



RF Module

Model Library Manual



VERSION 4.3b

 COMSOL

RF Module Model Library Manual

© 1998–2013 COMSOL

Protected by U.S. Patents 7,519,518; 7,596,474; and 7,623,991. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/sla) and may be used or copied only under the terms of the license agreement.

COMSOL, COMSOL Multiphysics, Capture the Concept, COMSOL Desktop, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/tm.

Version:

May 2013

COMSOL 4.3b

Contact Information

Visit the Contact Us 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
- Download COMSOL: www.comsol.com/support/download
- Product Updates: www.comsol.com/support/updates
- COMSOL Community: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Center: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM021002

Optimizing a Bow-Tie Antenna

Introduction

A bow-tie antenna patterned on a dielectric substrate is optimized by changing the length of the arms and the flare angle to reduce the magnitude of S_{11} , the reflection coefficient. The two geometric dimensions that are used as design variables directly control the size and shape of the antenna, and also affect the dimensions of the dielectric substrate. The gradient-free Nelder-Mead optimization method is used to improve the objective function.

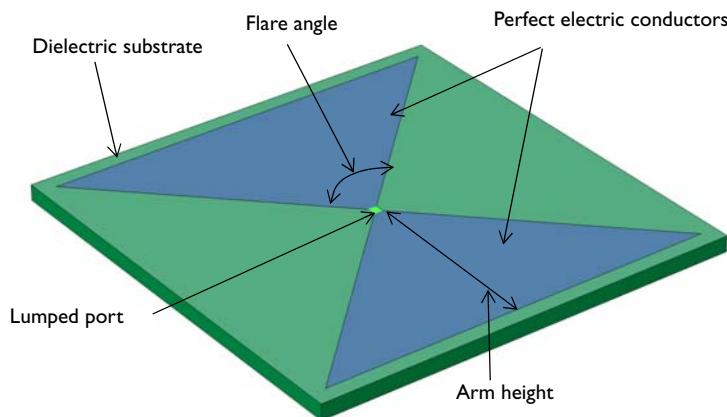


Figure 1: Outline of a bow-tie antenna on a dielectric substrate.

Model Definition

The model of the bow-tie antenna consists of a rectangular dielectric substrate with two triangular faces on top representing a metal pattern, as shown in [Figure 1](#). The flare angle and height of the arms define the antenna's shape. It is desirable to keep the size of the substrate as small as possible, so it is defined to extend a distance of 2 mm around the outline of the pattern. A lumped port excitation applied to a small rectangular face between the antenna arms mimics a $50\ \Omega$ transmission line feed.

The two design variables, flare angle and arm height, control a total of four dimensions in the model. The height and width of the rectangular dielectric substrate are defined to be slightly larger than the profile of the antenna. Thus, the two design variables

explicitly control two other geometric dimensions in the model which directly affect the antenna characteristics.

The reflection coefficient, $|S_{11}|$, is minimized by using the gradient-free optimization method. Because the objective function is non-analytic, it is not possible to analytically compute the derivatives with respect to the design variables. Furthermore, the design parameters will introduce significant geometric changes to the domains. Both of these reasons motivate the use of the gradient-free optimization method, which numerically approximates the gradients of the objective function. Using this approximate gradient information, the objective function is iteratively improved until the design variables are converged within the desired tolerance.

Results and Discussion

The model is solved for a single frequency, 2.45 GHz, and the optimizer adjusts the flare angle and height of the arms such that the reflection coefficient is reduced. For the initial, unoptimized, design $|S_{11}| = 0.62$, while $|S_{11}| = 0.20$ for the optimized design. The presence of the variably-sized dielectric substrate would make it quite difficult to use any analytic relationships to achieve this optimized design. The optimized values for the height and flare angle are 17.02 mm and 51.84°, respectively.

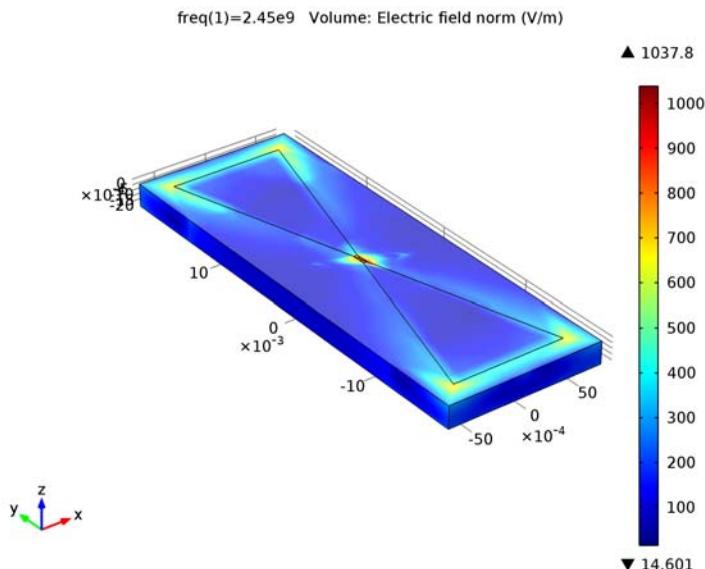


Figure 2: The electric field norm in an optimized antenna layout.

Model Library path: RF_Module/Antennas/bowtie_antenna_optimization

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 6 Click **Finish**.

GLOBAL DEFINITIONS

Define the model parameters under Global Definitions.

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
f0	2.45[GHz]	Operating frequency
L0	c_const/f0	Free space wavelength
h0	0.025[m]	Height of antenna arm
theta	90[deg]	Flare angle
Gap	L0/100	Gap at feed point
Padding	2[mm]	Padding around antenna
Thickness	2[mm]	Substrate thickness

Follow the instructions below to construct the model geometry. First create the geometry of a bow-tie antenna.

GEOMETRY I

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

Square I

- 1 In the **Model Builder** window, under **Model I>Geometry I>Work Plane I** right-click **Plane Geometry** and choose **Square**.
- 2 In the **Square** settings window, locate the **Size** section.
- 3 In the **Side length** edit field, type **L0**.
- 4 Locate the **Rotation Angle** section. In the **Rotation** edit field, type **theta/2**.
- 5 Click the **Build Selected** button.

Mirror I

- 1 Right-click **Plane Geometry** and choose **Transforms>Mirror**.
- 2 Select the object **sqI** only.
- 3 In the **Mirror** settings window, locate the **Input** section.
- 4 Select the **Keep input objects** check box.
- 5 Click the **Build Selected** button.

Rectangle I

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type **L0**.
- 4 In the **Height** edit field, type **h0**.
- 5 Locate the **Position** section. From the **Base** list, choose **Center**.
- 6 In the **yw** edit field, type **h0/2**.
- 7 Click the **Build Selected** button.

Intersection I

- 1 Right-click **Plane Geometry** and choose **Boolean Operations>Intersection**.
- 2 Click in the **Graphics** window, press **Ctrl+A** to highlight all objects, and then right-click to confirm the selection.
- 3 Click the **Build Selected** button.

Mirror 2

- 1 Right-click **Plane Geometry** and choose **Transforms>Mirror**.
- 2 Select the object **intI** only.

- 3** In the **Mirror** settings window, locate the **Normal Vector to Line of Reflection** section.
- 4** In the **xw** edit field, type 0.
- 5** In the **yw** edit field, type 1.
- 6** Locate the **Input** section. Select the **Keep input objects** check box.
- 7** Click the **Build Selected** button.

Rectangle 2

- 1** Right-click **Plane Geometry** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type $\tan(\theta/2) * \text{Gap}$.
- 4** In the **Height** edit field, type **Gap**.
- 5** Locate the **Position** section. From the **Base** list, choose **Center**.
- 6** Click the **Build Selected** button.

Rectangle 3

- 1** Right-click **Plane Geometry** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type $2 * h_0 * \tan(\theta/2) + 2 * \text{Padding}$.
- 4** In the **Height** edit field, type $2 * h_0 + 2 * \text{Padding}$.
- 5** Locate the **Position** section. From the **Base** list, choose **Center**.
- 6** Click the **Build Selected** button.

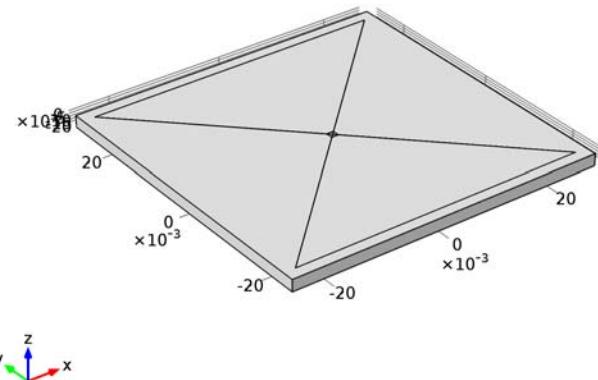
Extrude 1

- 1** In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane 1** and choose **Extrude**.
- 2** Select the object **wpl.r3** only.
- 3** In the **Extrude** settings window, locate the **Distances from Plane** section.
- 4** In the table, enter the following settings:

Distances (m)
Thickness

- 5** Select the **Reverse direction** check box.
- 6** Click the **Build Selected** button.

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



The unoptimized geometry of the bow-tie antenna

Sphere 1

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Sphere**.
- 2 In the **Sphere** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type **L0**.
- 4 Click to expand the **Layers** section. Click the **Build Selected** button.
- 5 In the table, enter the following settings:

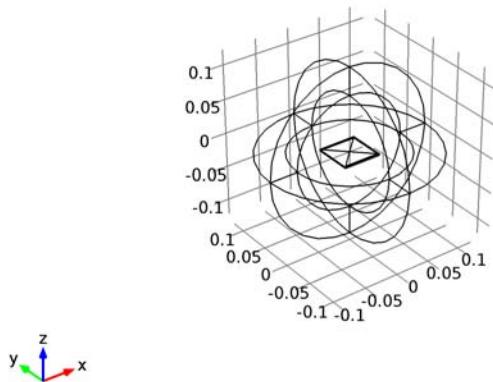
Thickness (m)
L0/3

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

Form Union

- 1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Form Union**.
- 2 In the **Settings** window, click **Build All**.

- 3 Click the **Wireframe Rendering** button on the Graphics toolbar.



The final geometry should look like that displayed in the figure above.

DEFINITIONS

Add a domain selection for the perfectly matched layer.

Explicit 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 Click **Paste Selection**.
- 4 Go to the **Paste Selection** dialog box.
- 5 In the **Selection** edit field, type **1-4,7-10**.
- 6 Click the **OK** button.
- 7 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.
- 8 Go to the **Rename Explicit** dialog box and type **PML** in the **New name** edit field.
- 9 Click **OK**.

Add the perfectly matched layer feature on the exterior domains.

Perfectly Matched Layer 1

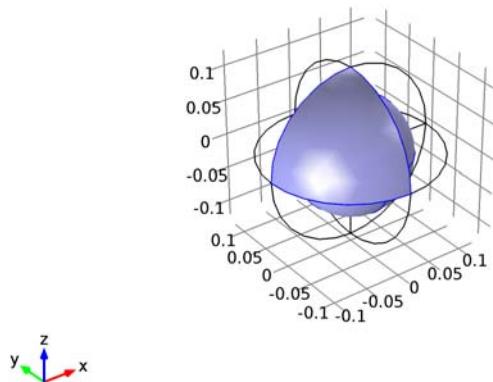
- 1 Right-click **Definitions** and choose **Perfectly Matched Layer**.

- 2 In the **Perfectly Matched Layer** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **PML**.
- 4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

View 1

Hide few outer domains to view the inner part of the model domain.

- 1 In the **Model Builder** window, under **Model 1 >Definitions** right-click **View 1** and choose **Hide Geometric Entities**.
- 2 Select Domains 2 and 5 only.



Now, set up the Electromagnetic Waves physics.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Perfect Electric Conductor 2

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.
- 2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.
- 3 Click **Paste Selection**.
- 4 Go to the **Paste Selection** dialog box.
- 5 In the **Selection** edit field, type **18, 19**.

- 6 Click the **OK** button.

Lumped Port 1

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 In the **Lumped Port** settings window, locate the **Boundary Selection** section.
- 3 Click **Paste Selection**.
- 4 Go to the **Paste Selection** dialog box.
- 5 In the **Selection** edit field, type 20-22,34.
- 6 Click the **OK** button.
- 7 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 8 From the **Wave excitation at this port** list, choose **On**.

MATERIALS

Material Browser

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
- 2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.
- 3 Click **Add Material to Model**.
- 4 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
- 5 In the **Material Browser** settings window, In the tree, select **Built-In>FR4 (Circuit Board)**.
- 6 Click **Add Material to Model**.

FR4 (Circuit Board)

- 1 In the **Model Builder** window, under **Model 1>Materials** click **FR4 (Circuit Board)**.
- 2 Select Domain 6 only.

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Size**.

Size

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.

- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type L0/5.
- 5 In the **Minimum element size** edit field, type 0.0198.
- 6 In the **Maximum element growth rate** edit field, type 1.85.
- 7 In the **Resolution of curvature** edit field, type 0.9.
- 8 In the **Resolution of narrow regions** edit field, type 0.2.

Size I

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size I**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 6 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type $L0/5/\sqrt{4.5}$.
- 8 Select the **Minimum element size** check box.
- 9 In the associated edit field, type 2[mm].

Free Tetrahedral I

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Click **Paste Selection**.
- 5 Go to the **Paste Selection** dialog box.
- 6 In the **Selection** edit field, type 5-6.
- 7 Click the **OK** button.

Swept I

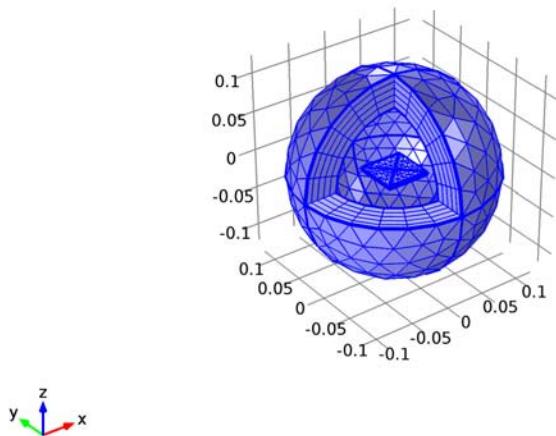
Right-click **Mesh 1** and choose **Swept**.

Distribution I

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

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

Compare the mesh with the figure shown below.



STUDY I

Add the optimization study node and assign the initial values, lower bounds, and upper bounds of the optimizing parameters. The objective function is an absolute value of the reflection coefficient S_{11} .

Optimization

- 1 In the **Model Builder** window, right-click **Study I** and choose **Optimization**.

- 2 In the **Optimization** settings window, locate the **Objective Function** section.

- 3 In the table, enter the following settings:

Expression	Description
abs(mod1.emw.S11)	

- 4 Locate the **Control Variables and Parameters** section. Click **Add**.

- 5 Click **Add**.

- 6** In the table, enter the following settings:

Parameter names	Initial value	Lower bound	Upper bound
h0	0.03	0.01	0.03
theta	60[deg]	30[deg]	90[deg]

- 7** Click to expand the **Output While Solving** section. Clear the **Keep objective values in table** check box.

Step 1: Frequency Domain

- 1** In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2** In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3** In the **Frequencies** edit field, type f_0 .
- 4** In the **Model Builder** window, click **Study 1**.
- 5** In the **Study** settings window, locate the **Study Settings** section.
- 6** Clear the **Generate default plots** check box.
- 7** Click the **Compute** button.

RESULTS

Data Sets

Create a data sets to visualize the results only in an antenna domain.

- 1** In the **Model Builder** window, under **Results>Data Sets** right-click **Solution 1** and choose **Duplicate**.
- 2** Right-click **Results>Data Sets>Solution 2** and choose **Add Selection**.
- 3** In the **Selection** settings window, locate the **Geometric Entity Selection** section.
- 4** From the **Geometric entity level** list, choose **Domain**.
- 5** Select Domain 6 only.

Use the following instructions to reproduce [Figure 2](#).

3D Plot Group 1

- 1** In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2** In the **3D Plot Group** settings window, locate the **Data** section.
- 3** From the **Data set** list, choose **Solution 2**.
- 4** Right-click **Results>3D Plot Group 1** and choose **Volume**.
- 5** Right-click **Results>3D Plot Group 1>Volume 1** and choose **Plot**.

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

Calculate the impedance of the lumped port, optimized parameters (height of an antenna arm and flare angle), and the absolute value of the reflection coefficient S_{11} using Global Evaluation nodes.

Derived Values

- I In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.
- 2 In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>Lumped port impedance (emw.Zport_I)**.
- 3 Click the **Evaluate** button.
- 4 In the **Model Builder** window, right-click **Derived Values** and choose **Global Evaluation**.
- 5 In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Height of antenna arm (h0)**.
- 6 Right-click **Results>Derived Values>Global Evaluation 2** and choose **Evaluate>Table I - Global Evaluation I (emw.Zport_I)**.
- 7 Right-click **Derived Values** and choose **Global Evaluation**.
- 8 In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Flare angle (theta)**.
- 9 In the **Global Evaluation** settings window, locate the **Expression** section.
- 10 From the **Unit** list, choose $^{\circ}$.
- II Right-click **Results>Derived Values>Global Evaluation 3** and choose **Evaluate>Table I - Global Evaluation I (emw.Zport_I)**.
- I2 Right-click **Derived Values** and choose **Global Evaluation**.
- I3 In the **Global Evaluation** settings window, locate the **Expression** section.
- I4 In the **Expression** edit field, type `abs(emw.S11)`.
- I5 Right-click **Results>Derived Values>Global Evaluation 4** and choose **Evaluate>Table I - Global Evaluation I (emw.Zport_I)**.

Branch-Line Coupler

Introduction

A branch line coupler, also known as 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.

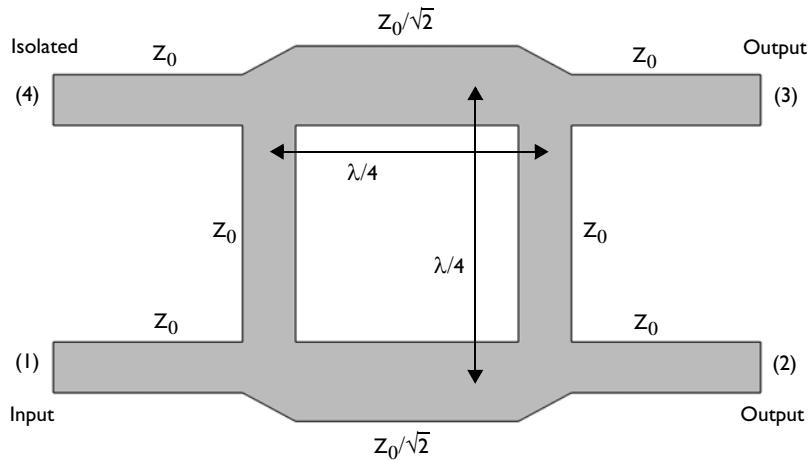


Figure 1: The geometry of a branch line coupler is symmetric.

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 the represent the microstrip line at each port.

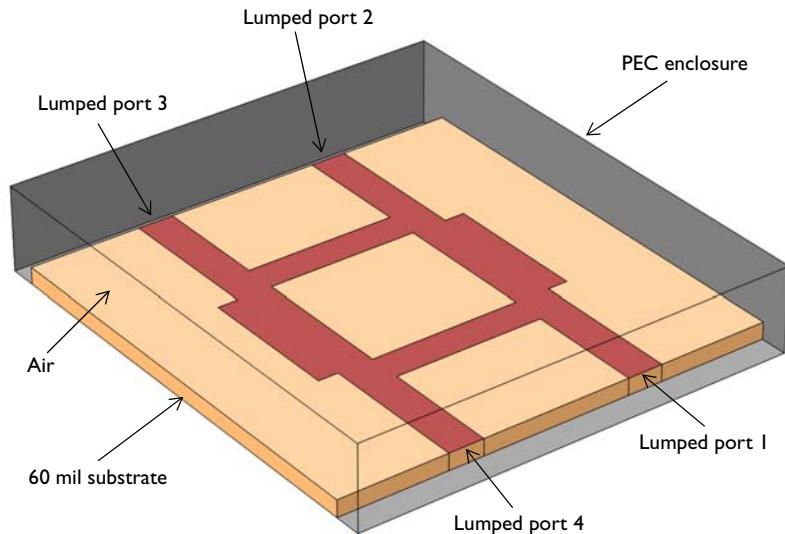


Figure 2: The model of the branch line coupler. Some exterior faces are removed for visualization.

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° .

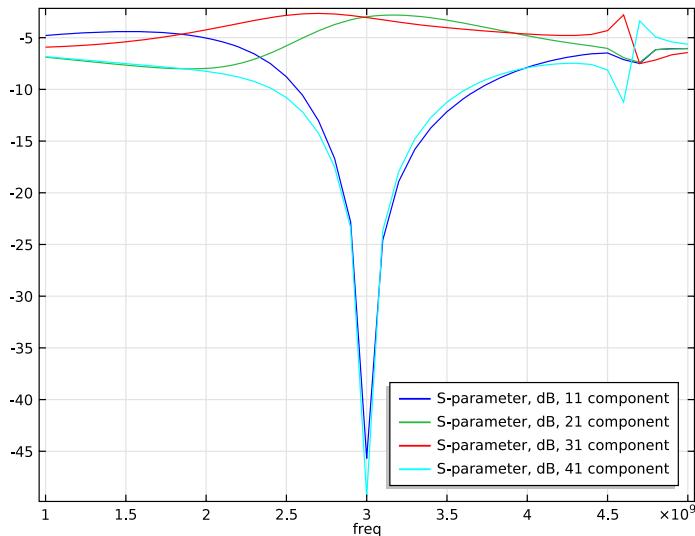


Figure 3: The frequency response of the branch line coupler shows good input matching (S_{11}) and isolation (S_{41}) around 3 GHz. The coupled signal at two output ports (S_{21} and S_{31}) is about -3 dB at 3 GHz.

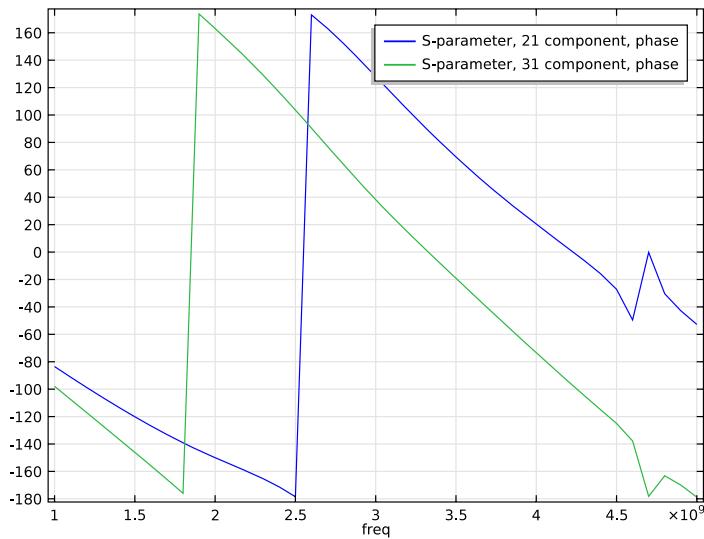


Figure 4: The phases on two output ports show approximately 90-degree shift at 3 GHz.

Because the metallic housing works as a rectangular cavity, there is a resonance observed around 4.6 GHz. This is the dominant TE₁₀₁ 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*, Wiley, 1998.
-

Model Library path: RF_Module/Passive_Devices/branch_line_coupler

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

- 3** In the table, enter the following settings:

Name	Expression	Description
thickness	60[mil]	Substrate thickness
l_s	40[mm]	Length, substrate
w_line2	5[mm]	Width, line 2
l_line2	13[mm]	Length, line 2
l_line1	(l_s-l_line2)/2	Length, line 1
w_line1	3.2[mm]	Width, line 1
w_line3	3[mm]	Width, line 3
l_line3	13.6[mm]	Length, line 3
f_min	1[GHz]	Minimum frequency in sweep
f_max	5[GHz]	Maximum frequency in sweep
lda_min	c_const/f_max	Minimum wavelength, air
h_max	0.2*lda_min	Maximum element size, air
h_s	h_max/sqrt(3.38)	Maximum element size, substrate

Here, 'mil' refers to the unit millinch.

GEOMETRY I

- In the **Model Builder** window, under **Model I** click **Geometry I**.
- In the **Geometry** settings window, locate the **Units** section.
- From the **Length unit** list, choose **mm**.
- Right-click **Model I>Geometry I** and choose **Work Plane** to add an *xy*-plane for the coupler layout.

Rectangle I

- In the **Model Builder** window, under **Model I>Geometry I>Work Plane I** right-click **Plane Geometry** and choose **Rectangle**.
- In the **Rectangle** settings window, locate the **Size** section.
- In the **Width** edit field, type $2*w_line1+l_line3$.
- In the **Height** edit field, type l_s .
- Locate the **Position** section. From the **Base** list, choose **Center**.
- Click the **Build Selected** button.

Rectangle 2

- 1** Right-click **Plane Geometry** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type $w_line2*2+l_line3$.
- 4** In the **Height** edit field, type l_line2 .
- 5** Locate the **Position** section. From the **Base** list, choose **Center**.
- 6** Click the **Build All** button.

Rectangle 3

- 1** Right-click **Plane Geometry** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type l_line3 .
- 4** In the **Height** edit field, type l_line2 .
- 5** Locate the **Position** section. In the **xw** edit field, type $-l_line3/2$.
- 6** In the **yw** edit field, type $l_line2/2+w_line3$.
- 7** Click the **Build Selected** button.
- 8** Click the **Zoom Extents** button on the Graphics toolbar.

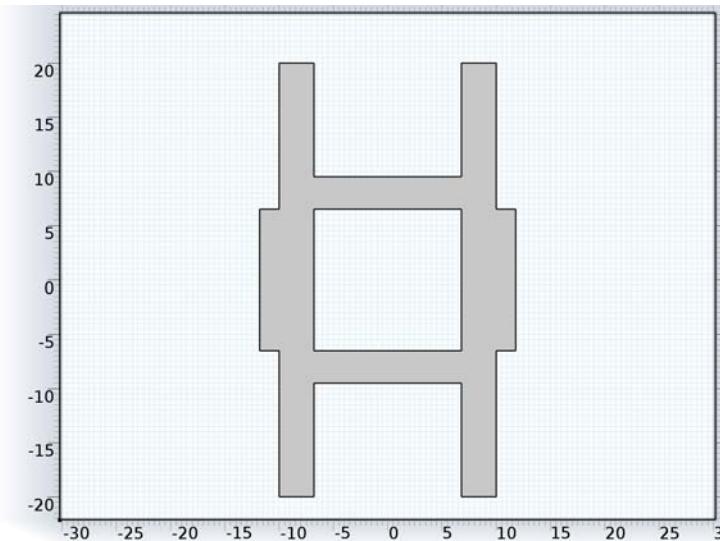
Array 1

- 1** Right-click **Plane Geometry** and choose **Transforms>Array**.
- 2** Select the object **r3** only.
- 3** In the **Array** settings window, locate the **Size** section.
- 4** From the **Array type** list, choose **Linear**.
- 5** In the **Size** edit field, type 3.
- 6** Locate the **Displacement** section. In the **yw** edit field, type $-l_line2-w_line3$.
- 7** Click the **Build All** button.

Difference 1

- 1** Right-click **Plane Geometry** and choose **Boolean Operations>Difference**.
- 2** Select the objects **r2** and **r1** only.
- 3** In the **Difference** settings window, locate the **Difference** section.
- 4** Under **Objects to subtract**, click **Activate Selection**.
- 5** Select the three rectangles belonging to the array object (**arr1**).
- 6** Clear the **Keep interior boundaries** check box.

- 7** Click the **Build All** button.



Work Plane I

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 I

- 1** In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane I** and choose **Extrude**.
- 2** In the **Extrude** settings window, locate the **Distances from Plane** section.
- 3** In the table, enter the following settings:

Distances (mm)
thickness

- 4** Click the **Build All** button.
- 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

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `l_s`.
- 4 In the **Depth** edit field, type `l_s`.
- 5 In the **Height** edit field, type `thickness`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **z** edit field, type `thickness/2`.
- 8 Click the **Build All** button.

Union 1

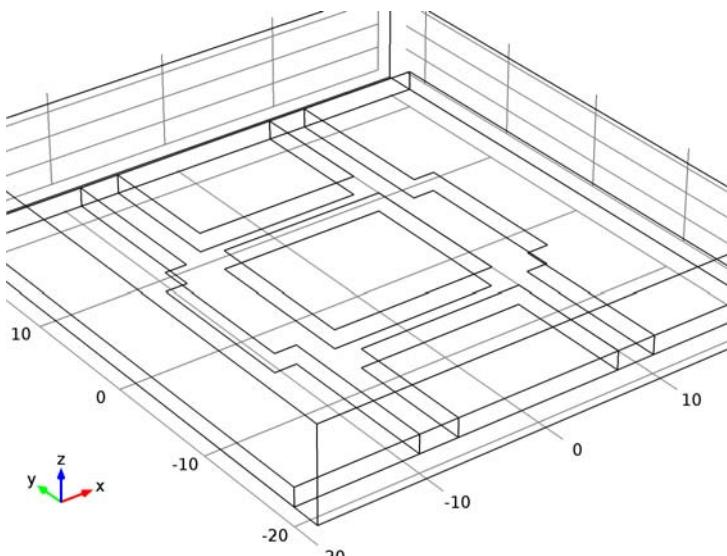
- 1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.
- 2 In the **Union** settings window, locate the **Union** section.
- 3 Clear the **Keep interior boundaries** check box.
- 4 Select the objects `ext1` and `blk1` only.
- 5 Locate the **Selections of Resulting Entities** section. Select the **Create selections** check box.
- 6 Click the **Build All** button.
- 7 Right-click **Model 1>Geometry 1>Union 1** and choose **Rename**.
- 8 Go to the **Rename Union** dialog box and type **Substrate** in the **New name** edit field.
- 9 Click **OK**.

Block 2

- 1 Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `l_s`.
- 4 In the **Depth** edit field, type `l_s+1_s/8`.
- 5 In the **Height** edit field, type `thickness*5`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **z** edit field, type `thickness*5/2`.
- 8 Click the **Build All** button.
- 9 Right-click **Model 1>Geometry 1>Block 2** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type **Package** in the **New name** edit field.

II Click **OK**.

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 I

- 1 In the **Model Builder** window, under **Model I** right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 13 only.
- 5 Right-click **Model I>Definitions>Explicit I** and choose **Rename**.
- 6 Go to the **Rename Explicit** dialog box and type **Microstrip Line** in the **New name** edit field.
- 7 Click **OK**.

View I

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

- 1 In the **Model Builder** window, under **Model 1 >Definitions** right-click **View 1** and choose **Hide Geometric Entities**.
- 2 In the **Hide Geometric Entities** settings window, 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

Now set up the physics. The default boundary condition is perfect electric conductor, which is applied to all exterior boundaries. Assign perfect electric conductor on the interior boundary on the microstrip lines. Choose Microstrip Line boundary from the selections already defined.

Perfect Electric Conductor 2

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.
- 2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Microstrip Line**.

Lumped Port 1

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 Select Boundary 24 only.
- 3 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4 From the **Wave excitation at this port** list, choose **On**.

Lumped Port 2

- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 Select Boundary 25 only.

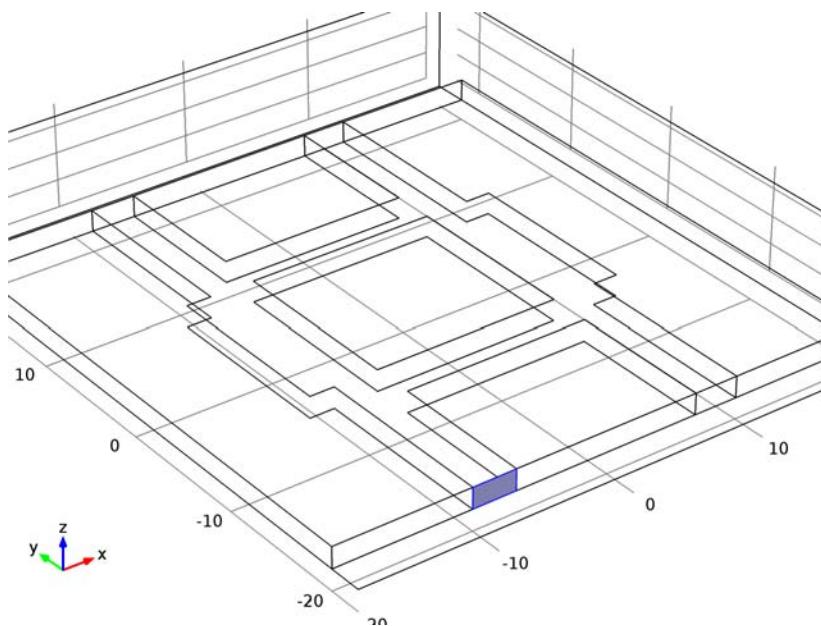
Lumped Port 3

- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 Select Boundary 15 only.

Lumped Port 4

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

- 2** Select Boundary 14 only.



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

MATERIALS

Assign material properties on the model. First, set all domains with the built-in air.

Material Browser

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

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

Material 2

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Selection** list, choose **Substrate**.

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

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

- 5** Right-click **Model 1>Materials>Material 2** and choose **Rename**.

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

- 7** Click **OK**.

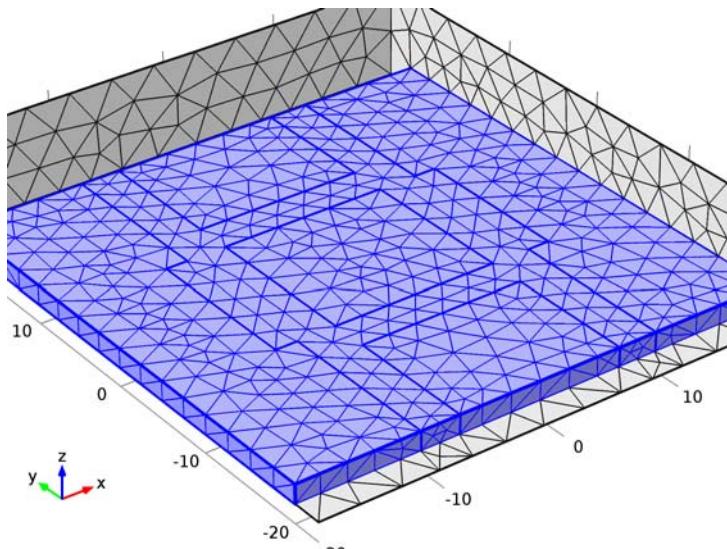
MESH 1

The maximum mesh sizes for air (h_{max}) and the substrate (h_s) are several times larger than the dimensions of the model. Therefore, you can use a global size setting determined by the geometry.

- 1** In the **Mesh** settings window, locate the **Mesh Settings** section.

- 2** From the **Element size** list, choose **Fine**.

- 3** Click the **Build All** button.



Three exterior boundaries are hidden in this view.

STUDY 1

Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type `range(f_min,0.1[GHz],f_max)`.
- 4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

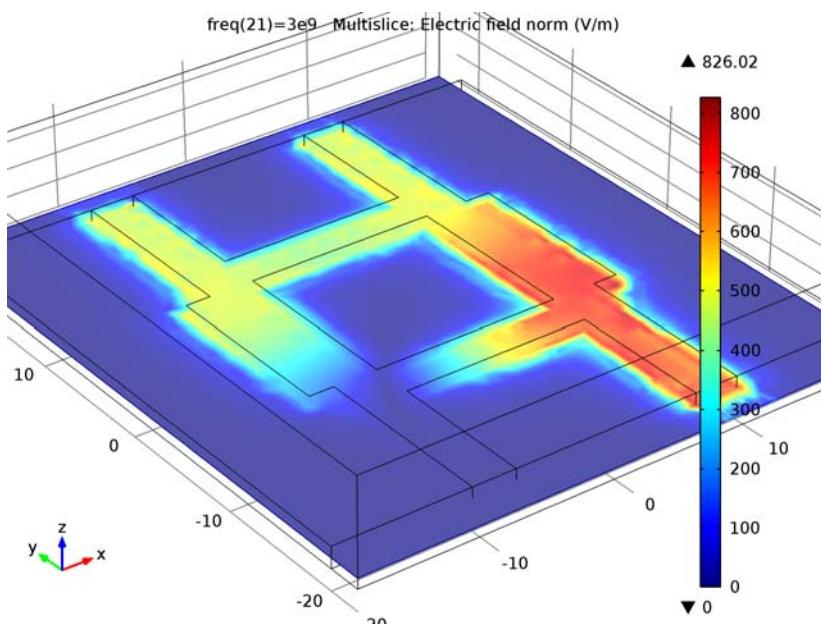
RESULTS

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 **3D Plot Group** settings window, locate the **Data** section.
- 2 From the **Parameter value (freq)** list, choose **3e9**.
- 3 In the **Model Builder** window, under **Results>Electric Field (emw)** click **Multislice**.
- 4 In the **Multislice** settings window, locate the **Multiplane Data** section.
- 5 Find the **x-planes** subsection. In the **Planes** edit field, type **0**.
- 6 Find the **y-planes** subsection. In the **Planes** edit field, type **0**.
- 7 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.
- 8 In the **Coordinates** edit field, type **thickness/2**.

- 9 Click the **Plot** button.



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

ID Plot Group 2

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, 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**.
- 5 Right-click **Results>ID Plot Group 2** and choose **Global**.
- 6 In the **Global** settings window, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 11 component (emw.S11dB)**.
- 7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 21 component (emw.S21dB)**.
- 8 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 31 component (emw.S31dB)**.

9 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 41 component (emw.S41dB)**.

10 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Expression** check box.

II Click the **Plot** button.

Compare the resulting plot with that shown in [Figure 3](#).

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

ID Plot Group 3

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

2 In the **ID Plot Group** settings window, locate the **Title** section.

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

4 Right-click **Results>ID Plot Group 3** and choose **Global**.

5 In the **Global** settings window, locate the **y-Axis Data** section.

6 In the table, enter the following settings:

Expression	Unit	Description
<code>arg(emw.S21)</code>	rad	S-parameter, 21 component, phase
<code>arg(emw.S31)</code>	rad	S-parameter, 31 component, phase

7 Click the **Plot** button.

Change the unit from radians to degrees.

8 In the table, enter the following settings:

Unit
deg
deg

9 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Expression** check box.

10 Click the **Plot** button.

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.

Derived Values

- 1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.
- 2 In the **Global Evaluation** settings window, locate the **Data** section.
- 3 From the **Parameter selection (freq)** list, choose **From list**.
- 4 In the **Parameter values (freq)** list, select **3e9**.
- 5 Locate the **Expression** section. In the **Expression** edit field, type `arg(emw.S21)-arg(emw.S31)`.
- 6 From the **Unit** list, choose \circ .
- 7 Click the **Evaluate** button.

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 will result 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

$$Q_{TE_{101}} = \frac{1,1107\eta}{R_s \left(1 + \frac{a}{2b}\right)}, f_{TE_{101}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}} \sqrt{\left(\frac{\pi}{a}\right)^2 + \left(\frac{\pi}{c}\right)^2} \quad (1)$$

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

$$Q_{TM_{010}} = \frac{1,2025\eta}{R_s \left(1 + \frac{a}{h}\right)}, f_{TM_{010}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}} \sqrt{\left(\frac{2,4049}{a}\right)^2} \quad (2)$$

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

$$Q_{TM_{011}} = \frac{1,004\eta}{R_s}, f_{TM_{011}} = \frac{2,744}{2\pi a \sqrt{\mu\varepsilon}} \quad (3)$$

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 I: 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.

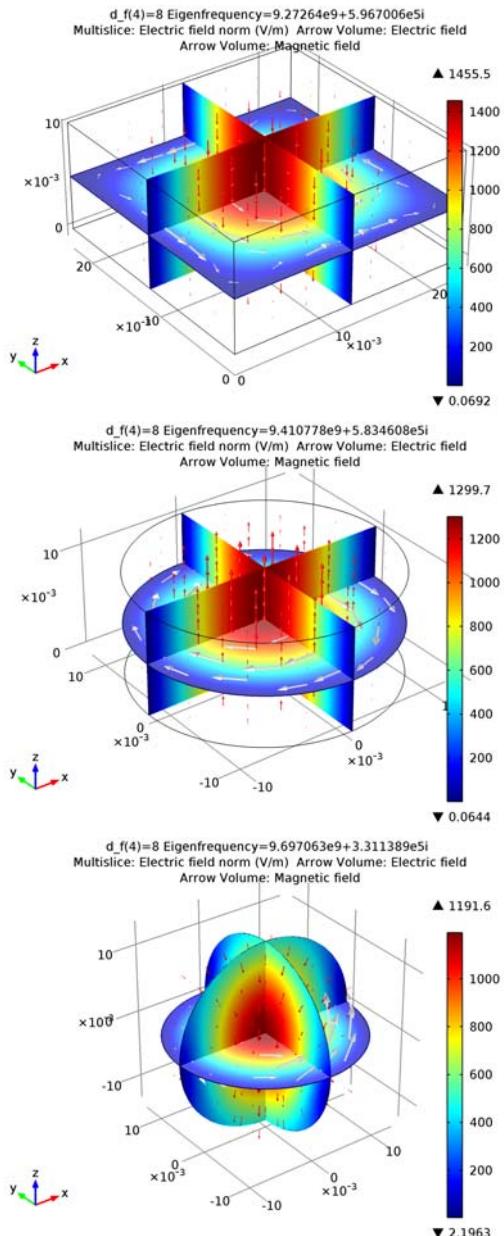


Figure 1: Arrow plots of electric and magnetic fields. Slice plot of electric field.

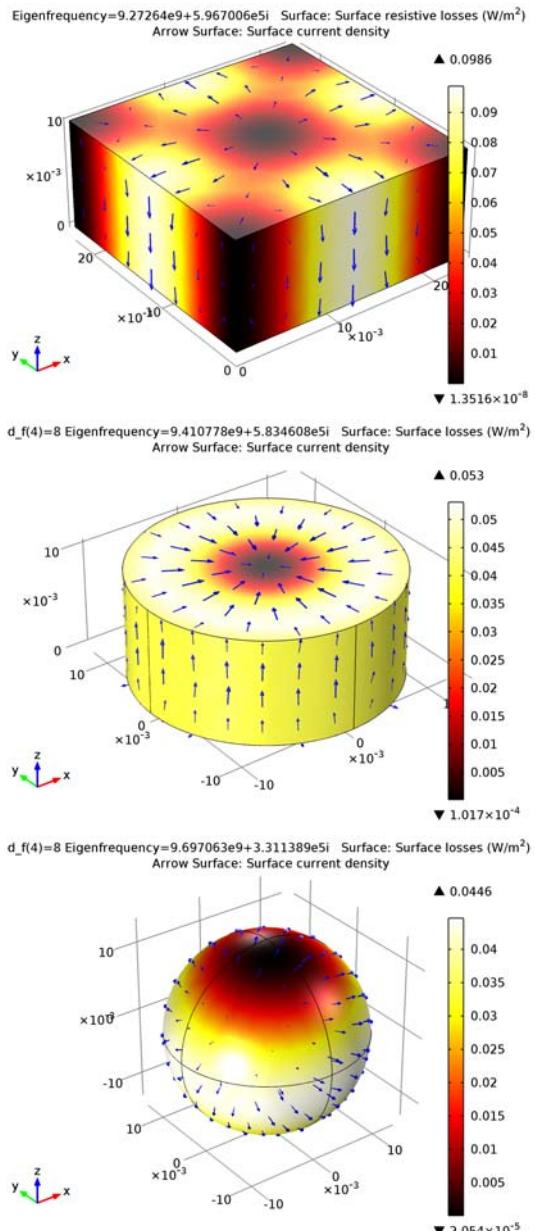


Figure 2: Arrow plots of surface currents. Surface plot of surface losses.

Notes About the COMSOL Implementation

Solve this model 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 solves a nonlinear eigenvalue problem. It is necessary to provide a frequency at which to initially evaluate the frequency-dependent surface losses. In the Eigenvalue Solver settings window you therefore need to specify the linearization point; see the modeling instructions for details.

Reference

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, Wiley, 1989.
-

Model Library path: RF_Module/Verification_Models/cavity_resonators

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Eigenfrequency**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 Click **Load from File**.

- 4 Browse to the model's Model Library 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

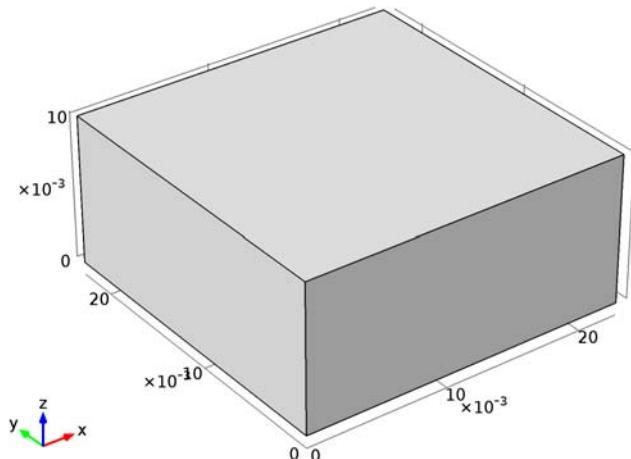
$f_{TE101_analytic_r} = 9.273$ GHz, $Q_{TE101_analytic_r} = 7770$ for the rectangular cavity, $f_{TM010_analytic_c} = 9.412$ GHz, $Q_{TM010_analytic_c} = 8065$ for the cylindrical cavity, $f_{TM011_analytic_s} = 9.698$ GHz, and $Q_{TM011_analytic_s} = 14594$ for the spherical cavity.

GEOMETRY I

Create a block for the rectangular cavity.

Block 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `a_r`.
- 4 In the **Depth** edit field, type `a_r`.
- 5 In the **Height** edit field, type `b_r`.
- 6 Click the **Build All** button.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Now set up the physics. Override the default perfect electric conductor condition on the exterior boundaries by an **Impedance Boundary Condition**.

Impedance Boundary Condition 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Impedance Boundary Condition**.
- 2 In the **Impedance Boundary Condition** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the model. First, set all domains with vacuum.

Material 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon0	1
Relative permeability	mu0	1
Electrical conductivity	sigma	0

- 4 Right-click **Model 1>Materials>Material 1** and choose **Rename**.
- 5 Go to the **Rename Material** dialog box and type **Vacuum** in the **New name** edit field.
- 6 Click **OK**.

Define a lossy conductive material for all exterior boundaries.

Material 2

- 1 Right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **All boundaries**.

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

Property	Name	Value
Relative permittivity	epsilon_r	1
Relative permeability	mu_r	1
Electrical conductivity	sigma	sigma_wall

6 Right-click **Model 1>Materials>Material 2** and choose **Rename**.

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

8 Click **OK**.

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

- 1** In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Model Couplings>Integration**.
- 2** In the **Integration** settings window, locate the **Operator Name** section.
- 3** In the **Operator name** edit field, type **int_v**.
- 4** Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 2

- 1** In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.
- 2** In the **Integration** settings window, locate the **Operator Name** section.
- 3** In the **Operator name** edit field, type **int_s**.
- 4** Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 5** From the **Selection** list, choose **All boundaries**.

Variables 1

- 1** Right-click **Definitions** and choose **Variables**.
- 2** In the **Variables** settings window, locate the **Variables** section.
- 3** Click **Load from File**.

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

The `emw.` prefix is for the physics interface, **Electromagnetic Waves, Frequency Domain** in the first model. `Wav` and `Qsh` are Energy density time average and Surface losses, respectively.

MESH I

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

- 1 In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **Free Tetrahedral**.

Size

- 1 In the **Model Builder** window, under **Model I>Mesh I** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `h_max_r/d_f`.
- 5 In the **Maximum element growth rate** edit field, type 2.
- 6 In the **Resolution of curvature** edit field, type 1.
- 7 In the **Resolution of narrow regions** edit field, type 0.1.
- 8 Click the **Build All** button.

STUDY I

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 I** node, then click **Step I: Eigenfrequency**.
- 2 In the **Eigenfrequency** settings window, locate the **Study Settings** section.
- 3 In the **Desired number of eigenfrequencies** edit field, type 1.
- 4 In the **Search for eigenfrequencies around** edit field, type `9e9`.

For the nonlinear eigenvalue problem, it is necessary to specify the linearization point for convergence.

Solver 1

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.
- 2 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1** click **Eigenvalue Solver 1**.
- 3 In the **Eigenvalue Solver** settings window, locate the **Values of Linearization Point** section.
- 4 Find the **Value of eigenvalue linearization point** subsection. In the **Point** edit field, type **9e9**.

Add a Parametric Sweep as a function of the discretization factor, **d_f**.

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names	Parameter value list
d_f	1 2 4 8

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

- 1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Volume**.
- 2 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field (emw.Ex,emw.Ey,emw.Ez)**.
- 3 Click the **Plot** button.
- 4 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Arrow Volume**.
- 5 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field (emw.Hx,emw.Hy,emw.Hz)**.

6 Locate the **Arrow Positioning** section. Find the **z grid points** subsection. In the **Points** edit field, type 1.

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

8 Click the **Plot** button.

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

Compare the resulting plot with that shown in [Figure 1](#), top.

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

3D Plot Group 2

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

2 Right-click **3D Plot Group 2** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Heating and losses>Surface resistive losses (emw.Qsrh)**.

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

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

6 In the **Arrow Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Currents and charge>Surface current density (emw.Jsx,...,emw.Jsz)**.

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

8 Click the **Plot** button.

9 Right-click **3D Plot Group 2** and choose **Rename**.

10 Go to the **Rename 3D Plot Group** dialog box and type **Surface Losses (emw)** in the **New name** edit field.

II Click **OK**.

ROOT

Next, set up a model for the cylindrical cavity.

I In the **Model Builder** window, right-click the root node and choose **Add Model**.

MODEL WIZARD

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

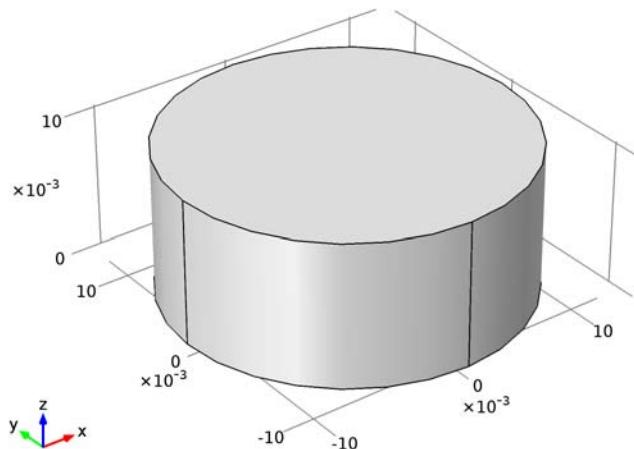
2 Click **Next**.

- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Custom Studies>Empty Study**.
- 6 Click **Finish**.

GEOOMETRY 2

Cylinder 1

- 1 In the **Model Builder** window, under **Model 2** right-click **Geometry 2** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type **a_c**.
- 4 In the **Height** edit field, type **height_c**.
- 5 Click the **Build All** button.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2

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 **Model 2** right-click **Electromagnetic Waves, Frequency Domain 2** and choose **Impedance Boundary Condition**.

- 2 In the **Impedance Boundary Condition** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the second model. Set all domains with vacuum.

Material 3

- 1 In the **Model Builder** window, under **Model 2** right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	1
Relative permeability	mu_r	1
Electrical conductivity	sigma	0

- 4 Right-click **Model 2>Materials>Material 3** and choose **Rename**.
- 5 Go to the **Rename Material** dialog box and type **Vacuum** in the **New name** edit field.
- 6 Click **OK**.

Set all exterior boundaries with a lossy conductive material.

Material 4

- 1 Right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **All boundaries**.
- 5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	1
Relative permeability	mu_r	1
Electrical conductivity	sigma	sigma_wall

- 6 Right-click **Model 2>Materials>Material 4** and choose **Rename**.
- 7 Go to the **Rename Material** dialog box and type **Lossy Wall** in the **New name** edit field.

8 Click OK.

Add variables and two integration coupling operators. The purpose of these are same as what you have done for the first model.

DEFINITIONS

Integration 3

- 1** In the **Model Builder** window, under **Model 2** right-click **Definitions** and choose **Model Couplings>Integration**.
- 2** In the **Integration** settings window, locate the **Operator Name** section.
- 3** In the **Operator name** edit field, type **int_v**.
- 4** Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 4

- 1** In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.
- 2** In the **Integration** settings window, locate the **Operator Name** section.
- 3** In the **Operator name** edit field, type **int_s**.
- 4** Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 5** From the **Selection** list, choose **All boundaries**.

Variables 2a

- 1** Right-click **Definitions** and choose **Variables**.
- 2** In the **Variables** settings window, locate the **Variables** section.
- 3** Click **Load from File**.
- 4** Browse to the model's Model Library 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.

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

Size

- 1 In the **Model Builder** window, under **Model 2>Mesh 2** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type h_{max_c/d_f} .
- 5 In the **Maximum element growth rate** edit field, type 2.
- 6 In the **Resolution of curvature** edit field, type 1.
- 7 In the **Resolution of narrow regions** edit field, type 0.1.
- 8 Click the **Build All** button.

STUDY 2

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

Step 1: Eigenfrequency 1

- 1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Eigenfrequency 1**.
- 2 In the **Eigenfrequency** settings window, locate the **Physics and Variables Selection** section.
- 3 In the table, enter the following settings:

Physics	Solve for
Electromagnetic Waves, Frequency Domain (emw)	x
Electromagnetic Waves, Frequency Domain 2 (emw2)	✓

Solver 7

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Show Default Solver**.
- 2 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 7** click **Eigenvalue Solver 1**.
- 3 In the **Eigenvalue Solver** settings window, locate the **Values of Linearization Point** section.
- 4 Find the **Value of eigenvalue linearization point** subsection. In the **Point** edit field, type 9e9.
- 5 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Electric Field (emw2)

- 1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw2)** and choose **Arrow Volume**.
- 2 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 2>Electric>Electric field (emw2.Ex,emw2.Ey,emw2.Ez)**.
- 3 Click the **Plot** button.
- 4 In the **Model Builder** window, right-click **Electric Field (emw2)** and choose **Arrow Volume**.
- 5 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 2>Magnetic>Magnetic field (emw2.Hx,emw2.Hy,emw2.Hz)**.
- 6 Locate the **Arrow Positioning** section. Find the **z grid points** subsection. In the **Points** edit field, type 1.
- 7 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.
- 8 Click the **Plot** button.
- 9 Click the **Zoom Extents** button on the Graphics toolbar.

The plot should now look like that in [Figure 1](#), middle.

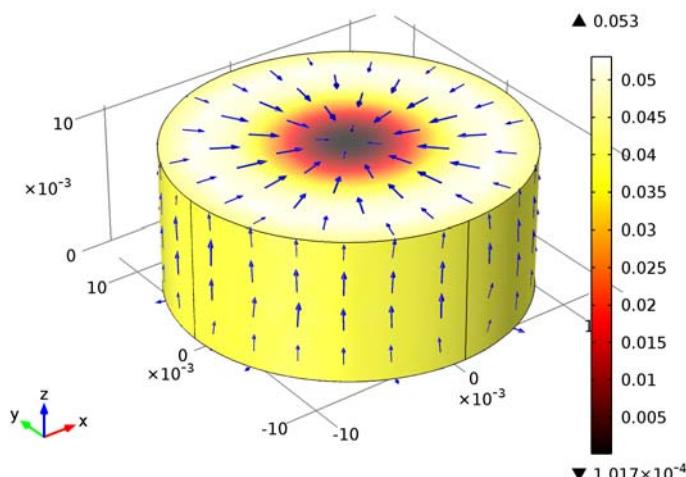
Again, add a surface plot of the surface losses and an arrow plot of the surface current ([Figure 2](#), middle).

3D Plot Group 4

- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Solution 6**.
- 4 Right-click **Results>3D Plot Group 4** and choose **Surface**.
- 5 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 2>Heating and losses>Surface losses (emw2.Qsh)**.
- 6 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.
- 7 In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Arrow Surface**.

- 8 In the **Arrow Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 2>Currents and charge>Surface current density (emw2.Jsx,...,emw2.Jsz)**.
- 9 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 10 Right-click **3D Plot Group 4** and choose **Rename**.
- 11 Go to the **Rename 3D Plot Group** dialog box and type **Surface Losses (emw2)** in the **New name** edit field.
- 12 Click **OK**.

$d_f(4)=8$ Eigenfrequency=9.410778e9+5.834608e5i Surface: Surface losses (W/m^2)
Arrow Surface: Surface current density



ROOT

Now add a model for the spherical cavity.

- 1 In the **Model Builder** window, right-click the root node and choose **Add Model**.

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Next**.

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

6 Click **Finish**.

GEOMETRY 3

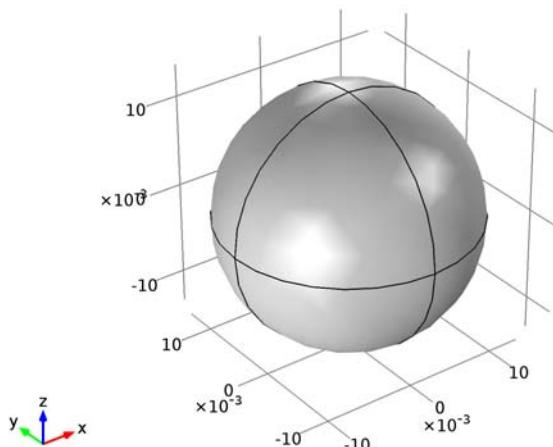
Sphere 1

1 In the **Model Builder** window, under **Model 3** right-click **Geometry 3** and choose **Sphere**.

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

3 In the **Radius** edit field, type a_s .

4 Click the **Build All** button.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 3

Set up the third physics interface.

