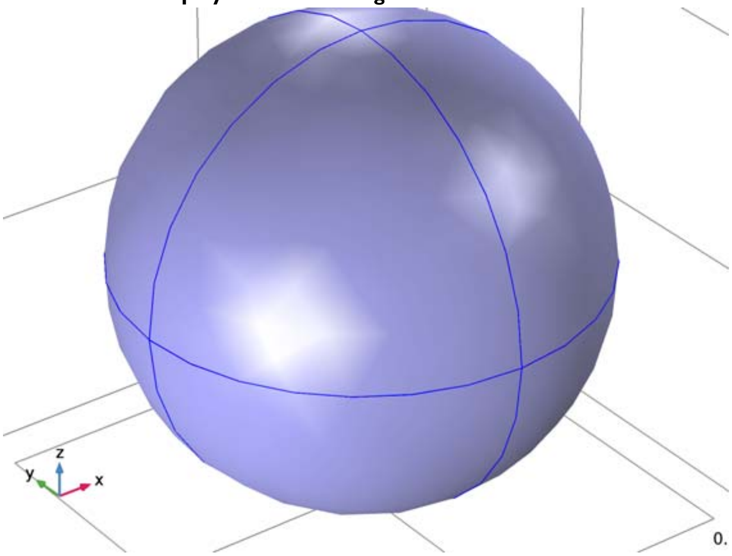
1 On the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundary 4 only.

5 Select the **Group by continuous tangent** check box.



Define the operators to evaluate the average field around the coil.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

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

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

4 From the **Selection** list, choose **Circle**.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

4 From the **Selection** list, choose **Circle**.

Define the variables to evaluate the axial ratio of the magnetic field and the standard deviation of the electric field.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`Bleft`	`abs(emw.Bx+j*emw.By)`	T	`Left hand rotating component of magnetic flux`
`Bright`	`abs(emw.Bx-j*emw.By)`	T	`Right hand rotating component of magnetic flux`
`BaxialratiodB`	`20*log10((Bright+Bleft)/(Bright-Bleft))`		`Magnetic flux axial ratio`
`intBaxialratiodB`	`intop1(abs(BaxialratiodB))`	m	`Integration of magnetic flux circularity around the phantom`
`stdev`	`sqrt(aveop1(emw.normE^2)-aveop1(emw.normE)^2)`	V/m	`Standard deviation of E norm`

View 1

Hide the outermost boundaries to view the interior parts when setting up the physics.

Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** 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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1-5, 8, 23, 26, 29, 63-65, 68-71` in the **Selection** text field.

6 Click **OK**.

MATERIALS

Use the material properties of air for all the domains in the model.

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	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Set up the physics for the model. Define PEC condition for the coil boundaries and provide quadrature excitation for the coil using lumped ports. Make use of lumped elements to model the capacitors in the coil.

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

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

Perfect Electric Conductor 3

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

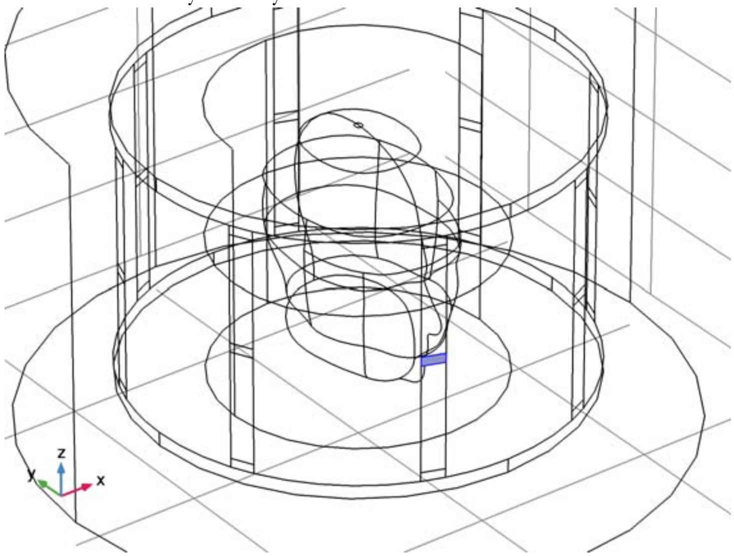
2 Select Boundaries 5, 6, 65, and 78 only.

Lumped Port 1

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

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

3 Select Boundary 37 only.



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

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

5 In the V_0 text field, type `5000`.

Lumped Port 2

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

2 Select Boundary 101 only.

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

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

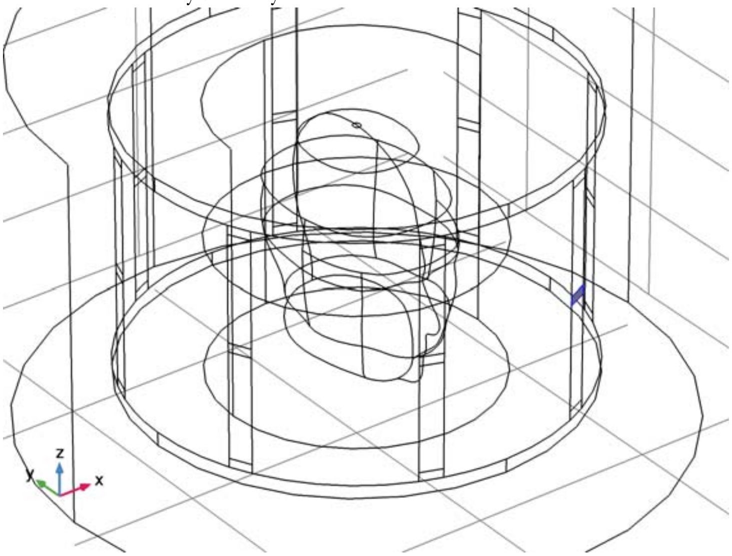
5 In the V_0 text field, type `5000`.

6 In the θ_{in} text field, type `pi/2`.

Lumped Element 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 91 only.



3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 96 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 3

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 86 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 4

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 32 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 5

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 15 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 6

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 20 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 7

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 35 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 8

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 89 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 9

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 99 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 10

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 94 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 11

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 84 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 12

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 30 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 13

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 13 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 14

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 18 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 15

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 39 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 16

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 93 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 17

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 103 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 18

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 98 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 19

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 88 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 20

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 34 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 21

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 17 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Lumped Element 22

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 22 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Capacitor**.

5 In the C_{element} text field, type `c_value`.

Scattering Boundary Condition 1

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

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

3 From the **Selection** list, choose **Absorbing boundaries**.

STUDY 1

Add a parametric sweep for the capacitance of the lumped elements in the coil.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
c_value		

5 Click **Range**.

6 In the **Range** dialog box, type `20` in the **Start** text field.

7 In the **Step** text field, type `0.5`.

8 In the **Stop** text field, type `30`.

9 Click **Replace**.

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

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
c_value	`range(20,0.5,30)`	pF

12 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

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

Study 1/Solution 1 (sol1)

Add a selection for the domains around the phantom to visualize the fields.

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domains 3–6 only.

Add a slice plot for the magnetic field and arrow plots to view the direction of the magnetic field.

3D Plot Group 2

1 On the **Results** toolbar, click **3D Plot Group**.

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

3 From the **Parameter value (c_value (pF))** list, choose **28.5**.

4 On the **3D Plot Group 2** toolbar, click **Slice**.

Slice 1

1 In the **Model Builder** window, under **Results>3D Plot Group 2** click **Slice 1**.

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

3 From the **Plane** list, choose **XY-planes**.

4 In the **Planes** text field, type `1`.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>emw.normB - Magnetic flux density norm**.

6 On the **3D Plot Group 2** toolbar, click **Arrow Volume**.

Arrow Volume 1

1 In the **Model Builder** window, under **Results>3D Plot Group 2** click **Arrow Volume 1**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>emw.Bx,...,emw.Bz - Magnetic flux density**.

3 On the **3D Plot Group 2** toolbar, click **Arrow Volume**.

Arrow Volume 2

1 In the **Model Builder** window, under **Results>3D Plot Group 2** click **Arrow Volume 2**.

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

3 In the **X component** text field, type `imag(emw.Bx)`.

4 In the **Y component** text field, type `imag(emw.By)`.

5 In the **Z component** text field, type `imag(emw.Bz)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

7 On the **3D Plot Group 2** toolbar, click **Plot**.

8 Click the **Go to XY View** button on the **Graphics** toolbar.

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

The plot shows the homogeneous and circularly polarized magnetic field around the air phantom. It is plotted in Figure 2.

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Global**.

3 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>Definitions>Variables>intBaxialratiodB - Integration of magnetic flux circularity around the phantom**.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 On the **1D Plot Group 3** toolbar, click **Plot**.

Global 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Global**.
3. 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>Definitions>Variables>stdev - Standard deviation of E norm**.

1D Plot Group 4

1. In the **Model Builder** window, under **Results** click **1D Plot Group 4**.
2. On the **1D Plot Group 4** toolbar, click **Plot**.

   It is obvious from the 1D plots that the optimum capacitance value for obtaining homogeneous magnetic field is around 28.5 pF. It is plotted in Figure 4 and Figure 5.

GLOBAL DEFINITIONS

Parameters

Now modify the capacitance of the lumped elements and rerun the model with the human head phantom.

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

Name	Expression	Value	Description
`c_value`	`28.5[pF]`	2.85E-11 F	`Capacitance used on the rungs`

MATERIALS

Material 2 (mat2)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. Select Domains 5 and 6 only.
3. In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	40	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0.9	S/m	Basic

ADD STUDY

1 On 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> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Frequency Domain

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

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

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

4 In the **Frequencies** text field, type `63.87[MHz]`.

5 On the **Home** toolbar, click **Compute**.

RESULTS

Study 2/Solution 2 (sol2)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 2/ Solution 2 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

Add a selection for the domains around the phantom to visualize the fields.

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

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

4 Select Domains 3–6 only.

3D Plot Group 6

1 On the **Results** toolbar, click **3D Plot Group**.

2 On the **3D Plot Group 6** toolbar, click **Slice**.

3D Plot Group 6

1 In the **Model Builder** window, under **Results** click **3D Plot Group 6**.

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

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

Slice 1

1 In the **Model Builder** window, under **Results>3D Plot Group 6** click **Slice 1**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>emw.normB - Magnetic flux density norm**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.

4 In the **Planes** text field, type `1`.

3D Plot Group 6

1 In the **Model Builder** window, under **Results** click **3D Plot Group 6**.

2 On the **3D Plot Group 6** toolbar, click **Arrow Volume**.

Arrow Volume 1

1 In the **Model Builder** window, under **Results>3D Plot Group 6** click **Arrow Volume 1**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>emw.Bx,...,emw.Bz - Magnetic flux density**.

3D Plot Group 6

1 In the **Model Builder** window, under **Results** click **3D Plot Group 6**.

2 On the **3D Plot Group 6** toolbar, click **Arrow Volume**.

Arrow Volume 2

1 In the **Model Builder** window, under **Results>3D Plot Group 6** click **Arrow Volume 2**.

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

3 In the **X component** text field, type `imag(emw.Bx)`.

4 In the **Y component** text field, type `imag(emw.By)`.

5 In the **Z component** text field, type `imag(emw.Bz)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

3D Plot Group 6

1 In the **Model Builder** window, under **Results** click **3D Plot Group 6**.

2 On the **3D Plot Group 6** toolbar, click **Plot**.

The plot shows that the magnetic field is homogeneous and circularly polarized even when the coil is loaded with the human head phantom. It is plotted in Figure 3.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

Evaluate the axial ratio of the magnetic field and the standard deviation of the electric field with the human head phantom.

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>intBaxialratiodB - Integration of magnetic flux circularity around the phantom**.

5 Click **Evaluate**.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>stdev - Standard deviation of E norm**.

5 Click **Evaluate**.

Appendix: Geometry 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 Click **Done**.

GEOMETRY 1

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 Click **Load from File**.

3 Browse to the model's Application Libraries folder and double-click the file `mri_coil_parameters.txt`.

4 In the table, enter the following settings:

Name	Expression	Value	Description
`t_ring`	`0.015[m]`	0.015 m	

GEOMETRY 1

Cylinder 1 (cyl1)

1 On 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 `r_coil`.

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

5 Locate the **Position** section. In the **z** text field, type `-h_coil/2`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`t_ring`

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

9 Select the **Layers on top** check box.

Work Plane 1 (wp1)

1 On 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 `-h_coil/2+t_ring`.

4 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `r_coil`.

4 In the **Sector angle** text field, type `6`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-22.5`.

Convert to Curve 1 (ccur1)

1 On the **Work Plane** toolbar, click **Conversions** and choose **Convert to Curve**.

2 Select the object **c1** only.

3 Right-click **Convert to Curve 1 (ccur1)** and choose **Build Selected**.

4 On the **Work Plane** toolbar, click **Delete**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** click **Delete Entities 1 (del1)**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 Click the **Zoom In** button on the **Graphics** toolbar.

5 Click the **Zoom In** button on the **Graphics** toolbar.

6 On the object **ccur1**, select Boundaries 2 and 3 only.

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

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 From the **Input object handling** list, choose **Keep**.

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

Distances (m)
`h_coil-2*t_ring`

Extrude 2 (ext2)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
l_element

Move 1 (mov1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **ext2** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **z** text field, type `0 (h_coil-2*t_ring)/2-l_element/2 (h_coil-2*t_ring)-l_element`.

Rotate 1 (rot1)

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

2 Select the objects **mov1(2)**, **ext1**, **mov1(3)**, and **mov1(1)** only.

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

4 In the **Rotation** text field, type `0 45 90 135 180 225 270 315`.

5 On the **Geometry** toolbar, click **Build All**.

Convert to Surface 1 (csur1)

1 On the **Geometry** toolbar, click **Conversions** and choose **Convert to Surface**.

2 Click the **Select Box** button on the **Graphics** toolbar.

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

4 Right-click **Convert to Surface 1 (csur1)** and choose **Build Selected**.

5 On the **Geometry** toolbar, click **Delete**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Delete Entities 1 (del1)**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

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

5 On the object **csur1**, select Boundaries 3–6, 9, 10, 21, 22, 33, 34, 36, 39, 51, 52, 63, and 64 only.

Cylinder 2 (cyl2)

1 On 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 `0.15`.

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

5 Locate the **Position** section. In the **z** text field, type `-h_coil/2`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`h_coil/2`

7 Clear the **Layers on side** check box.

8 Select the **Layers on top** check box.

Cylinder 3 (cyl3)

1 On 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 `r_coil+0.1`.

4 In the **Height** text field, type `h_coil+0.1`.

5 Locate the **Position** section. In the **z** text field, type `-(h_coil+0.1)/2`.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `0.5`.

4 On the **Geometry** toolbar, click **Build All**.

Created in COMSOL Multiphysics 5.3

Microwave Filter on PCB

This example analyzes the transfer function of a low-pass filter on a printed circuit board.

Introduction

Microstrip filters can be fabricated directly on a printed circuit board (PCB) with a microstrip line going from the input to the output. Along the microstrip line there are a number of stubs of certain lengths and widths. The design of the filter involves choosing the impedance of the microstrip line, the impedance of the stub microstrips, and the stub lengths. This particular filter is based on a textbook example from Ref. 2. It is also used as example in Ref. 1, which contains results from other simulation tools and methods and is freely available online. The filter has a seven-pole low-pass Chebyshev response with a cutoff frequency of 1 GHz.

Model Definition

The model uses the Electromagnetic Waves interface that solves the vector Helmholtz wave equation. The cutoff frequency of the filter is 1 GHz by design, and the dielectric layer of the PCB has a relative permittivity of 10.8. The metal layers are modeled as perfect electric conductors with zero thickness, thereby avoiding a dense meshing of thin conductive layers. The width of the microstrip line is 0.1 mm and the width of the stubs is 5 mm.

The characteristics of the filter are sensitive to the placement and length of the stubs; therefore this example also analyzes the change in filter characteristics as a function of mechanical deformation. This is done by adding Solid Mechanics and Moving Mesh interfaces. The Moving Mesh interface is necessary to enable the Electromagnetic Waves interface to account for the deformation of the PCB. The deformation comes from a uniform load across the board with fixed input and output faces.

Because the filter cutoff should be close to 1 GHz, the frequency is swept from 750 MHz up to 1.5 GHz. The first solution step performs this sweep for the Electromagnetic Waves interface without any mechanical deformation. Then a uniform load of 40 N is applied to the PCB, generating a large deformation of the board. The Solid Mechanics interface calculates the deformation, and the Moving Mesh interface applies this deformation to the coordinate system that the Electromagnetic Waves interface uses. After this step, the frequency sweep is performed again for the Electromagnetic Waves interface using the parametric solver.

This example accounts for the structural deformation in the sense that it solves for the electromagnetic fields on the deformed geometry, as if the PCB was manufactured in the deformed shape—free of stress.

Results and Discussion

The purpose of this simulation is to analyze how the S-parameter curve changes when a force of 40 N is applied on the circuit board. This force bends the PCB significantly, as you can see in Figure 1.

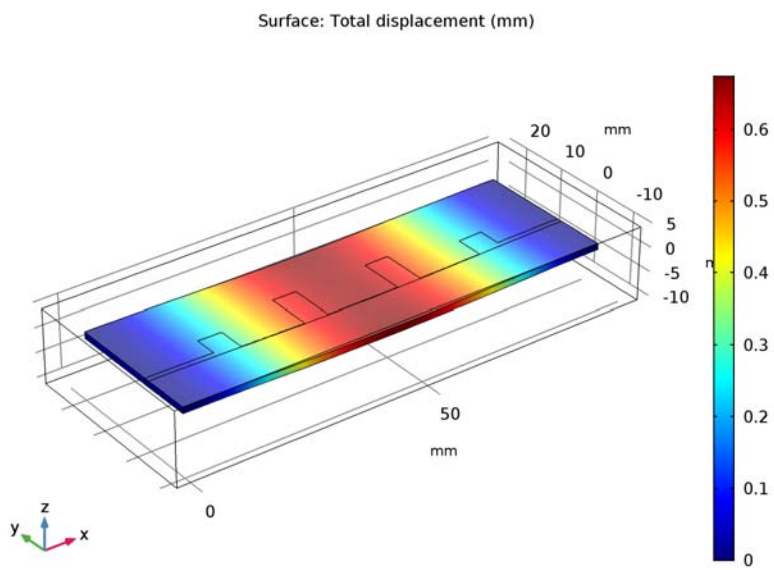


Figure 1: The graph shows the total displacement of the PCB, due to the load.

Although the PCB deformation is fairly large the S-parameter curve does not change that much. The cutoff frequency is shifted less than 10 MHz when the force is applied. Figure 2 displays the difference between the S-parameter curves with and without an applied force.

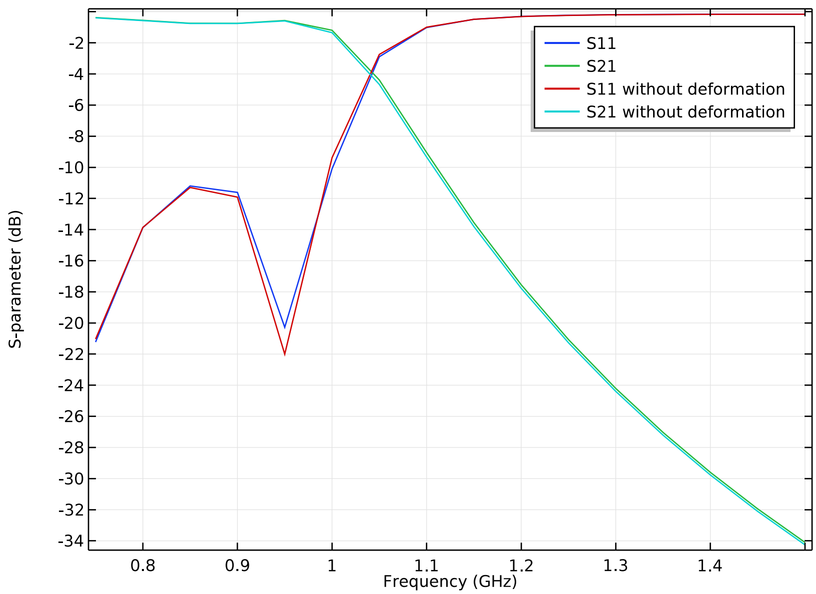


Figure 2: A comparison between the S_{11} and S_{21} parameters before and after a force of 40 N has been applied on the PCB. The red and turquoise lines correspond to the S-parameter curves for the filter with an applied force.

References

1. D.V. Tosic and M. Potrebic, "Software Tools for Research and Education," *Microwave Review*, vol. 12, no. 2, pp. 45–54, 2006.

2. J.-S.G. Hong and M.J. Lancaster, *Microstrip Filters for RF/Microwave Applications*, John Wiley & Sons, 2001.

Application Library path: `RF_Module/Filters/pcb_microwave_filter_with_stress`

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 **Preset Studies>Frequency Domain**.
6 Click **Done**.

GLOBAL DEFINITIONS

The following steps define the parameters for the frequency sweep.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
fstart	750[MHz]	7.5E8 Hz	Start frequency
fstop	1.5[GHz]	1.5E9 Hz	Stop frequency
fstep	50[MHz]	5E7 Hz	Frequency step

GEOMETRY 1

Set mm as the default unit for length.

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**.

Block 1 (blk1)

1 On 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 `89.49`.
4 In the **Depth** text field, type `29.54`.

5 In the **Height** text field, type `1.27`.

6 Locate the **Position** section. In the **y** text field, type `-10`.

7 Click **Build Selected**.

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

Work Plane 1 (wp1)

1 On 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 `1.27`.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** click **Plane Geometry**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `13.88`.

4 In the **Height** text field, type `1.125`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `5`.

4 In the **Height** text field, type `5.86`.

5 Locate the **Position** section. In the **xw** text field, type `13.88`.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `13.32`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **xw** text field, type `18.88`.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `5`.

4 In the **Height** text field, type `9.54`.

5 Locate the **Position** section. In the **xw** text field, type `32.2`.

Rectangle 5 (r5)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `15.09`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **xw** text field, type `37.2`.

Mirror 1 (mir1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **r2**, **r4**, **r1**, and **r3** only.

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

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

5 Locate the **Point on Line of Reflection** section. In the **xw** text field, type `44.745`.

Union 1 (uni1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

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

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

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

Work Plane 2 (wp2)

1 On 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 `1.27`.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>**
Work Plane 2 (wp2) click **Plane Geometry**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **yw** to `1.125`.

Mirror 1 (mir1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **b1** only.

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

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

5 Locate the **Point on Line of Reflection** section. In the **xw** text field, type `44.745`.

6 Click **Build Selected**.

The next step is to add boundary faces for the input and output ports by extruding the lines.

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

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (mm)
-1.27

Block 2 (blk2)

1 On 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 `100`.

4 In the **Depth** text field, type `40`.

5 In the **Height** text field, type `15`.

6 Locate the **Position** section. In the **x** text field, type `-5`.

7 In the **y** text field, type `-15`.

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

9 On the **Geometry** toolbar, click **Build All**.

ADD MATERIAL

1 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-In>FR4 (Circuit Board)**.
3 Click **Add to Component** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat2)**.
2 Select Domain 2 only.

The relative permittivity is modified to agree with the value used in Ref. 1. The FR4 material is selected to provide parameters for the solid mechanics simulation.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.
4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	10.8	1	Basic

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Scattering Boundary Condition 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Scattering Boundary Condition**.
2 Select Boundaries 1–5 and 18 only.

Lumped Port 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 10 only.

For the first port, wave excitation is **on** by default. This port excites the microstrip line.

Lumped Port 2

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

2 Select Boundary 16 only.

Perfect Electric Conductor 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Perfect Electric Conductor**.

2 Select Boundaries 8 and 11 only.

These boundaries represent the microstrip line and the ground plane on the PCB.

MESH 1

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

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

DEFINITIONS

Probe plotting is a convenient technique to plot while solving, which is very useful for parameter sweeps. It is possible to discover problems before the solution step has finished, and then stop the sweep to save time. It is also useful in situations when the solver does more steps than it stores in the output. The probe plot will contain all steps that the solver takes.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `S11` in the **Variable name** text field.

3 Locate the **Expression** section. Click **emw.S11dB - S11** in the upper-right corner of the section. In the **Model Builder** window, right-click **Definitions** and choose **Global Variable Probe**.

4 In the **Settings** window for **Global Variable Probe**, type `S21` in the **Variable name** text field.

5 Locate the **Expression** section. Click **emw.S21dB - S21** in the upper-right corner of the section.

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**, click to expand the **Results while solving** section.
3 Locate the **Results While Solving** section. Select the **Plot** check box.
4 Locate the **Study Settings** section. Click **Range**.
5 In the **Range** dialog box, type `fstart` in the **Start** text field.
6 In the **Step** text field, type `fstep`.
7 In the **Stop** text field, type `fstop`.
8 Click **Replace**.
9 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The **Electric Field** plot group under the **Results** node, shows the norm of the electric field. You can change the frequency by selecting another value from the Parameter value (freq) list box.

The Probe 1D Plot Group 2 displays the S_{11}- and S_{21}-parameters for the frequency sweep.

COMPONENT 1 (COMP1)

The following instructions adds physics from the Solid Mechanics and the Moving Mesh interfaces for the simulations of the deformed PCB.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.
4 Click **Add to Component** in the window toolbar.

ADD PHYSICS

1 Go to the **Add Physics** window.
2 In the tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.
3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 2 only.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

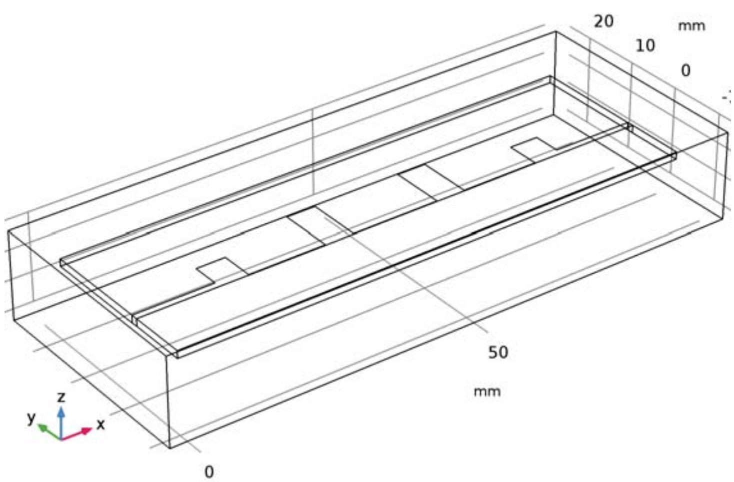
3 In the table, enter the following settings:

Name	Expression	Value	Description
fload	40[N]	40 N	Load on PCB

The following steps describe how to measure the volume of the PCB and then copy and paste the value in a parameter definition.

GEOMETRY 1

1 Click the **Select Domains** button on the **Graphics** toolbar.



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

3 On the object **fin**, select Domain 2 only.

4 On the **Geometry** toolbar, click **Measure**.

Click the Measure button from the toolbar. The volume of the PCB domain is displayed in the Messages window.

Copy the volume of the PCB from the Messages table.

GLOBAL DEFINITIONS

Parameters

(by pasting in the previously copied volume)

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
V	3357.0[mm^3]	3.357E-6 m^3	Volume of PCB

MOVING MESH (ALE)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Moving Mesh (ale)**.

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

Prescribed Deformation 1

1 Right-click **Moving Mesh (ale)** and choose **Prescribed Deformation**.

2 Select Domain 2 only.

3 In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Mesh Displacement** section.

4 In the d_x text-field array, type `u` on the first row.

5 In the d_y text-field array, type `v` on the second row.

6 In the d_z text-field array, type `w` on the third row.

Free Deformation 1

1 Right-click **Moving Mesh (ale)** and choose **Free Deformation**.

2 Select Domain 1 only.

Prescribed Mesh Displacement 2

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

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

3 From the **Selection** list, choose **All boundaries**.

4 Select Boundaries 6–17 only.

5 Locate the **Prescribed Mesh Displacement** section. In the d_x text field, type u.

6 In the d_y text field, type v.

7 In the d_z text field, type w.

SOLID MECHANICS (SOLID)

Body Load 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Volume Forces>Body Load**.

2 Select Domain 2 only.

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_V$ vector as

0	x
0	y
-fload/V	z

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundaries 6, 10, 12, and 15–17 only.

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 **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Solid Mechanics** and **Moving Mesh**.

ADD STUDY

1 On 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 **Custom Studies>Preset Studies for Some Physics Interfaces>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electromagnetic Waves, Frequency Domain (emw)**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 2: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Solid Mechanics** and **Moving Mesh**.

4 Locate the **Study Settings** section. Click **Range**.

5 In the **Range** dialog box, type `fstart` in the **Start** text field.

6 In the **Step** text field, type `fstep`.

7 In the **Stop** text field, type `fstop`.

8 Click **Replace**.

9 In the **Settings** window for **Frequency Domain**, locate the **Results While Solving** section.

10 Select the **Plot** check box.

11 From the **Plot group** list, choose **S-Parameter (emw)**.

12 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw) 1

The default plot shows a Multislice plot of the norm of the electric field for the last frequency in the sweep. The plot can be updated for any of the frequencies used, by selecting another frequency from the Parameter value (freq) list box.

S-Parameter (emw) 1

To compare the S-parameters for the initial and the stressed PCB, add the S-parameter from the first different solutions.

Global 1

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

Global 2

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

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

3 From the **Data set** 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 without deformation
emw.S21dB	dB	S21 without deformation

5 On the **S-Parameter (emw) 1** toolbar, click **Plot**.

You should now see the plot in Figure 2.

Stress (solid)

In the **Model Builder** window, expand the **Results>Stress (solid)** node.

Deformation

You may expand all subnodes under Results>Stress (solid) to reach Deformation.

1 In the **Model Builder** window, expand the **Results>Stress (solid)>Surface 1** node.

2 Right-click **Deformation** and choose **Disable**.

Surface 1

1 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>solid.disp - Total displacement**.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

3 From the **Data set** list, choose **Study 2/Solution Store 1 (sol3)**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 On the **Stress (solid)** toolbar, click **Plot**.

You should now see the plot in Figure 1.

Created in COMSOL Multiphysics 5.3

Polarized Circular Ports

Introduction

This example of a circular waveguide demonstrates how to excite and terminate ports with degenerate port modes. In particular it shows how to model and excite the TE11 mode of circular waveguides in 3D.

Model Definition

A straight piece of circular waveguide with perfect metallic walls is excited by a linearly polarized TE_{11} mode at one end and ideally terminated at the other end.

The TE_{11} mode of a circular waveguide is degenerate, meaning that there is an infinite number of possible variants of the TE_{11} mode that only differ in polarization. Any type of polarization (for example circular polarization) of the TE_{11} mode can be constructed by or decomposed into two linearly polarized modes. The direction of polarization of the first one can be chosen freely and the second one is obtained from the first one by a rotation of 90 degrees around the waveguide axis.

As a general structure may change the polarization of the incident field as it is transmitted or reflected, ideal termination means that any circular waveguide port that operates in the TE_{11} mode need to have two port features which are tuned to mutually orthogonal, linear polarizations of the TE_{11} mode respectively. The reference directions for the two port features are subject to a manual choice but must differ by a rotation of 90 degrees around the waveguide axis.

The Port subfeature, Circular Port Reference Axis is used to determine the reference direction for the polarization of each mode/port by means of selecting two vertices (points) on the port circumference. In this example, extra vertices that are equally distributed along the port edge are added to the geometry to allow for the definition of the desired reference directions.

With the stipulated excitation using the two mutually orthogonal TE_{11} ports as boundary conditions, the following equation is solved for the electric field vector $\mathbf{E}$ inside the waveguide:

$$\nabla\times(\mu_r^{-1}\nabla\times\mathbf{E})-k_0^2\left(\varepsilon_r-\frac{j\sigma}{\omega\varepsilon_0}\right)\mathbf{E} = 0$$

Results and Discussion

The first TE_{11} mode of the inport is shown in Figure 1.

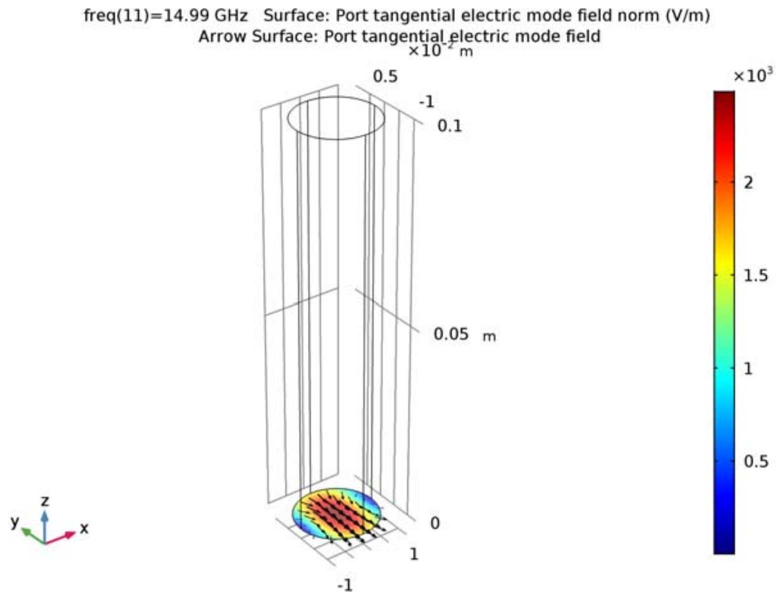


Figure 1: The first TE_{11} mode of the inport

Note: Depending on the details of the mesh, which in turn may depend on the origin of the CAD geometry, a mode that is rotated 180 degrees may be found.

The first TE_{11} mode of the outport is shown in Figure 2.

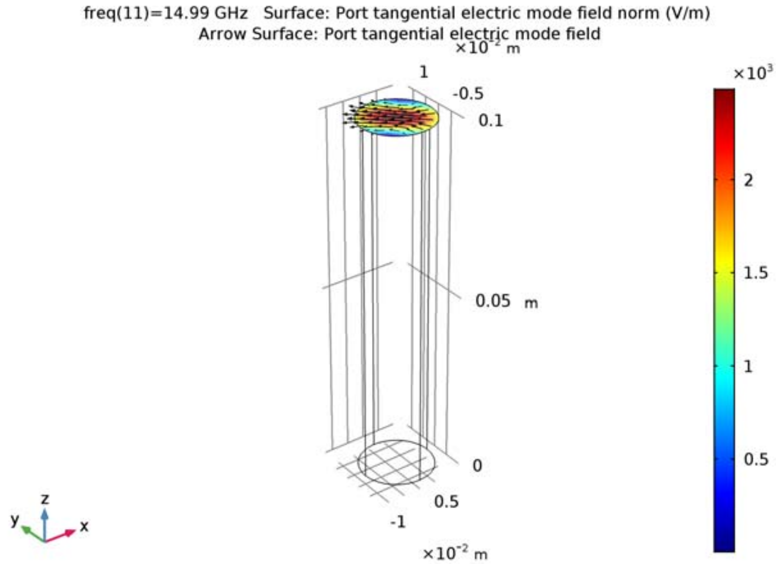


Figure 2: The first TE_{11} mode of the outport.

Note: Depending on the details of the mesh, which in turn may depend on the origin of the CAD geometry, a mode that is rotated 180 degrees may be found.

The transmission coefficients between the inport and outport modes are shown in Figure 3.

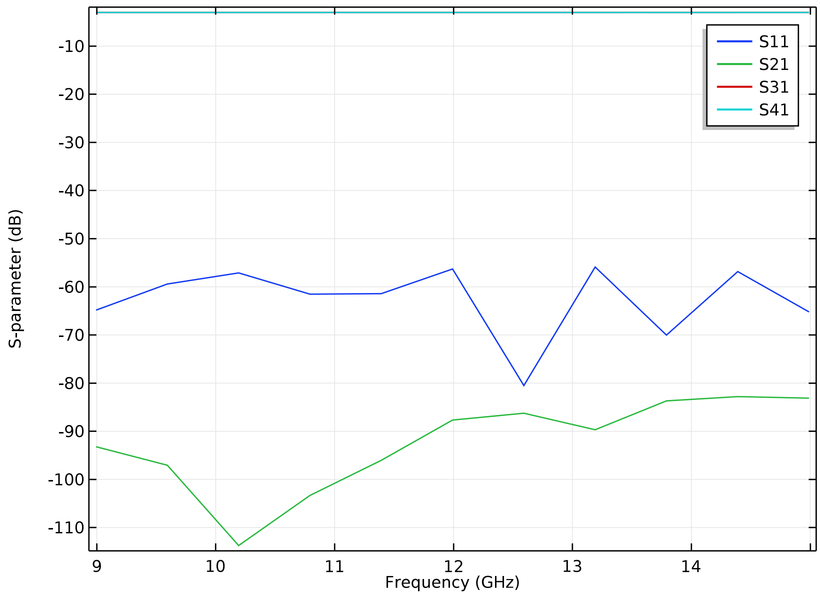


Figure 3: The transmission coefficients between inport modes and outport modes are plotted as a function of frequency. As the port modes are misaligned by 45 degrees the transmission coefficients approach the -3dB level.

Application Library path: `RF_Module/Tutorials/polarized_circular_ports`

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 **Preset Studies>Frequency Domain**.
6 Click **Done**.

GLOBAL DEFINITIONS

Add a parameter for the operating frequency.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
`frq`	`c_const/0.03[m]`	9.993E9 1/s	`Operating frequency`

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 In the **Frequencies** text field, type `range(0.9*frq,(1.5*frq-(0.9*frq))/10,1.5*frq)`.

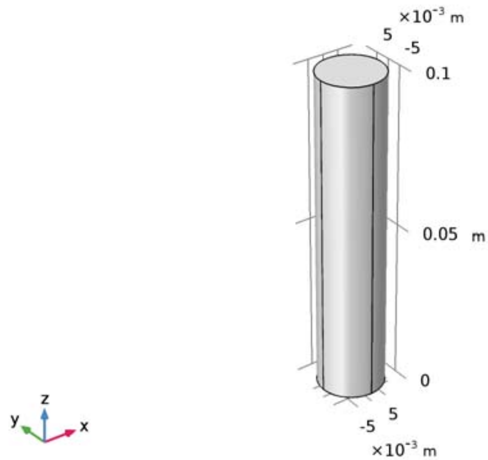
GEOMETRY 1

The geometry is essentially a cylinder.

Cylinder 1 (cyl1)

1 On 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 `0.01`.
4 In the **Height** text field, type `0.1`.

5 Click **Build Selected**.



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

You need to add a reference direction for the port polarization. Add a couple of lines on the cylinder end to generate extra vertices. This is done in a work plane.

Work Plane 1 (wp1)

1 On 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 `0.1`.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** click **Plane Geometry**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **yw** to `0.01`.

5 In row **2**, set **yw** to `-0.01`.

Rotate 1 (rot1)

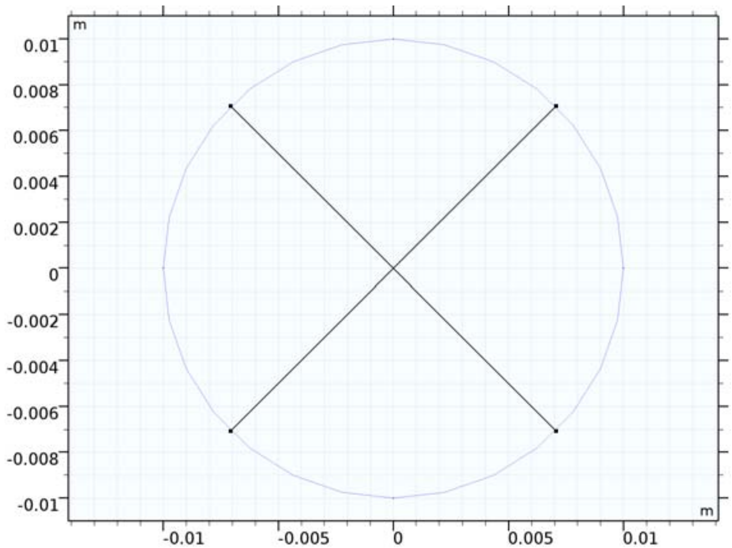
1 On the **Work Plane** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **b1** only.

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

4 In the **Rotation** text field, type `45 135`.

5 Click **Build Selected**.

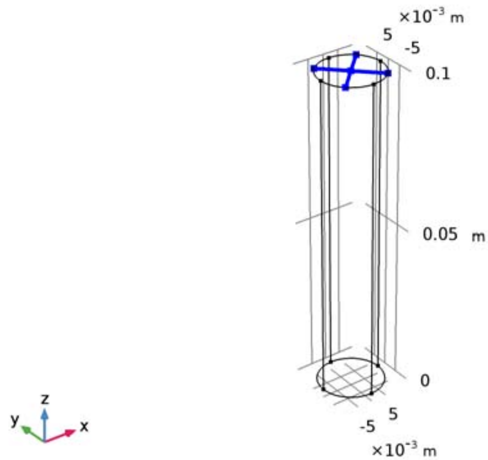


6 In the **Model Builder** window, click **Geometry 1**.

Ignore Edges 1 (ige1)

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 7, 8, 13, and 14 only.



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

4 Clear the **Ignore adjacent vertices** check box.

5 On the **Geometry** toolbar, click **Build All**.

MATERIALS

Next, add a material for the interior (air) of the waveguide.

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	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

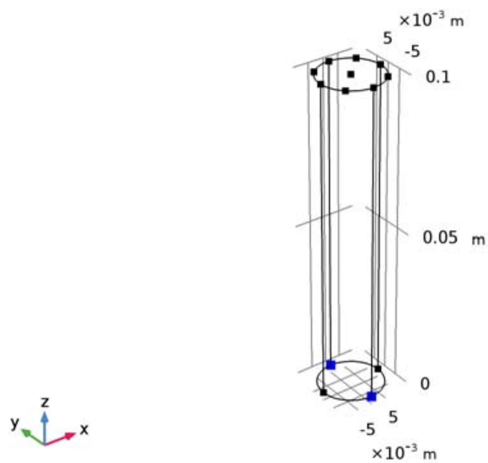
Set up one inport and three outports.

Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.
2 Select Boundary 3 only.
3 In the **Settings** window for **Port**, locate the **Port Properties** section.
4 From the **Type of port** list, choose **Circular**.

Circular Port Reference Axis 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)> Port 1** and choose **Circular Port Reference Axis**.
2 In the **Settings** window for **Circular Port Reference Axis**, locate the **Point Selection** section.
3 Click **Clear Selection**.
4 Select Points 5 and 8 only.



Port 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.
2 Select Boundary 3 only.
3 In the **Settings** window for **Port**, locate the **Port Properties** section.
4 From the **Type of port** list, choose **Circular**.

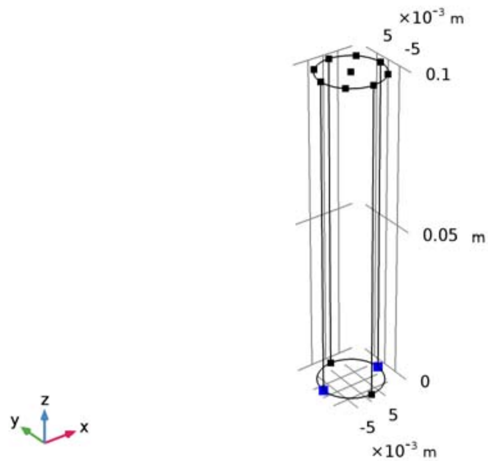
Circular Port Reference Axis 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>** **Port 2** and choose **Circular Port Reference Axis**.