Impedance Boundary Condition 1

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

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

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

MATERIALS

Assign material properties on the model. Set all domains with vacuum.

Material 5

- 1 In the **Model Builder** window, under **Model 3** right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	1
Relative permeability	mu_r	1
Electrical conductivity	sigma	0

- 4 Right-click **Model 3>Materials>Material 5** and choose **Rename**.
- 5 Go to the **Rename Material** dialog box and type Vacuum in the **New name** edit field.
- 6 Click **OK**.

Set all exterior boundaries with a lossy conductive material.

Material 6

- 1 Right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **All boundaries**.
- 5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	1
Relative permeability	mu_r	1
Electrical conductivity	sigma	sigma_wall

- 6 Right-click **Model 3>Materials>Material 6** and choose **Rename**.
- 7 Go to the **Rename Material** dialog box and type Lossy Wall in the **New name** edit field.
- 8 Click **OK**.

DEFINITIONS

Add variables and two integration coupling operators.

Integration 5

- 1 In the **Model Builder** window, under **Model 3** right-click **Definitions** and choose **Model Couplings>Integration**.
- 2 In the **Integration** settings window, locate the **Operator Name** section.
- 3 In the **Operator name** edit field, type `int_v`.
- 4 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 6

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.
- 2 In the **Integration** settings window, locate the **Operator Name** section.
- 3 In the **Operator name** edit field, type `int_s`.
- 4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 5 From the **Selection** list, choose **All boundaries**.

Variables 3a

- 1 Right-click **Definitions** and choose **Variables**.
- 2 In the **Variables** settings window, locate the **Variables** section.
- 3 Click **Load from File**.
- 4 Browse to the model's Model Library 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

In the **Model Builder** window, under **Model 3** right-click **Mesh 3** and choose **Free Tetrahedral**.

Size

- 1 In the **Model Builder** window, under **Model 3>Mesh 3** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `h_max_s/d_f`.
- 5 In the **Maximum element growth rate** edit field, type `2`.

- 6 In the **Resolution of curvature** edit field, type 1.
- 7 In the **Resolution of narrow regions** edit field, type 0.1.
- 8 Click the **Build All** button.

STUDY 3

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

Step 1: Eigenfrequency 1

- 1 In the **Model Builder** window, under **Study 3** click **Step 1: Eigenfrequency 1**.
- 2 In the **Eigenfrequency** settings window, locate the **Physics and Variables Selection** section.
- 3 In the table, enter the following settings:

Physics	Solve for
Electromagnetic Waves, Frequency Domain (emw)	x
Electromagnetic Waves, Frequency Domain 2 (emw2)	x
Electromagnetic Waves, Frequency Domain 3 (emw3)	✓

Solver 13

- 1 In the **Model Builder** window, right-click **Study 3** and choose **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Study 3>Solver Configurations>Solver 13** node, then click **Eigenvalue Solver 1**.
- 3 In the **Eigenvalue Solver** settings window, locate the **Values of Linearization Point** section.
- 4 Find the **Value of eigenvalue linearization point** subsection. In the **Point** edit field, type 9e9.
- 5 In the **Model Builder** window, right-click **Study 3** and choose **Compute**.

RESULTS

Electric Field (emw3)

- 1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw3)** and choose **Arrow Volume**.
- 2 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 3>Electric>Electric field (emw3.Ex,emw3.Ey,emw3.Ez)**.

- 3 In the **Model Builder** window, right-click **Electric Field (emw3)** and choose **Arrow Volume**.
- 4 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 3>Magnetic>Magnetic field (emw3.Hx,emw3.Hy,emw3.Hz)**.
- 5 Locate the **Arrow Positioning** section. Find the **z grid points** subsection. In the **Points** edit field, type 1.
- 6 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.
- 7 Click the **Plot** button.
- 8 Click the **Zoom Extents** button on the Graphics toolbar.

Compare the resulting plot with that shown in [Figure 1](#), bottom.

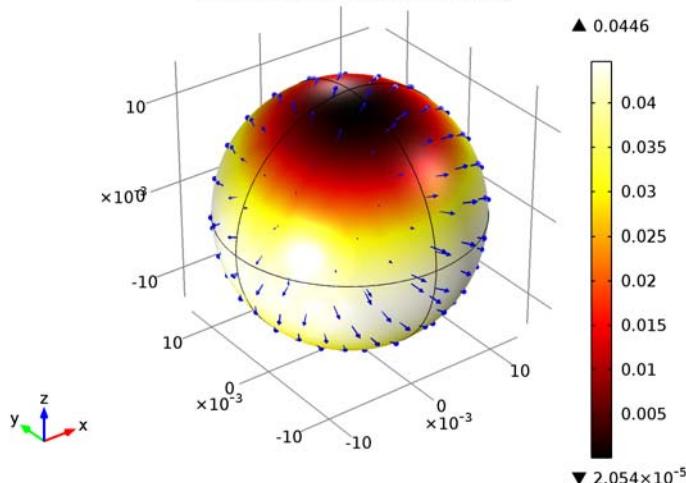
Again, add a surface plot of the surface losses and an arrow plot of the surface current ([Figure 2](#), bottom).

3D Plot Group 6

- I In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Solution 12**.
- 4 Right-click **Results>3D Plot Group 6** and choose **Surface**.
- 5 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 3>Heating and losses>Surface losses (emw3.Qsh)**.
- 6 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.
- 7 In the **Model Builder** window, right-click **3D Plot Group 6** and choose **Arrow Surface**.
- 8 In the **Arrow Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain 3>Currents and charge>Surface current density (emw3.Jsx,...,emw3.Jsz)**.
- 9 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 10 Right-click **3D Plot Group 6** and choose **Rename**.
- II Go to the **Rename 3D Plot Group** dialog box and type **Surface Losses (emw3)** in the **New name** edit field.

I2 Click OK.

$d_f(4)=8$ Eigenfrequency=9.697063e9+3.311389e5i Surface: Surface losses (W/m^2)
 Arrow Surface: Surface current density

**Derived Values**

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

- I** In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.
- 2** In the **Global Evaluation** settings window, locate the **Data** section.
- 3** From the **Data set** list, choose **Solution 2**.
- 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 **Expression** section. From the menu, choose **Definitions>Q-factor, computed from eigenvalue (Q_computed)**.
- 7** Click the **Evaluate** button.
- 8** In the **Global Evaluation** settings window, locate the **Data** section.
- 9** From the **Data set** list, choose **Solution 6**.
- 10** Click the **Evaluate** button.
- II** In the **Global Evaluation** settings window, locate the **Data** section.
- 12** From the **Data set** list, choose **Solution 12**.

- 13 Click the **Evaluate** button.
- 14 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 1** and choose **Duplicate**.
- 15 In the **Global Evaluation** settings window, locate the **Data** section.
- 16 From the **Data set** list, choose **Solution 2**.
- 17 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Q-factor, definition (Q_definition)**.
- 18 Right-click **Results>Derived Values>Global Evaluation 2** and choose **Evaluate>New Table**.
- 19 In the **Global Evaluation** settings window, locate the **Data** section.
- 20 From the **Data set** list, choose **Solution 6**.
- 21 Click the **Evaluate** button.
- 22 In the **Global Evaluation** settings window, locate the **Data** section.
- 23 From the **Data set** list, choose **Solution 12**.
- 24 Click the **Evaluate** button.
- 25 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 1** and choose **Duplicate**.
- 26 In the **Global Evaluation** settings window, locate the **Data** section.
- 27 From the **Data set** list, choose **Solution 2**.
- 28 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Frequency, simulated (frequency)**.
- 29 Right-click **Results>Derived Values>Global Evaluation 3** and choose **Evaluate>New Table**.
- 30 In the **Global Evaluation** settings window, locate the **Data** section.
- 31 From the **Data set** list, choose **Solution 6**.
- 32 Click the **Evaluate** button.
- 33 From the **Data set** list, choose **Solution 12**.
- 34 Click the **Evaluate** button.

Connecting a 3D Electromagnetic Wave Model to an Electrical Circuit

Introduction

A model 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.

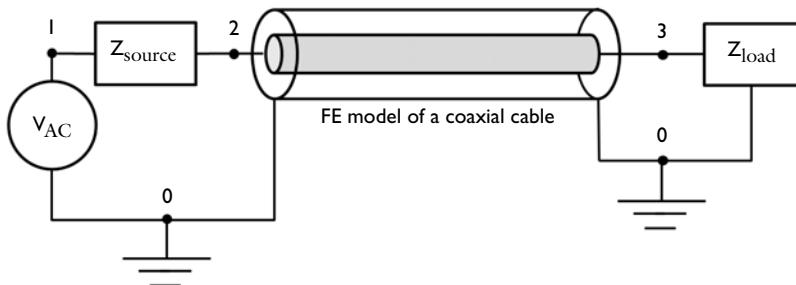


Figure 1: Schematic of a section of a coaxial transmission line connected to a voltage source, source impedance, and load.

Model Definition

The geometry in this model 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.

When solving such a model, some changes to the solver settings may be needed, since the default solver suggestions may not be appropriate. The appropriate way to solve this problem is with a Fully Coupled solver, using the default Iterative solver. Follow the Modeling Instructions for the solver settings.

Results and Discussion

[Figure 2](#) is a combined plot of the default electric field norm, magnetic field, and power flow.

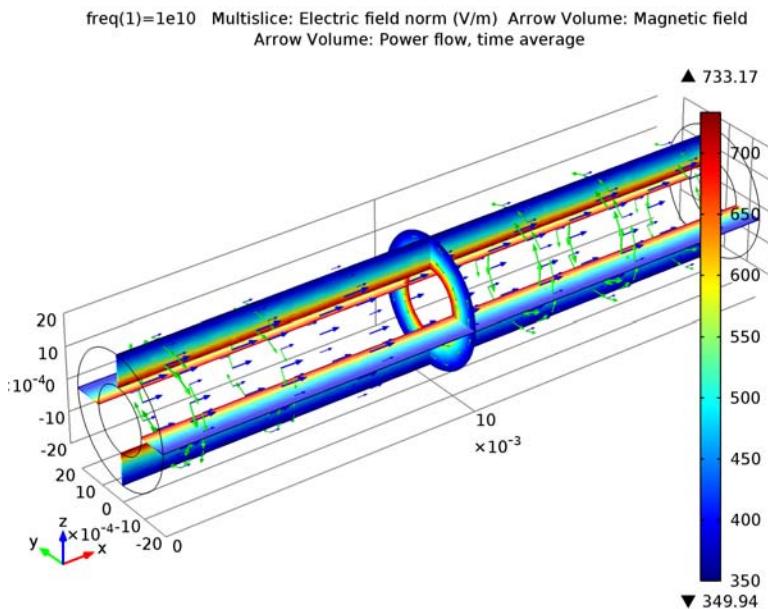


Figure 2: Electric field norm (multislices) and magnetic field, and power flow (green, blue arrows) inside the coaxial cable.

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

Model Library path: RF_Module/Transmission_Lines_and_Waveguides/
coaxial_cable_circuit

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 In the **Add physics** tree, select **AC/DC>Electrical Circuit (cir)**.
- 6 Click **Add Selected**.
- 7 Click **Next**.
- 8 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Frequency Domain**.
- 9 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
r_coax	1[mm]	Coax inner radius
R_coax	2[mm]	Coax outer radius
L_coax	20[mm]	Length of coax core into cavity
f0	10[GHz]	Frequency
lda0	c_const/f0	Wavelength, air
h_max	0.2*lda0	Maximum mesh element size, air
Z_coax	Z0_const/(2*pi)*log(R_coax/r_coax)	Analytical impedance

Here, c_{const} and Z_0_{const} are predefined COMSOL constants for the speed of the light and the wave impedance in vacuum, respectively.

GEOMETRY I

Create the geometry of the coaxial cable using two cylinders.

Cylinder 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type **R_coax**.
- 4 In the **Height** edit field, type **L_coax**.
- 5 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.
- 6 In the **x** edit field, type **1**.
- 7 In the **z** edit field, type **0**.
- 8 Right-click **Model 1>Geometry 1>Cylinder 1** and choose **Rename**.
- 9 Go to the **Rename Cylinder** dialog box and type **Coax_outer** in the **New name** edit field.
- 10 Click **OK**.

Cylinder 2

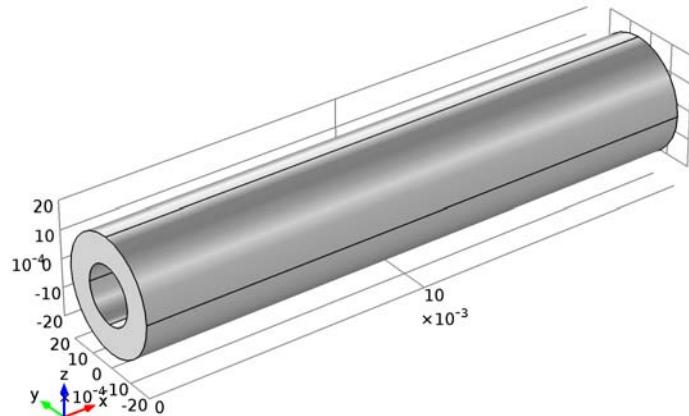
- 1 Right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type **r_coax**.
- 4 In the **Height** edit field, type **L_coax**.
- 5 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.
- 6 In the **x** edit field, type **1**.
- 7 In the **z** edit field, type **0**.
- 8 Right-click **Model 1>Geometry 1>Cylinder 2** and choose **Rename**.
- 9 Go to the **Rename Cylinder** dialog box and type **Coax_inner** in the **New name** edit field.
- 10 Click **OK**.

Difference 1

- 1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.
- 2 Select the object **cyl1** only.
- 3 In the **Difference** settings window, locate the **Difference** section.
- 4 Under **Objects to subtract**, click **Activate Selection**.

5 Select the object **cyl2** only.

6 Click the **Build All** button.



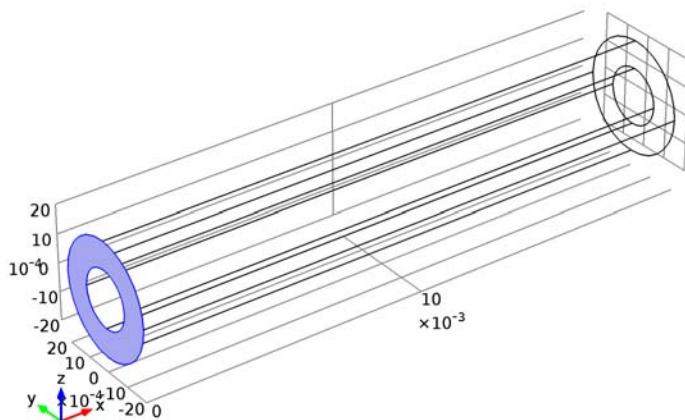
7 Click the **Wireframe Rendering** button on the Graphics toolbar.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Lumped Port 1

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

- 2 Select Boundary 1 only.



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

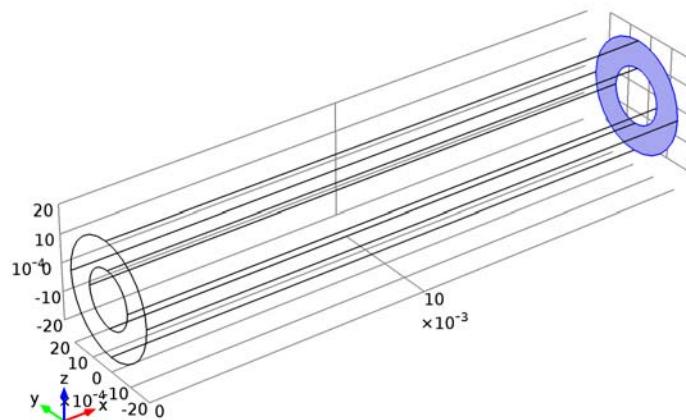
- 4 From the **Type of port** list, choose **Coaxial**.

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

Lumped Port 2

- I In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.

- 2** Select Boundary 10 only.



- 3** In the **Lumped Port** settings window, locate the **Port Properties** section.
4 From the **Type of port** list, choose **Coaxial**.
5 From the **Terminal type** list, choose **Circuit**.

ELECTRICAL CIRCUIT

Voltage Source 1

- 1** In the **Model Builder** window, under **Model 1** right-click **Electrical Circuit** and choose **Voltage Source**.
2 In the **Voltage Source** settings window, locate the **Device Parameters** section.
3 From the **Source type** list, choose **AC-source**.

Resistor 1

- 1** In the **Model Builder** window, right-click **Electrical Circuit** and choose **Resistor**.
2 In the **Resistor** settings window, 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 edit field, type `Z_coax`.

Resistor 2

- 1** Right-click **Electrical Circuit** and choose **Resistor**.
- 2** In the **Resistor** settings window, 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 edit field, type `Z_coax`.

ELECTRICAL CIRCUIT

External I Vs. U 1

- 1** In the **Model Builder** window, under **Model 1** right-click **Electrical Circuit** and choose **External I Vs. U**.
- 2** In the **External I Vs. U** settings window, 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 **Port voltage (emw/lport1)**.

External I Vs. U 2

- 1** In the **Model Builder** window, right-click **Electrical Circuit** and choose **External I Vs. U**.
- 2** In the **External I Vs. U** settings window, 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 **Port voltage (emw/lport2)**.

MATERIALS

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

Material Browser

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

MESH 1

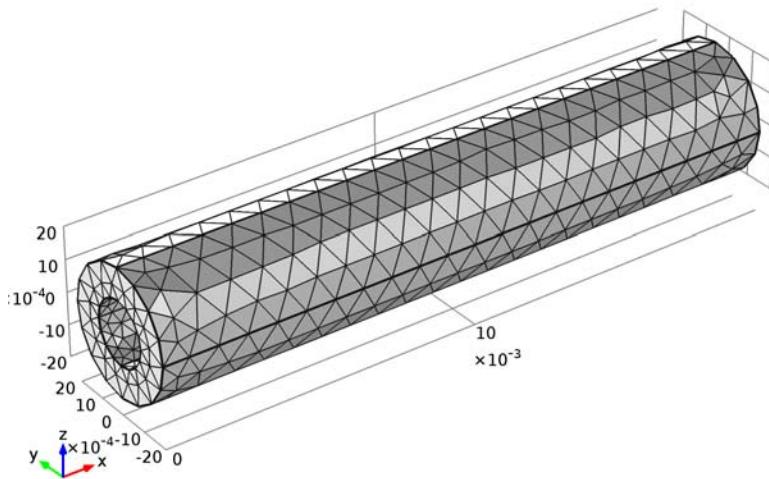
Choose the maximum mesh size in the air domain smaller than 0.2 wavelengths using the parameter **h_max**.

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

Size

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Fine**.
- 4 Click the **Custom** button.
- 5 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type **h_max**.

- 6 Click the **Build All** button.



STUDY I

Step 1: Frequency Domain

- 1 In the **Model Builder** window, expand the **Study I** node, then click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type f_0 .

Solver I

- 1 In the **Model Builder** window, right-click **Study I** and choose **Show Default Solver**.
- 2 Expand the **Solver I** node.
- 3 Right-click **Stationary Solver I** and choose **Fully Coupled**.
- 4 Right-click **Study I** and choose **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.

- 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 **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field (emw.Hx,emw.Hy,emw.Hz)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.
- 5 Right-click **Electric Field (emw)** and choose **Arrow Volume**.
- 6 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Energy and power>Power flow, time average (emw.Poavx,...,emw.Poavz)**.
- 7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

Compare the resulting plot with that shown in [Figure 2](#).

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.

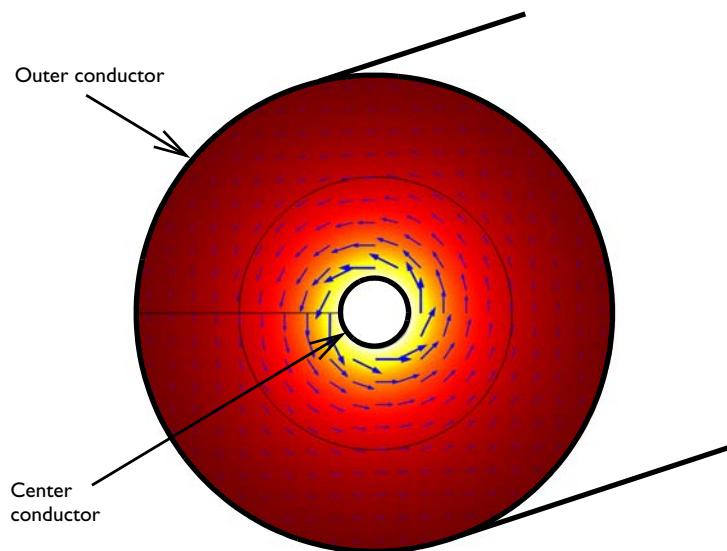


Figure 1: Cross section of a coaxial cable. The arrows visualize the magnetic field.

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 model, assume perfect conductors

and a lossless dielectric with relative permittivity $\epsilon_r = 2.4$. 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:

$$V = V_i - V_o = - \int_{r_o}^{r_i} \mathbf{E} \cdot d\mathbf{r} \quad (1)$$

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:

$$I = \oint_C \mathbf{H} \cdot d\mathbf{r} \quad (2)$$

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

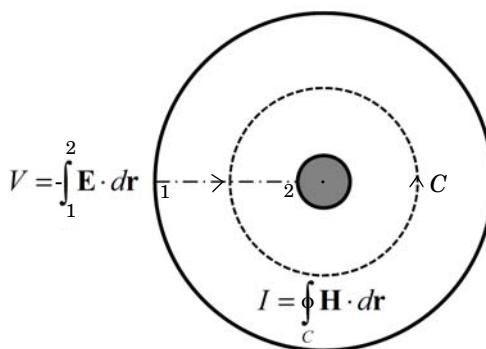


Figure 2: The impedance of a coaxial cable can be found from the voltage, V , and current, I , which are computed via line integrals as shown.

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

$$Z_{0,\text{analytic}} = \frac{1}{2\pi} \sqrt{\frac{\mu_0}{\epsilon_r \epsilon_0}} \log\left(\frac{r_o}{r_i}\right) \approx 74.5 \Omega$$

Results and Discussion

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

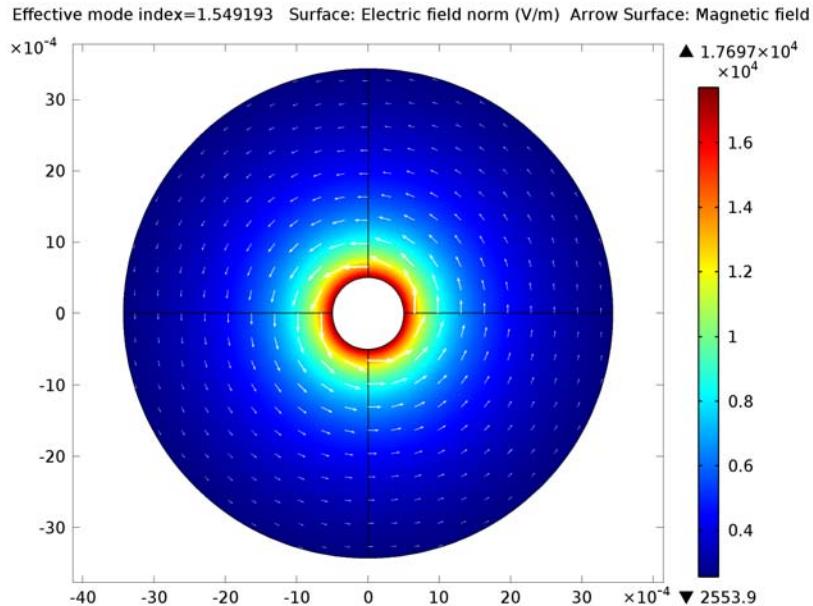


Figure 3: Electric field magnitude (surface) and magnetic field (arrows) inside the coaxial cable.

The impedance computed with the default mesh is $Z_0 = 74.65 \Omega$. As the mesh is refined, the result will approach the analytic value of 74.5Ω .

Notes About the COMSOL Implementation

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

Model Library path: RF_Module/Verification_Models/
coaxial_cable_impedance

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 5 Click **Add Selected**.
- 6 Click **Next**.
- 7 Find the **Studies** subsection. In the tree, select **Preset Studies>Mode Analysis**.
- 8 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
r_i	0.5[mm]	Coax inner radius
r_o	3.43[mm]	Coax outer radius
eps_r	2.4	Relative dielectric constant
Z0_analytic	(Z0_const/ (2*pi*sqrt(eps_r))) *log(r_o/r_i)	Characteristic impedance, analytic

Here $Z_0\text{const}$ is a predefined COMSOL constant for the characteristic impedance of vacuum, $Z_0 = \sqrt{\mu_0/\epsilon_0}$. From the Value column you can read off the value $Z_{0,\text{analytic}} = 74.53 \Omega$.

GEOMETRY I

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

Circle I

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Circle**.

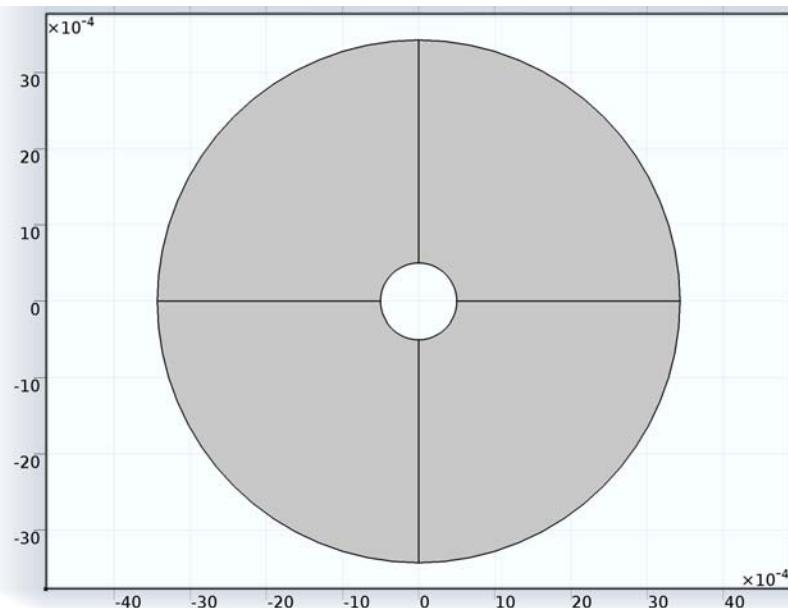
- 2** In the **Circle** settings window, locate the **Object Type** section.
- 3** From the **Type** list, choose **Curve**.
- 4** Locate the **Size and Shape** section. In the **Radius** edit field, type r_o .
- 5** Click to expand the **Layers** section. In the table, enter the following settings:

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

- 6** Click the **Build Selected** button.

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

- 1** In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2** In the **Material** settings window, locate the **Material Contents** section.

- 3** In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	eps_r
Relative permeability	mur	1
Electrical conductivity	sigma	0

- 4** Right-click **Model 1>Materials>Material 1** and choose **Rename**.
5 Go to the **Rename Material** dialog box and type **Insulator** in the **New name** edit field.
6 Click **OK**.

DEFINITIONS

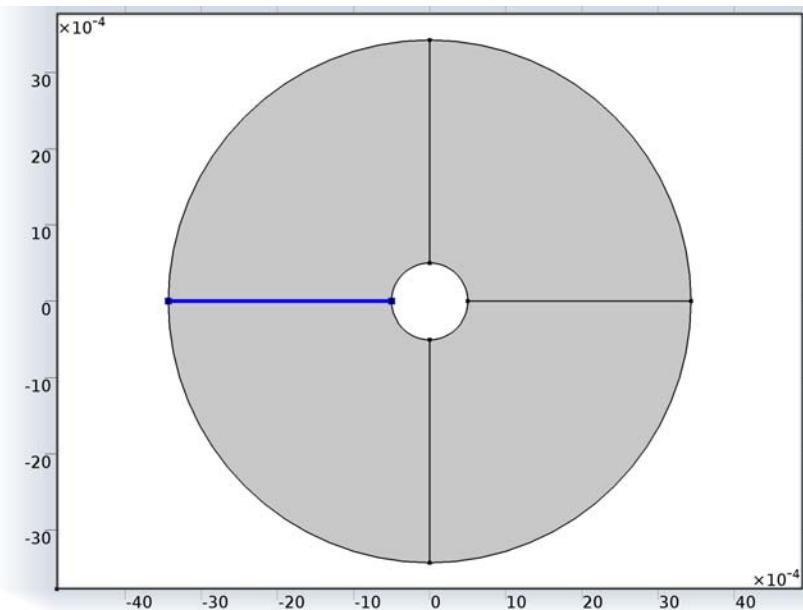
Add a variable for the characteristic impedance computed as the voltage between the conductors divided by the current through the cable. For this purpose, you need two integration coupling operators.

Integration 1

- In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.
- In the **Integration** settings window, locate the **Operator Name** section.
- In the **Operator name** edit field, type **int_rad**.
- Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 1 only.

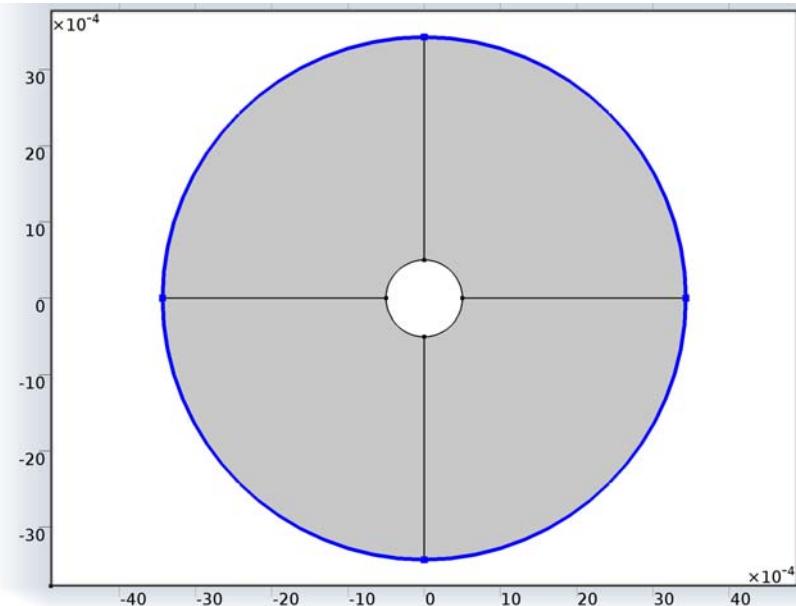
Any of the four interior boundaries that connect the two conductors would do.



Integration 2

- 1** Right-click **Definitions** and choose **Model Couplings>Integration**.
- 2** In the **Integration** settings window, locate the **Operator Name** section.
- 3** In the **Operator name** edit field, type `int_circ`.
- 4** Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

- 5** 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 I

- 1 Right-click **Definitions** and choose **Variables**.
- 2 In the **Variables** settings window, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
V	<code>int_rad(-emw.Ex*t1x-emw.Ey*t1y)</code>	Voltage
I	<code>-int_circ(emw.Hx*t1x+emw.Hy*t1y)</code>	Current
Z0_model	<code>V/I</code>	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.

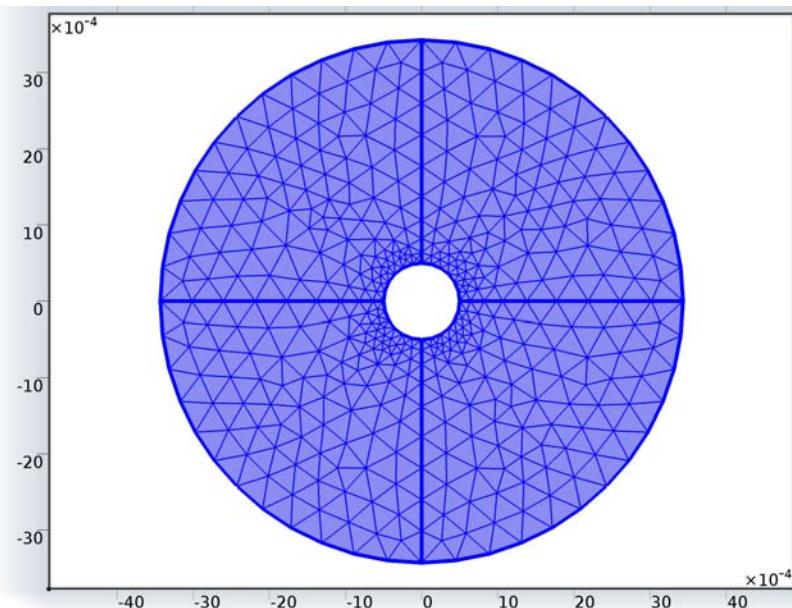
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

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

MESH I

Use the default mesh.

- I In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **Build All**.



STUDY I

Step 1: Mode Analysis

- I In the **Model Builder** window, under **Study I** click **Step 1: Mode Analysis**.
- 2 In the **Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Desired number of modes** edit field, type 1.
- 4 In the **Search for modes around** edit field, type `sqrt(eps_r)`.
- 5 In the **Model Builder** window, right-click **Study I** and choose **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.

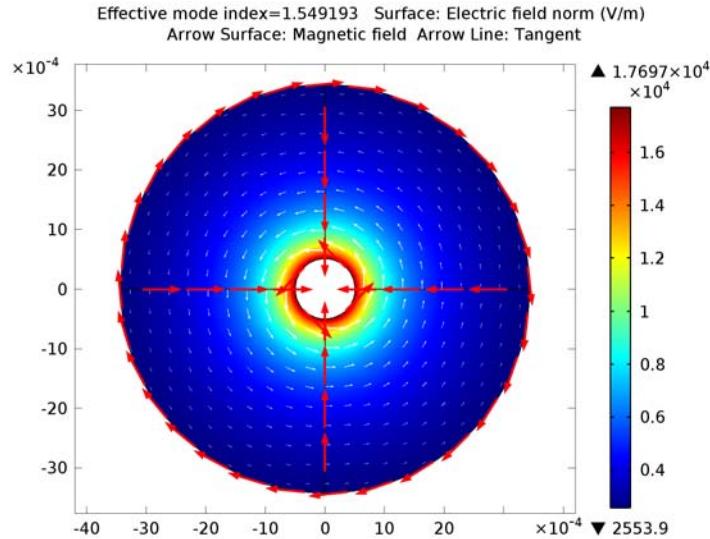
- 1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Surface**.
- 2 In the **Arrow Surface** settings window, locate the **Arrow Positioning** section.
- 3 Find the **x grid points** subsection. In the **Points** edit field, type 21.
- 4 Find the **y grid points** subsection. In the **Points** edit field, type 21.
- 5 Use the slider to adjust the arrow lengths.
- 6 From the **Color** list, choose **White**.
- 7 Click the **Plot** button.
- 8 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:

- 1 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Arrow Line**.
- 2 In the **Arrow Line** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Geometry>Tangent (tx,ty)**.
- 3 Locate the **Coloring and Style** section. In the **Number of arrows** edit field, type 50.

- 4** Click the **Plot** button.



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 I** and choose **Disable** to retrieve the result plot.

Finish by computing the characteristic impedance.

Derived Values

- 1** In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.
- 2** In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Characteristic impedance (Z0_model)**.
- 3** Click the **Evaluate** button.

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

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 model 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 depicted 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.

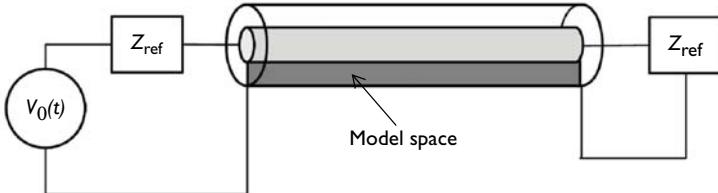


Figure 1: Schematic of a section of a coaxial transmission line connected to a transient voltage source and a load.

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 $I(t) = V_0(t) / Z_{\text{ref}}$ 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 $(0.37 - 0.10) / 2 \text{ ns} = 0.135 \text{ ns}$. This matches the expected value of L_{coax} / c , where $L_{\text{coax}} = 40 \text{ mm}$ is the length of the line and c is the speed of light in air.

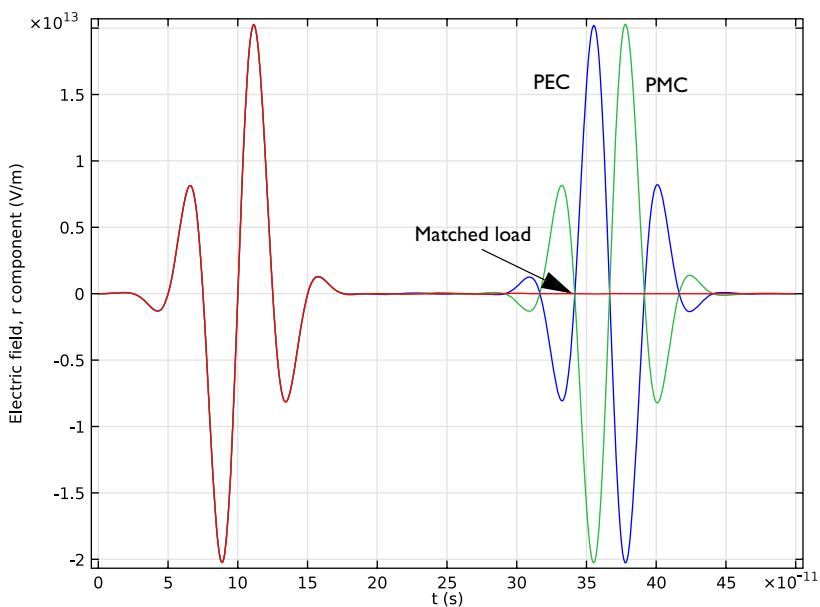


Figure 2: Radial component of electric field at the input port versus time for three different termination conditions: short (blue), open (green), and matched load (red).

Model Library path: RF_Module/Verification_Models/
coaxial_cable_transient

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D axisymmetric** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Transient (temw)**.
- 5 Click **Add Selected**.

6 Click **Next**.

7 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

8 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Description
r_coax	1[mm]	Coax inner radius
R_coax	2[mm]	Coax outer radius
L_coax	40[mm]	Length of coax core into cavity
f	20[GHz]	Pulse frequency
L	c_const/f	Wavelength, free space
T	1/f	Period
h_max	min(L/ 8,(R_coax-r_coax)/ 2)	Maximum element size

Next, define the excitation, $V_0(t)$, in terms of a Gaussian pulse and a sine function. Define a Gaussian pulse.

Gaussian Pulse I

1 Right-click **Global Definitions** and choose **Functions>Gaussian Pulse**.

2 In the **Gaussian Pulse** settings window, locate the **Function Name** section.

3 In the **Function name** edit field, type `gauss_pulse`.

4 Locate the **Parameters** section. In the **Location** edit field, type `2*T`.

5 In the **Standard deviation** edit field, type `T/2`.

Now use this pulse in an analytic function for $V_0(t)$:

Analytic I

1 Right-click **Global Definitions** and choose **Functions>Analytic**.

2 In the **Analytic** settings window, locate the **Function Name** section.

3 In the **Function name** edit field, type `V0`.

4 Locate the **Definition** section. In the **Expression** edit field, type `gauss_pulse(t)*sin(2*pi*f*t)`.

5 In the **Arguments** edit field, type `t`.

6 Locate the **Units** section. In the **Arguments** edit field, type `s`.

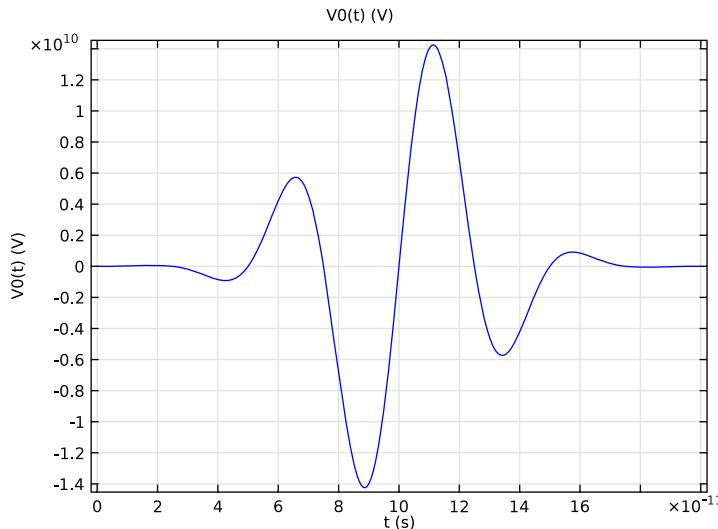
7 In the **Function** edit field, type `V`.

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

8 Click to expand the **Plot Parameters** section. In the table, enter the following settings:

Lower limit	Upper limit
0	0.2[ns]

9 Click to collapse the **Plot Parameters** section. Click the **Plot** button.



GEOMETRY I

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

Rectangle I

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

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

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

- 4 In the **Height** edit field, type `L_coax`.
- 5 Locate the **Position** section. In the `r` edit field, type `r_coax`.
- 6 Click the **Build All** button.

DEFINITIONS

Set up a point probe for plotting the electric field component E_r while solving. You will also use this plot to reproduce [Figure 2](#).

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Probes>Domain Point Probe**.
- 2 In the **Domain Point Probe** settings window, 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 **Model 1>Definitions>Domain Point Probe 1** node, then click **Point Probe Expression 1**.
- 6 In the **Point Probe Expression** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Transient>Electric>Electric field>Electric field, r component (temw.Er)**.

ELECTROMAGNETIC WAVES, TRANSIENT

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

Lumped Port 1

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Transient** and choose **Lumped Port**.
- 2 Select Boundary 2 only (the bottom boundary).
- 3 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4 From the **Wave excitation at this port** list, choose **On**.
- 5 In the `V0` edit field, type `V0(t)`.
- 6 Locate the **Settings** section. In the `Zref` edit 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 Right-click **Electromagnetic Waves, Transient** 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** and choose **Lumped Port**.
- 2 Select Boundary 3 only.
- 3 In the **Lumped Port** settings window, locate the **Settings** section.
- 4 In the Z_{ref} edit field, type $(Z_0_const/2/\pi)*\log(R_{coax}/r_{coax})$.

MATERIALS

Material Browser

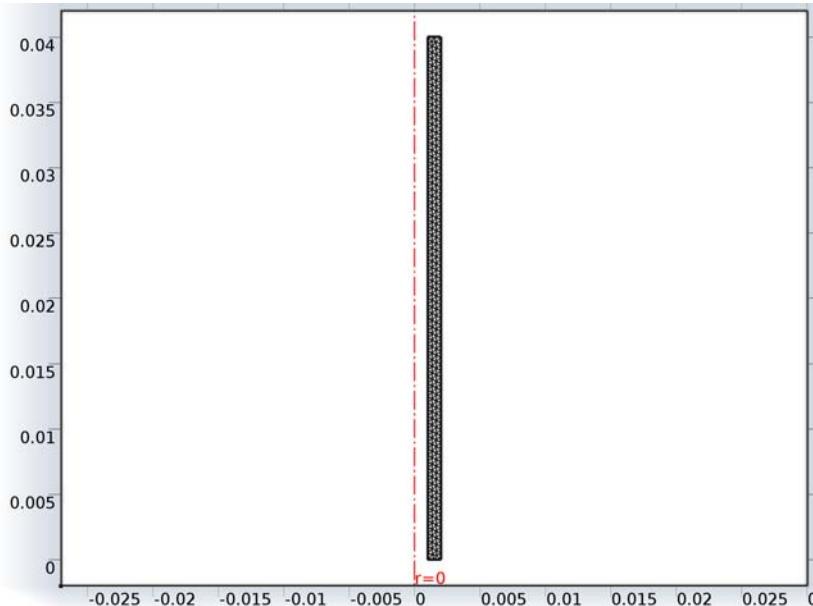
- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
- 2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.
- 3 Click **Add Material to Model**.

MESH 1

Size

- 1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Free Triangular**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type h_{max} .

- 5 Click the **Build All** button.



STUDY 1

Step 1: Time Dependent

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
 - 2 In the **Time Dependent** settings window, locate the **Study Settings** section.
 - 3 In the **Times** edit field, type range $(0, T/24, 10*T)$.
 - 4 Select the **Relative tolerance** check box.
 - 5 In the associated edit 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.
- 6 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.
 - 7 In the **Physics and Variables Selection** section, under **Model 1>Electromagnetic Waves, Transient**: Ctrl-click first **Perfect Magnetic Conductor 1** and then **Lumped Port 2** so both are selected. Then right-click and choose **Disable** (or use the **Disable** button).

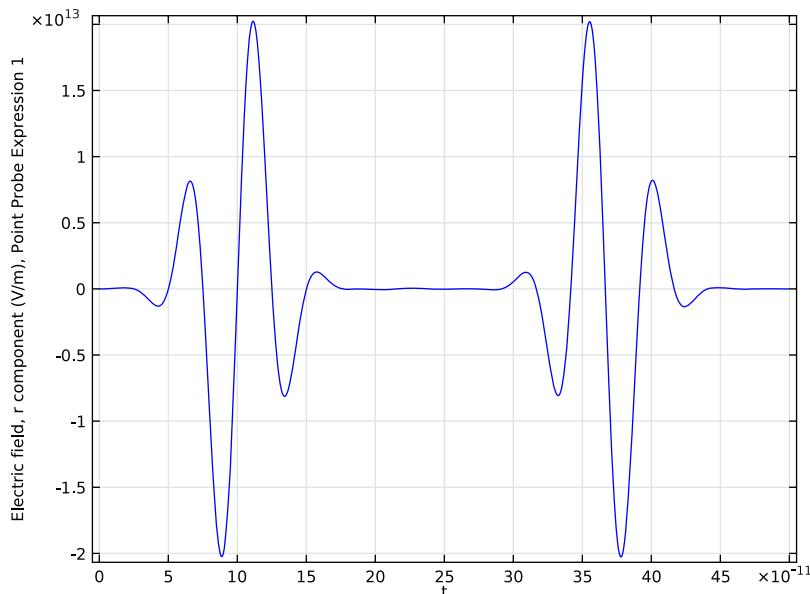
STUDY 1

I In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

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

RESULTS*Probe ID Plot Group 2*

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

*Electric Field*

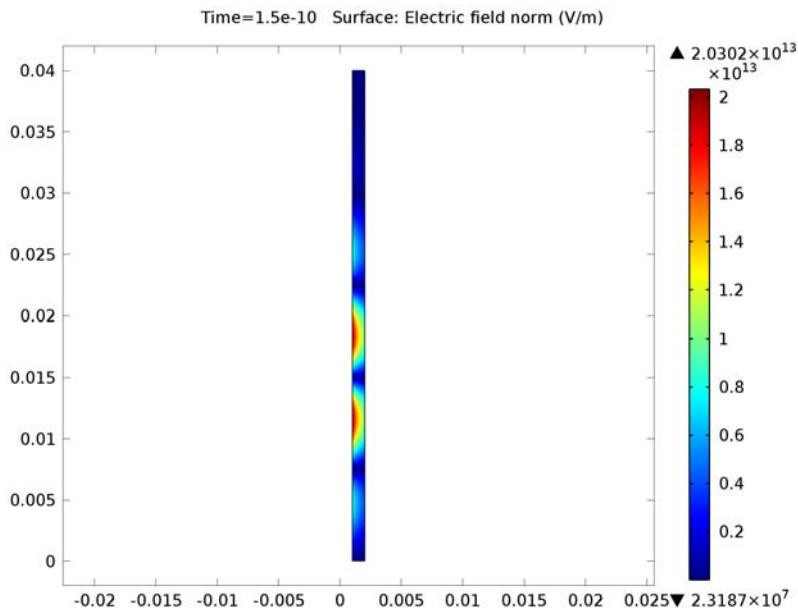
The default surface plot shows the electric field in the coax at the end of the simulation interval. Because the transient has died out, the solution you see is only noise. Modify the time to get a more interesting plot.

I In the **Model Builder** window, click **Electric Field**.

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

3 From the **Time** list, choose **1.5e-10**.

- 4** Click the **Plot** button.



Now turn to the open termination case.

DEFINITIONS

- I** In the **Model Builder** window, under **Model 1>Definitions>Domain Point Probe 1** click **Point Probe Expression 1**.
- 2** In the **Point Probe Expression** settings window, locate the **Table and Window Settings** section.
- 3** 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

Perfect Magnetic Conductor 1

In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Transient** right-click **Perfect Magnetic Conductor 1** and choose **Enable**.

STUDY 1

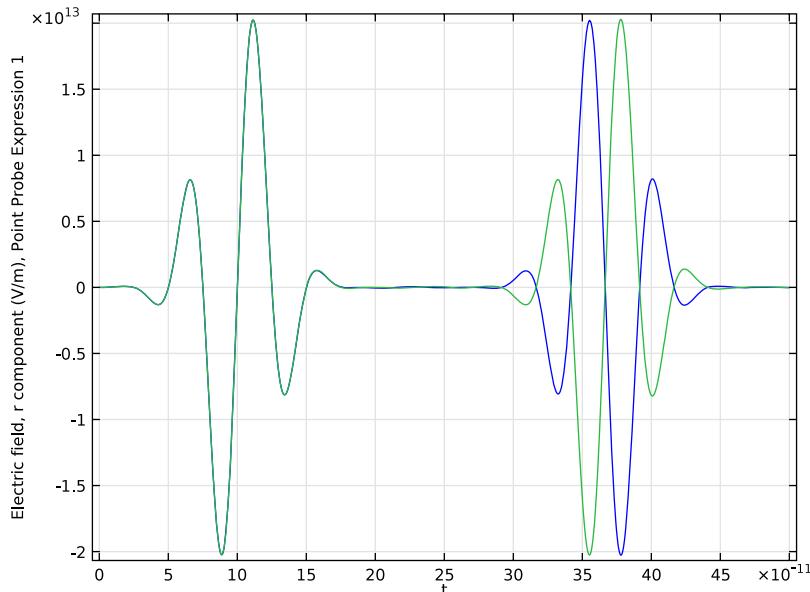
- I** In the **Model Builder** window, click **Study 1**.

- 2** In the **Study** settings window, locate the **Study Settings** section.
- 3** Clear the **Generate default plots** check box.
- 4** Click the **Compute** button.

RESULTS

Probe ID Plot Group 2

The reflected waves for the short and open terminations are 180° out of phase.



Finally, activate the matched load case.

DEFINITIONS

- 1** In the **Model Builder** window, under **Model 1>Definitions>Domain Point Probe 1** click **Point Probe Expression 1**.
- 2** In the **Point Probe Expression** settings window, click to expand the **Table and Window Settings** section.
- 3** From the **Output table** list, choose **New table**.

STUDY I

ELECTROMAGNETIC WAVES, TRANSIENT

Lumped Port 2

- 1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Transient** 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 I

In the **Model Builder** window, right-click **Study I** and choose **Compute**.

RESULTS

Probe ID Plot Group 2

- 1 In the **ID Plot Group** settings window, locate the **Plot Settings** section.
- 2 Select the **x-axis label** check box.
- 3 In the associated edit field, type **t (s)**.
- 4 Select the **y-axis label** check box.
- 5 In the associated edit field, type **Electric field, r component (V/m)**.
- 6 Click the **Plot** button.

The plot should now look like that in [Figure 2](#), with the red graph corresponding to the matched case.

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 $50\ \Omega$ 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.

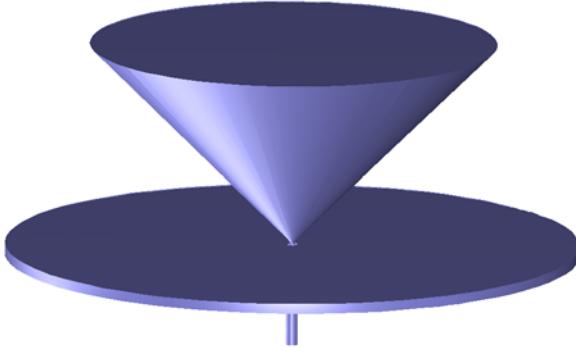


Figure 1: The geometry of the antenna. The central conductor of the coaxial cable is connected to the metallic cone, and the cable screen is connected to the finite ground plane.

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 :

$$\nabla \times \left(\frac{1}{\epsilon} \nabla \times H_\varphi \right) - \mu \omega^2 H_\varphi = 0$$

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, 50Ω , to

obtain maximum transmission into free space. This is quite well fulfilled in the high frequency range.

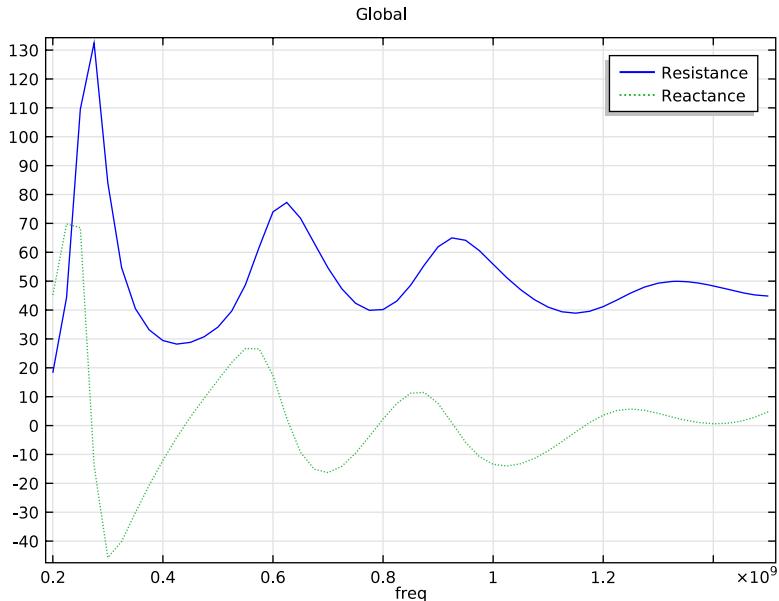


Figure 2: The antenna impedance in Ω as a function of frequency from 200 MHz to 1.5 GHz. The solid line shows the radiation resistance and the dotted line represents the reactance.

[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](#).

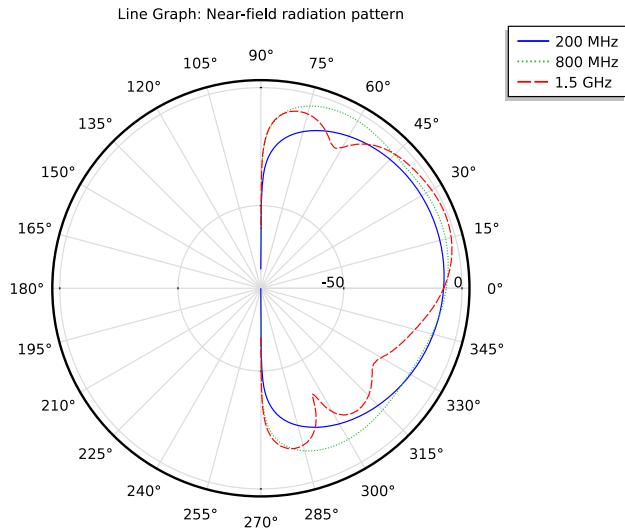


Figure 3: Polar plot of the antenna radiation pattern in the near field versus the elevation angle for 200 MHz, 800 MHz, and 1.5 GHz. The scale is logarithmic.

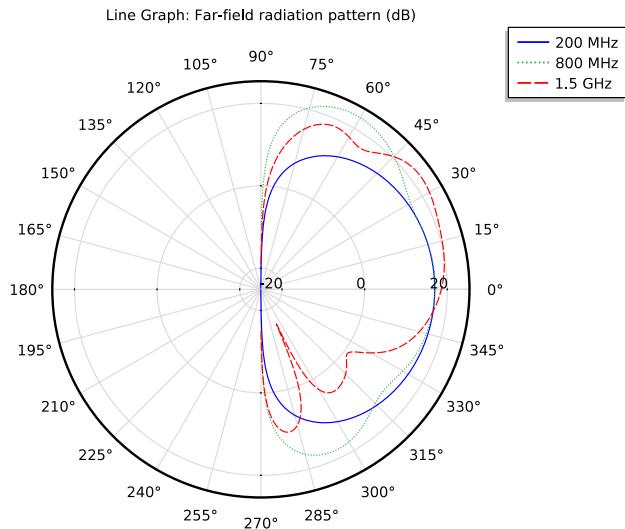


Figure 4: Polar plot of the antenna radiation pattern for the far field versus the elevation angle for 200 MHz, 800 MHz, and 1.5 GHz. This plot is normalized differently but has a shape resembling the near field.

As the frequency increases the antenna impedance gets closer to 50Ω , which means that a voltage generator connected to the input of the antenna should have an output impedance of 50Ω .