2 In the **Settings** window for **Circular Port Reference Axis**, locate the **Point Selection** section.

3 Click **Clear Selection**.

4 Select Points 1 and 12 only.



Port 3

1 In the **Model Builder** window, right-click **Electromagnetic Waves,** **Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 4 only.

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

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

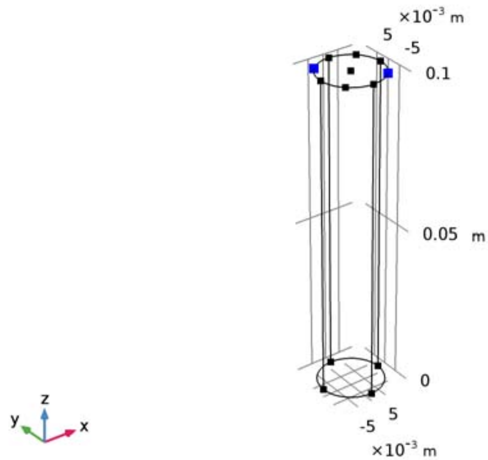
Circular Port Reference Axis 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>** **Port 3** and choose **Circular Port Reference Axis**.

2 In the **Settings** window for **Circular Port Reference Axis**, locate the **Point Selection** section.

3 Click **Clear Selection**.

4 Select Points 4 and 10 only.



Port 4

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 4 only.

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

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

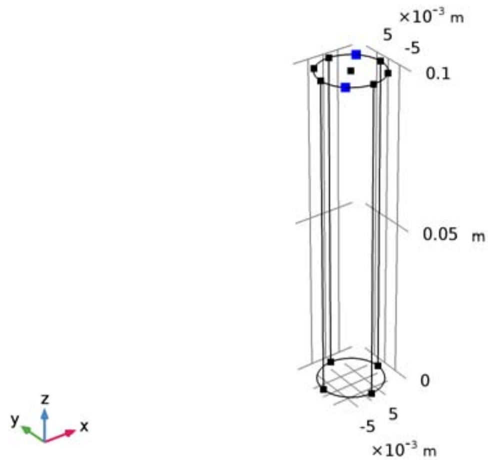
Circular Port Reference Axis 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)> Port 4** and choose **Circular Port Reference Axis**.

2 In the **Settings** window for **Circular Port Reference Axis**, locate the **Point Selection** section.

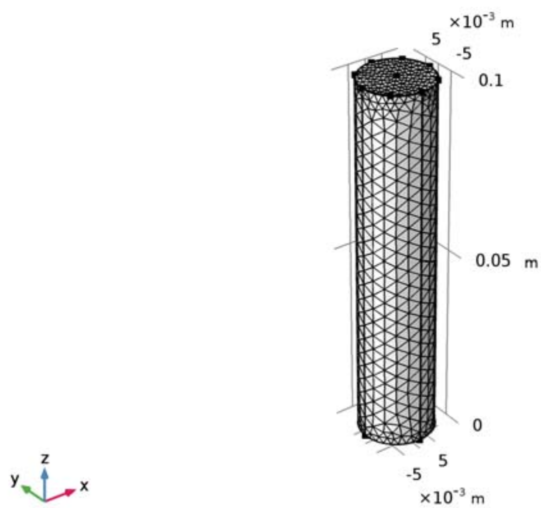
3 Click **Clear Selection**.

4 Select Points 3 and 11 only.



MESH 1

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



STUDY 1

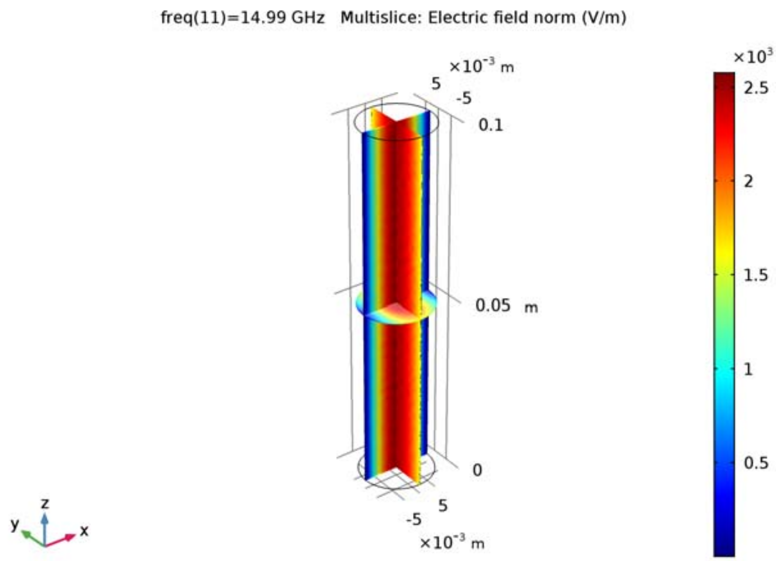
Step 1: Frequency Domain

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

RESULTS

Electric Field (emw)

Inspect the electric field norm.

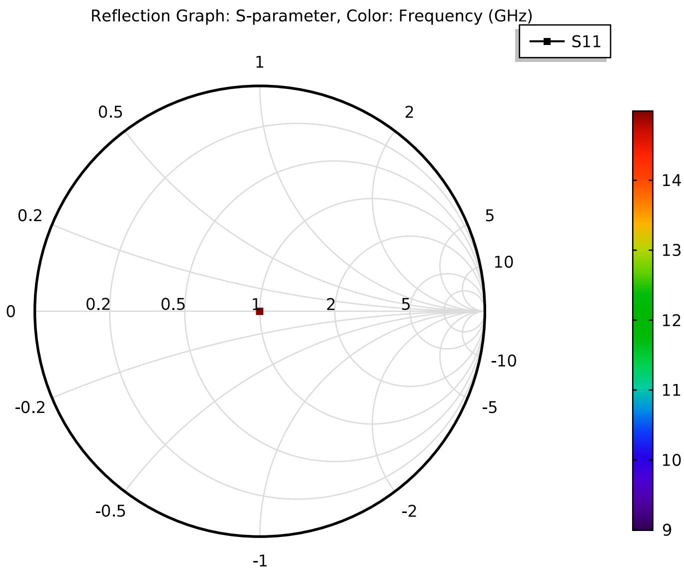


Next, inspect the S-parameters representing transmission.

S-Parameter (emw)

As expected, the transmitted energy is evenly divided between the outport modes (Figure 3).

Smith Plot (emw)



3D Plot Group 4

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

2 In the **Settings** window for **3D Plot Group**, type `Port 1` in the **Label** text field.

Surface 1

1 Right-click **Port 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>emw.normtEmode_1 - Port tangential electric mode field norm**.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Port 1** and choose **Arrow Surface**.

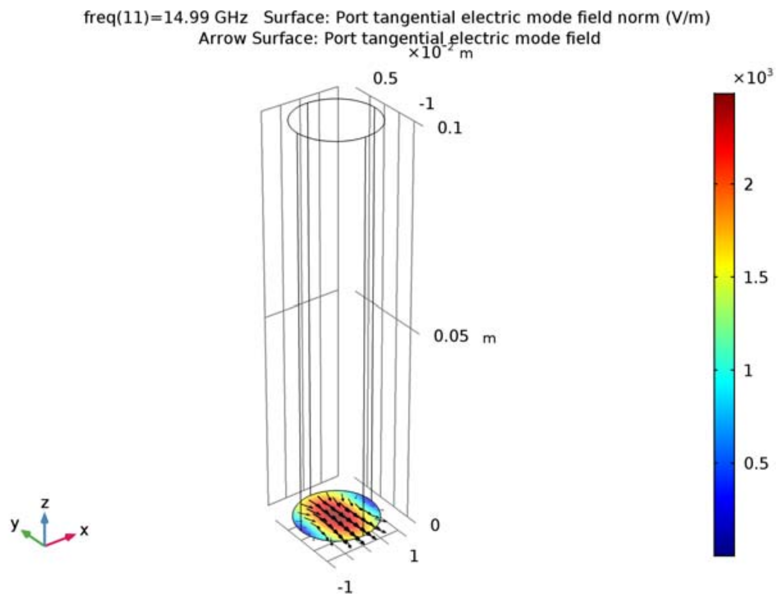
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Ports>emw.tEmodex_1,...,emw.tEmodez_1 - Port tangential electric mode field**.

3 Locate the **Coloring and Style** section. In the **Number of arrows** text field, type `1000`.

4 From the **Color** list, choose **Black**.

5 On the **Port 1** toolbar, click **Plot**.

6 Click **Plot**.



Port 1.1

1. Right-click **Port 1** and choose **Duplicate**.
2. In the **Settings** window for **3D Plot Group**, type `Port 2` in the **Label** text field.

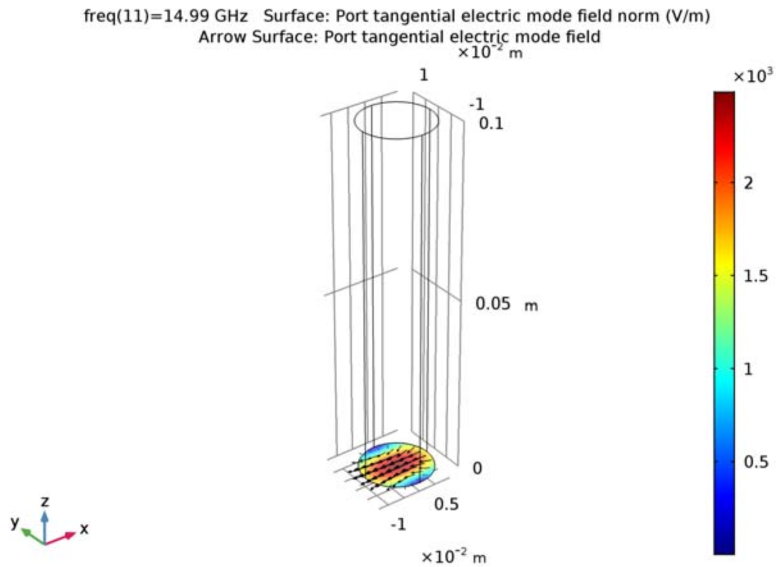
Surface 1

1. In the **Model Builder** window, expand the **Results>Port 2** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `emw.normtEmode_2`.

Arrow Surface 1

1. In the **Model Builder** window, under **Results>Port 2** click **Arrow Surface 1**.
2. In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
3. In the **X component** text field, type `emw.tEmodex_2`.
4. In the **Y component** text field, type `emw.tEmodey_2`.
5. In the **Z component** text field, type `emw.tEmodez_2`.

6 On the **Port 2** toolbar, click **Plot**.



Port 2.1

1 In the **Model Builder** window, under **Results** right-click **Port 2** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Port 3` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Port 3** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `emw.normtEmode_3`.

Arrow Surface 1

1 In the **Model Builder** window, under **Results>Port 3** click **Arrow Surface 1**.

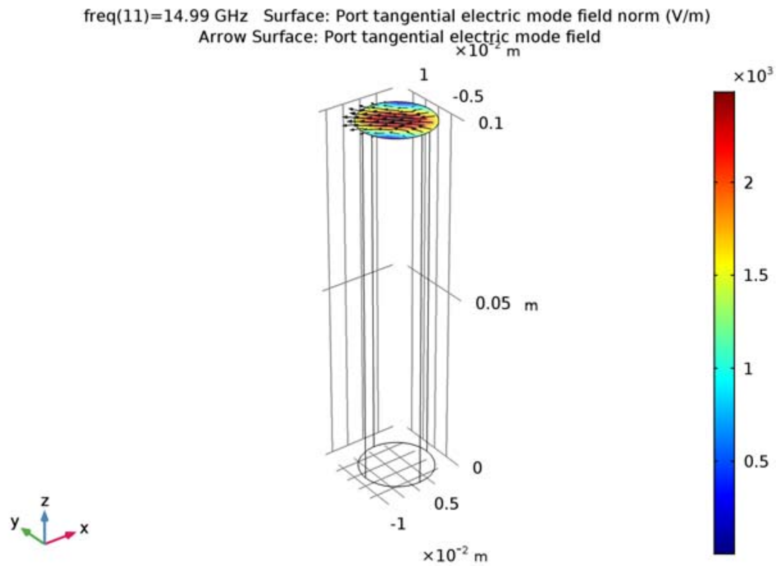
2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **X component** text field, type `emw.tEmodex_3`.

4 In the **Y component** text field, type `emw.tEmodey_3`.

5 In the **Z component** text field, type `emw.tEmodez_3`.

6 On the **Port 3** toolbar, click **Plot**.



Port 3.1

1 In the **Model Builder** window, under **Results** right-click **Port 3** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Port 4` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Port 4** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `emw.normtEmode_4`.

Arrow Surface 1

1 In the **Model Builder** window, under **Results>Port 4** click **Arrow Surface 1**.

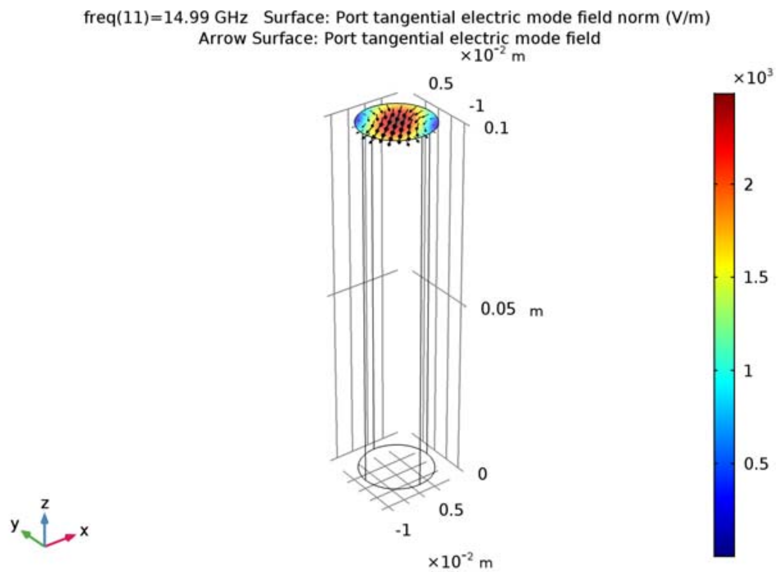
2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **X component** text field, type `emw.tEmodex_4`.

4 In the **Y component** text field, type `emw.tEmodey_4`.

5 In the **Z component** text field, type `emw.tEmodez_4`.

6 On the **Port 4** toolbar, click **Plot**.



Derived Values

Next, display numerical values for the transmission at the highest frequency.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

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

3 From the **Parameter selection (freq)** list, choose **Last**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S31dB - S31**.

5 Click **Evaluate**.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

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

3 From the **Parameter selection (freq)** list, choose **Last**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S41dB - S41**.

5 Click **Evaluate**.

As expected, the result is about -3 dB for both modes.

Radar Cross Section

Introduction

This tutorial model demonstrates the use of a background field in an electromagnetic scattering problem. Although this example is a boat hit by a radar, this same technique can be used in any situation where an isolated object meets electromagnetic waves from a distant source. For example, several orders of magnitude smaller, an equally common application is plasmon resonant nanoparticles. Besides setting up the background field and sweeping it over a range of angles of incidence, this example also shows you how to compute the far-field and the radar cross section (RCS).

Model Definition

This example computes the interaction between a boat and the incident field from a radar transmitter. The transmitter is considered to be distant enough that this field can be treated as a plane wave. This makes it possible to exclude the transmitter from the model geometry and look only at the boat and its immediate surroundings.

Although the modeling procedure is similar in 3D, this example is in 2D to quickly set up and solve. In order to focus on the concepts, the geometry is intentionally kept very simple (see Figure 1).

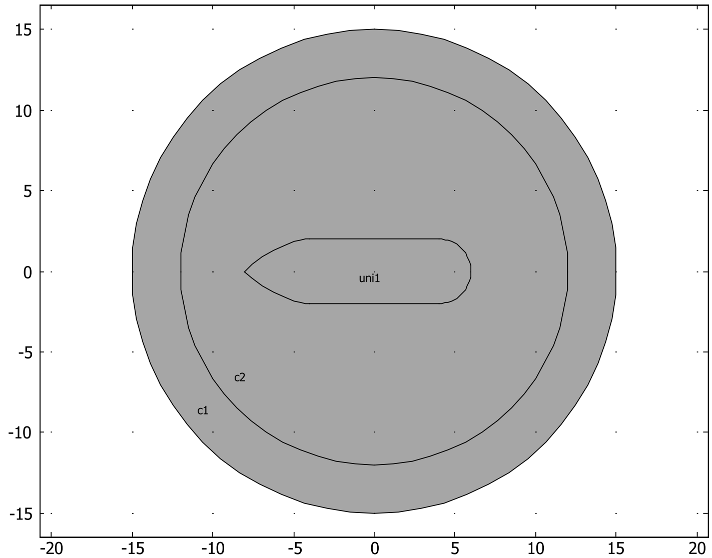


Figure 1: The model geometry. The boat has a length of 14 m and is surrounded by air - the cut plane lies above the water surface.

The inner circle in the geometry represents air surrounding the boat. The outer circle is a Perfectly Matched Layer (PML) which minimizes unphysical reflections of the scattered wave as it leaves the model domain. The radius of the inner circle is an important model consideration. To get good results, the inner circle needs to completely surround the boat. It also must extend to a considerable fraction of the wavelength, as well as the characteristic length of the evanescent wave outside it. However, in practice, and in order to minimize time and memory usage, do not make the circle too large. For the purpose of this example, the inner circle is bigger than necessary to provide a good view of the near field. The radius of the outer circle does not matter as long as it allows for the meshing needs of the PML, which is 5–6 mesh elements across.

The background electromagnetic field from the radar is described by its out-of-plane electric field component:

$$\mathbf{E}_b = \exp(jk_0(x\cos\phi + y\sin\phi))\mathbf{e}_z$$

In this equation:

- j is the imaginary unit;

- $k_0 = 2\pi f/c$ is the wave number in vacuum;
- $c = 3\cdot 10^8$ m/s is the speed of light;
- $f = 100$ MHz is the frequency;
- ϕ is the angle of incidence; and
- $\phi = 0$ corresponds to a wave incident from the positive x direction (and hence propagating in the negative x direction, from the right to the left in the model).

The time-harmonic wave equation is then solved for the relative field, $\mathbf{E}_{\text{rel}} = \mathbf{E} - \mathbf{E}_b$, where $\mathbf{E}$ is the total, measurable field.

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}_{\text{rel}}) - \left(\varepsilon_r - j\frac{\sigma}{\omega\varepsilon_0}\right) k_0^2 \mathbf{E}_{\text{rel}} = 0$$

The relative field is the difference between the measured field caused by the presence of the boat and the background field. It is utilized to describe how detectable the boat is with the radar - its RCS. The RCS of a 3D scatterer is defined as

$$\sigma_{3\text{D}} = \lim_{r \to \infty} 4\pi r^2 \frac{|\mathbf{E}_{\text{rel}}|^2}{|\mathbf{E}_b|^2}$$

For the 2D scatterer in this example, the RCS per unit length is used to address its monostatic scattering characteristics at the angle where the incident wave comes from, which is given by

$$\sigma_{2\text{D}} = \lim_{r \to \infty} 2\pi r \frac{|\mathbf{E}_{\text{rel}}|^2}{|\mathbf{E}_b|^2}$$

where the relative field as a function of radius is calculated with the help of COMSOL's built-in far-field computation, $\mathbf{E}_{\text{far}}$. The RCS of a 3D model which has a constant geometrical cross-section of a 2D model can also be estimated from the RCS per unit length by

$$\sigma_{3\text{D}} = \sigma_{2\text{D}} \frac{2l^2}{\lambda}$$

where l is the length of a scatterer and λ is the wavelength.

Results and Discussion

Using a parametric solver, the results are for the full range of angles of incidence, from 0 to 359 degrees in 1 degree steps. Figure 2 shows the norm of the total field caused by a background plane wave incident from a 30-degree angle. The reflections on the boat create a standing wave pattern. The boat casts a shadow on the lower left side.

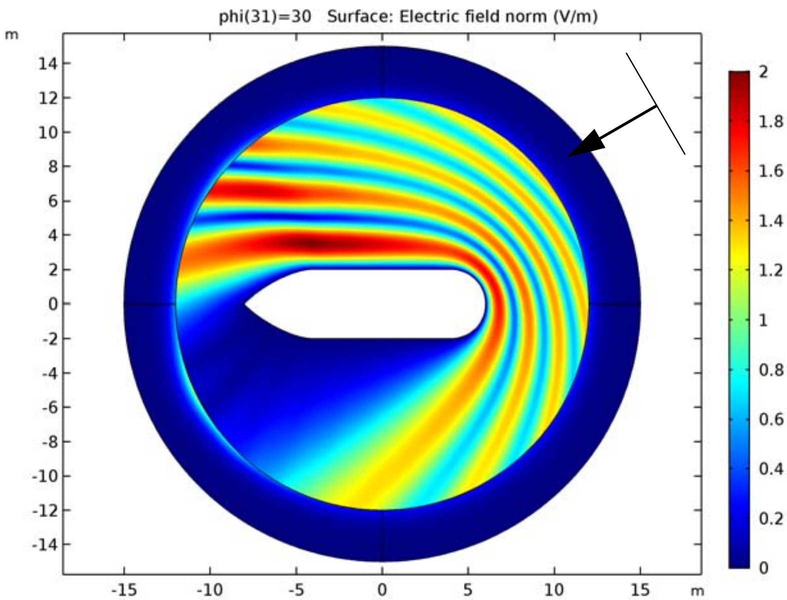


Figure 2: The total field norm for a 30 degree angle of incidence. The arrow represents the propagation direction of the incident background field.

You can also visualize the relative field sent out from the boat. Figure 3 shows both its instantaneous value at a zero phase and its norm. Note that the lack of standing waves in the latter indicates that the PML does its job of absorbing the outgoing field without any reflections.

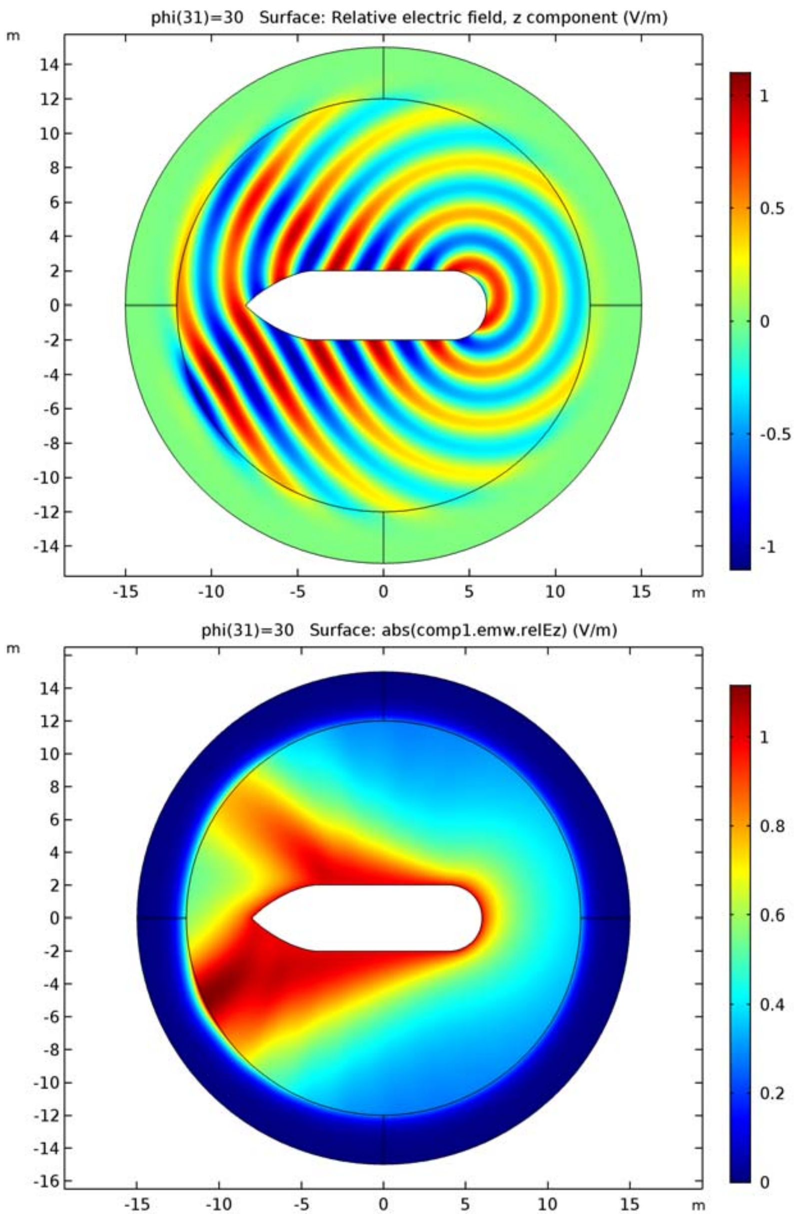


Figure 3: The instantaneous value (top) and magnitude (bottom) of the relative electric field sent out by the boat for a 30-degree angle of incidence.

As shown in these near field plots, you can guess that a distant observer would see peaks in the relative field centered around 150 and 210 degrees. This is confirmed by the far-field plot in Figure 4. The far-field computation uses the Stratton-Chu formula (see Radar Cross Section in the *RF Module User's Guide*) with the relative electric field on the boundaries of the boat as the input. Note that the relative field is the only component for which it makes sense to evaluate the far field. While the relative field falls off with the distance from the boat, the incident field amplitude remains constant. Hence the total field is non-trivial only at finite distances from the boat.

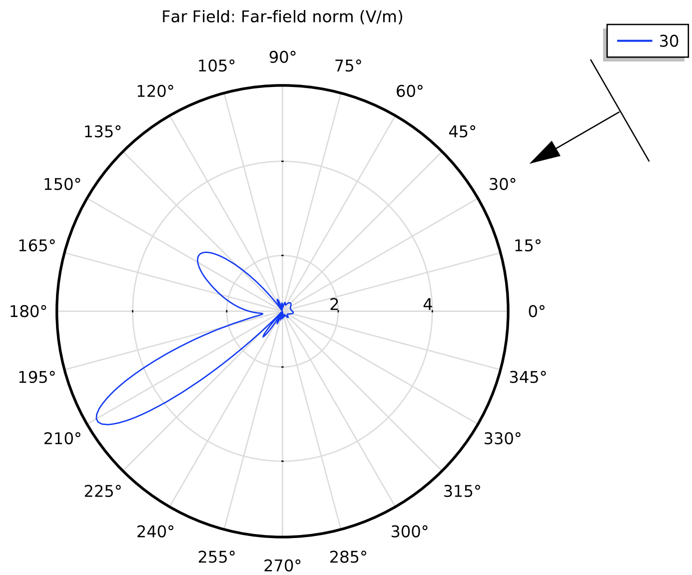


Figure 4: Far-field radiation plot for a 30-degree angle of incidence. The distance to the center represents the far field in dB.

Figure 5 shows the RCS per unit length. Compared to the other plots, which show various aspects of the fields for a specific angle of incidence, this output is generally only possible by solving for the complete range of angles, from 0 to 360 degrees. In this example, one could have avoided solving for half of this range by noting that because of the geometry symmetry, the results in the upper and lower half-plane are identical.

Except for the symmetry, a main feature of the RCS per unit length plot is the prominent peak at 90 degrees, due to the flat side of the boat. If the radar is in this direction, much of the field that hits the boat reflects back towards it. The peak around 135 degrees is explained similarly, but the side of the bow replaces the side of the boat. This peak is lower and wider because the boundary is shorter and slightly bent. There is a dip at 180 degrees because most of the field that hits the bow from a straight angle reflects to the sides.

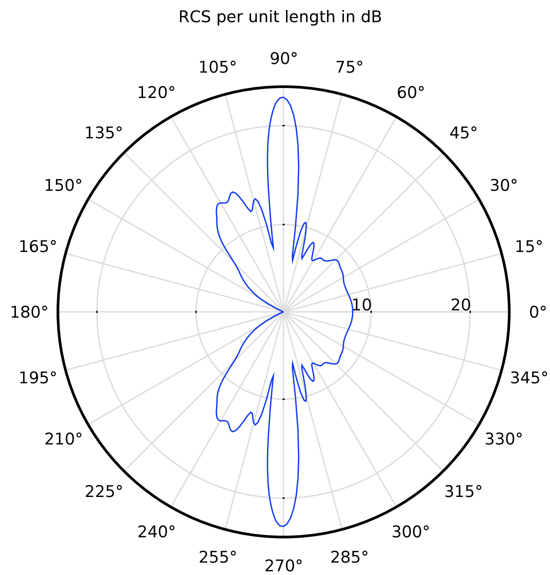


Figure 5: Polar plot of the RCS per unit length as a function of the angle of incidence.

Application Library path: `RF_Module/Scattering_and_RCS/radar_cross_section`

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 **2D**.

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 **Preset 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 In the **Frequencies** text field, type `100[MHz]`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
phi	0[deg]	0 rad	Angle of incidence, degrees

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `15`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	3

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `8`.

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

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

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type 2.

4 Locate the **Position** section. In the **x** text field, type 4.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Quadratic**.

4 Find the **Control points** subsection. In row **1**, set **x** to -4 and **y** to 2.

5 In row **2**, set **x** to -6 and **y** to 2.>

6 In row **3**, set **x** to -8.

7 Find the **Added segments** subsection. Click **Add Quadratic**.

8 Find the **Control points** subsection. In row **2**, set **x** to -6 and **y** to -2.

9 In row **3**, set **x** to -4 and **y** to -2.

10 Find the **Added segments** subsection. Click **Add Linear**.

11 Find the **Control points** subsection. Click **Close Curve**.

Union 1 (uni1)

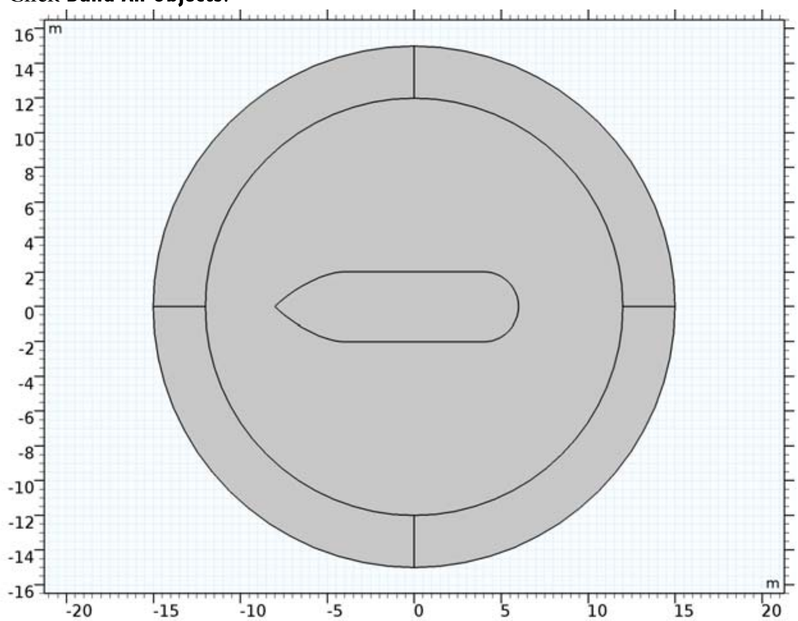
1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **b1**, **r1**, and **c2** only.

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build All Objects**.



DEFINITIONS

General Extrusion 1 (genext1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.
2. In the **Settings** window for **General Extrusion**, type `anglemap` in the **Operator name** text field.
3. Select Domain 3 only.
4. Locate the **Destination Map** section. In the **x-expression** text field, type `x*cos(phi[deg])`.
5. In the **y-expression** text field, type `x*sin(phi[deg])`.

   The operator that you just defined, anglemap, will return the value of its argument as evaluated in the point (x*cos(phi[deg]),x*sin(phi[deg])). This point has the same angle from the origin as that of the incident field.

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Boat Boundaries` in the **Label** text field.
3. Select Domain 4 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

You will select the boundaries of the boat several times throughout this tutorial. The Named Selection you just defined makes this more convenient.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1, 2, 5, and 6 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

The PML will make sure that the scattered field from the boat is almost completely absorbed before what remains of it reflects on the exterior boundaries of the model. Note that the background field is not affected by any of this - it is by definition what you are setting it to be.

ADD MATERIAL

1 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Aluminum**.

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

MATERIALS

Aluminum (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat2)**.

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

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

4 From the **Selection** list, choose **Boat Boundaries**.

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 Select Domains 1–3, 5, and 6 only.

Because you will be using a boundary condition to represent the boat, the field inside it will be identically zero and is not necessary to solve for. Removing the boat domain this way will save time and memory.

3 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Settings** section.

4 From the **Formulation** list, choose **Scattered field**.

The option you just selected lets you enter a background field. The expression you will use, represents a plane wave coming in from an angle phi. In this expression, the wave propagation constant in vacuum, emw.k0, is automatically provided by the physics interface. Once you have solved the model, you can plot the background field to verify that you have set it up correctly.

5 Specify the $\mathbf{E}_b$ vector as

0	x
0	y
1[V/m]*exp(j*emw.k0*(x*cos(phi[deg])+y*sin(phi[deg])))	z

Impedance Boundary Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Impedance Boundary Condition**.

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

3 From the **Selection** list, choose **Boat Boundaries**.

The use of an impedance boundary condition assumes that the skin depth in the material is much less than the material thickness. At 200 MHz, the skin depth in Aluminum is of the order of microns, so it is safe to say this is the case. You could even have used a perfect electric conductor condition instead, with largely unchanged results. In the case of scattering on a nanoparticle, the skin depth is often of the same order of magnitude as the particle itself. In such situations, you should not use the impedance boundary condition but rather keep the material domain active.

4 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

MESH 1

In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

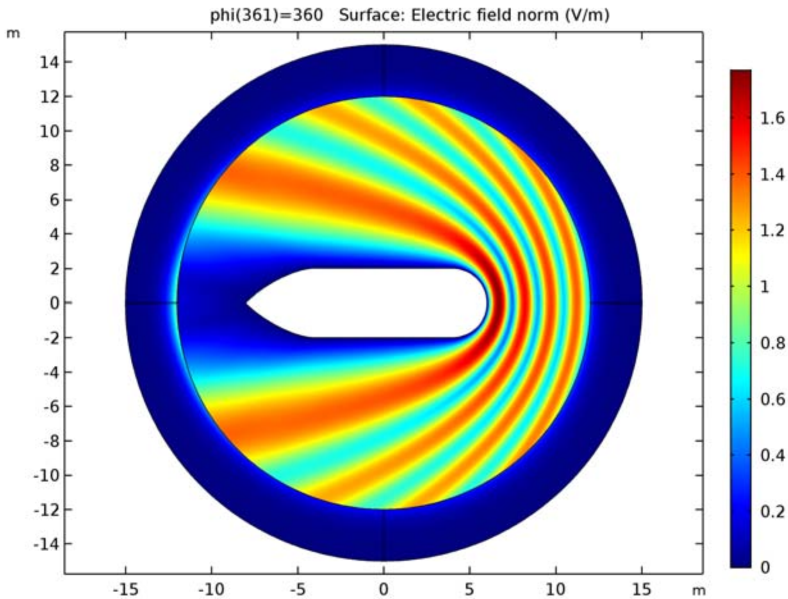
Parameter name	**Parameter value list**	**Parameter unit**
phi	`range(0,1,360)`	

You are solving this model for incidence angles from 0 to 360 degrees, in steps of 1 degree. Smaller steps would give you more accurate RCS plots, but the total solution time as well as the size of the model file increases linearly with the number of angles.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw)



The default plot shows the norm of the total electric field for an angle of incidence equal to 360 degrees. The total electric field is the actual, measurable, physical field. The plot is dominated by a standing wave pattern caused by the reflections mainly on the stern and the sides of the boat. As you might expect with the wave coming in almost from the right, a wake is forming to the left of the boat, beyond the bow. To reproduce Figure 2, see what the field looks like with a 30 degree angle of incidence.

1. In the **Model Builder** window, click **Electric Field (emw)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Parameter value (phi)** list, choose **30**.
4. On the **Electric Field (emw)** toolbar, click **Plot**.

   At 30 degrees, the wake widens. It is also possible to discern an increased field above and to the left of the boat, due to the reflections on its upper side. Try also plotting the instantaneous value of the total and the relative field.

Surface

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Electric>Electric field>emw.Ez - Electric field, z component**.

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

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Electric>Relative electric field>emw.relEz - Relative electric field, z component**.

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

Because the relative field is the difference between the observed and the background field, its magnitude will increase both in the wake and where the total field is enhanced by reflections. This trend is even clearer if you plot the absolute value of the relative field.

6 Locate the **Expression** section. In the **Expression** text field, type `abs(comp1.emw.relEz)`.

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

The plot should now resemble the bottom plot in Figure 3. You can get a quantitative measure of how the reflected field is radiating out in different directions from a plot of the far field as a function of the angle. A common way to do this is as a polar plot. You already have a default polar plot that you can make slight changes to.

2D Far Field (emw)

1 In the **Model Builder** window, under **Results** click **2D Far Field (emw)**.

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

3 From the **Parameter selection (phi)** list, choose **From list**.

4 In the **Parameter values (phi)** list, select **30**.

Far Field 1

1 In the **Model Builder** window, expand the **2D Far Field (emw)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `360`.

4 On the **2D Far Field (emw)** toolbar, click **Plot**.

To conclude, create a similar plot of the monostatic RCS per unit length. While the far-field plot is for one specific angle of incidence at a time, this plot will visualize the back scattering as a function of the angle of incidence.

Point Graph 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **Polar Plot Group**.
2. In the **Model Builder** window, right-click **Polar Plot Group 3** and choose **Point Graph**.
3. Select Point 13 only.
4. In the **Settings** window for **Point Graph**, locate the **r-Axis Data** section.
5. In the **Expression** text field, type `10*log10(anglemap(emw.bRCS2D))`.
6. Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
7. In the **Title** text area, type `RCS per unit length in dB`.
8. Locate the θ **Angle Data** section. From the **Parameter** list, choose **Expression**.
9. In the **Expression** text field, type `phi[deg]`.
10. On the **Polar Plot Group 3** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.3

Computing the Radar Cross Section of a Perfectly Conducting Sphere

General Description

This classic benchmark problem in computational electromagnetics is about computing the monostatic *radar cross section* (RCS) of a perfectly conducting sphere in free space, illuminated by a linearly polarized plane wave. The RCS is computed for sphere radius to free space wavelength ratios ranging from 0.1 to 0.8 and is compared to an exact analytical solution. This region represents the lower half of a transition zone between a long wavelength asymptotic solution, “Rayleigh scattering,” and a short wavelength asymptotic solution, “Geometrical Optics.” The transition zone is known as the “Mie region” after the originator of the exact solution. A mesh convergence study is performed for the first scattering resonance at a sphere radius to free space wavelength ratio of approximately 0.16364.

Model Setup

GEOMETRY

Due to symmetry, it is sufficient to model only one quarter of the sphere. Figure 1 shows the geometry and boundary conditions.

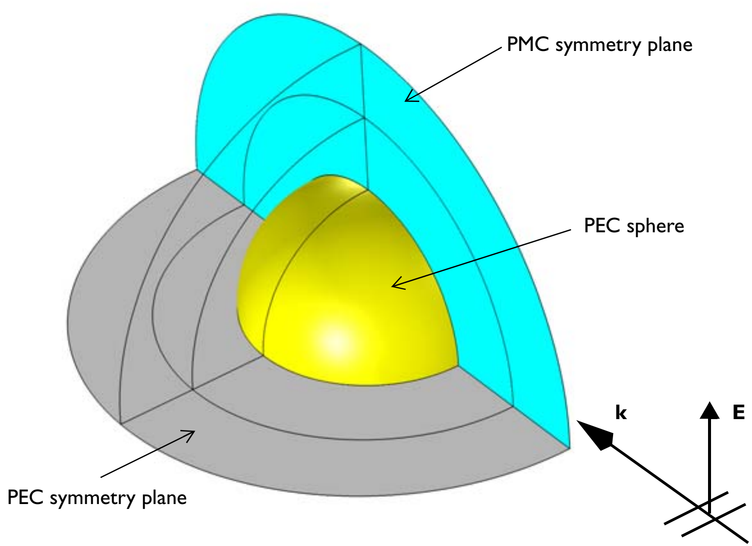


Figure 1: The computational domain for computing the RCS of a PEC sphere in free space. Due to symmetry, it is sufficient to model one quarter of the sphere.

The geometry consists of two concentric spherical shells. The innermost shell, adjacent to the sphere, represents the free space domain, and the second shell represents a perfectly matched layer (PML) region that is used to provide an approximately reflection free termination of the, in reality unbounded, free space domain.

EQUATION

The model is set up and solved using a frequency domain formulation for the scattered electric field. The incident plane wave travels in the positive x direction, with the electric field polarized along the z-axis. The governing frequency domain equation can be written in the form

$$\nabla \times (\mu_{\mathrm{r}}^{-1} \nabla \times (\mathbf{E}_i + \mathbf{E}_{sc})) - k_0^2 \varepsilon_{\mathrm{rc}} (\mathbf{E}_i + \mathbf{E}_{sc}) = \mathbf{0}$$

where the scattered electric field $\mathbf{E}_{\mathrm{sc}}$ is the dependent variable and the incident electric field $\mathbf{E}_i = (0,0,E_z)$, with

$$E_z = 1[V/m]e^{-jk_0 x}$$

The equation is discretized using second order edge elements (also known as vector elements, Nedelec elements, or curl-conforming elements). It is well known that in order to resolve the wave field, one should strive for 10 or more discretization points per wavelength. The combination of using second-order elements and 8 elements per wavelength fulfills this criterion with some margin. To respect the geometry, a mesh that is somewhat finer for the longest wavelengths is required on the surface of the scatterer. A maximum element size of half the radius is used on those boundaries. The PML region requires special meshing as described under the section Perfectly Matched Layer below.

BOUNDARY CONDITIONS

The sphere has perfect electric conductor (PEC) boundaries. The PEC boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{0}$$

sets the tangential component of the electric field to zero. It is used for the modeling of lossless metallic surfaces or as a symmetry type boundary condition. It imposes symmetry for magnetic fields and “magnetic currents” and antisymmetry for electric fields and electric currents.

PEC boundary conditions and perfect magnetic conductor (PMC) boundary conditions apply on the symmetry planes used to subdivide the sphere model.

The PMC boundary condition

$$\mathbf{n} \times \mathbf{H} = \mathbf{0}$$

sets the tangential component of the magnetic field and thus also the surface current density to zero. On external boundaries, this can be interpreted as a “high surface impedance” boundary condition or used as a symmetry type boundary condition. It imposes symmetry for electric fields and electric currents and antisymmetry for magnetic fields and “magnetic currents.”

PERFECTLY MATCHED LAYER

The PML region, the second concentric shell around the sphere, provides an approximately reflection free termination of the computational domain by applying a complex-valued coordinate stretching in the radial (outward) direction. For good accuracy, there should be at least five elements through the thickness of the PML. This condition is usually most efficiently met by using a swept mesh so that the effective element quality becomes insensitive to the scaling in the radial direction. The mesh used in this example is shown in Figure 2. It consists of a free tetrahedral mesh around the sphere and a swept mesh in the PML domain.

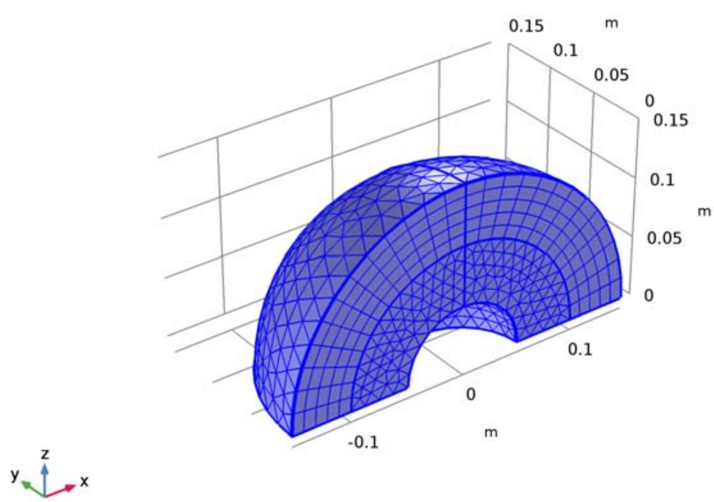


Figure 2: A free tetrahedral mesh is used in the free-space region around the sphere, and a swept mesh is used in the PML region.

The free space region around the sphere is defined to be the far-field domain. This specifies that a near-field to far-field calculation is done on the boundary of this domain, which takes the computed electric fields around the sphere and uses the Stratton-Chu equation to find the scattered electric field infinitely far away from the origin.

In 3D, this is:

$$\mathbf{E}_p = \frac{jk}{4\pi}\mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk\mathbf{r} \cdot \mathbf{r}_0) dS$$

For scattering problems, the far field in COMSOL is identical to what in physics is known as the "scattering amplitude."

The radiating or scattering object is located in the vicinity of the origin, while the far-field point p is taken at infinity but with a well-defined angular position (θ, φ).

In the above formulas,

- $\mathbf{E}$ and $\mathbf{H}$ are the fields on the "aperture"—the surface S enclosing the sphere.
- $\mathbf{r}_0$ is the unit vector pointing from the origin to the field point p. If the field points lie on a spherical surface S', $\mathbf{r}_0$ is the unit normal to S'.
- $\mathbf{n}$ is the unit normal to the surface S.
- η is the wave impedance:

$$\eta = \sqrt{\mu / \varepsilon}$$

- k is the wave number.
- λ is the wavelength.
- $\mathbf{r}$ is the radius vector (not a unit vector) of the surface S.
- $\mathbf{E}_p$ is the calculated far field in the direction from the origin towards point p.

The unit vector $\mathbf{r}_0$ can be interpreted as the direction defined by the angular position (θ, φ) and $\mathbf{E}_p$ is the far field in this direction.

Results and Discussion

Figure 3 compares the simulation result for the RCS with the analytic solution computed using the scattered component of the electric field from this example and equation 11-247 in Ref. 1. As the figure shows, there is good agreement between the analytic solution determined in this manner and the finite-element model.

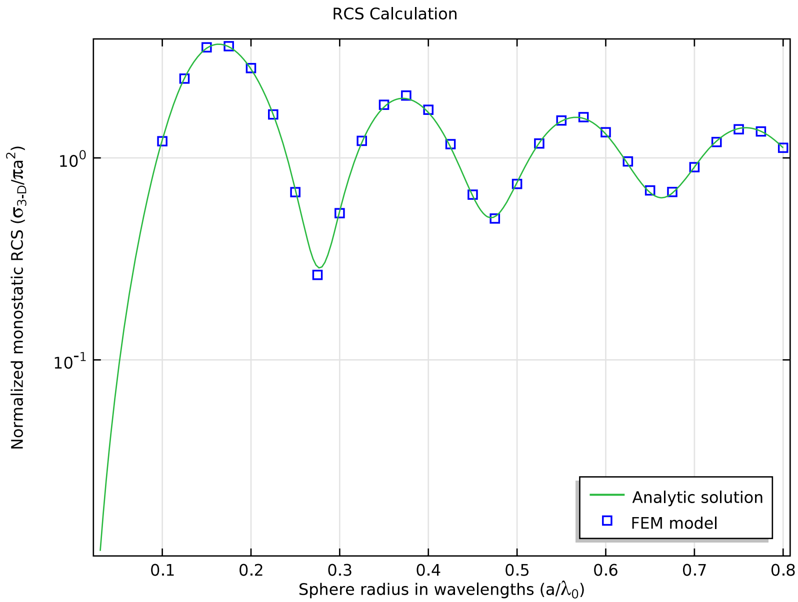


Figure 3: Comparison of the analytic solution and the COMSOL Multiphysics model of the RCS of a PEC sphere in free space.

Mesh Convergence

For the wavelength corresponding to the first maximum in the RCS plot in Figure 3, a mesh convergence study is performed to validate that the model converges toward a unique solution when refining the mesh isotropically. The model is solved in a parametric sweep over the number of mesh elements per wavelength. In the PML, the mesh density is not changed in the radial outward direction (that is, in the sweep direction for the swept mesh). The PML is resolved by 5 element layers in this direction which is sufficient to resolve the exponential damping in the radial direction. Thus the error contribution from the PML is not expected to decrease by adding more element layers. The main error contribution from the PML is due to the fact that it is not perfectly absorbing because of finite thickness and damping rather than mesh density. Thus, it is expected to give a contribution to the error in the computed RCS that does not decrease when refining the mesh.

Figure 4 shows the mesh convergence. The displayed error is the difference between the RCS from the finite element model and the exact solution from equation 11-247 in Ref. 1.

As mentioned, the PML is expected to yield an error contribution which cannot be eliminated by refining the mesh. As there is no sign of stagnation in the convergence plot, this error contribution must be smaller than 0.1%. The RCS plot in Figure 3 corresponds to 8 elements per wavelength, that is a relative error of about 3% at the wavelength of the maximum in the RCS versus wavelength curve.

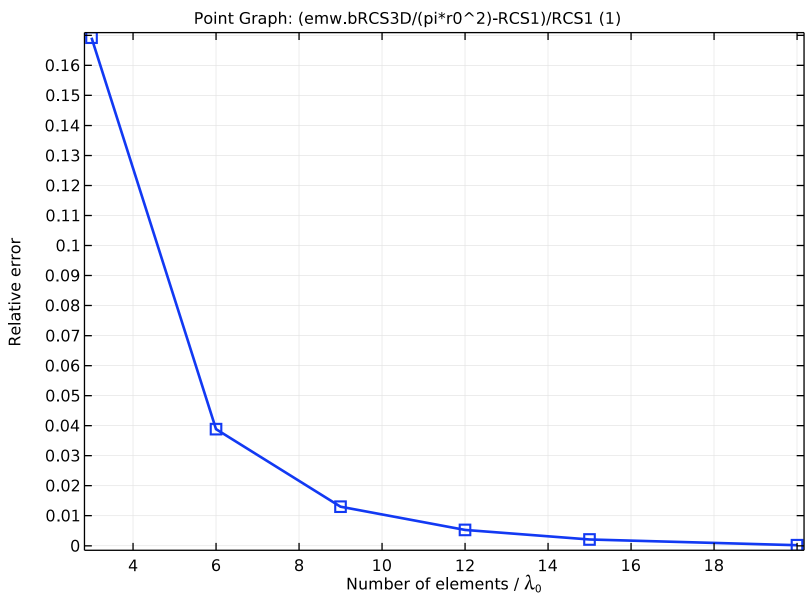


Figure 4: Mesh convergence for the difference in backscattering (monostatic) RCS between the COMSOL model and the exact solution.

Reference

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, John Wiley & Sons, 1989.

Application Library path: `RF_Module/Verification_Examples/rcs_sphere`

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 **Preset Studies>Frequency Domain**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
r_lda	0.5	0.5	Sphere radius in wavelengths
r0	5[cm]	0.05 m	Sphere radius
lda	r0/r_lda	0.1 m	Wavelength
k0	2*pi/lda	62.83 1/m	Wavenumber
f0	c_const/lda	2.998E9 1/s	Frequency
t_air	lda/2	0.05 m	Thickness of air around sphere
t_pml	lda/2	0.05 m	Thickness of PML
h_size	8	8	Number of elements per wavelength
E0	1[V/m]	1 V/m	Incident field magnitude

GEOMETRY 1

First, create a sphere with two layer definitions. The outermost layer represents the PMLs and the core represents the PEC sphere for RCS analysis. The median layer is the air domain.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.
2 In the **Settings** window for **Sphere**, locate the **Size** section.
3 In the **Radius** text field, type `r0+t_air+t_pml`.
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	t_pml
Layer 2	t_air

5 Click **Build All Objects**.

DEFINITIONS

Add a view with a different angle of perspective.

Camera

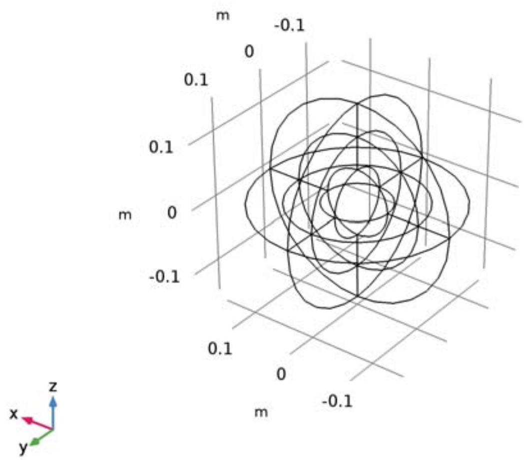
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

Change only the sign of y in the Position and Up Vector sections:

2 In the **Model Builder** window, expand the **View 2** node, then click **Camera**.
3 In the **Settings** window for **Camera**, locate the **Position** section.
4 In the **y** text field, type `1.871`.
5 Locate the **Up Vector** section. In the **y** text field, type `-0.412`.
6 Click **Update**.

Choose wireframe rendering to get a better view of the interior parts.

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



GEOMETRY 1

Due to the symmetry of the structure, it is sufficient to model only one quarter of the sphere. Delete the domains which are not part of the modeling domain.

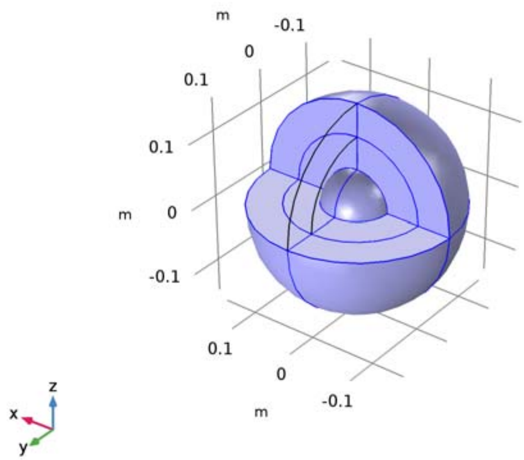
Delete Entities 1 (del1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **sph1**, select Domains 1–3, 5–7, and 9–15 only.



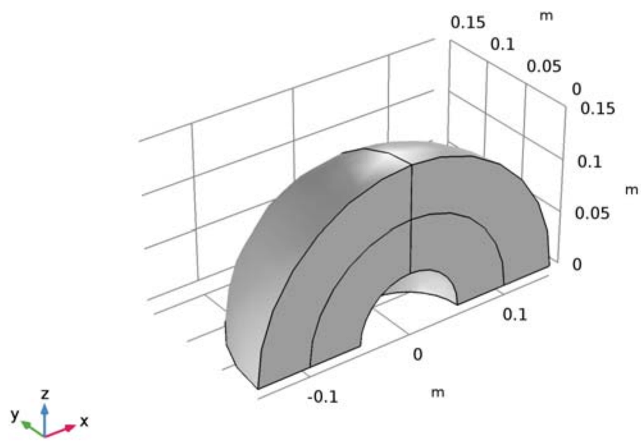
5 Click **Build All Objects**.

DEFINITIONS

View 1

After removing unnecessary domains, change the view to the first view definition which gives a better angle showing all layers.

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



This is the modeling domain for RCS analysis.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. You will solve the model for the scattered field, which requires background electric field (E-field) information. The background plane wave is traveling in the positive x direction, with the electric field polarized along the z-axis. The default boundary condition is perfect electric conductor, which applies to all exterior boundaries including the boundaries perpendicular to the background E-field polarization.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Settings** section.
3. From the **Formulation** list, choose **Scattered field**.
4. Specify the $\mathbf{E}_b$ vector as

0	x
0	y
E0*exp(-j*k0*x)	z

Choose wireframe rendering in the current view to get a better view of the interior parts.

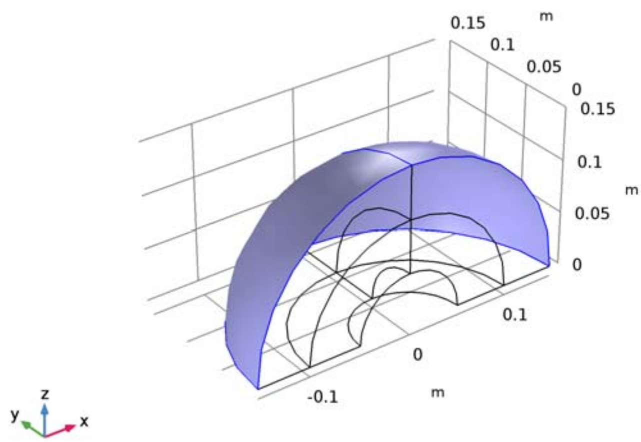
5 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

6 Locate the **Physics-Controlled Mesh** section. Clear the **Enable** check box.

Scattering Boundary Condition 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** and choose **Scattering Boundary Condition**.

2 Select Boundaries 3 and 14 only.



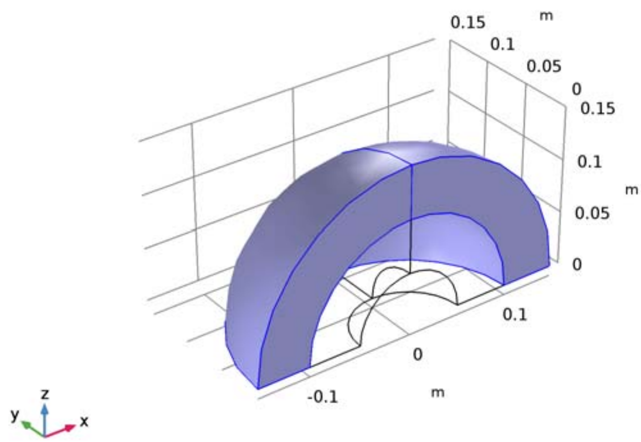
DEFINITIONS

The outermost domains from the center of the sphere are the PMLs.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 4 only.



3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

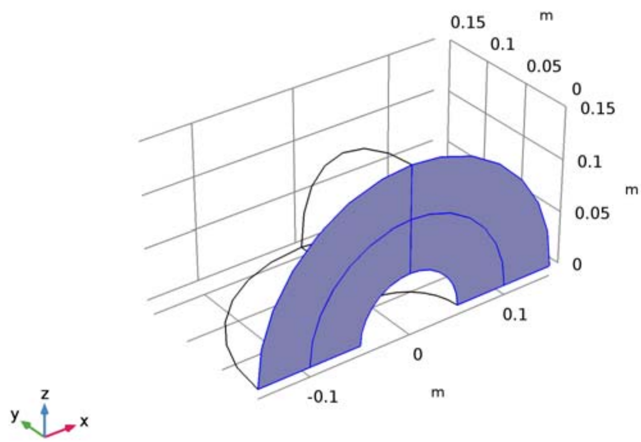
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Set PMC on the boundaries parallel to the background E-field polarization.

Perfect Magnetic Conductor 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 1, 4, 9, and 12 only.



Far-Field Calculation 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

2 In the **Model Builder** window, expand the **Far-Field Domain 1** node, then click **Far-Field Calculation 1**.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the y=0 plane** check box.

5 Select the **Symmetry in the z=0 plane** check box.

6 From the **Symmetry type** list, choose **Symmetry in H (PEC)**.

MATERIALS

Next, assign material properties. Use air for all domains.

ADD MATERIAL

1 On 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.

MATERIALS

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

MESH 1

Use a tetrahedral mesh for the air domains.

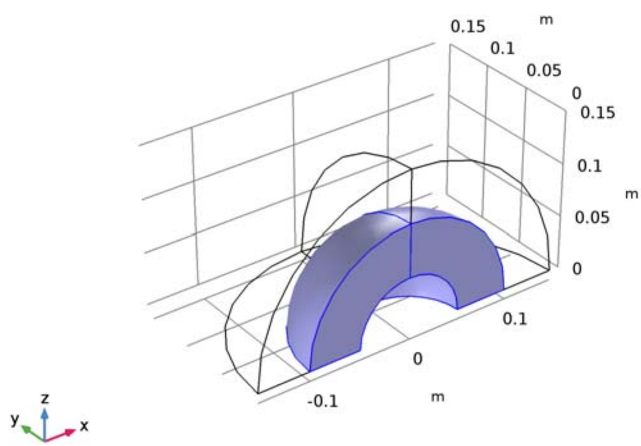
Free Tetrahedral 1

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

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

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

4 Select Domains 2 and 3 only.



The maximum mesh size is at most 0.2 wavelengths in free space. In this model, use 0.125 wavelengths.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lda/h_size`.

5 In the **Minimum element size** text field, type `lda/h_size`.

Use a swept mesh for the PML domains.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

3 In the **Settings** window for **Distribution**, click **Build All**.

Compare the mesh with that shown in Figure 2.

STUDY 1

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_lda	`range(0.1,0.025,0.8)`	

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 In the **Frequencies** text field, type `f0`.

4 In the **Model Builder** window, click **Study 1**.

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Clear the **Generate default plots** check box.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Follow the instructions below to reproduce the plot in Figure 3. First, show the computed RCS values using square markers.

1D Plot Group 1

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

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

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

5 Find the **Type and data** subsection. Clear the **Unit** check box.

6 Clear the **Description** check box.

7 Clear the **Type** check box.

8 Find the **User** subsection. In the **Prefix** text field, type `RCS Calculation`.

9 Locate the **Plot Settings** section. Select the **x-axis label** check box.

10 In the associated text field, type `Sphere radius in wavelengths (a/\lambda<sub>0</sub>)`.

11 Select the **y-axis label** check box.

12 In the associated text field, type `Normalized monostatic RCS (\sigma<sub>3-D</sub>/\pi a<sup>2</sup>)`.

13 Locate the **Axis** section. Select the **y-axis log scale** check box.

Point Graph 1

1 Right-click **1D Plot Group 1** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `emw.bRCS3D/(pi*r0^2)`.

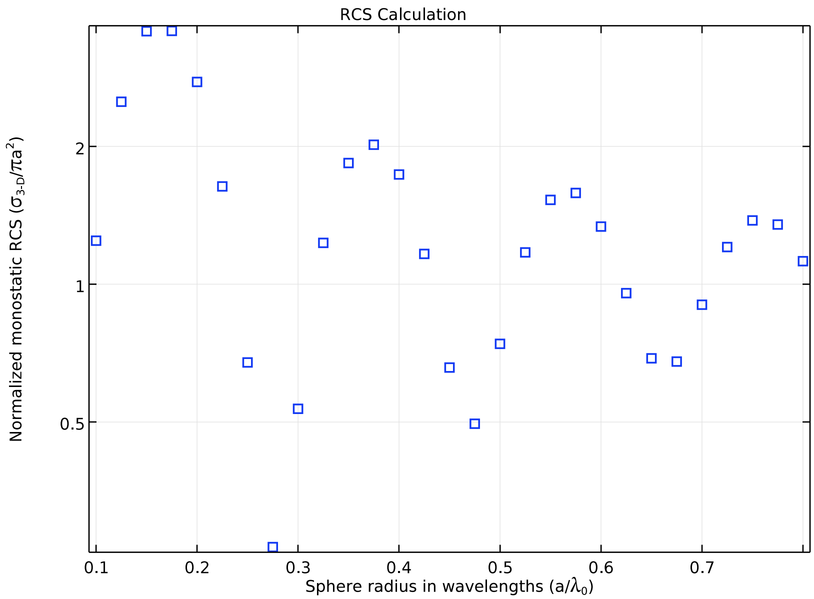
5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

9 On the **1D Plot Group 1** toolbar, click **Plot**.



The observed RCS graph pattern is oscillatory in the Mie region.

Next, proceed to perform the mesh convergence study at the first resonance in the Mie region.

Start by extending the parameter list with the resonant radius and the associated theoretical RCS value.

GLOBAL DEFINITIONS

Parameters

Add the two rows at the end and change the first line according to.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`r_lda`	`r1`	0.1636	`Sphere radius in wavelengths`
`r0`	`5[cm]`	0.05 m	`Sphere radius`

Name	Expression	Value	Description
lda	r0/r_lda	0.3056 m	Wavelength
k0	2*pi/lda	20.56 1/m	Wavenumber
f0	c_const/lda	9.811E8 1/s	Frequency
t_air	lda/2	0.1528 m	Thickness of air around sphere
t_pml	lda/2	0.1528 m	Thickness of PML
h_size	8	8	Number of elements per wavelength
E0	1[V/m]	1 V/m	Incident field magnitude
r1	0.16363636363636364	0.1636	Relative radius at 1st resonance
RCS1	3.6549540474068576	3.655	RCS at 1st resonance

MESH 1

Add a new mesh with some tweaks to make sure that the curvature of the sphere is always resolved.

This is to avoid inverted mesh elements.

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

MESH 2

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

COMPONENT 1 (COMP1)

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

MESH 2

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Mesh 2** node, then click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Minimum element size** text field, type `if(lda/h_size>r0/2,r0/2,lda/h_size)`.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 2** right-click **Free Tetrahedral 1** and choose **Size**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `7,10` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 Click the **Custom** button.

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

10 In the associated text field, type `if(lda/h_size>r0/2,r0/2,lda/h_size)`.

ROOT

Add a new frequency domain study for the mesh convergence analysis.

ADD STUDY

1 On 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> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

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

STUDY 2

A parametric sweep is needed to loop over the mesh sizes.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
h_size	3 6 9 12 15 20	

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `f0`.

4 In the **Model Builder** window, click **Study 2**.

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Clear the **Generate default plots** check box.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Continue to plot the relative error versus elements per wavelength.

1D Plot Group 2

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

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

3 From the **Data set** list, choose **Study 2/Parametric Solutions 2 (sol33)**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Number of elements / \lambda<sub>0</sub>`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Relative error`.

Point Graph 1

1 Right-click **1D Plot Group 2** and choose **Point Graph**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `(emw.bRCS3D/(pi*r0^2)-RCS1)/RCS1`.

8 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **h_size**.

9 From the **Parameter** list, choose **Expression**.

10 In the **Expression** text field, type `h_size`.

11 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

12 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

14 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

15 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

16 On the **1D Plot Group 2** toolbar, click **Plot**.

Compare the convergence plot with that shown in Figure 4.

RF Heating

Introduction

This is a model of an RF waveguide bend with a dielectric block inside. There are electromagnetic losses in the block as well as on the waveguide walls which cause the assembly to heat up over time. The material properties of the block are functions of temperature. The transient thermal behavior, as well as the steady-state solution, are computed.

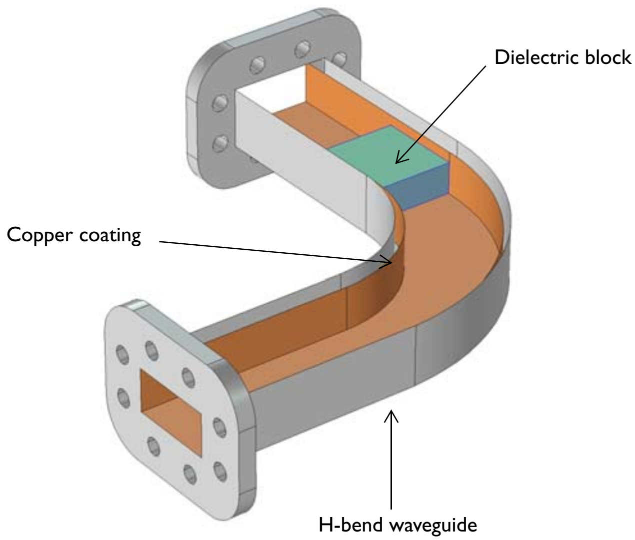


Figure 1: A waveguide bend with a dielectric block inside. Top boundaries of the waveguide are removed only for visualization.

Model Definition

The waveguide bend shown in Figure 1 is connected to a 100 W power source, operating at 10 GHz, via a rectangular waveguide operating in the TE_{10} mode. The other end of the bend is also connected to a rectangular waveguide operating in the TE_{10} mode. The objective of such a bend is primarily to change the direction of propagation of the energy. Here, however, a block of dielectric is introduced as an example of a lossy material interacting with an electromagnetic field.

The waveguide is made of aluminum. To reduce surface losses, the inside walls are coated with copper, a high-conductivity metal. The dielectric block is modeled as having electrical conductivity of $\sigma = 0$, relative permeability of $\mu_r = 1$, and a relative permittivity of $\varepsilon_r = 2.1$, with a loss tangent that is a function of temperature, $\delta = 0.001(1 + T/300\ \mathrm{K})$. The thermal conductivity of this block is also a function of temperature, $k = 0.3(1 + T/300\ \mathrm{K})$ W/m/K. Furthermore, the density is 2200 $\mathrm{kg/m^3}$ and the specific heat is 1050 J/kg/K. These are generic properties representative of a dielectric material.

At the operating frequency, the skin depth of the copper coating is much smaller than the dimensions of the waveguide, that is, the electromagnetic fields penetrate a negligible distance into the walls. This means that the electromagnetic losses can be localized entirely on the surface, and that there is no need to solve Maxwell's equations inside of the walls themselves. Thus, Maxwell's equations only need to be solved in the air domain inside of the waveguide, as well as inside of the block. The heat transfer equation is solved in the block as well as the waveguide walls.

The objective of the analysis is to observe how the assembly of the dielectric block and waveguide heat up over time, as well as to find the steady-state temperature. The waveguide is initially assumed to be at a constant temperature throughout. After the power source is turned on, the electromagnetic fields interact with the highly conductive interior boundaries of the waveguide, as well as the lossy dielectric block. The losses in the block and on the walls are sources of heat that raises the temperature. The block is assumed to be in perfect thermal contact with the walls of the waveguide, that is, any heat generated in the block is conducted away into the walls. The outside boundaries of the walls are assumed to be facing ambient air, which leads to free convective cooling off of these faces. This example uses an averaged heat transfer coefficient to represent this free convection to ambient air.

The model solves two governing equations: Maxwell's equations, which describe the electromagnetic fields, and the heat transfer equation, which describes the temperature. It is assumed that the operating frequency is much higher than any thermal transients, and thus it is possible to solve the problem either in a frequency-transient or a frequency-stationary sense.

A *frequency-transient* simulation solves Maxwell's equations in the frequency domain. This implicitly assumes that all material properties used to solve Maxwell's equations are constant over a single period of oscillation of the electromagnetic wave. The heat transfer equation is, on the other hand, is solved transiently. The electromagnetic fields are only recomputed when the material properties have changed significantly, as determined by a criterion involving the relative tolerance of the time-dependent solver. The objective of the analysis is to determine the change in temperature from given initial conditions and how long these changes take.

A *frequency-stationary* simulation solves Maxwell's equations in the frequency domain, but it solves the stationary heat transfer equation under the assumption that all initial transient variations have died out. Although no transient information is obtainable, this computation is significantly faster than a frequency-transient analysis and gives the steady-state temperature distribution.

Results and Discussion

Figure 2 plots the peak temperature within the dielectric block over time, showing that it takes several minutes for the block to reach thermal equilibrium.

Figure 3 plots the fields inside of the waveguide, as well as the temperature of the assembly, for the steady-state temperature solution after all thermal transients have died out. The dielectric block shows a significant temperature variation, which affects the thermal conductivity and loss tangent, plotted in Figure 4.

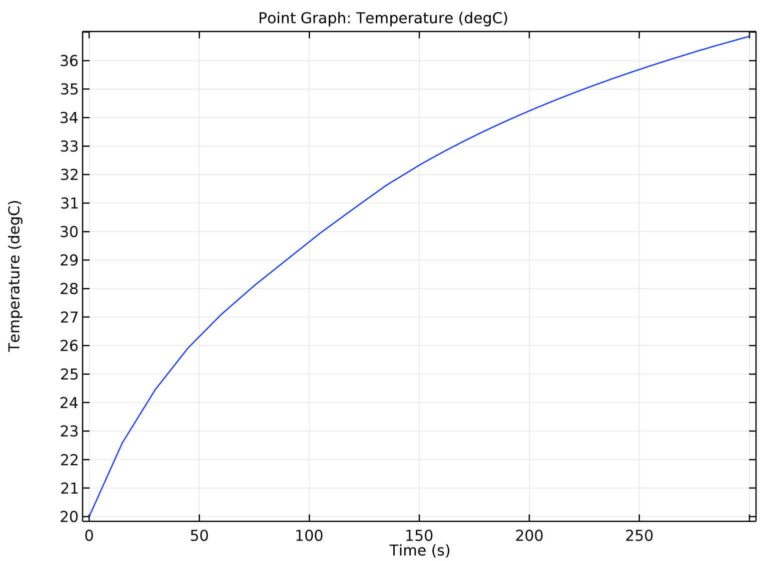


Figure 2: The maximum temperature, evaluated over the volume of the block, is plotted as a function of temperature.

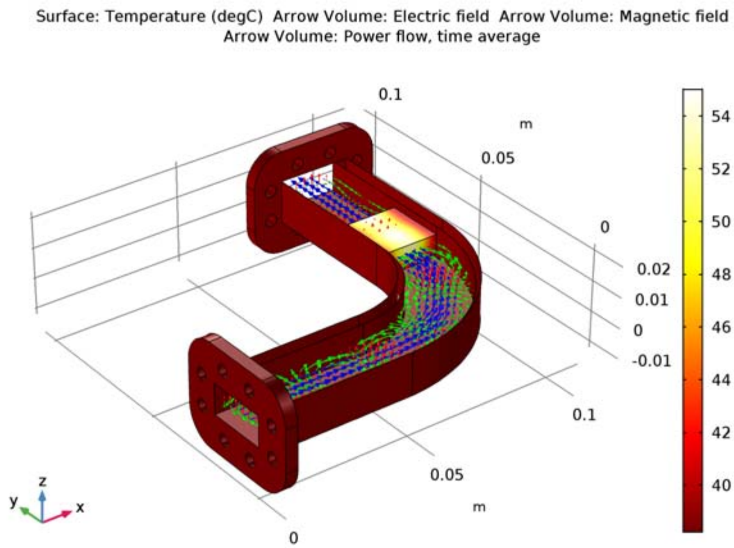


Figure 3: The electric fields (red arrows) magnetic fields (green arrows) and power flow (blue arrows) are shown inside of the waveguide. The steady-state temperature is plotted on the block and waveguide walls.

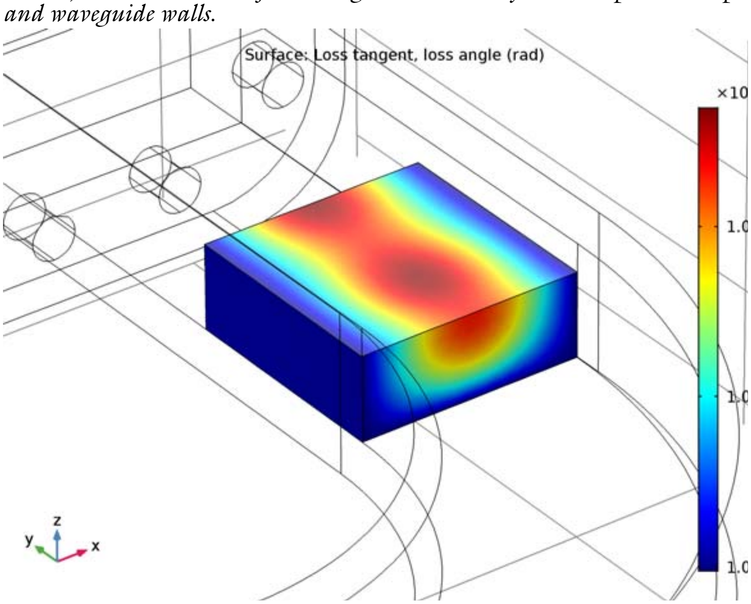


Figure 4: The loss tangent within the dielectric block for the steady-state solution shows that the variation in temperature affects the material properties.

Application Library path: `RF_Module/Microwave_Heating/rf_heating`

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 **Heat Transfer>Electromagnetic Heating> Microwave Heating**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Frequency-Transient**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
f0	10[GHz]	1E10 Hz	Current frequency
lda0	c_const/f0	0.02998 m	Wavelength, air
h_max	0.2*lda0	0.005996 m	Maximum mesh element size, air

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

GEOMETRY 1

First, import the geometry of the waveguide including a dielectric block inside the waveguide.

Import 1 (imp1)

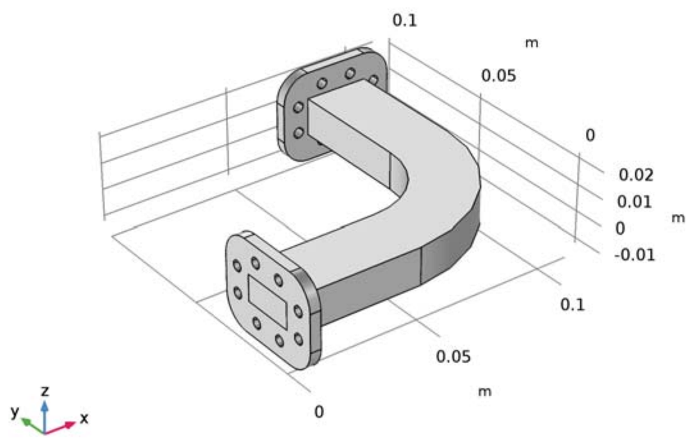
1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `rf_heating.mphbin`.

5 Click **Import**.



Use the wireframe rendering to see the inner parts of the waveguide.

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

DEFINITIONS

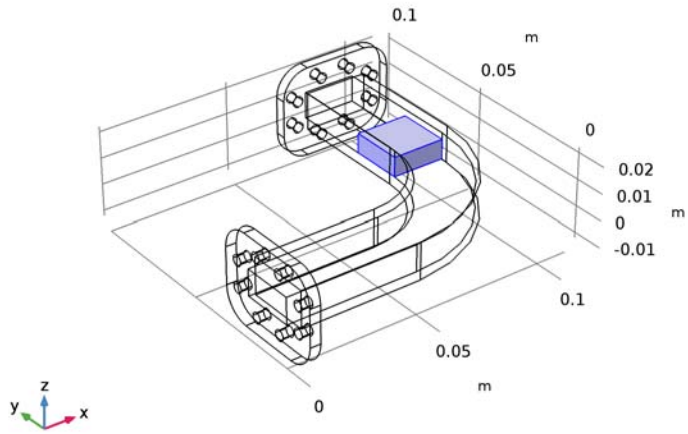
Create a set of selections for use before setting up the physics. First, create a selection for the dielectric block.

Explicit 1

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

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

3 Select Domain 3 only.



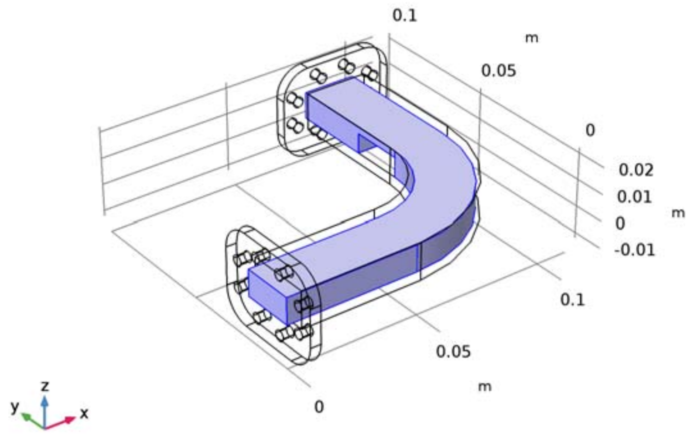
Add a selection for the air-filled region inside the waveguide.

Explicit 2

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

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

3 Select Domain 2 only.



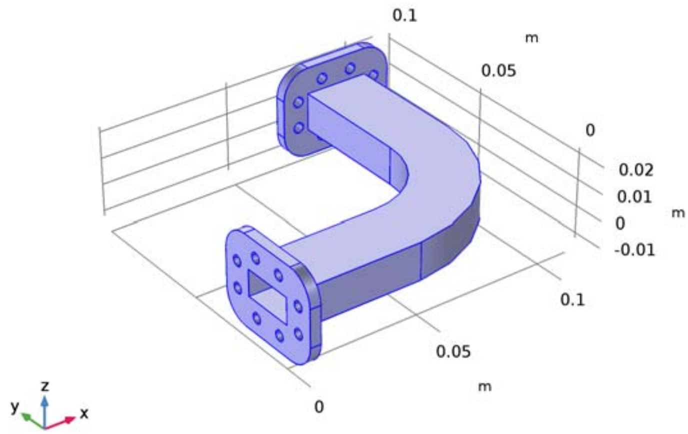
Add a selection for the waveguide structure.

Explicit 3

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

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

3 Select Domain 1 only.



Add a selection for the inner surface of the waveguide.

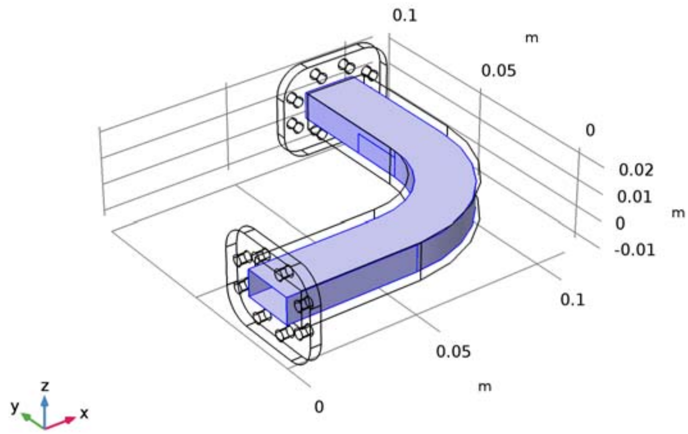
Explicit 4

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

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

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

4 Select Boundaries 16–18, 35, 53, 54, 72, 74, 75, 78, 96, and 97 only.



Add a selection for the outer surface of the waveguide.

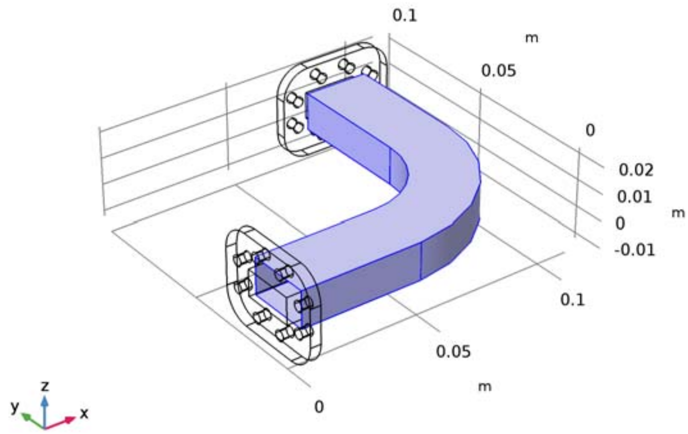
Explicit 5

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

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

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

4 Select Boundaries 48–52, 55, 69, and 98 only.



To get a better view, suppress some of the boundaries. Furthermore, by assigning the resulting settings to a View node, you can easily return to the same view later by clicking the **Go to View 2** button on the Graphics toolbar.

View 2

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

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

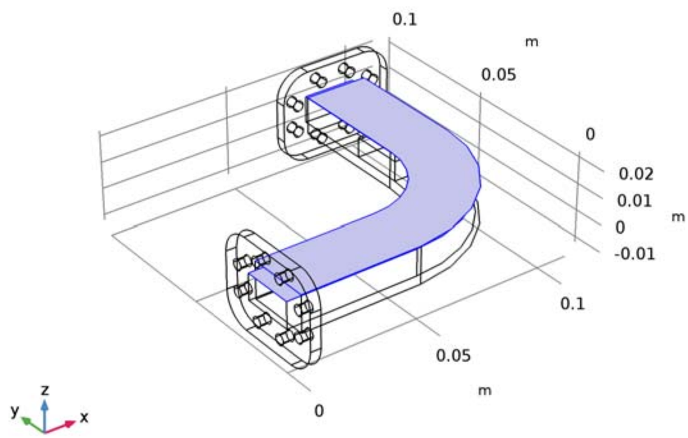
Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **View 2** 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 18 and 50 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.

3 In the list, select **1**.

4 Click **Remove from Selection**.

5 Select Domains 2 and 3 only.

Wave Equation, Electric 1

1 In the **Model Builder** window, expand the **Electromagnetic Waves, Frequency Domain (emw)** node, then click **Wave Equation, Electric 1**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Loss tangent, loss angle**.

HEAT TRANSFER IN SOLIDS (HT)

On the **Physics** toolbar, click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Heat Transfer in Solids (ht)**.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.
2. In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.
3. In the list, select **2**.
4. Click **Remove from Selection**.
5. Select Domains 1 and 3 only.

Heat Flux 1

1. In the **Model Builder** window, right-click **Heat Transfer in Solids (ht)** and choose **Heat Flux**.
2. In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Waveguide outside surfaces**.
4. Locate the **Heat Flux** section. Click the **Convective heat flux** button.
5. In the h text field, type `5`.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Physics-Controlled Mesh** section.
3. Clear the **Enable** check box.

Wave Equation, Electric 2

1. Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** and choose **Wave Equation, Electric**.
2. In the **Settings** window for **Wave Equation, Electric**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Air**.

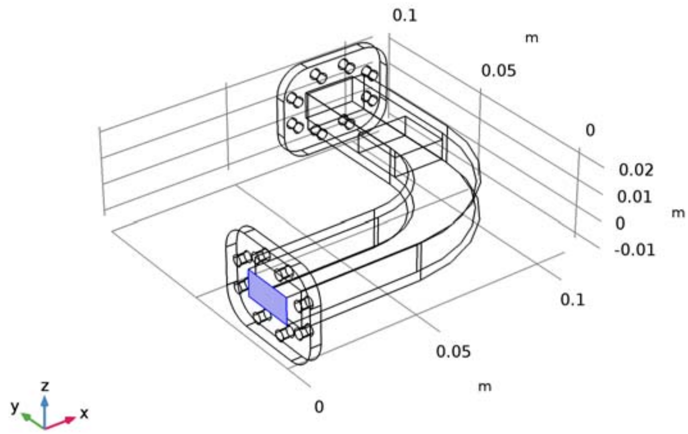
Impedance Boundary Condition 1

1. In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Impedance Boundary Condition**.
2. In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Waveguide inside surfaces**.

Port 1

1. Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 15 only.



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

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

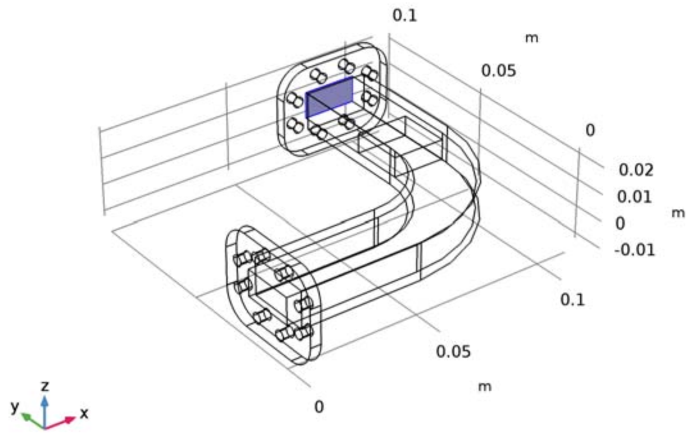
For the first port, wave excitation is **on** by default.

5 In the P_{in} text field, type `100`.

Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 79 only.



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

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

MATERIALS

Next, assign material properties on the model. Begin by specifying Aluminum for the waveguide structure.