Model Library path: RF_Module/Antennas/conical_antenna

Modeling Instructions

MODEL WIZARD

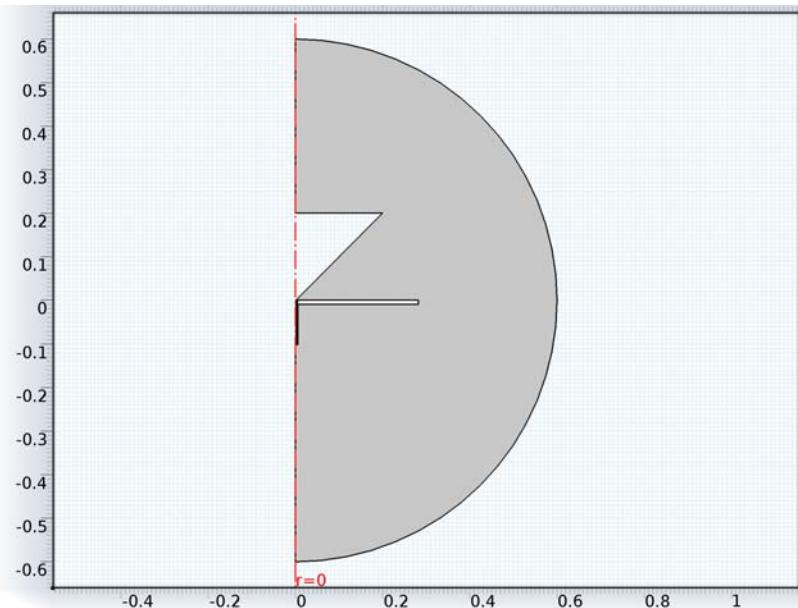
- 1 Go to the **Model Wizard** window.
- 2 Click the **2D axisymmetric** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GEOMETRY I

Import I

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Import**.
- 2 In the **Import** settings window, locate the **Import** section.
- 3 Click the **Browse** button.
- 4 Browse to the model's Model Library folder and double-click the file **conical_antenna.mphbin**.

- 5 Click the **Import** button.



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 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
Z_t1	50[ohm]	characteristic transmission line impedance

DEFINITIONS

Variables 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.
- 2 In the **Variables** settings window, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
Z	$Z_{\text{tl}} * (1 + \text{emw.S11}) / (1 - \text{emw.S11})$	Antenna impedance

`emw.S11` is the name of the automatically computed reflection S-parameter.

DEFINITIONS

Define the following selections in order to get easy access to some frequently used domains and boundaries.

Explicit 1

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Model Builder** window, under **Model 1>Definitions** right-click **Explicit 1** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Air** in the **New name** edit field.
- 4 Click **OK**.
- 5 Select Domain 1 only.

Explicit 2

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 2** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Dielectric** in the **New name** edit field.
- 4 Click **OK**.

The dielectric domain is inside the coaxial cable just below the cone. It is easier to select it if you zoom in a little.

- 5 Select Domain 2 only.

Explicit 3

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

- 2** Right-click **Explicit 3** and choose **Rename**.
- 3** Go to the **Rename Explicit** dialog box and type **Outer Air Boundaries** in the **New name** edit field.
- 4** Click **OK**.
- 5** In the **Explicit** settings window, locate the **Input Entities** section.
- 6** From the **Geometric entity level** list, choose **Boundary**.
- 7** 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.

MATERIALS

Material Browser

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

Air

- 1** In the **Model Builder** window, under **Model 1>Materials** click **Air**.
- 2** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Selection** list, choose **Air**.

Material 2

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Selection** list, choose **Dielectric**.
- 4** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	2.07
Relative permeability	mu_r	1
Electric conductivity	sigma	0

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Port 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** 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 **Port** settings window, locate the **Port Properties** section.
- 4 From the **Type of port** list, choose **Coaxial**.
- 5 From the **Wave excitation at this port** list, choose **On**.

Scattering Boundary Condition 1

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Scattering Boundary Condition**.
- 2 In the **Scattering Boundary Condition** settings window, 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** 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 **Far-Field Calculation** settings window, 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 **Model 1** right-click **Mesh 1** and choose **Free Triangular**.

Because this is a relatively small 2D model, you could easily use one of the finer mesh defaults and get a good accuracy. The following settings however are designed to get you a fine mesh mostly where it is needed.

Size

- 1** In the **Size** settings window, locate the **Element Size Parameters** section.

- 2** In the **Maximum element size** edit field, type **25[mm]**.

This global maximum element size makes sure that the mesh everywhere resolves the wavelength.

Size 1

- 1** In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

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

- 3** From the **Geometric entity level** list, choose **Domain**.

- 4** From the **Selection** list, choose **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 edit field, type **0.5[mm]**.

You now have an especially fine mesh inside the coaxial cable, where the wave is produced.

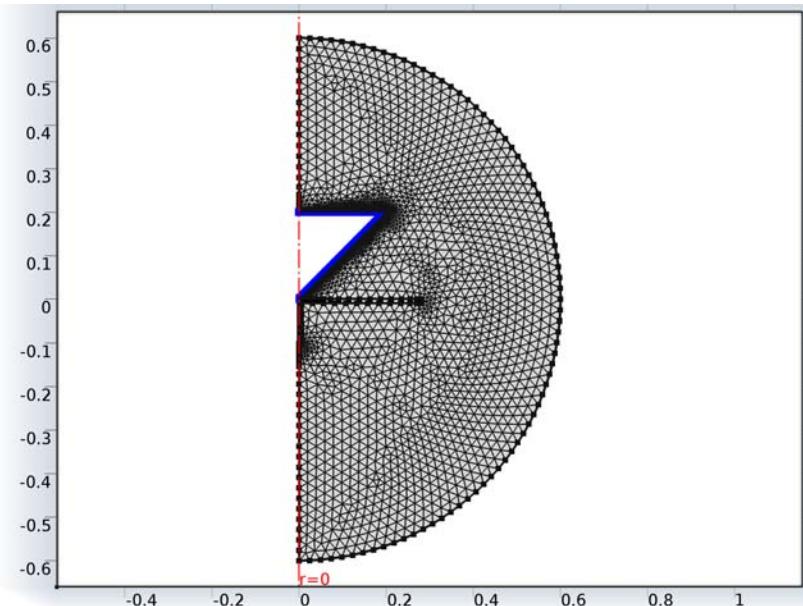
Size 2

- 1** Right-click **Free Triangular 1** and choose **Size**.

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

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

- 4 Select Boundaries 4 and 8 only.
 - 5 Locate the **Element Size** section. Click the **Custom** button.
 - 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
 - 7 In the associated edit field, type $2.5[\text{mm}]$.
- This gives a fine mesh on the surface of the antenna.
- 8 Click the **Build All** button.



STUDY I

Step 1: Frequency Domain

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

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

- 3 In the **Frequencies** edit field, type **range(200e6,25e6,1.5e9)**.

The frequency range you just entered runs from 200 MHz to 1.5 GHz in steps of 25 MHz.

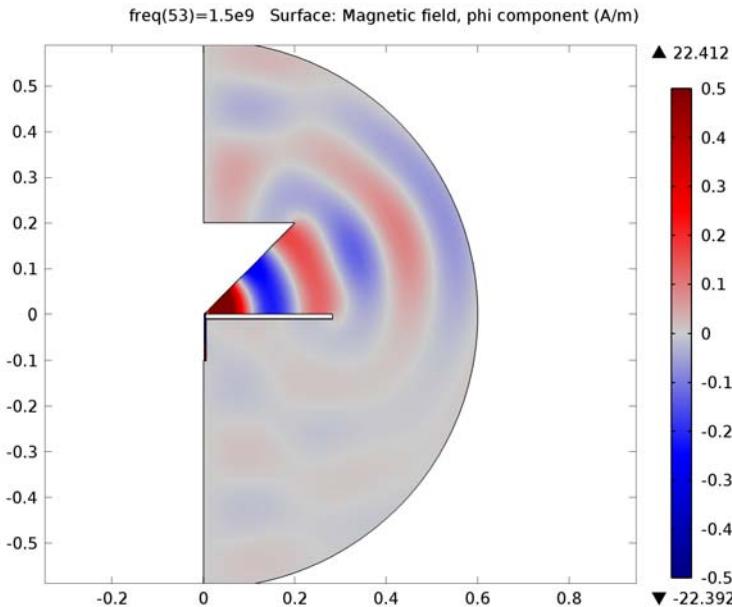
- 4 In the **Model Builder** window, right-click **Study I** and choose **Compute**.

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.

- 1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Surface**.
- 2 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>Magnetic field, phi component (emw.Hphi)**.
- 3 Click to expand the **Range** section. Select the **Manual color range** check box.
- 4 In the **Minimum** edit field, type -0.5.
- 5 In the **Maximum** edit field, type 0.5.
- 6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.
- 7 Click the **Plot** button.
- 8 Click the **Zoom Extents** button on the Graphics toolbar.



To plot the impedance as a function of the frequency, set up a 1D plot.

ID Plot Group 4

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Right-click **ID Plot Group 4** and choose **Global**.
- 3 In the **Global** settings window, locate the **y-Axis Data** section.
- 4 In the table, enter the following settings:

Expression	Description
real(Z)	Resistance
imag(Z)	Reactance

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

- 6 Click the **Plot** button.

You have now reproduced [Figure 2](#). Next, visualize the near-field and far-field radiation patterns using polar plots.

Polar Plot Group 5

- 1 In the **Model Builder** window, right-click **Results** 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 **Polar Plot Group** settings window, locate the **Data** section.
- 3 From the **Parameter selection (freq)** list, choose **From list**.
- 4 In the **Parameter values (freq)** list, choose **2e8**, **8e8**, and **1.5e9**.
- 5 Right-click **Results>Polar Plot Group 5** and choose **Line Graph**.
- 6 In the **Line Graph** settings window, locate the **Selection** section.
- 7 From the **Selection** list, choose **Outer Air Boundaries**.
- 8 Click **Replace Expression** in the upper-right corner of the **r-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Energy and power>Power outflow, time average (emw.nPoav)**.
- 9 Locate the **r-Axis Data** section. In the **Expression** edit 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.
- 10 Select the **Description** check box.

- I1** In the associated edit field, type **Near-field radiation pattern**.
- I2** Locate the **θ Angle Data** section. From the **Parameter** list, choose **Expression**.
- I3** In the **Expression** edit field, type `atan2(z,r)`.
- I4** Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.
- I5** Click to collapse the **Coloring and Style** section. Click to expand the **Legends** section. Select the **Show legends** check box.
- I6** From the **Legends** list, choose **Manual**.
- I7** In the table, enter the following settings:

Legends
200 MHz
800 MHz
1.5 GHz

- I8** Click to collapse the **Legends** section. 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.

Polar Plot Group 6

- I1** In the **Model Builder** window, right-click **Polar Plot Group 5** and choose **Duplicate**.
- I2** In the **Model Builder** window, expand the **Results>Polar Plot Group 6** node, then click **Line Graph 1**.
- I3** In the **Line Graph** settings window, click **Replace Expression** in the upper-right corner of the **r-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Global>Far-field norm, dB (emw.normdBfar)**.
- I4** Locate the **r-Axis Data** section. In the **Description** edit field, type **Far-field radiation pattern**.
- I5** Click the **Plot** button.

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.

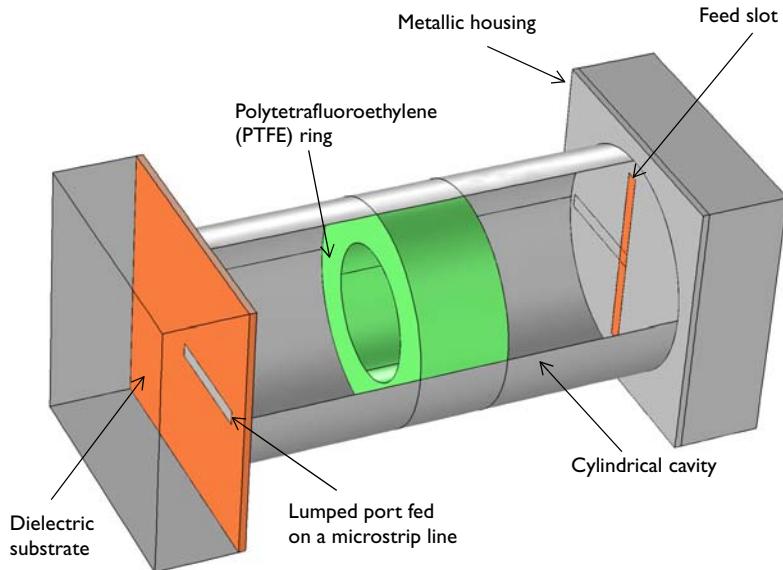


Figure 1: An evanescent mode cavity filter. The signal fed from a microstrip line is slot coupled into the cylindrical cavity loaded with a PTFE ring.

Model Definition

The resonant frequency of the empty cylindrical waveguide cavity TE₁₁₁ mode can be calculated from the equation

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

where a and d are the radius and length of the cylinder, respectively, and p'_{nm} is the m th root of the Bessel function $J'_n(x)$. The TE₁₁₁ 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 model was a computation (not presented here) of the TE₁₁₁ 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, and the end of each microstrip line over the slots is shorted. The cavity is partially filled with a ring of PTFE, $\epsilon_r = 2.1$, 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.

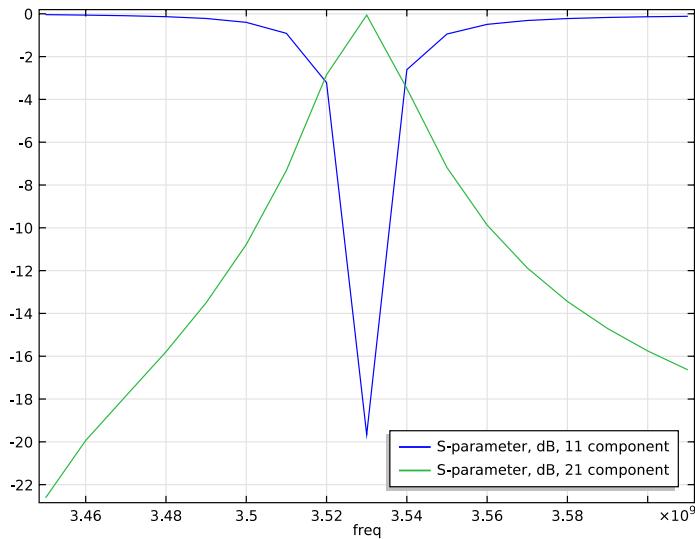


Figure 2: The frequency response of the filter shows bandpass filter characteristics. The center frequency is lower than the dominant mode resonant frequency of the metallic cavity.

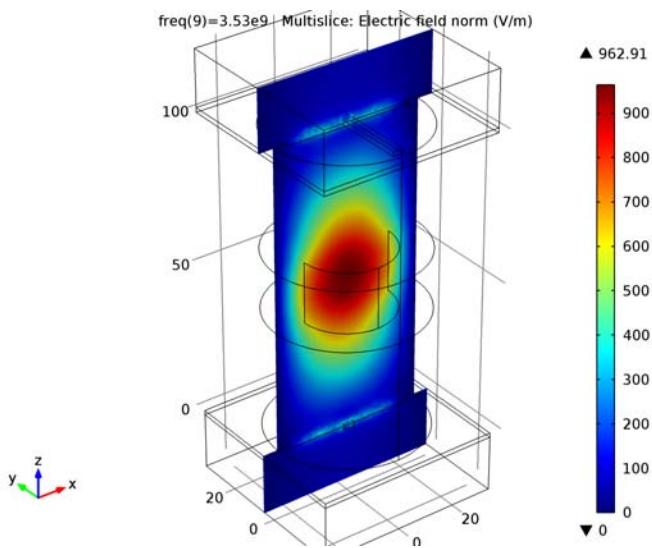


Figure 3: The dielectric tube inside the cavity does not distort the electric field distribution at resonance significantly.

Reference

1. D.M. Pozar, *Microwave Engineering*, Wiley, 1998.
-

Model Library path: RF_Module/Passive_Devices/
cylindrical_cavity_filter_evanescent

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

- 3** In the table, enter the following settings:

Name	Expression	Description
d	60[mil]	Substrate thickness
l_slot	42[mm]	Slot length
w_slot	3[mm]	Slot width
f_min	3.45[GHz]	Minimum frequency in sweep
f_max	3.61[GHz]	Maximum frequency in sweep
lda_min	c_const/f_max	Minimum wavelength, air
h_max	0.2*lda_min	Maximum element size, air

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

GEOMETRY I

- 1** In the **Model Builder** window, under **Model I** click **Geometry I**.
2 In the **Geometry** settings window, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Create a cylindrical cavity.

Cylinder I

- 1** Right-click **Model I > Geometry I** and choose **Cylinder**.
2 In the **Cylinder** settings window, locate the **Size and Shape** section.
3 In the **Radius** edit field, type 25.
4 In the **Height** edit field, type 100.
5 Click the **Build Selected** button.
6 Right-click **Model I > Geometry I > Cylinder I** and choose **Rename**.
7 Go to the **Rename Cylinder** dialog box and type **Cavity** in the **New name** edit field.
8 Click **OK**.

Create a coupling slot.

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

Rectangle I

- 1** In the **Model Builder** window, under **Model I > Geometry I > Work Plane I** right-click **Plane Geometry** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.
3 In the **Width** edit field, type **l_slot**.

- 4 In the **Height** edit field, type `w_slot`.
- 5 Locate the **Position** section. From the **Base** list, choose **Center**.
- 6 Click the **Build Selected** button.

Create a substrate.

Block 1

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `60`.
- 4 In the **Depth** edit field, type `60`.
- 5 In the **Height** edit field, type `d`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **z** edit field, type `-d/2`.
- 8 Click the **Build Selected** button.
- 9 Right-click **Model 1>Geometry 1>Block 1** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type `Bottom_plate` in the **New name** edit field.
- II Click **OK**.

Create a 50 ohm microstrip line.

Block 2

- 1 Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `3.2`.
- 4 In the **Depth** edit field, type `25`.
- 5 In the **Height** edit field, type `d`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **y** edit field, type `25/2-w_slot/2`.
- 8 In the **z** edit field, type `-d/2`.
- 9 Click the **Build Selected** button.
- 10 Right-click **Model 1>Geometry 1>Block 2** and choose **Rename**.
- II Go to the **Rename Block** dialog box and type `Bottom_feed` in the **New name** edit field.

I2 Click OK.

Create a metallic housing.

Block 3

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

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

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

4 In the **Depth** edit field, type 60.

5 In the **Height** edit field, type 20.

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

7 In the **z** edit field, type -10.

8 Click the **Build Selected** button.

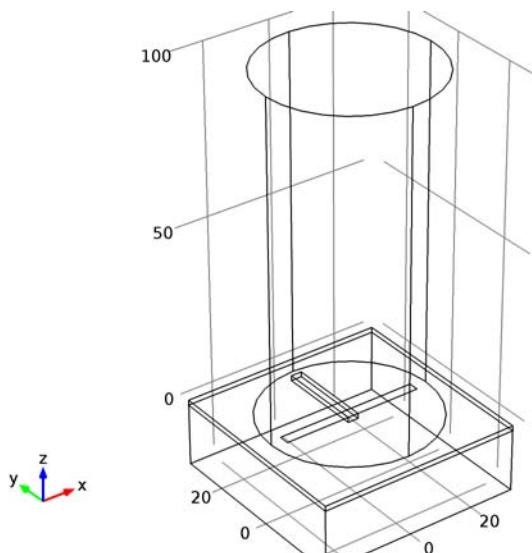
9 Click the **Zoom Extents** button on the Graphics toolbar.

I0 Right-click **Model 1>Geometry 1>Block 3** and choose **Rename**.

II Go to the **Rename Block** dialog box and type **Housing** in the **New name** edit field.

I2 Click OK.

I3 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

- 1 Right-click **Geometry 1** and choose **Transforms>Rotate**.
- 2 Select all objects except for the cylinder, that is, **blk3**, **wpl**, **blk1**, and **blk2**.
- 3 In the **Rotate** settings window, locate the **Rotation Angle** section.
- 4 In the **Rotation** edit field, type 0, 180.
- 5 Locate the **Point on Axis of Rotation** section. In the **z** edit field, type 50.
- 6 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **Cartesian**.
- 7 In the **x** edit field, type 1.
- 8 In the **z** edit field, type 0.
- 9 Click the **Build Selected** button.
- 10 Click the **Zoom Extents** button on the Graphics toolbar.

Create a dielectric ring.

Cylinder 2

- 1 Right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type 25.
- 4 In the **Height** edit field, type 20.
- 5 Locate the **Position** section. In the **z** edit field, type 40.

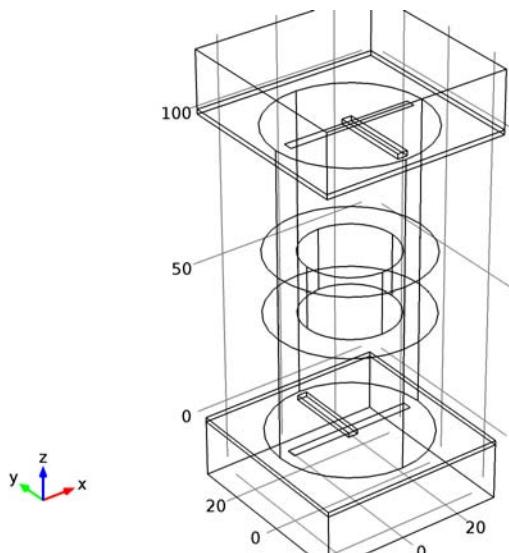
Cylinder 3

- 1 Right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type 15.
- 4 In the **Height** edit field, type 20.
- 5 Locate the **Position** section. In the **z** edit field, type 40.

Difference 1

- 1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.
- 2 Select the object **cyl2** only.
- 3 In the **Difference** settings window, locate the **Difference** section.
- 4 Under **Objects to subtract**, click **Activate Selection**.
- 5 Select the object **cyl3** only.

6 Click the **Build All** button.



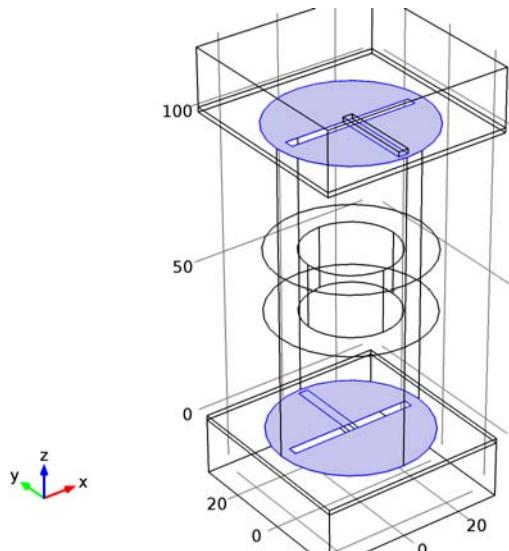
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

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

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

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

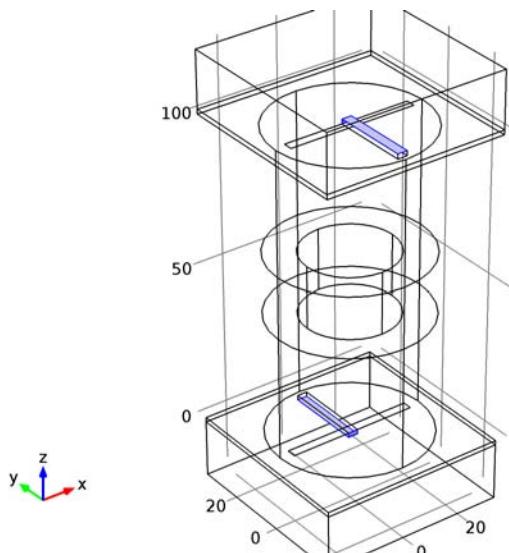


Proceed to define boundary condition for the shorted microstrip lines.

Perfect Electric Conductor 3

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

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



Lumped Port 1

- 1** Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2** Select Boundary 44 only.
- 3** In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4** From the **Wave excitation at this port** list, choose **On**.

Lumped Port 2

- 1** Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2** Select Boundary 34 only.

MATERIALS

Material Browser

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

Create a substrate material.

Material 2

- 1 In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2 Select Domains 2, 3, 7, and 8 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

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

- 5 Right-click **Model 1>Materials>Material 2** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type **Substrate** in the **New name** edit field.
- 7 Click **OK**.

Create a dielectric ring material.

Material 3

- 1 Right-click **Materials** and choose **Material**.
- 2 Select Domain 6 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilonr	2.1
Relative permeability	mur	1
Electrical conductivity	sigma	0

- 5 Right-click **Model 1>Materials>Material 3** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type **PTFE** in the **New name** edit field.
- 7 Click **OK**.

MESH 1

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

Size

The maximum element size, 14, is below 0.2 times the minimum wavelength over the frequency sweep, so you could keep this setting as is. However, to get a setting that varies with the frequency range in use, specify the maximum element size as follows:

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size Parameters** section.
- 3 In the **Maximum element size** edit field, type `h_max`.

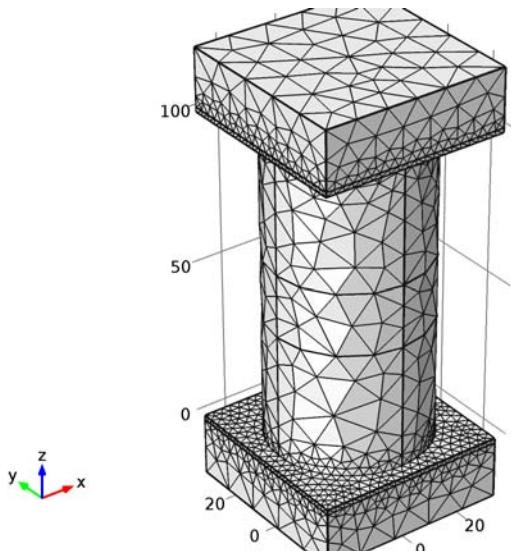
Size 1

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

To increase the accuracy of the S-parameter calculations, refine the meshes on the lumped ports.

- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 34 and 44 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type 1.

- 8** Click the **Build All** button.



STUDY I

Step 1: Frequency Domain

- 1** In the **Model Builder** window, under **Study I** click **Step 1: Frequency Domain**.
- 2** In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3** In the **Frequencies** edit field, type `range(f_min,0.01[GHz],f_max)`.
- 4** In the **Model Builder** window, right-click **Study I** and choose **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 **3D Plot Group** settings window, locate the **Data** section.
- 2** From the **Parameter value (freq)** list, choose **3.53e9**.
- 3** In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
- 4** In the **Multislice** settings window, locate the **Multiplane Data** section.
- 5** Find the **x-planes** subsection. In the **Planes** edit field, type **0**.

6 Find the **z-planes** subsection. In the **Planes** edit field, type 0.

7 Click the **Plot** button.

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

Proceed with plotting the scattering parameters ([Figure 2](#)).

ID Plot Group 2

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

2 In the **ID Plot Group** settings window, click to expand the **Legend** section.

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

4 Right-click **Results>ID Plot Group 2** and choose **Global**.

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

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

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

8 Click the **Plot** button.

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.

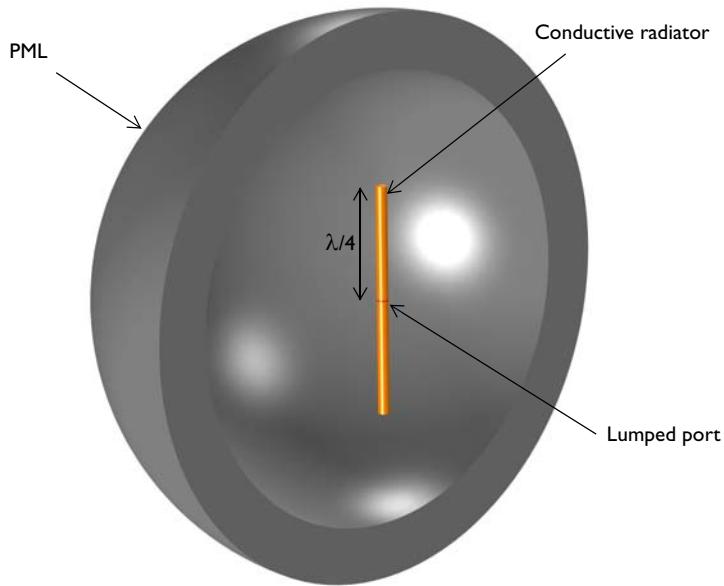


Figure 1: A dipole antenna. The model consists of two cylindrical arms of conductive material with a voltage source in between. A region of free space bounded by a perfectly matched layer (PML) surrounds the antenna.

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 will approach 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 $119 + 30i \Omega$, 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.

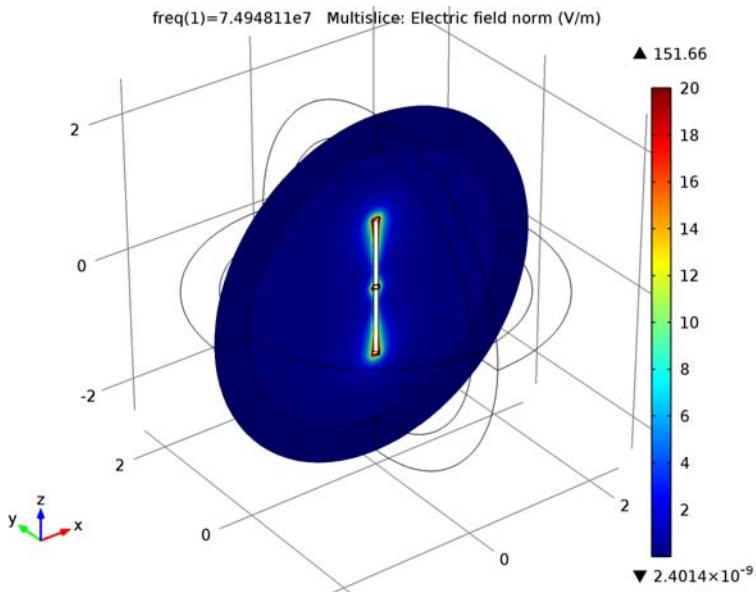


Figure 2: A slice plot of the electric field magnitude around the antenna.

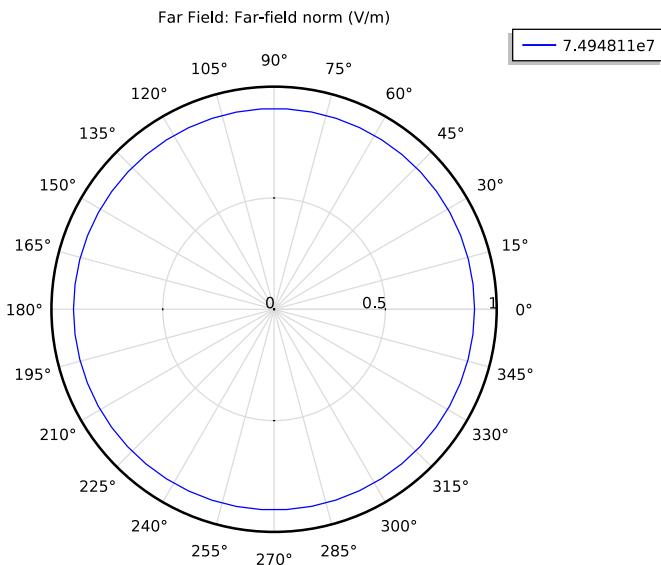


Figure 3: The polar plot of the far field pattern in the xy -plane is isotropic.

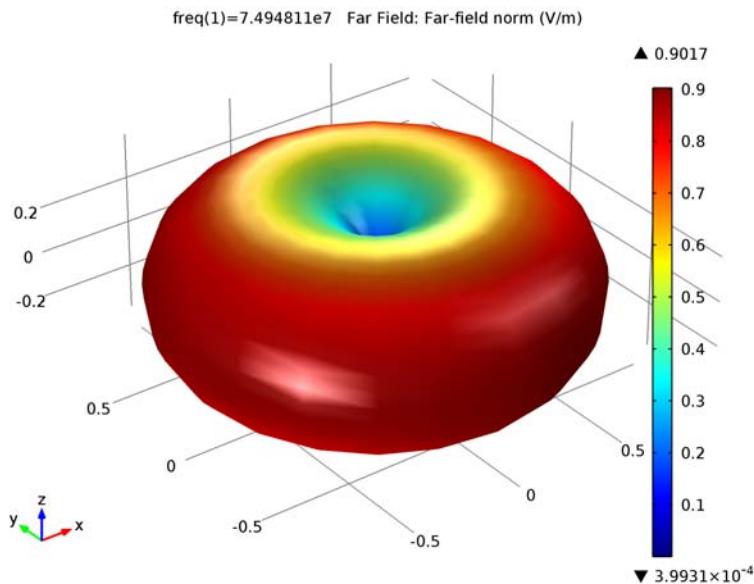


Figure 4: A 3D visualization of the far-field pattern of the dipole shows the expected torus-shaped pattern.

Model Library path: RF_Module/Antennas/dipole_antenna

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
lambda0	4[m]	Operating wavelength
arm_length	lambda0/4	Dipole antenna arm length
r_antenna	arm_length/20	Dipole antenna arm radius
gap_size	arm_length/100	Gap between arms

GEOMETRY I

Create a sphere with a layer. The outer layer presents the PML.

Sphere 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Sphere**.
- 2 In the **Sphere** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type $2.5 * \text{arm_length}$.
- 4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	$0.5 * \text{arm_length}$

- 5 Click the **Build Selected** button.

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

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type r_antenna .
- 4 In the **Height** edit field, type $2 * \text{arm_length} + \text{gap_size}$.

5 Locate the **Position** section. In the **z** edit field, type $-(\text{arm_length}+\text{gap_size}/2)$.

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 Click the **Build Selected** button.

The domain inside the antenna radiators is not part of the model analysis.

Difference 1

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

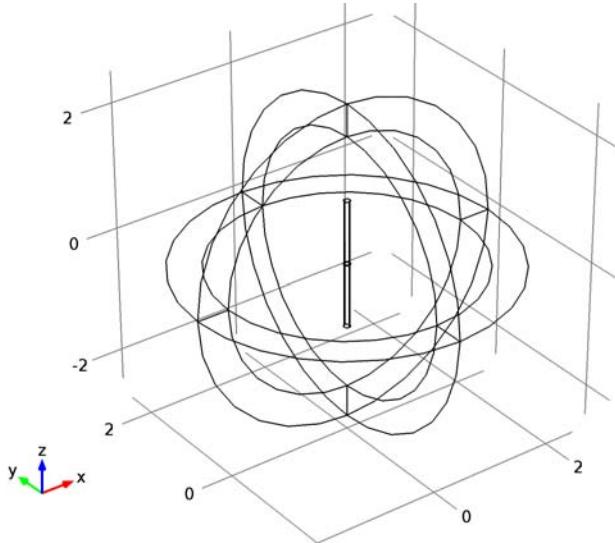
2 Select the object **sph1** only.

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

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

5 Select the object **cyl1** only.

6 Click the **Build All** button.

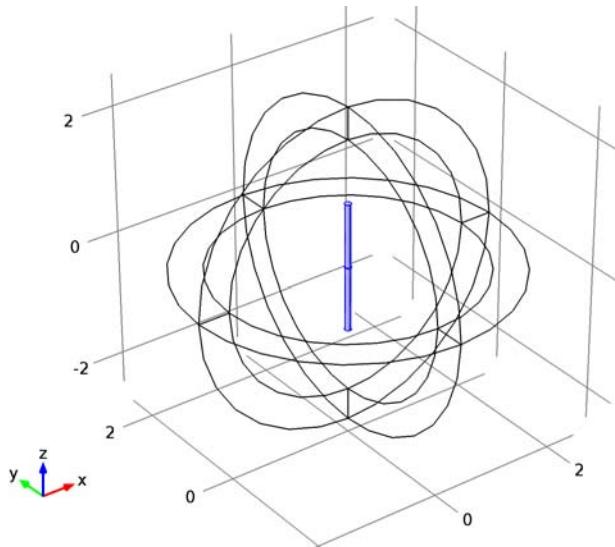


DEFINITIONS

Create a set of selections for use before setting up the physics. First, create a selection for the antenna radiator surface.

Explicit 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Click **Paste Selection**.
- 5 Go to the **Paste Selection** dialog box.
- 6 In the **Selection** edit field, type **13-15, 18-20, 28, 30, 39, 41**.
- 7 Click the **OK** button.



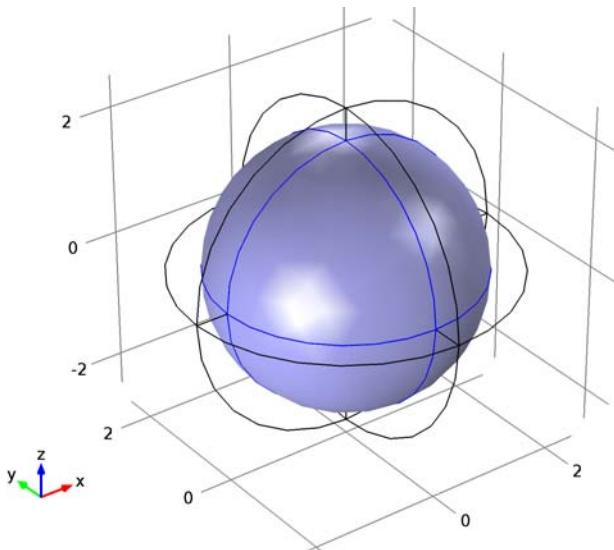
- 8 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.
- 9 Go to the **Rename Explicit** dialog box and type **Antenna** in the **New name** edit field.
- 10 Click **OK**.

Add a selection for the far-field domain.

Explicit 2

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

2 Select Domain 5 only.



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

4 Go to the **Rename Explicit** dialog box and type **Far-field domain** in the **New name** edit field.

5 Click **OK**.

Add a selection for the far-field calculation. This is the outer surface of the far-field domain.

Explicit 3

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

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

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

4 Select Boundaries 9–12, 26, 27, 34, and 37 only.

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

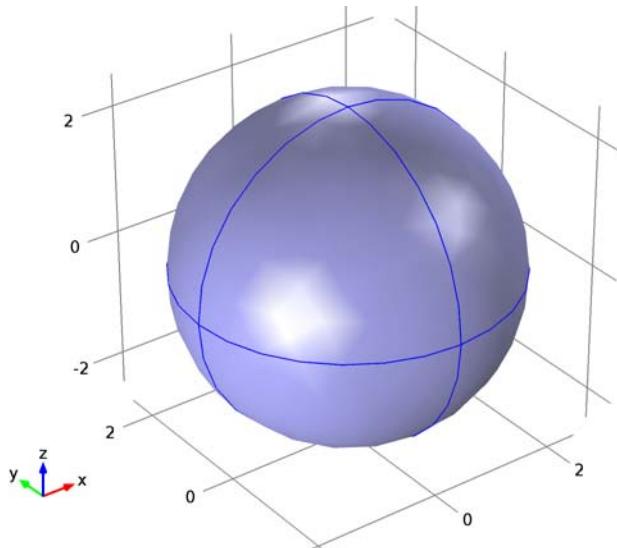
6 Go to the **Rename Explicit** dialog box and type **Far-field calculation** in the **New name** edit field.

7 Click **OK**.

Add a perfectly matched layer on the outermost domain of the sphere.

Perfectly Matched Layer 1

- 1 Right-click **Definitions** and choose **Perfectly Matched Layer**.
- 2 Select Domains 1–4 and 6–9 only.
- 3 In the **Perfectly Matched Layer** settings window, locate the **Geometry** section.
- 4 From the **Type** list, choose **Spherical**.

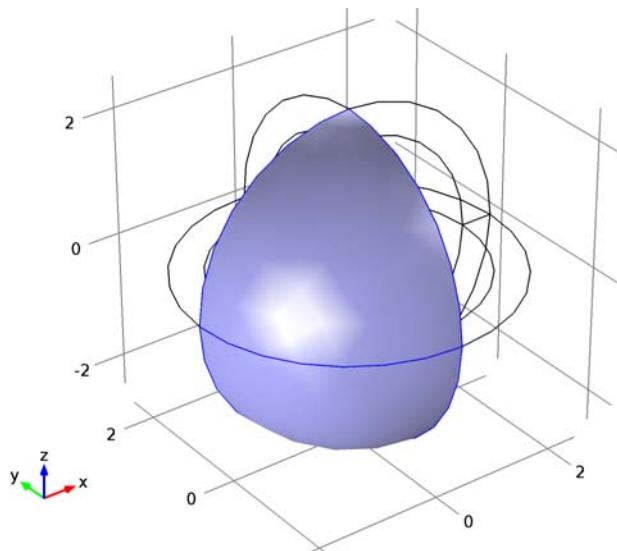


View 1

Suppress some domains and boundaries. This helps to see the interior parts when setting up the physics and reviewing the mesh.

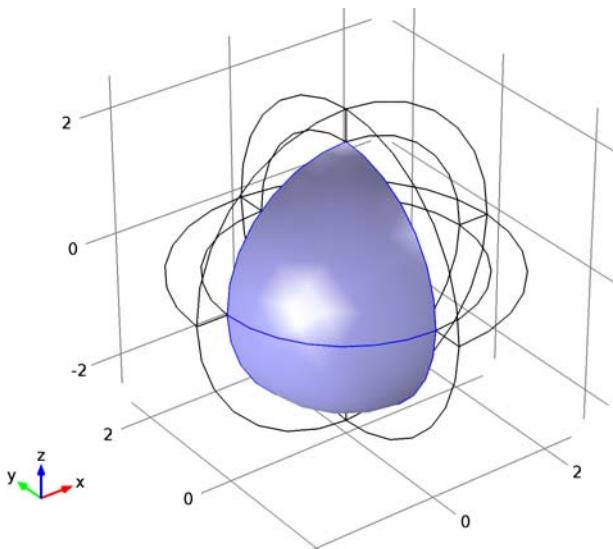
- 1 In the **Model Builder** window, under **Model 1 > Definitions** right-click **View 1** and choose **Hide Geometric Entities**.

- 2** Select Domains 1 and 2 only.



- 3** In the **Model Builder** window, right-click **View 1** and choose **Hide Geometric Entities**.
- 4** In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.
- 5** From the **Geometric entity level** list, choose **Boundary**.

- 6 Select Boundaries 9 and 10 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Set up the physics for the model. Add an **Impedance Boundary Condition** that overrides the default PEC boundary condition on the antenna radiator surface.

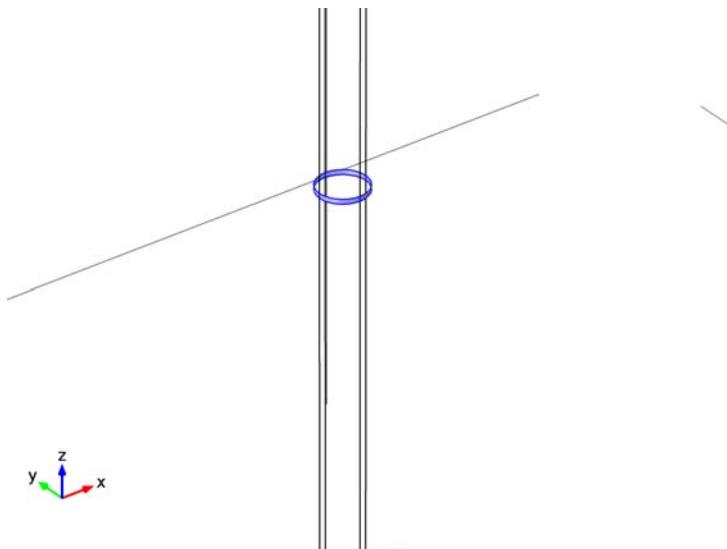
Impedance Boundary Condition 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Impedance Boundary Condition**.
- 2 In the **Impedance Boundary Condition** settings window, 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** 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 **Lumped Port** settings window, locate the **Port Properties** section.
5 From the **Type of port** list, choose **User defined**.
6 In the h_{port} edit field, type `gap_size`.
7 In the w_{port} edit field, type `2*pi*r_antenna`.
8 In the **a_h** table, enter the following settings:

0	x
0	y
1	z

- 9** From the **Wave excitation at this port** list, choose **On**.
10 Click the **Zoom Extents** button on the Graphics toolbar.

Far-Field Domain I

- 1** Right-click **Electromagnetic Waves, Frequency Domain** and choose **Far-Field Domain**.
2 In the **Far-Field Domain** settings window, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Far-field domain**.

Far-Field Calculation I

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

- 2 In the **Far-Field Calculation** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Far-field calculation**.

MATERIALS

Assign air as the material for all domains and override the antenna radiator surface with copper.

Material Browser

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
- 2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.
- 3 Click **Add Material to Model**.
- 4 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
- 5 In the **Material Browser** settings window, In the tree, select **Built-In>Copper**.
- 6 Click **Add Material to Model**.

Copper

- 1 In the **Model Builder** window, under **Model 1>Materials** click **Copper**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **Antenna**.

MESH 1

The maximum mesh size is 0.2 wavelengths in free space. To evaluate the antenna radiator with the level of the 2nd-order polynomial, set the maximum element size smaller than the cylinder radius.

Size 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Far-field domain**.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type $\lambda_0/5$.

Size 2

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **Antenna**.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type `r_antenna/1.5`.

Free Tetrahedral 1

- 1 Right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Far-field domain**.

Use a swept mesh for the PML.

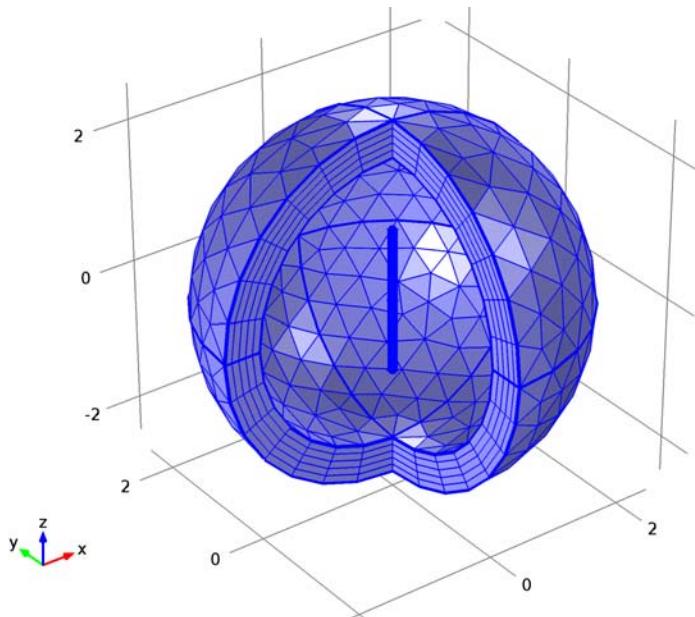
Swept 1

Right-click **Mesh 1** and choose **Swept**.

Distribution 1

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

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



STUDY 1

Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type c_const/λ_0 .
- 4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

The default plot shows the E-field norm, 2D far-field polar plot, and 3D far-field radiation pattern.

Electric Field (emw)

- 1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
- 2 In the **Multislice** settings window, locate the **Multiplane Data** section.
- 3 Find the **x-planes** subsection. In the **Planes** edit field, type 0.
- 4 Find the **z-planes** subsection. In the **Planes** edit field, type 0.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 Set the **Maximum** value to 1.

The results show the E-field norm distribution on the antenna radiators. It is plotted in [Figure 2](#).

Polar Plot Group 2

Adjust the axis range.

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

2 In the **Polar Plot Group** settings window, click to expand the **Axis** section.

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

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

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

6 Click the **Plot** button.

The plotted H-plane pattern is omni-directional on the *xy*-plane as shown in [Figure 3](#).

3D Plot Group 3

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

Compare the reproduced plot with [Figure 4](#).

Derived Values

Finish the result analysis by evaluating the port impedance.

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

2 In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>Lumped port impedance (emw.Zport_1)**.

3 Click the **Evaluate** button.

Fresnel Equations

Introduction

A plane electromagnetic wave propagating through free space ($n = 1$) as shown in [Figure 1](#) is incident at an angle upon an infinite dielectric medium ($n = 1.5$) and is partially reflected and partially transmitted. 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 at an angle 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 will be no reflections at an incident angle of roughly 56° , known as the *Brewster angle*.

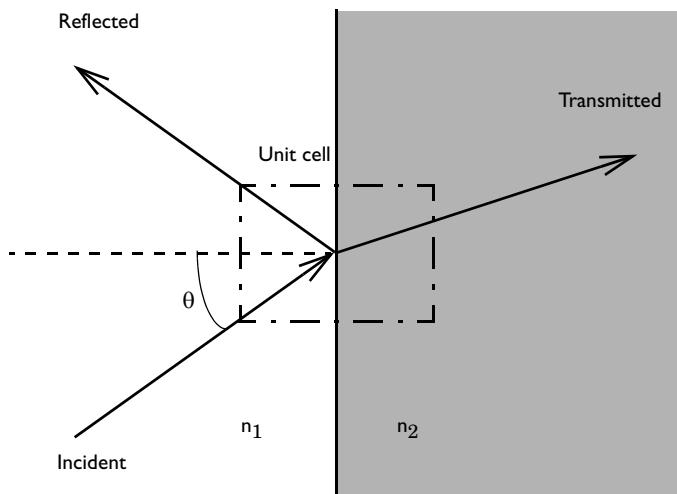


Figure 1: A plane wave propagating through free space incident upon an infinite dielectric medium.

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. Reflection and transmission coefficients for s-polarization and p-polarization are defined respectively as

$$r_s = \frac{n_1 \cos \theta_{\text{incident}} - n_2 \cos \theta_{\text{transmitted}}}{n_1 \cos \theta_{\text{incident}} + n_2 \cos \theta_{\text{transmitted}}} \quad (1)$$

$$t_s = \frac{2n_1 \cos \theta_{\text{incident}}}{n_1 \cos \theta_{\text{incident}} + n_2 \cos \theta_{\text{transmitted}}} \quad (2)$$

$$r_p = \frac{n_2 \cos \theta_{\text{incident}} - n_1 \cos \theta_{\text{transmitted}}}{n_1 \cos \theta_{\text{transmitted}} + n_2 \cos \theta_{\text{incident}}} \quad (3)$$

$$t_p = \frac{2n_1 \cos \theta_{\text{incident}}}{n_1 \cos \theta_{\text{transmitted}} + n_2 \cos \theta_{\text{incident}}} \quad (4)$$

Reflectance and transmittance are defined as

$$R = |r|^2 \quad (5)$$

$$T = \frac{n_2 \cos \theta_{\text{transmitted}}}{n_1 \cos \theta_{\text{incident}}} |t|^2 \quad (6)$$

The Brewster angle at which $r_p = 0$ is defined as

$$\theta_B = \text{atan} \frac{n_2}{n_1} \quad (7)$$

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.

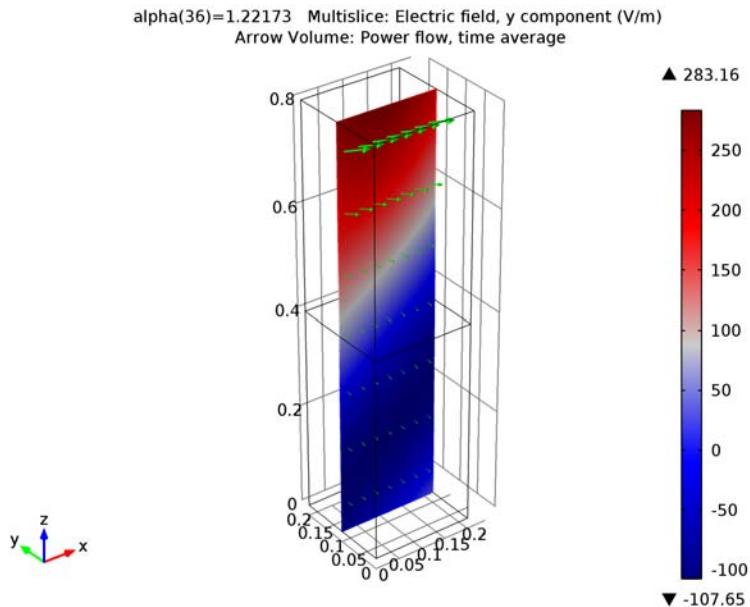


Figure 2: Electric field, E_y (slice) and power flow (arrows) for TE incidence at 70° inside the unit cell.

For the TM case, [Figure 3](#) visualizes the y component of the magnetic-field distribution instead, again in combination with the power flow.

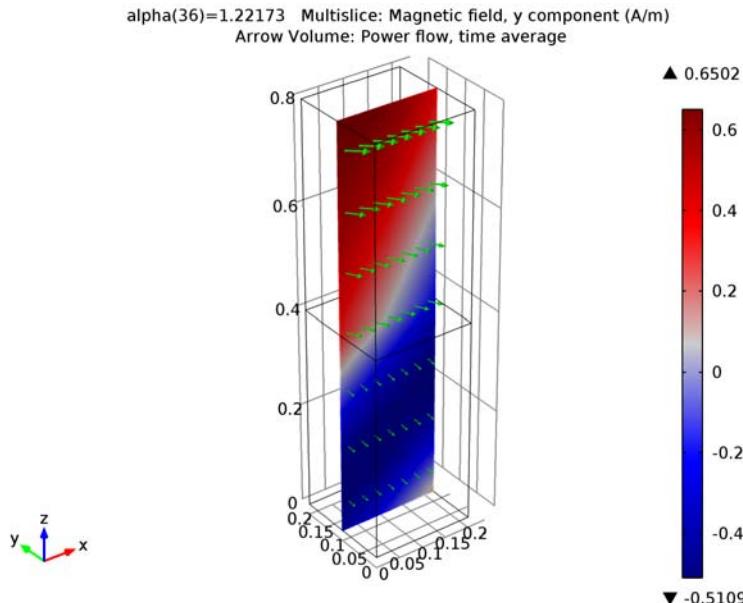


Figure 3: Magnetic field, H_y (slice) and power flow (arrows) for TM incidence at 70° inside the unit cell.

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.

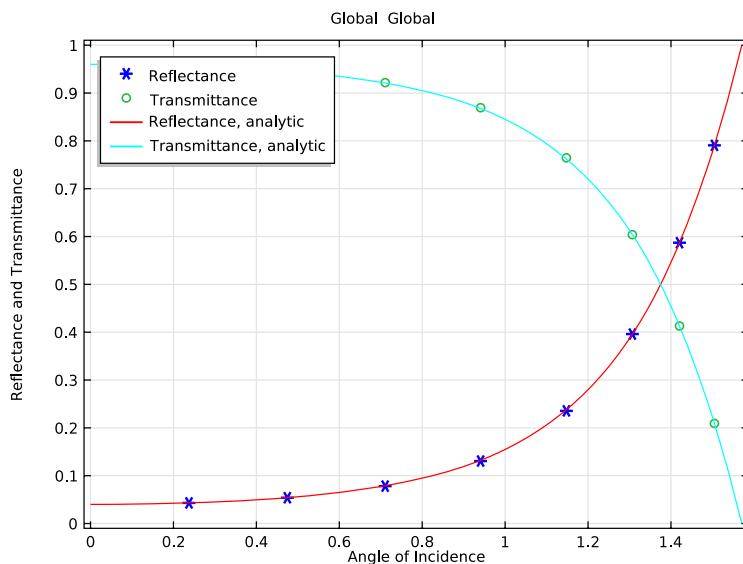


Figure 4: The reflectance and transmittance for TE incidence agree well with the analytic solutions.

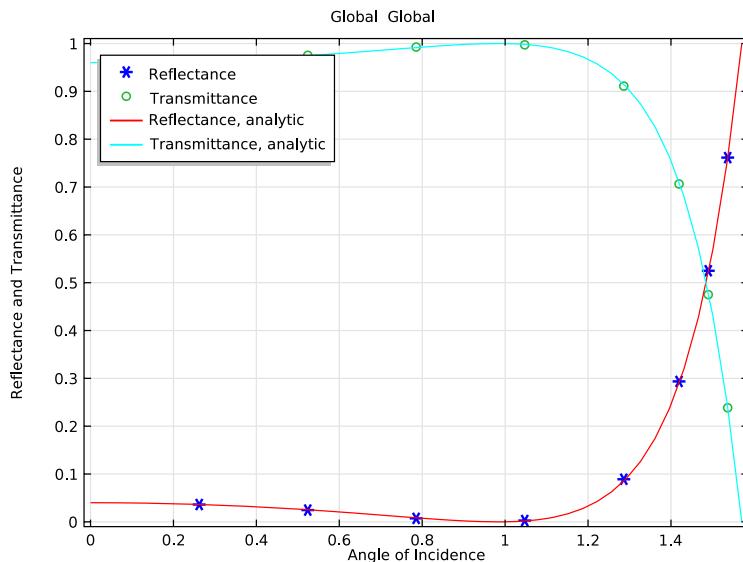


Figure 5: The reflectance and transmittance for TM incidence agree well with the analytic solutions. The Brewster angle is also observed at the expected location.

References

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, Wiley, 1989.
 2. B.E.A. Saleh and M.C. Teich, *Fundamentals of Photonics*, Wiley, 1991.
-

Model Library path: RF_Module/Verification_Models/fresnel_equations

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

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

Parameters

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

- 3** In the table, enter the following settings:

Name	Expression	Description
n_air	1	Refractive index, air
n_slab	1.5	Refractive index, slab
lda0	1[m]	Wavelength
f0	c_const/lda0	Frequency
alpha	70[deg]	Angle of incidence
beta	asin(n_air*sin(alpha)/n_slab)	Refraction angle
h_max	lda0/6	Maximum element size, air
alpha_brewster	atan(n_slab/n_air)	Brewster angle, TM only
r_s	(n_air*cos(alpha)-n_slab*cos(beta))/ (n_air*cos(alpha)+n_slab*cos(beta))	Reflection coefficient, TE
r_p	(n_slab*cos(alpha)-n_air*cos(beta))/ (n_air*cos(beta)+n_slab*cos(alpha))	Reflection coefficient, TM
t_s	(2*n_air*cos(alpha))/ (n_air*cos(alpha)+n_slab*cos(beta))	Transmission coefficient, TE
t_p	(2*n_air*cos(alpha))/ (n_air*cos(beta)+n_slab*cos(alpha))	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.

DEFINITIONS

Variables I

- In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.
- In the **Variables** settings window, locate the **Variables** section.

- 3** In the table, enter the following settings:

Name	Expression	Description
ka	emw.k0	Propagation constant, air
kax	ka*sin(alpha)	kx for incident wave
kay	0	ky for incident wave
kaz	ka*cos(alpha)	kz for incident wave
kb	n_slab*emw.k0	Propagation constant, slab
kbx	kb*sin(beta)	kx for refracted wave
kby	0	ky for refracted wave
kbz	kb*cos(beta)	kz for refracted wave

GEOMETRY I

First, create a block composed of two domains. Use layers to split the block.