ADD MATERIAL

1 On 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>Aluminum**.

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

MATERIALS

Aluminum (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat1)**.

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

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

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

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

MATERIALS

Air (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat2)**.

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

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

Material 3 (mat3)

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 `Dielectric` in the **Label** text field.

3 Select Domain 3 only.

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

Property	Name	Value	Unit	Property group
Relative permittivity (real part)	epsilonPrim	2.1	1	Dielectric losses
Loss tangent, loss angle	delta	0.001*(T/300[K])	rad	Loss tangent, loss angle
Relative permeability	mur	1	1	Basic
Thermal conductivity	k	0.3[W/m/K]*(T/300[K])	W/(m·K)	Basic
Density	rho	2200	kg/m³	Basic
Heat capacity at constant pressure	Cp	1050	J/(kg·K)	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Copper**.

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

MATERIALS

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

Copper (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat4)**.

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

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

4 From the **Selection** list, choose **Waveguide inside surfaces**.

MESH 1

Choose the maximum mesh size in the air domain smaller than 0.2 wavelengths using the parameter h_max that you defined earlier. For the dielectric materials, scale the mesh size by the inverse of the square root of the relative dielectric constant.

Size 1

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

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

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

4 From the **Selection** list, choose **Air**.

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 text field, type `h_max`.

Size 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Dielectric**.

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

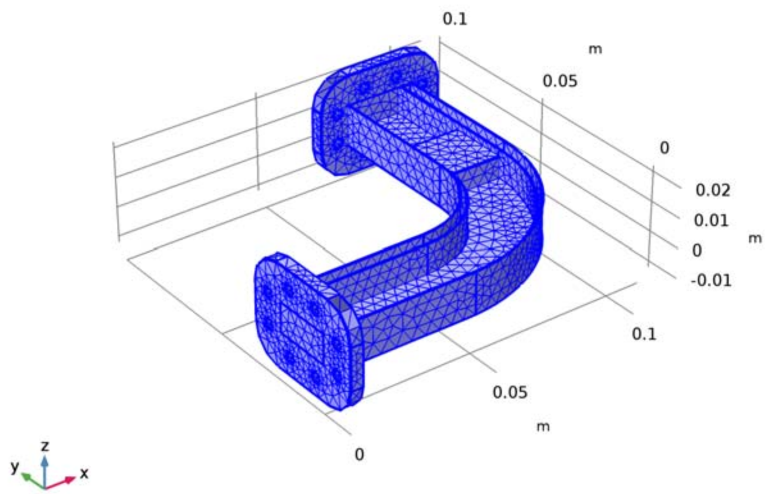
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `h_max/sqrt(2.1)`.

Free Tetrahedral 1

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

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.



STUDY 1

Step 1: Frequency-Transient

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency-Transient**.

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

3 In the **Times** text field, type `range(0,15,300)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.001`.

6 In the **Frequency** text field, type `f0`.

7 In the **Model Builder** window, click **Study 1**.

8 In the **Settings** window for **Study**, locate the **Study Settings** section.

9 Clear the **Generate default plots** check box.

10 On the **Home** toolbar, click **Compute**.

RESULTS

Data Sets

Plot the transient response of the peak temperature.

1D Plot Group 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Evaluation>Maximum**.

2 On the **Results** toolbar, click **1D Plot Group**.

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

4 From the **Data set** list, choose **Maximum 1**.

Point Graph 1

1 Right-click **1D Plot Group 1** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1> Heat Transfer in Solids>Temperature>T - Temperature**.

3 Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.

4 On the **1D Plot Group 1** toolbar, click **Plot**.

Compare the resulting plot with that shown in Figure 2.

Next, add a Frequency-Stationary study to evaluate the peak temperature which can be observed with the **Frequency-Transient** study after applying a enough long time so the peak temperature is saturated.

ADD STUDY

1 On 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 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency-Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Frequency-Stationary**.

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

3 In the **Frequency** text field, type `f0`.

4 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The default plots show the distribution of the electric field norm and the temperature. For the temperature plot, first change the unit to the degree Celsius and then, add arrow plots of the electric fields, magnetic fields, and power flow.

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Electric>emw.Ex,emw.Ey,emw.Ez - Electric field**.

3 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type 40.

4 Find the **Y grid points** subsection. In the **Points** text field, type 40.

5 Find the **Z grid points** subsection. In the **Points** text field, type 1.

6 On the **Temperature (ht)** toolbar, click **Plot**.

Arrow Volume 2

1 Right-click **Results>Temperature (ht)>Arrow Volume 1** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Magnetic>emw.Hx,emw.Hy,emw.Hz - Magnetic field**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

Arrow Volume 3

1 Right-click **Results>Temperature (ht)>Arrow Volume 2** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>**

Electromagnetic Waves, Frequency Domain>Energy and power>emw.Poavx,...,emw.Poavz - Power flow, time average.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

Compare the resulting plot with that shown in Figure 3.

Finally, reproduce the plot of the loss tangent on the dielectric block shown in Figure 4.

To create the plot, reuse the plot group named Isothermal Contours (ht).

Isothermal Contours (ht)

1 In the **Model Builder** window, under **Results** click **Isothermal Contours (ht)**.

2 In the **Settings** window for **3D Plot Group**, type `Loss Tangent (emw)` in the **Label** text field.

Isosurface

In the **Model Builder** window, expand the **Results>Loss Tangent (emw)** node.

Surface 1

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

2 In the **Model Builder** window, under **Results** right-click **Loss Tangent (emw)** and choose **Surface**.

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Material properties>emw.delta - Loss tangent, loss angle**.

4 On the **Loss Tangent (emw)** toolbar, click **Plot**.

Click the Zoom Box button on the Graphics toolbar and then use the mouse to zoom in.

Created in COMSOL Multiphysics 5.3

Fast Prototyping of a Butler Matrix Beamforming Network

Introduction

A Butler matrix is a passive beamforming feed network. It is a cost-effective feed network for phased array antennas because the circuit can be fabricated in the form of microstrip lines and it is viable to perform beam scanning without deploying expensive active devices. This example guides how to design such a circuit efficiently using the Transmission Line physics interface. The results show the logarithmic voltage on the Butler matrix beamforming circuit at 30 GHz and the arithmetic phase progression at each output port.

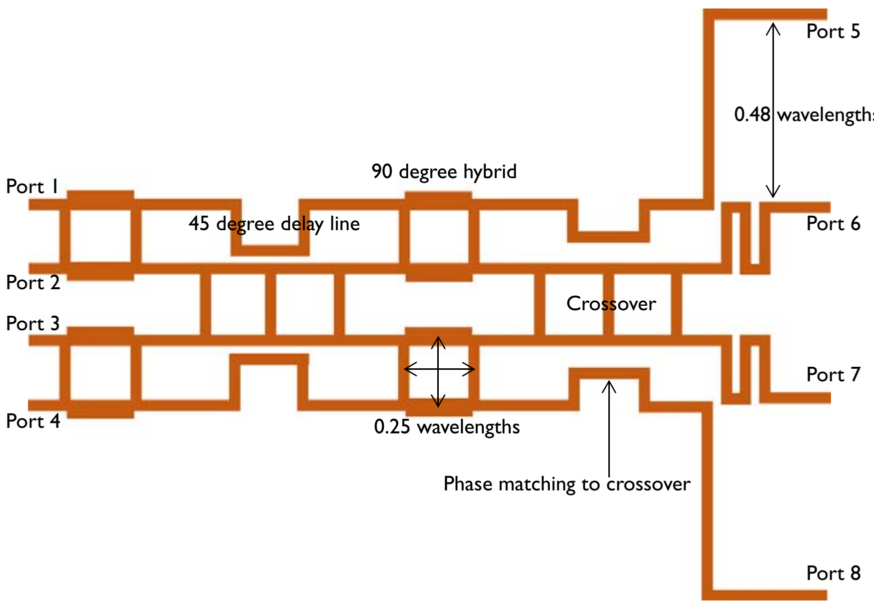


Figure 1: A microstrip 4×4 Butler matrix beamforming network for a phased array antenna

Model Definition

The butler matrix beamforming network consists of a few subsections: 90 degree hybrid, 45 degree delay line, crossover, transition matching the output phase to that of crossover, and inner and outer front-ends. Since these subsections are repeatedly used in the entire structure, the geometry building process can be simplified by adding these subsections as the Geometry Parts under Global Definition node and reusing them as necessary.

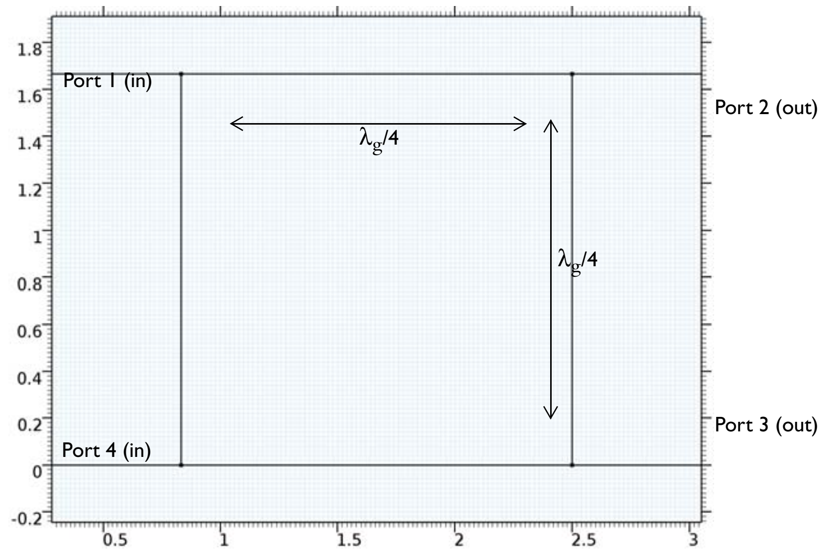


Figure 2: The part geometry of a 90 degree hybrid (branch-line coupler)

The geometry of a 90 degree hybrid, also known as a branch-line coupler is shown in Figure 2. Ref. 1 discusses the design characteristics and its S-parameters calculated using even-odd model analysis. A full 3D COMSOL model is available in Ref. 2. The 90 degree hybrid splits the input power equally into two output ports (-3 dB) with a 90 degree phase difference. Because the geometry is symmetric, the response of the circuit is reciprocal regardless of the input port configuration. In this example, the input ports are located on the left side and there is no coupled power between the input ports that is also described by its S-parameter matrix:

$$[S] = \frac{-1}{\sqrt{2}}\begin{bmatrix} 0 & j & 1 & 0 \\ j & 0 & 0 & 1 \\ 1 & 0 & 0 & j \\ 0 & 1 & j & 0 \end{bmatrix}$$

Figure 3 describes a delay line geometry providing a 45 degrees phase lag than the output phase of the crossover. Figure 4 shows a transition part that matches the output phase to that of the crossover.

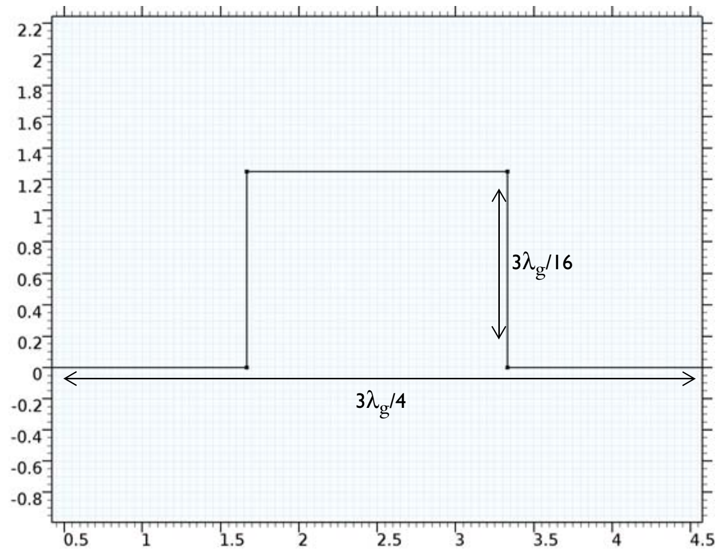


Figure 3: The part geometry of a 45 degree delay line that is 0.125 wavelengths longer than the crossover part.

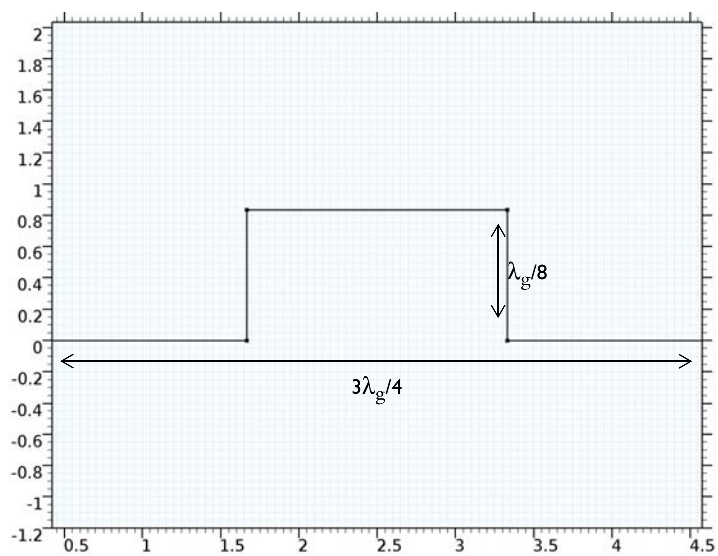


Figure 4: The part geometry of a transition structure. The electrical length is same as that in the input signal path of the crossover part.

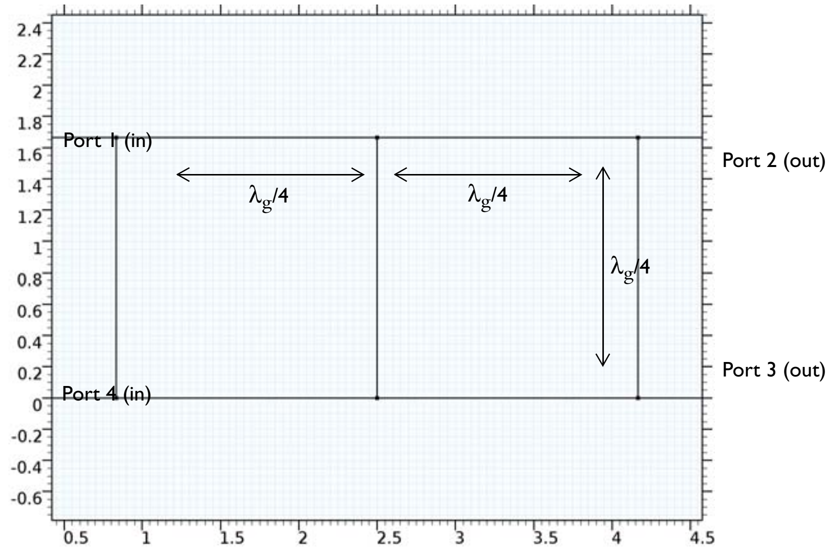


Figure 5: The part geometry of a crossover structure. The port definition is only for the subsection analysis.

The geometry of a crossover in Figure 5 is analogous of a two-section cascaded branch-line coupler, but it consists of only 50 Ω lines. Its behavior can be analyzed with the same even-odd analysis method (Ref. 1) used for the branch-line coupler characterization. The even-odd analysis transforms the four-port network into two decoupled two-port networks. After the transformation, each cascaded two port-network can be described via ABCD parameters.

If the circuit is normalized by the 50 Ω reference impedance, the ABCD parameters for each section are

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix}_{\text{open, shunt}} = \begin{bmatrix} 1 & 0 \\ j & 1 \end{bmatrix}$$

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix}_{\text{short, shunt}} = \begin{bmatrix} 1 & 0 \\ -j & 1 \end{bmatrix}$$

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix}_{50\Omega, \frac{\lambda}{4}\text{line}} = \begin{bmatrix} 0 & j \\ j & 0 \end{bmatrix}$$

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix}_{\text{even}} = \begin{bmatrix} 1 & 0 \\ j & 1 \end{bmatrix}\begin{bmatrix} 0 & j \\ j & 0 \end{bmatrix}\begin{bmatrix} 1 & 0 \\ j & 1 \end{bmatrix}\begin{bmatrix} 0 & j \\ j & 0 \end{bmatrix}\begin{bmatrix} 1 & 0 \\ j & 1 \end{bmatrix} = \begin{bmatrix} 0 & -j \\ -j & 0 \end{bmatrix}$$

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix}_{\text{odd}} = \begin{bmatrix} 1 & 0 \\ -j & 1 \end{bmatrix}\begin{bmatrix} 0 & j \\ j & 0 \end{bmatrix}\begin{bmatrix} 1 & 0 \\ -j & 1 \end{bmatrix}\begin{bmatrix} 0 & j \\ j & 0 \end{bmatrix}\begin{bmatrix} 1 & 0 \\ -j & 1 \end{bmatrix} = \begin{bmatrix} 0 & j \\ j & 0 \end{bmatrix}$$

The reflection and transmission coefficients from ABCD are defined as

$$\Gamma = \frac{A+B-C-D}{A+B+C+D}$$

$$T = \frac{2}{A+B+C+D}$$

The wave amplitude at each port is

$$B_1 = (\Gamma_{\text{even}} + \Gamma_{\text{odd}})/2 = 0$$

$$B_2 = (T_{\text{even}} + T_{\text{odd}})/2 = 0$$

$$B_3 = (T_{\text{even}} - T_{\text{odd}})/2 = j$$

$$B_4 = (\Gamma_{\text{even}} + \Gamma_{\text{odd}})/2 = 0$$

Because it is a passive and reciprocal network, the S-parameters are

$$\begin{bmatrix} S \end{bmatrix} = \begin{bmatrix} 0 & 0 & j & 0 \\ 0 & 0 & 0 & j \\ j & 0 & 0 & 0 \\ 0 & j & 0 & 0 \end{bmatrix}$$

The two input ports are isolated from each other. The input signal from the upper left side flows to the output at the lower right side while the input signal from the lower left side flows to the output at the upper right side. The ladder-shape crossover structure works like X-shape crossover lines.

Figure 6 and Figure 7 show the geometry of front-end parts that adjust the distance between output ports from a quarter-wave length to 0.48 wavelengths without distorting the output phase relation. The higher gain of an antenna array can be realized by increasing the distance between antenna elements, but this will result in an undesirable higher sidelobe level and a grating lobe. The given spacing configuration for antenna array elements provides the antenna radiation pattern with a reasonable gain and sidelobe level.

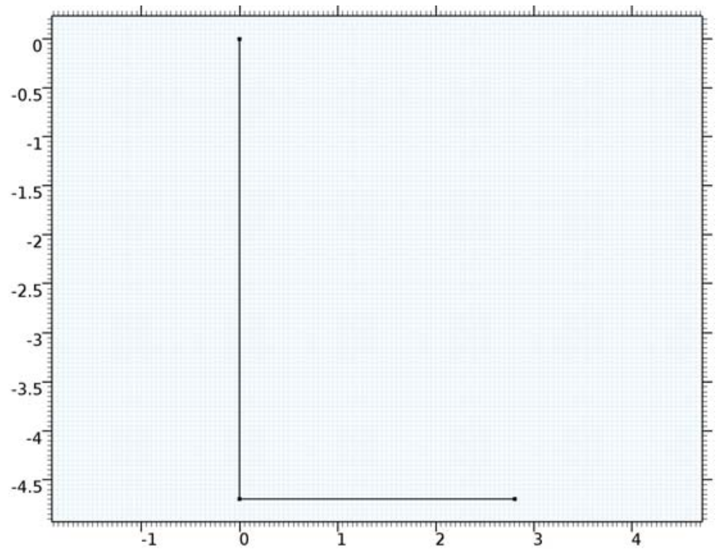


Figure 6: The part geometry of an outer front-end structure.

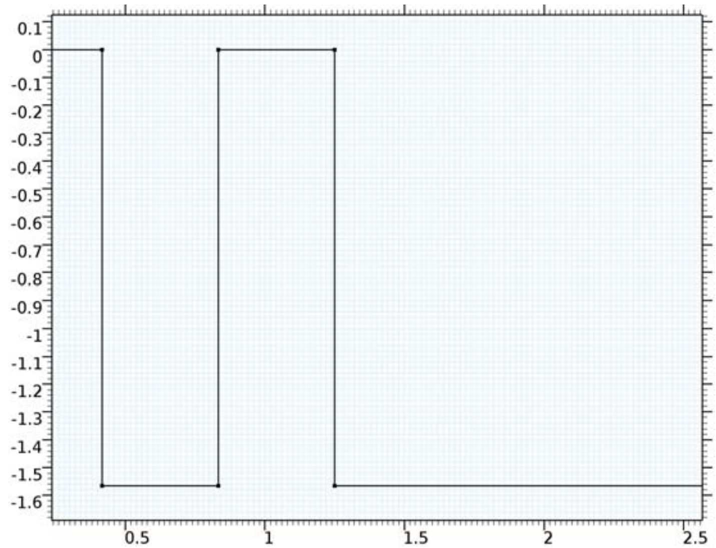


Figure 7: The part geometry of an inner front-end structure.

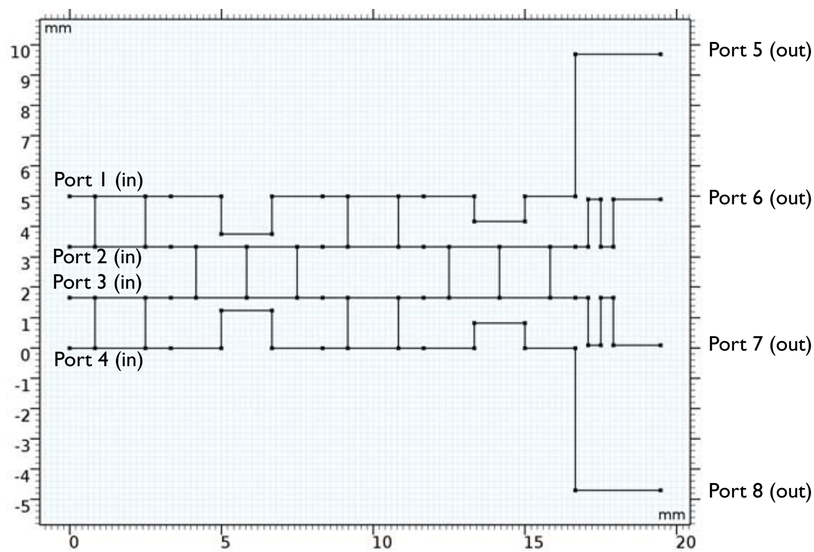


Figure 8: The finalized geometry of a butler matrix beamforming network.

By combining four 90 degree hybrids (branch-line couplers), two 45 degree delay lines, two phase matching transitions, two inner front-ends, and two outer front-ends, the geometry for the butler matrix beamforming network is completed (Figure 8).

All transmission line distributed element parameters except for a few branch-lines are set based on a 50 Ω microstrip line built on a 20 mil lossless substrate with permittivity $\varepsilon_r = 3.38$ and 1 oz copper. The accurate values can be calculated accurately from Ref. 3.

TABLE 1: CALCULATED TRANSMISSION LINE PARAMETERS OF A 50 Ω MICROSTRIP LINE.

R	L	G	C
12.41 Ω/m	272.9 nH/m	0 S/m	107.1 pF/m

The contribution of the distributed resistance on the insertion loss with the given substrate properties is less than 0.05 dB. To make the modeling steps simpler in this example, the approximated parameter values in Table 2 are used for a 50 Ω microstrip line.

TABLE 2: SIMPLIFIED TRANSMISSION LINE PARAMETERS OF A 50 Ω MICROSTRIP LINE.

R	L	G	C
0 Ω/m	250 nH/m	0 S/m	100 pF/m

The transmission line parameters with a different characteristic impedance value, $Z_0/\sqrt{2}$ for the branch-lines, are adjusted using the normalized impedance. The distributed inductance is proportionally scaled and the distributed capacitance is inversely scaled by the normalized impedance of the microstrip line.

In order to excite ports one by one, the port sweep option in the transmission line physics interface is activated and combined with a parametric sweep in the study steps. Each port is terminated by a lumped port with 50 Ω reference characteristic impedance

Results and Discussion

The default plot show the real value of the voltage on the transmission lines. The default input expression is changed to plot the logarithmic value of the voltage (Figure 9). The plot shows that port 1, port 2 and port 3 have no coupled power (below -100 dB) from the excited port 4.

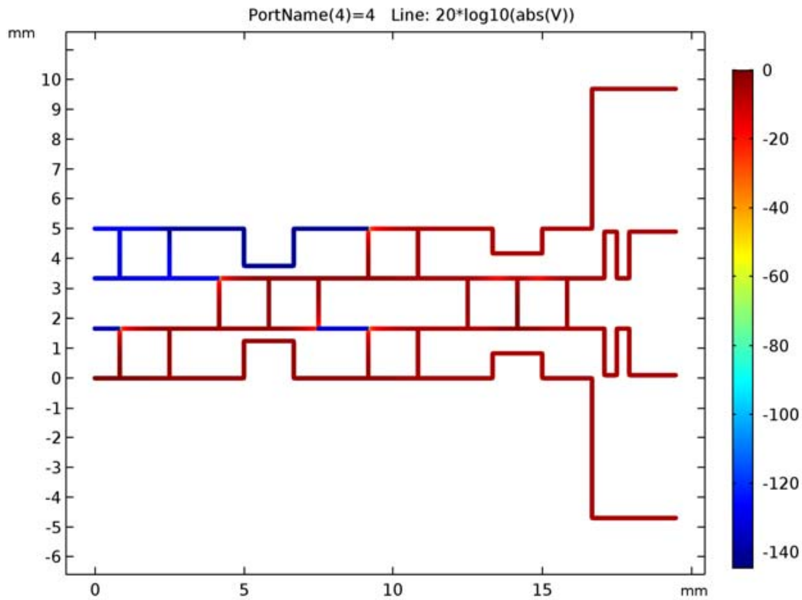


Figure 9: The dB-scaled voltage on the transmission lines when port 4 is excited. Port1, port 2 and port 3 are isolated below -100 dB.

In Figure 10, the minimum range of the dB-scaled voltage plot is set to -10 dB to get a closer look at the level of each output port. The input voltage is equally distributed to all four output ports (- 6dB).

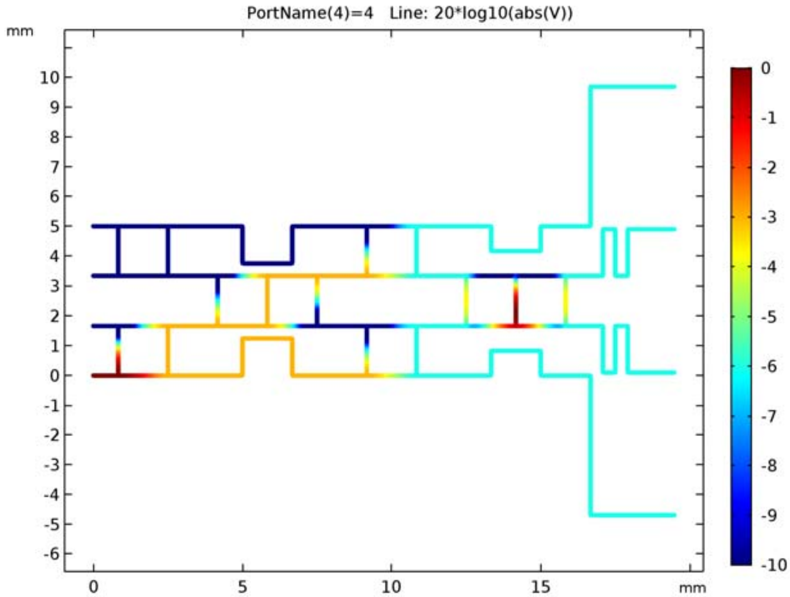


Figure 10: The range of the dB-scaled voltage plot is adjusted to see the output voltage level.

Table 3 shows the evaluated phase at each output port.

TABLE 3: THE EVALUATED PHASE OF VOLTAGE AT EACH PORT

	PORT 5	**PORT 6**	**PORT 7**	**PORT 8**
PORT 1 EXCITED	-90°	-135°	-180°	135°
PORT 2 EXCITED	-180°	-45°	90°	-135°
PORT 3 EXCITED	-135°	90°	-45°	-180°
PORT 4 EXCITED	135°	-180°	-135°	-90°

By adjusting some of the evaluated angles, the phase at each port can be configured in an arithmetic order and the resulted phase progression is summarized in Table 4. If the butler matrix beamforming network is excited in the order of port 3 (-135 degrees), port 1 (-45 degress), port 4 (45 degrees) and port 2 (135 degrees), and connected to a 4×1 antenna array, the antenna radiation pattern will be steered from one side to the other side (Figure 11). Note that the antenna array model in Figure 11 is not included in this example.

TABLE 4: THE EVALUATED PHASE OF VOLTAGE AT EACH PORT (PHASE ADJUSTED)

	PORT 5	PORT 6	PORT 7	PORT 8	PHASE PROGRESSION
PORT 1 EXCITED	-90°	-135°	-180°	-225°	-45°
PORT 2 EXCITED	-180°	-45°	90°	225°	135°
PORT 3 EXCITED	225°	90°	-45°	180°	-135°
PORT 4 EXCITED	-225°	-180°	-135°	-90°	45°

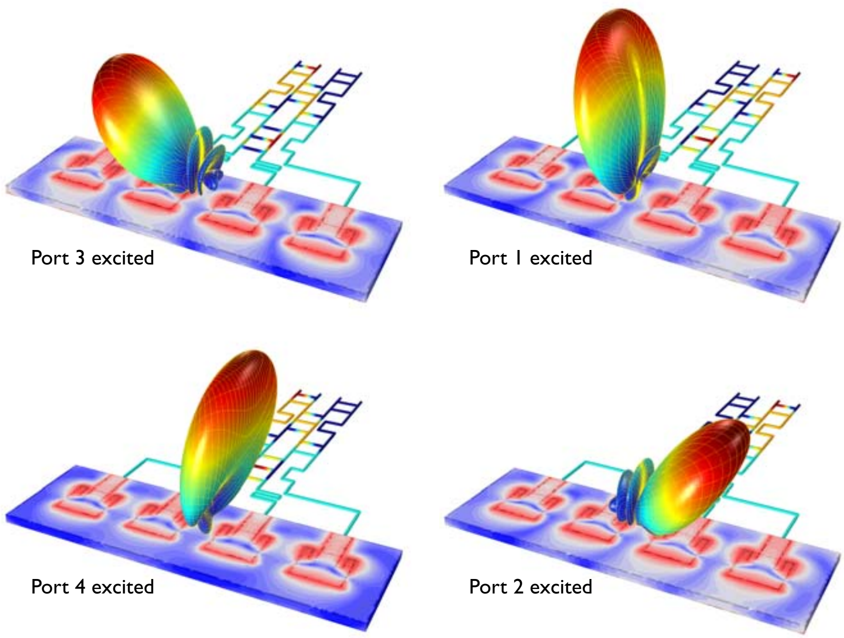


Figure 11: The far-field radiation pattern of a 4×1 microstrip patch antenna array connected to the butler matrix beamforming network. The antenna model is not included in this example.

References

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

2. COMSOL Application Gallery, "*Branch-Line Coupler*", https://www.comsol.com/model/branch-line-coupler-11727

3. COMSOL Application Gallery, “*Transmission Line Parameter Calculator*”, https://www.comsol.com/model/transmission-line-parameter-calculator-22351

Application Library path: `RF_Module/Couplers_and_Power_Dividers/transmission_line_butler`

Model Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Radio Frequency>Transmission Line (tl)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model’s Application Libraries folder and double-click the file `transmission_line_butler_parameters.txt`.

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

The 4x4 Butler matrix beamforming network in this example consists of a few parts that are repeatedly shown in the geometry. To make the modeling process more efficient, define these as **Geometry Parts** and reuse them as necessary.

5 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts**.

6 Right-click **Global Definitions>Geometry Parts** and choose **2D Part**.

PART 1

1 In the **Settings** window for **Part**, type `90 Degree Hybrid` in the **Label** text field.

2 Locate the **Units** section. From the **Length unit** list, choose **mm**.

Bézier Polygon 1 (b1)

On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

90 DEGREE HYBRID

Bézier Polygon 1 (b1)

1 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

2 Find the **Added segments** subsection. Click **Add Linear**.

3 Find the **Control points** subsection. In row **2**, set **x** to `ul*2`.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `ul/2`.

5 In row **2**, set **x** to `ul/2`.

6 In row **2**, set **y** to `ul`.

Rotate 1 (rot1)

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

2 Click the **Select All** button on the **Graphics** toolbar.

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

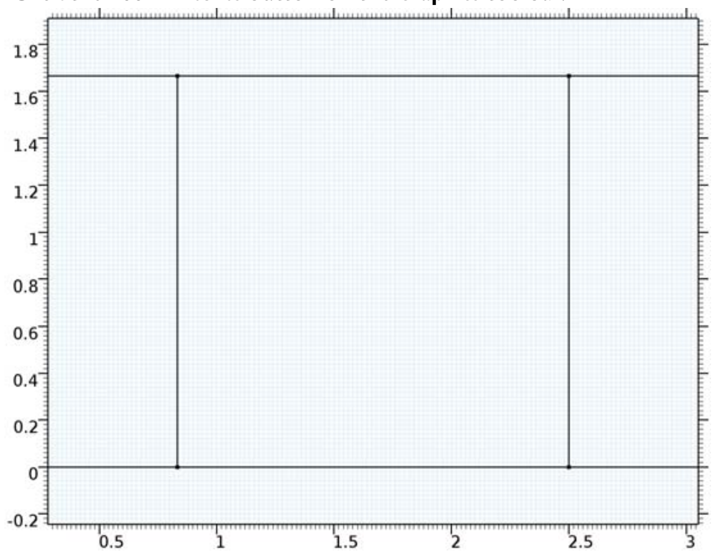
4 In the **Rotation** text field, type `0 180`.

5 Locate the **Center of Rotation** section. In the **x** text field, type `ul`.

6 In the **y** text field, type `ul/2`.

7 Click **Build Selected**.

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



GLOBAL DEFINITIONS

Right-click **Geometry Parts** and choose **2D Part**.

PART 2

1. In the **Settings** window for **Part**, type `45 Degree Delay` in the **Label** text field.
2. Locate the **Units** section. From the **Length unit** list, choose **mm**.

Bézier Polygon 1 (b1)

On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

45 DEGREE DELAY

Bézier Polygon 1 (b1)

1. In the **Settings** window for **Bézier Polygon**, locate the **General** section.
2. From the **Type** list, choose **Open curve**.
3. Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.
4. Find the **Control points** subsection. In row **2**, set **x** to `ul`.
5. Find the **Added segments** subsection. Click **Add Linear**.
6. Find the **Control points** subsection. In row **2**, set **y** to `ul*0.75`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **x** to `ul*2`.

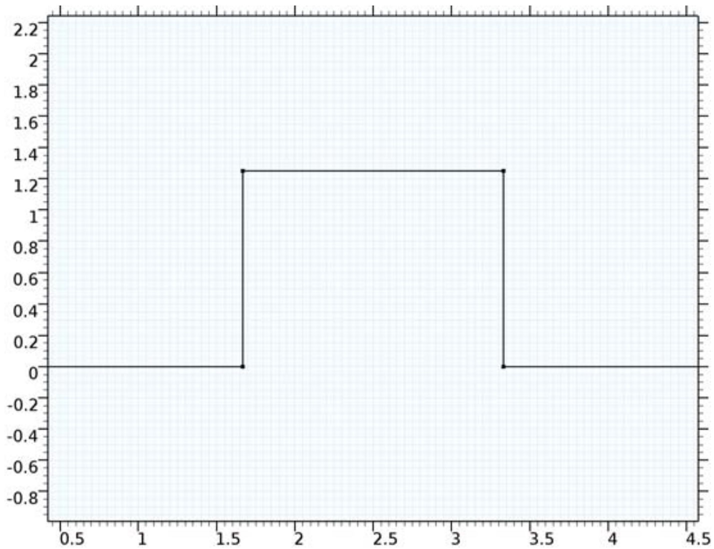
9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. In row **2**, set **y** to `0`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **x** to `ul*3`.

13 Click **Build Selected**.



GLOBAL DEFINITIONS

Right-click **Geometry Parts** and choose **2D Part**.

PART 3

1 In the **Settings** window for **Part**, type `Transition` in the **Label** text field.

2 Locate the **Units** section. From the **Length unit** list, choose **mm**.

3 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

TRANSITION

Bézier Polygon 1 (b1)

1 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

2 From the **Type** list, choose **Open curve**.

3 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to `ul`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **y** to `ul/2`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **x** to `ul*2`.

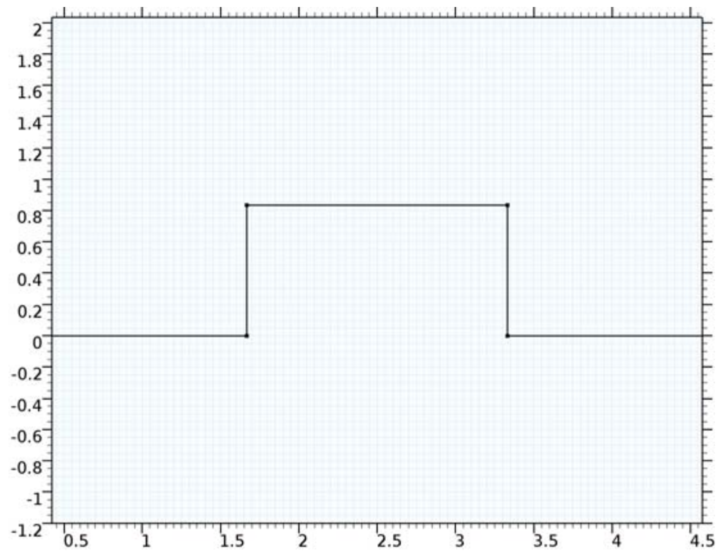
9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. In row **2**, set **y** to `0`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **x** to `ul*3`.

13 Click **Build Selected**.



GLOBAL DEFINITIONS

Right-click **Geometry Parts** and choose **2D Part**.

PART 4

1 In the **Settings** window for **Part**, type `Crossover` in the **Label** text field.

2 Locate the **Units** section. From the **Length unit** list, choose **mm**.

3 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

CROSSOVER

Bézier Polygon 1 (b1)

1 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

2 Find the **Added segments** subsection. Click **Add Linear**.

3 Find the **Control points** subsection. In row **2**, set **x** to `ul*3`.

Bézier Polygon 2 (b2)

1 Right-click **Global Definitions>Geometry Parts>Crossover>Bézier Polygon 1 (b1)** and choose **Duplicate**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Control points** subsection. In row **1**, set **y** to `ul`.

4 In row **2**, set **y** to `ul`.

Bézier Polygon 3 (b3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `ul/2`.

5 In row **2**, set **x** to `ul/2`.

6 In row **2**, set **y** to `ul`.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

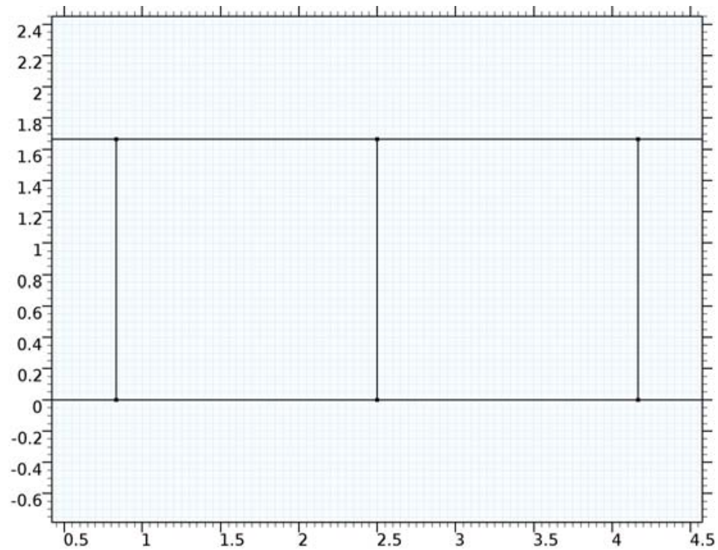
2 Select the object **b3** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `3`.

5 Locate the **Displacement** section. In the **x** text field, type `ul`.

6 Click **Build Selected**.



GLOBAL DEFINITIONS

In the **Model Builder** window, under **Global Definitions** right-click **Geometry Parts** and choose **2D Part**.

PART 5

1 In the **Settings** window for **Part**, type `Front-end, outer` in the **Label** text field.

2 Locate the **Units** section. From the **Length unit** list, choose **mm**.

Bézier Polygon 1 (b1)

On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

FRONT-END, OUTER

Bézier Polygon 1 (b1)

1 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

2 From the **Type** list, choose **Open curve**.

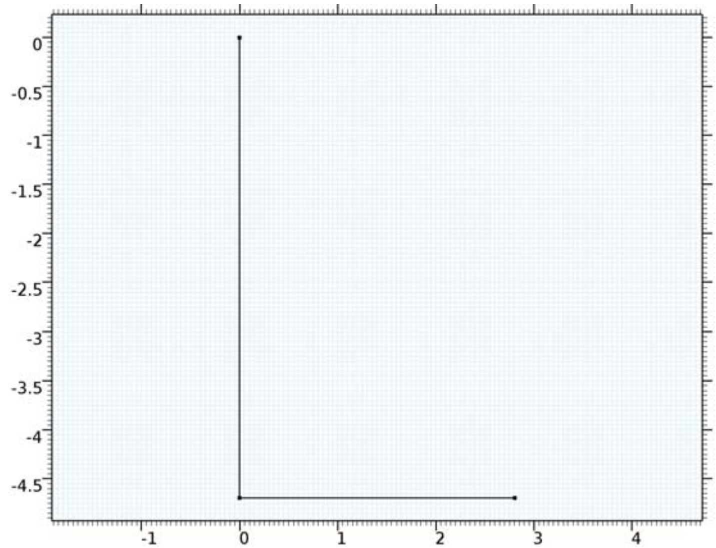
3 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **y** to `-1.5*(array_d-ul)`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **x** to `-1.5*array_d+ul*6`.

7 Click **Build Selected**.



GLOBAL DEFINITIONS

Right-click **Geometry Parts** and choose **2D Part**.

PART 6

1 In the **Settings** window for **Part**, type `Front-end, inner` in the **Label** text field.

2 Locate the **Units** section. From the **Length unit** list, choose **mm**.

3 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

FRONT-END, INNER

Bézier Polygon 1 (b1)

1 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

2 From the **Type** list, choose **Open curve**.

3 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to `ul*0.25`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **y** to `-(array_d-ul)/2`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **x** to `ul*0.5`.

9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. In row **2**, set **y** to `0`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **x** to `ul*0.75`.

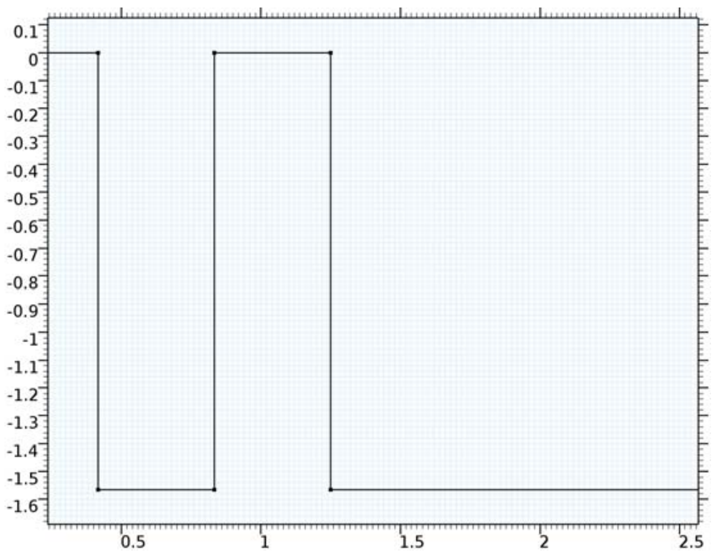
13 Find the **Added segments** subsection. Click **Add Linear**.