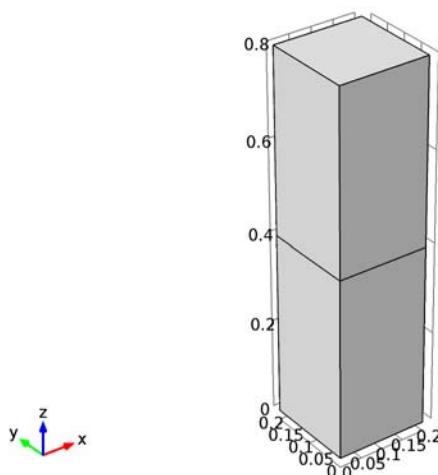
Block I

- 1** In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Block**.
- 2** In the **Block** settings window, locate the **Size and Shape** section.
- 3** In the **Width** edit field, type **0.2**.
- 4** In the **Depth** edit field, type **0.2**.
- 5** In the **Height** edit 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 the **Build All** button.

- 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

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.

Wave Equation, Electric I

- 1 In the **Model Builder** window, expand the **Model 1>Electromagnetic Waves, Frequency Domain** node, then click **Wave Equation, Electric I**.
- 2 In the **Wave Equation, Electric** settings window, locate the **Electric Displacement Field** section.
- 3 From the **Electric displacement field model** list, choose **Refractive index**.

The wave is excited from the port on the top.

Port I

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- 2 Select Boundary 7 only.

- 3** In the **Port** settings window, locate the **Port Properties** section.
- 4** From the **Wave excitation at this port** list, choose **On**.
- 5** Locate the **Port Mode Settings** section. In the **E₀** table, enter the following settings:

0	x
$\exp(-i*kax*x) [V/m]$	y
0	z

- 6** In the β edit field, type `abs(kaz)`.

Port 2

- 1** Right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- 2** Select Boundary 3 only.
- 3** In the **Port** settings window, locate the **Port Mode Settings** section.
- 4** In the **E₀** table, enter the following settings:

0	x
$\exp(-i*kbx*x) [V/m]$	y
0	z

- 5** In the β edit field, type `abs(kbz)`.

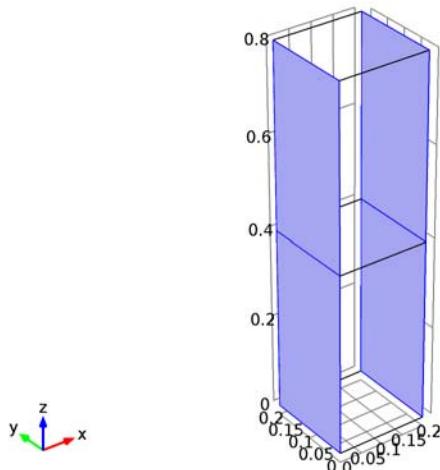
The bottom surface is an observation port. The S_{21} -parameter from Port 1 and Port 2 provides the transmission characteristics.

The E-field polarization has E_y only and the boundaries are always either parallel or perpendicular to the E-field polarization. Apply periodic boundary conditions on the boundaries parallel to the E-field except those you already assigned to the ports.

Periodic Condition 1

- 1** Right-click **Electromagnetic Waves, Frequency Domain** and choose **Periodic Condition**.
- 2** Select Boundaries 1, 4, 10, and 11 only.
- 3** In the **Periodic Condition** settings window, locate the **Periodicity Settings** section.
- 4** From the **Type of periodicity** list, choose **Floquet periodicity**.
- 5** In the **k_F** table, enter the following settings:

kax	x
0	y
0	z

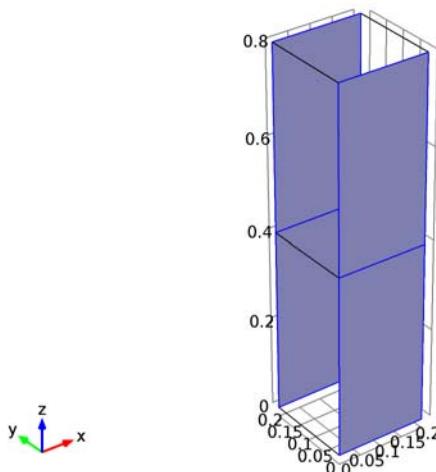


Apply a perfect electric conductor condition on the boundaries perpendicular to the E-field. This condition creates a virtually infinite modeling space.

Perfect Electric Conductor 2

- I Right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

- 2** Select Boundaries 2, 5, 8, and 9 only.



MATERIALS

Now set up the material properties based on refractive index. The top half is filled with air.

Material 1

- 1** In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2** Select Domain 2 only.
- 3** In the **Material** settings window, locate the **Material Contents** section.
- 4** In the table, enter the following settings:

Property	Name	Value
Refractive index	n	n_air

- 5** Right-click **Model 1>Materials>Material 1** and choose **Rename**.
- 6** Go to the **Rename Material** dialog box and type **Air** in the **New name** edit field.
- 7** Click **OK**.

The bottom half is glass.

Material 2

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

- 2** Select Domain 1 only.
- 3** In the **Material** settings window, locate the **Material Contents** section.
- 4** In the table, enter the following settings:

Property	Name	Value
Refractive index	n	n_slab

- 5** Right-click **Model 1>Materials>Material 2** and choose **Rename**.
- 6** Go to the **Rename Material** dialog box and type Glass in the **New name** edit field.
- 7** Click **OK**.

MESH 1

The periodic boundary condition performs better if the mesh is identical on the periodicity boundaries. This is especially important when dealing with vector degrees of freedom, as will be the case in the TM version of this model. The maximum element size is smaller than 0.2 times the wavelength. The bottom half domain is scaled inversely by the refractive index of the material.

- 1** In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

Size

- 1** In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
- 2** In the **Size** settings window, locate the **Element Size** section.
- 3** Click the **Custom** button.
- 4** Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type h_{max} .

Size 1

- 1** In the **Model Builder** window, under **Model 1>Mesh 1** click **Size 1**.
- 2** In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Geometric entity level** list, choose **Domain**.
- 4** Select Domain 1 only.
- 5** Locate the **Element Size** section. Click the **Custom** button.
- 6** Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7** In the associated edit field, type h_{max}/n_{slab} .

Free Triangular I

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 1 and 4 only.

Copy Face I

1 Right-click **Mesh 1** and choose **More Operations>Copy Face**.

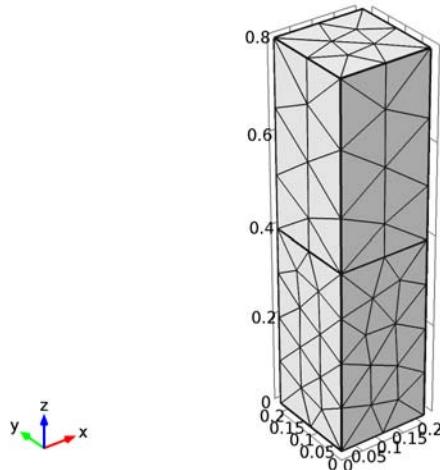
2 Select Boundaries 1 and 4 only.

3 In the **Copy Face** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundaries 10 and 11 only.

Free Tetrahedral I

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

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



STUDY I

Step 1: Frequency Domain

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

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

3 In the **Frequencies** edit field, type **f0**.

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names	Parameter value list
alpha	range(0,2[deg],90[deg])

Use a direct solver instead of an iterative one for faster convergence.

Solver 1

- 1 Right-click **Study 1** and choose **Show Default Solver**.
- 2 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** right-click **Direct** and choose **Enable**.
- 3 Click the **Compute** button.

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.

- 1 In the **Model Builder** window, under **Results>Electric Field (emw)** click **Multislice**.
- 2 In the **Multislice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field, y component (emw.Ey)**.
- 3 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** edit field, type 0.
- 4 Find the **z-planes** subsection. In the **Planes** edit field, type 0.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.
- 6 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Arrow Volume**.
- 7 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Energy and power>Power flow, time average (emw.Poavx,...,emw.Poavz)**.

- 8 Locate the **Arrow Positioning** section. Find the **y grid points** subsection. In the **Points** edit field, type 1.
- 9 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.
- 10 In the **Model Builder** window, click **Electric Field (emw)**.
- 11 In the **3D Plot Group** settings window, locate the **Data** section.
- 12 From the **Parameter value (alpha)** list, choose **1.22173**.
- 13 Click the **Plot** button.
- 14 Click the **Zoom Extents** button on the Graphics toolbar.

The plot should look like that in [Figure 2](#).

Add a 1D plot to see the reflection and transmission versus the angle of incidence.

ID Plot Group 2

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, locate the **Plot Settings** section.
- 3 Select the **x-axis label** check box.
- 4 In the associated edit field, type **Angle of Incidence**.
- 5 Select the **y-axis label** check box.
- 6 In the associated edit field, type **Reflectance** and **Transmittance**.
- 7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.
- 8 Right-click **Results>ID Plot Group 2** and choose **Global**.
- 9 In the **Global** settings window, locate the **y-Axis Data** section.
- 10 In the table, enter the following settings:

Expression	Unit	Description
<code>abs(emw.S11)^2</code>	1	Reflectance
<code>abs(emw.S21)^2</code>	1	Transmittance

- 11 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 12 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 13 Click to expand the **Legends** section. From the **Legends** list, choose **Automatic**.
- 14 Find the **Include** subsection. Clear the **Expression** check box.
- 15 In the **Model Builder** window, right-click **ID Plot Group 2** and choose **Global**.
- 16 In the **Global** settings window, locate the **y-Axis Data** section.

- I7** In the table, enter the following settings:

Expression	Unit	Description
$\text{abs}(\text{r}_\text{s})^2$		Reflectance, analytic
$\text{n}_\text{slab} \cdot \cos(\beta) / (\text{n}_\text{air} \cdot \cos(\alpha)) \cdot \text{abs}(\text{t}_\text{s})^2$		Transmittance, analytic

- I8** Locate the **Legends** section. From the **Legends** list, choose **Automatic**.

- I9** Find the **Include** subsection. Clear the **Expression** check box.

- I10** Click the **Plot** button.

- I11** Right-click **ID Plot Group 2** and choose **Rename**.

- I12** Go to the **Rename ID Plot Group** dialog box and type **Reflection** and **Transmission** in the **New name** edit field.

- I13** Click **OK**.

Compare the resulting plots with [Figure 4](#).

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 are effectless. Thus, the H-field is perpendicular to the plane of incidence and it is convenient to solve the model for the H-field.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Port 1

- In the **Port** settings window, locate the **Port Mode Settings** section.
- From the **Input quantity** list, choose **Magnetic field**.
- In the **H₀** table, enter the following settings:

0	x
$\exp(-i \cdot k_\text{ax} \cdot x)$ [A/m]	y
0	z

Port 2

- In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 2**.
- In the **Port** settings window, locate the **Port Mode Settings** section.
- From the **Input quantity** list, choose **Magnetic field**.

- 4** In the **H₀** table, enter the following settings:

0	x
$\exp(-i*k_{bx}*x) [A/m]$	y
0	z

Perfect Electric Conductor 2

The model utilizes the H-field for the TM case and the remaining boundaries need to be perfect magnetic conductors.

- I** In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** right-click **Perfect Electric Conductor 2** and choose **Disable**.

Perfect Magnetic Conductor 1

- I** Right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Magnetic Conductor**.

- 2** Select Boundaries 2, 5, 8, and 9 only.

To keep the solution and plots for the TE case, do as follows:

STUDY 1

Solver 1

- I** In the **Model Builder** window, under **Study 1>Solver Configurations** right-click **Solver 1** and choose **Solution>Copy**.

RESULTS

Electric Field (emw)

- I** In the **Model Builder** window, under **Results** Ctrl-click to select both **Results>Electric Field (emw)** and **Results>Reflection and Transmission**, then right-click and choose **Duplicate**.
- 2** In the **3D Plot Group** settings window, locate the **Data** section.
- 3** From the **Data set** list, choose **Solution 2**.

Reflection and Transmission

- I** In the **Model Builder** window, under **Results** click **Reflection and Transmission**.
- 2** In the **ID Plot Group** settings window, locate the **Data** section.
- 3** From the **Data set** list, choose **Solution 2**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS*Electric Field (emw) 1*

- 1 In the **Model Builder** window, under **Results>Electric Field (emw) 1** click **Multislice 1**.
- 2 In the **Multislice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>Magnetic field, y component (emw.Hy)**.
- 3 Click the **Plot** button.

This reproduces [Figure 3](#).

Reflection and Transmission 1

- 1 In the **Model Builder** window, expand the **Results>Reflection and Transmission 1** node, then click **Global 2**.
- 2 In the **Global** settings window, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
$\text{abs}(r_p)^2$		Reflectance, analytic
$n_{\text{slab}} \cdot \cos(\beta) / (n_{\text{air}} \cdot \cos(\alpha)) \cdot \text{abs}(t_p)^2$		Transmittance, analytic

- 4 Click the **Plot** button.

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 model 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 model

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.

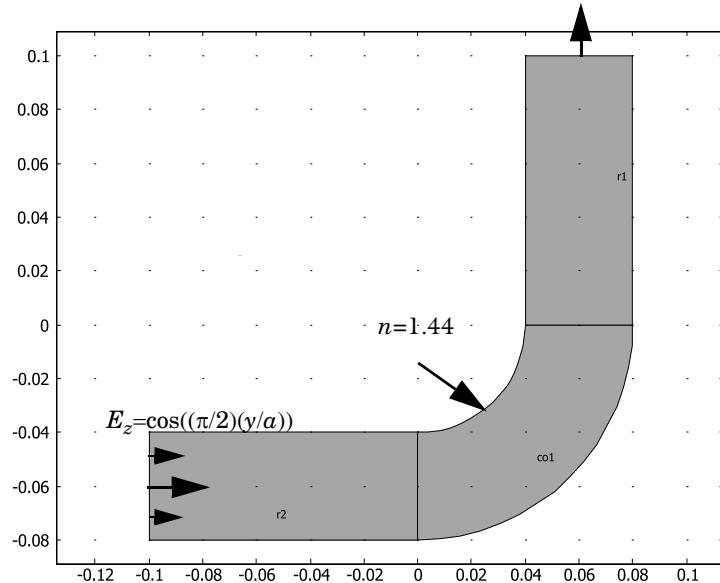


Figure 1: The geometry of the model.

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.

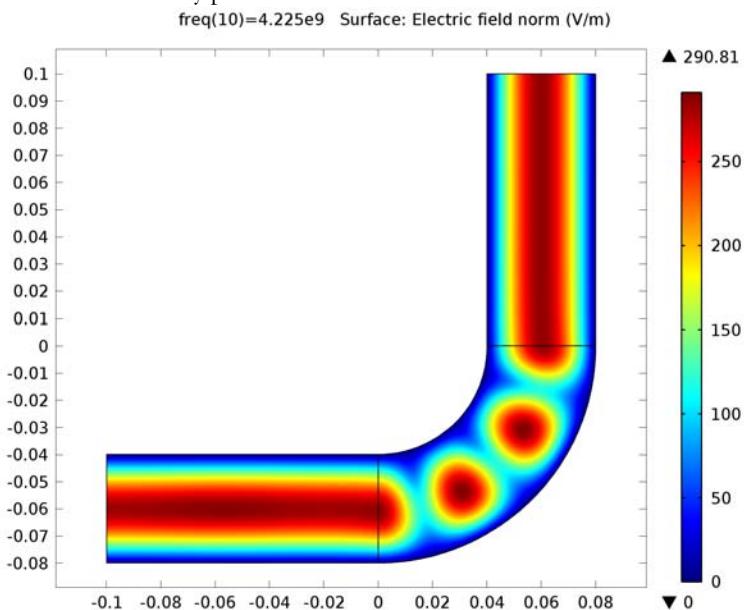


Figure 2: The electric field norm at a frequency of 4.225 GHz.

Figure 3 shows the S-parameters in a dB scale. The result agrees very well with that of the 3D model.

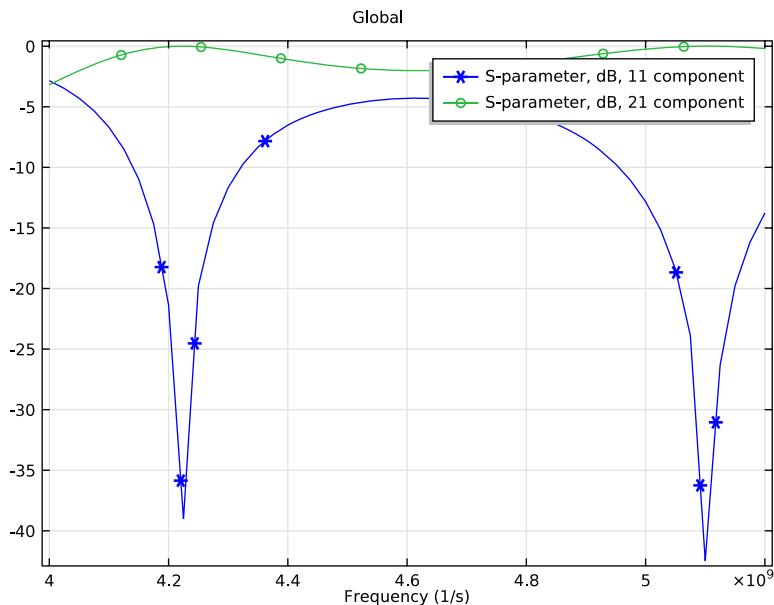


Figure 3: The S-parameters, in a dB scale, as functions of the frequency.

Model Library path: RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_2d

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

7 Click **Finish**.

GEOMETRY I

Circle 1

- 1** In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Circle**.
- 2** In the **Circle** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type **0.08**.

Circle 2

- 1** In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.
- 2** In the **Circle** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type **0.04**.

Square 1

- 1** Right-click **Geometry 1** and choose **Square**.
- 2** In the **Square** settings window, locate the **Size** section.
- 3** In the **Side length** edit field, type **0.08**.
- 4** Locate the **Position** section. In the **y** edit field, type **-0.08**.

Compose 1

- 1** Right-click **Geometry 1** and choose **Boolean Operations>Compose**.
- 2** Click in the **Graphics** window, press **Ctrl+A** to highlight all objects, and then right-click to confirm the selection.
- 3** In the **Compose** settings window, locate the **Compose** section.
- 4** In the **Set formula** edit field, type **sq1*(c1-c2)**.

Rectangle 1

- 1** Right-click **Geometry 1** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type **0.04**.
- 4** In the **Height** edit field, type **0.1**.
- 5** Locate the **Position** section. In the **x** edit field, type **0.04**.

Rectangle 2

- 1** Right-click **Geometry 1** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.

- 3** In the **Width** edit field, type 0.1.
- 4** In the **Height** edit field, type 0.04.
- 5** Locate the **Position** section. In the **x** edit field, type -0.1.
- 6** In the **y** edit field, type -.08.
- 7** Click the **Build All** button.
- 8** Click the **Zoom Extents** button on the Graphics toolbar.

MATERIALS

Material 1

- 1** In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2** Right-click **Material 1** and choose **Rename**.
- 3** Go to the **Rename Material** dialog box and type Air in the **New name** edit field.
- 4** Click **OK**.
- 5** Select Domains 1 and 3 only.
- 6** In the **Material** settings window, locate the **Material Properties** section.
- 7** In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index (n)**.
- 8** Click **Add to Material**.
- 9** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Refractive index	n	1

Material 2

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** Right-click **Material 2** and choose **Rename**.
- 3** Go to the **Rename Material** dialog box and type Silica Glass in the **New name** edit field.
- 4** Click **OK**.
- 5** Select Domain 2 only.
- 6** In the **Material** settings window, locate the **Material Properties** section.
- 7** In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index (n)**.
- 8** Click **Add to Material**.

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

Property	Name	Value
Refractive index	n	1.44

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

With TE waves, only the z-component of the electric field needs to be solved for.

- In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Components** section.
- From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Wave Equation, Electric I

- In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Wave Equation, Electric I**.
- In the **Wave Equation, Electric** settings window, locate the **Electric Displacement Field** section.
- 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

- In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- Select Boundary 1 only.
- In the **Port** settings window, locate the **Port Properties** section.
- From the **Type of port** list, choose **Rectangular**.
- From the **Wave excitation at this port** list, choose **On**.

Port 2

- Right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- Select Boundary 7 only.
- In the **Port** settings window, locate the **Port Properties** section.
- In the **Port name** edit field, type 2.
- From the **Type of port** list, choose **Rectangular**.

MESH I

- 1** In the **Model Builder** window, under **Model I** click **Mesh I**.
- 2** In the **Mesh** settings window, locate the **Mesh Settings** section.
- 3** From the **Element size** list, choose **Extra fine**.
- 4** Click the **Build All** button.

STUDY I*Step I: Frequency Domain*

- 1** In the **Model Builder** window, under **Study I** click **Step I: Frequency Domain**.
- 2** In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3** In the **Frequencies** edit field, type `range(4e9,0.025e9,5.2e9)`.
This gives you a range of frequencies from 4 GHz to 5.2 GHz, with a pitch of 0.025 GHz.
- 4** In the **Model Builder** window, right-click **Study I** and choose **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 **2D Plot Group** settings window, locate the **Data** section.
- 3** From the **Parameter value (freq)** list, choose **4.225e9**.
- 4** Click the **Plot** button.

Finally, plot the S-parameters.

ID Plot Group 2

- 1** In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2** Right-click **ID Plot Group 2** and choose **Global**.
- 3** In the **Global** settings window, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, II component (emw.SIIdB)**.

- 4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 2I component (emw.S21dB)**.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 In the **Expression** edit field, type `freq`.
- 7 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 8 Locate the **Legends** section. Select the **Show legends** check box.
- 9 Click the **Plot** button.

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 model examines a TE wave, one that has no electric field component in the direction of propagation. More specifically, for this model you select the frequency and waveguide dimension so that TE₁₀ 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 model, to make the simulation and the results more interesting, the bend will be filled with Silica glass, a dielectric medium.

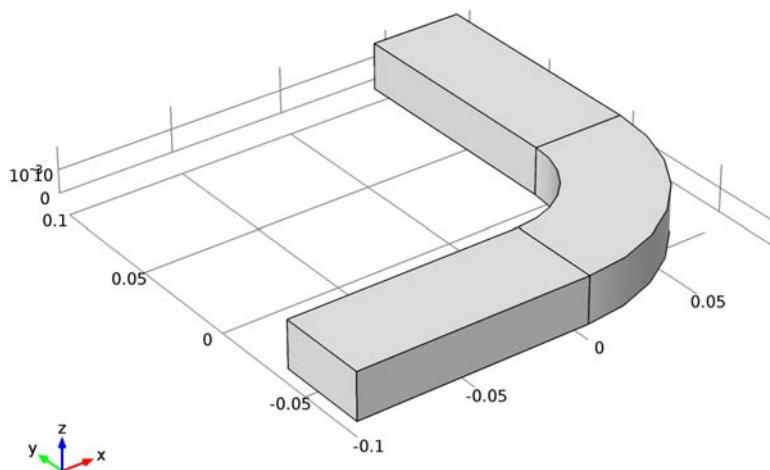
The model also shows how to systematically compute and export all S-parameters to a Touchstone file.

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.

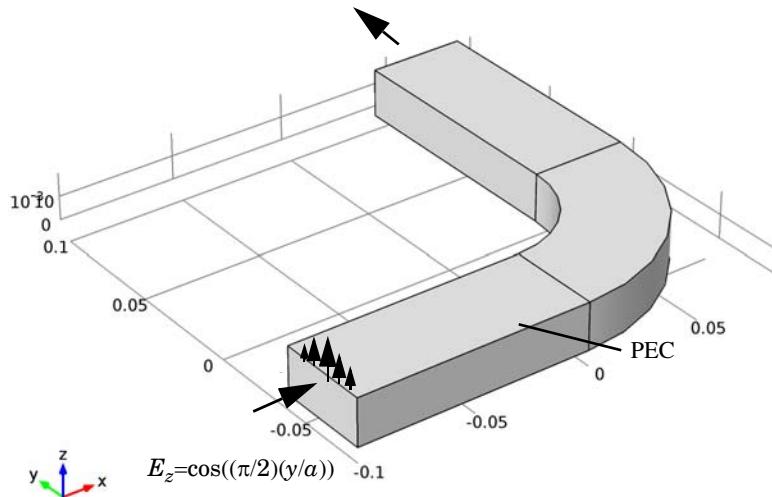
The geometry is as follows:



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 physics 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 $\mathbf{E} = \text{Re}\{(0, 0, \sin(\pi(a-y)/(2a))e^{j\omega t})\} = \text{Re}\{\mathbf{E}e^{j\omega t}\}$, 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

$$(v_c)_{mn} = \frac{c}{2} \sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

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 $(v_c)_{10} = 3.7$ GHz, $(v_c)_{01} = 7.5$ GHz, $(v_c)_{11} = 8.4$ GHz. The frequencies used in this model 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 will be 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.

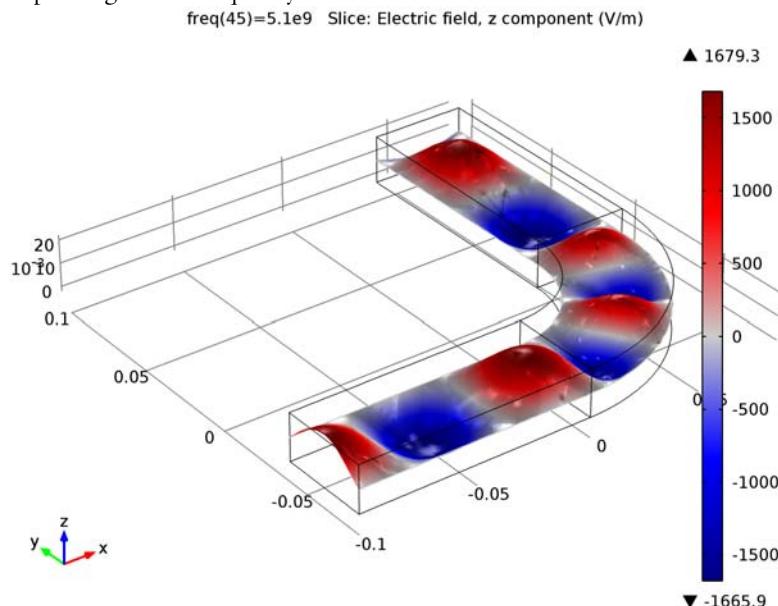


Figure 1: The z-component of the electric field for a frequency of 5.1 GHz.

The S-parameters are shown as functions of the frequency in [Figure 2](#).

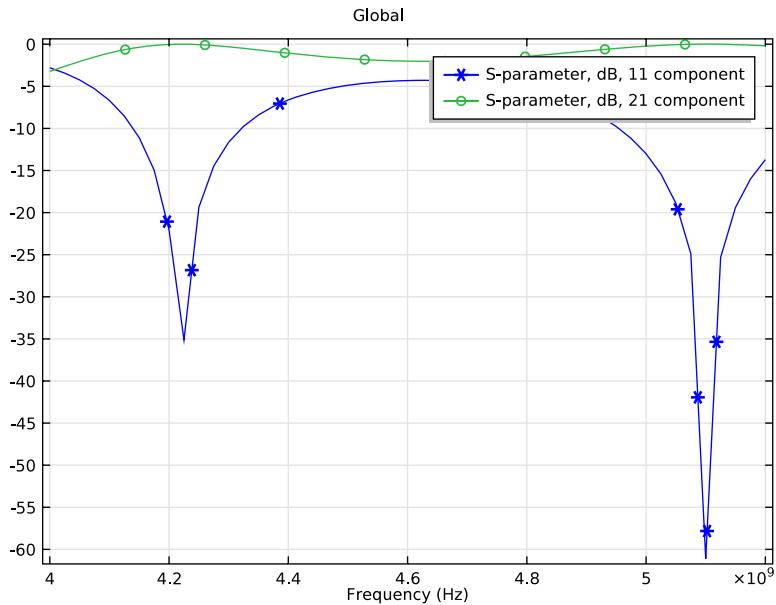


Figure 2: The S-parameters, on a dB scale, as a function of the frequency.

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.)

Model Library path: RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_3d

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

6 Click **Finish**.

G E O M E T R Y I

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

Circle 1

- 1** In the **Model Builder** window, under **Model I>Geometry I>Work Plane I** right-click **Plane Geometry** and choose **Circle**.
- 2** In the **Circle** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type **0.08**.

Circle 2

- 1** Right-click **Plane Geometry** and choose **Circle**.
- 2** In the **Circle** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type **0.04**.

Square 1

- 1** Right-click **Plane Geometry** and choose **Square**.
- 2** In the **Square** settings window, locate the **Size** section.
- 3** In the **Side length** edit field, type **0.08**.
- 4** Locate the **Position** section. In the **yw** edit field, type **-0.08**.

Compose I

- 1** Right-click **Plane Geometry** and choose **Boolean Operations>Compose**.
- 2** Click in the **Graphics** window, press **Ctrl+A** to highlight all objects, and then right-click to confirm the selection.
- 3** In the **Compose** settings window, locate the **Compose** section.
- 4** In the **Set formula** edit field, type **sq1*(c1-c2)**.

Rectangle 1

- 1** Right-click **Plane Geometry** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type **0.04**.
- 4** In the **Height** edit field, type **0.1**.

- 5 Locate the **Position** section. In the **xw** edit field, type **0.04**.

Rectangle 2

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type **0.1**.
- 4 In the **Height** edit field, type **0.04**.
- 5 Locate the **Position** section. In the **xw** edit field, type **-0.1**.
- 6 In the **yw** edit field, type **-0.08**.
- 7 Click the **Build All** button.

Extrude 1

- 1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane 1** and choose **Extrude**.
- 2 In the **Extrude** settings window, locate the **Distances from Plane** section.
- 3 In the table, enter the following settings:

Distances (m)
0.02

- 4 Click the **Build All** button.
- 5 Click the **Zoom Extents** button on the Graphics toolbar.

MATERIALS

Material 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2 Right-click **Material 1** and choose **Rename**.
- 3 Go to the **Rename Material** dialog box and type **Air** in the **New name** edit field.
- 4 Click **OK**.
- 5 Select Domains 1 and 3 only.
- 6 In the **Material** settings window, locate the **Material Properties** section.
- 7 In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index (n)**.
- 8 Click **Add to Material**.

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

Property	Name	Value
Refractive index	n	1

Material 2

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** Right-click **Material 2** and choose **Rename**.
- 3** Go to the **Rename Material** dialog box and type **Silica Glass** in the **New name** edit field.
- 4** Click **OK**.
- 5** Select Domain 2 only.
- 6** In the **Material** settings window, locate the **Material Properties** section.
- 7** In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index (n)**.
- 8** Click **Add to Material**.
- 9** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Refractive index	n	1.44

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Wave Equation, Electric 1

- 1** In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Wave Equation, Electric 1**.
- 2** In the **Wave Equation, Electric** settings window, 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** and choose **Port**.
- 2** Select Boundary 1 only.
- 3** In the **Port** settings window, locate the **Port Properties** section.
- 4** From the **Type of port** list, choose **Rectangular**.
- 5** From the **Wave excitation at this port** list, choose **On**.

Port 2

- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- 2 Select Boundary 15 only.
- 3 In the **Port** settings window, locate the **Port Properties** section.
- 4 In the **Port name** edit 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 I

When modeling electromagnetic waves (as well as any other type of waves), it is important to consider the wavelength when creating a mesh. In order to get accurate results, you need to resolve each wavelength with at least some 4-5 mesh elements. To get the most accuracy out of a limited number of elements, it is therefore important that you use a proportionally finer mesh in the bend, where the wavelength is shorter due to the dielectric medium.

Size

- 1 In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **Free Tetrahedral**.
- 2 In the **Size** settings window, locate the **Element Size Parameters** section.
- 3 In the **Maximum element size** edit field, type $3e8 / (5.2e9 * 5)$.

In the air, this gives 5 elements per wavelength at the highest frequency, 5.2 GHz.

Size I

- 1 In the **Model Builder** window, under **Model I>Mesh I** right-click **Free Tetrahedral I** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

- 7 In the associated edit field, type $3e8 / (5.2e9 * 1.44 * 5)$.

This setting gives 5 elements per wavelength in the Silica glass, with its refractive index of 1.44.

- 8 Click the **Build All** button.

If you look closely at the mesh, you can see that it is indeed a bit finer in the bend than elsewhere.

STUDY 1

Step 1: Frequency Domain

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

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

- 3 In the **Frequencies** edit field, type `range(4e9,0.025e9,5.2e9)`.

The range command you just entered means that you will get solutions for frequencies from 4 GHz to 5.2 GHz, with a pitch of 25 MHz. The solution process should only take a few minutes, but if you want to speed it up, you can increase the pitch.

- 4 In the **Model Builder** window, right-click **Study 1** and choose **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.

ID Plot Group 2

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

- 2 Right-click **ID Plot Group 2** and choose **Global**.

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

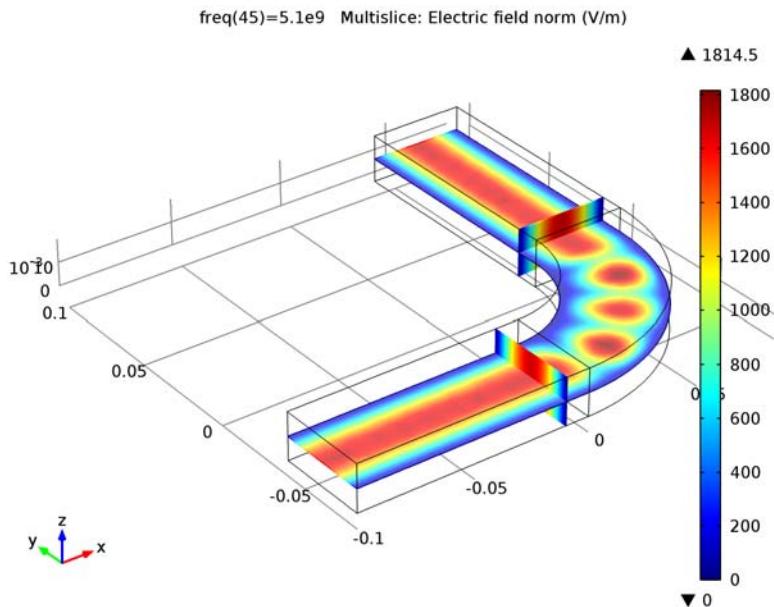
- 4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 21 component (emw.S21dB)**.
- 5 Click to expand the **x-Axis Data** section. Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Frequency (emw.freq)**.
- 6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 7 Click the **Plot** button.

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.

Electric Field (emw)

- 1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Parameter value (freq)** list, choose **5.1e9**.

- 4 Click the **Plot** button.



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.

- 5 In the **Model Builder** window, under **Results>Electric Field (emw)** right-click **Multislice** and choose **Delete**.
- 6 Click **Yes** to confirm.
- 7 Right-click **Electric Field (emw)** and choose **Slice**.
- 8 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field>Electric field, z component (emw.Ez)**.
- 9 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
- 10 In the **Planes** edit field, type 1.
- 11 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

I2 Click the **Plot** button.

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.

I3 Click to expand the **Range** section. Locate the **Coloring and Style** section. Select the **Symmetrize color range** check box.**I4** Right-click **Results>Electric Field (emw)>Slice 1** and choose **Deformation**.**I5** In the **Deformation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field (emw.Ex,emw.Ey,emw.Ez)**.**I6** Click the **Plot** button.

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**I** In the **Model Builder** window, under **Model 1** click **Electromagnetic Waves, Frequency Domain**.**2** In the **Electromagnetic Waves, Frequency Domain** settings window, 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***I** In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.**2** In the **Parameters** settings window, locate the **Parameters** section.**3** In the table, enter the following settings:

Name	Expression
PortName	1

STUDY 1*Parametric Sweep***I** In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.**2** In the **Parametric Sweep** settings window, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter names	Parameter value list
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 Right-click **Study 1** and choose **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.

ID Plot Group 2

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>ID Plot Group 2** click **Global 1**.

2 In the **Global** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

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:

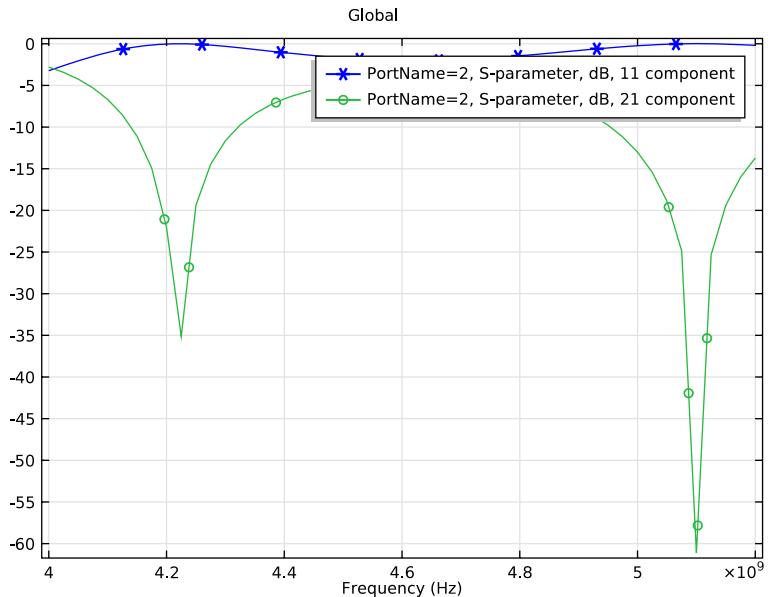
4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 12 component (emw.S12dB)**.

5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 22 component (emw.S22dB)**.

6 Locate the **Data** section. From the **Parameter selection (PortName)** list, choose **From list**.

7 From the **Parameter selection (PortName)** list, choose **Last**.

8 Click the **Plot** button.



Impedance Matching of a Lossy Ferrite 3-port Circulator

For a description of this model, including detailed step-by-step instructions showing how to build it, see the book *Introduction to the RF Module*.

Model Library path: RF_Module/Ferrimagnetic_Devices/
lossy_circulator_3d

Parameterized Circulator Geometry

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 model, including detailed step-by-step instructions showing how to build it, see the book *Introduction to the RF Module*.

Model Library path: RF_Module/Ferrimagnetic_Devices/
lossy_circulator_3d_geom

Defining a Mapped Dielectric Distribution of a Metamaterial Lens

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.

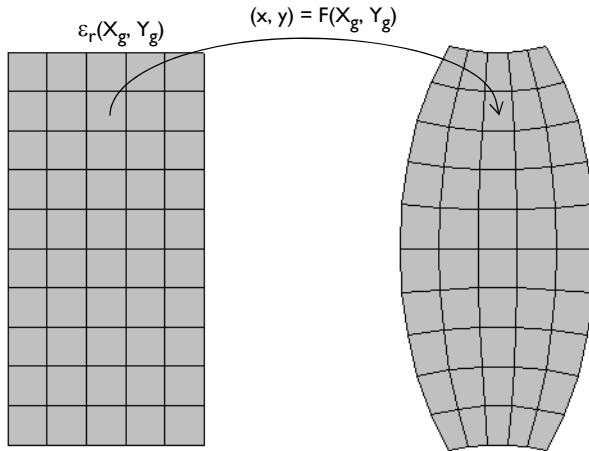


Figure 1: A convex metamaterial lens. Both the shape and the dielectric distribution are defined on a rectangular domain, and mapped into the deformed state.

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.

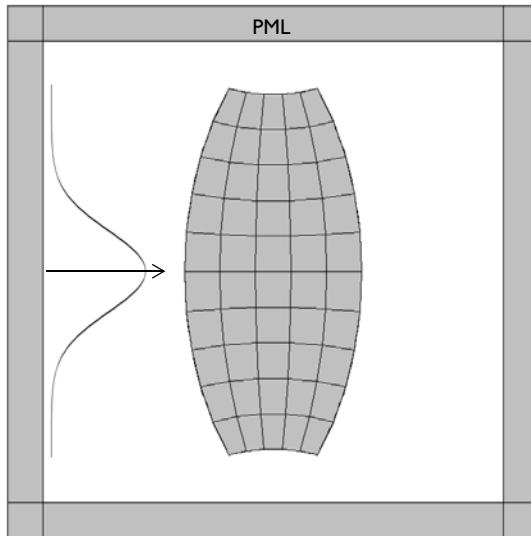


Figure 2: The modeling domain consists of the metamaterial lens in an air domain, and a surrounding PML. A Gaussian beam is incident from the left.

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:

$$\epsilon_r = \left(1 + \frac{1}{2}Y_g^2\right)^2$$

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 functionality is used to define the mapping of the dielectric from the initially rectangular domain onto the desired shape. The deformation and the dielectric distribution within the lens domain is completely specified via the above functions.

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.

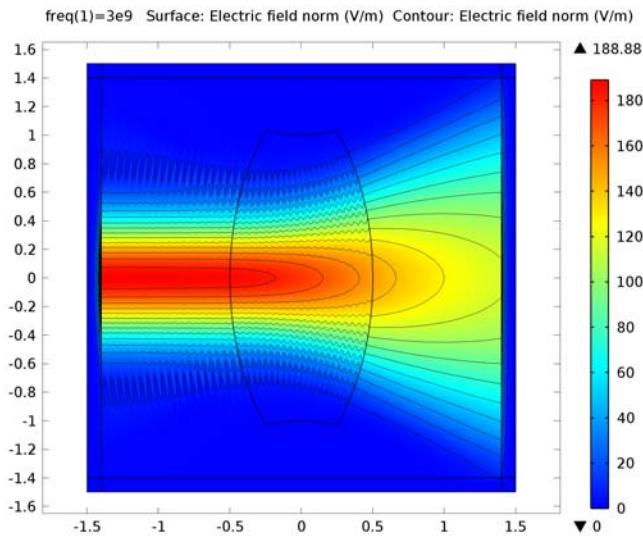


Figure 3: The norm of the electric field shows the Gaussian beam diffracted by the metamaterial lens.

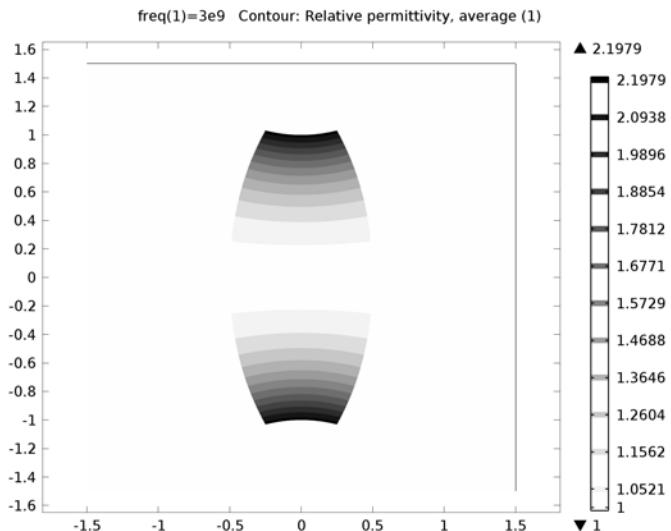


Figure 4: Contour plot of the dielectric distribution.

Model Library path: RF_Module/Tutorial_Models/
mapped_metamaterial_distribution

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.
- 5 Click **Add Selected**.
- 6 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 7 Click **Add Selected**.
- 8 Click **Next**.
- 9 Find the **Studies** subsection. In the tree, select **Custom Studies>Preset Studies for Some Physics>Stationary**.
- 10 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
f0	3[GHz]	Operating frequency
lda0	c_const/f0	Free space wavelength
w0	lda0*4	Gaussian beam waist size

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

G E O M E T R Y I

First, create a square for the entire model domain. Add a layer on each side of the square.

Square 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Square**.
- 2 In the **Square** settings window, locate the **Size** section.
- 3 In the **Side length** edit 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	1da0

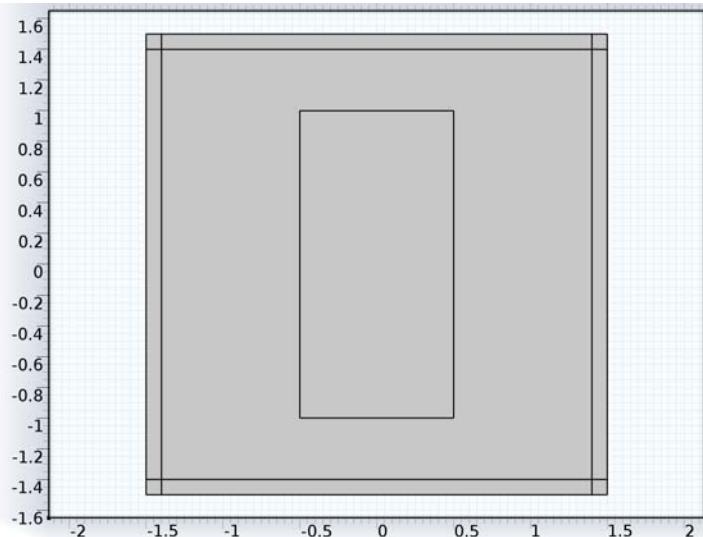
- 6 Select the **Layers to the left** check box.
- 7 Select the **Layers to the right** check box.
- 8 Select the **Layers on top** check box.
- 9 Click the **Build Selected** button.

Add a rectangle for the lens.

Rectangle 1

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Height** edit field, type 2.
- 4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 Click the **Build All** button.



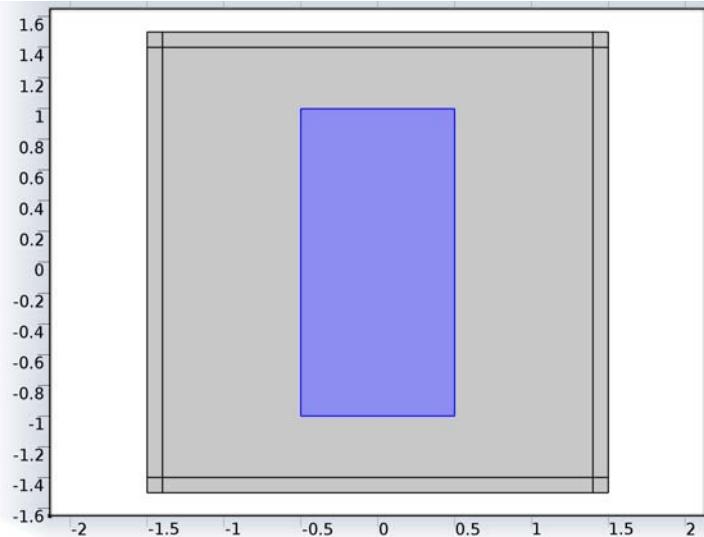
DEFINITIONS

Add a selection for the lens domain which will be recalled frequently while setting up the model properties.

Explicit 1

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

2 Select Domain 7 only.



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

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

5 Click **OK**.

Next, add a set of variables for the shape and the dielectric distribution of the metamaterial lens.

Variables 1a

1 Right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, 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:

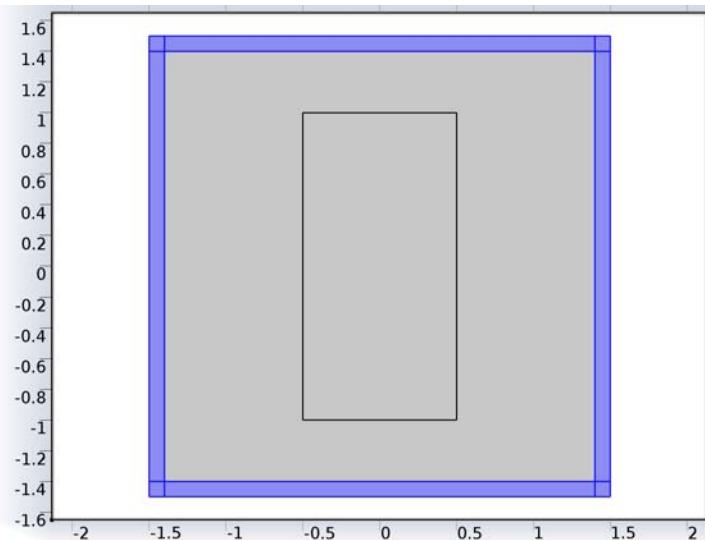
Name	Expression	Description
xp	$0.5[m]*Xg[1/m]*(2-(Yg[1/m])^2)$	Mapping of $Xg \rightarrow x$
yp	$Yg*(1+(0.5*(xp[1/m])^2))$	Mapping of $Yg \rightarrow y$
erp	$(1+0.5*(Yg[1/m])^2)^2$	Dielectric distribution

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

- 1 Right-click **Definitions** and choose **Perfectly Matched Layer**.
- 2 Select Domains 1–4, 6, and 8–10 only.



DEFORMED GEOMETRY

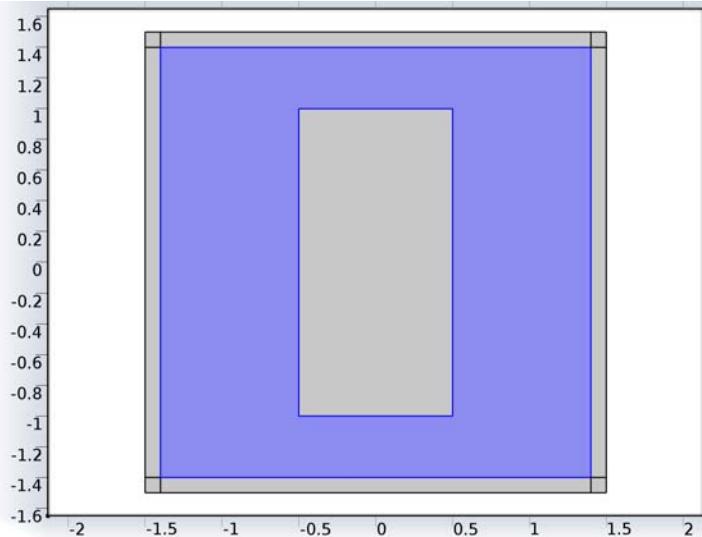
Set up Deformed Geometry. You need to specify Free Deformation, Prescribed Mesh Displacement and Prescribed Deformation.

- 1 In the **Model Builder** window, under **Model 1** click **Deformed Geometry**.
- 2 In the **Deformed Geometry** settings window, locate the **Frame Settings** section.
- 3 From the **Geometry shape order** list, choose **1**.

Free Deformation 1

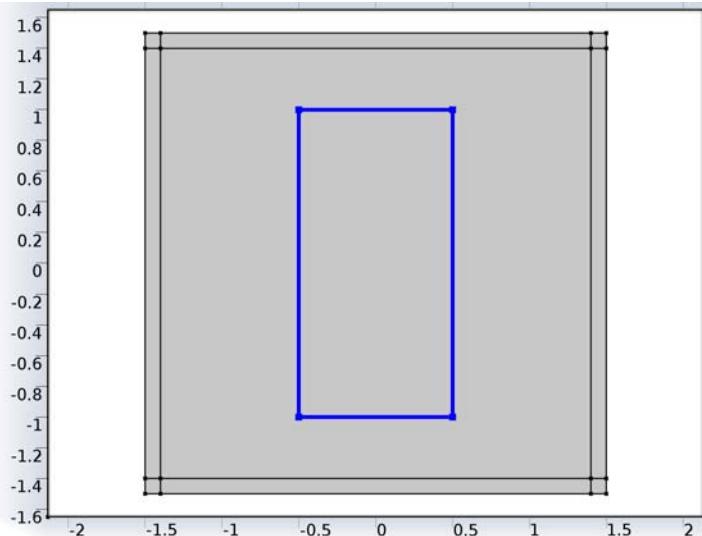
- 1 Right-click **Model 1>Deformed Geometry** and choose **Free Deformation**.

2 Select Domain 5 only.



Prescribed Mesh Displacement 2

- 1 In the **Model Builder** window, right-click **Deformed Geometry** and choose **Prescribed Mesh Displacement**.
- 2 Select Boundaries 15–18 only.



- 3 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.
- 4 In the d_x edit field, type $xp-Xg$.
- 5 In the d_y edit field, type $yp-Yg$.

Prescribed Deformation 1

- 1 Right-click **Deformed Geometry** and choose **Prescribed Deformation**.
- 2 In the **Prescribed Deformation** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Lens**.
- 4 In the **Prescribed Deformation** settings window, locate the **Prescribed Mesh Displacement** section.
- 5 In the d_x edit-field array, type $xp-Xg$ on the first row.
- 6 In the d_y edit-field array, type $yp-Yg$ on the 2nd row.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

In Electromagnetic Waves, the dielectric distribution is configured via the user-defined variable ϵ_{rp} and the Gaussian beam is modeled as entering the domain from the left side, via a surface current excitation.

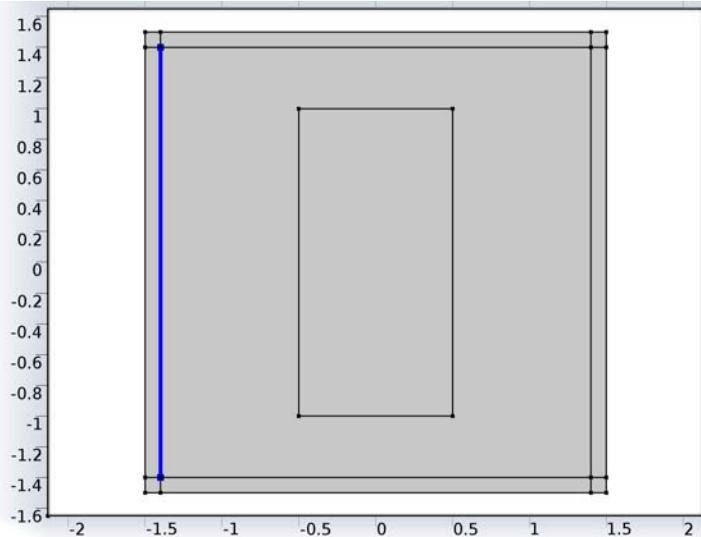
Wave Equation, Electric 2

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Wave Equation, Electric**.
- 2 In the **Wave Equation, Electric** settings window, 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 edit field, type ϵ_{rp} .
- 5 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**.

Surface Current 1

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Surface Current**.

2 Select Boundary 10 only.



3 In the **Surface Current** settings window, locate the **Surface Current** section.

4 In the \mathbf{J}_{s0} table, enter the following settings:

0	x
0	y
$\exp(-(\mathbf{y}/\mathbf{w}0)^2)$	z

MATERIALS

Set all domain with vacuum. The lens domain material properties are explicitly configured by Wave Equation, Electric 2 in Electromagnetic Waves.

Material Browser

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

MESH 1

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

Size

- 1** In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.
- 2** In the **Size** settings window, locate the **Element Size** section.
- 3** Click the **Custom** button.
- 4** Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `lda0/10`.
- 5** In the **Minimum element size** edit field, type `0.0012`.
- 6** In the **Model Builder** window, right-click **Mesh 1** and choose **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 **Stationary** settings window, locate the **Physics and Variables Selection** section.
- 3** In the table, enter the following settings:

Physics	Solve for
Electromagnetic Waves, Frequency Domain	x

Add a Frequency Domain study step and set as solved only for Electromagnetic Waves, Frequency Domain.

Step 2: Frequency Domain

- 1** In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Frequency Domain**.
- 2** In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3** In the **Frequencies** edit field, type `f0`.
- 4** Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics	Solve for
Deformed Geometry	x

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

RESULTS

The default plot shows the magnitude of electric fields. Change the default color pattern and add a contour plot for the magnitude.

Electric Field (emw)

- 1 In the **Model Builder** window, under **Results>Electric Field (emw)** click **Surface**.
- 2 In the **Surface** settings window, locate the **Coloring and Style** section.
- 3 From the **Color table** list, choose **RainbowLight**.
- 4 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Contour**.
- 5 In the **Contour** settings window, locate the **Levels** section.
- 6 In the **Total levels** edit field, type 14.
- 7 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 8 From the **Color** list, choose **Black**.
- 9 Clear the **Color legend** check box.

See [Figure 3](#) to compare the reproduced plot.

Add a filled contour plot describing the dielectric distribution over the lens.

2D Plot Group 2

- 1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.
- 2 Right-click **2D Plot Group 2** and choose **Contour**.
- 3 In the **Contour** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Material properties>Relative permittivity, average (emw.epsrAv)**.
- 4 In the **Contour** settings window, locate the **Levels** section.
- 5 In the **Total levels** edit field, type 12.
- 6 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.
- 7 From the **Color table** list, choose **GrayScale**.
- 8 Select the **Reverse color table** check box.

The plot for the dielectric distribution is shown in [Figure 4](#).

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 500 W, 2.45 GHz microwave source via a rectangular waveguide operating in the TE₁₀ mode. Near the bottom of the oven there is a cylindrical glass plate with a spherical potato placed on top of it. A part of the potato is cut away for mechanical stability, which also facilitates the creation of a finite element mesh in the region where it is in contact with the plate. Symmetry is utilized by simulating only half of the problem. The symmetry cut is applied vertically through the oven, waveguide, potato, and plate. [Figure 1](#) below shows the reduced geometry.