14 Find the **Control points** subsection. In row **2**, set **y** to `-(array_d-ul)/2`.

15 Find the **Added segments** subsection. Click **Add Linear**.

16 Find the **Control points** subsection. In row **2**, set **x** to `-1.5*array_d+ul*6`.

17 Click **Build Selected**.



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**.

90 Degree Hybrid 2 (pi2)

1 On the **Geometry** toolbar, click **Parts** and choose **90 Degree Hybrid**.

2 On the **Geometry** toolbar, click **Parts** and choose **90 Degree Hybrid**.

3 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

4 In the **x-displacement** text field, type `ul*5`.

5 Click **Build Selected**.

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

45 Degree Delay 1 (pi3)

1 On the **Geometry** toolbar, click **Parts** and choose **45 Degree Delay**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `ul*2`.

Transition 1 (pi4)

1 On the **Geometry** toolbar, click **Parts** and choose **Transition**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `ul*7`.

4 Click **Build Selected**.

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

Front-end, outer 1 (pi5)

1 On the **Geometry** toolbar, click **Parts** and choose **Front-end, outer**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `ul*10`.

4 Click **Build Selected**.

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

Front-end, inner 1 (pi6)

1 On the **Geometry** toolbar, click **Parts** and choose **Front-end, inner**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `ul*10`.

4 In the **y-displacement** text field, type `ul`.

5 Click **Build Selected**.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Click the **Select All** button on the **Graphics** toolbar.

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

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

5 Locate the **Point on Line of Reflection** section. In the **y** text field, type `ul*1.5`.

6 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

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

8 Click **Build Selected**.

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

Crossover 1 (pi7)

1 On the **Geometry** toolbar, click **Parts** and choose **Crossover**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `ul*2`.

4 In the **y-displacement** text field, type `ul`.

Crossover 2 (pi8)

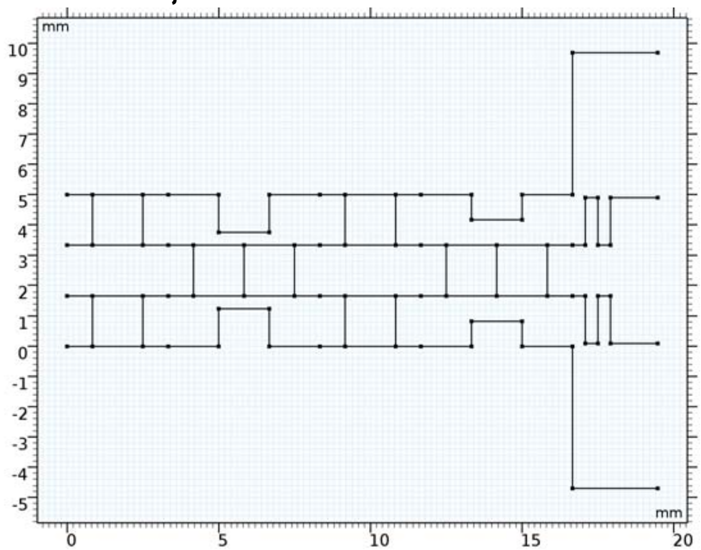
1 On the **Geometry** toolbar, click **Parts** and choose **Crossover**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `ul*7`.

4 In the **y-displacement** text field, type `ul`.

5 Click **Build All Objects**.



TRANSMISSION LINE (TL)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Transmission Line (tl)**.
2. In the **Settings** window for **Transmission Line**, locate the **Port Sweep Settings** section.
3. Select the **Activate port sweep** check box.

   Activate the port sweep option in the physics interface and combine with a parametric sweep. This will excite ports one by one.

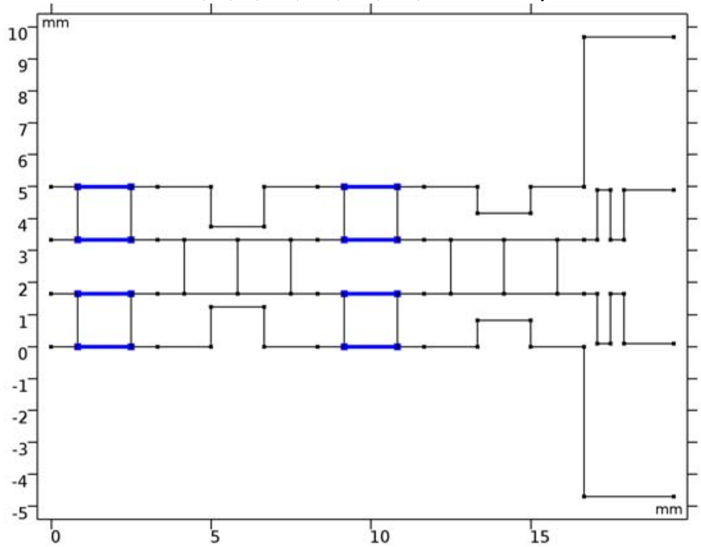
Transmission Line Equation 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Transmission Line (tl)** click **Transmission Line Equation 1**.
2. In the **Settings** window for **Transmission Line Equation**, locate the **Transmission Line Equation** section.
3. In the L text field, type `L0`.
4. In the C text field, type `C0`.

Transmission Line Equation 2

1. In the **Model Builder** window, right-click **Transmission Line (tl)** and choose **Transmission Line Equation**.

2 Select Boundaries 6, 7, 9, 10, 43, 44, 46, and 47 only.



Set the impedance of the selected transmission lines (branch-lines in the 90 degree hybrid) to `Z0/sqrt(2)` by adjusting the distributed inductance and capacitance values.

3 In the **Settings** window for **Transmission Line Equation**, locate the **Transmission Line Equation** section.

4 In the L text field, type `L0*z1`.

5 In the C text field, type `C0/z1`.

Lumped Port 1

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 4 only.

See Figure 8 to confirm the lumped port configuration.

Lumped Port 2

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 3 only.

Lumped Port 3

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 2 only.

Lumped Port 4

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 1 only.

Lumped Port 5

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 82 only.

Lumped Port 6

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 81 only.

Lumped Port 7

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 80 only.

Lumped Port 8

1 Right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 79 only.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

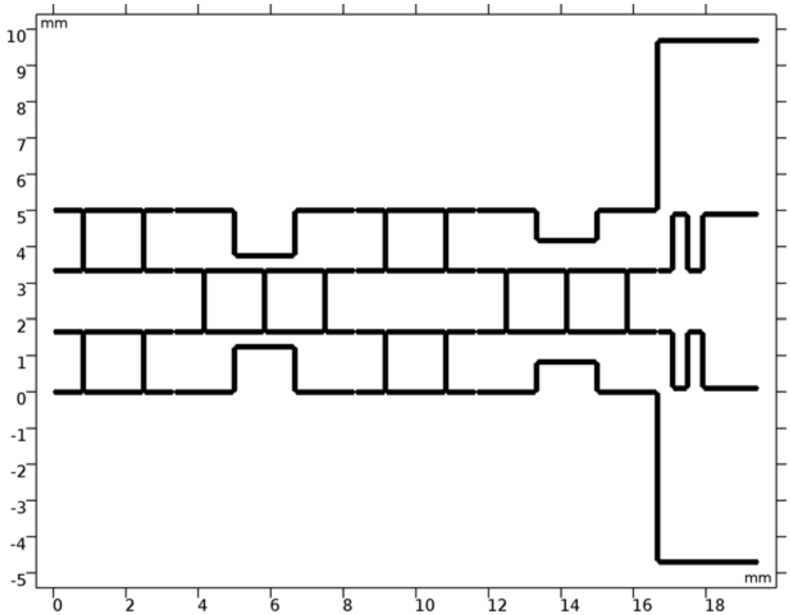
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `ul/15`.

5 Click the **Select Boundaries** button on the **Graphics** toolbar.

6 Click **Build All**.



STUDY 1

Parametric Sweep

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

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

3 Click **Add**.

4 Click to select row number 1 in the table.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
PortName	1 2 3 4	

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 In the **Frequencies** text field, type `f0`.

4 On the **Study** toolbar, click **Compute**.

RESULTS

Line Graph

1 In the **Model Builder** window, expand the **2D Plot Group 1** node, then click **Line Graph**.

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

3 In the **Expression** text field, type `20*log10(abs(V))`.

4 On the **2D Plot Group 1** toolbar, click **Plot**.

Other input ports (port 1, port2 and port 3) are fully isolated from the excited port 4. See Figure 9.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Minimum** text field, type `-10`.

7 In the **Maximum** text field, type `0`.

8 On the **2D Plot Group 1** toolbar, click **Plot**.

Figure 10 shows that the input power to port 4 is equally split into all output ports (-6 dB).

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`arg(tl.Vport_5)`	`deg`	`Port 5 phase`
`arg(tl.Vport_6)`	`deg`	`Port 6 phase`
`arg(tl.Vport_7)`	`deg`	`Port 7 phase`
`arg(tl.Vport_8)`	`deg`	`Port 8 phase`

4 Click **Evaluate**.

Compare the evaluated values to those in Table 3.

Fast Modeling of a Transmission Line Low-Pass Filter

Introduction

One way to design a filter is to utilize the element values of well-known filter prototypes such as maximally flat or equal-ripple low-pass filters. It is easier to fabricate a distributed element filter on a microwave substrate than a lumped element filter since it is cumbersome to find off-the-shelf capacitors and inductors exactly matched to the frequency-scaled element values of the filter prototype. This example demonstrates the design process of a distributed element filter using Richard's transformation, Kuroda's identity, and the Transmission Line physics interface. This approach is very fast compared to solving Maxwell's equations in 3D. The model simulates a three-element 0.5 dB equal-ripple low-pass filter that has a cutoff frequency at 4 GHz. The resulting S-parameter plot shows a low-pass frequency response that is also periodically observed at higher frequency range.

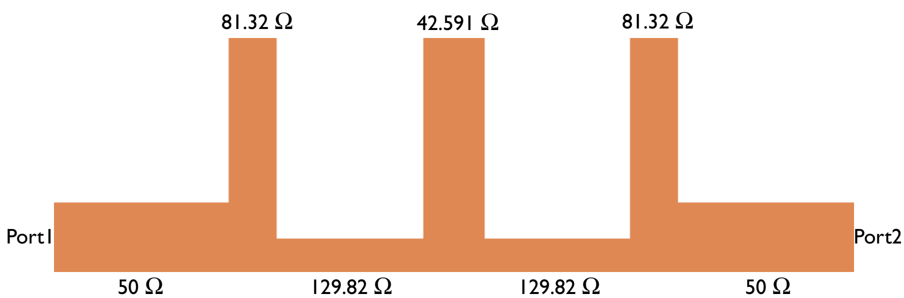


Figure 1: Microstrip low-pass filter circuit. The impedance for each unit length (0.125 wavelengths) stub is calculated from the element values of a three element 0.5 dB equal-ripple low-pass filter.

Model Definition

The modeling process of a low-pass filter can be summarized as

- Define a filter type such as maximally flat or equal-ripple.
- Identify element values for the filter prototype.
- Convert the inductors and capacitors in the lumped element filter to series and shunt stubs by using Richard's transformation.
- Apply Kuroda's identity to convert short-circuited series stubs to open-circuited shunt stubs.
- Scale the impedance of stubs by the reference characteristic impedance (50 Ω) and set the length of stubs to 0.125 wavelengths defined by the cutoff frequency.

Ref. 1 provides the element values for a 0.5 dB equal-ripple low-pass filter. The element values for a three element prototype are also shown in Table 1.

TABLE 1: 0.5 DECIBEL EQUAL-RIPPLE LOW-PASS FILTER ELEMENT VALUES, N = 3

g_1	g_2	g_3	g_4
1.5963	1.0967	1.5963	1

These values are unscaled inductance and capacitance in a lumped element circuit that need to be converted to distributed elements. Richard's transformation converts an inductor to a short-circuited stub and a capacitor to an open-circuited stub, respectively. The model is based on a three element prototype beginning with a series inductor. Two series inductors are transformed to series stubs and one shunt capacitor is transformed to a shunt stub. The normalized impedance of the open-circuited stub is the same as the lumped element value of the inductor (Equation 1) and that for the short-circuited stub is the inverse of the lumped element value of the capacitor (Equation 2).

$$Z\mathrm{stub}_{\mathrm{short-circuited}} = L \tag{1}$$

$$Z\mathrm{stub}_{\mathrm{open-circuited}} = \frac{1}{C} \tag{2}$$

The short-circuited series stub is not easily realizable as a microstrip circuit so it has to be transformed again using Kuroda's identity that will convert a short-circuited series to an open-circuited shunt stub. A unit length (0.125 wavelengths) transmission line element must be added at each end of the input and output of the filter before applying Kuroda's identity. During this transformation, the impedance of the stub and an additional unit length microstrip line element is scaled by n^2 (Equation 3).

$$n^2 = 1 + \frac{1}{Z\mathrm{series,\ stub}_{\mathrm{short-circuited}}} \tag{3}$$

$$Z\mathrm{shunt,\ stub}_{\mathrm{open-circuited}} = n^2 \tag{4}$$

$$Z\mathrm{unit,\ 0{,}125\lambda} = n^2 Z\mathrm{stub}_{\mathrm{short-circuited}} \tag{5}$$

The location of the converted open-circuited stub and the added unit length microstrip line element is swapped to complete the filter geometry. Finally, the impedance is scaled by the reference characteristic impedance, 50 Ω.

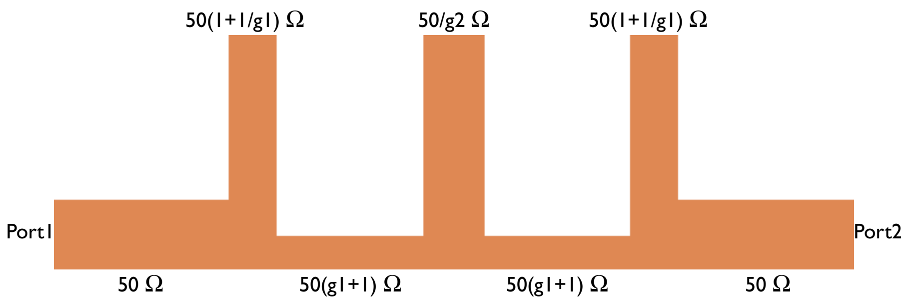


Figure 2: The three element filter design using lumped element prototype element values.

The filter geometry is built with six lines (Bézier polygons) on a two-dimensional space. The properties of each line representing a microstrip line with a different characteristic impedance are configured by Transmission Line Equation features.

The transmission line parameters for a 50 Ω microstrip line built on a 20 mil lossless substrate with permittivity $\varepsilon_r = 3.38$ and 1 oz copper can be calculated accurately from Ref. 2.

TABLE 2: CALCULATED TRANSMISSION LINE PARAMETERS OF A 50 Ω MICROSTRIP LINE.

R	L	G	C
12.41 Ω/m	272.9 nH/m	0 S/m	107.1 pF/m

The contribution of the distributed resistance on the insertion loss with the given substrate properties is less than 0.05 dB. To make the modeling steps simpler in this example, the approximated parameter values in Table 3 are used for a 50 Ω microstrip line.

TABLE 3: SIMPLIFIED TRANSMISSION LINE PARAMETERS OF A 50 Ω MICROSTRIP LINE.

R	L	G	C
0 Ω/m	250 nH/m	0 S/m	100 pF/m

Other transmission line parameters with different characteristic impedance values are adjusted using the normalized impedance. The distributed inductance is proportionally scaled and the distributed capacitance is inversely scaled by the normalized impedance of the microstrip line.

Results and Discussion

The S-parameters, S_{11} and S_{21} of the low-pass filter is plotted in Figure 3. The cutoff is shown at the intended frequency 4 GHz. The ripple of S_{21} is 0.5 dB.

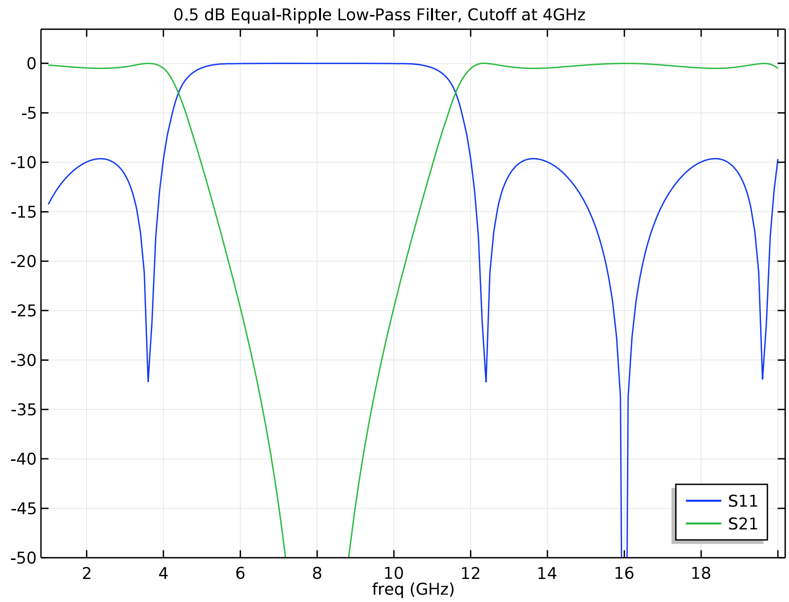


Figure 3: The frequency response of the 0.5dB equal-ripple low-pass filter.

The passband is observed again at the frequency hight 12 GHz. It is a distributed element filter so the frequency response is periodic.

References

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

2. COMSOL Application Gallery, "*Transmission Line Parameter Calculator*", https://www.comsol.com/model/transmission-line-parameter-calculator-22351

Application Library path: `RF_Module/Filters/transmission_line_lpf`

Model Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Radio Frequency>Transmission Line (tl)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `transmission_line_lpf_parameters.txt`.

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**.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
3 Find the **Added segments** subsection. Click **Add Linear**.
4 Find the **Control points** subsection. In row **1**, set **x** to `-ul-0.5`.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `-ul`.

5 In row **2**, set **x** to `ul`.

Bézier Polygon 3 (b3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `ul`.

5 In row **2**, set **x** to `ul+0.5`.

Bézier Polygon 4 (b4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `-ul`.

5 In row **2**, set **x** to `-ul`.

6 In row **2**, set **y** to `ul`.

Bézier Polygon 5 (b5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **y** to `ul`.

Bézier Polygon 6 (b6)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

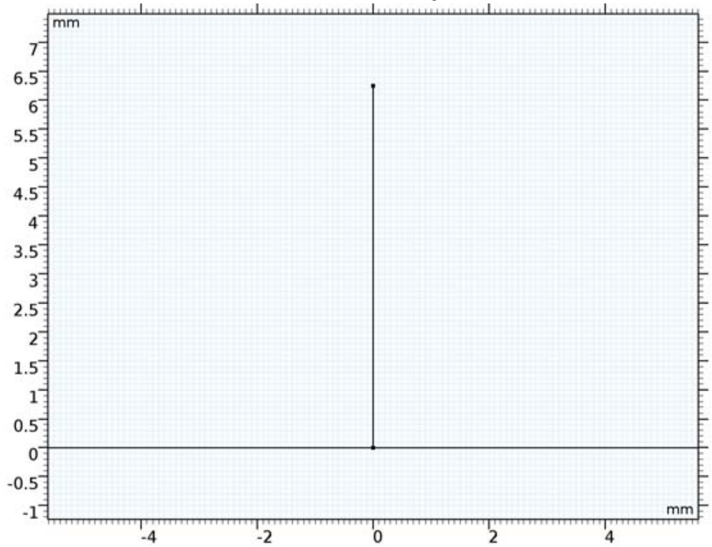
4 Find the **Control points** subsection. In row **1**, set **x** to `ul`.

5 In row **2**, set **x** to `ul`.

6 In row **2**, set **y** to `ul`.

7 On the **Geometry** toolbar, click **Build All**.

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



TRANSMISSION LINE (TL)

Lumped Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 1 only.

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

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

Lumped Port 2

1 In the **Model Builder** window, right-click **Transmission Line (tl)** and choose **Lumped Port**.

2 Select Point 8 only.

Transmission Line Equation 1

Set the input parameters of the transmission line that are configured for 50Ω.

1 In the **Model Builder** window, under **Component 1 (comp1)>Transmission Line (tl)** click **Transmission Line Equation 1**.

2 In the **Settings** window for **Transmission Line Equation**, locate the **Transmission Line Equation** section.

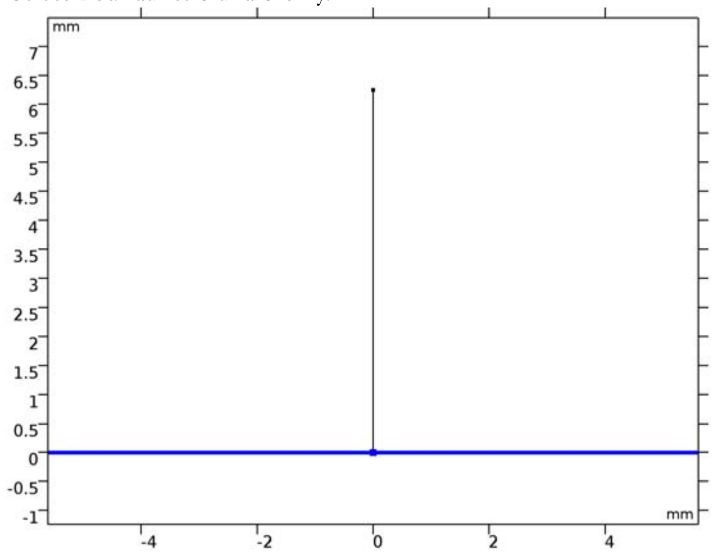
3 In the L text field, type `L0`.

4 In the C text field, type `C0`.

Transmission Line Equation 2

1 In the **Model Builder** window, right-click **Transmission Line (tl)** and choose **Transmission Line Equation**.

2 Select Boundaries 3 and 5 only.



3 In the **Settings** window for **Transmission Line Equation**, locate the **Transmission Line Equation** section.

4 In the L text field, type `L0*z1_1`.

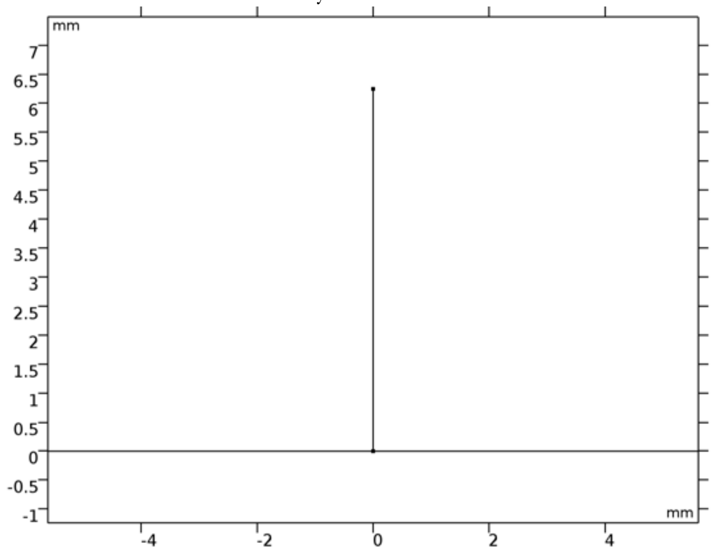
5 In the C text field, type `C0/z1_1`.

The input parameters are scaled by the normalized impedance for 129.82Ω.

Transmission Line Equation 3

1 Right-click **Transmission Line (tl)** and choose **Transmission Line Equation**.

2 Select Boundaries 2 and 6 only.



3 In the **Settings** window for **Transmission Line Equation**, locate the **Transmission Line Equation** section.

4 In the L text field, type `L0*z1_2`.

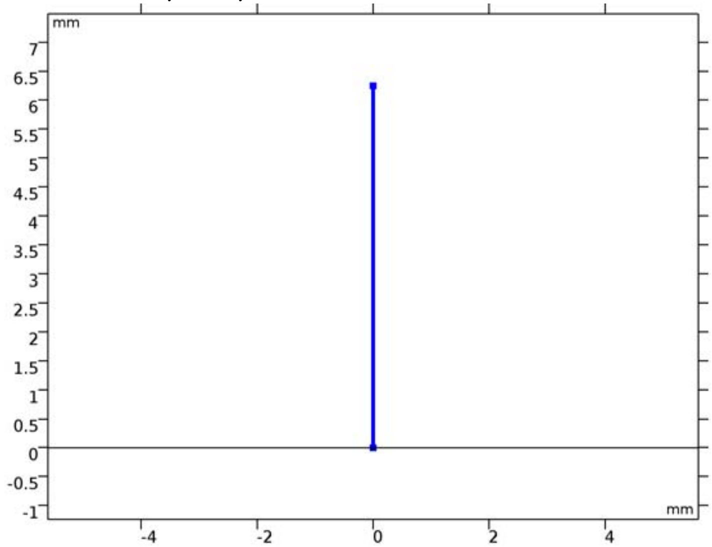
5 In the C text field, type `C0/z1_2`.

The input parameters are scaled by the normalized impedance for 81.32Ω.

Transmission Line Equation 4

1 Right-click **Transmission Line (tl)** and choose **Transmission Line Equation**.

2 Select Boundary 4 only.



3 In the **Settings** window for **Transmission Line Equation**, locate the **Transmission Line Equation** section.

4 In the L text field, type `L0*z2`.

5 In the C text field, type `C0/z2`.

The input parameters are scaled by the normalized impedance for 42.592Ω.

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 In the **Frequencies** text field, type `range(1[GHz],0.1[GHz],20[GHz])`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Line Graph

1 In the **Model Builder** window, expand the **2D Plot Group 1** node, then click **Line Graph**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 From the **Line type** list, choose **Tube**.

4 On the **2D Plot Group 1** toolbar, click **Plot**.

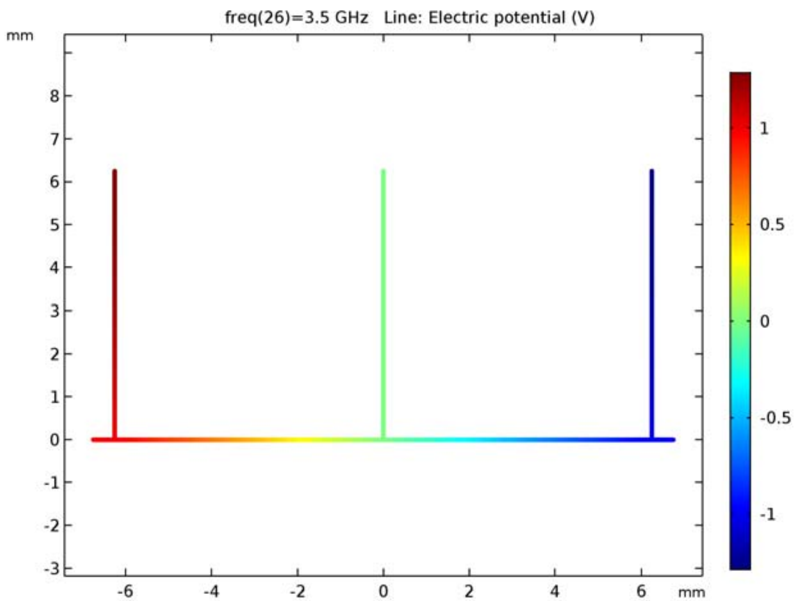
2D Plot Group 1

1 In the **Model Builder** window, under **Results** click **2D Plot Group 1**.

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

3 From the **Parameter value (freq (GHz))** list, choose **3.5**.

4 On the **2D Plot Group 1** toolbar, click **Plot**.



This is the voltage plot at 3.5 GHz that is inside the passband.

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Global**.

3 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>Transmission Line>Ports>S-parameter, dB>tl.S11dB - S11**.

4 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Transmission Line>Ports>S-parameter, dB>tl.S21dB - S21**.

5 On the **1D Plot Group 2** toolbar, click **Plot**.

1D Plot Group 2

1. In the **Model Builder** window, under **Results** click **1D Plot Group 2**.
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 `0.5 dB Equal-Ripple Low-Pass Filter, Cutoff at 4GHz`.
5. Locate the **Axis** section. Select the **Manual axis limits** check box.
6. In the **y minimum** text field, type `-50`.
7. Locate the **Legend** section. From the **Position** list, choose **Lower right**.

   Compare the resulting plot with that shown in Figure 3.

Created in COMSOL Multiphysics 5.3

Tunable Evanescent Mode Cavity Filter Using a Piezo Actuator

Introduction

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

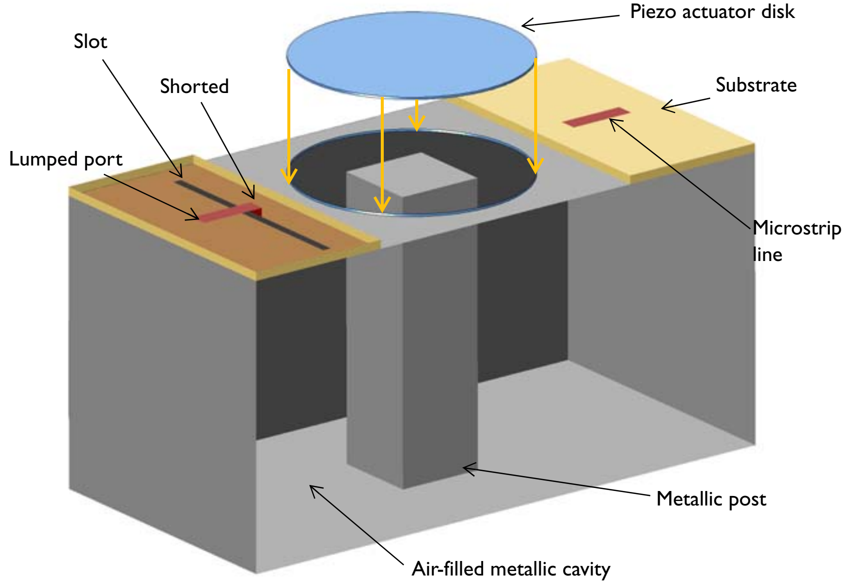


Figure 1: A tunable evanescent mode cavity filter is composed of a rectangular cavity with a metallic post, a piezo actuator disk, and slot-coupled microstrip lines. There is a small gap between the top of the post and the bottom side of the piezo actuator. The front part of the cavity wall is removed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

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

Model Definition

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

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

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

By adding a metallic post and creating reactance inside the cavity, the resonance frequency can be lowered. The cavity is air filled and the height of the post is slightly smaller than b, which creates a small gap between the top of the post and the cavity where the electric fields are confined. Two shorted 50 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 between the top of the post and the bottom of the piezo device, strong electric fields are observed.

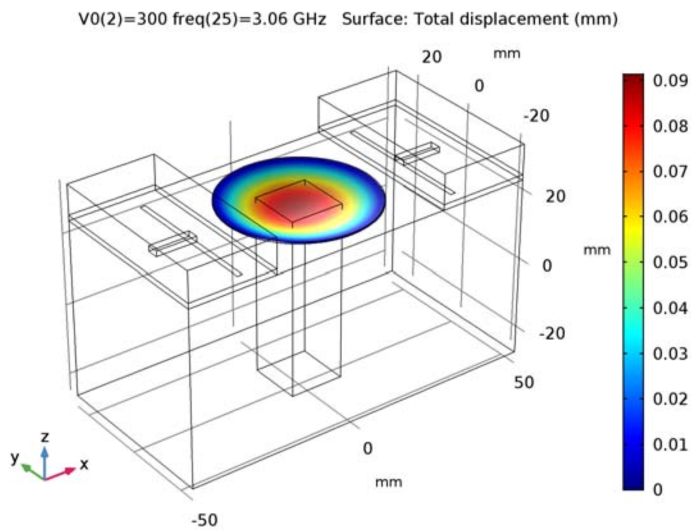


Figure 2: This plot shows the total piezo displacement when 300 V is applied on the actuator. The visualization is exaggerated to emphasize the deflection.

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

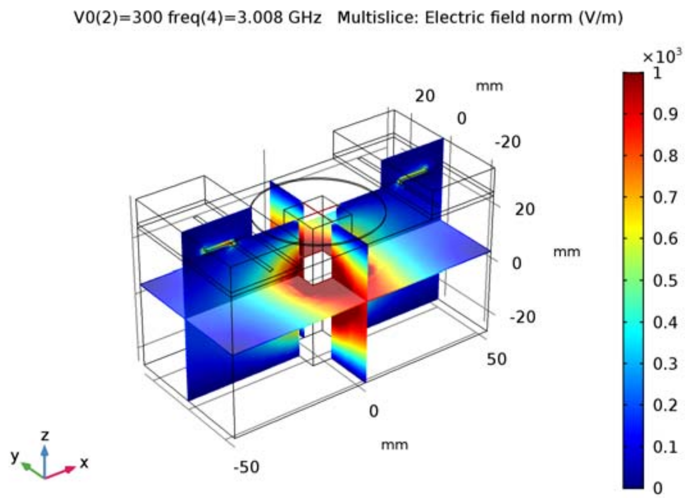


Figure 3: The dominant mode inside the cavity is observed from the electric field distribution plot.

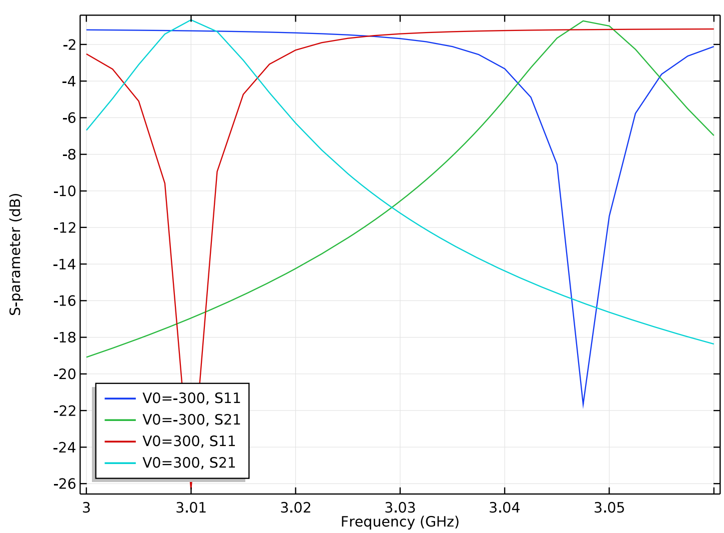


Figure 4: The deflection for the piezo device controlled by the input bias can shift the resonance frequency of the filter.

Notes About the COMSOL Implementation

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

Application Library path: `RF_Module/Filters/tunable_cavity_filter`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **Empty Study**.
10 Click **Done**.

STUDY 1

Step 1: Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

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

3 In the table, clear the **Solve for** check box for the **Electromagnetic Waves, Frequency Domain (emw)** interface.

Step 2: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.

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

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

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for following interfaces:

Physics interface
Solid Mechanics (solid)
Electrostatics (es)
Moving Mesh (ale)

GEOMETRY 1

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

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

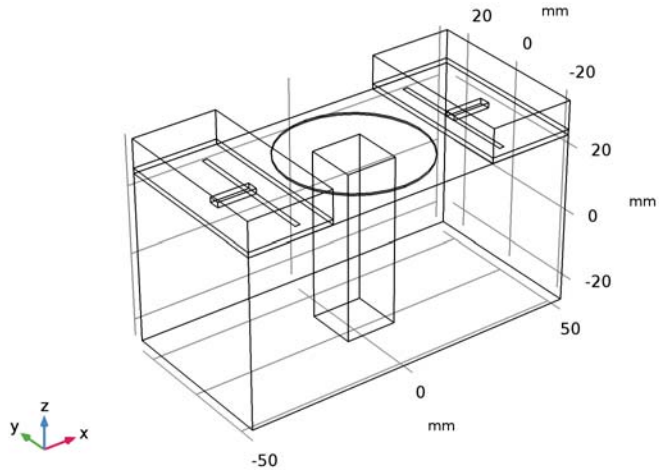
Cavity (blk1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Cavity (blk1)**.

2 In the **Settings** window for **Block**, click **Build All Objects**.

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

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



GLOBAL DEFINITIONS

Parameters

Add parameters that are not related to geometry.

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

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

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

DEFINITIONS

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

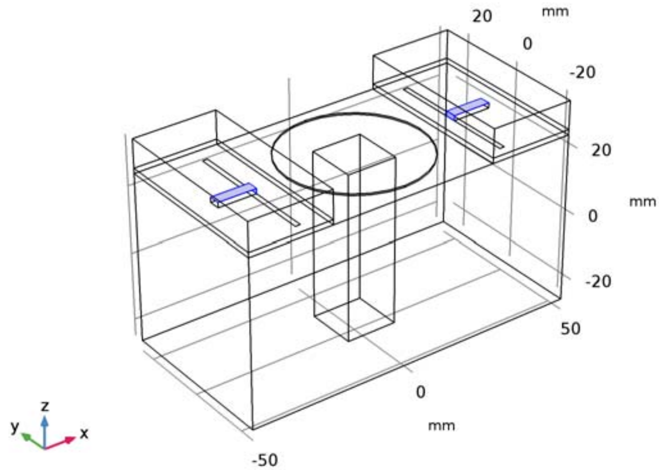
Explicit 1

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

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

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

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



Add a selection for the ground.

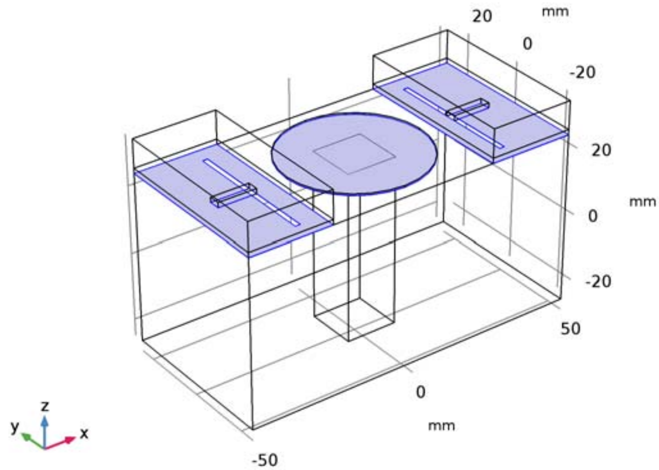
Explicit 2

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

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

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

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



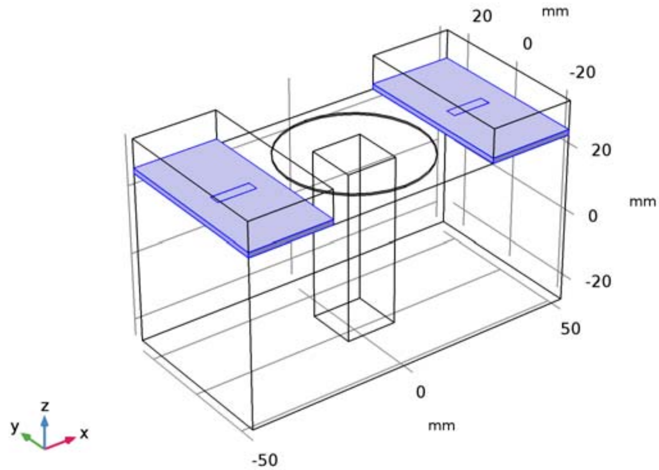
Add a selection for the substrate.

Explicit 3

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

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

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



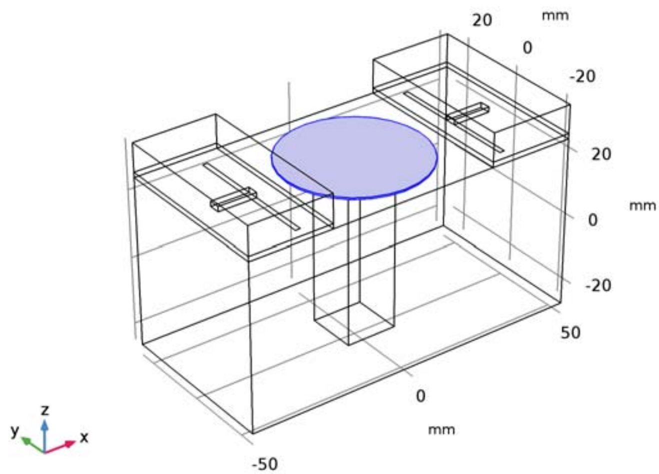
Add a selection for the piezo actuator disk.

Explicit 4

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

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

3 Select Domain 5 only.



Add a selection for fixed edges of the piezo disk.

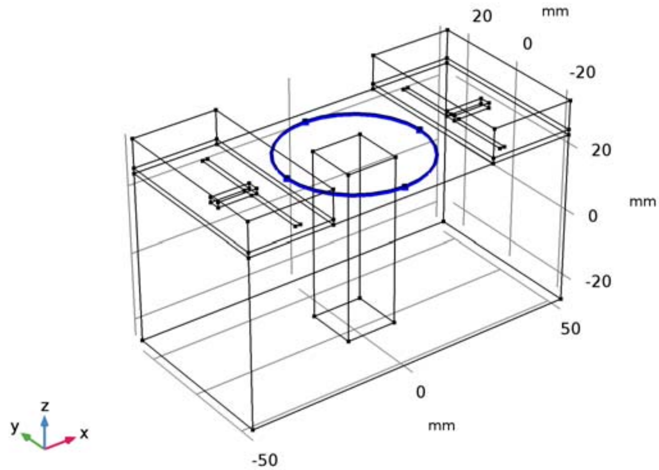
Explicit 5

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

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

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

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



Add a selection for the open boundaries of RF domain.

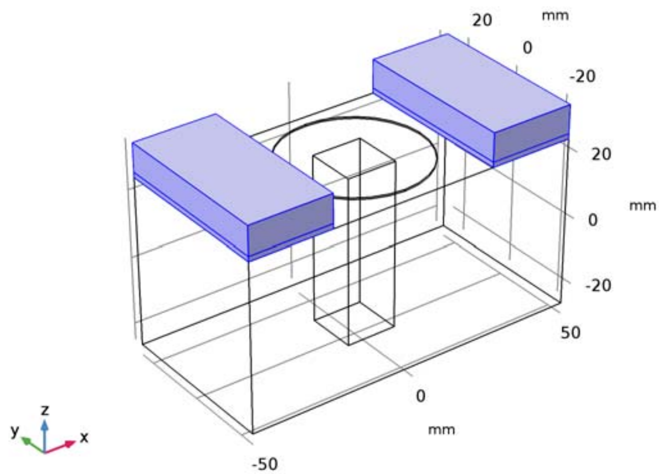
Explicit 6

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

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

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

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



Before defining materials, specify domains for each physics.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Piezo actuator**.

ELECTROSTATICS (ES)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Electrostatics (es)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics (es)**.