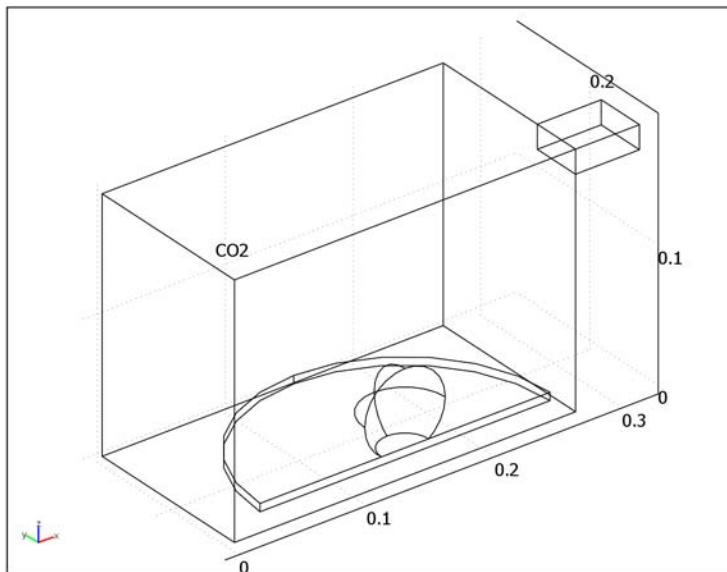


Figure 1: Geometry of microwave oven, potato, and waveguide feed.

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 TE₁₀ mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes are given analytically from the relation

$$(v_c)_{mn} = \frac{c}{2} \sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers and c denotes the speed of light. For the TE₁₀ mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 7.8$ cm and $b = 1.8$ cm), the TE₁₀ 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 v is given by the expression

$$\beta = \frac{2\pi}{c} \sqrt{v^2 - v_c^2}$$

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:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \left(\epsilon_r - \frac{j\sigma}{\omega \epsilon_0} \right) \mathbf{E} = 0$$

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 $\mu_r = \epsilon_r = 1$. In the potato the same parameters are used except for the permittivity which is set to $\epsilon_r = 65 - 20j$ where the imaginary part accounts for dielectric losses. The glass plate has $\sigma = 0$, $\mu_r = 1$ and $\epsilon_r = 2.55$.

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.

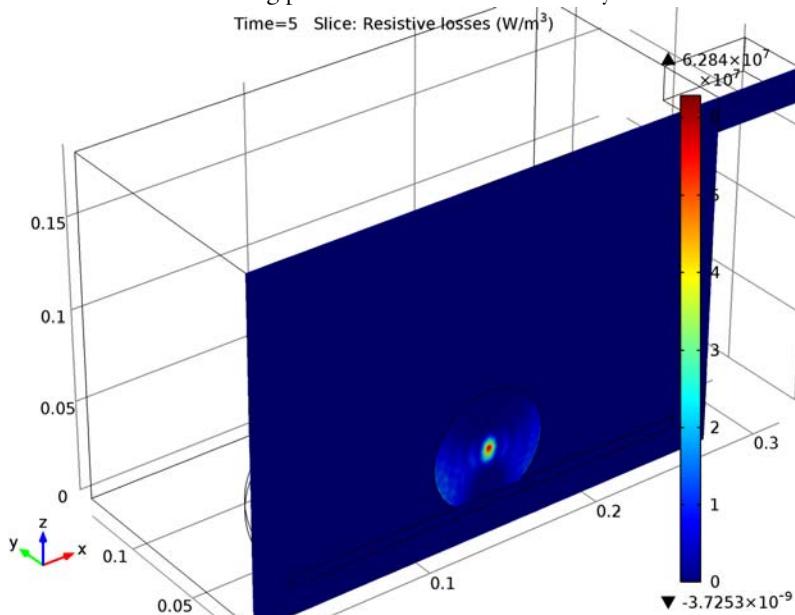


Figure 2: Dissipated microwave power distribution (W/m^3).

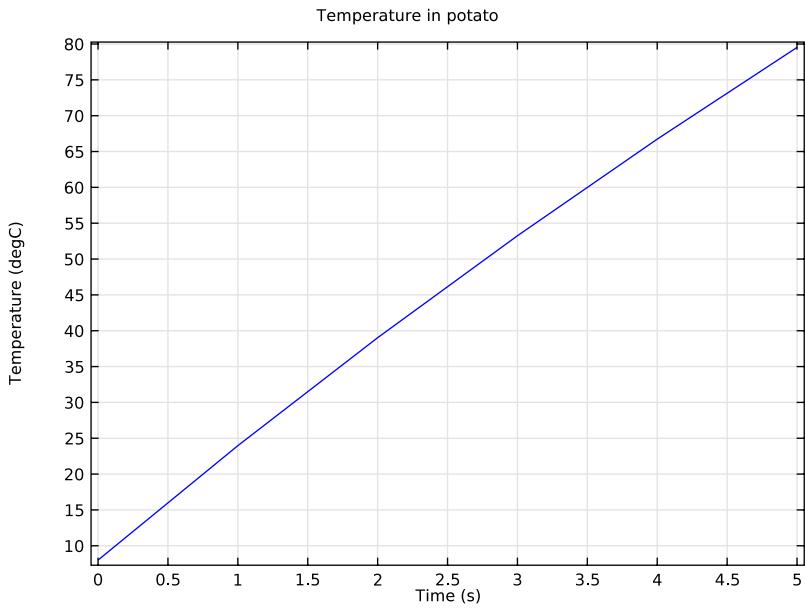


Figure 3: Temperature in the center of the potato during the first 5 seconds of heating.

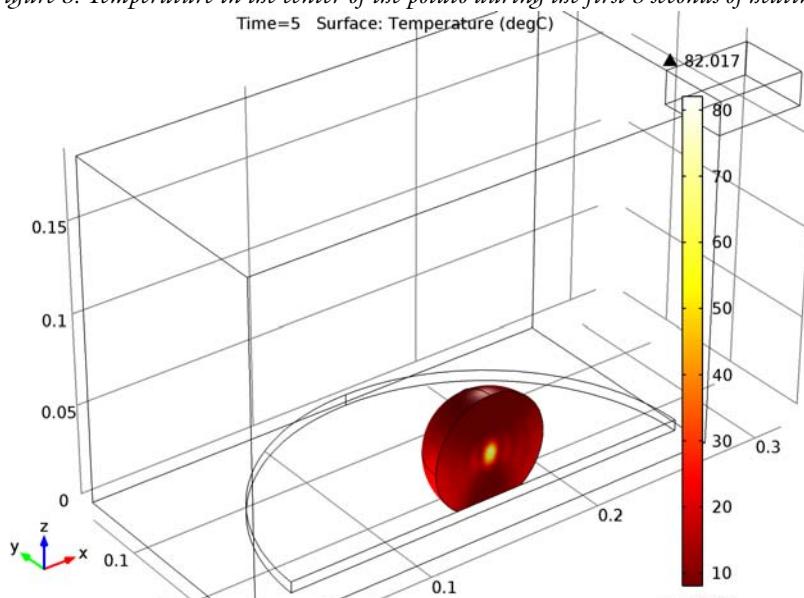


Figure 4: Temperature distribution after 5 seconds of heating.

Model Library path: RF_Module/Microwave_Heating/microwave_oven

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Heat Transfer>Electromagnetic Heating>Microwave Heating (mh)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency-Transient**.
- 6 Click **Finish**.

GLOBAL DEFINITIONS

First, define a set of parameters for creating the geometry.

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
w0	267[mm]	Oven width
do	270[mm]	Oven depth
ho	188[mm]	Oven height
wg	50[mm]	Waveguide width
dg	78[mm]	Waveguide depth
hg	18[mm]	Waveguide height
rp	113.5[mm]	Glass plate radius
hp	6[mm]	Glass plate height
bp	15[mm]	Glass plate base
rpot	31.5[mm]	Potato radius
T0	8[degC]	Initial potato temperature

GEOMETRY I*Block 1*

- 1** In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Block**.
- 2** In the **Block** settings window, locate the **Size and Shape** section.
- 3** In the **Width** edit field, type **wo**.
- 4** In the **Depth** edit field, type **do/2**.
- 5** In the **Height** edit field, type **ho**.

Block 2

- 1** In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.
- 2** In the **Block** settings window, locate the **Size and Shape** section.
- 3** In the **Width** edit field, type **wg**.
- 4** In the **Depth** edit field, type **dg/2**.
- 5** In the **Height** edit field, type **hg**.
- 6** Locate the **Position** section. In the **x** edit field, type **wo**.
- 7** In the **z** edit field, type **ho-hg**.

Cylinder 1

- 1** Right-click **Geometry 1** and choose **Cylinder**.
- 2** In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type **rp**.
- 4** In the **Height** edit field, type **hp**.
- 5** Locate the **Position** section. In the **x** edit field, type **wo/2**.
- 6** In the **z** edit field, type **bp**.

Sphere 1

- 1** Right-click **Geometry 1** and choose **Sphere**.
- 2** In the **Sphere** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type **rpot**.
- 4** Locate the **Position** section. In the **x** edit field, type **wo/2**.
- 5** In the **z** edit field, type **rpot+bp**.
- 6** Click the **Build All** button.

The sphere you have created for the potato now overlaps the glass plate. This in itself is not a problem, but where the sphere touches the bottom of the glass plate, you

risk getting very thin mesh elements. To avoid this problem, you will delete the part of the sphere that overlaps the cylinder. To retain the cylinder after this operation, begin by making a copy of it.

Copy |

- 1 Right-click **Geometry 1** and choose **Transforms>Copy**.
- 2 Select the object **cyl1** only. The object cyl1 is the cylinder.

Difference |

- 1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.
- 2 Select the object **sph1** only to add it to the **Objects to add** list. The object sph1 is the sphere.
- 3 In the **Difference** settings window, locate the **Difference** section.
- 4 Under **Objects to subtract**, click **Activate Selection**.
- 5 Select the object **cyl1** only.
- 6 Click the **Build All** button.

Finally, make a geometric operation to keep only the part of the potato and the plate that overlaps the half oven.

Compose |

- 1 Right-click **Geometry 1** and choose **Boolean Operations>Compose**.
- 2 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.
- 3 In the **Compose** settings window, locate the **Compose** section.
- 4 In the **Set formula** edit field, type $(blk1+blk2)*(dif1+copy1)$.
- 5 Select the **Keep input objects** check box.
- 6 Click the **Build All** button.

Delete Entities |

- 1 Right-click **Geometry 1** and choose **Delete Entities**.
- 2 In the **Delete Entities** settings window, locate the **Entities or Objects to Delete** section.
- 3 From the **Geometric entity level** list, choose **Object**.
- 4 Select the objects **dif1** and **copy1** only.
- 5 Click the **Build All** button.
- 6 Click the **Wireframe Rendering** button on the Graphics toolbar.

DEFINITIONS

Create the following selections definitions in order to make Domain and Boundary selections easier as you walk through these model instructions. Note that if you have problems finding certain numbers, you can always choose View > Selection List.

Explicit 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 1** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Potato** in the **New name** edit field.
- 4 Click **OK**.
- 5 Select Domain 3 only.

Explicit 2

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 2** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Plate** in the **New name** edit field.
- 4 Click **OK**.
- 5 Select Domain 2 only.

Explicit 3

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 3** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Air** in the **New name** edit field.
- 4 Click **OK**.
- 5 Select Domains 1 and 4 only.

Explicit 4

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 4** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **No Heat Transfer** in the **New name** edit field.
- 4 Click **OK**.
- 5 Select Domains 1, 2, and 4 only.

Explicit 5

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

- 2 Right-click **Explicit 5** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Port Boundary** in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Explicit** settings window, locate the **Input Entities** section.
- 6 From the **Geometric entity level** list, choose **Boundary**.
- 7 Select Boundary 23 only.

Explicit 6

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 6** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Symmetry Boundaries** in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Explicit** settings window, locate the **Input Entities** section.
- 6 From the **Geometric entity level** list, choose **Boundary**.
- 7 Select Boundaries 2, 7, 10, and 19 only.

Explicit 7

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.
- 2 Right-click **Explicit 7** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type **Metal Boundaries** in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Explicit** settings window, locate the **Input Entities** section.
- 6 From the **Geometric entity level** list, choose **Boundary**.
- 7 Select Boundaries 1, 3–5, 17, and 20–22 only.

MATERIALS

Next, define the materials. Air and Copper are already in the Material Library.

Material Browser

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

3 Click **Add Material to Model.**

Air

- 1** In the **Model Builder** window, under **Model 1>Materials** click **Air**.
- 2** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Selection** list, choose **Air**.

Material 2

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** Right-click **Material 2** and choose **Rename**.
- 3** Go to the **Rename Material** dialog box and type **Potato** in the **New name** edit field.
- 4** Click **OK**.
- 5** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 6** From the **Selection** list, choose **Potato**.
- 7** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Electric conductivity	sigma	0
Relative permittivity	epsilon_r	65 - 20*j
Relative permeability	mur	1
Thermal conductivity	k	0.55
Density	rho	1050
Heat capacity at constant pressure	Cp	3.64e3

Material 3

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** Right-click **Material 3** and choose **Rename**.
- 3** Go to the **Rename Material** dialog box and type **Glass** in the **New name** edit field.
- 4** Click **OK**.
- 5** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 6** From the **Selection** list, choose **Plate**.

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

Property	Name	Value
Electric conductivity	sigma	0
Relative permittivity	epsilon_r	2.55
Relative permeability	mu_r	1

You do not need to define the listed thermal properties, as the glass plate will not be in the thermal part of the model.

Material Browser

- In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
- In the **Material Browser** settings window, In the tree, select **Built-In>Copper**.
- Click **Add Material to Model**.

Copper

- In the **Model Builder** window, under **Model 1>Materials** click **Copper**.
- In the **Material** settings window, locate the **Geometric Entity Selection** section.
- From the **Geometric entity level** list, choose **Boundary**.
- From the **Selection** list, choose **Metal Boundaries**.

MICROWAVE HEATING

It is now time to set up the physics. The first thing to do is to assign an Electromagnetic Waves model to the domains not taking part in the thermal problem. This will override the Microwave Heating model which by default is present everywhere.

Wave Equation, Electric I

- In the **Model Builder** window, under **Model 1** right-click **Microwave Heating** and choose the domain condition **Electromagnetic Waves>Wave Equation, Electric**.
- In the **Wave Equation, Electric** settings window, locate the **Domain Selection** section.
- From the **Selection** list, choose **No Heat Transfer**.

For the electromagnetic part of the problem, begin by defining the input port.

Port I

- In the **Model Builder** window, right-click **Microwave Heating** and choose the boundary condition **Electromagnetic Waves>Port**.
- In the **Port** settings window, locate the **Boundary Selection** section.
- From the **Selection** list, choose **Port Boundary**.

4 Locate the **Port Properties** section. From the **Wave excitation at this port** list, choose **On**.

5 In the P_{in} edit field, type 500.

6 Locate the **Port Mode Settings** section. In the \mathbf{E}_0 table, enter the following settings:

0	x
0	y
$\cos(\pi y/dg) [V/m]$	z

7 In the β edit field, type $2*\pi/c_const*sqrt(freq^2-c_const^2/(4*dg^2))$.

This is the propagation constant for the first propagating mode.

Next, set up the remaining boundary conditions.

Impedance Boundary Condition 1

1 Right-click **Microwave Heating** and choose the boundary condition **Electromagnetic Waves>Impedance Boundary Condition**.

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

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

Perfect Magnetic Conductor 1

1 Right-click **Microwave Heating** and choose the boundary condition **Electromagnetic Waves>Perfect Magnetic Conductor**.

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

3 From the **Selection** list, choose **Symmetry Boundaries**.

This concludes the electromagnetic part of the physics. The thermal part will automatically get the heat source from the microwave heating model). Thermal insulation is the default boundary condition, so the only setting you need to make is the initial temperature.

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Microwave Heating** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the T edit field, type T0.

MESH I

To ensure convergence and get an accurate result, the mesh in this model is required to everywhere resolve the wavelength. This is most critical in the potato, where the high permittivity results in a wavelength of just above 15 mm.

Free Tetrahedral I

In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **Free Tetrahedral**.

Size I

- 1 In the **Model Builder** window, under **Model I > Mesh I** right-click **Free Tetrahedral I** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Potato**.
- 5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.
- 6 Click the **Custom** button.
- 7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 8 In the associated edit field, type **6[mm]**.
- 9 Click the **Build All** button.

STUDY I

Step I: Frequency-Transient

- 1 In the **Model Builder** window, under **Study I** click **Step I: Frequency-Transient**.
- 2 In the **Frequency-Transient** settings window, locate the **Study Settings** section.
- 3 In the **Times** edit field, type **range(0,1,5)**. This will give you output at every second from $t = 0$ s to $t = 5$ s.
- 4 In the **Frequency** edit field, type **2.45 [GHz]**.
- 5 In the **Model Builder** window, right-click **Study I** and choose **Compute**.

RESULTS

Temperature (mh)

The Graphics window shows the temperature distribution on the surface of the potato after 5 s. Change the unit to degC to reproduce [Figure 4](#).

- 1 In the **Model Builder** window, under **Results>Temperature (mh)** click **Surface 1**.
- 2 In the **Surface** settings window, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.
- 4 Click the **Plot** button.

Create another plot group to plot the resistive heating on the symmetry plane.

3D Plot Group 2

- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 Right-click **3D Plot Group 2** and choose **Slice**.
- 3 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Heating and losses>Resistive losses (mh.Qrh)**.
- 4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.
- 5 From the **Entry method** list, choose **Coordinates**.
- 6 Click the **Plot** button.

The plot should now look like [Figure 2](#). Next, add a nice visualization of the electromagnetic fields to the temperature plot.

Temperature (mh)

- 1 In the **Model Builder** window, under **Results** right-click **Temperature (mh)** and choose **Slice**.
- 2 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Electric>Electric field>Electric field, z component (Ez)**.
- 3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
- 4 From the **Entry method** list, choose **Coordinates**.
- 5 In the **z-coordinates** edit field, type **0.1**.
- 6 Right-click **Results>Temperature (mh)>Slice 1** and choose **Deformation**.
- 7 In the **Deformation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Electric>Electric field (Ex,Ey,Ez)**.
- 8 Click the **Plot** button.

Derived Values

Make a volume integral of the microwave heating to find out how much of the energy is absorbed in the potato.

- 1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Integration>Volume Integration**.

Select one point in time for the output. Since the material parameters of the potato are independent of the temperature, it does not matter which time you choose.

- 2 In the **Volume Integration** settings window, locate the **Data** section.

- 3 From the **Time selection** list, choose **From list**.

- 4 In the **Times** list, select **0**.

- 5 Locate the **Selection** section. From the **Selection** list, choose **Potato**.

- 6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Heat Transfer in Solids)>Total heat source (mh.Qtot)**.

- 7 Click the **Evaluate** button.

The result is 312 W. Finally, to reproduce [Figure 3](#), create a plot of temperature in the center of the potato as a function of time.

Data Sets

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Point 3D**.

- 2 In the **Cut Point 3D** settings window, locate the **Point Data** section.

- 3 In the **x** edit field, type 0.134.

- 4 In the **y** edit field, type 0.

- 5 In the **z** edit field, type 0.047.

1D Plot Group 3

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

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

- 3 From the **Data set** list, choose **Cut Point 3D 1**.

- 4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

- 5 In the **Title** text area, type **Temperature in potato**.

- 6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

- 7 In the associated edit field, type **Time (s)**.

- 8 Right-click **Results>ID Plot Group 3** and choose **Point Graph**.
- 9 In the **Point Graph** settings window, locate the **y-Axis Data** section.
- 10 From the **Unit** list, choose **degC**.
- 11 Click the **Plot** button.

Microwave Filter on PCB

This model 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. The entire layout including the dielectric layer is imported from an ODB++(X) file using the ECAD Import feature.

Model Definition

The model uses the Electromagnetic Waves interface that solves the vector Helmholtz wave equation. The PCB layout of the Chebyshev filter is imported using the ECAD Import feature, which creates the entire 3D geometry from the layout and stackup information of an ODB++ file.

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 ECAD import has an option that causes the import engine to ignore the thickness of metal layers and insert them as faces on the dielectric layer. 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 model 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 model 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. In reality, the applied stress and strain will also cause stress-optical effects, see the model [Stress-Optical Effects in a Photonic Waveguide](#). Given relevant material data, such effects could rather easily be incorporated in this model.

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](#).

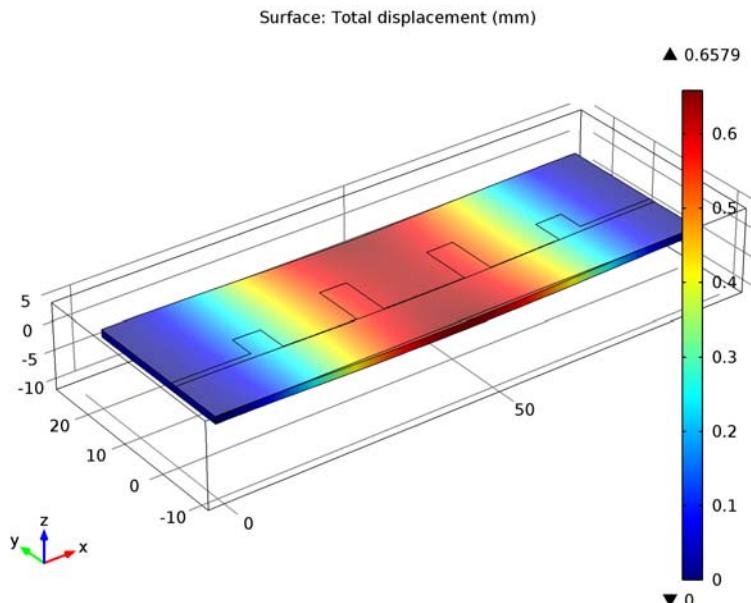


Figure 1: The graph shows the total displacement of the PCB, due to the load.

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.

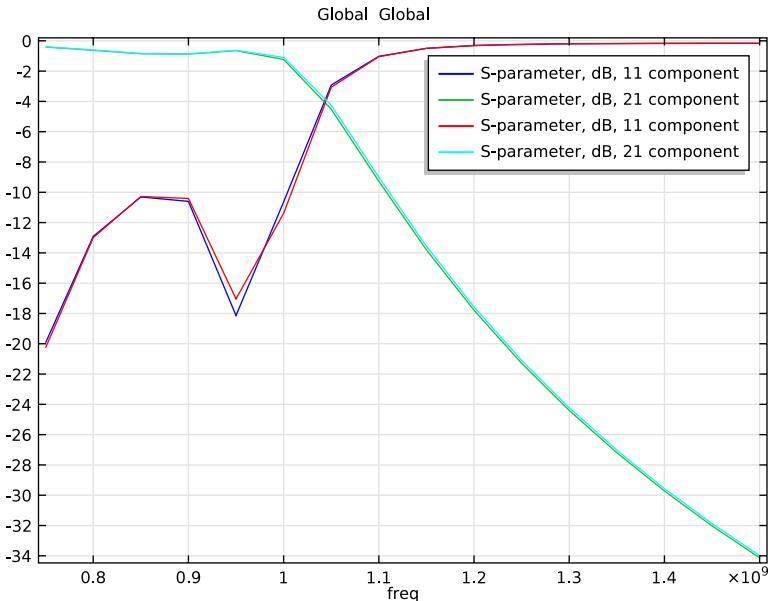


Figure 2: A comparison between the S_{11} and S_{21} parameters before and after a force of 40 N has been applied on the PCB. The red and turquoise lines correspond to the S-parameter curves for the filter with an applied force.

References

1. D.V. Tosic and M. Potrebic, “Software Tools for Research and Education,” *Microwave Review*, vol. 12, no. 2, p. 45, 2006.
 2. J.-S.G. Hong and M.J. Lancaster, *Microstrip Filters for RF/Microwave Applications*, John Wiley, 2001.
-

Model Library path: RF_Module/Passive_Devices/
pcb_microwave_filter_with_stress

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 6 Click **Finish**.

GLOBAL DEFINITIONS

The following steps define the parameters for the frequency sweep.

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
fstart	750[MHz]	Start frequency
fstop	1.5[GHz]	Stop frequency
fstep	50[MHz]	Frequency step

GEOMETRY I

Set mm as the default unit for length.

- 1 In the **Model Builder** window, under **Model I** click **Geometry I**.
- 2 In the **Geometry** settings window, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.

The major part of the geometry is imported using the ECAD Import tool, which automatically reads a PCB layout and extrudes the layers to a 3D geometry.

Block I

- 1 Right-click **Model I>Geometry I** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type 100.

- 4** In the **Depth** edit field, type 40.
- 5** In the **Height** edit field, type 15.
- 6** Locate the **Position** section. In the **x** edit field, type -5.
- 7** In the **y** edit field, type -15.
- 8** In the **z** edit field, type -10.

Import /

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

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

3 From the **Geometry import** list, choose **ECAD file (ODB++)**.

4 Click the **Browse** button.

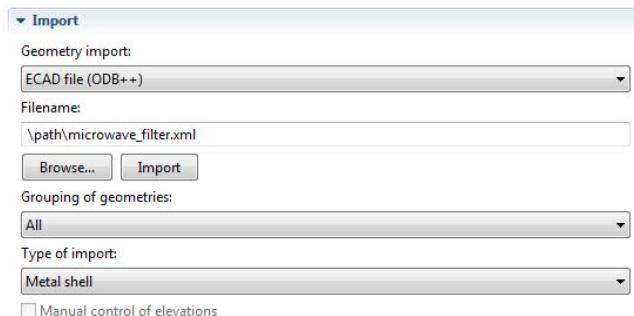
5 Browse to the model's Model Library folder and double-click the file `pcb_microwave_filter_with_stress.xml`.

It is located in the same folder as specified in the Model Library Path on page ??.

This path originates from the folder **models** under the COMSOL Multiphysics installation folder.

6 From the **Type of import** list, choose **Metal shell**.

This will import all metal layers as faces, which drastically reduce the problem size in this model without compromising the accuracy.



7 Click the **Import** button.

The PCB is now placed within the drawn block. To see the PCB, select wireframe rendering.

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

The next step is to add boundary faces for the input and output ports.

Work Plane 1

- 1 Right-click **Geometry 1** and choose **Work Plane**.
- 2 In the **Work Plane** settings window, locate the **Plane Definition** section.
- 3 From the **Plane type** list, choose **Face parallel**.
- 4 On the object **imp1**, select Boundary 2 only.

Rectangle 1

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

The size and position of the rectangle, as defined below, will perfectly fit it to the microstrip line.

- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type **1.125**.
- 4 In the **Height** edit field, type **1.27**.
- 5 Locate the **Position** section. In the **xw** edit field, type **-4.77**.
- 6 In the **yw** edit field, type **-0.635**.

Work Plane 1

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

This action embeds the 2D rectangle in the 3D geometry.

Work Plane 2

- 1 Right-click **Geometry 1** and choose **Work Plane**.
- 2 In the **Work Plane** settings window, locate the **Plane Definition** section.
- 3 From the **Plane type** list, choose **Face parallel**.
- 4 On the object **imp1**, select Boundary 5 only.

Rectangle 1

- 1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 2** right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type **1.125**.
- 4 In the **Height** edit field, type **1.27**.
- 5 Locate the **Position** section. In the **xw** edit field, type **3.645**.
- 6 In the **yw** edit field, type **-0.635**.

Work Plane 2

- In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane 2** and choose **Build Selected**.

MATERIALS

Air

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

FR4 (Circuit Board)

- In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
- In the **Material Browser** settings window, In the tree, select **Built-In>FR4 (Circuit Board)**.
- Click **Add Material to Model**.
- In the **Model Builder** window, under **Model 1>Materials** click **FR4 (Circuit Board)**.
- 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.

- In the **Material** settings window, locate the **Material Contents** section.
- In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon0r	10.8

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Scattering Boundary Condition 1

- In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Scattering Boundary Condition**.
- Select Boundaries 1–5 and 18 only.

Lumped Port 1

- In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- Select Boundary 10 only.

- 3 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4 From the **Wave excitation at this port** list, choose **On**. This port excites the microstrip line.

Lumped Port 2

- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 Select Boundary 16 only.

Perfect Electric Conductor 2

- 1 Right-click **Electromagnetic Waves, Frequency Domain** 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 I

Free Triangular I

- 1 In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **More Operations>Free Triangular**.
- 2 Select Boundaries 10 and 16 only.

Size I

- 1 Right-click **Model I>Mesh I>Free Triangular I** and choose **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 5 In the associated edit field, type 1.

The Maximum element size is reduced to 1 mm to increase the accuracy of the S-parameter calculations.

- 6 In the **Model Builder** window, right-click **Mesh I** and choose **Free Tetrahedral**.

Size

- 1 In the **Model Builder** window, under **Model I>Mesh I** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Coarse**.
- 4 Click the **Custom** button.

- 5 Locate the **Element Size Parameters** section. In the **Minimum element size** edit field, type 0.1.

Allow the Minimum element size to be as small as 0.1 mm to generate a fine mesh along the thin microstrip.

- 6 Click the **Build All** button.

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.

Global Variable Probe 1

- 1 In the **Model Builder** window, under **Model 1>Definitions** click **Global Variable Probe 1**.
- 2 In the **Global Variable Probe** settings window, locate the **Probe Settings** section.
- 3 In the **Probe variable** edit field, type S11.
- 4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 11 component (emw.S11dB)**.

Global Variable Probe 2

- 5 In the **Model Builder** window, under **Model 1>Definitions** click **Global Variable Probe 2**.
- 6 In the **Global Variable Probe** settings window, locate the **Probe Settings** section.
- 7 In the **Probe variable** edit field, type S21.
- 8 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 21 component (emw.S21dB)**.

STUDY 1

Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, click to expand the **Results While Solving** section.
- 3 Select the **Plot** check box.
- 4 Locate the **Study Settings** section. Click the **Range** button.

- 5 Go to the **Range** dialog box.
- 6 In the **Start** edit field, type `fstart`.
- 7 In the **Stop** edit field, type `fstop`.
- 8 In the **Step** edit field, type `fstep`.
- 9 Click the **Replace** button.
- 10 Click the **Compute** button.

RESULTS

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 ID Plot Group 2** displays the S_{11} - and S_{21} -parameters for the frequency sweep.

ADDITION OF SOLID MECHANICS AND MOVING MESH TO THE MODEL

The following instructions adds physics from the Solid Mechanics and the Moving Mesh interfaces for the simulations of the deformed PCB.

- 1 In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 3 Click **Add Selected**.
- 4 In the **Add physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.
- 5 Click **Add Selected**.
- 6 Click **Finish**.

SOLID MECHANICS

- 1 In the **Model Builder** window, under **Model 1** click **Solid Mechanics**.
- 2 Select Domain 2 only.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.

- 3** In the table, enter the following settings:

Name	Expression	Description
fload	40[N]	Load on PCB

DEFINITION OF PCB VOLUME

The following steps describe how to measure the volume of the PCB and then copy and paste the value in a parameter definition.

GEOMETRY I

Import I

- 1** Click the **Select Domains** button on the Graphics toolbar.

- 2** In the **Model Builder** window, under **Model I>Geometry I** click **Import I**.

- 3** On the object **impl**, highlight Domain 1 only.

- 4** 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 (by pasting in the previously copied volume).

- 5** In the **Model Builder** window, under **Global Definitions** click **Parameters**.

- 6** In the **Parameters** settings window, locate the **Parameters** section.

- 7** In the table, enter the following settings:

Name	Expression	Description
V	3357.0[mm^3]	Volume of PCB

MOVING MESH

In the **Model Builder** window, expand the **Model I>Moving Mesh** node.

Prescribed Deformation I

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

- 2** Select Domain 2 only.

- 3** In the **Prescribed Deformation** settings window, locate the **Prescribed Mesh Displacement** section.

- 4** In the d_x edit-field array, type u on the first row.

- 5** In the d_y edit-field array, type v on the 2nd row.

- 6** In the d_z edit-field array, type w on the 3rd row.

Free Deformation 1

- 1** Right-click **Moving Mesh** and choose **Free Deformation**.
- 2** Select Domain 1 only.

Prescribed Mesh Displacement 2

- 1** Right-click **Moving Mesh** and choose **Prescribed Mesh Displacement**.
- 2** In the **Prescribed Mesh Displacement** settings window, locate the **Boundary Selection** section.
- 3** From the **Selection** list, choose **All boundaries**.
- 4** Select Boundaries 6–17 only.
- 5** Locate the **Prescribed Mesh Displacement** section. In the d_x edit field, type u .
- 6** In the d_y edit field, type v .
- 7** In the d_z edit field, type w .

SOLID MECHANICS*Body Load 1*

- 1** In the **Model Builder** window, under **Model 1** right-click **Solid Mechanics** and choose **Body Load**.
- 2** Select Domain 2 only.
- 3** In the **Body Load** settings window, locate the **Force** section.
- 4** In the **F_V** table, enter the following settings:

0	x
0	y
-fload/V	z

Fixed Constraint 1

- 1** In the **Model Builder** window, right-click **Solid Mechanics** 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 **Frequency Domain** settings window, locate the **Physics and Variables Selection** section.

- 3** In the table, enter the following settings:

Physics	Solve for
Solid Mechanics	x
Moving Mesh	x

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Find the **Studies** subsection. In the tree, select **Custom Studies>Preset Studies for Some Physics>Stationary**.
- 3 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	Solve for
Electromagnetic Waves, Frequency Domain (emw)	x

- 4** Click **Finish**.

STUDY 2

Step 2: Frequency Domain

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Study Steps>Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Physics and Variables Selection** section.
- 3 In the table, enter the following settings:

Physics	Solve for
Solid Mechanics	x
Moving Mesh	x

- 4 Locate the **Study Settings** section. Click the **Range** button.
- 5 Go to the **Range** dialog box.
- 6 In the **Start** edit field, type **fstart**.
- 7 In the **Stop** edit field, type **fstop**.
- 8 In the **Step** edit field, type **fstep**.

- 9 Click the **Replace** button.
- 10 In the **Frequency Domain** settings window, locate the **Results While Solving** section.
- 11 Select the **Plot** check box.
- 12 From the **Plot group** list, choose **Probe ID Plot Group 2**.
- 13 Right-click **Study 2** and choose **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.

To compare the S-parameters for the initial and the stressed PCB, make a new 1D Plot Group and add the S-parameters from the two different solutions.

ID Plot Group 5

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Right-click **ID Plot Group 5** and choose **Global**.
- 3 In the **Global** settings window, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 1I component (emw.S11dB)**.
- 4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 2I component (emw.S21dB)**.
- 5 In the **Model Builder** window, right-click **ID Plot Group 5** and choose **Global**.
- 6 In the **Global** settings window, locate the **Data** section.
- 7 From the **Data set** list, choose **Solution 3**.
- 8 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **S-parameter, dB, 1I component (emw.S11dB)**.
- 9 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **S-parameter, dB, 2I component (emw.S21dB)**.
- 10 Click the **Plot** button.

You should now see the plot in [Figure 2](#).

Stress (solid)

- 1 In the **Model Builder** window, under **Results>Stress (solid)>Surface 1** right-click **Deformation** and choose **Disable**.
- 2 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
- 3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Solid Mechanics>Displacement>Total displacement (solid.disp)**.
- 4 In the **Model Builder** window, click **Stress (solid)**.
- 5 In the **3D Plot Group** settings window, locate the **Data** section.
- 6 From the **Data set** list, choose **Solution 4**.
- 7 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.
- 8 Click the **Plot** button.

You should now see the plot in [Figure 1](#).

Plasmonic Wire Grating

Introduction

A plane electromagnetic wave is incident on a wire grating on a dielectric substrate. The model computes transmission and reflection coefficients for the refraction, specular reflection, and first order diffraction.

Model Definition

Figure 1 shows the considered grating, with a gold wire on a dielectric material with refractive index n_β . The grating constant, or the distance between the wires, is d . A plane-polarized wave traveling through a medium with refractive index n_α is incident on the grating, at an angle α in a plane perpendicular to the grating.

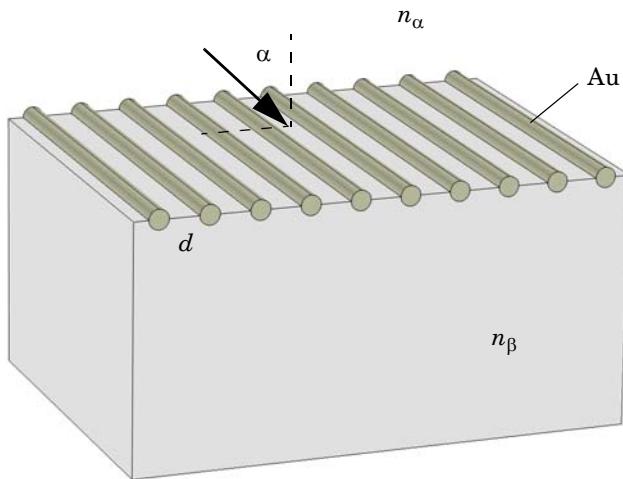


Figure 1: The modeled grating. The model considers a unit cell of a slice through this geometry. The grating is assumed to consist of an infinite number of infinitely long wires.

If the wavelengths involved in the model are sufficiently short compared to the grating constant, one or several diffraction orders can be present. The diagram in Figure 2

shows two transmissive paths taken by light incident on adjacent cells of the grating, exactly one grating constant apart.

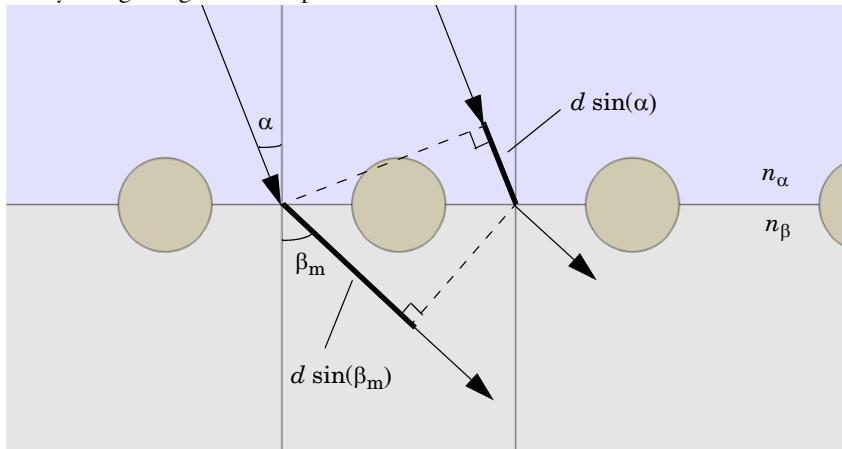


Figure 2: The geometric path lengths of two transmitted parallel beams. The optical path length is the geometric path length multiplied by the local refractive index.

The criterion for positive interference is that the difference in optical path length along the two paths equals an integer number of vacuum wavelengths, or:

$$m\lambda_0 = d(n_\beta \sin \beta_m - n_\alpha \sin \alpha) \quad (1)$$

with $m = 0, \pm 1, \pm 2, \dots$, λ_0 the vacuum wavelength, and β_m the transmitted diffracted beam of order m . For $m = 0$, this reduces to refraction, as described by Snell's law:

$$\sin \beta_0 = \frac{n_\alpha}{n_\beta} \sin \alpha$$

Because the sine functions can only vary between -1 and 1, the existence of higher diffraction order requires that

$$-(n_\alpha + n_\beta) < \frac{m\lambda_0}{d} < (n_\alpha + n_\beta)$$

The model instructions cover only first order diffraction, and are hence only valid for under the condition

$$2\lambda_0 > d(n_\alpha |\sin \alpha| + n_\beta) \quad (2)$$

Note that for the special cases of perpendicular and grazing incidence, the right-hand side of the inequality evaluates to $d n_\beta$ and $d(n_\alpha + n_\beta)$ respectively.

Figure 3 shows the corresponding paths of the reflected light.

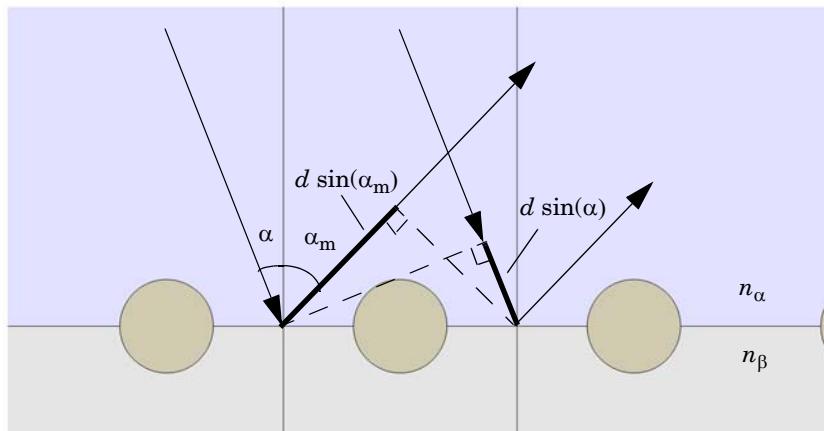


Figure 3: The geometric path lengths of two parallel reflected beams.

For positive interference we get

$$m\lambda_0 = d n_\alpha (\sin \alpha_m - \sin \alpha), \quad (3)$$

where α_m is the reflected beam of diffraction order m . Setting $m = 0$ in this equation renders

$$\sin \alpha_0 = \sin \alpha,$$

or specular reflection. The condition for no reflected diffracted beams of order 2 or greater being present is

$$2\lambda_0 > d n_\alpha (1 + |\sin \alpha|). \quad (4)$$

The model uses $n_\alpha = 1$ for air and $n_\beta = 1.2$ for the dielectric substrate. Allowing for arbitrary angles of incidence and with a grating constant $d = 400$ nm, Equation 2 sets the validity limit to vacuum wavelengths greater than 440 nm. The model uses $\lambda_0 = 441$ nm. For the wire, a complex-valued permittivity of $-1.75 - 5.4i$ approximates that of gold at the corresponding frequency.

The performance of the grating depends on the polarization of the incident wave. Therefore both a transverse electric (TE) and a transverse magnetic (TM) case are considered. The TE wave has the electric field component in the z direction, out of the modeling xy -plane. For the TM wave, the electric field vector is pointing in the xy -plane and perpendicular to the direction of propagation, whereas the magnetic field

has only a component in the z direction. The angle of incidence is for both cases swept from 0 to $\pi/2$, with a pitch of $\pi/40$.

Results and Discussion

As an example of the output from the model, [Figure 4](#) and [Figure 5](#) show the electric field norm for an angle of incidence equal to $\pi/5$, for the TE and TM case respectively.

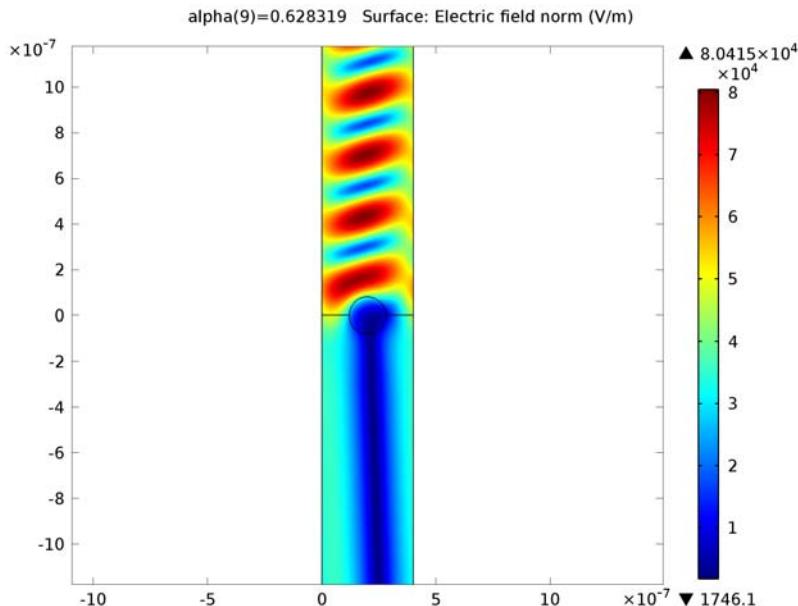


Figure 4: Electric field norm for TE incidence at $\pi/5$.

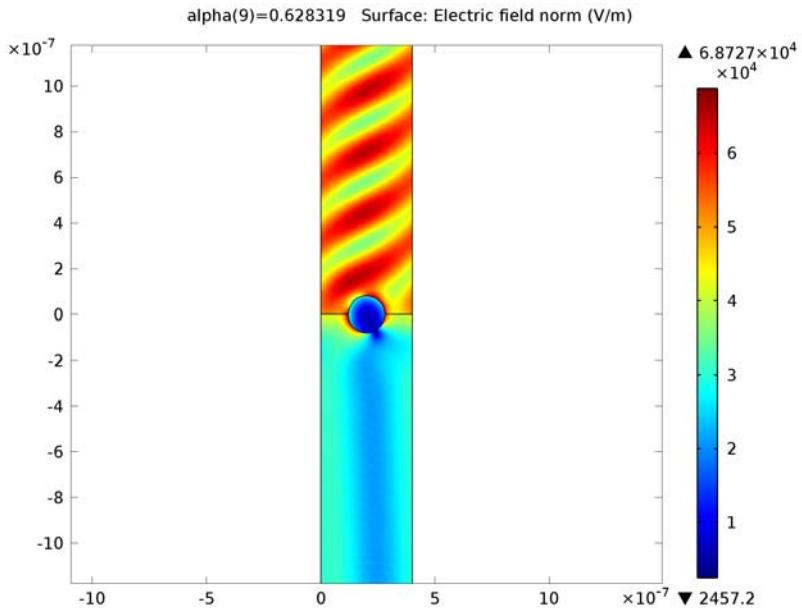


Figure 5: Electric field norm for TM incidence at $\pi/5$.

All the computed transmission and reflection coefficients for TE incidence are plotted in Figure 6. R_0 , the coefficient for specular reflection, increases rather steadily with the angle of incidence. This is both because of reflection in the material interface and because the wave “sees” the wire as increasingly wider at greater angles—the same effect as achieved by a Venetian blind. T_0 , the refracted but not diffracted transmission, decreases accordingly. For the considered wavelength to period length ratio, the transmitted diffracted beam T_{-1} is propagating only for nearly perpendicular incidence. The reflected diffraction order R_1 would need a shorter wavelength or a larger grating period to show up. Instead, the most prominent diffraction orders are R_{-1} and T_1 .

Note first that the sum of all coefficients is consistently less than 1. This is because of the dielectric losses in the wire. This is even more apparent for TM incidence, as Figure 7 shows. Here, approximately half of the wave is absorbed in the wire. Another important feature of the TM case is that there is very little specular reflection (R_0) around 60 degrees.

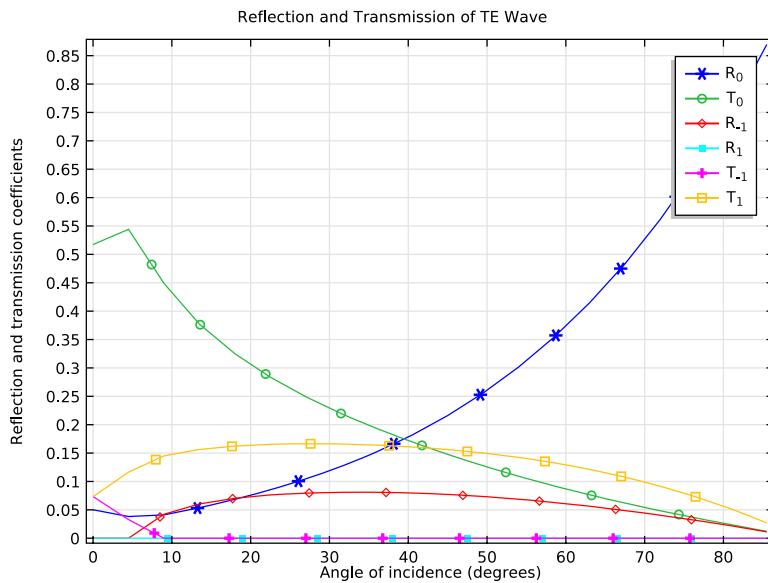


Figure 6: Transmission and reflection coefficients for TE incidence.

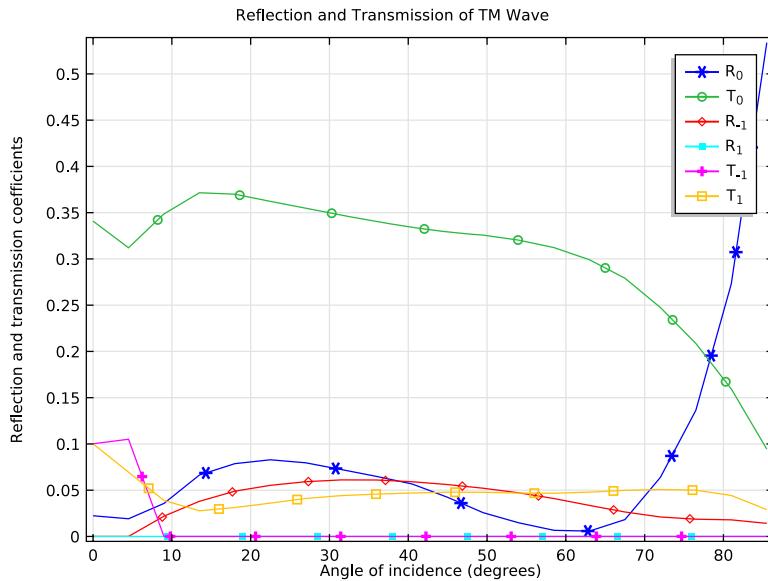


Figure 7: Transmission and reflection coefficients for TM incidence.

Notes About the COMSOL Implementation

The model is set up for one unit cell of the grating, flanked by Floquet boundary conditions describing the periodicity. As applied, this condition states that the solution on one side of the unit equals the solution on the other side multiplied by a complex-valued phase factor. The phase shift between the boundaries is evaluated from the perpendicular component of the wave vector. Because the periodicity boundaries are parallel with the y -axis, only the x -component is required. Note that due to the continuity of the field, the phase factor will be the same for the refracted and reflected beams as for the incident wave.

Port conditions are used both for specifying the incident wave and for letting the resulting solution leave the model without any non-physical reflections. In order to achieve perfect transmission through the port boundaries, one port for each mode ($m = 0, m = -1, m = 1$) in each direction must be present. This gives a total of 6 ports.

The input to each periodic port is an electric or magnetic field amplitude vector and an angle of incidence. The angle of incidence is defined as

$$\mathbf{k} \times \mathbf{n} = k \sin \alpha \mathbf{z},$$

where \mathbf{k} is the propagation vector of the incident wave, \mathbf{n} is the normalized normal vector, k is the wave number, α is the angle of incidence, and \mathbf{z} is the unit vector in the z direction. Note that this definition means that the angle of incidence on the opposite sides have opposite signs. To automatically create ports for the diffraction orders, you also provide the refractive index at the port boundary and the maximum frequency (which in this model is the single frequency that is used).

The below table lists the parameters names used in the model. “Internal” means that the variable is not provided as an input parameter

TABLE 4-1: PARAMETER NAMES

MODEL DESCRIPTION	MODEL	DESCRIPTION
n_α	na	Refractive index, air
n_β	nb	Refractive index, dielectric
α	alpha	Angle of incidence
α_1	Internal	Reflected diffraction angle, order 1
α_{-1}	Internal	Reflected diffraction angle, order -1
β_0	beta	Refraction angle

TABLE 4-I: PARAMETER NAMES

MODEL DESCRIPTION	MODEL	DESCRIPTION
β_1	Internal	Refracted diffraction angle, order 1
β_{-1}	Internal	Refracted diffraction angle, order -1

Model Library path: RF_Module/Tutorial_Models/plasmonic_wire_grating

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

- 3** In the table, enter the following settings:

Name	Expression	Description
na	1	Refractive index, air
nb	1.2	Refractive index, dielectric
d	400 [nm]	Grating constant
lam0	441 [nm]	Vacuum wavelength
f0	c_const/lam0	Frequency
alpha	0	Angle of incidence
beta	asin(na*sin(alpha)/nb)	Refraction angle

Although the angle of incidence will not remain constant at 0, it needs to be specified as a parameter to be accessible to the parametric solver.

GEOMETRY I

Create the geometry entirely in terms of the grating constant, for easy scalability.

Rectangle 1

- In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.
- In the **Rectangle** settings window, locate the **Size** section.
- In the **Width** edit field, type **d**.
- In the **Height** edit field, type **3*d**.
- Click the **Build Selected** button.

Rectangle 2

- In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.
- In the **Rectangle** settings window, locate the **Size** section.
- In the **Width** edit field, type **d**.
- In the **Height** edit field, type **3*d**.
- Locate the **Position** section. In the **y** edit field, type **-3*d**.
- Click the **Build Selected** button.

Circle 1

- Right-click **Geometry 1** and choose **Circle**.
- In the **Circle** settings window, locate the **Size and Shape** section.
- In the **Radius** edit field, type **d/5**.

4 Locate the **Position** section. In the **x** edit field, type $d/2$.

5 Click the **Build Selected** button.

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

The geometry now consists of two rectangular domains for the air and the dielectric, and a circle centered on their intersection. You can remove the line through the circle if you first create a union of the objects.

Union I

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

2 From the **Edit** menu, choose **Select All**.

3 Click the **Build Selected** button.

Delete Entities /

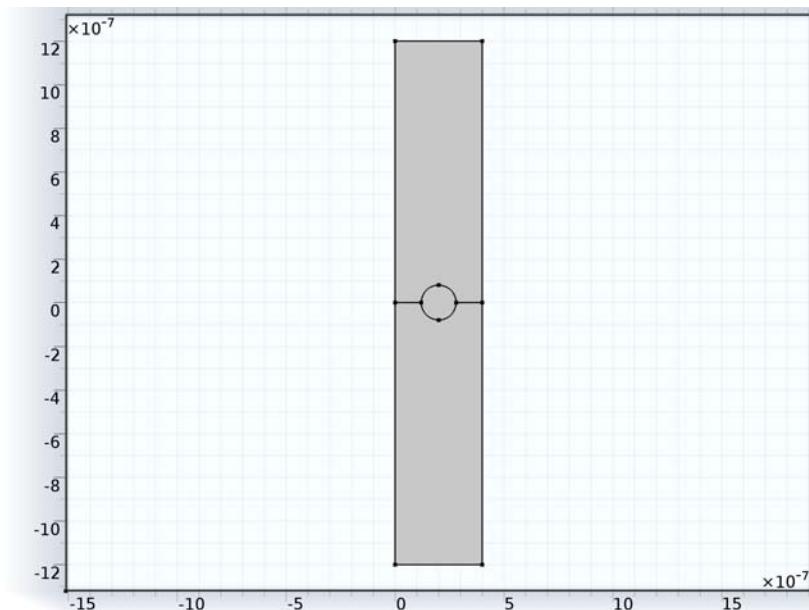
1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **unil**, select Boundary 6 only. (This is the horizontal diameter of the circle in the center of the geometry).

3 Click the **Build Selected** button.

Form Union

- 1** In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Form Union** and choose **Build Selected**.

**ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN**

Before setting up the materials, define which constitutive relations you want to use in the Electromagnetic Waves interface.

Wave Equation, Electric 1

- 1** In the **Model Builder** window, expand the **Electromagnetic Waves, Frequency Domain** node, then click **Wave Equation, Electric 1**.
- 2** In the **Wave Equation, Electric** settings window, locate the **Electric Displacement Field** section.
- 3** From the **Electric displacement field model** list, choose **Refractive index**.

Wave Equation, Electric 2

- 1** In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Wave Equation, Electric**.
- 2** Select Domain 3 only.

- 3 In the **Wave Equation, Electric** settings window, locate the **Electric Displacement Field** section.
- 4 From the **Electric displacement field model** list, choose **Dielectric loss**.

MATERIALS

Material 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Name	Value
Refractive index	n	na

- 4 Right-click **Model 1>Materials>Material 1** and choose **Rename**.
- 5 Go to the **Rename Material** dialog box and type **Air** in the **New name** edit field.
- 6 Click **OK**.

Material 2

- 1 Right-click **Materials** and choose **Material**.
- 2 Select Domain 1 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

Property	Name	Value
Refractive index	n	nb

- 5 Right-click **Model 1>Materials>Material 2** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type **Dielectric** in the **New name** edit field.
- 7 Click **OK**.

Material 3

- 1 Right-click **Materials** and choose **Material**.
- 2 Select Domain 3 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.

- 4** In the table, enter the following settings:

Property	Name	Value
Relative permittivity (imaginary part)	epsilonBis	5.4
Relative permittivity (real part)	epsilonPrim	-1.75
Relative permeability	mur	1
Electric conductivity	sigma	0

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

- 6** Go to the **Rename Material** dialog box and type **Gold** in the **New name** edit field.

- 7** Click **OK**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

In the first version of this model, you will assume a TE-polarized wave. This means that E_x and E_y will be zero throughout the geometry, and that you consequently only need to solve for E_z .

- 1** In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Components** section.

- 2** From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Now define the excitation port. A periodic port assumes that the structure is periodic and simplifies the setup of ports for the diffraction orders.

Port 1

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

- 2** Select Boundary 5 only.

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

- 4** From the **Type of port** list, choose **Periodic**.

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

Notice the you define the electric field by only setting the amplitude. A phase factor should not be entered.

- 6** Locate the **Port Mode Settings** section. In the **E₀** table, enter the following settings:

0	x
0	y
1	z

- 7 In the α edit field, type `alpha`.
- 8 In the n edit field, type `na`.
- 9 In the f_{\max} edit field, type `f0`.

The order in which you set up the ports will determine how the S-parameters are labeled. You have just created Port 1 for the excitation. If you set up the next port for the transmission of the purely refracted beam, the S_{21} -parameter will contain information on the zero order transmission.

Port 2

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- 2 Select Boundary 2 only.
- 3 In the **Port** settings window, locate the **Port Properties** section.
- 4 From the **Type of port** list, choose **Periodic**.
- 5 Locate the **Port Mode Settings** section. In the \mathbf{E}_0 table, enter the following settings:

0	x
0	y
1	z

The angle of incidence on the exit side corresponds to the angle of incidence an incident wave on that side would have to provide the correct propagation angle in the material. Notice that this also means that the sign is opposite that on the entry side.

- 6 In the α edit field, type `-beta`.
- 7 In the n edit field, type `nb`.
- 8 In the f_{\max} edit field, type `f0`.

Port 1

Continue with the ports for the reflected diffraction orders. Since these diffraction orders are not propagating at normal incidence, you have to add the ports manually.

Diffraction Order 1

- 1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** right-click **Port 1** and choose **Diffraction Order**.
- 2 In the **Diffraction Order** settings window, locate the **Port Mode Settings** section.
- 3 From the **Components** list, choose **Out-of-plane vector**.

- 4 In the **m** edit field, type -1.

Diffraction Order 2

- 1 In the **Model Builder** window, right-click **Diffraction Order 1** and choose **Duplicate**.
- 2 In the **Diffraction Order** settings window, locate the **Port Properties** section.
- 3 In the **Port name** edit field, type 4.
- 4 Locate the **Port Mode Settings** section. In the **m** edit field, type 1.

The transmitted diffraction orders are propagating at normal incidence. Thus, you can create them automatically by clicking the **Compute Diffraction Orders** button.

Port 2

- 1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 2**.
- 2 In the **Port** settings window, locate the **Port Mode Settings** section.
- 3 Click the **Compute Diffraction Orders** button.

You now find the **Diffraction Order** ports as subfeatures to **Port 2**.

- 4 In the **Model Builder** window, expand the **Port 2** node.

Periodic Condition 1

- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Periodic Condition**.
- 2 Select Boundaries 1, 3, 7, and 8 only.
- 3 In the **Periodic Condition** settings window, locate the **Periodicity Settings** section.
- 4 From the **Type of periodicity** list, choose **Floquet periodicity**.

The wave vector in the direction for the periodicity is used by the periodic port. Thus, you can use that wave vector also for the Floquet periodic condition.

- 5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

MESH 1

The periodic boundary conditions perform better if the mesh is identical on the periodicity boundaries. This is especially important when dealing with vector degrees of freedom, as will be the case in the TM version of this model.

- 1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.
- 2 In the **Mesh** settings window, locate the **Mesh Settings** section.
- 3 From the **Sequence type** list, choose **User-controlled mesh**.

Free Triangular 1

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Delete**.
- 2 Click **Yes** to confirm.

Size

- 1 In the **Size** settings window, locate the **Element Size** section.
- 2 From the **Predefined** list, choose **Extra fine**.

Edge 1

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Edge**.
- 2 Select Boundaries 1 and 3 only.

Copy Edge 1

- 1 Right-click **Mesh 1** and choose **More Operations>Copy Edge**.
- 2 Select Boundary 3 only.
- 3 In the **Copy Edge** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundary 8 only.

Copy Edge 2

- 1 Right-click **Mesh 1** and choose **More Operations>Copy Edge**.
- 2 Select Boundary 1 only.
- 3 In the **Copy Edge** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundary 7 only.

Free Triangular 1

- 1 Right-click **Mesh 1** and choose **Free Triangular**.
- 2 In the **Settings** window, click **Build All**.

To set up the study to sweep for the angle of incidence, some modifications of the solver is required.

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 **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type f_0 .

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names
alpha

- 5 Click **Range**.
- 6 Go to the **Range** dialog box.
- 7 In the **Start** edit field, type 0.
- 8 In the **Stop** edit field, type $\pi/2 - \pi/40$.
- 9 In the **Step** edit field, type $\pi/40$.
- 10 Click the **Replace** button.
- II Right-click **Study 1** and choose **Compute**.

RESULTS*Electric Field (emw)*

The default plot shows the electric field norm for the last solution, almost tangential incidence. Look at a more interesting angle of incidence.

- 1 In the **2D Plot Group** settings window, locate the **Data** section.
- 2 From the **Parameter value (alpha)** list, choose **0.628319**.
- 3 Click the **Plot** button.
- 4 Click the **Zoom Extents** button on the Graphics toolbar.

The plot should now look like [Figure 4](#).

Rename the plot group to make it clear that it shows the TE solution.

- 5 Right-click **Results>Electric Field (emw)** and choose **Rename**.
- 6 Go to the **Rename 2D Plot Group** dialog box and type **2D Plot Group TE** in the **New name** edit field.
- 7 Click **OK**.

Add a 1D plot to look at the various orders of reflection and transmission versus the angle of incidence.

ID Plot Group 2

- I Right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, locate the **Title** section.
- 3 From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type **Reflection and Transmission of TE Wave**.
- 5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 6 In the associated edit field, type **Angle of incidence (degrees)**.
- 7 Select the **y-axis label** check box.
- 8 In the associated edit field, type **Reflection and transmission coefficients**.
- 9 Right-click **Results>ID Plot Group 2** and choose **Global**.

Since the diffraction orders are not propagating for all angles, use a logic expression to set the S-parameter to zero when the wave is evanescent and to multiply with one when it is propagating.

- 10 In the **Global** settings window, locate the **y-Axis Data** section.
- II In the table, enter the following settings:

Expression
<code>abs(emw.S11)^2</code>
<code>abs(emw.S21)^2</code>
<code>abs(emw.S31)^2*(imag(emw.beta_3)==0)</code>
<code>abs(emw.S41)^2*(imag(emw.beta_4)==0)</code>
<code>abs(emw.S51)^2*(imag(emw.beta_5)==0)</code>
<code>abs(emw.S61)^2*(imag(emw.beta_6)==0)</code>

- 12 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 13 In the **Expression** edit field, type `alpha*180/pi`.
- 14 Click to expand the **Legends** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 15 Locate the **Legends** section. From the **Legends** list, choose **Manual**.
- 16 In the table, enter the following settings:

Legends
<code>R<sub>0</sub></code>
<code>T<sub>0</sub></code>
<code>R<sub>-1</sub></code>

Legends

R₁

T₋₁

T₁

I7 Click the **Plot** button.

I8 In the **Model Builder** window, right-click **ID Plot Group 2** and choose **Rename**.

I9 Go to the **Rename ID Plot Group** dialog box and type **1D Plot Group TE** in the **New name** edit field.

I0 Click **OK**.

The plot should now look like [Figure 6](#).

The remaining instructions show to alter the physics so that you solve for an incident TM wave.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

I In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Components** section.

2 From the **Electric field components solved for** list, choose **In-plane vector**.

You will now solve for E_x and E_y instead of E_z ; for a TM wave, E_z is zero.

Port 1

The easiest way to specify a TM wave is to define the magnetic field, since only the z component is used.

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

2 In the **Port** settings window, locate the **Port Mode Settings** section.

3 From the **Input quantity** list, choose **Magnetic field**.

4 In the **H₀** table, enter the following settings:

0	x
0	y
1	z

Diffraction Order 1

I In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain>Port 1** click **Diffraction Order 1**.

2 In the **Diffraction Order** settings window, locate the **Port Mode Settings** section.

3 From the **Components** list, choose **In-plane vector**.

Diffraction Order 2

1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain>Port 1** click **Diffraction Order 2**.

2 In the **Diffraction Order** settings window, locate the **Port Mode Settings** section.

3 From the **Components** list, choose **In-plane vector**.

Port 2

1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 2**.

2 In the **Port** settings window, locate the **Port Mode Settings** section.

3 From the **Input quantity** list, choose **Magnetic field**.

4 In the **H₀** table, enter the following settings:

0	x
0	y
1	z

5 Click the **Compute Diffraction Orders** button to change components for the diffraction orders that are propagating at normal incidence.

R O O T

Add a new study in order not to overwrite the TE solution.

1 In the **Model Builder** window, right-click the root node and choose **Add Study**.

M O D E L W I Z A R D

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

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

3 Click **Finish**.

S T U D Y 2

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Frequency Domain**.

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

- 3 In the **Frequencies** edit field, type f_0 .

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 2** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names
alpha

- 5 Click **Range**.
- 6 Go to the **Range** dialog box.
- 7 In the **Start** edit field, type 0.
- 8 In the **Stop** edit field, type $\pi/2 - \pi/40$.
- 9 In the **Step** edit field, type $\pi/40$.
- 10 Click the **Replace** button.
- II Right-click **Study 2** and choose **Compute**.

RESULTS

Electric Field (emw)

- 1 In the **2D Plot Group** settings window, locate the **Data** section.
- 2 From the **Parameter value (alpha)** list, choose **0.628319**.
- 3 Click the **Plot** button.
- 4 Click the **Zoom Extents** button on the Graphics toolbar.
- 5 Right-click **Results>Electric Field (emw)** and choose **Rename**.
- 6 Go to the **Rename 2D Plot Group** dialog box and type **2D Plot Group TM** in the **New name** edit field.
- 7 Click **OK**.

You have now reproduced [Figure 5](#).

For the transmission and the reflection of the TM waves, copy and reuse the 1D plot for the TE waves.

1D Plot Group TE I

- 1 In the **Model Builder** window, under **Results** right-click **ID Plot Group TE** and choose **Duplicate**.
- 2 In the **ID Plot Group** settings window, locate the **Title** section.
- 3 In the **Title** text area, type **Reflection and Transmission of TM Wave**.
- 4 Locate the **Data** section. From the **Data set** list, choose **Solution 2**.
- 5 Click the **Plot** button.
- 6 Right-click **Results>ID Plot Group TE I** and choose **Rename**.
- 7 Go to the **Rename ID Plot Group** dialog box and type **1D Plot Group TM** in the **New name** edit field.
- 8 Click **OK**.

Compare the resulting plot with that in [Figure 7](#).

Quarter-Wave Transformer

Introduction

Transmission lines are used when the frequency of the electromagnetic signals is so high that the wave nature of the signals must be taken into account. A consequence of the wave nature is that the signals are reflected if there are abrupt changes of the characteristic impedance along the transmission line. Similarly, the load impedance, Z_L , at the end of the transmission line must match its characteristic impedance, Z_0 . Otherwise there will be reflections from the transmission line's end.

A quarter-wave transformer (see [Figure 1](#)) is a component that can be inserted between the transmission line and the load to match the load impedance to the transmission line's characteristic impedance. To get this functionality, the transformer must be a quarter of a wavelength long and the relation between the impedances involved must be

$$\frac{Z_{in}}{Z} = \frac{Z}{Z_L} \quad (1)$$

If the length and the impedance requirements are fulfilled, the load impedance will not give rise to any reflections.

Typically, the characteristic impedance of transmission lines, Z_0 , is 50Ω . Thus, Z_{in} in [Equation 1](#) should be set to

$$Z_{in} = Z_0 = 50 \Omega \quad (2)$$

when solving for the characteristic impedance of the quarter-wave transformer, Z .

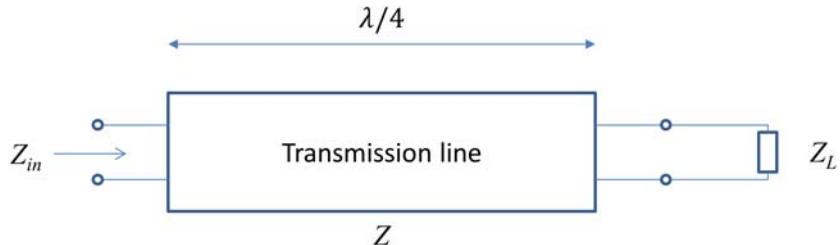


Figure 1: Schematic of a quarter-wave transformer. The input impedance is Z_{in} , the impedance of the transformer transmission line is Z , and the load impedance is Z_L .

This model exemplifies some of the characteristics of a quarter-wave transformer. In particular, the model simulation shows that the transformer only provides matching for one particular frequency, namely that for which the transformer is a quarter of a wavelength long.

Model Definition

The 1D geometry of the model consists of two line intervals. Each line interval represents a separate transmission line, with different electrical parameters (distributed capacitance and inductance) and lengths.

To excite and terminate the transmission lines, use lumped ports. This also makes it easy to obtain the reflection (S_{11}) and transmission (S_{21}) coefficients for the system.

Results and Discussion

As an example of the output from the model, [Figure 2](#) shows the voltage amplitude distribution along the transmission lines for a frequency where the quarter-wave transformer matches the load impedance to the characteristic impedance of the incoming transmission line. The figure shows that the amplitude is constant, indicating that there is no reflection and therefore no standing waves. [Figure 3](#) shows the frequency spectrum for the same transformer. As is evident from the graph, the

quarter-wave transformer only operates without reflection in a certain wavelength range.

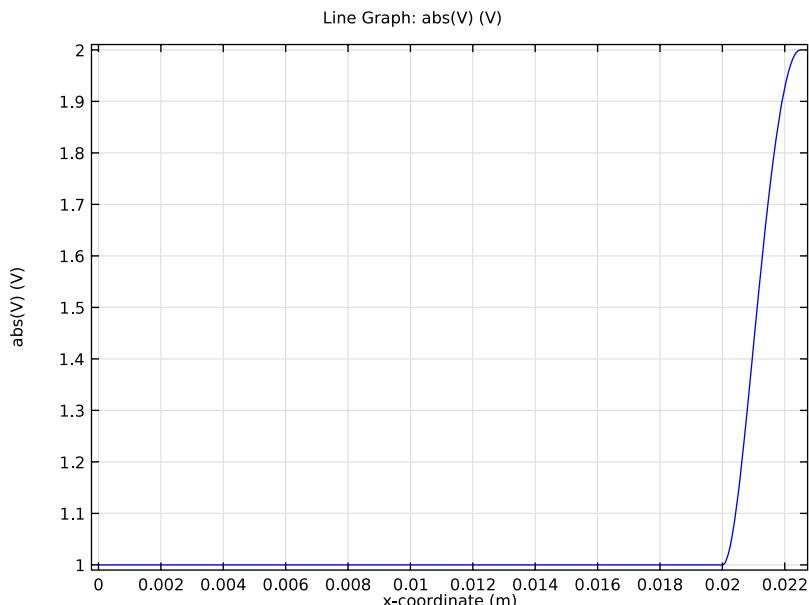


Figure 2: Absolute value of the voltage versus the x-coordinate. The quarter-wave transformer starts at x-coordinate 0.02 m.

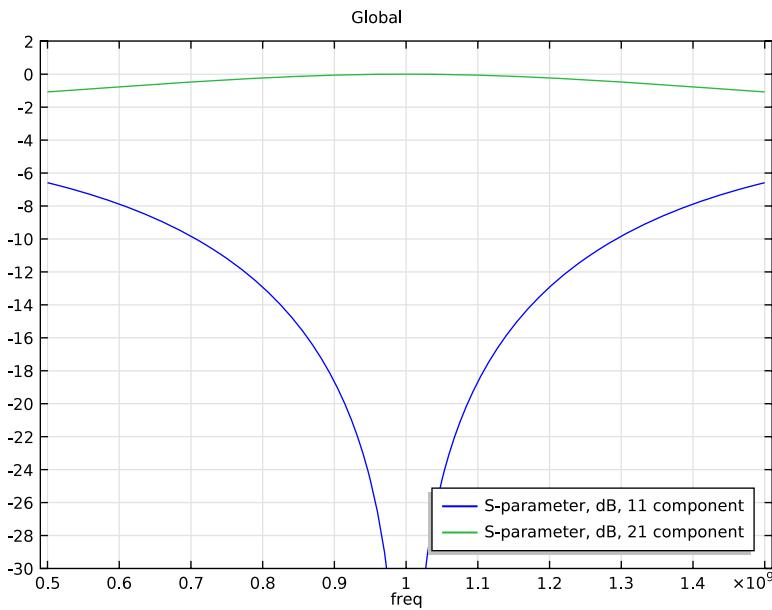


Figure 3: Spectral response for the transmission line. Notice that the transmission coefficient (S_{21}) peaks at the frequency (1 GHz) for which the transformer is a quarter-wave long. At that frequency the reflection coefficient (S_{11}) is zero (approaches negative infinity which the dB scale used in the graph).

Model Library path: RF_Module/Transmission_Lines_and_Waveguides/
quarter_wave_transformer

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **ID** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Radio Frequency>Transmission Line (tl)**.
- 5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

7 Click **Finish**.

GLOBAL DEFINITIONS

First add some parameters that defines the electrical and geometrical properties of the transmission lines.

Parameters

- 1** In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2** In the **Parameters** settings window, locate the **Parameters** section.
- 3** In the table, enter the following settings:

Name	Expression	Description
L1	$2.5e-6[H/m]$	Distributed inductance, first transmission line
C1	$1e-9[F/m]$	Distributed capacitance, first transmission line
f	$1[GHz]$	Frequency
wl1	$1/(f*sqrt(L1*C1))$	Wavelength, first transmission line
d1	wl1	Length, first transmission line
Z1	$sqrt(L1/C1)$	Characteristic impedance, first transmission line
ZL	$4*Z1$	Terminating impedance
Z2	$sqrt(Z1*ZL)$	Characteristic impedance, second transmission line
C2	C1	Distributed capacitance, second transmission line
L2	$C2*Z2^2$	Distributed inductance, second transmission line
wl2	$1/(f*sqrt(L2*C2))$	Wavelength, second transmission line
d2	wl2/4	Length, second transmission line
hmax	$d2/10$	Maximum discretization step

GEOMETRY I

Set up the geometry as two intervals.

Interval I

- 1** In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Interval**.

- 2** In the **Interval** settings window, locate the **Interval** section.
- 3** From the **Number of intervals** list, choose **Many**.
- 4** In the **Points** edit field, type $0, d1, d1+d2$.
- 5** Click the **Build All** button.

TRANSMISSION LINE

Assign the first transmission line the distributed capacitance and inductance C1 and L1, respectively.

Transmission Line Equation 1

- 1** In the **Model Builder** window, under **Model 1>Transmission Line** click **Transmission Line Equation 1**.
- 2** In the **Transmission Line Equation** settings window, locate the **Transmission Line Equation** section.
- 3** In the **L** edit field, type **L1**.
- 4** In the **C** edit field, type **C1**.

Now define the second transmission line by adding a transmission line equation feature to the second interval.

Transmission Line Equation 2

- 1** In the **Model Builder** window, right-click **Transmission Line** and choose **Transmission Line Equation**.
 - 2** Click the **Zoom Extents** button on the Graphics toolbar to make the size of the transmission line suitable for selecting the second interval.
 - 3** Select Domain 2 only.
- Add the electrical parameters for the second transmission line.
- 4** In the **Transmission Line Equation** settings window, locate the **Transmission Line Equation** section.
 - 5** In the **L** edit field, type **L2**.
 - 6** In the **C** edit field, type **C2**.

Replace the default absorbing boundary condition with lumped ports. With the lumped ports, it is easy to excite the transmission line and also to plot the S-parameters, that is, the reflection and transmission coefficient, for the transmission line.

Lumped Port 1

- 1 Right-click **Transmission Line** and choose **Lumped Port**.
- 2 Select Boundary 1 only. Select that this port shall be excited. You can use the default voltage for the port.
- 3 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4 From the **Wave excitation at this port** list, choose **On**.

Lumped Port 2

- 1 Right-click **Transmission Line** and choose **Lumped Port**.
- 2 Select Boundary 3 only. This lumped port should have a different characteristic impedance than the first lumped port and the two transmission lines.
- 3 In the **Lumped Port** settings window, locate the **Settings** section.
- 4 In the Z_{ref} edit field, type ZL .

MESH I

Let the mesh have a maximum subinterval that is one tenth of the quarter-wave part of the transmission line.

- 1 In the **Model Builder** window, under **Model I** click **Mesh I**.
- 2 In the **Mesh** settings window, locate the **Mesh Settings** section.
- 3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

- 1 In the **Model Builder** window, under **Model I>Mesh I** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type $hmax$.

STUDY I

Step 1: Frequency Domain

Set the frequency for the frequency-domain study and create a first default plot.

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type f .
- 4 In the **Model Builder** window, right-click **Study I** and choose **Compute**.

RESULTS

To clearly demonstrate that the quarter-wave transformer works, replace the plot expression with the absolute value of the voltage.

Electric Potential (tl)

- 1 In the **Model Builder** window, expand the **Electric Potential (tl)** node, then click **Line graph**.
- 2 In the **Line Graph** settings window, locate the **y-Axis Data** section.
- 3 In the **Expression** edit field, type `abs(V)`.
- 4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Geometry>Coordinate>x-coordinate (x)**.
- 5 Click the **Plot** button.

You should now have a graph as in [Figure 2](#). Notice that the left part of the curve is flat, with a unit amplitude, indicating that there are no standing waves, despite the fact that the second lumped port has a load impedance that normally would not be matched with the transmission line.

STUDY I

Step 1: Frequency Domain

Now modify the study settings to create a frequency sweep around 1 GHz, but first define the frequency sweep parameters.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
df	500[MHz]	Half of frequency sweep
fstep	10[MHz]	Frequency step

STUDY I

Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.

- 3** Click the **Range** button.
- 4** Go to the **Range** dialog box.
- 5** In the **Start** edit field, type `f-df`.
- 6** In the **Step** edit field, type `fstep`.
- 7** In the **Stop** edit field, type `f+df`.
- 8** Click the **Replace** button.
- 9** In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Create a new plot group for a global plot of the S_{11} (reflection) and S_{21} (transmission) coefficients.

ID Plot Group 2

- 1** In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2** Right-click **ID Plot Group 2** and choose **Global**.
- 3** In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Transmission Line>Ports>S-parameter, dB>S-parameter, dB, II component (tl.S11dB)**.
- 4** Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Transmission Line>Ports>S-parameter, dB>S-parameter, dB, 2I component (tl.S21dB)**.
- 5** Click the **Plot** button.

Modify the *y*-axis limits to show that S_{21} actually has its maximum value for the frequency where S_{11} is at its' minimum value.

- 6** In the **Model Builder** window, click **ID Plot Group 2**.
- 7** In the **ID Plot Group** settings window, click to expand the **Axis** section.
- 8** Select the **Manual axis limits** check box.
- 9** In the **y minimum** edit field, type `-30`.
- 10** In the **y maximum** edit field, type `2`.

Move the legend panel, so it doesn't cover the curves.

- II** Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.
- 12** Click the **Plot** button.

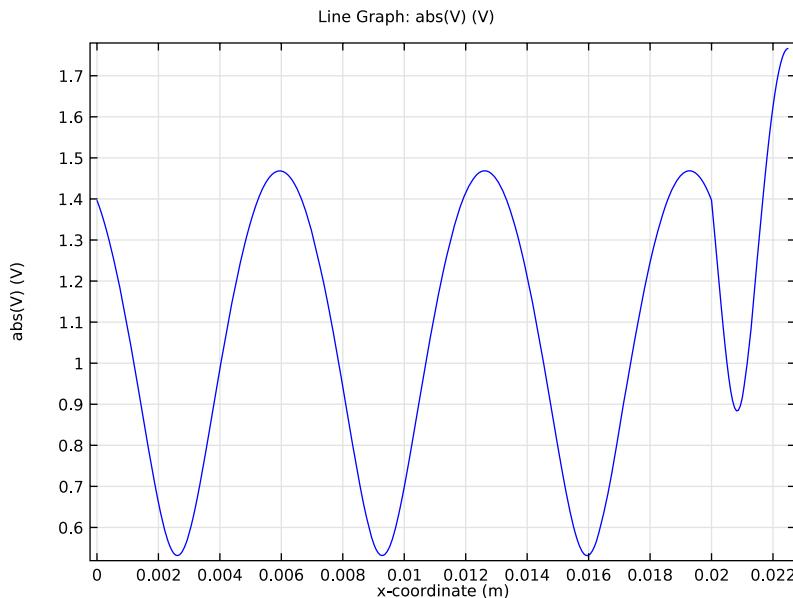
You should now have a plot of the spectrum for S_{11} and S_{21} , similar to the one in [Figure 3](#).

Electric Potential (tl)

To demonstrate that the quarter-wave transformer only eliminates the reflection at one frequency, plot the last frequency in the first plot group.

- 1 In the **Model Builder** window, under **Results** click **Electric Potential (tl)**.
- 2 In the **ID Plot Group** settings window, locate the **Data** section.
- 3 From the **Parameter selection (freq)** list, choose **Last**.
- 4 Click the **Plot** button.

Notice that the curve in the left part of the plot is not flat. The sinusoidal oscillation in the absolute value of the voltage is a signature of the standing wave that appears when there is a reflection point along the transmission line. For the selected frequency the quarter-wave transformer is not a quarter-wave long and, thus, there are now reflections.



To demonstrate that the quarter-wave transformer not only should have a matched length, but also a matched characteristic impedance, set the characteristic impedance of the second lumped port to 50 ohms.

TRANSMISSION LINE

Lumped Port 2

- 1 In the **Model Builder** window, under **Model 1>Transmission Line** click **Lumped Port 2**.
- 2 In the **Lumped Port** settings window, locate the **Settings** section.
- 3 In the Z_{ref} edit field, type 50.

Compute the spectral plot again.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

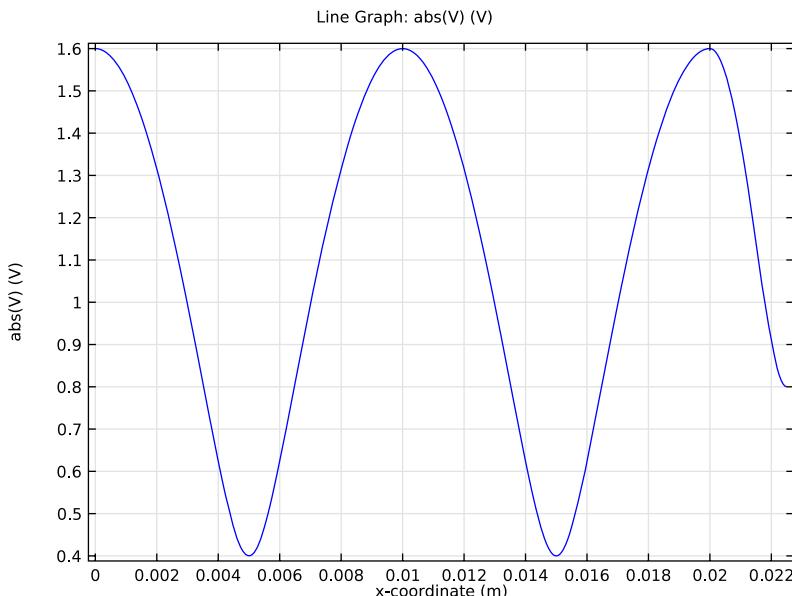
Electric Potential (tl)

Select the central frequency (1 GHz) in plot group 1.

- 1 In the **ID Plot Group** settings window, locate the **Data** section.
- 2 From the **Parameter selection (freq)** list, choose **From list**.
- 3 In the **Parameter values (freq)** list, select **10e8**.

4 Click the **Plot** button.

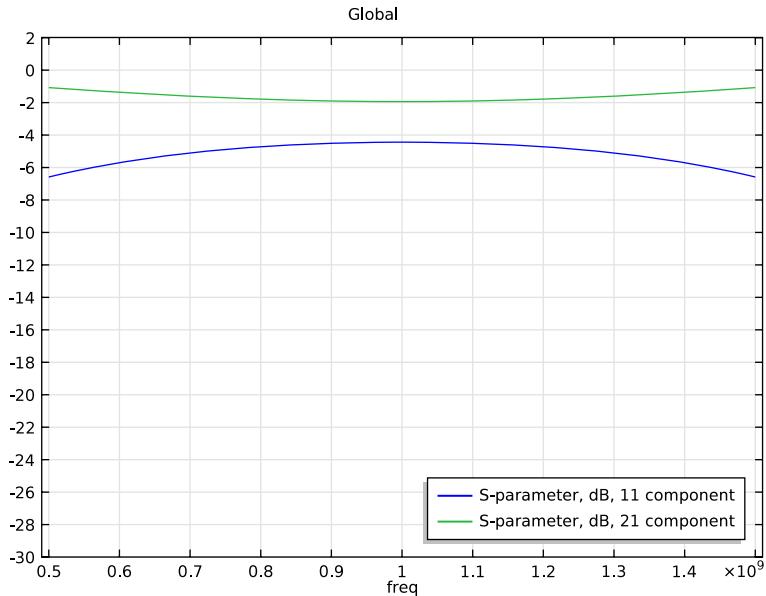
Notice that there is now a standing wave also at the center frequency.



Select the second plot group to see the spectral response.

ID Plot Group 2

Notice that there is still a resonance at the center frequency. However, as was already indicated by the spatial plot, there is considerable reflection also at the resonance frequency.



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.

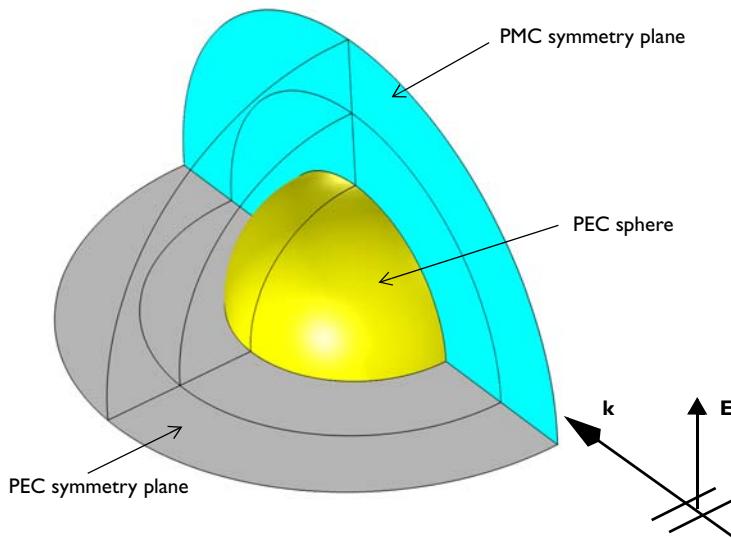


Figure 1: The computational domain for computing the RCS of a PEC sphere in free space. Due to symmetry, it is sufficient to model one quarter of the sphere.

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

$$\nabla \times (\mu_r^{-1} \nabla \times (\mathbf{E}_i + \mathbf{E}_{sc})) - k_0^2 \epsilon_{rc} (\mathbf{E}_i + \mathbf{E}_{sc}) = \mathbf{0}$$

where the scattered electric field \mathbf{E}_{sc} is the dependent variable and the incident electric field $\mathbf{E}_i = (0, 0, E_z)$, with

$$E_z = 1[V/m]e^{-jk_0x}$$

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 (outwards) 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 model is shown in [Figure 2](#). It consists of a free tetrahedral mesh around the sphere and a swept mesh in the PML domain.

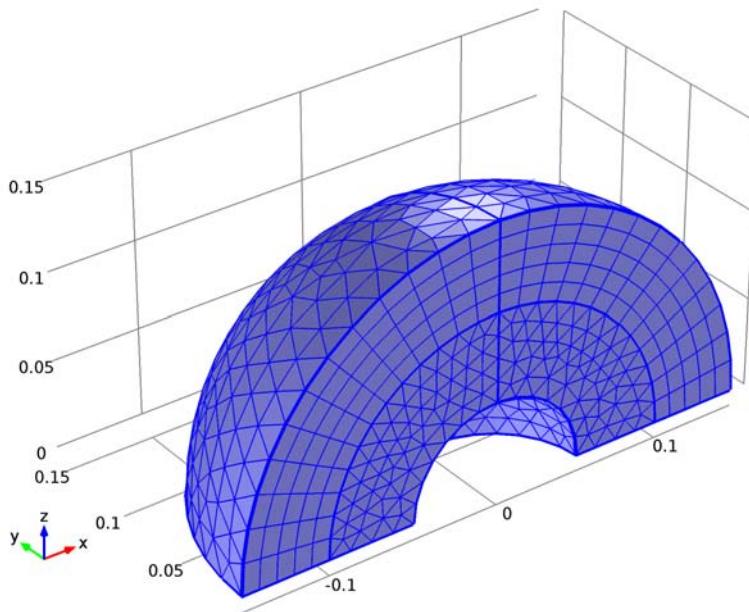


Figure 2: A free tetrahedral mesh is used in the free-space region around the sphere, and a swept mesh is used in the PML region.

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:

$$\eta = \sqrt{\mu/\epsilon}$$

- 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 model 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.

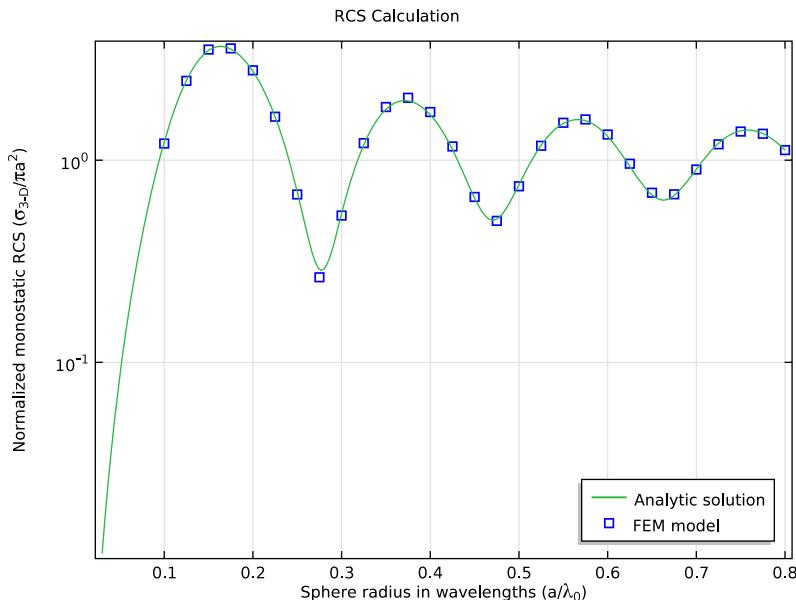


Figure 3: Comparison of the analytic solution and the COMSOL Multiphysics model of the RCS of a PEC sphere in free space.

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 towards 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 outwards 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 will not decrease when refining the mesh.

In [Figure 4](#), the mesh convergence is shown. 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 vs. wavelength curve.

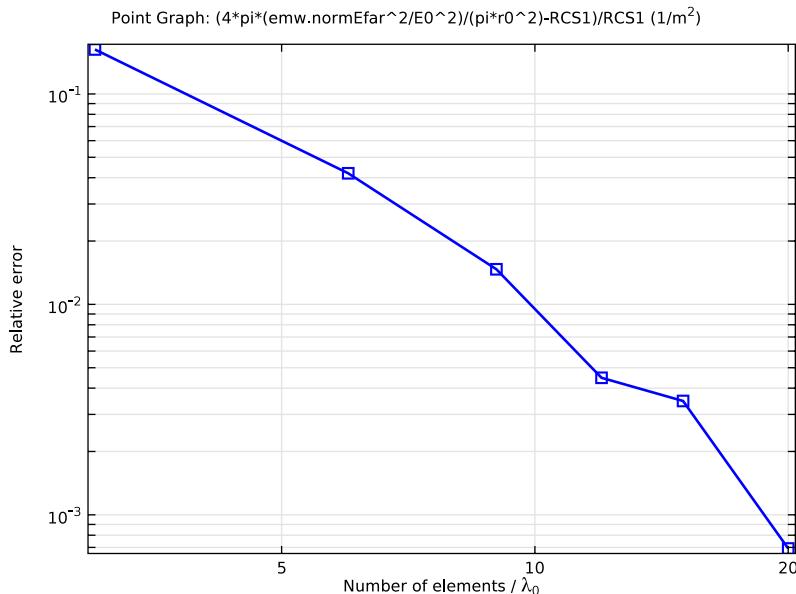


Figure 4: Mesh convergence for the difference in backscattering (monostatic) RCS between the COMSOL model and the exact solution.

Reference

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

Model Library path: RF_Module/Verification_Models/rccs_sphere

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
r_lda	0.5	Sphere radius in wavelengths
r0	5[cm]	Sphere radius
lda	r0/r_lda	Wavelength
k0	2*pi/lda	Wavenumber
f0	c_const/lda	Frequency
t_air	lda/2	Thickness of air around sphere
t_pml	lda/2	Thickness of PML
h_size	8	Number of elements per wavelength
E0	1[V/m]	Incident field magnitude

GEOMETRY I

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

- 1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Sphere**.
- 2 In the **Sphere** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type $r_0+t_{\text{air}}+t_{\text{pml}}$.
- 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 the **Build All** button.

DEFINITIONS

Add a view with a different angle of perspective.

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

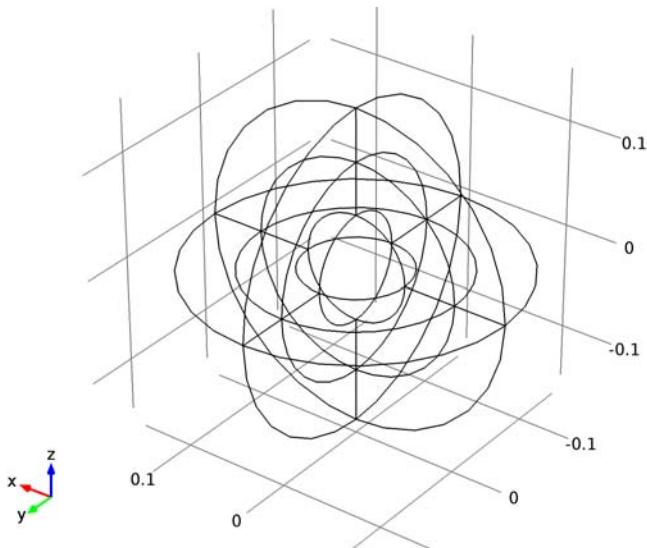
Camera

Change only the sign of y in the Position and Up Vector sections:

- 1 In the **Model Builder** window, expand the **View 2** node, then click **Camera**.
- 2 In the **Camera** settings window, locate the **Position** section.
- 3 In the **y** edit field, type 1.871 .
- 4 Locate the **Up Vector** section. In the **y** edit field, type -0.412 .
- 5 Click the **Apply** button.

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

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



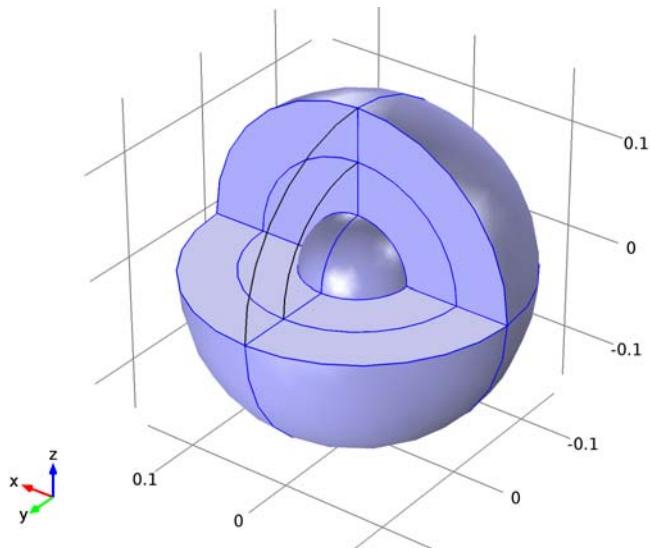
GEOMETRY I

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 I

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Delete Entities**.
- 2 In the **Delete Entities** settings window, 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.

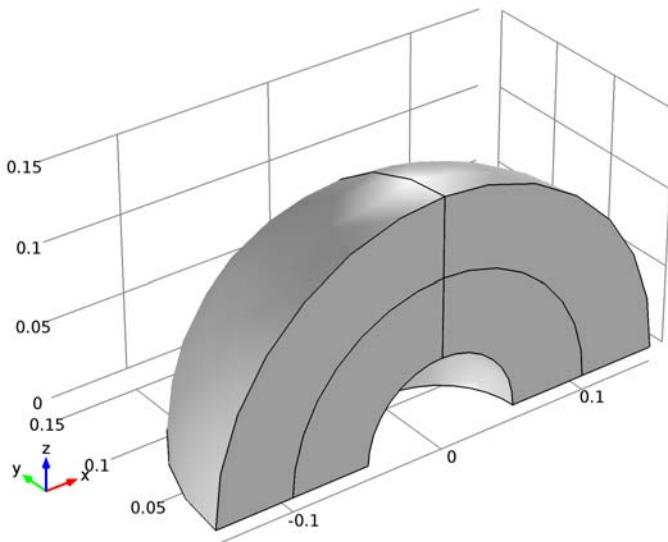


- 5 Click the **Build All** button.

After removing unnecessary domains, change the view to the first view definition which gives a better angle showing all layers.

- 6 Click the **Go to View 1** button on the Graphics toolbar.

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



This is the modeling domain for RCS analysis.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

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 **Model 1** click **Electromagnetic Waves, Frequency Domain**.
- 2 In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Settings** section.
- 3 From the **Solve for** list, choose **Scattered field**.

- 4** In the \mathbf{E}_b table, enter the following settings:

0	x
0	y
$E_0 \cdot \exp(-j \cdot k_0 \cdot x)$	z

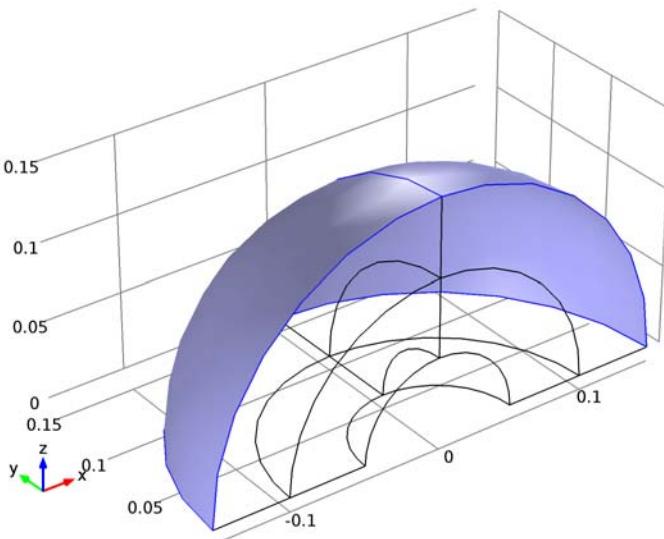
Choose wireframe rendering in the current view to get a better view of the interior parts.

- 5** Click the **Wireframe Rendering** button on the Graphics toolbar.

Scattering Boundary Condition 1

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

- 2** Select Boundaries 3 and 14 only.



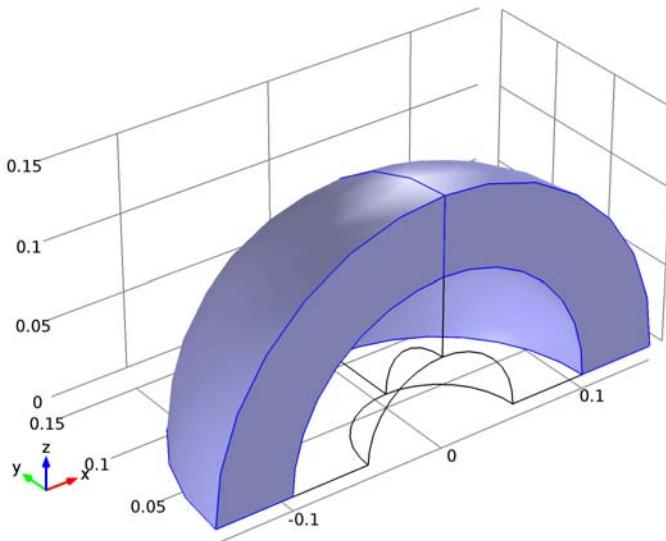
DEFINITIONS

The outermost domains from the center of the sphere are the PMLs.

Perfectly Matched Layer 1

- I** In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Perfectly Matched Layer**.

- 2 Select Domains 1 and 4 only.



- 3 In the **Perfectly Matched Layer** settings window, locate the **Geometry** section.

- 4 From the **Type** list, choose **Spherical**.

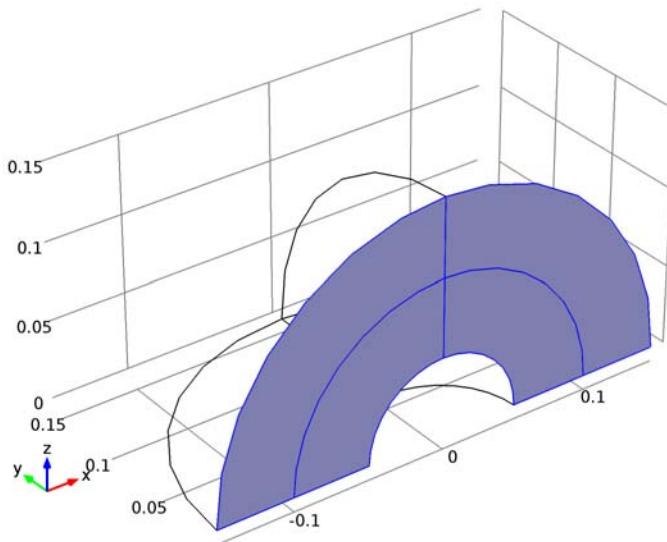
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Set PMC on the boundaries parallel to the background E-field polarization.

Perfect Magnetic Conductor I

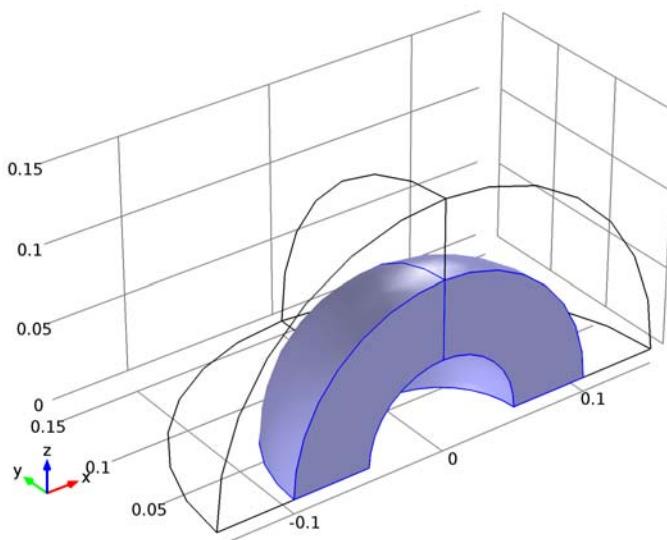
- I In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 1, 4, 9, and 12 only.



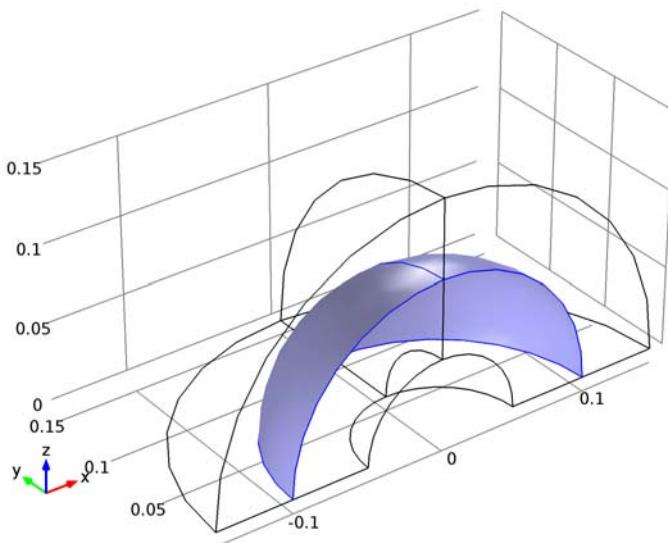
Far-Field Domain 1

- I In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Far-Field Domain**.
- 2** Select Domains 2 and 3 only.



Far-Field Calculation 1

- 1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain>Far-Field Domain 1** click **Far-Field Calculation 1**.
- 2 In the **Far-Field Calculation** settings window, locate the **Boundary Selection** section.
- 3 Click **Clear Selection**.
- 4 Select Boundaries 6 and 13 only.



- 5 Locate the **Far-Field Calculation** section. Select the **Symmetry in the y=0 plane** check box.
- 6 Select the **Symmetry in the z=0 plane** check box.
- 7 From the **Symmetry type** list, choose **Symmetry in H (PEC)**.

MATERIALS

Next, assign material properties. Use air for all domains.

Material Browser

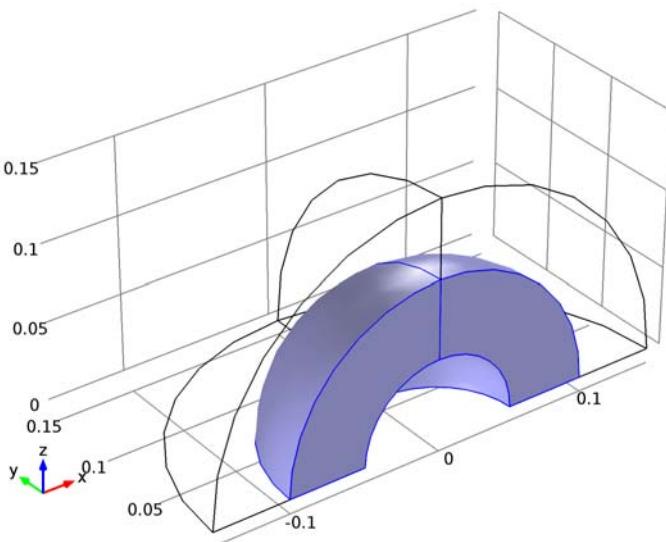
- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
- 2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.
- 3 Click **Add Material to Model**.

MESH I

Use a tetrahedral mesh for the air domains.

Free Tetrahedral I

- 1 In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **Free Tetrahedral**.
- 2 In the **Free Tetrahedral** settings window, 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 **Model I>Mesh I** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `lda/h_size`.
- 5 In the **Minimum element size** edit field, type `lda/h_size`.

Use a swept mesh for the PML domains.

Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.
- 2 In the **Settings** window, click **Build All**.

Compare the mesh with that shown in [Figure 2](#).

STUDY 1*Parametric Sweep*

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names	Parameter value list
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 **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type **f0**.
- 4 In the **Model Builder** window, click **Study 1**.
- 5 In the **Study** settings window, locate the **Study Settings** section.
- 6 Clear the **Generate default plots** check box.
- 7 Click the **Compute** button.

RESULTS

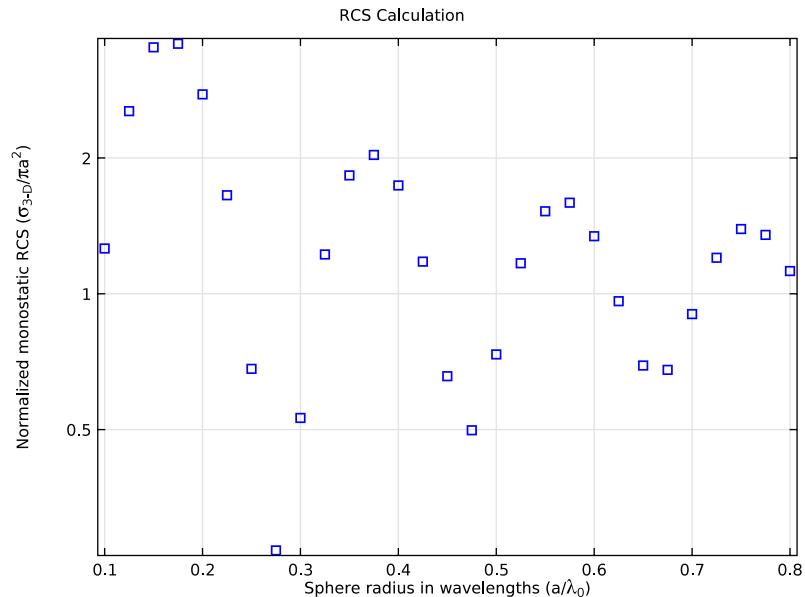
Follow the instructions below to reproduce the plot in [Figure 3](#). First, show the computed RCS values using square markers.

ID Plot Group 1

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Solution 2**.

- 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** edit field, type **RCS Calculation**.
- 9 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 10 In the associated edit field, type **Sphere radius in wavelengths (a/lambda₀)**.
- 11 Select the **y-axis label** check box.
- 12 In the associated edit field, type **Normalized monostatic RCS (sigma_{3-D}/pi a²)**.
- 13 Locate the **Axis** section. Select the **y-axis log scale** check box.
- 14 Right-click **Results>ID Plot Group I** and choose **Point Graph**.
- 15 Select Point 2 only.
- 16 In the **Point Graph** settings window, locate the **y-Axis Data** section.
- 17 In the **Expression** edit field, type **4*pi*(emw.normEfar^2/E0^2)/(pi*r0^2)**.
- 18 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.
- 19 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 20 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.
- 21 From the **Positioning** list, choose **In data points**.

22 Click the **Plot** button.



The observed RCS graph pattern is oscillatory in the Mie region.

GLOBAL DEFINITIONS

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.

Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
r_lda	r1	Sphere radius in wavelengths
r0	5[cm]	Sphere radius
lda	r0/r_lda	Wavelength
k0	2*pi/lda	Wavenumber
f0	c_const/lda	Frequency
t_air	lda/2	Thickness of air around sphere

Name	Expression	Description
t_pml	$1da/2$	Thickness of PML
h_size	8	Number of elements per wavelength
E0	$1[V/m]$	Incident field magnitude
r1	0.16363636363636364	Relative radius at 1st resonance
RCS1	3.6549540474068576	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.

- I In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Duplicate**.

MESH 2

In the **Model Builder** window, expand the **Model 1>Meshes** node.

Size

- I In the **Model Builder** window, expand the **Mesh 2** node, then click **Size**.
- 2 In the **Size** settings window, locate the **Element Size Parameters** section.
- 3 In the **Minimum element size** edit field, type $r0/2$.

Size 1

- I In the **Model Builder** window, under **Model 1>Meshes>Mesh 2** right-click **Free Tetrahedral 1** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Click **Paste Selection**.
- 5 Go to the **Paste Selection** dialog box.
- 6 In the **Selection** edit field, type $7, 10$.
- 7 Click the **OK** button.
- 8 In the **Size** settings window, locate the **Element Size** section.
- 9 Click the **Custom** button.
- 10 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- II In the associated edit field, type $r0/2$.

ROOT

Add a new frequency domain study for the mesh convergence analysis.

- In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

- Go to the **Model Wizard** window.
- Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- Click **Finish**.

STUDY 2

A parametric sweep is needed to loop over the mesh sizes.

Parametric Sweep

- In the **Model Builder** window, right-click **Study 2** and choose **Parametric Sweep**.
- In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- Click **Add**.
- In the table, enter the following settings:

Parameter names	Parameter value list
h_size	3 6 9 12 15 20

Step 1: Frequency Domain

- In the **Model Builder** window, under **Study 2** click **Step 1: Frequency Domain**.
- In the **Frequency Domain** settings window, locate the **Study Settings** section.
- In the **Frequencies** edit field, type f_0 .
- In the **Model Builder** window, click **Study 2**.
- In the **Study** settings window, locate the **Study Settings** section.
- Clear the **Generate default plots** check box.
- Right-click **Study 2** and choose **Compute**.

RESULTS

Continue to plot the relative error versus elements per wavelength.

ID Plot Group 2

- In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- In the **ID Plot Group** settings window, locate the **Data** section.
- From the **Data set** list, choose **Solution 4**.

- 4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 5 In the associated edit field, type Number of elements / λ_0 .
- 6 Select the **y-axis label** check box.
- 7 In the associated edit field, type **Relative error**.
- 8 Right-click **Results>ID Plot Group 2** and choose **Point Graph**.
- 9 In the **Point Graph** settings window, locate the **Selection** section.
- 10 Click **Paste Selection**.
- 11 Go to the **Paste Selection** dialog box.
- 12 In the **Selection** edit field, type 2.
- 13 Click the **OK** button.
- 14 In the **Point Graph** settings window, locate the **y-Axis Data** section.
- 15 In the **Expression** edit field, type $(4\pi(\epsilon_r \cdot \text{normEfar}^2 / E_0^2) / (\pi r_0^2 - RCS1)) / RCS1$.
- 16 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **h_size**.
- 17 From the **Parameter** list, choose **Expression**.
- 18 In the **Expression** edit field, type **h_size**.
- 19 Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** edit field, type 2.
- 20 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.
- 21 From the **Positioning** list, choose **In data points**.
- 22 Click the **x-Axis Log Scale** button on the Graphics toolbar.
- 23 Click the **y-Axis Log Scale** button on the Graphics toolbar.

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.

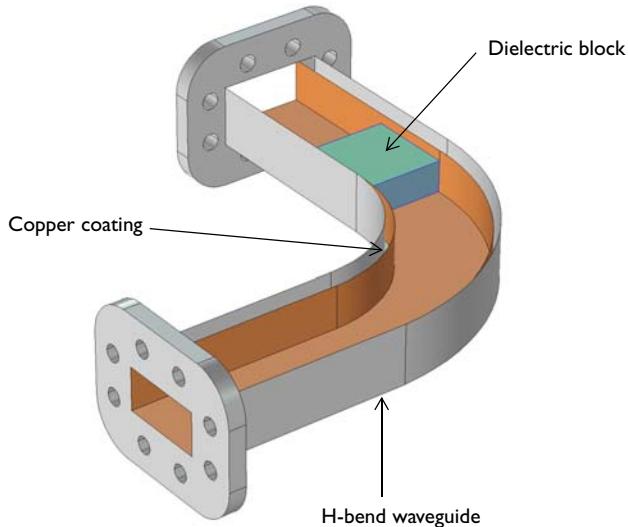


Figure 1: A waveguide bend with a dielectric block inside. Top boundaries of the waveguide are removed only for visualization.

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 $\epsilon_r = 2.1$, with a loss tangent that is a function of temperature, $\delta = 0.001(1 + T/300 \text{ K})$. The thermal conductivity of this block is also a function of temperature, $k = 0.3(1 + T/300 \text{ K}) \text{ W/m/K}$. Furthermore, the density is 2200 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 will raise 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 will be conducted away into the walls. The outside boundaries of the walls are assumed to be facing ambient air, which will lead to free convective cooling off of these faces. This model 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](#).