2 In the **Settings** window for **Electrostatics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Piezo actuator**.

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 (ALE)

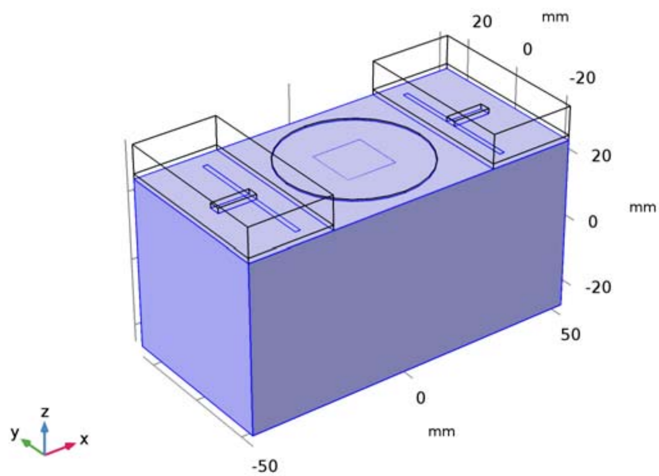
On the **Physics** toolbar, click **Electrostatics (es)** and choose **Moving Mesh (ale)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Moving Mesh (ale)**.

2 In the **Settings** window for **Moving Mesh**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.



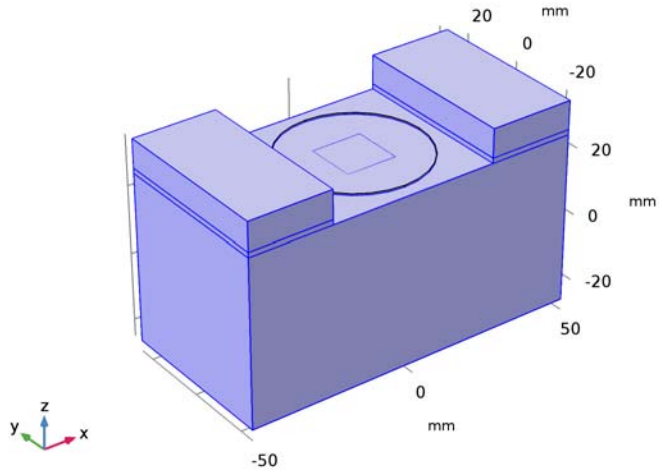
Set up the **Electromagnetic Waves, Frequency Domain** interface. Suppress the piezo actuator disk domain.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

On the **Physics** toolbar, click **Moving Mesh (ale)** and choose **Electromagnetic Waves, Frequency Domain (emw)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 Select Domains 1–4 and 6–8 only.



Assign material properties. Use three materials for this model: PZT-5H, air, and a user-defined substrate.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Lead Zirconate Titanate (PZT-5H) (mat1)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Piezo actuator**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 Click **Add to Component** in the window toolbar.

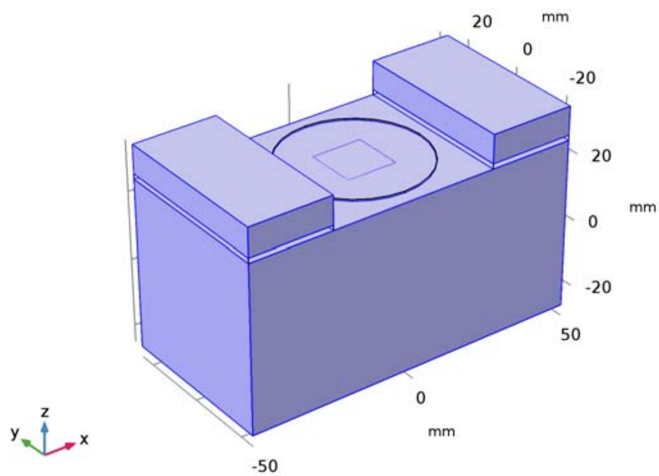
MATERIALS

Air (mat2)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat2)**.

3 Select Domains 1, 3, and 7 only.



Material 3 (mat3)

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 **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Substrate**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

SOLID MECHANICS (SOLID)

Now, set up the physics constraints. Start by assuming the bottom rim of piezo actuator part is attached on the same size circular aperture of the cavity top and no deflection is expected.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node.

Fixed Constraint 1

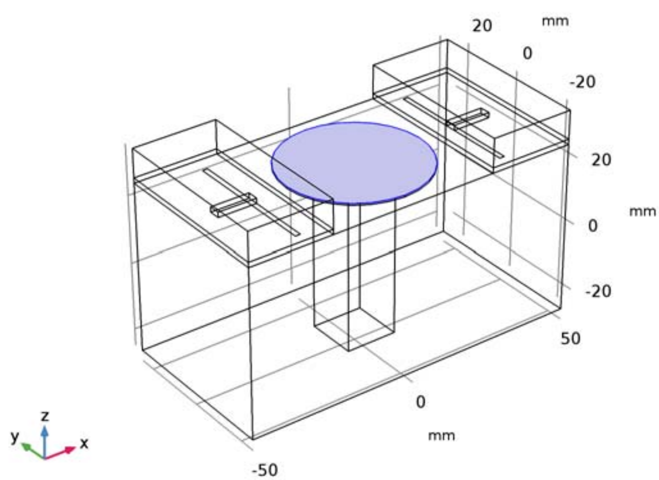
1 Right-click **Solid Mechanics (solid)** and choose **Edges>Fixed Constraint**.
2 In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.
3 From the **Selection** list, choose **Piezo fixed edges**.

ELECTROSTATICS (ES)

Next, define the Electrostatics constraints.

Electric Potential 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose **Electric Potential**.
2 Select Boundary 29 only.
3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.
4 In the V_0 text field, type `-V0`.



Ground 1

1. In the **Model Builder** window, right-click **Electrostatics (es)** and choose **Ground**.
2. In the **Settings** window for **Ground**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Ground**.

MOVING MESH (ALE)

Prescribed Mesh Displacement 1

1. In the **Model Builder** window, expand the **Moving Mesh (ale)** node, then click **Prescribed Mesh Displacement 1**.
2. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.
3. In the d_x text field, type `u`.
4. In the d_y text field, type `v`.
5. In the d_z text field, type `w`.

Free Deformation 1

1. In the **Model Builder** window, right-click **Moving Mesh (ale)** and choose **Free Deformation**.
2. Select Domain 1 only.

Prescribed Mesh Displacement 2

1. Right-click **Moving Mesh (ale)** and choose **Prescribed Mesh Displacement**.
2. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type `1-27,29-57` in the **Selection** text field.
5. Click **OK**.

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. In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Feed line**.

Perfect Electric Conductor 3

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Perfect Electric Conductor**.
2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Ground**.

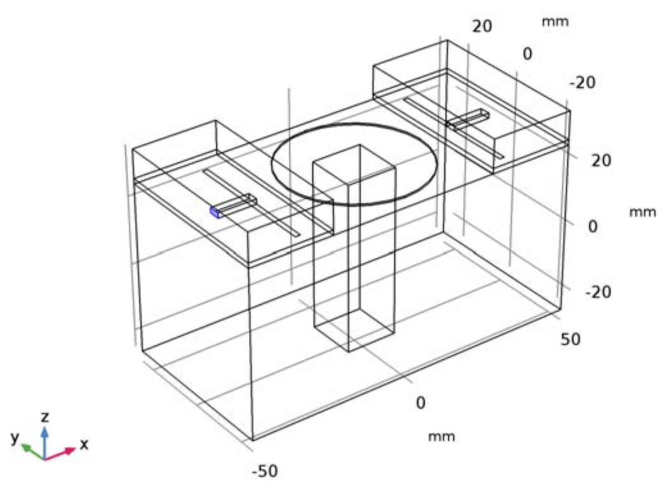
Scattering Boundary Condition 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Scattering Boundary Condition**.
2 In the **Settings** window for **Scattering Boundary Condition**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Open boundaries**.

Lumped Port 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.
2 Select Boundary 14 only.

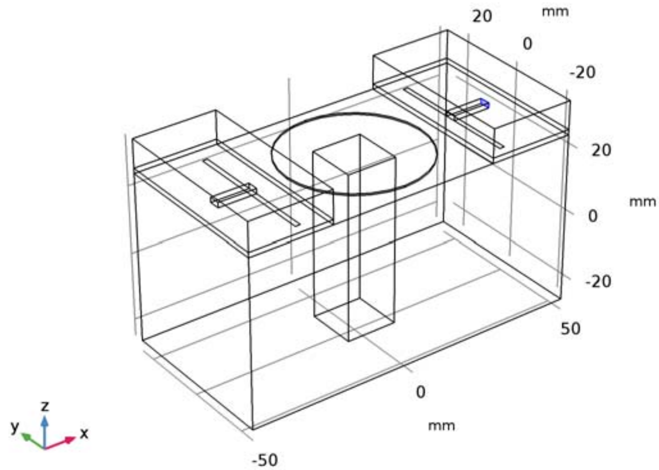
   For the first port, wave excitation is **on** by default.



Lumped Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 54 only.



MESH 1

Adjust the maximum and minimum element size manually on the piezo actuator to reduce the memory cost of running the study without compromising the accuracy. Then, use swept mesh for the piezo actuator to handle the structural deformation more efficiently.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.
3. From the **Sequence type** list, choose **User-controlled mesh**.

Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size 1**.
2. In the **Settings** window for **Size**, locate the **Element Size Parameters** section.
3. In the **Maximum element size** text field, type 10.
4. In the **Minimum element size** text field, type 1.

Free Tetrahedral 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
2. In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1–4 and 6–8 only.

Select all domains except for the piezo actuator.

Distribution 1

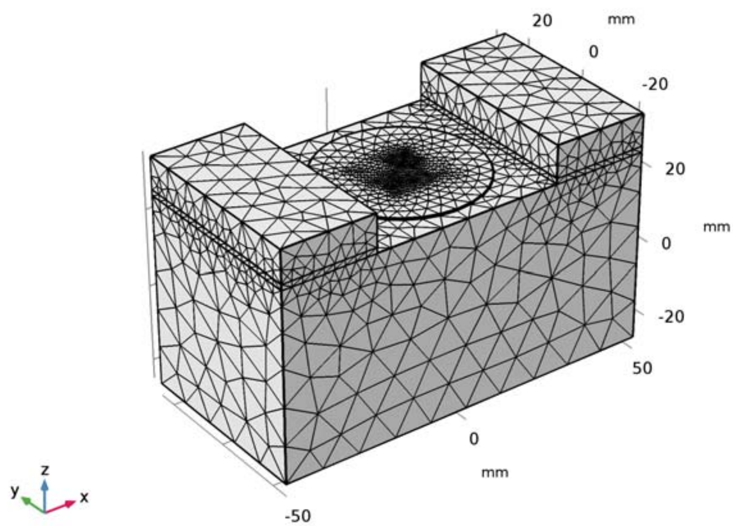
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 2.

5 Click **Build All**.



STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0	-300,300	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

Replace the default stress plot by displacement plot.

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement** node, then click **Surface 1**.

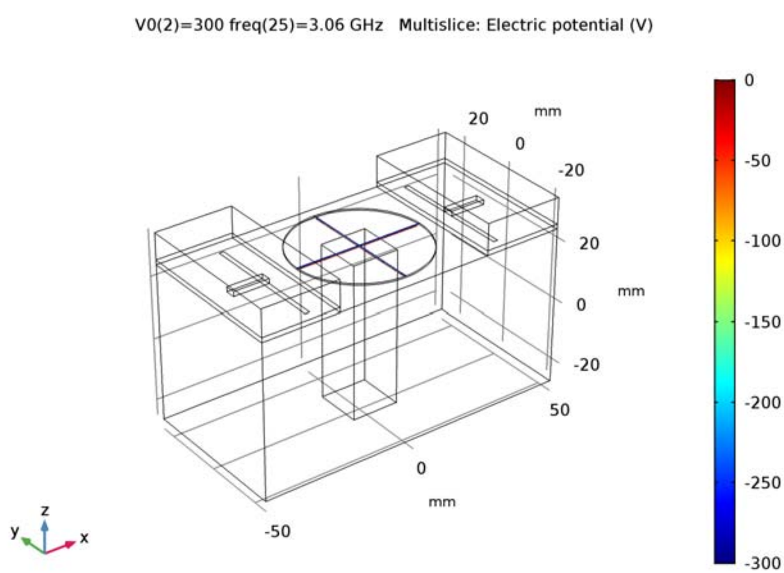
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>solid.disp - Total displacement**.

3 On the **Displacement** toolbar, click **Plot**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

This plot shows the deflected piezo actuator disk; compare with Figure 2.

Electric Potential (es)



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 **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Parameter value (freq (GHz))** list, choose **3.008**.
4 On the **Electric Field (emw)** toolbar, click **Plot**.

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
2 In the **Settings** window for **Multislice**, click to expand the **Range** section.
3 Select the **Manual color range** check box.
4 In the **Maximum** text field, type 1000.
5 On the **Electric Field (emw)** toolbar, click **Plot**.

   The resulting plot shows strong electric fields resulting from the dominant resonance at the center of the cavity as well as in the gap between the metallic post and the ceiling of the cavity.Compare the plot with that shown in Figure 3.

S-Parameter (emw)

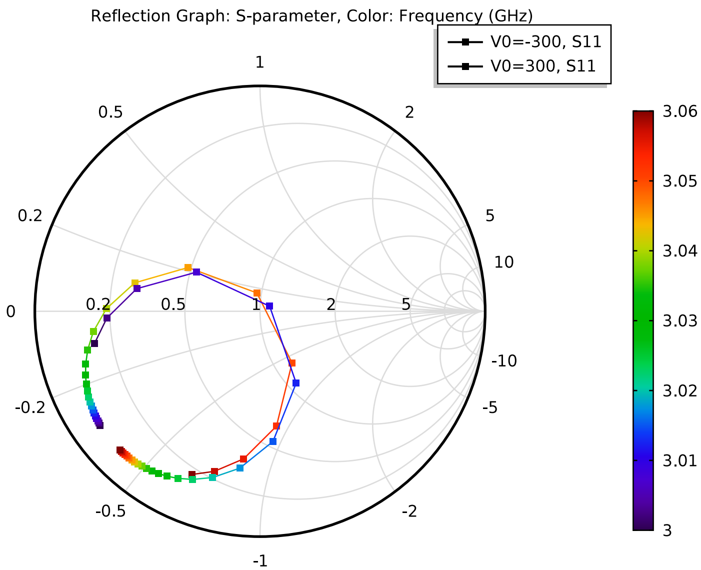
1 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.
2 From the **Position** list, choose **Lower left**.

Global 1

1 In the **Model Builder** window, expand the **S-Parameter (emw)** node, then click **Global 1**.
2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.
3 From the **Unit** list, choose **GHz**.
4 On the **S-Parameter (emw)** toolbar, click **Plot**.

   The plotted S-parameters show the frequency shift as a function of the input bias on the piezo actuator; compare with Figure 4.

Smith Plot (emw)



Appendix: Geometry Modeling Instructions

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

GLOBAL DEFINITIONS

Load geometrical parameters from a file.

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `tunable_cavity_filter_parameters.txt`.

GEOMETRY 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**.

First, create a block for the cavity.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Cavity` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `100`.

4 In the **Depth** text field, type `50`.

5 In the **Height** text field, type `50`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Add a substrate block.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Substrate` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `25`.

4 In the **Depth** text field, type `50`.

5 In the **Height** text field, type `thickness`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `-37.5`.

8 In the **z** text field, type `25+thickness/2`.

Add a block for the air domain.

Block 3 (blk3)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Air block` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `25`.

4 In the **Depth** text field, type `50`.

5 In the **Height** text field, type `10`.

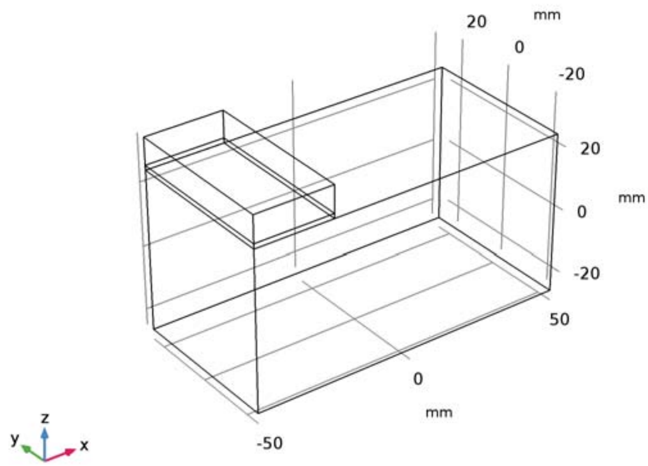
6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `-37.5`.

8 In the **z** text field, type `30`.

9 Right-click **Air block** and choose **Build Selected**.

10 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



Add a block for the microstrip line feed.

Block 4 (blk4)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Feed_line` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `l_feed+w_slot`.

4 In the **Depth** text field, type `3.2`.

5 In the **Height** text field, type `thickness`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `-x_slot-l_feed/2`.

8 In the **z** text field, type `25+thickness/2`.

Add a work plane where you will draw a slot.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `25`.

4 Click **Show Work Plane**.

Plane Geometry

Add a rectangle for the slot.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `w_slot`.

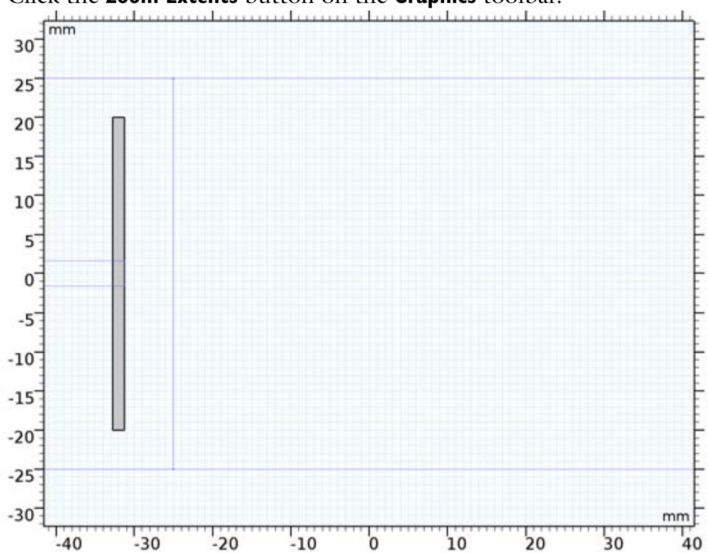
4 In the **Height** text field, type `l_slot`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **xw** text field, type `-x_slot`.

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.



9 In the **Model Builder** window, click **Geometry 1**.

Generate the 2nd slot coupled microstrip line by mirroring some geometries.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **wp1**, **blk3**, **blk2**, and **blk4** 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**.

Add a block for the metal post in the middle of the cavity.

Block 5 (blk5)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Post` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `15`.

4 In the **Depth** text field, type `15`.

5 In the **Height** text field, type `50-gap_post`.

6 Locate the **Position** section. In the **x** text field, type `-7.5`.

7 In the **y** text field, type `-7.5`.

8 In the **z** text field, type `-25`.

9 Right-click **Post** and choose **Build Selected**.

Add a cylinder for the piezo actuator disk.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `Piezo actuator` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `21`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **z** text field, type `25`.

6 Right-click **Piezo actuator** and choose **Build Selected**.

The inside of the metal post is not part of the modeling domain. Therefore, subtract it from the cavity.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **blk1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **blk5** only.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.3

Numerical Modeling of a UHF RFID Tag

Introduction

UHF RFID tags are widely used for identifying and tracking animals. This example simulates a passive radio-frequency identification (RFID) tag for the UHF frequency range.

With respect to the chip transponder's complex impedance, a reflection coefficient is computed. This is done using an approach that differs from the conventional scattering parameter analysis method by a real reference impedance value.

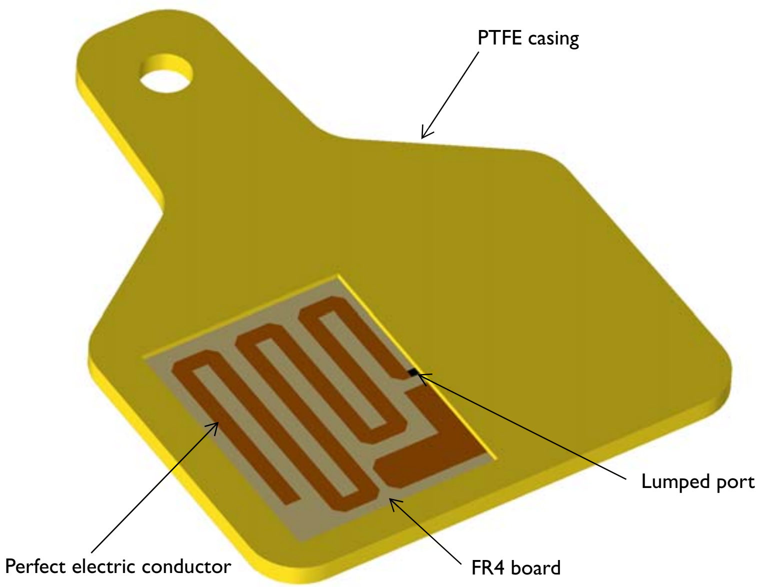


Figure 1: The RFID tag's geometry consists of copper traces patterned on an FR4 board that is enclosed by a low dielectric PTFE case. The surrounding air domain and perfectly matched layers, which are required for the simulation, are not included in this figure.

Model Definition

In this example, the RFID tag's operating frequency is 915 MHz. At this frequency, the metal part of the RFID tag can be modeled as a perfect electrical conductor (PEC), because while the copper traces patterned on the FR4 board are geometrically very thin, they are much thicker than the skin depth.

The entire circuit board is inserted inside a lossless PTFE casing. The tag is modeled in a spherical air domain, which is enclosed by perfectly matched layers (PML) that absorb all outgoing radiation from the tag.

A lumped port with a reference impedance of 50 Ω is used on the location of an RFID chip. This is done to excite the tag and evaluate the input impedance of the tag's antenna part, which is modeled as a meander line. An additional copper strip is placed adjacent to the meander line to control the impedance.

The conventional S-parameter works well only with a real reference impedance. However, the RFID chip's impedance is complex and the calculated S-parameter is not physical when a complex port reference impedance is used.

In Ref. 1, the power wave reflection coefficient term is introduced. It is applicable for evaluating the matching properties of an RFID tag:

$$\Gamma = \frac{Z_l - Z_{\mathrm{ref}}^*}{Z_l + Z_{\mathrm{ref}}}$$

where Z_l is the complex load impedance and Z_{ref} is the complex reference impedance.

Results and Discussion

Figure 2 shows the default E-field norm on the *xy*-plane. The field distribution plot indicates that the electric field is symmetrically confined along the meander line, as well as in the area between the meander line and impedance matching strip.

The far-field radiation pattern of the tag is shown in Figure 3. Noticeably, the tag's radiation pattern looks very similar to the radiation pattern of a half-wave dipole antenna.

The evaluated impedance of the tag is around $18+j124$ Ω and the power wave reflection coefficient, in dB, is below −15 dB.

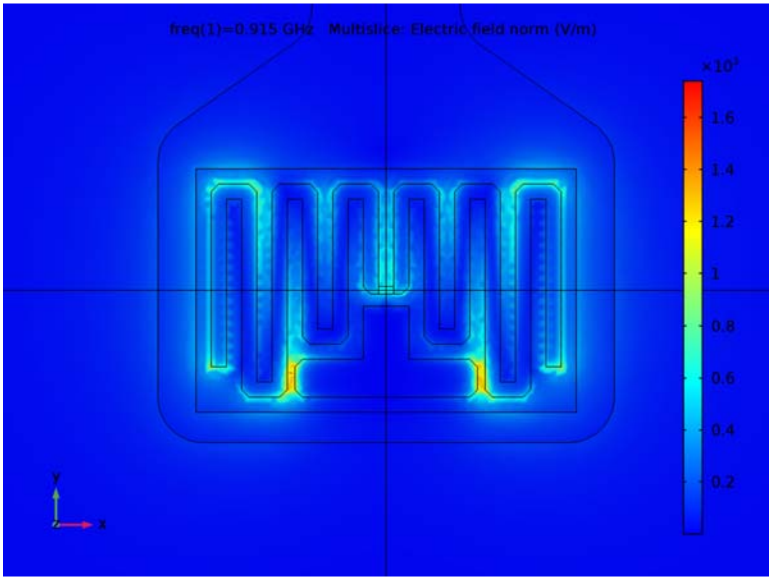


Figure 2: The E-field norm plot shows where the field is strongly confined in the tag.

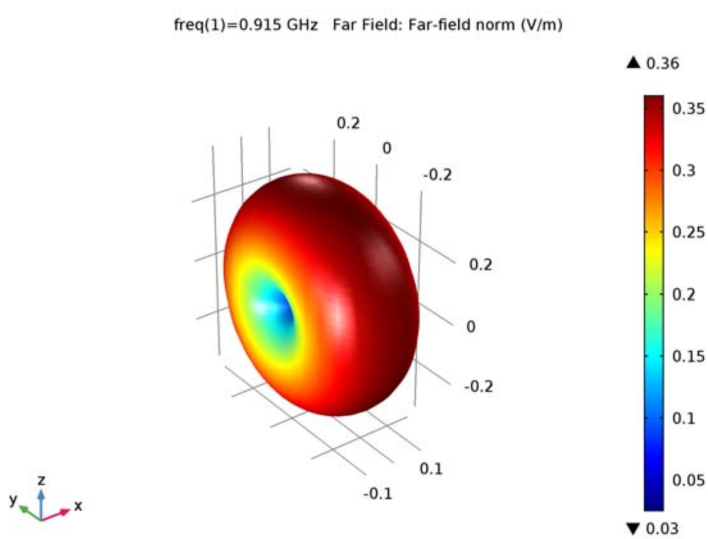


Figure 3: The far-field radiation pattern resembles that of a half-wave dipole antenna.

Reference

1. K. Kurokawa, “Power Waves and the Scattering Matrix,” *IEEE Transactions on Microwave Theory and Techniques*, Volume 13, 1965.

Model Library path: `RF_Module/Antennas/uhf_rfid_tag`

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 **Preset 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. In the **Frequencies** text field, type `915[MHz]`.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`Zc`	`15-j*125[ohm]`	(15-125i) Ω	`Chip impedance`

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**.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

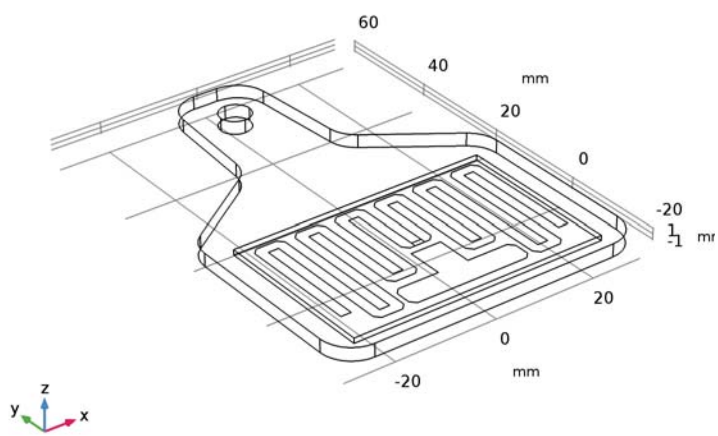
2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `uhf_rfid_tag.mphbin`.

5 Click **Import**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



Add a sphere for the air domain surrounding the RFID tag and perfectly matched layers that will be configured later on.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

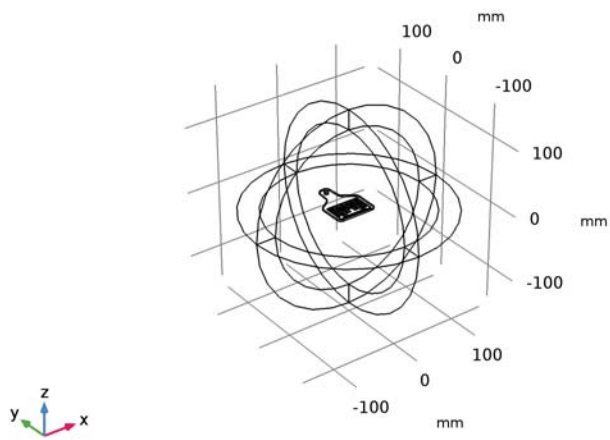
3 In the **Radius** text field, type 150.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	30

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.



DEFINITIONS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

Define a variable for calculating the reflection coefficient between two complex impedances.

Variables 1

1 In the **Settings** window for **Variables**, locate the **Variables** section.

2 In the table, enter the following settings:

Name	Expression	Unit	Description
`Gamma`	`(emw.Zport_1-conj(Zc))/(emw.Zport_1+Zc)`		`Reflection coefficient for complex impedance matching`

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1–4 and 9–12 only.

These are all of the outermost domains of the sphere.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

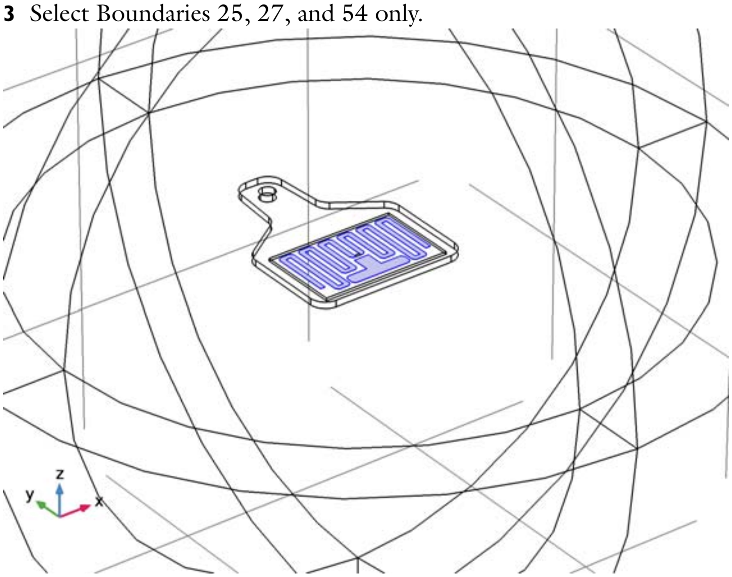
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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 Click the **Zoom In** button on the **Graphics** toolbar, a couple of times to get a clear view of the RFID tag.

3 Select Boundaries 25, 27, and 54 only.



Lumped Port 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 35 only.

For the first port, wave excitation is **on** by default.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

4 Click the **Zoom In** button on the **Graphics** toolbar.

5 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

ADD MATERIAL

1 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>FR4 (Circuit Board)**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat2)**.

2 Select Domain 7 only.

3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 3 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Select Domain 6 only.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	2.1	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.

To see the meshed structure of the device, remove some boundaries from the view.

DEFINITIONS

Hide for Physics 1

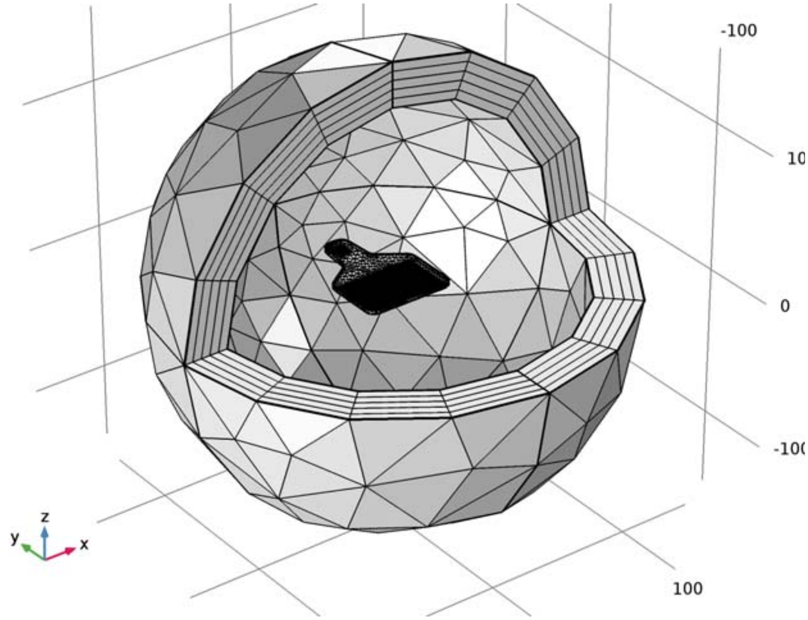
1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **View 1** and choose **Hide for Physics**.

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 6, 10, 16, 37, 40, and 42 only.

MESH 1



STUDY 1

Step 1: Frequency Domain

On the **Home** toolbar, click **Compute**.

RESULTS

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 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

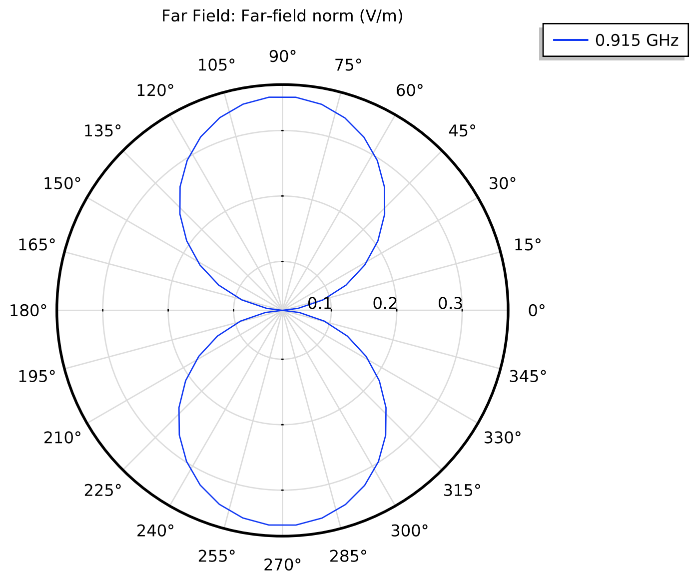
6 On the **Electric Field (emw)** toolbar, click **Plot**.

7 Click the **Go to XY View** button on the **Graphics** toolbar.

   Zoom in a couple of time to get a good view of the RFID tag.

   Compare the reproduced plot with Figure 2.

2D Far Field (emw)



The E-plane radiation pattern resembles that of a dipole antenna.

Far Field 1

1. In the **Model Builder** window, expand the **3D Far Field (emw)** node, then click **Far Field 1**.
2. In the **Settings** window for **Far Field**, locate the **Evaluation** section.
3. Find the **Angles** subsection. In the **Number of azimuth angles** text field, type 40.
4. On the **3D Far Field (emw)** toolbar, click **Plot**.

   Reproduce Figure 3.

Global Evaluation 2

1. On the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1> Electromagnetic Waves, Frequency Domain>Ports>emw.Zport_1 - Lumped port impedance**.
3. Click **Evaluate**.

Global Evaluation 3

1. On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(Gamma))`		

4 Click **Evaluate**.

Vivaldi Antenna

Introduction

A tapered slot antenna, also known as a Vivaldi antenna, is useful for wide-band applications. Here, an exponential function is used for the taper profile. The objective of this example is to compute the far-field pattern and to compute the impedance of the structure. Good matching is observed over a wide frequency band.

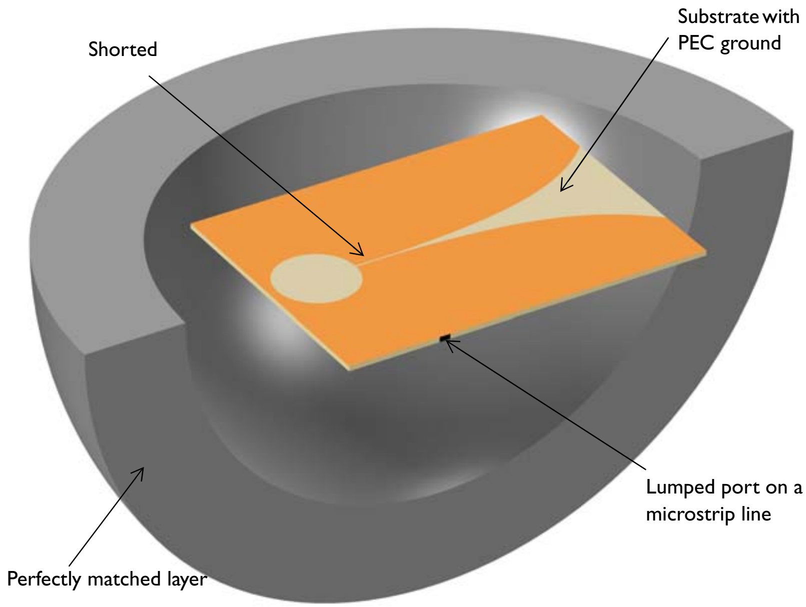


Figure 1: The Vivaldi antenna is realized on a thin dielectric substrate. The entire domain is bounded by a perfectly matched layer.

Model Definition

In this Vivaldi antenna model, the tapered slot is patterned with a perfect electric conductor (PEC) ground plane on the top of the dielectric substrate. A simple exponential function, $e^{0.044x}$ is used to create the tapered slot curves. One end of the slot is open to air and the other end is finished with a circular slot. On the bottom of the substrate, the shorted 50 Ω microstrip feed line is modeled as PEC surfaces. The entire modeling domain is bounded by a perfectly matched layer (PML) which acts like an anechoic chamber absorbing all radiated energy. To excite the antenna, a lumped port is used. The model is meshed using a tetrahedral mesh with approximately five elements per wavelength in each material and simulation frequency.

Results and Discussion

The simulated SWR plot, Figure 2, shows good wide-band matching properties. A Vivaldi antenna utilizes traveling waves generating a directive radiation pattern toward the open end of the tapered slot. The 3D far-field pattern in Figure 3 shows a directive radiation pattern.

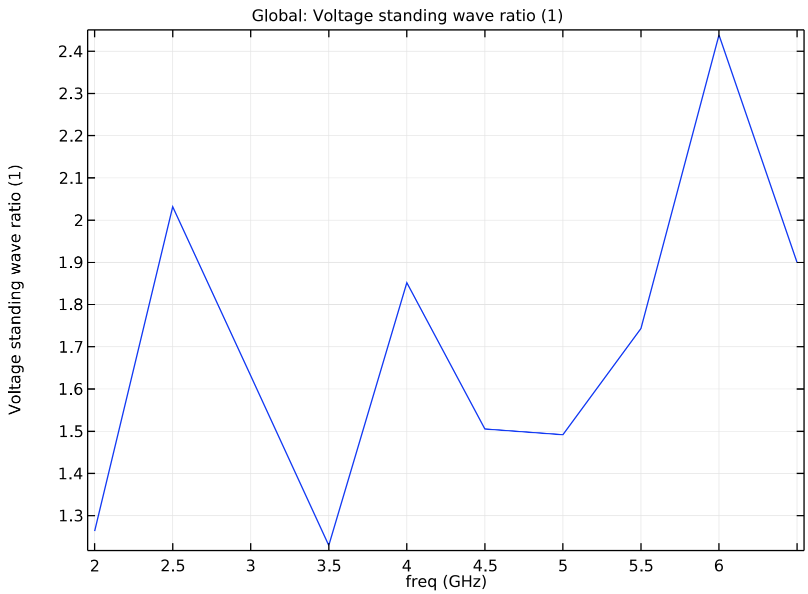


Figure 2: The frequency response SWR of the Vivaldi antenna shows wide-band impedance matching, better than 2:1 in most of the simulated frequency range.

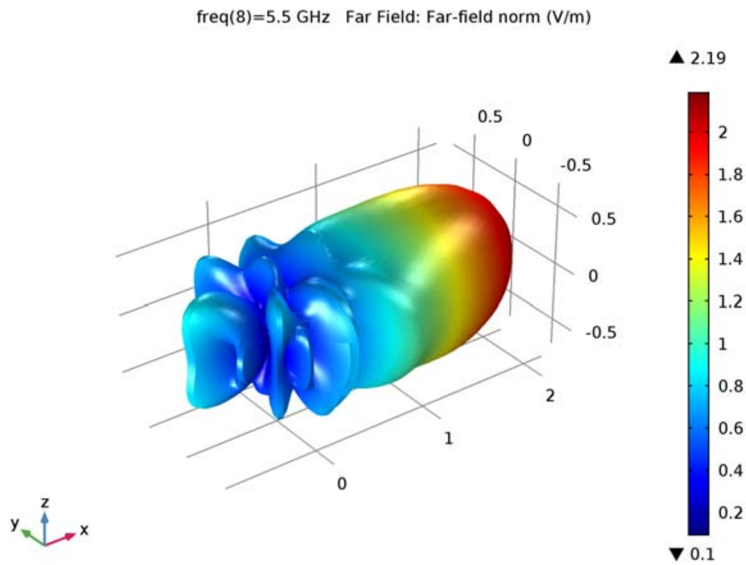


Figure 3: 3D far-field pattern at 5.5 GHz shows a directional radiation pattern.

References

1. D.M. Pozar, *Microwave Engineering*, John Wiley & Sons, 1998.
2. C.A. Balanis, *Antenna Theory*, John Wiley & Sons, 1997.

Application Library path: `RF_Module/Antennas/vivaldi_antenna`

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 **Preset 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 In the **Frequencies** text field, type `range(2[GHz],0.5[GHz],6.5[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**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
thickness	60[mil]	0.001524 m	Substrate thickness
w_slot	0.5[mm]	5E-4 m	Slot with

Here, `mil` refers to the unit milliinch.

GEOMETRY 1

Create a block for the antenna substrate.

Block 1 (blk1)

1 On 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 `110`.

4 In the **Depth** text field, type `80`.

5 In the **Height** text field, type `thickness`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Next, add a block for the 50Ω microstrip feed line.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Feed line` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `3.2`.

4 In the **Depth** text field, type `40+w_slot/2`.

5 In the **Height** text field, type `thickness`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `-26`.

8 In the **y** text field, type `-20+w_slot/4`.

Next, create a work plane where you will draw the Vivaldi antenna pattern. Use two parametric curves for the tapered slot.

Work Plane 1 (wp1)

1 On 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**.

Plane Geometry

Click the **Zoom Extents** button on the **Graphics** toolbar.

Add a parametric curve using the exponential profile.

Parametric Curve 1 (pc1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `70`.

4 Locate the **Expressions** section. In the **xw** text field, type `s-15`.

5 In the **yw** text field, type `exp(0.044*s)-1+w_slot/2`.

Generate the other parametric curve by mirroring the first one.

Mirror 1 (mir1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 In the **Settings** window for **Mirror**, locate the **Normal Vector to Line of Reflection** section.

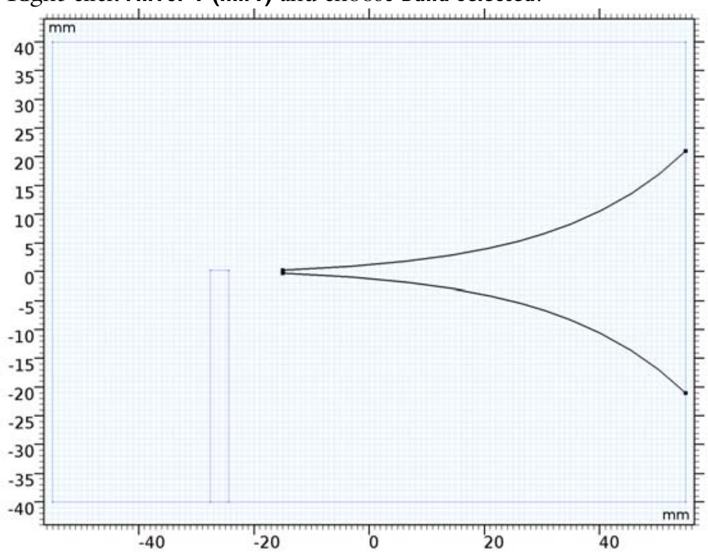
3 In the **yw** text field, type `1`.

4 In the **xw** text field, type `0`.

5 Locate the **Input** section. Select the **Keep input objects** check box.

6 Select the object **pc1** only.

7 Right-click **Mirror 1 (mir1)** and choose **Build Selected**.



Add a rectangle describing the thin slot connected to the tapered slot.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20`.