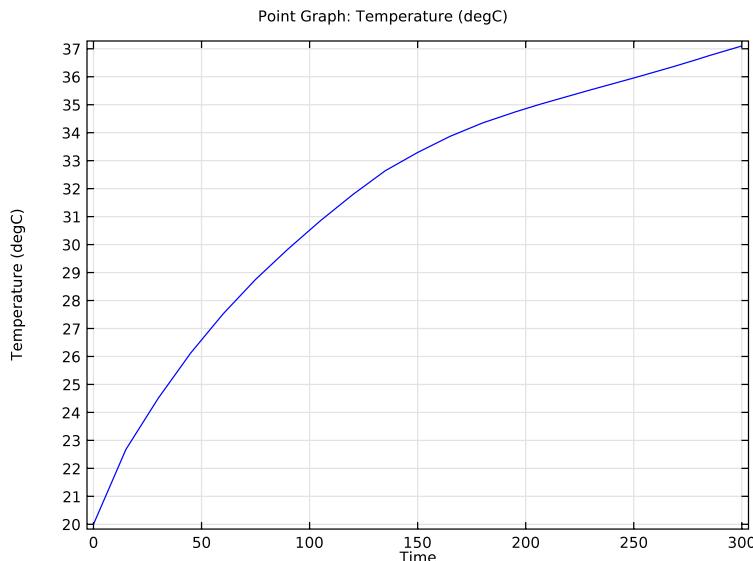


Figure 2: The maximum temperature, evaluated over the volume of the block, is plotted as a function of temperature.

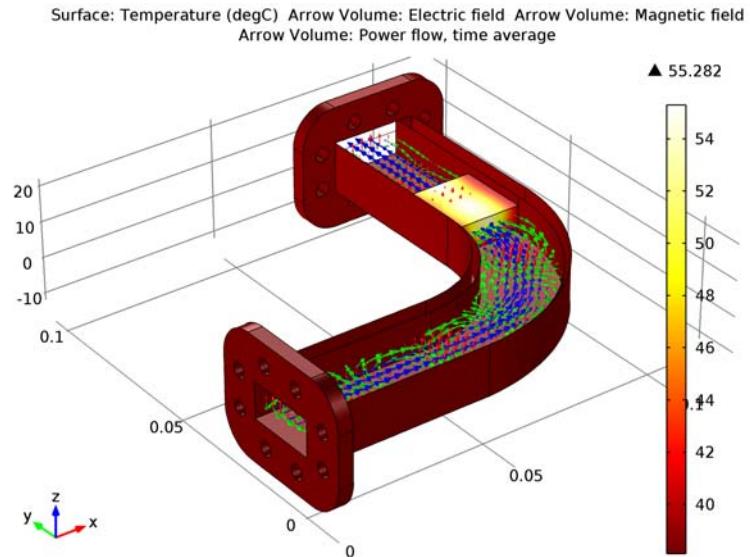


Figure 3: The electric fields (red arrows) magnetic fields (green arrows) and power flow (blue arrows) are shown inside of the waveguide. The steady-state temperature is plotted on the block and waveguide walls.

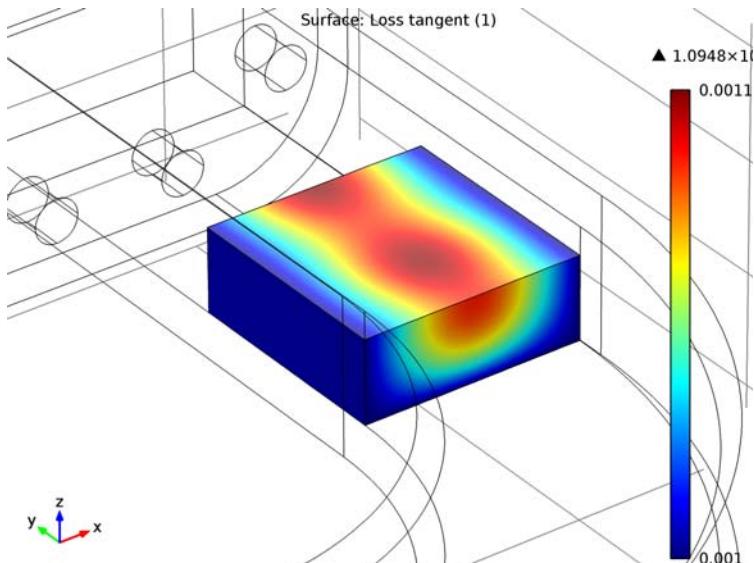


Figure 4: The loss tangent within the dielectric block for the steady-state solution shows that the variation in temperature affects the material properties.

Model Library path: RF_Module/Microwave_Heating/rf_heating

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Heat Transfer>Electromagnetic Heating>Microwave Heating (mh)**.
- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency-Transient**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS*Parameters*

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
f0	10[GHz]	Current frequency
lda0	c_const/f0	Wavelength, air
h_max	0.2*lda0	Maximum mesh element size, air

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

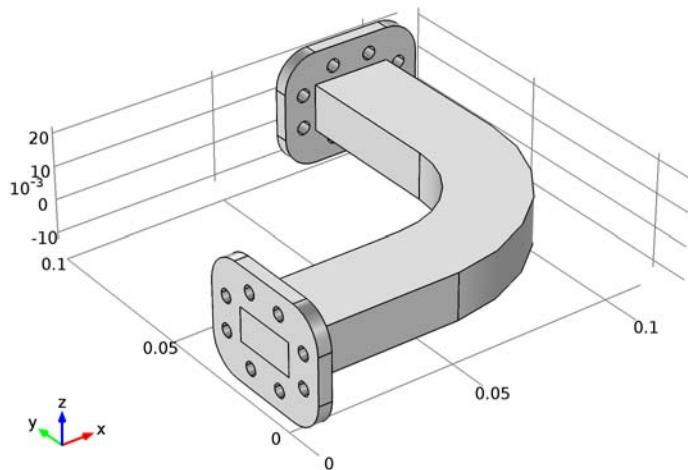
GEOMETRY I

First, import the geometry of the waveguide including a dielectric block inside the waveguide.

Import I

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Import**.

- 2 In the **Import** settings window, locate the **Import** section.
- 3 Click the **Browse** button.
- 4 Browse to the model's Model Library folder and double-click the file `rf_heating.mphbin`.
- 5 Click the **Import** button.



Use the wireframe rendering to see the inner parts of the waveguide.

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

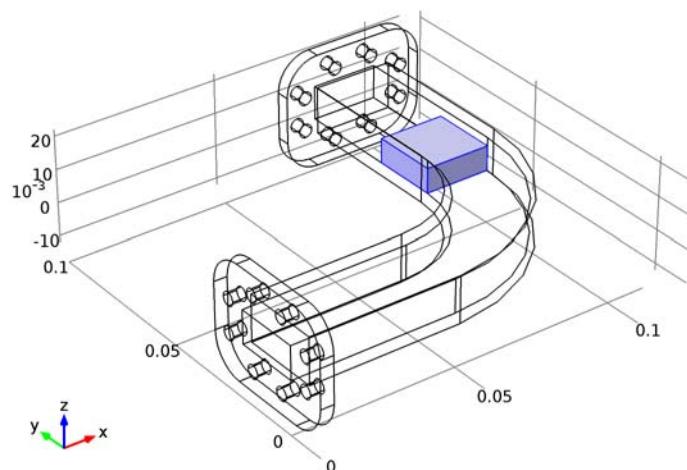
DEFINITIONS

Create a set of selections for use before setting up the physics. First, create a selection for the dielectric block.

Explicit 1

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

- 2** Select Domain 3 only.



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

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

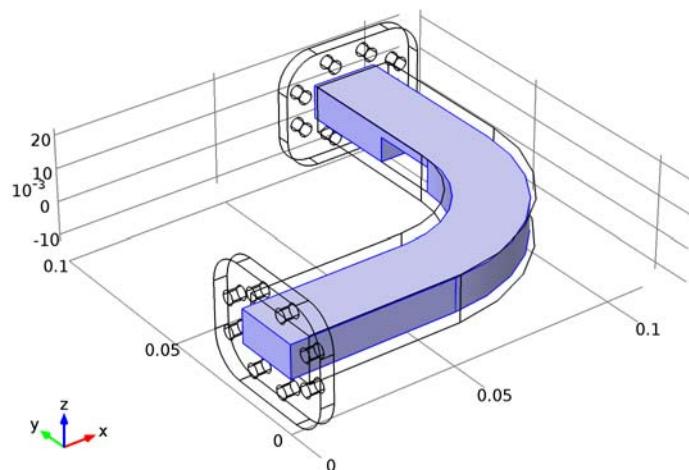
- 5** Click **OK**.

Add a selection for the air-filled region inside the waveguide.

Explicit 2

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

- 2** Select Domain 2 only.



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

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

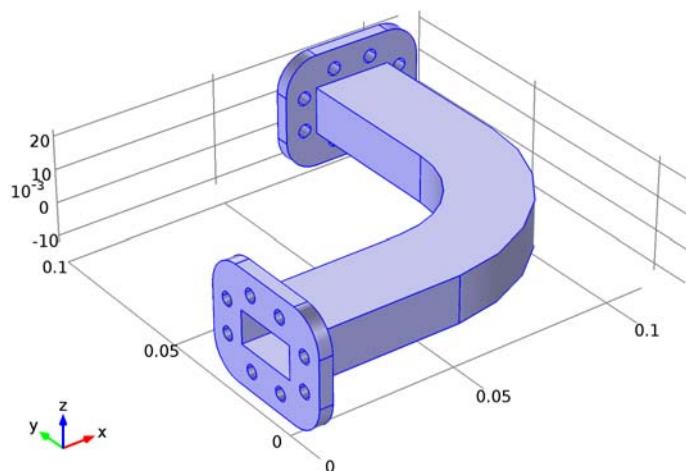
- 5** Click **OK**.

Add a selection for the waveguide structure.

Explicit 3

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

- 2 Select Domain 1 only.



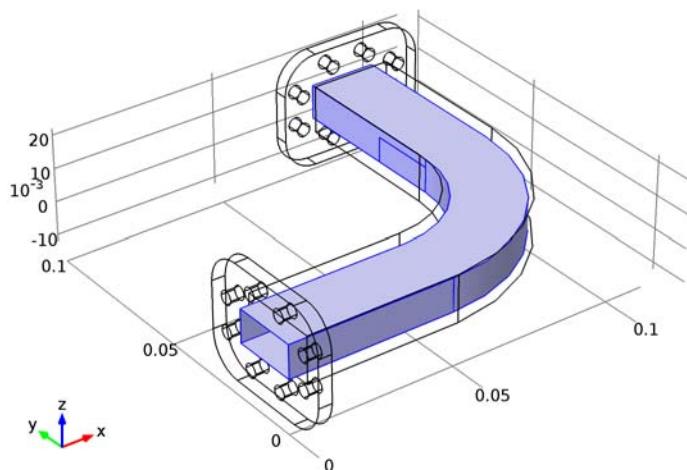
- 3 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.
- 4 Go to the **Rename Explicit** dialog box and type **Waveguide** in the **New name** edit field.
- 5 Click **OK**.

Add a selection for the inner surface of the waveguide.

Explicit 4

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundaries 16–18, 35, 53, 54, 72, 74, 75, 78, 96, and 97 only.



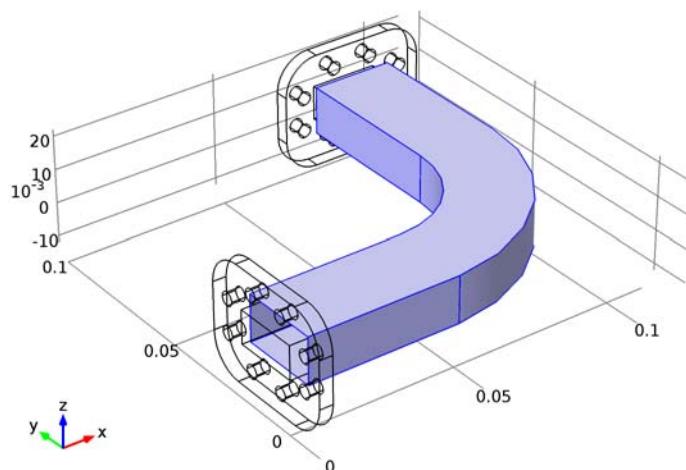
- 5 Right-click **Model 1>Definitions>Explicit 4** and choose **Rename**.
6 Go to the **Rename Explicit** dialog box and type **Waveguide inside surfaces** in the **New name** edit field.
7 Click **OK**.

Add a selection for the outer surface of the waveguide.

Explicit 5

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundaries 48–52, 55, 69, and 98 only.



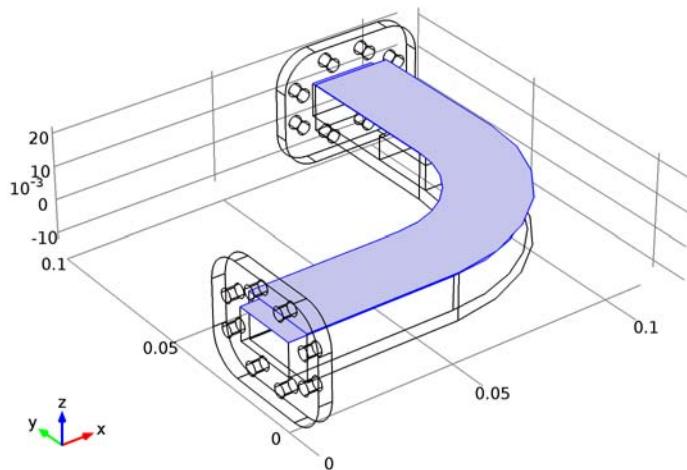
- 5 Right-click **Model 1>Definitions>Explicit 5** and choose **Rename**.
- 6 Go to the **Rename Explicit** dialog box and type **Waveguide outside surfaces** in the **New name** edit field.
- 7 Click **OK**.

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 Right-click **Definitions** and choose **View**.
- 2 Click the **Wireframe Rendering** button on the Graphics toolbar.
- 3 In the **Model Builder** window, under **Model 1>Definitions** right-click **View 2** and choose **Hide Geometric Entities**.
- 4 In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.
- 5 From the **Geometric entity level** list, choose **Boundary**.

- 6 Select Boundaries 18 and 50 only.



MICROWAVE HEATING

Microwave Heating Model I

- 1 In the **Model Builder** window, expand the **Model I>Microwave Heating** node, then click **Microwave Heating Model I**.
- 2 In the **Microwave Heating Model** settings window, locate the **Electric Displacement Field** section.
- 3 From the **Electric displacement field model** list, choose **Loss tangent**.

Heat Transfer in Solids I

- 1 In the **Model Builder** window, right-click **Microwave Heating** and choose the domain setting **Heat Transfer>Heat Transfer in Solids**.
- 2 In the **Heat Transfer in Solids** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Waveguide**.

Heat Flux I

- 1 Right-click **Microwave Heating** and choose the boundary condition **Heat Transfer>Heat Flux**.
- 2 In the **Heat Flux** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Waveguide outside surfaces**.

4 Locate the **Heat Flux** section. Click the **Inward heat flux** button.

5 In the *h* edit field, type 5.

Wave Equation, Electric I

1 Right-click **Microwave Heating** and choose the domain setting **Electromagnetic Waves>Wave Equation, Electric**.

2 In the **Wave Equation, Electric** settings window, locate the **Domain Selection** section.

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

Impedance Boundary Condition I

1 Right-click **Microwave Heating** and choose the boundary condition **Electromagnetic Waves>Impedance Boundary Condition**.

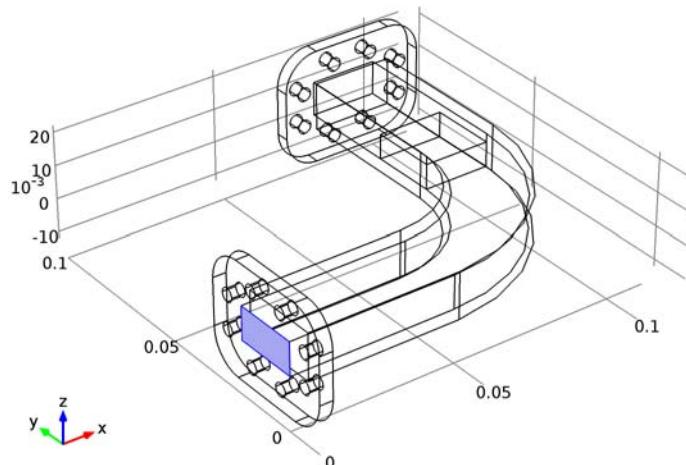
2 In the **Impedance Boundary Condition** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Waveguide inside surfaces**.

Port I

1 Right-click **Microwave Heating** and choose the boundary condition **Electromagnetic Waves>Port**.

2 Select Boundary 15 only.



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

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

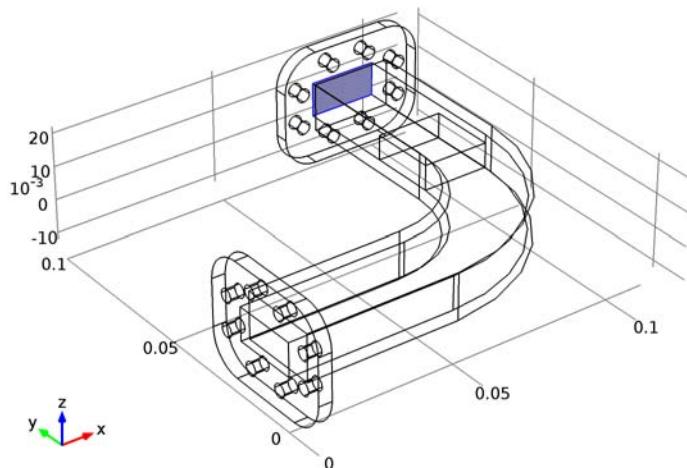
5 From the **Wave excitation at this port** list, choose **On**.

6 In the P_{in} edit field, type 100.

Port 2

1 Right-click **Microwave Heating** and choose the boundary condition **Electromagnetic Waves>Port**.

2 Select Boundary 79 only.



3 In the **Port** settings window, 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.

Material Browser

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

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Aluminum**.

4 Right-click and choose **Add Material to Model** from the menu.

Aluminum

- 1 In the **Model Builder** window, under **Model 1>Materials** click **Aluminum**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Waveguide**.

Material Browser

- 1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
- 2 In the **Material Browser** window, locate the **Materials** section.
- 3 In the tree, select **Built-In>Air**.
- 4 Right-click and choose **Add Material to Model** from the menu.

Air

- 1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Air**.

Material 3

- 1 In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2 Select Domain 3 only.
- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

Property	Name	Value
Relative permittivity (real part)	epsilonPrim	2.1
delta	delta	0.001*(T/300[K])
Relative permeability	mur	1
Thermal conductivity	k	0.3[W/m/K]*(T/300[K])
Density	rho	2200
Heat capacity at constant pressure	Cp	1050

- 5 Right-click **Material 3** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type **Dielectric** in the **New name** edit field.
- 7 Click **OK**.

Material Browser

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

- 2** In the **Material Browser** window, locate the **Materials** section.
- 3** In the tree, select **Built-In>Copper**.
- 4** Right-click and choose **Add Material to Model** from the menu.

Copper

- 1** In the **Model Builder** window, under **Model 1>Materials** click **Copper**.
- 2** In the **Material** settings window, 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

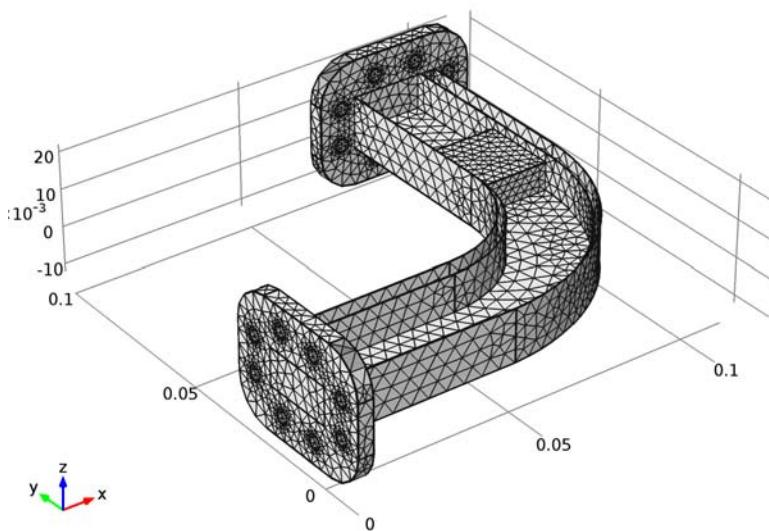
- 1** In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Size**.
- 2** In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Geometric entity level** list, choose **Domain**.
- 4** From the **Selection** list, choose **Air**.
- 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 edit field, type `h_max`.

Size 2

- 1** In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
- 2** In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Geometric entity level** list, choose **Domain**.
- 4** From the **Selection** list, choose **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 edit field, type `h_max/sqrt(4.5)`.

Free Tetrahedral I

- 1 Right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Settings** window, click **Build All**.



STUDY I

Step 1: Frequency-Transient

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency-Transient**.
- 2 In the **Frequency-Transient** settings window, locate the **Study Settings** section.
- 3 In the **Times** edit field, type range $(0, 15, 300)$.
- 4 Select the **Relative tolerance** check box.
- 5 In the associated edit field, type 0.001 .
- 6 In the **Frequency** edit field, type $f0$.
- 7 In the **Model Builder** window, click **Study 1**.
- 8 In the **Study** settings window, locate the **Study Settings** section.
- 9 Clear the **Generate default plots** check box.
- 10 Click the **Compute** button.

RESULTS

Data Sets

Plot the transient response of the peak temperature.

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **More Data Sets>Maximum**.

ID Plot Group 1

- 1 Right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Maximum 1**.
- 4 Right-click **Results>ID Plot Group 1** and choose **Point Graph**.
- 5 In the **Point Graph** settings window, locate the **y-Axis Data** section.
- 6 From the **Unit** list, choose **degC**.
- 7 Click the **Plot** button. 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.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency-Stationary**.
- 3 Click **Finish**.

STUDY 2

Step 1: Frequency-Stationary

- 1 In the **Model Builder** window, under **Study 2** click **Step 1: Frequency-Stationary**.
- 2 In the **Frequency-Stationary** settings window, locate the **Study Settings** section.
- 3 In the **Frequency** edit field, type **f0**.
- 4 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Temperature (mh)

The default plot shows the distribution of the temperature. First, change the unit to the degree Celsius and then, add arrow plots of the electric fields, magnetic fields, and power flow.

- 1 In the **Model Builder** window, expand the **Temperature (mh)** node, then click **Surface I**.
- 2 In the **Surface** settings window, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.
- 4 In the **Model Builder** window, right-click **Temperature (mh)** and choose **Arrow Volume**.
- 5 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Electric>Electric field (Ex,Ey,Ez)**.
- 6 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** edit field, type 40.
- 7 Find the **y grid points** subsection. In the **Points** edit field, type 40.
- 8 Find the **z grid points** subsection. In the **Points** edit field, type 1.
- 9 Click the **Plot** button.
- 10 In the **Model Builder** window, under **Results>Temperature (mh)** right-click **Arrow Volume I** and choose **Duplicate**.
- 11 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Magnetic>Magnetic field (mh.Hx,mh.Hy,mh.Hz)**.
- 12 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.
- 13 In the **Model Builder** window, under **Results>Temperature (mh)** right-click **Arrow Volume 2** and choose **Duplicate**.
- 14 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Energy and power>Power flow, time average (mh.Poavx,...,mh.Poavz)**.
- 15 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](#).

3D Plot Group 3

- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Solution 2**.
- 4 Right-click **Results>3D Plot Group 3** and choose **Surface**.
- 5 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Microwave Heating (Electromagnetic Waves)>Material properties>Loss tangent (mh.delta)**.
- 6 Click the **Plot** button.
- 7 Click the **Zoom Box** button on the Graphics toolbar and then use the mouse to zoom in.

Tunable Evanescent Mode Cavity Filter Using a Piezo Actuator

Introduction

An evanescent mode cavity filter can be realized by adding a structure inside of the cavity. This structure changes the resonant frequency below that of the dominant mode of the unfilled cavity. A piezo actuator is used to control the size of a small air gap which provides the tunability of the resonant frequency.

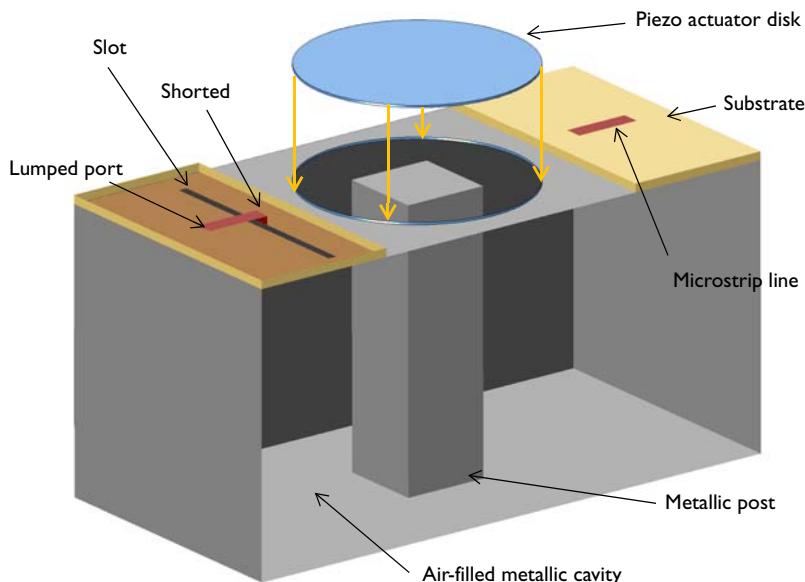


Figure 1: A tunable evanescent mode cavity filter is composed of a rectangular cavity with a metallic post, a piezo actuator disk, and slot-coupled microstrip lines. There is a small gap between the top of the post and the bottom side of the piezo actuator. The front part of the cavity wall is removed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

Note: In addition to the RF Module, this model requires one of the Acoustics Module, the MEMS Module, or the Structural Mechanics Module.

Model Definition

This example starts from a basic rectangular cavity filter, whose resonant frequencies are given by

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

where a and b are the waveguide aperture dimensions and d is the length of the waveguide cavity. In this model, the cavity width, height, and length are $a = 100$ mm, $b = 50$ mm, and $d = 50$ mm, respectively. The resulting resonant frequency of the dominant mode, TE₁₀₁, 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.

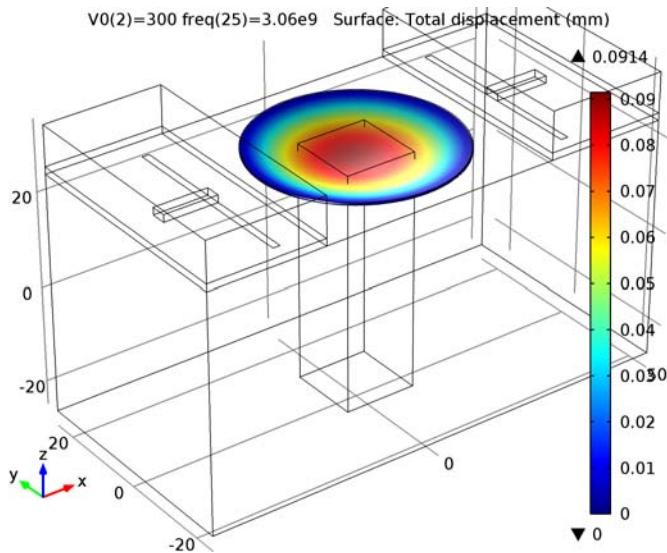


Figure 2: This plot shows the total piezo displacement when 300 V is applied on the actuator. The visualization is exaggerated to emphasize the deflection.

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

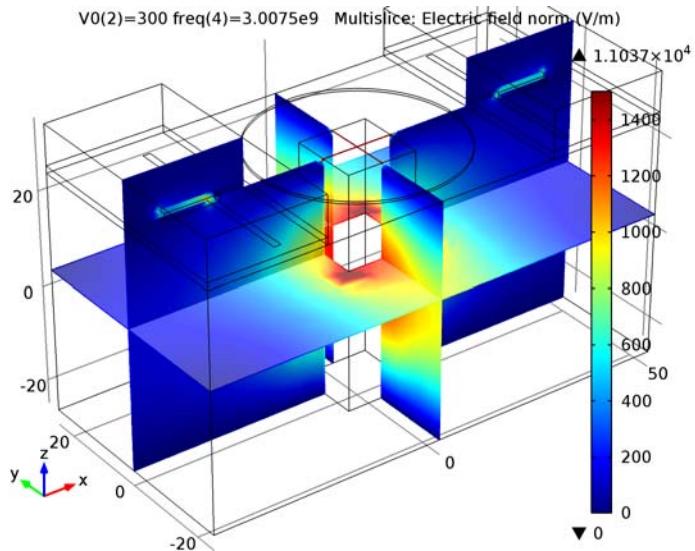


Figure 3: The dominant mode inside the cavity is observed from the electric field distribution plot.

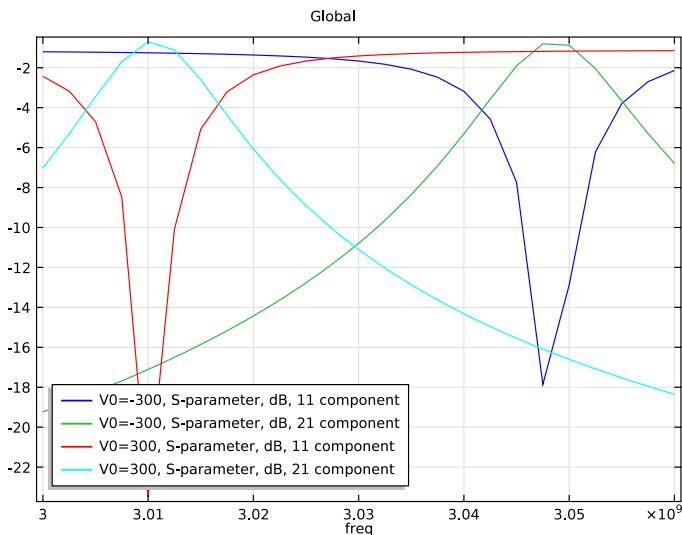


Figure 4: The deflection for the piezo device controlled by the input bias can shift the resonance frequency of the filter.

Notes About the COMSOL Implementation

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

Model Library path: RF_Module/Passive_Devices/tunable_cavity_filter

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Structural Mechanics>Piezoelectric Devices (pzd)**.
- 4 Click **Add Selected**.
- 5 In the **Add physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.
- 6 Click **Add Selected**.
- 7 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 8 Click **Add Selected**.
- 9 Click **Next**.
- 10 Find the **Studies** subsection. In the tree, select **Custom Studies>Empty Study**.
- II Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

- 3** In the table, enter the following settings:

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

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

GEOOMETRY I

- In the **Model Builder** window, under **Model I** click **Geometry I**.
- In the **Geometry** settings window, locate the **Units** section.
- From the **Length unit** list, choose **mm**.

First, create a block for the cavity.

Block I

- In the **Model Builder** window, right-click **Geometry I** and choose **Block**.
- In the **Block** settings window, locate the **Size and Shape** section.
- In the **Width** edit field, type 100.
- In the **Depth** edit field, type 50.
- In the **Height** edit field, type 50.
- Locate the **Position** section. From the **Base** list, choose **Center**.
- Right-click **Model I>Geometry I>Block I** and choose **Rename**.
- Go to the **Rename Block** dialog box and type **Cavity** in the **New name** edit field.
- Click **OK**.

Add a substrate block.

Block 2

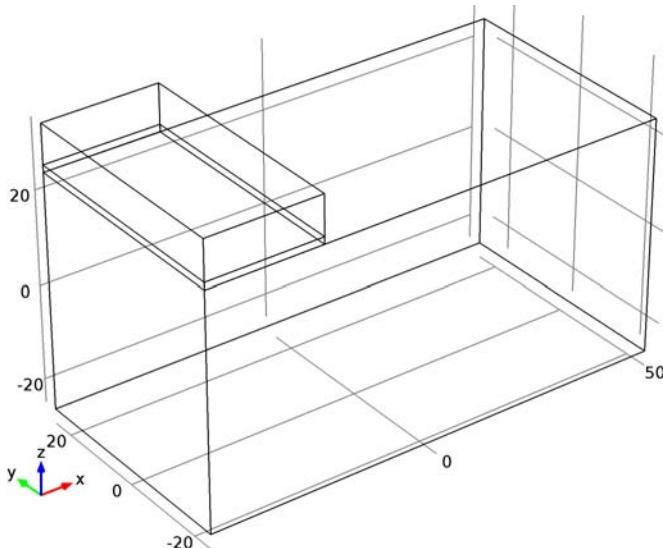
- 1 Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type 25.
- 4 In the **Depth** edit field, type 50.
- 5 In the **Height** edit field, type thickness.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** edit field, type -37.5.
- 8 In the **z** edit field, type $25+thickness/2$.
- 9 Right-click **Model 1>Geometry 1>Block 2** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type **Substrate** in the **New name** edit field.
- II Click **OK**.

Add a block for the air domain.

Block 3

- 1 Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type 25.
- 4 In the **Depth** edit field, type 50.
- 5 In the **Height** edit field, type 10.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** edit field, type -37.5.
- 8 In the **z** edit field, type 30.
- 9 Right-click **Model 1>Geometry 1>Block 3** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type **Air block** in the **New name** edit field.
- II Click **OK**.
- I2 Right-click **Model 1>Geometry 1>Block 3** and choose **Build Selected**.

- I3 Click the **Wireframe Rendering** button on the Graphics toolbar.



Add a block for the microstrip line feed.

Block 4

- I Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `l_feed+w_slot`.
- 4 In the **Depth** edit field, type `3.2`.
- 5 In the **Height** edit field, type `thickness`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** edit field, type `-x_slot-l_feed/2`.
- 8 In the **z** edit field, type `25+thickness/2`.
- 9 Right-click **Model 1>Geometry 1>Block 4** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type `Feed line` in the **New name** edit field.
- II Click **OK**.

Add a work plane where you will draw a slot.

Work Plane 1

- I Right-click **Geometry 1** and choose **Work Plane**.
- 2 In the **Work Plane** settings window, locate the **Plane Definition** section.

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

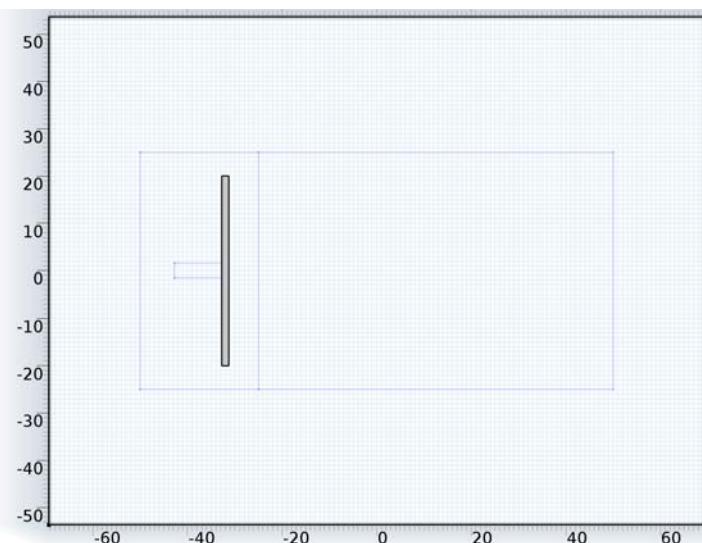
Plane Geometry

Add a rectangle for the slot.

- 1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.
- 2 Click the **Zoom Extents** button on the Graphics toolbar.

Rectangle 1

- 1 In the **Rectangle** settings window, locate the **Size** section.
- 2 In the **Width** edit field, type `w_slot`.
- 3 In the **Height** edit field, type `l_slot`.
- 4 Locate the **Position** section. From the **Base** list, choose **Center**.
- 5 In the **xw** edit field, type `-x_slot`.
- 6 Click the **Build Selected** button.



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

Mirror 1

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Mirror**.
- 2 Select the objects **blk3**, **blk4**, **blk2**, and **wpl** only.
- 3 In the **Mirror** settings window, locate the **Input** section.

- 4 Select the **Keep input objects** check box.
- 5 Locate the **Normal Vector to Plane of Reflection** section. In the **x** edit field, type 1.
- 6 In the **z** edit field, type 0.

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

Block 5

- I Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type 15.
- 4 In the **Depth** edit field, type 15.
- 5 In the **Height** edit field, type **h_post**.
- 6 Locate the **Position** section. In the **x** edit field, type -7.5.
- 7 In the **y** edit field, type -7.5.
- 8 In the **z** edit field, type -25.
- 9 Right-click **Model 1>Geometry 1>Block 5** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type Post in the **New name** edit field.
- II Click **OK**.

Add a cylinder for the piezo actuator disk.

Cylinder 1

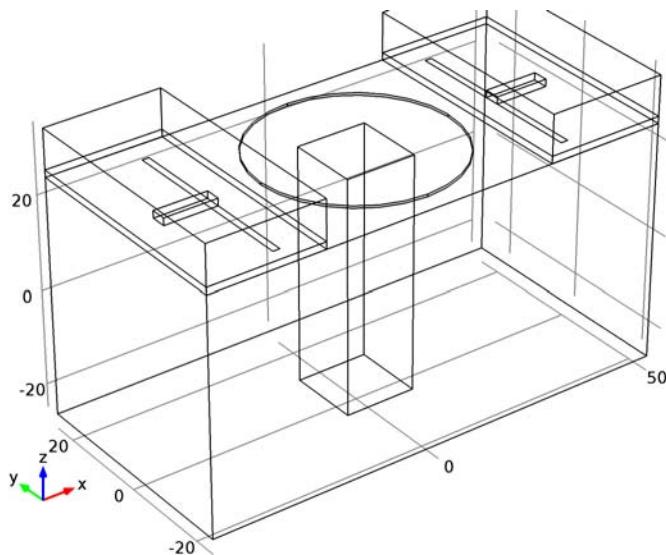
- I Right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type 21.
- 4 In the **Height** edit field, type 0.5.
- 5 Locate the **Position** section. In the **z** edit field, type 25.
- 6 Right-click **Model 1>Geometry 1>Cylinder 1** and choose **Rename**.
- 7 Go to the **Rename Cylinder** dialog box and type Piezo actuator in the **New name** edit field.
- 8 Click **OK**.

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

Difference 1

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

- 2 Select the object **blk1** only.
- 3 In the **Difference** settings window, locate the **Difference** section.
- 4 Under **Objects to subtract**, click **Activate Selection**.
- 5 Select the object **blk5** only.
- 6 Click the **Build All** button.



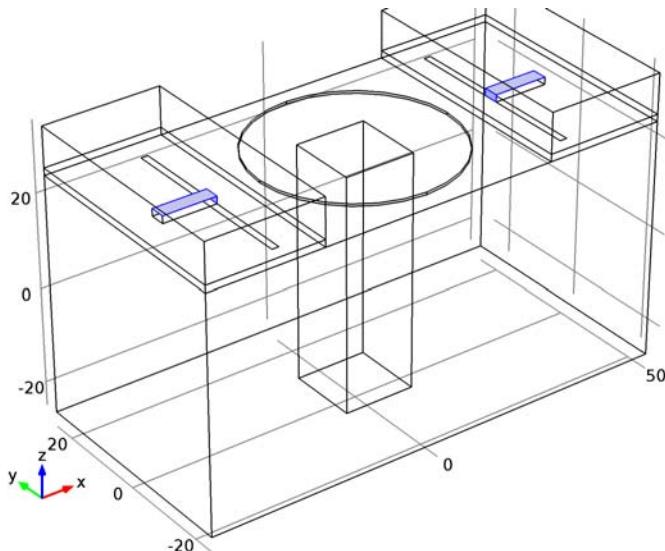
DEFINITIONS

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

Explicit 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

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



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

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

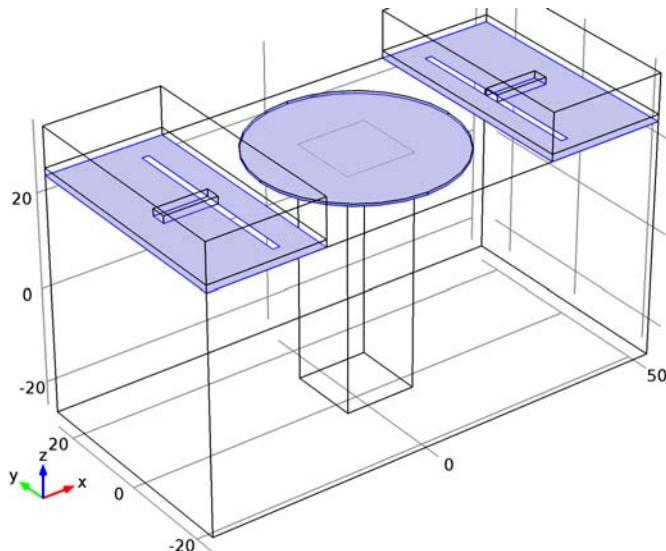
- 7 Click **OK**.

Add a selection for the ground.

Explicit 2

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

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



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

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

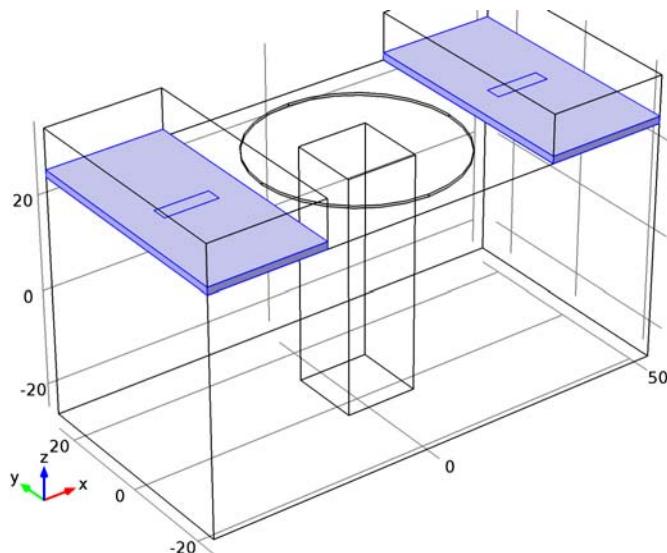
- 7 Click **OK**.

Add a selection for the substrate.

Explicit 3

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

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



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

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

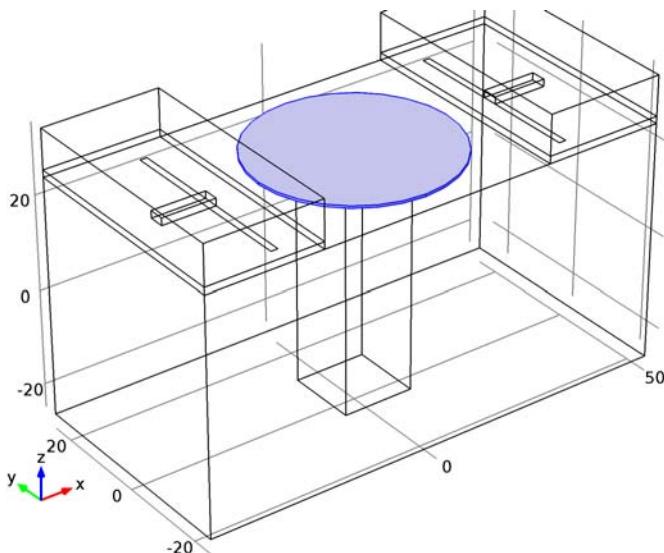
5 Click **OK**.

Add a selection for the piezo actuator disk.

Explicit 4

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

2 Select Domain 5 only.



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

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

5 Click **OK**.

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

PIEZOELECTRIC DEVICES

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

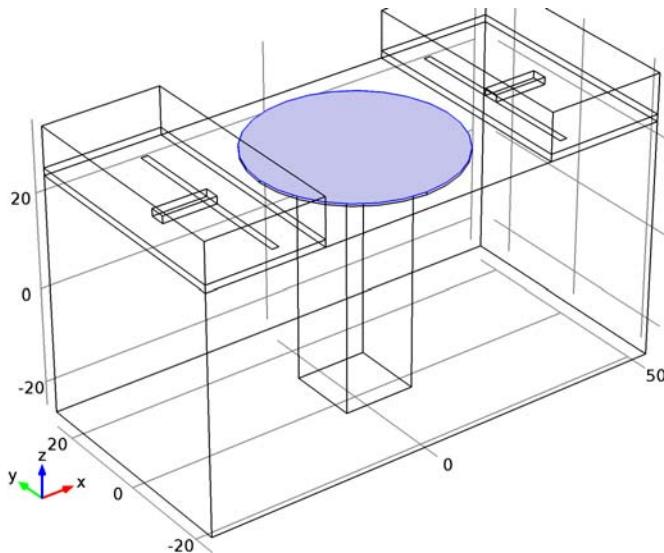
2 From the **Selection** list, choose **Piezo actuator**.

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

Electric Potential 1

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

2 Select Boundary 29 only.



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

4 In the V_0 edit field, type $-V_0$.

Ground 1

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

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

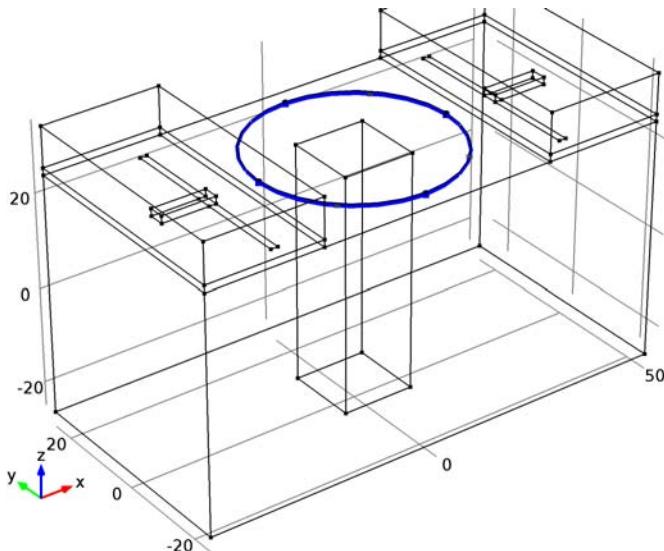
3 From the **Selection** list, choose **Ground**.

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

Fixed Constraint 1

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

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

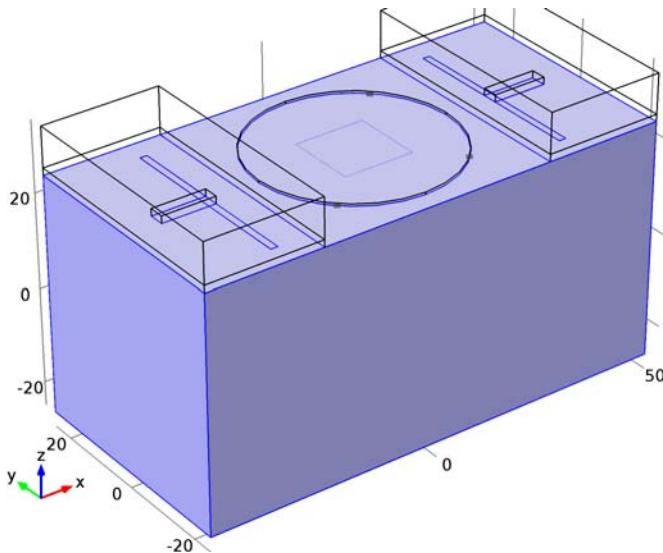


Set up the Moving Mesh interface. Because the substrate and air domains are deflection free regions, do not include them in this physics.

MOVING MESH

- 1** In the **Model Builder** window, under **Model 1** click **Moving Mesh**.
- 2** In the **Moving Mesh** settings window, locate the **Domain Selection** section.
- 3** Click **Clear Selection**.

- 4** Select Domain 1 only.



Prescribed Mesh Displacement 1

- 1** In the **Model Builder** window, under **Model 1 > Moving Mesh** click **Prescribed Mesh Displacement 1**.
- 2** In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.
- 3** In the d_x edit field, type u .
- 4** In the d_y edit field, type v .
- 5** In the d_z edit field, type w .

Free Deformation 1

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

Prescribed Mesh Displacement 2

- 1** Right-click **Moving Mesh** and choose **Prescribed Mesh Displacement**.
- 2** In the **Prescribed Mesh Displacement** settings window, locate the **Boundary Selection** section.
- 3** Click **Paste Selection**.
- 4** Go to the **Paste Selection** dialog box.

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

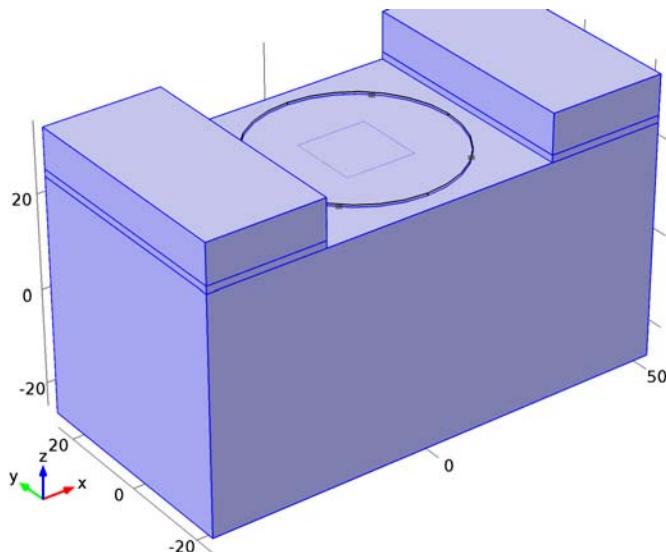
6 Click the **OK** button.

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

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

2 Select Domains 1–4 and 6–8 only.



Perfect Electric Conductor 2

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

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

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

Perfect Electric Conductor 3

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

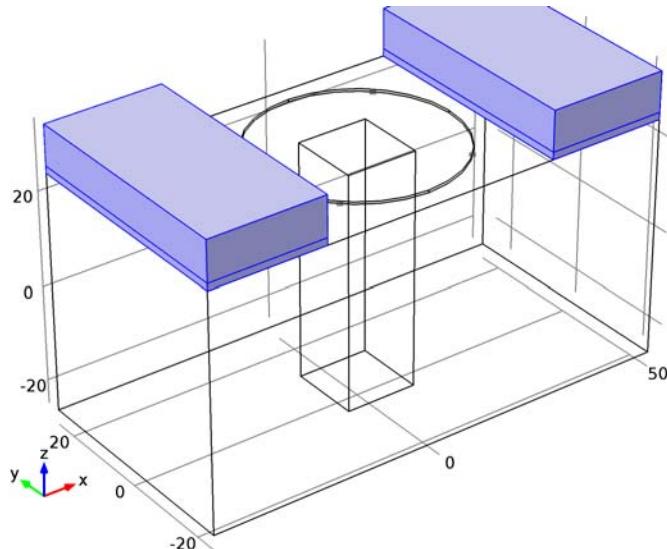
2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.

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

Scattering Boundary Condition 1

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

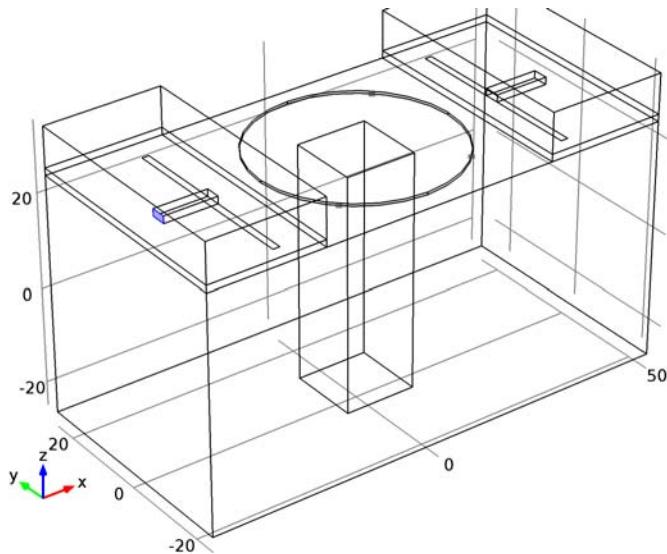
- 2 Select Boundaries 4, 5, 7, 8, 10, 12, 13, 23, 25, 37, 38, 40, 41, 43–45, 56, and 57 only.



Lumped Port 1

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

2 Select Boundary 14 only.



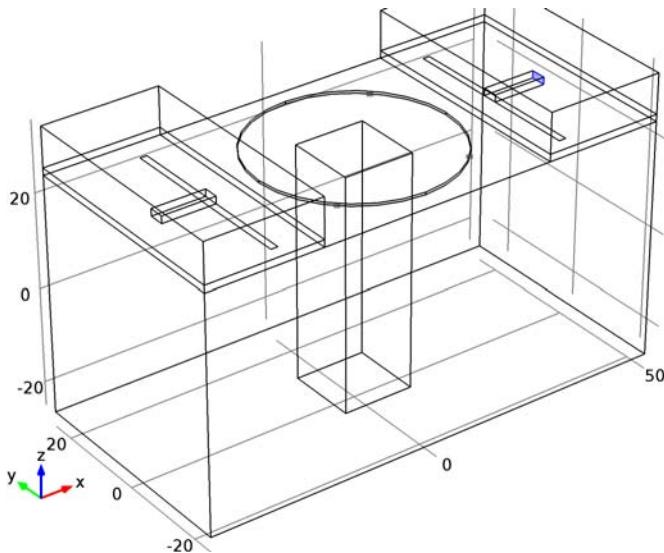
3 In the **Lumped Port** settings window, locate the **Port Properties** section.

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

Lumped Port 2

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

- 2** Select Boundary 54 only.



MATERIALS

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

Material Browser

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

Lead Zirconate Titanate (PZT-5H)

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

Material Browser

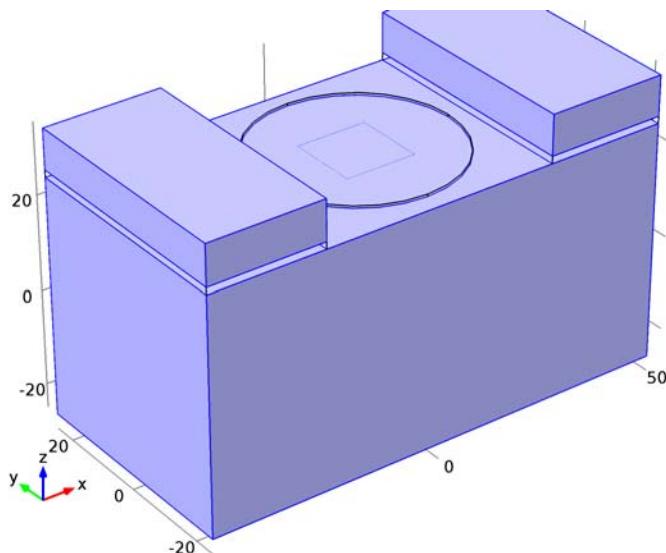
- 1** In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
- 2** In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click Add Material to Model.

Air

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

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



Material 3

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

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

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

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

5 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilon_r	3.38
Relative permeability	mu_r	1
Electrical conductivity	sigma	0

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

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

8 Click **OK**.

MESH I

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

Size 1

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

Size 2

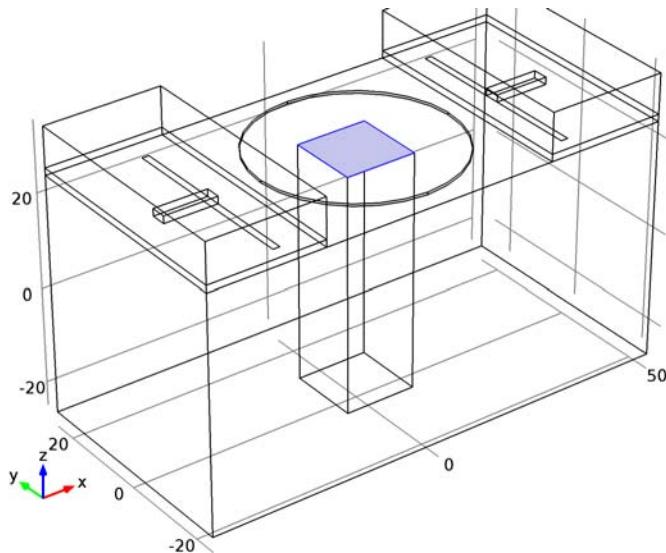
- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Substrate**.
- 5 Locate the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

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

Size 3

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

4 Select Boundary 32 only.

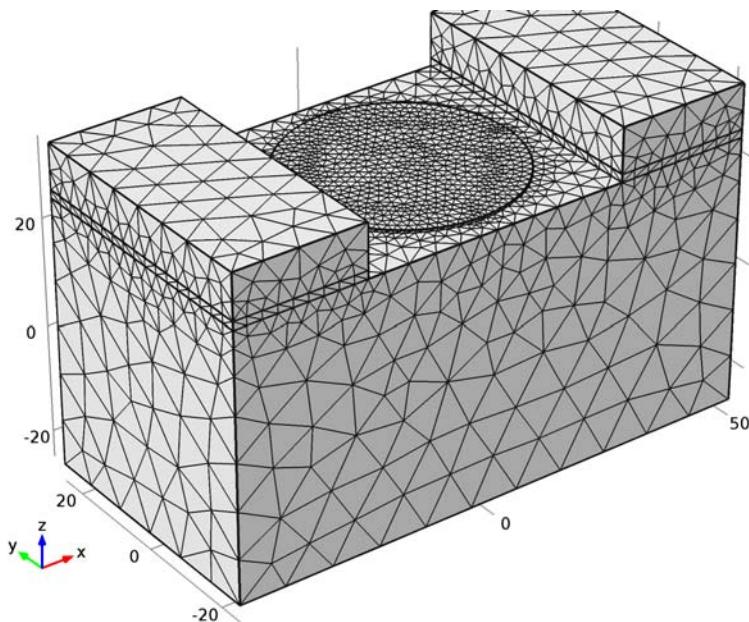


- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type $h_{\max}/10$.