4 In the **Height** text field, type `w_slot`.

5 Locate the **Position** section. In the **xw** text field, type `-35`.

6 In the **yw** text field, type `-w_slot/2`.

Add a circle attached to the end of the slot.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `12`.

4 Locate the **Position** section. In the **xw** text field, type `-40.5`.

Create a union of the circle and the rectangle to remove unnecessary boundaries.

Union 1 (uni1)

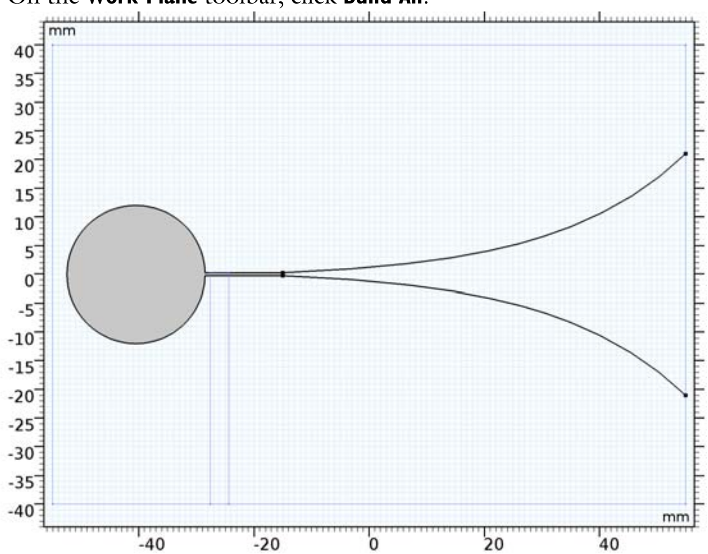
1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **r1** and **c1** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 On the **Work Plane** toolbar, click **Build All**.



6 In the **Model Builder** window, click **Geometry 1**.

Add a sphere for the PMLs. Use a layer definition to create a shell-type structure.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, type `PML` in the **Label** text field.

3 Locate the **Size** section. In the **Radius** text field, type `110`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

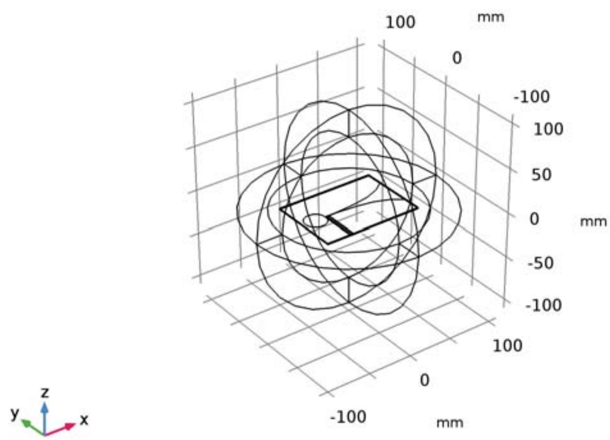
Layer name	Thickness (mm)
`Layer 1`	30

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Choose wireframe rendering to get a better view of the interior parts.

7 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



DEFINITIONS

Add a perfectly matched layer.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1–4 and 8–11 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

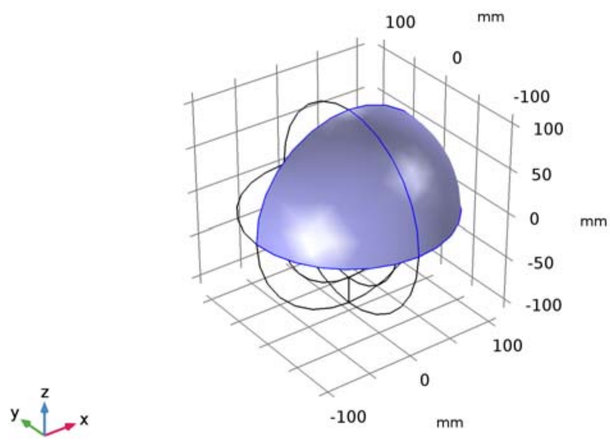
View 1

Hide some domains to get a better view of the interior parts when setting up the physics and reviewing the mesh.

Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **View 1** and choose **Hide for Physics**.

2 Select Domains 2 and 9 only.



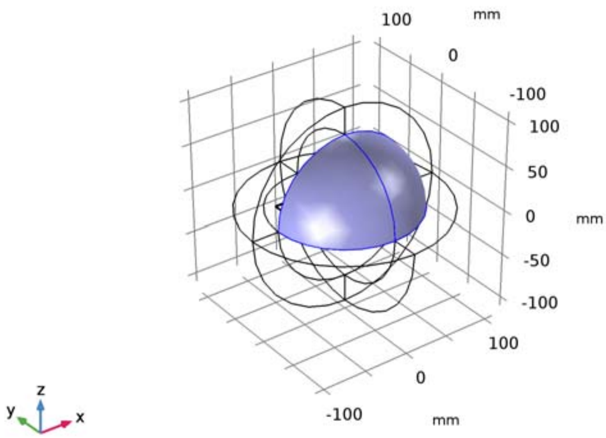
Hide for Physics 2

1 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 10 and 36 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. Use the selections already defined when assigning boundary conditions.

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 16, 21, 22, 24, and 27 only.

Lumped Port 1

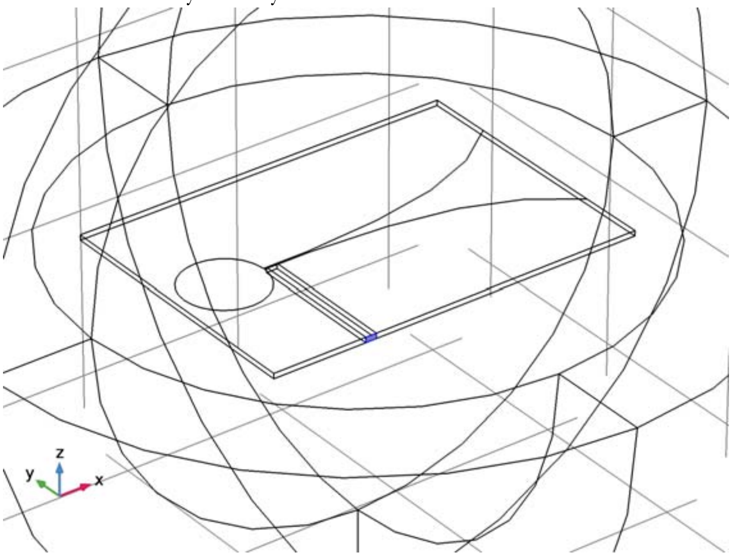
1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Far-Field Domain**.

2 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

3 Click the **Zoom In** button on the **Graphics** toolbar.

4 Click the **Zoom In** button on the **Graphics** toolbar.

5 Select Boundary 20 only.



For the first port, wave excitation is **on** by default.

MATERIALS

Assign material properties for the model. First, use air for all domains.

ADD MATERIAL

1 On 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.

MATERIALS

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Air (mat1)

Override the substrate with a dielectric material of $\varepsilon_r = 3.38$.

Material 2 (mat2)

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 6 and 7 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

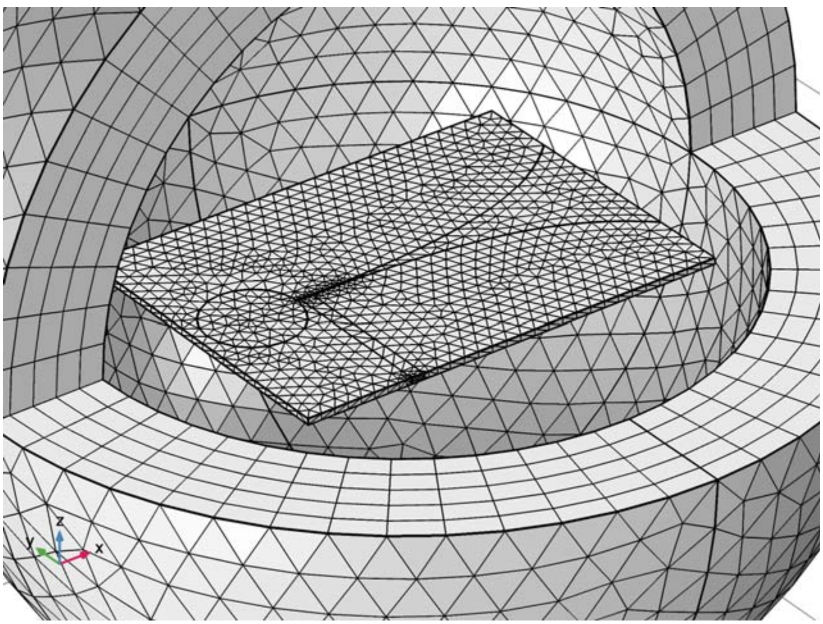
MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Coarse**.

4 Click **Build All**.



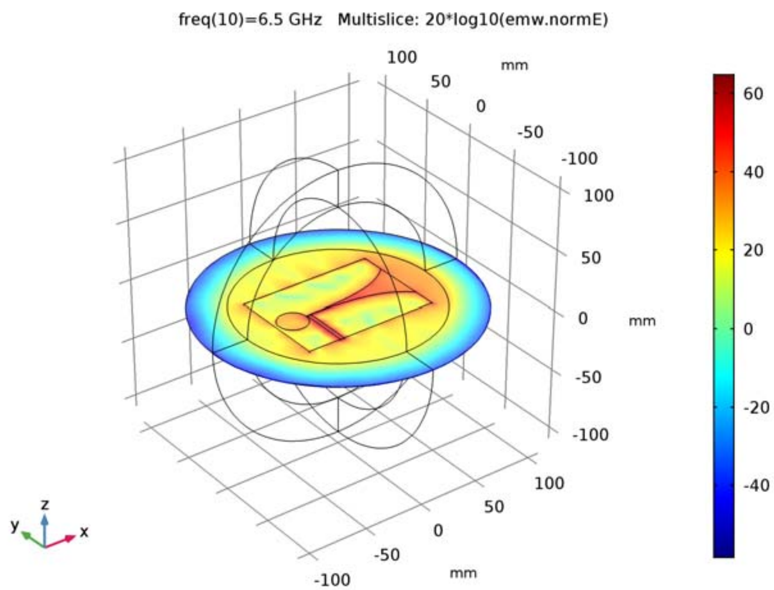
STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

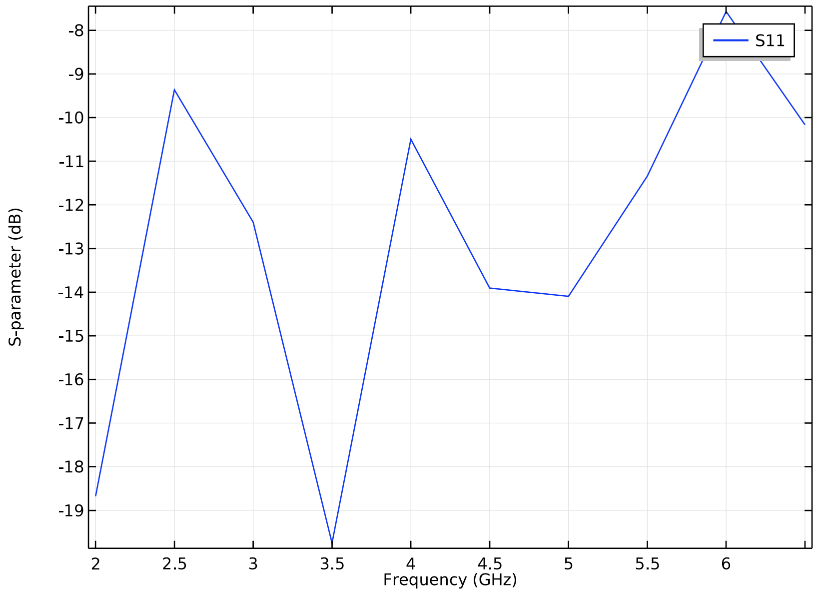
Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
2 In the **Settings** window for **Multislice**, locate the **Expression** section.
3 In the **Expression** text field, type `20*log10(emw.normE)`.
4 Locate the **Multiplane Data** section. Find the **X-planes** subsection. In the **Planes** text field, type `0`.
5 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.
6 On the **Electric Field (emw)** toolbar, click **Plot**.

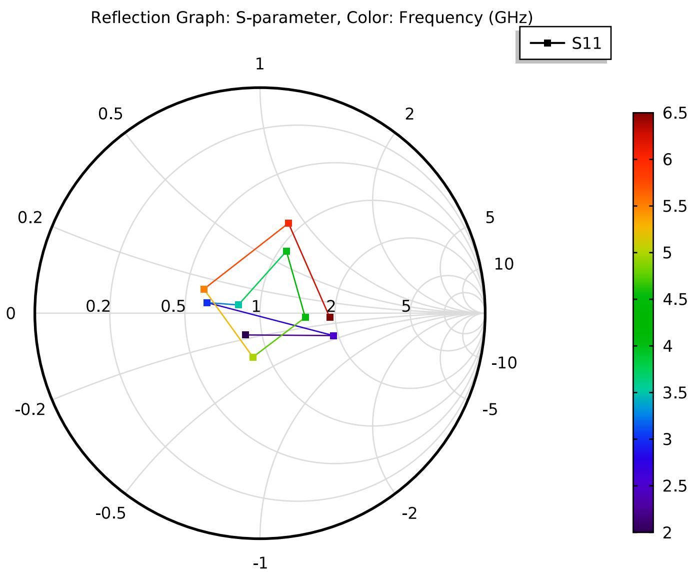


Strong electric fields are observed in the slot and microstrip line.

S-Parameter (emw)



Smith Plot (emw)



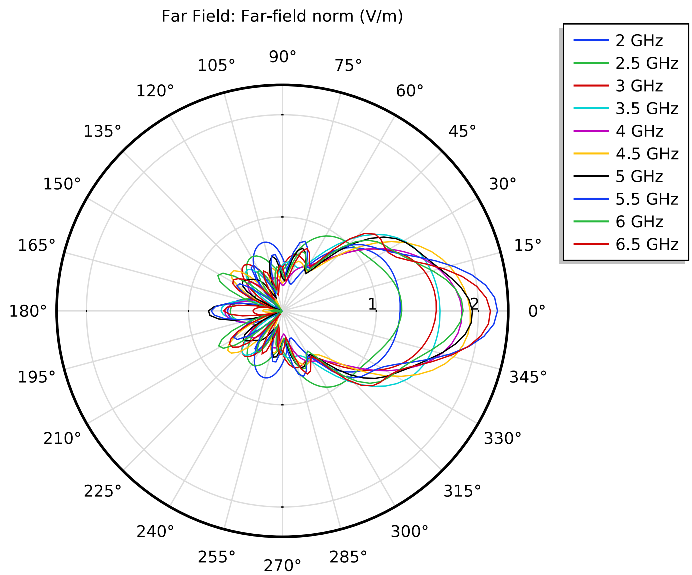
Far Field 1

1 In the **Model Builder** window, expand the **Results>2D Far Field (emw)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type 100.

4 On the **2D Far Field (emw)** toolbar, click **Plot**.



2D far-field radiation patterns in the /[xy/]-plane plotted for all frequencies.

3D Far Field (emw)

1 In the **Model Builder** window, under **Results** click **3D Far Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (GHz))** list, choose **5.5**.

Far Field 1

1 In the **Model Builder** window, expand the **3D Far Field (emw)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of elevation angles** text field, type 90.

4 In the **Number of azimuth angles** text field, type 90.

5 On the **3D Far Field (emw)** toolbar, click **Plot**.

Compare the resulting 3D radiation pattern plot with Figure 3.

Global 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Global**.

3 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>Electromagnetic Waves, Frequency Domain>Ports>emw.VSWR_1 - Voltage standing wave ratio**.

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 On the **1D Plot Group 6** toolbar, click **Plot**.

This VSWR plot replicates the wide-band frequency response shown in Figure 2.

Waveguide Adapter

Introduction

This is a model of an adapter for microwave propagation in the transition between a rectangular and an elliptical waveguide. Such waveguide adapters are designed to keep energy losses due to reflections at a minimum for the operating frequencies. To investigate the characteristics of the adapter, the simulation includes a wave traveling from a rectangular waveguide through the adapter and into an elliptical waveguide. The S-parameters are calculated as functions of the frequency. The involved frequencies are all in the single-mode range of the waveguide, that is, the frequency range where only one mode is propagating in the waveguide.

Model Definition

The waveguide adapter consists of a rectangular part smoothly transcending into an elliptical part as seen in Figure 1.

Figure 1: The geometry of the waveguide adapter.

The walls of manufactured waveguides are typically plated with a good conductor such as silver. The model approximates the walls by perfect conductors. This is represented by the boundary condition $\mathbf{n} \times \mathbf{E} = \mathbf{0}$.

The rectangular port is excited by a transverse electric (TE) wave, which is a wave that has no electric field component in the direction of propagation. This is what an incoming wave would look like after traveling through a straight rectangular waveguide with the same cross section as the rectangular part of the adapter. The excitation frequencies are selected so that the TE_{10} mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes can be achieved analytically from the relation

$$(\nu_c)_{mn} = \frac{c}{2}\sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers, and c is the speed of light. For the TE_{10} mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 2.286$ cm and $b = 1.016$ cm), the TE_{10} mode is the only propagating mode for frequencies between 6.6 GHz and 14.7 GHz.

Although the shape of the TE_{10} mode is known analytically, this example lets you compute it using a numerical port. This technique is very general, in that it allows the port boundary to have any shape. The solved equation is

$$\nabla\times(n^{-2}\nabla\times H_n) + (n^{-2}\beta^2 - k_0^2)H_n = 0$$

Here H_n is the component of the magnetic field perpendicular to the boundary, n the refractive index, β the propagation constant in the direction perpendicular to the boundary, and k_0 the free space wave number. The eigenvalues are $\lambda = -j\beta$.

The same equation is solved separately at the elliptical end of the waveguide. The elliptical port is passive, but the eigenmode is still used in the boundary condition of the 3D propagating wave simulation. The dimensions of the elliptical end of the waveguide are such that the frequency range for the lowest propagating mode overlaps that of the rectangular port.

With the stipulated excitation at the rectangular port and the numerically established mode shapes as boundary conditions, the following equation is solved for the electric field vector $\mathbf{E}$ inside the waveguide adapter:

$$\nabla\times(\mu_r^{-1}\nabla\times\mathbf{E}) - k_0^{\,2}\left(\varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0}\right)\mathbf{E} = 0$$

where μ_r denotes the relative permeability, j the imaginary unit, σ the conductivity, ω the angular frequency, ε_r the relative permittivity, and ε_0 the permittivity of free space. The model uses the following material properties for free space: $\sigma = 0$ and $\mu_r = \varepsilon_r = 1$.

Results

Figure 2 shows a single-mode wave propagating through the waveguide.

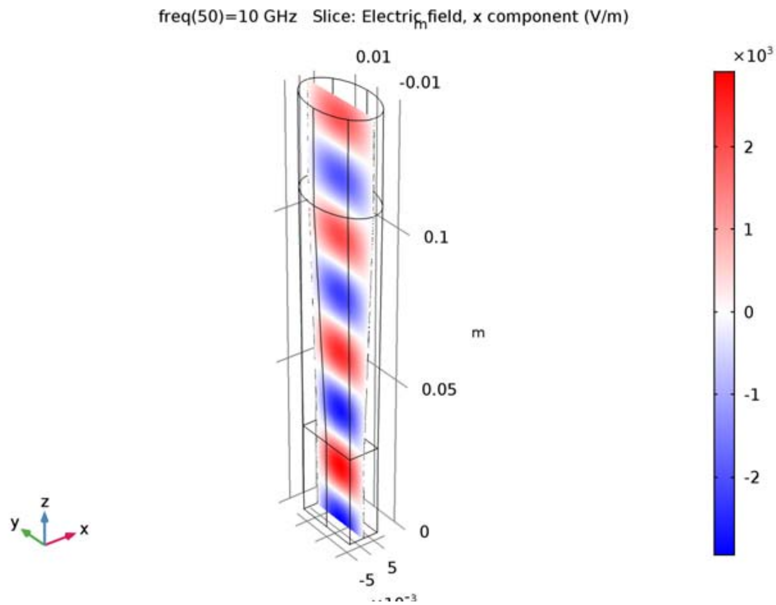


Figure 2: The x component of the propagating wave inside the waveguide adapter at the frequency 10 GHz.

Naming the rectangular port Port 1 and the elliptical port Port 2, the S-parameters describing the reflection and transmission of the wave are defined as follows:

$$S_{11} = \frac{\int_{\text{Port 1}} ((\mathbf{E}_c - \mathbf{E}_1) \cdot \mathbf{E}_1^*) dA_1}{\int_{\text{Port 1}} (\mathbf{E}_1 \cdot \mathbf{E}_1^*) dA_1}$$

$$S_{21} = \frac{\int_{\text{Port 2}} (\mathbf{E}_c \cdot \mathbf{E}_2^*) dA_2}{\int_{\text{Port 2}} (\mathbf{E}_2 \cdot \mathbf{E}_2^*) dA_2}$$

Here $\mathbf{E}_c$ is the calculated total field. $\mathbf{E}_1$ is the analytical field for the port excitation, and $\mathbf{E}_2$ is the eigenmode calculated from the boundary mode analysis and normalized with respect to the outgoing power flow. Figure 3 shows the S_{11} and S_{21} parameters as functions of the frequency.

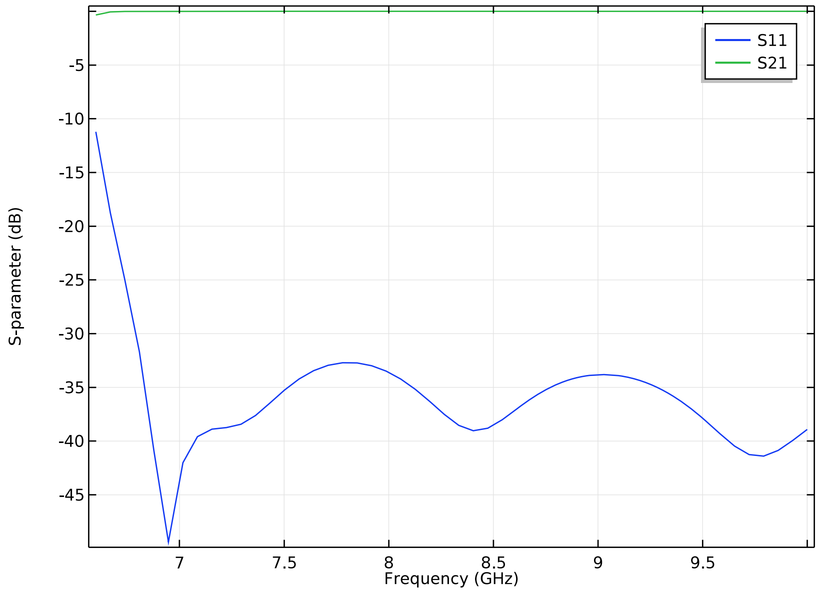


Figure 3: The S_{11} parameter and S_{21} parameter (in dB) as a function of the frequency. This parameter describes the reflections when the waveguide adapter is excited at the rectangular port and a measure of the part of the wave that is transmitted through the elliptical port when the waveguide adapter is excited at the rectangular port, respectively.

Application Library path: `RF_Module/Transmission_Lines_and_Waveguides/waveguide_adapter`

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 **Empty Study**.

6 Click **Done**.

STUDY 1

Step 1: Boundary Mode Analysis

1 On the **Study** toolbar, click **Study Steps** and choose **Other>Boundary Mode Analysis**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Mode analysis frequency** text field, type `7[GHz]`.

   The exact value of this frequency is not important. What matters is that it should be above the cutoff frequency for the fundamental mode, but below that for the next mode. This setting ensures that the boundary mode analysis finds the fundamental mode.

Add another boundary mode analysis, for the second port.

Step 2: Boundary Mode Analysis 2

1 On the **Study** toolbar, click **Study Steps** and choose **Other>Boundary Mode Analysis**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `2`.

4 In the **Mode analysis frequency** text field, type `7[GHz]`.

Finally, add the 3D equation for the propagating wave in the waveguide.

Step 3: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(6.6[GHz],3.4[GHz]/49,10[GHz])`.

Proceed to import the geometry.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `waveguide_adapter.mphbin`.

5 Click **Import**.

ADD MATERIAL

1 On 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.

MATERIALS

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Air (mat1)

By default, the first material you add applies on all domains so you need not alter any settings.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 In the **Settings** window for **Port**, locate the **Port Properties** section.

3 From the **Type of port** list, choose **Numeric**.

For the first port, wave excitation is **on** by default.

4 Select Boundary 13 only.

The wave enters the adapter through the port with a rectangular cross section.

Port 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 In the **Settings** window for **Port**, locate the **Port Properties** section.

3 From the **Type of port** list, choose **Numeric**.

4 In the **Port name** text field, type 2.

5 Select Boundary 6 only.

This is the exit port, the one with an elliptical cross-section.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

Now set up the study to find the boundary modes and use them when computing the field distribution over a range of frequencies.

Step 1: Boundary Mode Analysis

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Boundary Mode Analysis**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 Select the **Search for modes around** check box.

4 In the associated text field, type 50.

This value should be in the vicinity of the value that you expect the fundamental mode to have. If you do not know this in advance, you can experiment with some different values or estimate one from analytical formulas valid for cross-sections resembling yours.

5 From the **Transform** list, choose **Out-of-plane wave number**.

Step 2: Boundary Mode Analysis 2

1 In the **Model Builder** window, under **Study 1** click **Step 2: Boundary Mode Analysis 2**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 Select the **Search for modes around** check box.

4 In the associated text field, type 50.

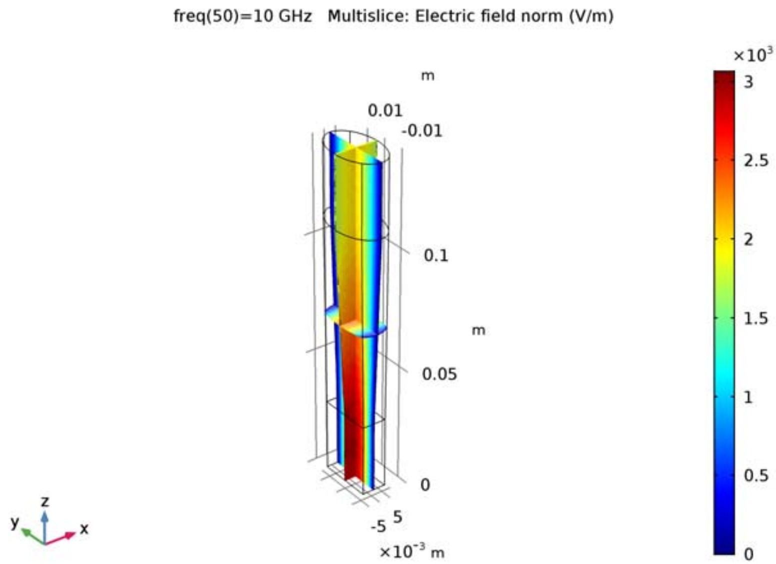
5 From the **Transform** list, choose **Out-of-plane wave number**.

6 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

1 Click the **Zoom Extents** button on the **Graphics** toolbar.



The default plot shows the norm of the electric field on slices through the waveguide; you can simplify and improve this plot.

Delete the Multislice plot.

Multislice

In the **Model Builder** window, expand the **Electric Field (emw)** node.

Slice 1

1 Right-click **Multislice** and choose **Delete**.

2 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Slice**.

3 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Electric>Electric field>emw.Ex - Electric field, x component**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 Locate the **Plane Data** section. In the **Planes** text field, type 1.

6 On the **Electric Field (emw)** toolbar, click **Plot**.

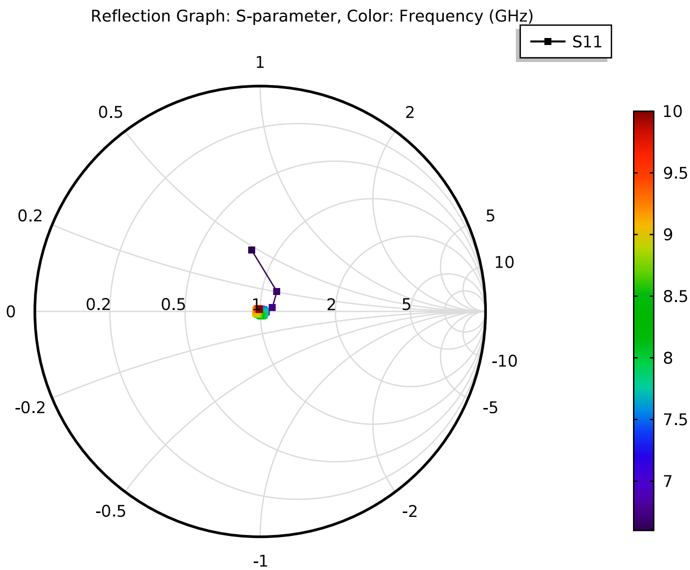
The plot now shows the x component of the electric field at the highest frequency, 10 GHz (compare with Figure 2). If you would like to see the field for other frequencies, you can select them by clicking on the **Electric Field (emw)** plot group.

Proceed by checking the plot of the S-parameters as functions of the frequency.

S-Parameter (emw)

Select the **S-Parameter (emw)** plot group under **Results** in Model Builder. The plot should closely resemble that in Figure 3.

Smith Plot (emw)



Created in COMSOL Multiphysics 5.3

Waveguide Iris Bandpass Filter

Introduction

A conductive diaphragm, an iris, placed transverse to a waveguide aperture causes a discontinuity and generates shunt reactance. Bandpass frequency response can be achieved from cascaded cavity resonators combined with such reactive elements, which can be created by inserting a series of iris elements inside the waveguide. This example consists of a WR-90 X-band waveguide and symmetrical inductive diaphragms (irises). The calculated S-parameters show good bandpass response and out-of-band rejection.

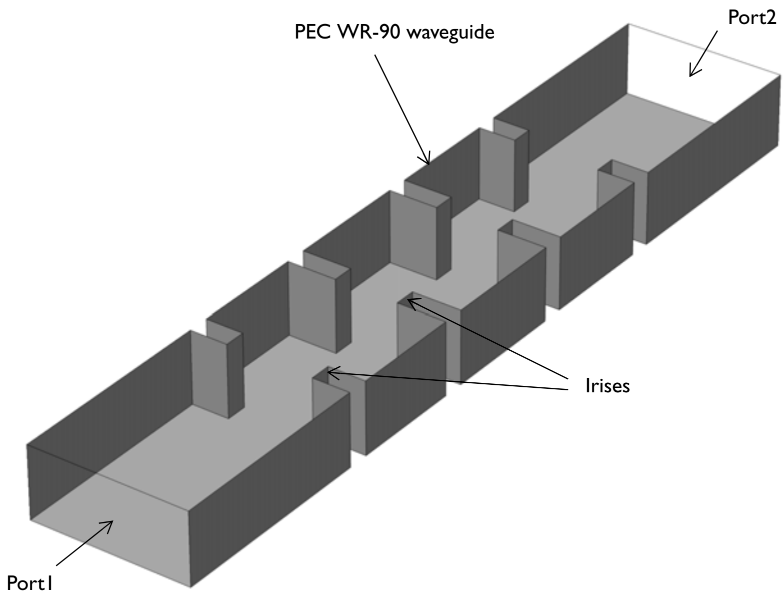


Figure 1: Symmetrical irises inside the waveguide generate shunt inductance. The top surface is removed to show the inside.

Model Definition

This example uses a WR-90 waveguide for X-band applications. The waveguide and iris parts are modeled as perfect electric conductors (PECs) and the inside of the waveguide is filled with air. On each end of the waveguide, a port boundary condition is applied with the predefined rectangular TE_{10} mode. Only one port is excited to observe the S-parameters of the example. The upper cut-off frequency can be approximately estimated using the resonant frequency of the biggest cavity located in the middle of the filter via

$$f_{nml} = \frac{c}{2\pi\sqrt{\varepsilon_r\mu_r}}\sqrt{\left(\frac{m\pi}{a}\right)^2 + \left(\frac{n\pi}{b}\right)^2 + \left(\frac{l\pi}{d}\right)^2}$$

where a and b are the dimensions of the waveguide's aperture and d is the length of the cavity. For this example, the values $a = 2.286$ cm, $b = 1.016$ cm, and $d = 1.73$ cm give a resonant frequency at the dominant mode, TE_{101}, of 10.87 GHz. Because the cavities are not completely closed but formed with the open irises, this estimation gives only an approximated value.

Results and Discussion

The default plot shows the norm of the electric fields in the waveguide; see Figure 2. At 10 GHz, the shape of the field distribution in each section of cavities looks like that of the dominant mode of a cavity.

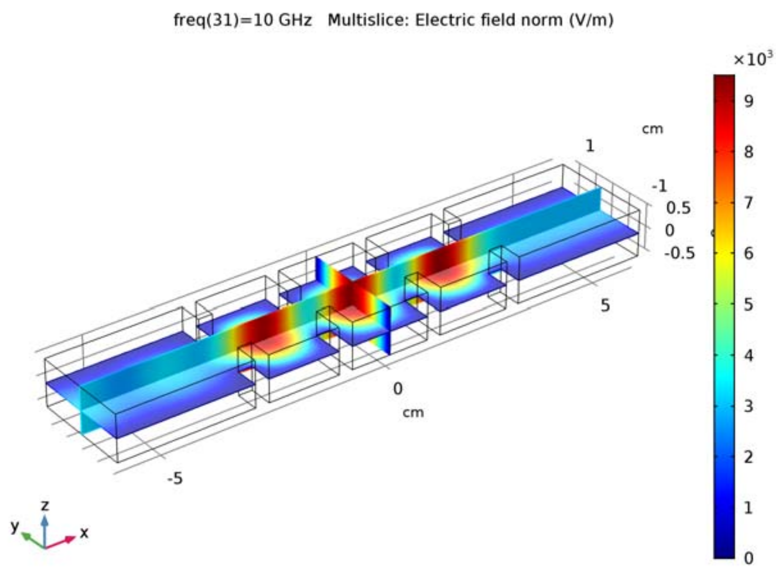


Figure 2: The E-field norm plot at the passband frequency shows the dominant mode resonance in each cavity formed by the irises.

Figure 3 shows the calculated S-parameters. The passband is around 10 GHz and good out-of-band rejection frequency response is observed.

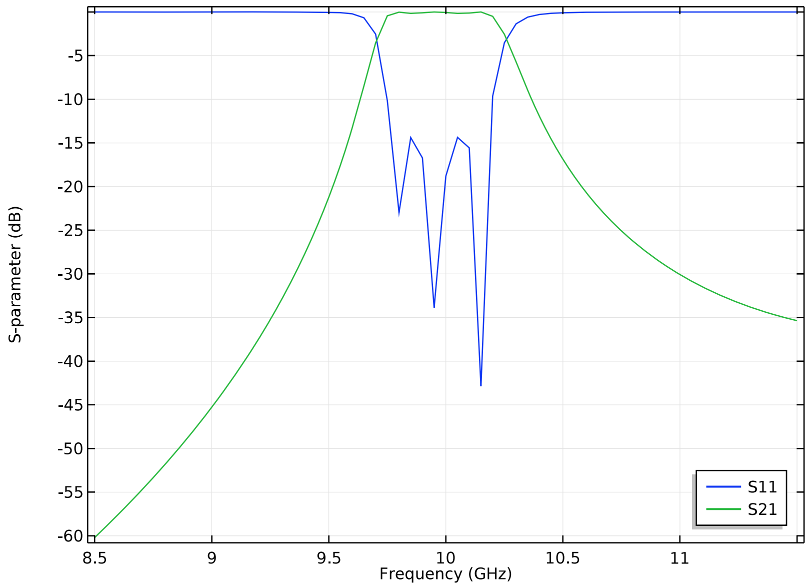


Figure 3: The calculated S-parameters show good matching characteristics as well as out-of-band rejection.

In Figure 4, the calculated S-parameters from the Frequency-Domain Modal method are plotted together with those of the discrete frequency sweep. While the frequency resolution of the Frequency-Domain Modal is five times finer than that of the discrete frequency sweep, the simulation time is four times faster to analyze the same filter.

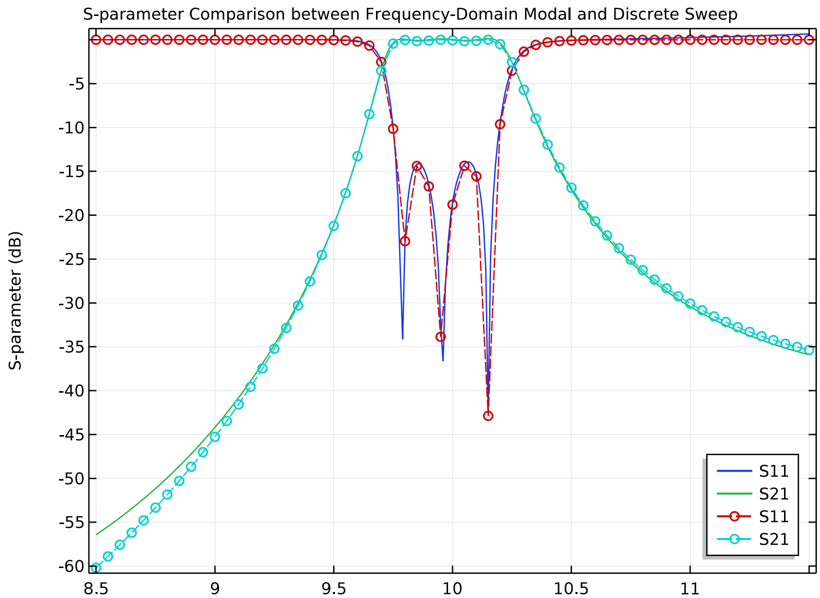


Figure 4: The calculated S-parameters from the Frequency-Domain Modal method are plotted with those of the discrete frequency sweep (dashed lines with circle makers).

References

1. D.M. Pozar, *Microwave Engineering*, John Wiley & Sons, 1998.

2. R.E. Collin, *Foundation of Microwave Engineering*, McGraw-Hill, 1992.

Application Library path: `RF_Module/Filters/waveguide_iris_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 **Preset 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 `8.5[GHz]` in the **Start** text field.
5 In the **Step** text field, type `0.05[GHz]`.
6 In the **Stop** text field, type `11.5[GHz]`.
7 Click **Replace**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `waveguide_iris_filter_parameters.txt`.

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 **cm**.

Create a block for the WR-90 waveguide.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `WR-90` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `12`.

4 In the **Depth** text field, type `w_wg`.

5 In the **Height** text field, type `h_wg`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 Click **Build All Objects**.

Choose wireframe rendering to get a view inside the waveguide when adding the irises.

8 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Next, add a block for generating the inner irises, which form the center cavity.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Iris1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `d_iris`.

4 In the **Depth** text field, type `l_iris1`.

5 In the **Height** text field, type `h_wg`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `spacing/2`.

8 In the **y** text field, type `(w_wg-l_iris1)/2`.

Add another block for generating the outer irises, which enclose the first and last cavities.

Block 3 (blk3)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Iris2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `d_iris`.

4 In the **Depth** text field, type `l_iris2`.

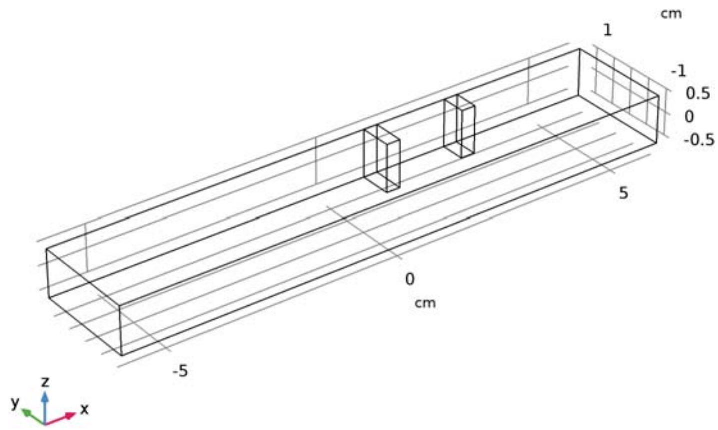
5 In the **Height** text field, type `h_wg`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `spacing*1.42`.

8 In the **y** text field, type `(w_wg-l_iris2)/2`.

9 Click **Build All Objects**.



Create symmetrical inductive diaphragms by mirroring the iris blocks a couple of times.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Select the objects **blk2** and **blk3** 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`.

Mirror 2 (mir2)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Select the objects **mir1(1)**, **mir1(2)**, **blk2**, and **blk3** 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 **y** text field, type `1`.
6 In the **z** text field, type `0`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

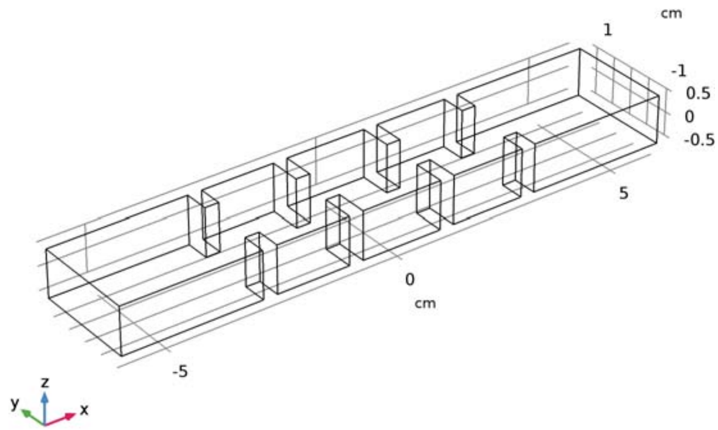
2 Select the object **blk1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the objects **mir1(1)**, **mir2(2)**, **mir2(3)**, **mir1(2)**, **mir2(1)**, **blk2**, **blk3**, and **mir2(4)** only.

6 Click **Build All Objects**.



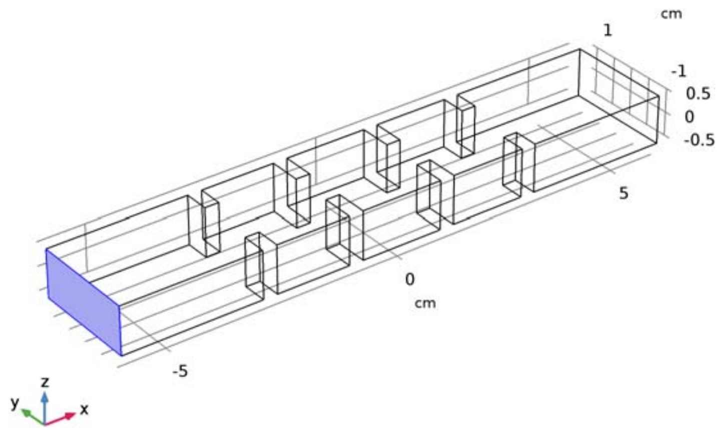
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now, set up the physics. The default boundary condition, Perfect Electric Conductor, applies to all exterior boundaries.

Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 1 only.



The excitation port

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

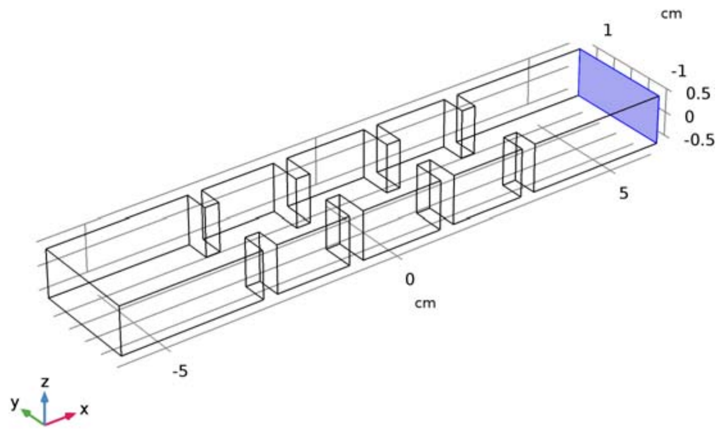
4 From the **Type of port** list, choose **Rectangular**.

For the first port, wave excitation is **on** by default.

Port 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 38 only.



The observation port

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Rectangular**.

MATERIALS

Assign material properties on the model. Use air for all domains.

ADD MATERIAL

1 On 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.

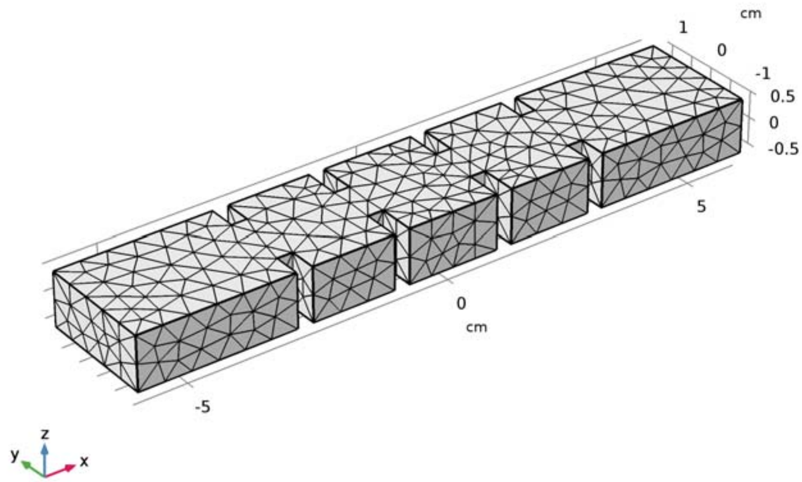
MATERIALS

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows the distribution of the norm of the electric field. Choose the center frequency of the passband.

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (GHz))** list, choose **10**.

4 On the **Electric Field (emw)** toolbar, click **Plot**.

The resonant E-field should be observed in the cavities. Compare with Figure 2.

S-Parameter (emw)

1 In the **Model Builder** window, under **Results** click **S-Parameter (emw)**.

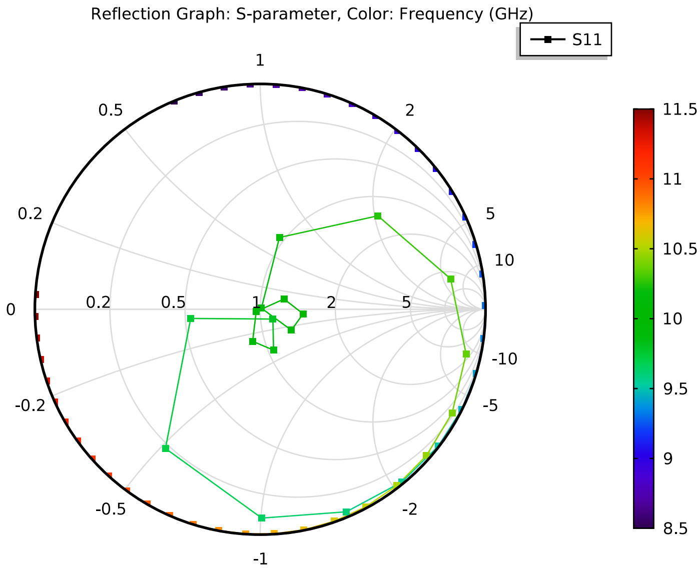
2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 On the **S-Parameter (emw)** toolbar, click **Plot**.

The resulting plot shows the S-parameters of the filter. Compare the plot with Figure 3.

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.

ROOT

On the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Frequency-Domain Modal**.

3 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Eigenfrequency

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency**.

3 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

4 In the **Search for eigenfrequencies around** text field, type `9.5[GHz]`.

Step 2: Frequency-Domain Modal

1 In the **Model Builder** window, under **Study 2** 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(8.5[GHz],0.01[GHz],11.5[GHz])`.

With a very fine frequency step simulation, the solutions contain a lot of data. As a result, the model file size will increase tremendously when it is saved. By selecting the Store fields in output check box in the Values of Dependent Variables section of the Frequency Domain study step settings, it is possible to define for what part of the model the computed solution should be saved. When only S-parameters are of interest, it is not necessary to store all of the field solutions. Instead, only store the field on the selections for the port boundaries, as those will be used for the S-parameter calculations.

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, click **OK**.

Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** 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, click **OK**.

STUDY 2

Step 2: Frequency-Domain Modal

1 In the **Model Builder** window, under **Study 2** click **Step 2: Frequency-Domain Modal**.

2 In the **Settings** window for **Frequency-Domain Modal**, click to expand the **Values of dependent variables** section.

3 Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

4 Under **Selections**, click **Add**.

5 In the **Add** dialog box, In the **Selections** list, choose **Explicit 1** and **Explicit 2**.

6 Click **OK**.

7 On the **Home** toolbar, click **Compute**.

RESULTS

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw) 1** node, then click **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **X-planes** subsection. From the **Entry method** list, choose **Coordinates**.

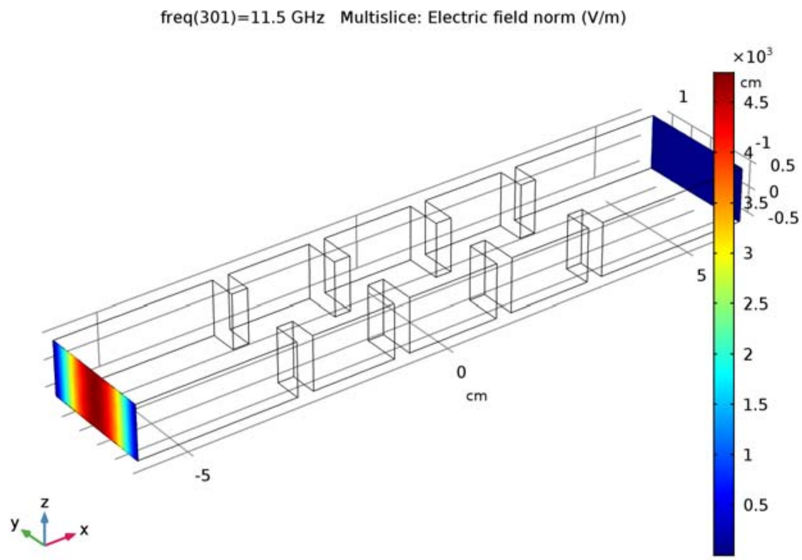
4 In the **Coordinates** text field, type `-6 6`.

5 Find the **Y-planes** subsection. From the **Entry method** list, choose **Number of planes**.

6 In the **Planes** text field, type `0`.

7 Find the **Z-planes** subsection. In the **Planes** text field, type `0`.

8 On the **Electric Field (emw) 1** toolbar, click **Plot**.



Since 11.5 GHz is not within the pass-band, the input power at the excitation port is not delivered to the observation port.

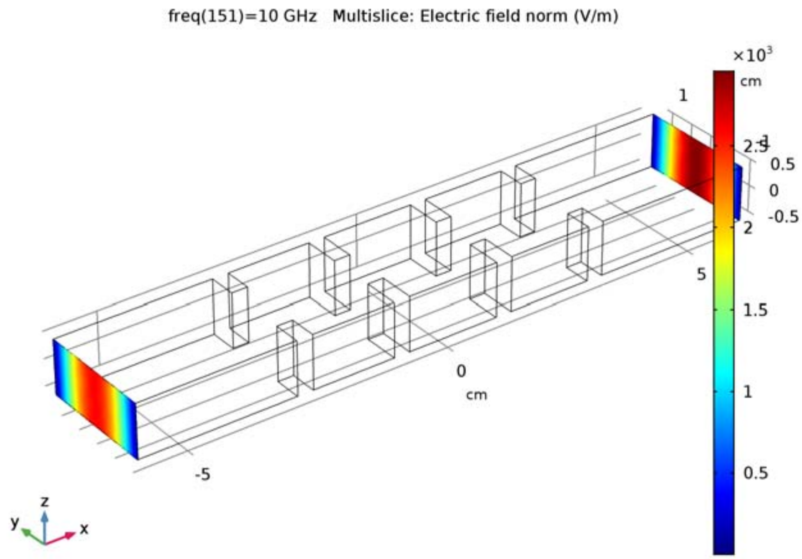
Electric Field (emw) 1

1 In the **Model Builder** window, under **Results** click **Electric Field (emw) 1**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (GHz))** list, choose **10**.

4 On the **Electric Field (emw) 1** toolbar, click **Plot**.



When the frequency of the plot is within the pass-band, the input power at the excitation port is delivered to the observation port.

Next, plot the calculated S-parameters from the Frequency-Domain Modal method together with those of the discrete frequency sweep.

Global 2

1. In the **Model Builder** window, under **Results** right-click **S-Parameter (emw) 1** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.
4. Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S11dB - S11**.
5. Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S21dB - S21**.
6. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

8 From the **Positioning** list, choose **In data points**.

9 On the **S-Parameter (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**, click to expand the **Title** section.

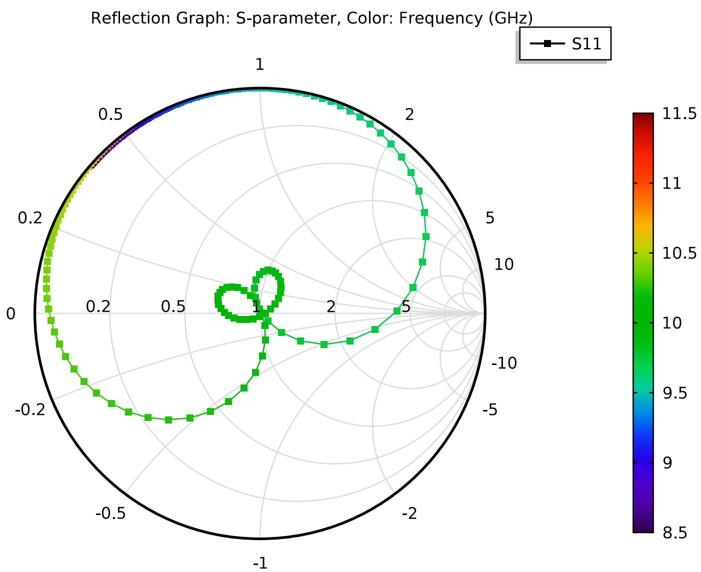
3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `S-parameter Comparison between Frequency-Domain Modal and Discrete Sweep`.

5 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

Compare the plot with Figure 4.

Smith Plot (emw) 1



Created in COMSOL Multiphysics 5.3

SMA Connectorized Wilkinson Power Divider

Introduction

Resistive power dividers and T-junction power dividers are two conventional types of three-port power dividers. Such dividers are either lossy or not matched to the system reference impedance at all ports. In addition, isolation between two coupled ports is not guaranteed. The Wilkinson power divider outperforms both the lossless T-junction divider and the resistive divider and does not have the issues mentioned above. This example shows how to model such a device.

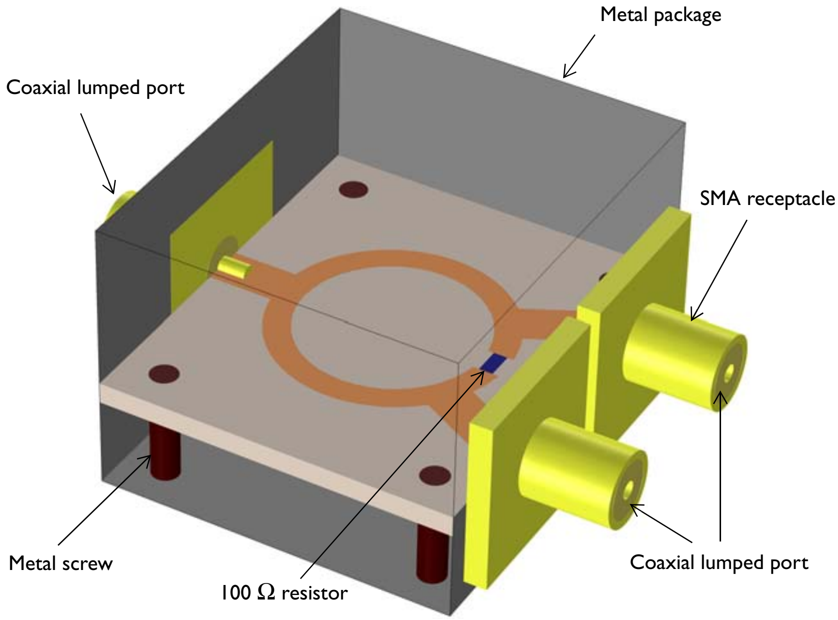


Figure 1: A Wilkinson power divider is fabricated on a 60 mil substrate. An SMA receptacle is added on each port and the circuit board is suspended in the metal package using screws.

Model Definition

The Wilkinson power divider is a three-port device composed of 50 Ω and 70.7 Ω microstrip lines on a dielectric substrate with a ground plane and a 100 Ω resistor mounted between two ports. The model also includes a metal enclosure, screws, and SMA receptacles connected to each port representing a complete package of a power divider shown in Figure 1. Except for the microstrip lines and ground plane, model all the SMA receptacles, screws, and the metal package using perfect electric conductor (PEC) boundaries. The SMA receptacle and screw domains enclosed by these PEC boundaries are not part of the example analysis, so they are set to PEC by default. The microstrip lines and ground plane made of 1 oz copper layers are modeled using a transition boundary condition with 35 μm thickness to address lossy conductive surfaces due to finite copper conductivity. The relative dielectric constant, ε_r, of the 60 mil substrate is 3.38. The boundaries facing the dielectric-filled coaxial connector of the SMA receptacles are specified as coaxial lumped ports. The 100 Ω resistor is realized via a uniform lumped port with100 Ω characteristic impedance.

Results and Discussion

Figure 2 shows the symmetric E-field norm distribution on the top of the substrate. The input energy is equally coupled to each output port.

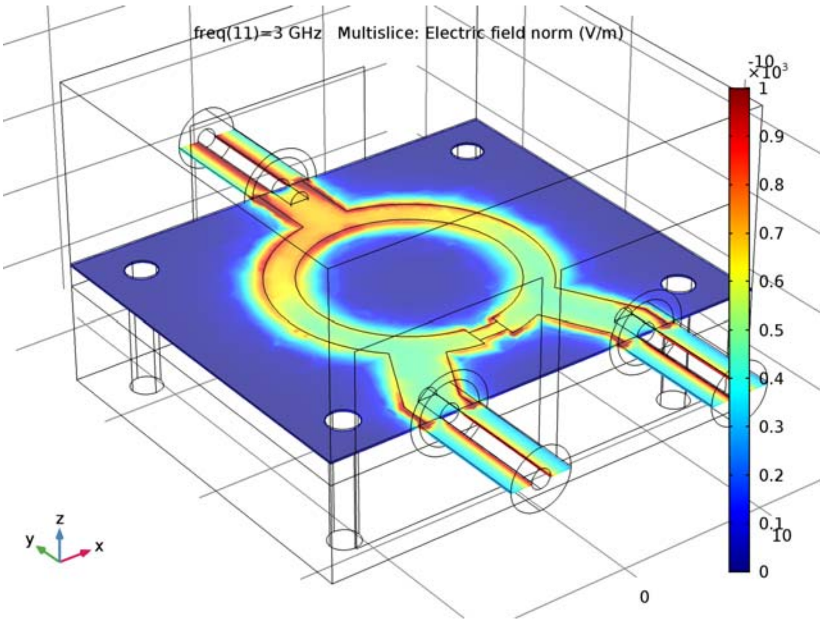


Figure 2: The E-field norm plot shows that the input is evenly split between the two output ports.

The S-parameters plotted in Figure 3 show the frequency response of the Wilkinson power divider. Good input impedance matching characteristics are observed and the coupled power at each output port is about –3 dB around 3 GHz.

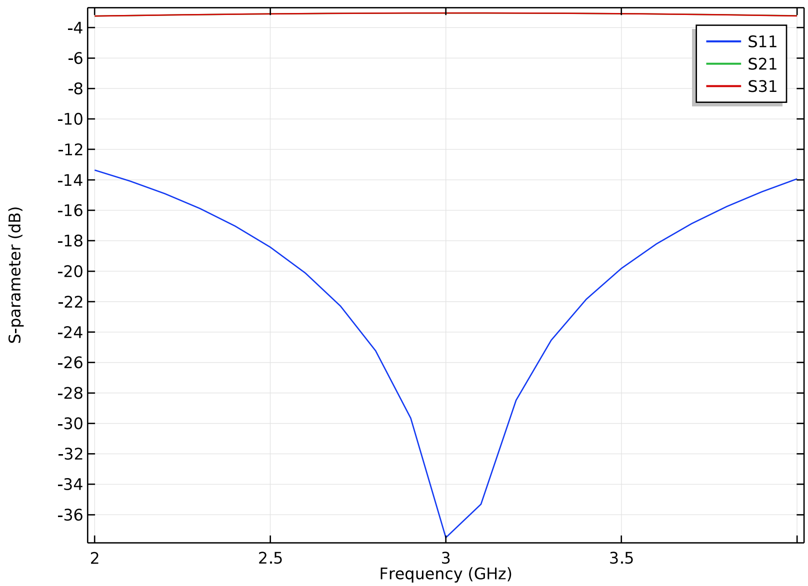


Figure 3: The S-parameters show very good input matching at 3 GHz and evenly divided power at the two output ports.

References

1. D.M. Pozar, *Microwave Engineering*, John Wiley & Sons, 1998.

2. R.E. Collin, *Foundation of Microwave Engineering*, McGraw-Hill, 1992.

Application Library path: `RF_Module/Couplers_and_Power_Dividers/wilkinson_power_divider`

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 **Preset 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 In the **Frequencies** text field, type `range(2[GHz],0.1[GHz],4[GHz])`.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `wilkinson_power_divider_parameters.txt`.

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**.

First, create the substrate.

Block 1 (blk1)

1 On 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 `w_subs`.

4 In the **Depth** text field, type `l_subs`.

5 In the **Height** text field, type `1.524`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **z** text field, type `-0.762`.

Add a block for the metal package.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Package` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_subs`.

4 In the **Depth** text field, type `l_subs`.

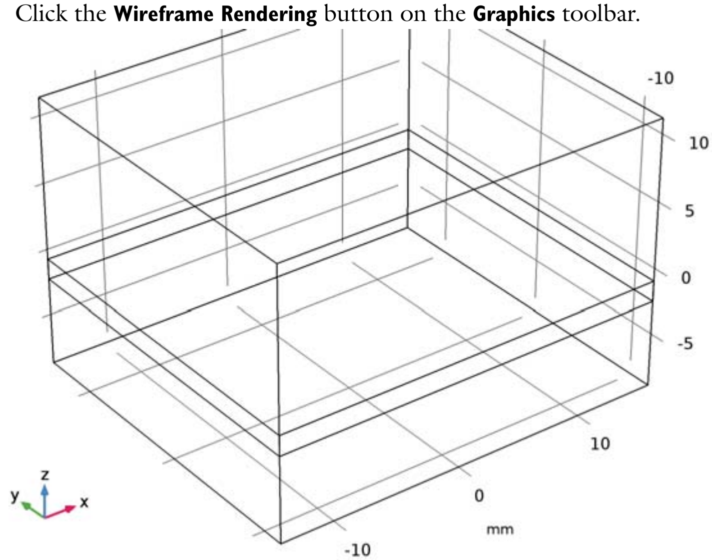
5 In the **Height** text field, type `20`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **z** text field, type `2`.

8 Right-click **Package** and choose **Build Selected**.

9 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



Add a work plane for drawing the layout of the power divider.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Plane Geometry

Add two circles to create the ring strip part.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, type `Ring outer` in the **Label** text field.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `r_ring`.

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, type `Ring inner` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_ring-1.87`.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Ring cut` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `2`.

4 In the **Height** text field, type `3`.

5 Locate the **Position** section. In the **xw** text field, type `-1`.

6 In the **yw** text field, type `-9`.

Difference 1 (dif1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

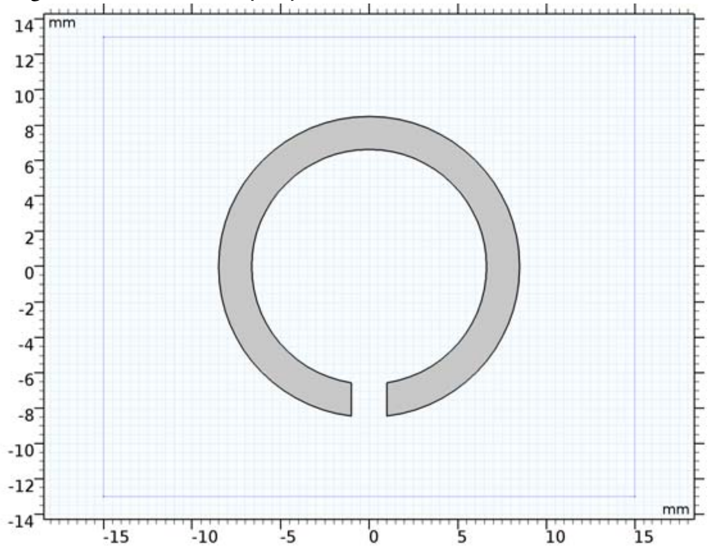
2 Select the object **c1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the objects **r1** and **c2** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.



Add a rectangle for the 100 ohm resistor.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Lumped element` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `2`.

4 Locate the **Position** section. In the **xw** text field, type `-1`.

5 In the **yw** text field, type `-8`.

Add rectangles for the 50 ohm microstrip feed lines.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `3.2`.

4 In the **Height** text field, type `5`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **yw** text field, type `10.5`.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `3.2`.

4 In the **Height** text field, type `2`.

5 Locate the **Position** section. In the **xw** text field, type `-7`.

6 From the **Base** list, choose **Center**.

7 In the **yw** text field, type `-12`.

Rectangle 5 (r5)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `3.2`.

4 In the **Height** text field, type `6`.

5 Locate the **Position** section. In the **xw** text field, type `-8.6`.

6 In the **yw** text field, type `-11`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-28`.

Mirror 1 (mir1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **r4** and **r5** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

Create a union of all objects except the small rectangle for the resistor (r2) to remove unnecessary boundaries.

Union 1 (uni1)

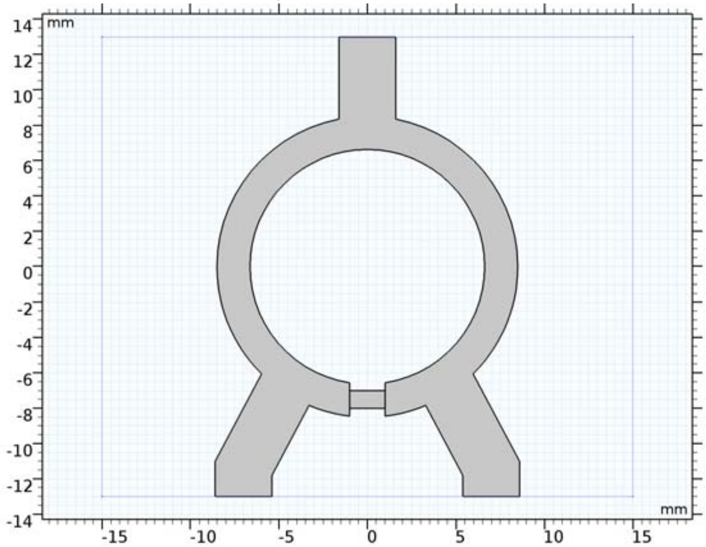
1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mir1(1)**, **dif1**, **mir1(2)**, **r3**, **r4**, and **r5** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 On the **Work Plane** toolbar, click **Build All**.



The power divider layout drawn on the substrate.

6 In the **Model Builder** window, click **Geometry 1**.

Create the coax SMA receptacle composed of the coax inner and outer conductors, the SMA connector part, and the flange.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `Coax inner` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_inner`.

4 In the **Height** text field, type `l_sma+2`.

5 Locate the **Position** section. In the **x** text field, type `-7`.

6 In the **y** text field, type `-l_subs/2-l_sma`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `SMA` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_outer+0.6`.

4 In the **Height** text field, type `l_sma`.

5 Locate the **Position** section. In the **x** text field, type `-7`.

6 In the **y** text field, type `-l_subs/2-l_sma`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Block 3 (blk3)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Flange` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `12.7`.

4 In the **Depth** text field, type `12.7`.

5 In the **Height** text field, type `1.65`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `-7`.

8 In the **y** text field, type `-(l_subs+1.65)/2`.

9 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Create a union of a couple of objects, the SMA connector, and the flange to remove unnecessary boundaries.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **cyl2** and **blk3** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Cylinder 3 (cyl3)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `Coax dielectric` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_outer`.

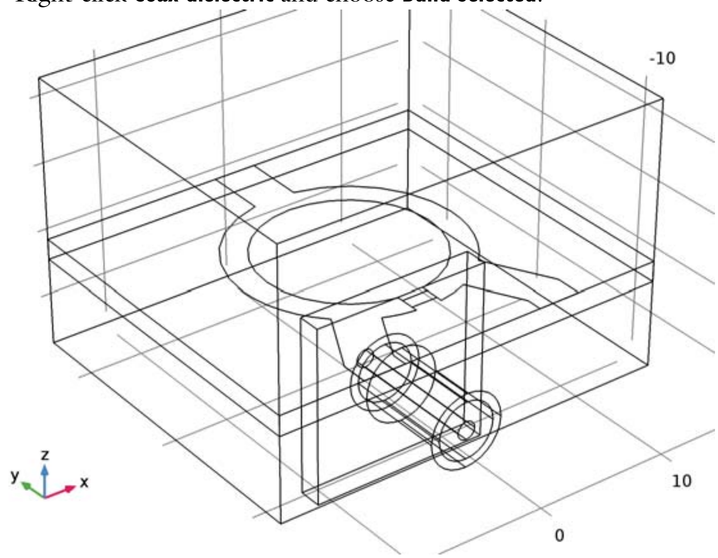
4 In the **Height** text field, type `l_sma`.

5 Locate the **Position** section. In the **x** text field, type `-7`.

6 In the **y** text field, type `-l_subs/2-l_sma`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

8 Right-click **Coax dielectric** and choose **Build Selected**.



Create two more SMA receptacles.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the objects **uni1**, **cyl1**, and **cyl3** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `7,14`.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **copy1(1)**, **copy1(3)**, and **copy1(5)** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `180`.

Add a cylinder for the metal screw.

Cylinder 4 (cyl4)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `Screw` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `8`.

4 Locate the **Position** section. In the **x** text field, type `-12`.

5 In the **y** text field, type `-10`.

6 In the **z** text field, type `-8`.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **cyl4** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

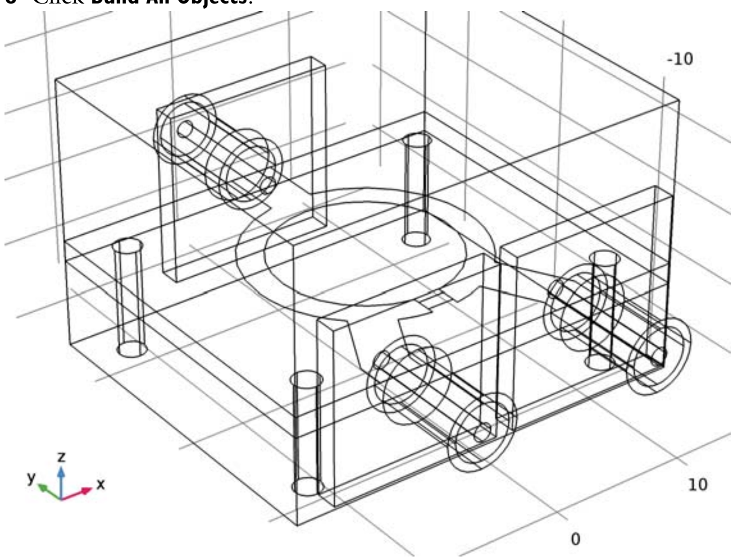
4 In the **x size** text field, type `2`.

5 In the **y size** text field, type `2`.

6 Locate the **Displacement** section. In the **x** text field, type `24`.

7 In the **y** text field, type `20`.

8 Click **Build All Objects**.



DEFINITIONS

Create a set of selections to use when setting up the physics. Begin with the microstrip line boundaries including the substrate ground plane.

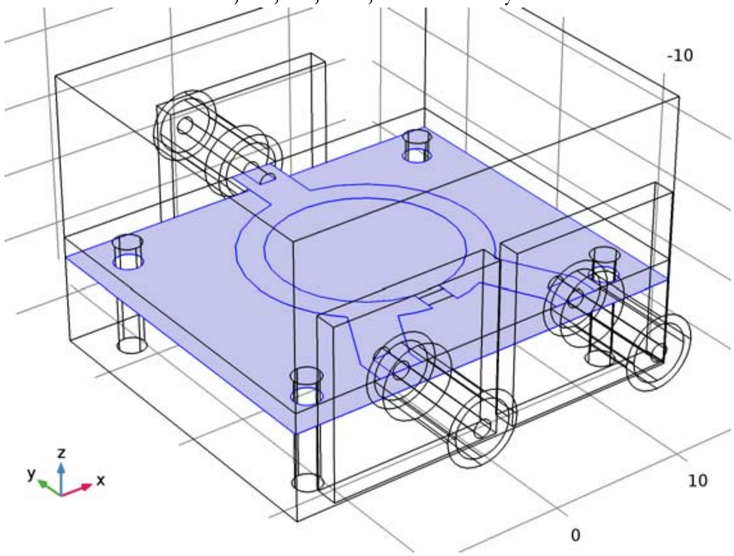
Explicit 1

1 On 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 6, 54, 61, 102, and 147 only.



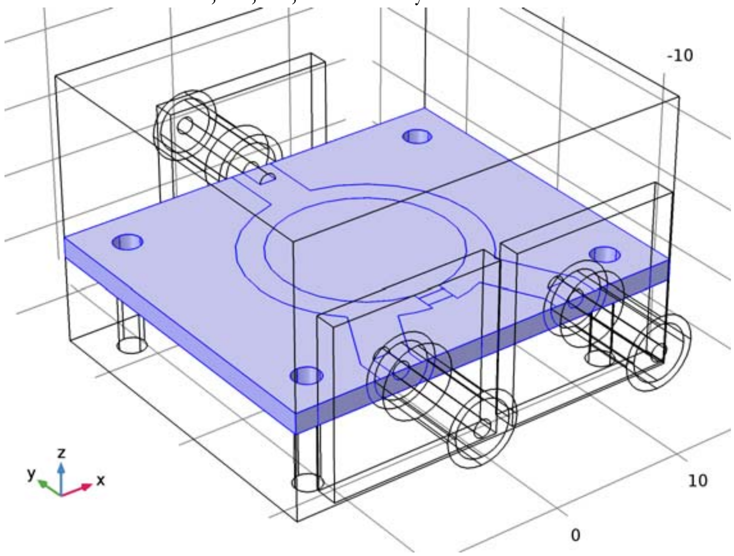
Add a selection for the substrate.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Substrate` in the **Label** text field.

3 Select Domains 2, 11, 15, and 21 only.



Add a selection for the coax dielectric (PTFE).

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Coax dielectric` in the **Label** text field.

3 Select Domains 9, 14, and 19 only.

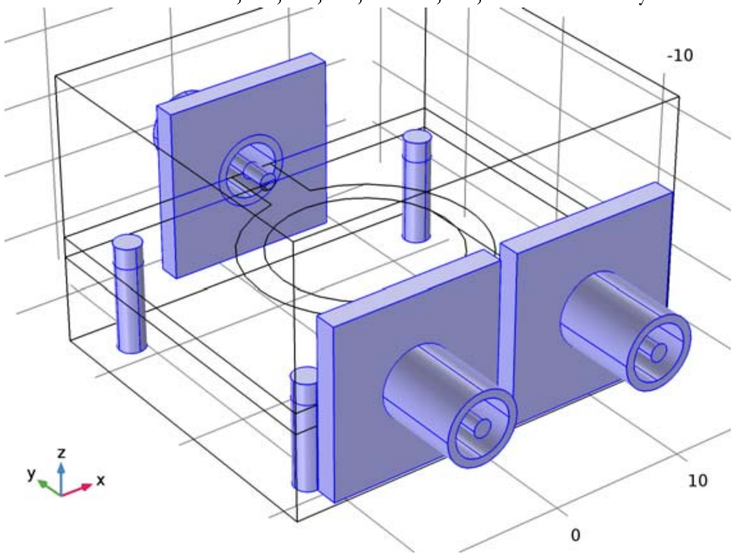
Add a selection for the domains consisting of metal. These domains are not part of the model analysis.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Metal volume` in the **Label** text field.

3 Select Domains 4–8, 10, 12, 13, 16–18, 20, and 22–26 only.



Define the model domain, which is the complement of the metal volume selection.

Complement 1

1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Model domain` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Metal volume** in the **Selections to invert** list.

5 Click **OK**.

View 1

Suppress some boundaries to get a view of the interior while setting the physics and mesh.

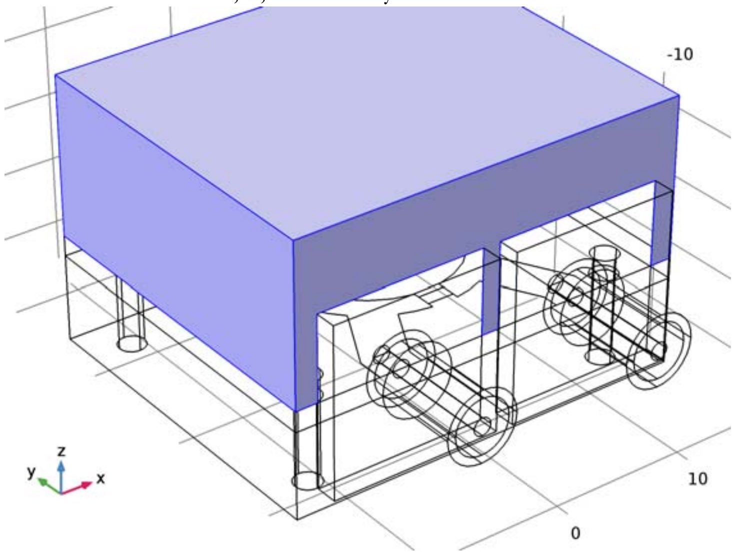
Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** 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 7, 8, and 10 only.



Now, set up the physics.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Model domain**.

Perfect Electric Conductor 1

The Perfect Electric Conductor applies by default to all exterior boundaries. After restricting the Electromagnetics Waves, Frequency Domain interface to the model domain, these outer boundaries include the coax SMA receptacles and the metal screws. Add a Transition Boundary Condition to the microstrip line and the substrate ground plane.

Transition Boundary Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Transition Boundary Condition**.

2 In the **Settings** window for **Transition Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Microstrip line**.

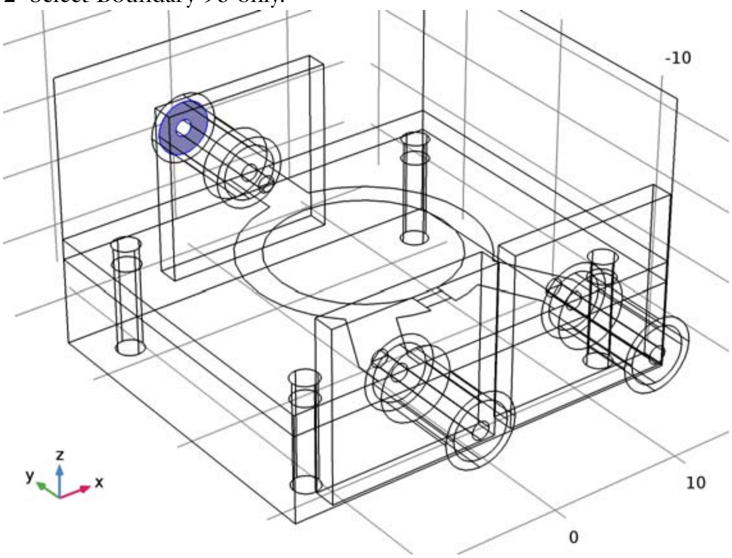
4 Locate the **Transition Boundary Condition** section. In the *d* text field, type `35[um]`.

Proceed with the Lumped Port conditions.

Lumped Port 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 95 only.



3 In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.

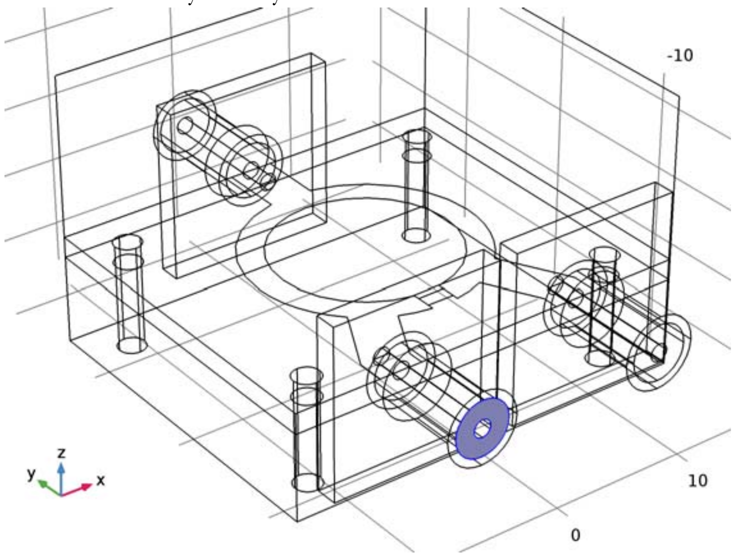
4 From the **Type of lumped port** list, choose **Coaxial**.

For the first port, wave excitation is **on** by default.

Lumped Port 2

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 51 only.



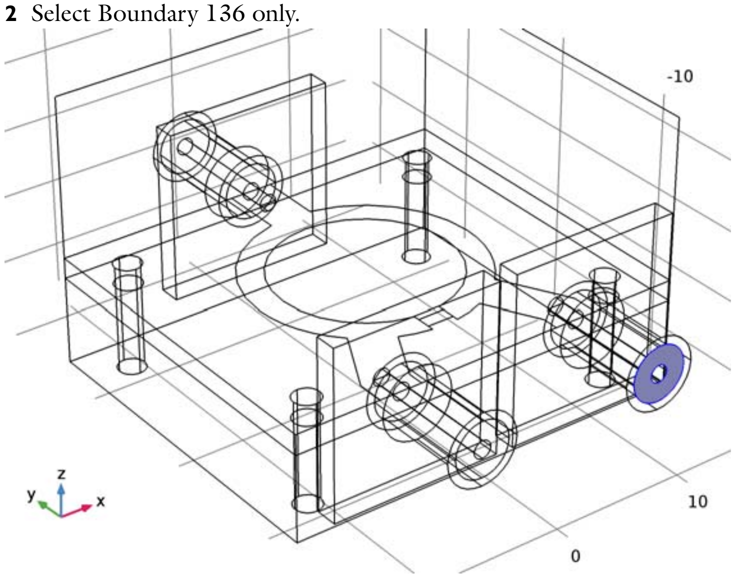
3 In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.

4 From the **Type of lumped port** list, choose **Coaxial**.

Lumped Port 3

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Port**.

2 Select Boundary 136 only.



3 In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.

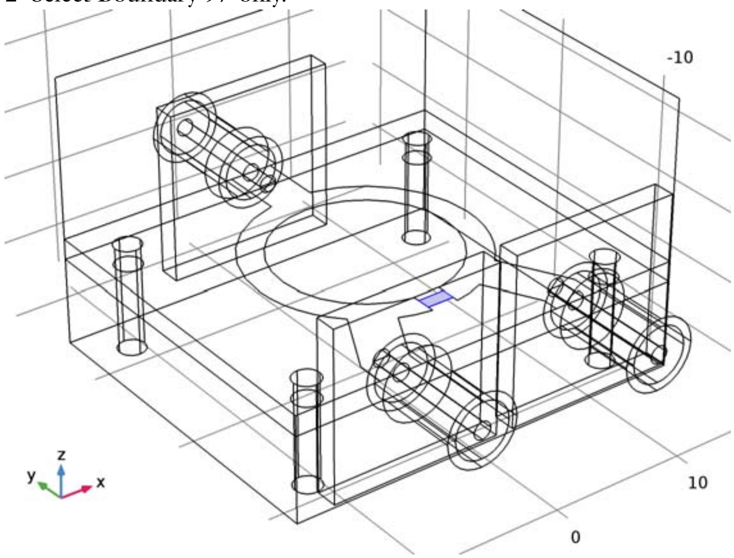
4 From the **Type of lumped port** list, choose **Coaxial**.

Add a lumped element for the 100Ω resistor.

Lumped Element 1

1 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Lumped Element**.

2 Select Boundary 97 only.



3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 In the Z_{element} text field, type `100[ohm]`.

MATERIALS

Next, assign material properties. First, specify air for all domains.

ADD MATERIAL

1 On 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 Go to the **Add Material** window.

6 In the tree, select **Built-In>Copper**.

7 Click **Add to Component** in the window toolbar.

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Microstrip line**.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Override the material for the substrate domains with a dielectric material of $\varepsilon_r = 3.38$.

Material 3 (mat3)

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	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

Similarly, override the coax dielectric domains with a material of $\varepsilon_r = 2.1$.

Material 4 (mat4)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `PTFE` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Coax dielectric**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

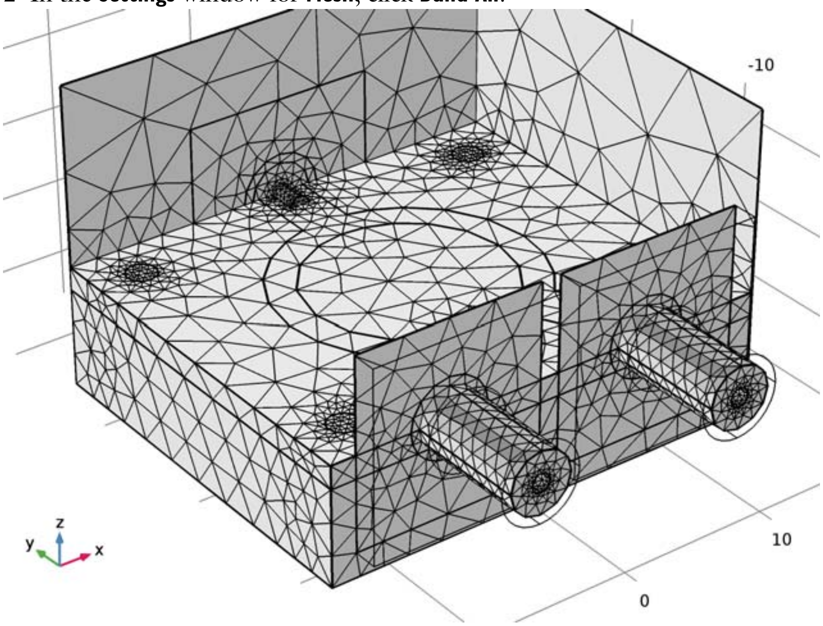
Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	2.1	1	Basic

Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows the E-field norm distribution. Change the settings to plot the E-field norm on the substrate.

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (GHz))** list, choose **3**.

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 -0.1.

7 Click to expand the **Range** section. Select the **Manual color range** check box.

8 In the **Maximum** text field, type 1000.

The resulting plot shows the E-field equally split between Port 2 and Port 3. Compare with Figure 2.

S-Parameter (emw)

The reproduced plot shows the calculated S-parameters. Compare with Figure 3.

Smith Plot (emw)