Free Tetrahedral 1

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

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



STUDY I

Step 1: Stationary

- 1** In the **Model Builder** window, right-click **Study I** and choose **Study Steps>Stationary**.
- 2** In the **Stationary** settings window, locate the **Physics and Variables Selection** section.
- 3** In the table, enter the following settings:

Physics	Solve for
Electromagnetic Waves, Frequency Domain	<input checked="" type="checkbox"/>

Step 2: Frequency Domain

- 1** Right-click **Study I** and choose **Study Steps>Frequency Domain**.
- 2** In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3** In the **Frequencies** edit field, type `range(f_min,0.0025[GHz],f_max)`.

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

Physics	Solve for
Piezoelectric Devices	x
Moving Mesh	x

Parametric Sweep

- 1 Right-click **Study 1** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names	Parameter value list
V0	-300, 300

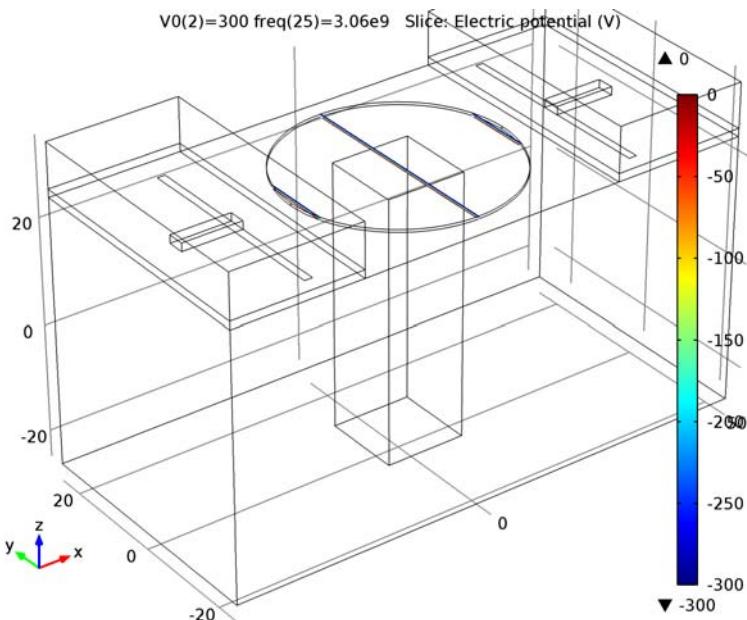
- 5 Right-click **Study 1** and choose **Compute**.

RESULTS

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

Displacement (pzd)

This plot shows the deflected piezo actuator disk; compare with [Figure 2](#).

Potential (pzd)

The electric potential inside the piezo actuator disk.

Electric Field (emw)

- 1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Parameter value (freq)** list, choose **3.0075e9**.
- 4 In the **Model Builder** window, under **Results>Electric Field (emw)** click **Multislice**.
- 5 In the **Multislice** settings window, click to expand the **Range** section.
- 6 Select the **Manual color range** check box.
- 7 Set the **Maximum** value to **3**.

The resulting plot shows strong electric fields resulting from the dominant resonance at the center of the cavity as well as in the gap between the metallic post and the ceiling of the cavity. Compare the plot with that shown in [Figure 3](#).

ID Plot Group 4

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Solution 3**.

- 4 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.
- 5 Right-click **Results>1D Plot Group 4** and choose **Global**.
- 6 In the **Global** settings window, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 11 component (emw.S11dB)**.
- 7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 21 component (emw.S21dB)**.
- 8 Click the **Plot** button.

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

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 model is to compute the far-field pattern and to compute the impedance of the structure. Good matching is observed over a wide frequency band.

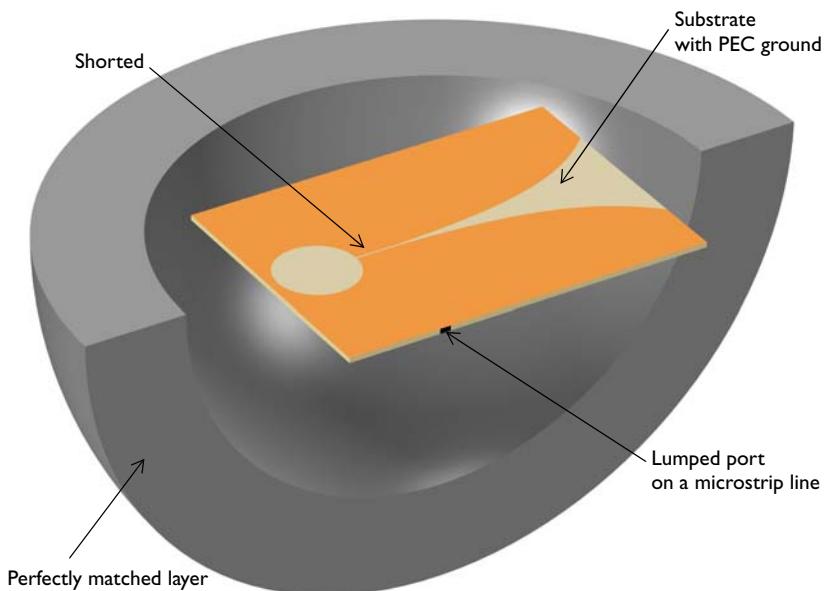


Figure 1: The Vivaldi antenna is realized on a thin dielectric substrate. The entire domain is bounded by a perfectly matched layer.

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.

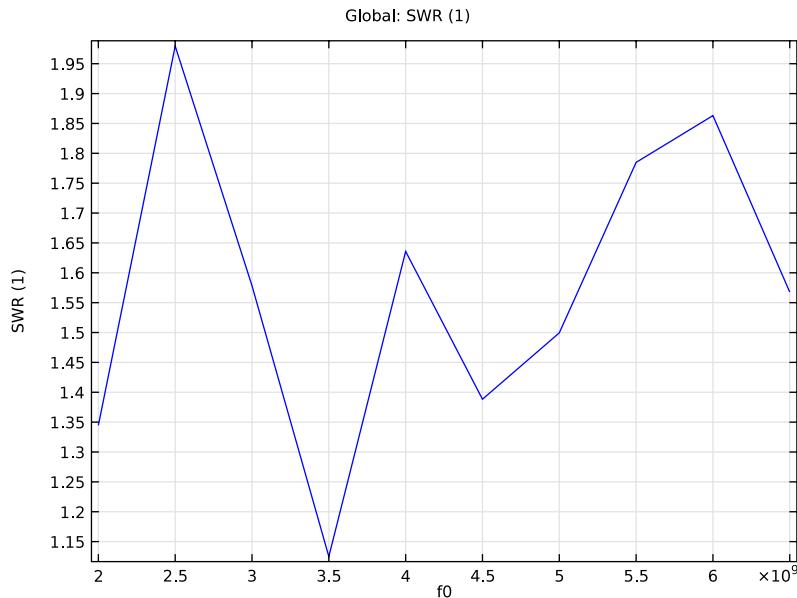


Figure 2: The frequency response SWR of the Vivaldi antenna shows wide-band impedance matching, better than 2:1 in the simulated frequency range.

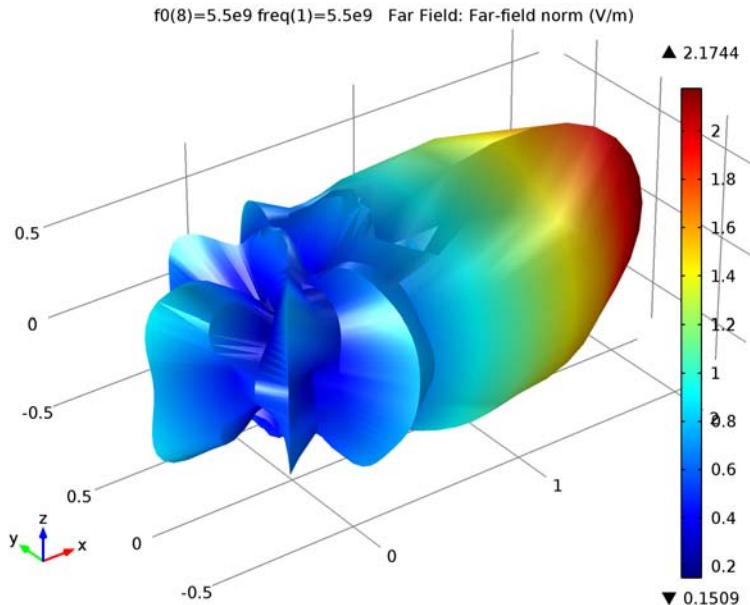


Figure 3: 3D far-field pattern at 5.5 GHz shows a directional radiation pattern.

References

1. D.M. Pozar, *Microwave Engineering*, Wiley, 1998.
 2. C.A. Balanis, *Antenna Theory*, Wiley, 1997.
-

Model Library path: RF_Module/Antennas/vivaldi_antenna

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

- 4 Click **Add Selected**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 7 Click **Finish**.

GEOMETRY I

- 1 In the **Model Builder** window, under **Model I** click **Geometry I**.
- 2 In the **Geometry** settings window, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Description
thickness	60[mil]	Substrate thickness
w_slot	0.5[mm]	Slot width
f_min	2.0[GHz]	Minimum frequency in sweep
f_max	6.5[GHz]	Maximum frequency in sweep
f0	f_max	Current frequency in sweep
lda0	c_const/f0	Current wavelength, air
h_max	0.2*lda0	Maximum element size, air

Here, 'mil' refers to the unit milliinch and **c_const** is a predefined COMSOL constant for the speed of light in vacuum.

GEOMETRY I

Create a block for the antenna substrate.

Block I

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type 110.
- 4 In the **Depth** edit field, type 80.

- 5 In the **Height** edit field, type **thickness**.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 Right-click **Model 1>Geometry 1>Block 1** and choose **Rename**.
- 8 Go to the **Rename Block** dialog box and type **Substrate** in the **New name** edit field.
- 9 Click **OK**.

Next, add a block for the 50 ohm microstrip feed line.

Block 2

- I Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type **3.2**.
- 4 In the **Depth** edit field, type **40+w_slot/2**.
- 5 In the **Height** edit field, type **thickness**.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** edit field, type **-26**.
- 8 In the **y** edit field, type **-20+w_slot/4**.
- 9 Right-click **Model 1>Geometry 1>Block 2** and choose **Rename**.
- 10 Go to the **Rename Block** dialog box and type **Feed line** in the **New name** edit field.
- II Click **OK**.

Next, create a work plane where you will draw the Vivaldi antenna pattern. Use two parametric curves for the tapered slot.

Work Plane 1

- I Right-click **Geometry 1** and choose **Work Plane**.
- 2 In the **Work Plane** settings window, locate the **Plane Definition** section.
- 3 In the **z-coordinate** edit field, type **thickness/2**.
- 4 Click the **Zoom Extents** button on the Graphics toolbar.

Add a parametric curve using the exponential profile.

Parametric Curve 1

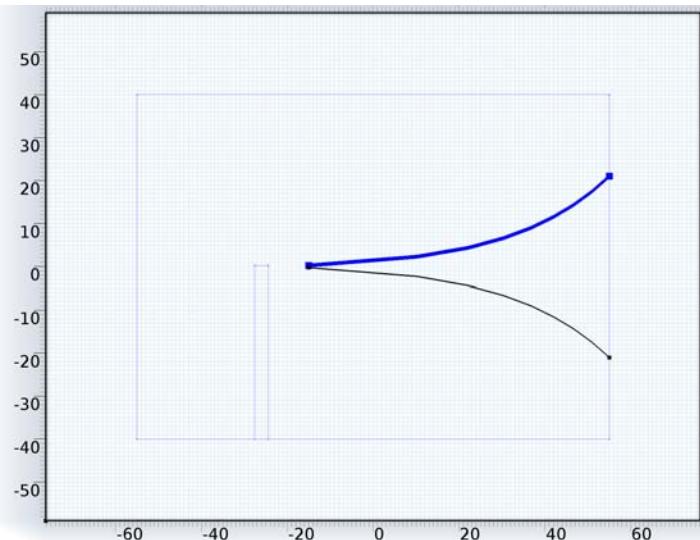
- I In the **Model Builder** window, right-click **Plane Geometry** and choose **Parametric Curve**.
- 2 In the **Parametric Curve** settings window, locate the **Parameter** section.
- 3 In the **Maximum** edit field, type **70**.
- 4 Locate the **Expressions** section. In the **xw** edit field, type **s - 15**.

- 5** In the **yw** edit field, type `exp(0.044*s)-1+w_slot/2.`

Generate the other parametric curve by mirroring the first one.

Mirror 1

- 1** In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Transforms>Mirror**.
- 2** In the **Mirror** settings window, locate the **Normal Vector to Line of Reflection** section.
- 3** In the **yw** edit field, type `1.`
- 4** In the **xw** edit field, type `0.`
- 5** Locate the **Input** section. Select the **Keep input objects** check box.
- 6** Select the object **pcl** only.
- 7** Click the **Build Selected** button.



Add a rectangle describing the thin slot connected to the tapered slot.

Rectangle 1

- 1** Right-click **Plane Geometry** and choose **Rectangle**.
- 2** In the **Rectangle** settings window, locate the **Size** section.
- 3** In the **Width** edit field, type `20.`
- 4** In the **Height** edit field, type `w_slot.`
- 5** Locate the **Position** section. In the **xw** edit field, type `-35.`

- 6** In the **yw** edit field, type $-w_{\text{slot}}/2$.

Add a circle attached to the end of the slot.

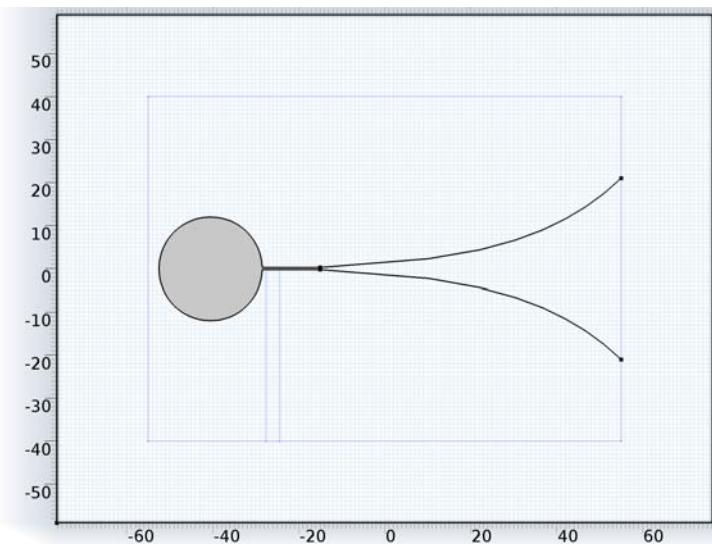
Circle 1

- 1** Right-click **Plane Geometry** and choose **Circle**.
- 2** In the **Circle** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type 12.
- 4** Locate the **Position** section. In the **xw** edit field, type -40.5 .

Create a union of the circle and the rectangle to remove unnecessary boundaries.

Union 1

- 1** Right-click **Plane Geometry** and choose **Boolean Operations>Union**.
- 2** Select the objects **c1** and **r1** only.
- 3** In the **Union** settings window, locate the **Union** section.
- 4** Clear the **Keep interior boundaries** check box.
- 5** Click the **Build All** button.



Add a sphere for the PMLs. Use a layer definition to create a shell-type structure.

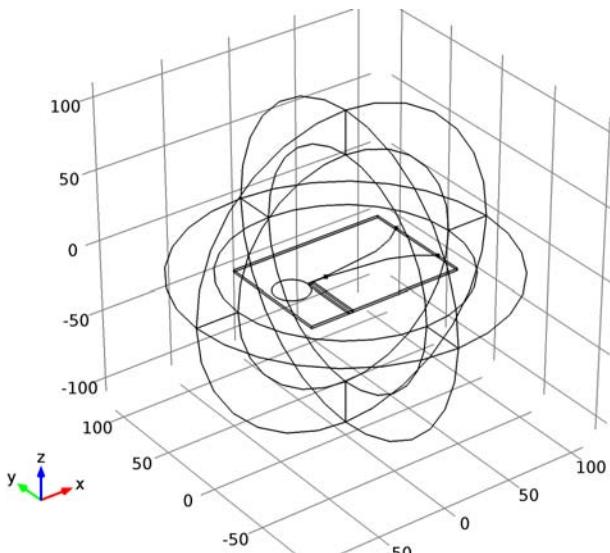
Sphere 1

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

- 2 In the **Sphere** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit 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 Right-click **Model 1>Geometry 1>Sphere 1** and choose **Rename**.
- 6 Go to the **Rename Sphere** dialog box and type PML in the **New name** edit field.
- 7 Click **OK**.
- 8 Click the **Build All** button.
- 9 Click the **Zoom Extents** button on the Graphics toolbar.
Choose wireframe rendering to get a better view of the interior parts.
- 10 Click the **Wireframe Rendering** button on the Graphics toolbar.

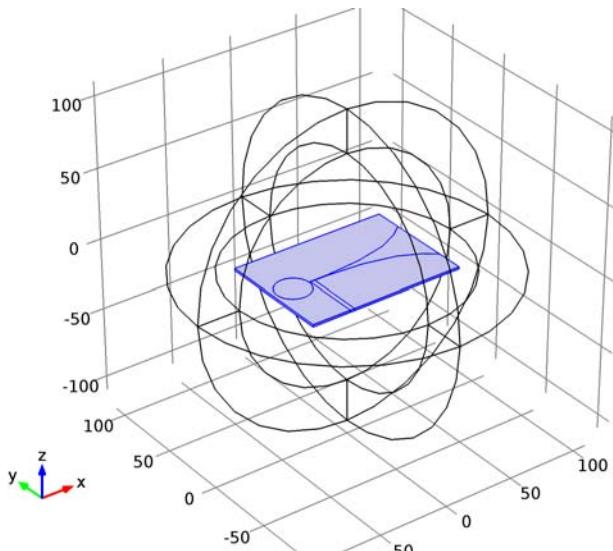


DEFINITIONS

Create a set of selections for use before setting up the physics. First, create a selection for the substrate.

Explicit 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
- 2 Select Domains 6 and 7 only.



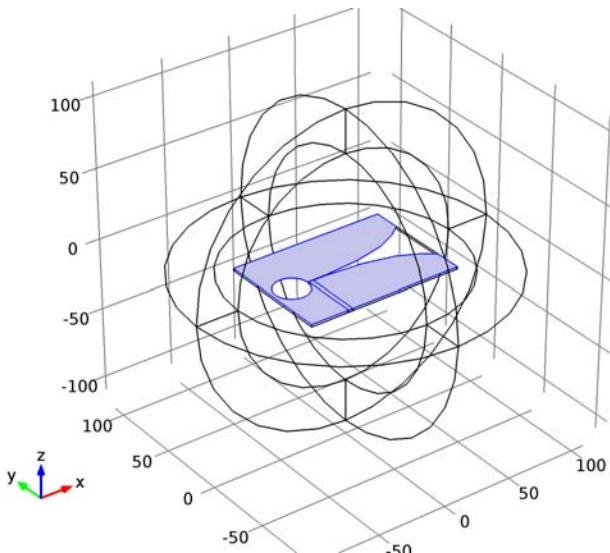
- 3 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.
- 4 Go to the **Rename Explicit** dialog box and type **Substrate** in the **New name** edit field.
- 5 Click **OK**.

Then, add a selection for the ground plane.

Explicit 2

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundaries 16, 22, and 27 only.



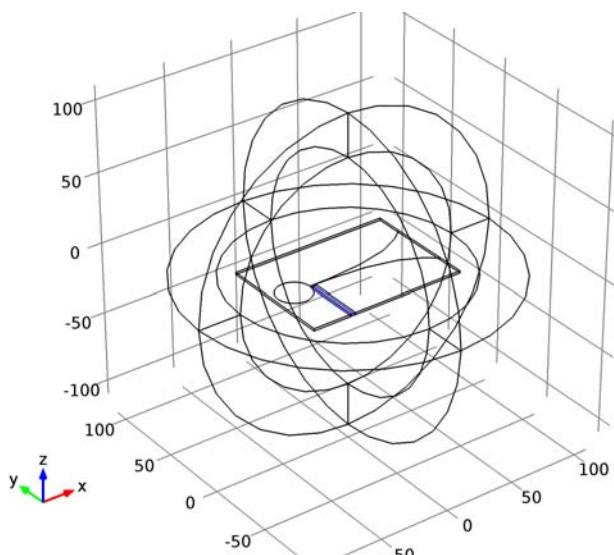
- 5 Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.
6 Go to the **Rename Explicit** dialog box and type **Ground** in the **New name** edit field.
7 Click **OK**.

Add a selection for the shorted microstrip line.

Explicit 3

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundaries 21 and 24 only.



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

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

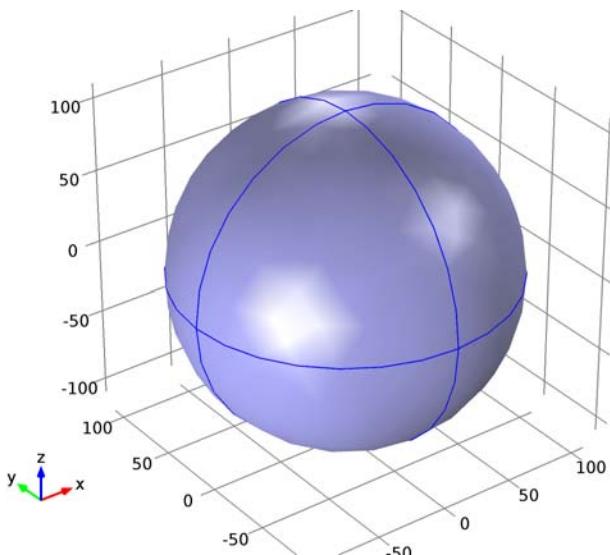
- 7 Click **OK**.

Add a selection for the PML.

Explicit 4

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

2 Select Domains 1–4 and 8–11 only.



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

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

5 Click **OK**.

Add a selection for the far-field domain.

Complement 1

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

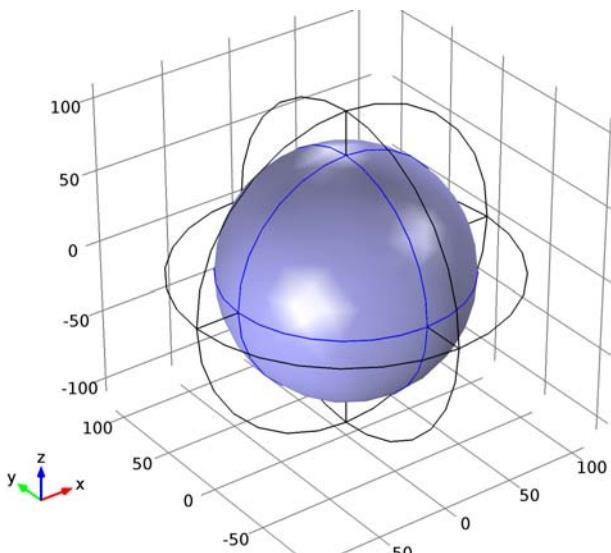
2 In the **Complement** settings window, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 Go to the **Add** dialog box.

5 In the **Selections to invert** list, select **PML**.

- 6** Click the **OK** button.



- 7** Right-click **Model 1>Definitions>Complement 1** and choose **Rename**.

- 8** Go to the **Rename Complement** dialog box and type **Far-field** in the **New name** edit field.
- 9** Click **OK**.

Next, add a selection for the far-field calculation boundaries. These are the outermost boundaries of the far-field domain.

Explicit 5

- 1** Right-click **Definitions** and choose **Selections>Explicit**.
- 2** In the **Explicit** settings window, locate the **Input Entities** section.
- 3** From the **Geometric entity level** list, choose **Boundary**.
- 4** Select Boundaries 9–12, 35, 36, 40, and 43 only.
- 5** Right-click **Model 1>Definitions>Explicit 5** and choose **Rename**.
- 6** Go to the **Rename Explicit** dialog box and type **Far-field calculation** in the **New name** edit field.
- 7** Click **OK**.

Next, add a selection for the scattering boundaries. These are the outermost boundaries of the PML.

Explicit 6

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 5–8, 33, 34, 39, and 44 only.
- 5 Right-click **Model 1>Definitions>Explicit 6** and choose **Rename**.
- 6 Go to the **Rename Explicit** dialog box and type Scattering boundary in the **New name** edit field.
- 7 Click **OK**.

Add a perfectly matched layer.

Perfectly Matched Layer 1

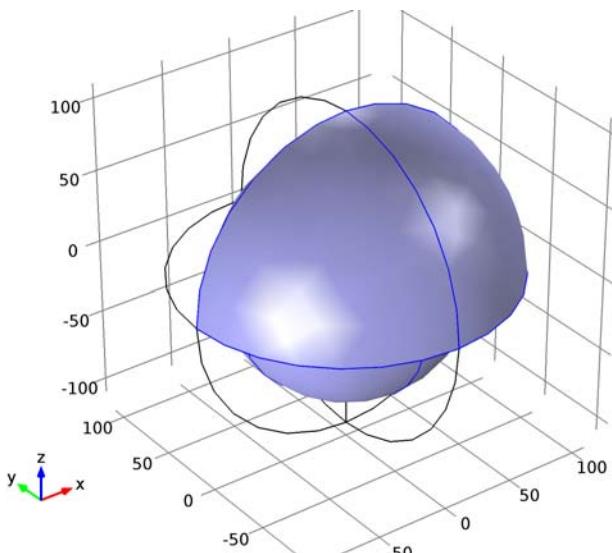
- 1 Right-click **Definitions** and choose **Perfectly Matched Layer**.
- 2 In the **Perfectly Matched Layer** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **PML**.
- 4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

View 1

Hide some domains to get a better view of the interior parts when setting up the physics and reviewing the mesh.

- 1 In the **Model Builder** window, under **Model 1>Definitions** right-click **View 1** and choose **Hide Geometric Entities**.

- 2** Select Domains 2, 5, and 9 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

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 **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.
- 2** In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.
- 3** From the **Selection** list, choose **Ground**.

Perfect Electric Conductor 3

- 1** In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.
- 2** In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.
- 3** From the **Selection** list, choose **Feed line**.

Scattering Boundary Condition 1

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

- 2 In the **Scattering Boundary Condition** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Scattering boundary**.

Far-Field Domain I

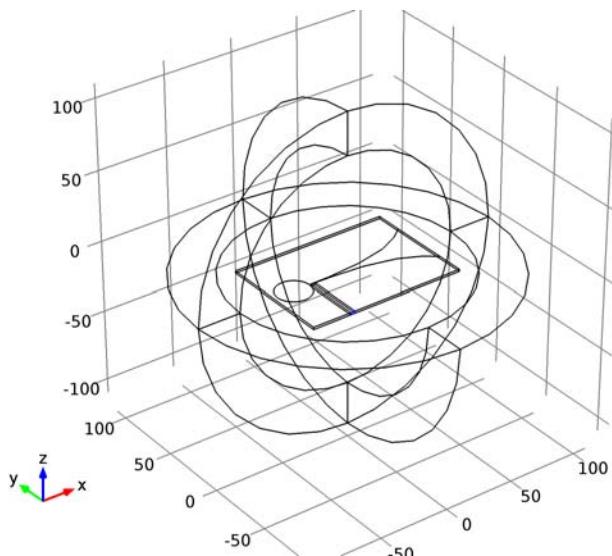
- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Far-Field Domain**.
- 2 In the **Far-Field Domain** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Far-field**.

Far-Field Calculation I

- 1 In the **Model Builder** window, expand the **Far-Field Domain I** node, then click **Far-Field Calculation I**.
- 2 In the **Far-Field Calculation** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Far-field calculation**.

Lumped Port I

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 Select Boundary 20 only.



- 3 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4 From the **Wave excitation at this port** list, choose **On**.

MATERIALS

Assign material properties for the model. First, use air for all domains.

Material Browser

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

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

Material 2

- 1 In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Substrate**.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

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

- 5 Right-click **Model 1>Materials>Material 2** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type **Substrate** in the **New name** edit field.
- 7 Click **OK**.

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 substrate, 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 **Model 1** right-click **Mesh 1** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Far-field**.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

- 7 In the associated edit field, type `h_max`.
- 8 Select the **Maximum element growth rate** check box.
- 9 In the associated edit field, type 2.

Size 2

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

Free Tetrahedral 1

- 1 Right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Far-field**.

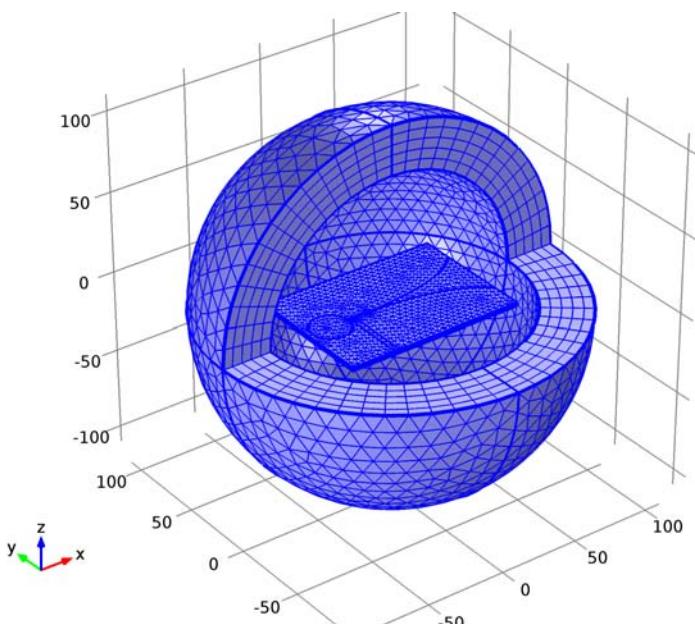
Swept 1

Right-click **Mesh 1** and choose **Swept**.

Distribution 1

- 1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

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



STUDY 1

Parametric Sweep

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.
- 2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter names	Parameter value list
f0	range(f_min,0.5[GHz],f_max)

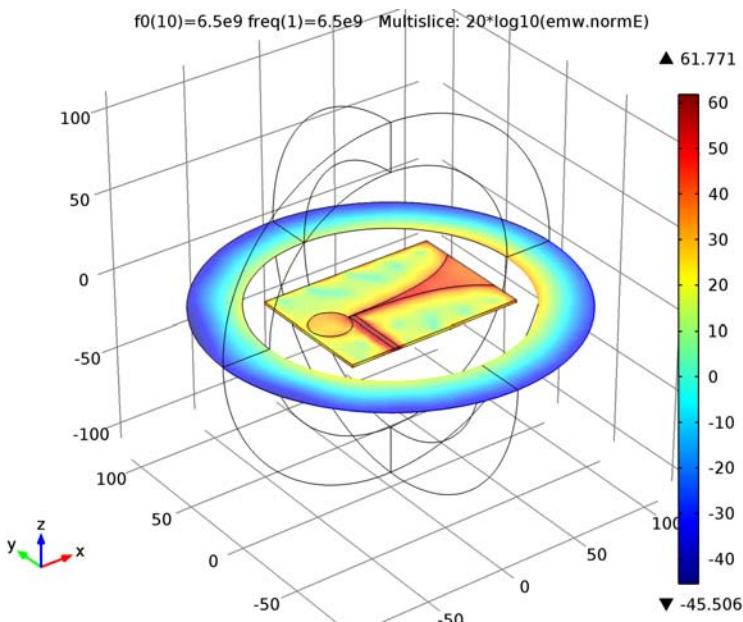
Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type f0.
- 4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

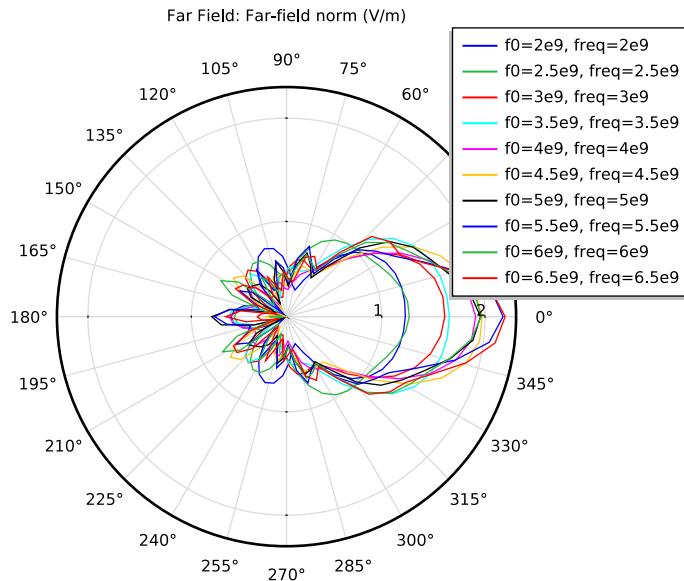
Electric Field (emw)

- 1 In the **Model Builder** window, under **Results>Electric Field (emw)** click **Multislice**.
- 2 In the **Multislice** settings window, locate the **Expression** section.
- 3 In the **Expression** edit field, type $20*\log10(\text{emw.normE})$.
- 4 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** edit field, type 0.
- 5 Find the **y-planes** subsection. In the **Planes** edit field, type 0.
- 6 Click the **Plot** button.



Strong electric fields are observed in the slot and microstrip line.

Polar Plot Group 2



*2D far-field radiation patterns in the *xy*-plane plotted for all frequencies.*

3D Plot Group 3

- 1 In the **Model Builder** window, under **Results** click **3D Plot Group 3**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Parameter value (f0)** list, choose **5.5e9**.
- 4 Click the **Plot** button.
- 5 Click the **Zoom Extents** button on the Graphics toolbar.

Compare the resulting 3D radiation pattern plot with [Figure 3](#).

ID Plot Group 4

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Solution 2**.
- 4 Right-click **Results>ID Plot Group 4** and choose **Global**.
- 5 In the **Global** settings window, locate the **x-Axis Data** section.
- 6 From the **Axis source data** list, choose **Outer solutions**.
- 7 Click to expand the **Legends** section. Clear the **Show legends** check box.

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

Expression	Description
$(1+\text{abs}(\text{emw.S11}))/\text{(1}-\text{abs}(\text{emw.S11}))$	SWR

- 9 Click the **Plot** button.

This SWR 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 transitioning 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 TE₁₀ mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes can be achieved analytically from the relation

$$(v_c)_{mn} = \frac{c}{2} \sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers, and c is the speed of light. For the TE₁₀ mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 2.286$ cm and $b = 1.016$ cm), the TE₁₀ mode is the only propagating mode for frequencies between 6.6 GHz and 14.7 GHz.

Although the shape of the TE₁₀ mode is known analytically, this model 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 **E** inside the waveguide adapter:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \left(\epsilon_r - \frac{j\sigma}{\omega \epsilon_0} \right) \mathbf{E} = 0$$

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 $\mu_r = \epsilon_r = 1$.

Results

Figure 2 shows a single-mode wave propagating through the waveguide.

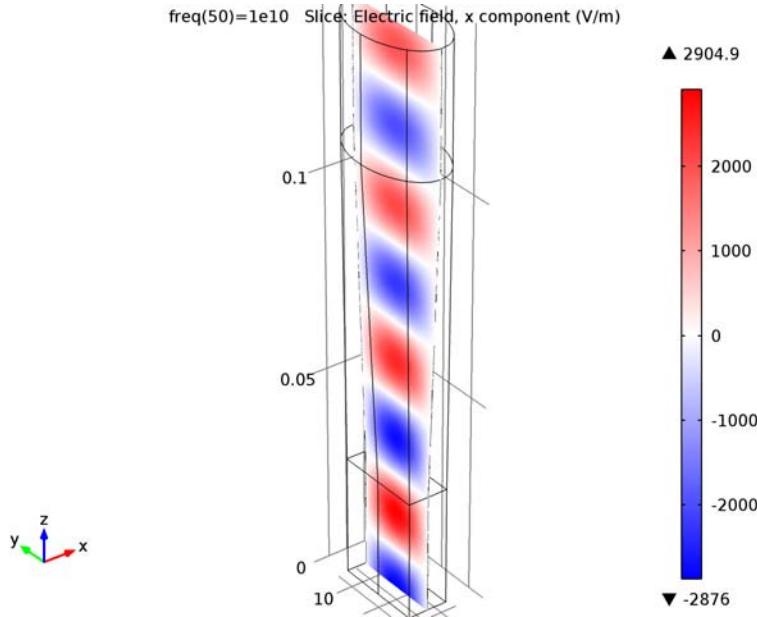


Figure 2: The x component of the propagating wave inside the waveguide adapter at the frequency 10 GHz.

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:

$$S_{11} = \frac{\int_{\text{Port 1}} ((E_c - E_1) \cdot E_1^*) dA_1}{\int_{\text{Port 1}} (E_1 \cdot E_1^*) dA_1}$$

$$S_{21} = \frac{\int_{\text{Port 2}} (E_c \cdot E_2^*) dA_2}{\int_{\text{Port 2}} (E_2 \cdot E_2^*) dA_2}$$

Here E_c is the calculated total field. E_1 is the analytical field for the port excitation, and E_2 is the eigenmode calculated from the boundary mode analysis and normalized

with respect to the outgoing power flow. [Figure 3](#) and [Figure 4](#) show the S_{11} and S_{21} parameters as functions of the frequency.

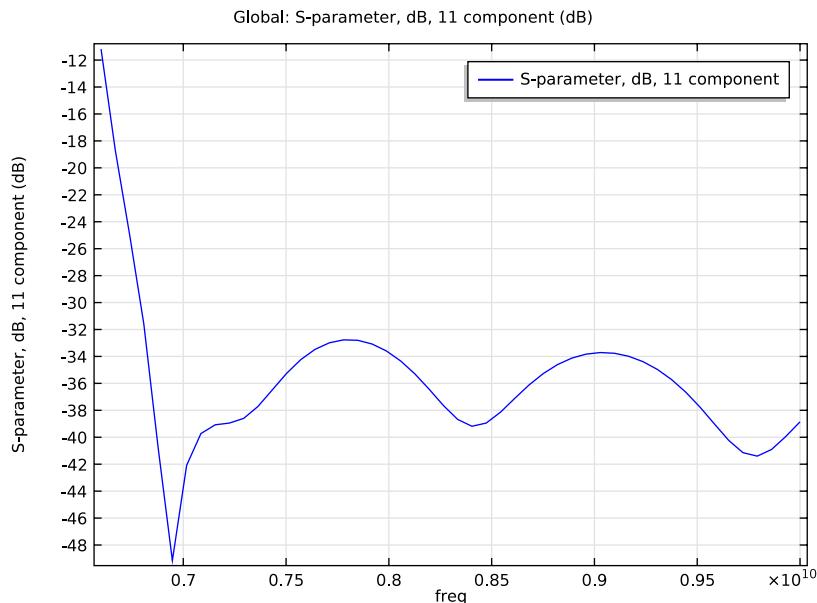


Figure 3: The S_{11} parameter (in dB) as a function of the frequency. This parameter describes the reflections when the waveguide adapter is excited at the rectangular port.

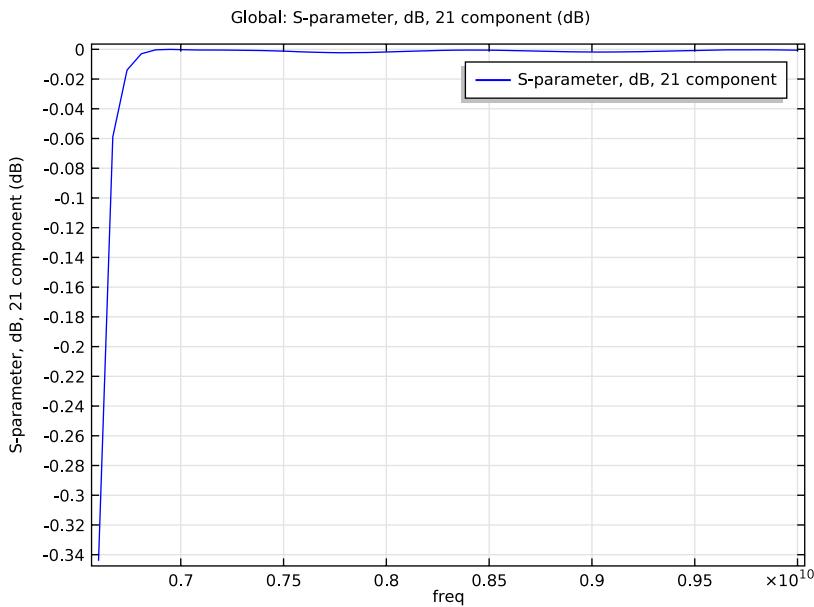


Figure 4: The S_{21} parameter (in dB) as a function of the frequency. This parameter is a measure of the part of the wave that is transmitted through the elliptical port when the waveguide adapter is excited at the rectangular port.

Model Library path: RF_Module/Transmission_Lines_and_Waveguides/waveguide_adapter

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Custom Studies>Empty Study**.

6 Click **Finish**.

STUDY 1

Step 1: Boundary Mode Analysis

- 1 In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Boundary Mode Analysis**.
- 2 In the **Model Builder** window, click **Step 1: Boundary Mode Analysis**.
- 3 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 4 In the **Mode analysis frequency** edit 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 In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Boundary Mode Analysis**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Port name** edit field, type **2**.
- 4 In the **Mode analysis frequency** edit field, type **7[GHz]**.

Finally, add the 3D equation for the propagating wave in the waveguide.

Step 3: Frequency Domain

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

Proceed to import the geometry.

GEOMETRY 1

Import 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.
- 2 In the **Import** settings window, locate the **Import** section.
- 3 Click the **Browse** button.
- 4 Browse to the model's Model Library folder and double-click the file **waveguide_adapter.mphbin**.

- 5 Click the **Import** button.

MATERIALS

Material Browser

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

Air

By default, the first material you add applies on all domains so you need not alter any settings.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Port 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Port**.
- 2 In the **Port** settings window, locate the **Port Properties** section.
- 3 From the **Type of port** list, choose **Numeric**.
- 4 From the **Wave excitation at this port** list, choose **On**.
- 5 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** and choose **Port**.
- 2 In the **Port** settings window, locate the **Port Properties** section.
- 3 From the **Type of port** list, choose **Numeric**.
- 4 In the **Port name** edit field, type 2.
- 5 Select Boundary 6 only.

This is the exit port, the one with an elliptical cross-section.

MESH I**Size**

- 1 In the **Model Builder** window, under **Model I** right-click **Mesh I** and choose **Free Tetrahedral**.
- 2 In the **Size** settings window, locate the **Element Size Parameters** section.
- 3 In the **Maximum element size** edit field, type 0.006.
- 4 Click the **Build All** button.

STUDY I

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 I** node, then click **Step 1: Boundary Mode Analysis**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Search for modes around** edit 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.

- 4 From the **Transform** list, choose **Out-of-plane wave number**.

Step 2: Boundary Mode Analysis 2

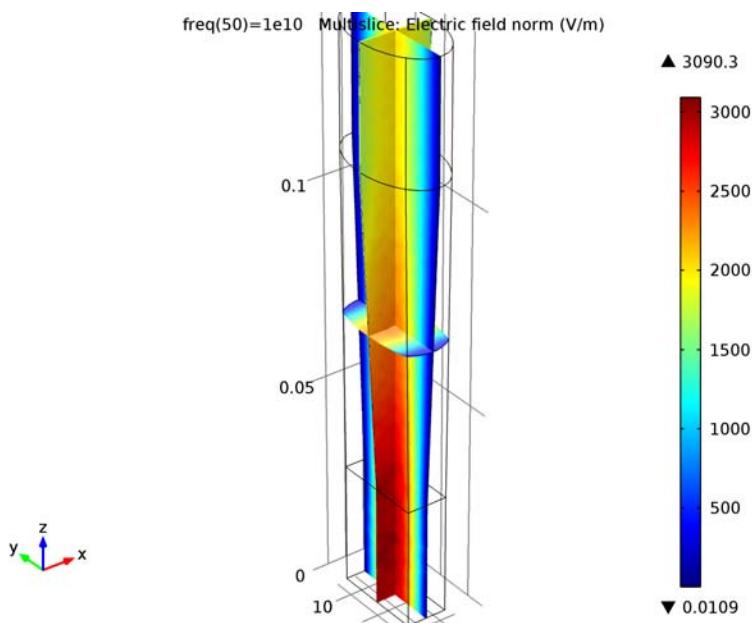
- 1 In the **Model Builder** window, under **Study I** click **Step 2: Boundary Mode Analysis 2**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Search for modes around** edit field, type 50.
- 4 From the **Transform** list, choose **Out-of-plane wave number**.

Step 3: Frequency Domain

- 1 In the **Model Builder** window, under **Study I** click **Step 3: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type range (6.6e9, 3.4e9/49, 1.0e10).
- 4 In the **Model Builder** window, right-click **Study I** and choose **Compute**.

RESULTS

Electric Field (emw)



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.

- 1 In the **Model Builder** window, expand the **Electric Field (emw)** node.
- 2 Right-click **Multislice** and choose **Delete**.
- 3 Click **Yes** to confirm.
- 4 Right-click **Electric Field (emw)** and choose **Slice**.
- 5 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field, x component (emw.Ex)**.
- 6 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.
- 7 Locate the **Plane Data** section. From the **Plane** list, choose **yz-planes**.
- 8 In the **Planes** edit field, type 1.

9 Click the **Plot** button.

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 Plot Group 3D (pg1).

Proceed to plot the S-parameters as functions of the frequency.

ID Plot Group 2

- 1** In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2** Right-click **ID Plot Group 2** and choose **Global**.
- 3** In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 11 component (emw.S11dB)**.
- 4** Click the **Plot** button.
This plot should closely resemble that in [Figure 3](#).
- 5** Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 21 component (emw.S21dB)**.
- 6** Click the **Plot** button.
This reproduces the plot in [Figure 4](#). Finally, add S_{11} to obtain both S-parameters in the same plot.
- 7** Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **S-parameter, dB, 11 component (emw.S11dB)**.
- 8** Click the **Plot** button.

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.

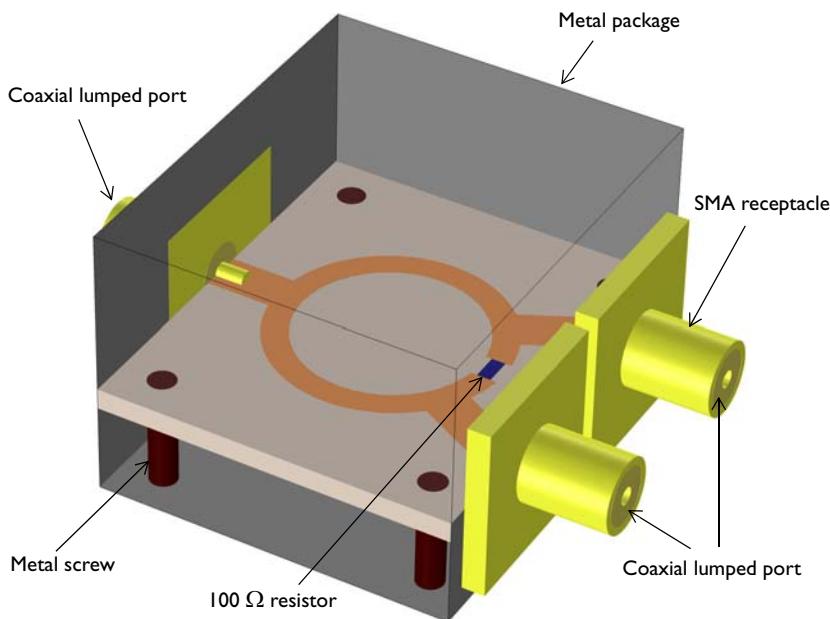


Figure 1: A Wilkinson power divider is fabricated on a 60 mil substrate. An SMA receptacle is added on each port and the circuit board is suspended in the metal package using screws.

Model Definition

The Wilkinson power divider is a three-port device composed of $50\ \Omega$ and $70.7\ \Omega$ microstrip lines on a dielectric substrate with a ground plane and a $100\ \Omega$ 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](#). Model all microstrip lines, 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 model analysis. 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 $100\ \Omega$ resistor is realized via a uniform lumped port with $100\ \Omega$ 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.

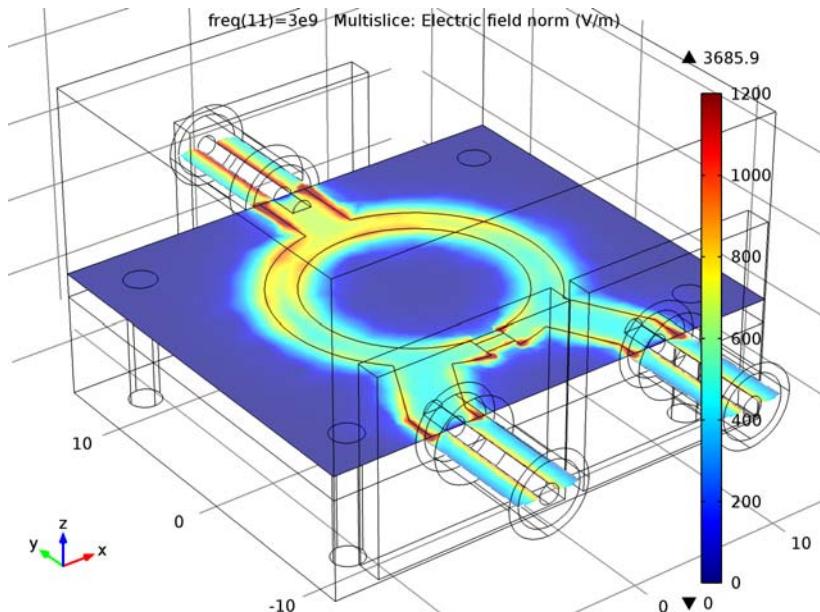


Figure 2: The E-field norm plot shows that the input is evenly split between the two output ports.

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.

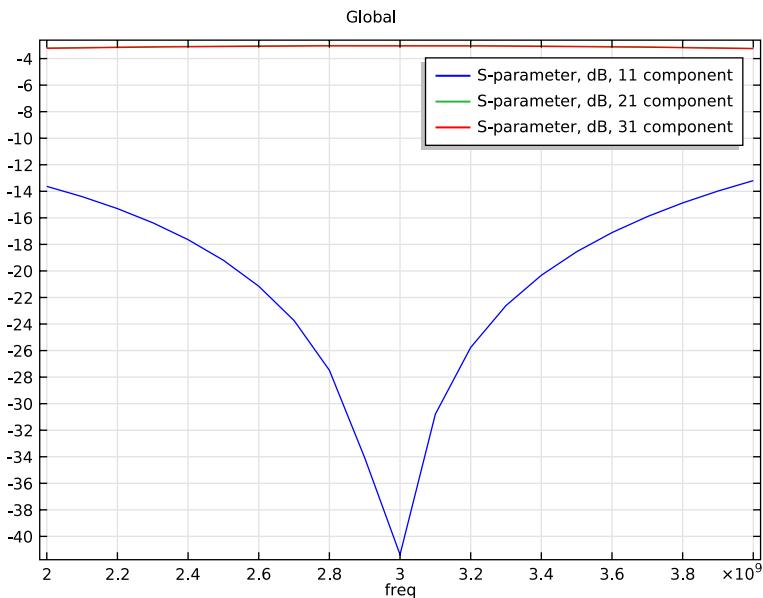


Figure 3: The S-parameters show very good input matching at 3 GHz and evenly divided power at the two output ports.

References

1. D.M. Pozar, *Microwave Engineering*, Wiley, 1998.
 2. R.E. Collin, *Foundation of Microwave Engineering*, McGraw-Hill, 1992.
-

Model Library path: RF_Module/Passive_Devices/wilkinson_power_divider

Modeling Instructions

MODEL WIZARD

- I Go to the **Model Wizard** window.

- 2** Click **Next**.
- 3** In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
- 4** Click **Next**.
- 5** Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
- 6** Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1** In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2** In the **Parameters** settings window, locate the **Parameters** section.
- 3** In the table, enter the following settings:

Name	Expression	Description
r_ring	8.5[mm]	Radius, microstrip line ring
w_subs	30[mm]	Width, substrate
l_subs	26[mm]	Length, susbstrate
r_inner	0.635[mm]	Radius, coax inner
r_outer	2.05[mm]	Radius, coax outer
l_sma	8[mm]	Length, SMA
f_min	2[GHz]	Minimum frequency in sweep
f_max	4[GHz]	Maximum frequency in sweep
lda0	c_const/f_max	Wavelength, air
h_max	0.2*lda0	Maximum mesh size, air

Here, c_{const} is a predefined COMSOL constant for the speed of light in vacuum.

GEOMETRY I

- 1** In the **Model Builder** window, under **Model I** click **Geometry I**.
- 2** In the **Geometry** settings window, locate the **Units** section.
- 3** From the **Length unit** list, choose **mm**.

First, create the substrate.

Block I

- 1** In the **Model Builder** window, right-click **Geometry I** and choose **Block**.
- 2** In the **Block** settings window, locate the **Size and Shape** section.

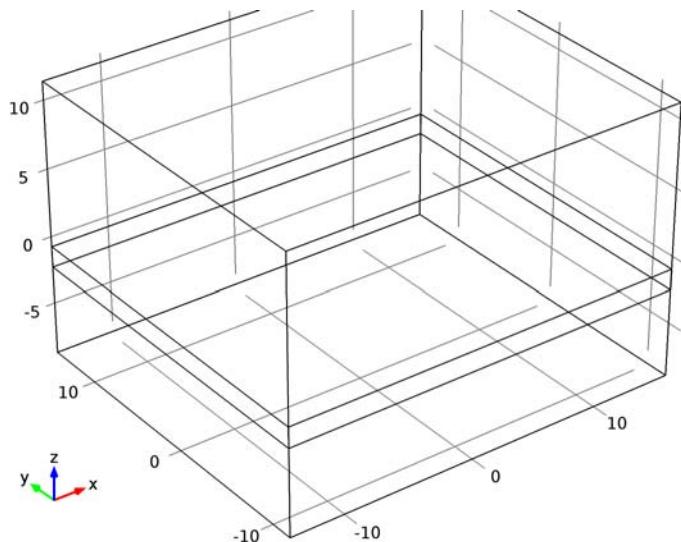
- 3 In the **Width** edit field, type `w_subs`.
- 4 In the **Depth** edit field, type `l_subs`.
- 5 In the **Height** edit field, type `1.524`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **z** edit field, type `-0.762`.
- 8 Right-click **Model 1>Geometry 1>Block 1** and choose **Rename**.
- 9 Go to the **Rename Block** dialog box and type **Substrate** in the **New name** edit field.
- 10 Click **OK**.

Add a block for the metal package.

Block 2

- 1 Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `w_subs`.
- 4 In the **Depth** edit field, type `l_subs`.
- 5 In the **Height** edit field, type `20`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **z** edit field, type `2`.
- 8 Click the **Build Selected** button.

- 9 Click the **Wireframe Rendering** button on the Graphics toolbar.



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

- 11 Go to the **Rename Block** dialog box and type **Package** in the **New name** edit field.

- 12 Click **OK**.

Add a work plane for drawing the layout of the power divider.

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

Plane Geometry

Add two circles to create the ring strip part.

Circle 1

- 1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Circle**.
- 2 Click the **Zoom Extents** button on the Graphics toolbar.
- 3 In the **Circle** settings window, locate the **Size and Shape** section.
- 4 In the **Radius** edit field, type **r_ring**.
- 5 Right-click **Model 1>Geometry 1>Work Plane 1>Plane Geometry>Circle 1** and choose **Rename**.
- 6 Go to the **Rename Circle** dialog box and type **Ring_outer** in the **New name** edit field.
- 7 Click **OK**.

Circle 2

- 1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Circle**.
- 2 In the **Circle** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type `r_ring-1.87`.
- 4 Right-click **Model 1>Geometry 1>Work Plane 1>Plane Geometry>Circle 2** and choose **Rename**.
- 5 Go to the **Rename Circle** dialog box and type `Ring_inner` in the **New name** edit field.
- 6 Click **OK**.

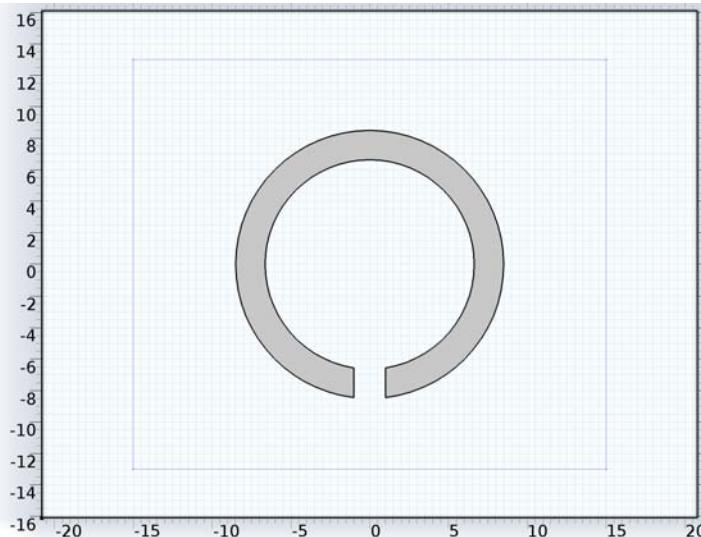
Rectangle 1

- 1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type `2`.
- 4 In the **Height** edit field, type `3`.
- 5 Locate the **Position** section. In the **xw** edit field, type `-1`.
- 6 In the **yw** edit field, type `-9`.
- 7 Right-click **Model 1>Geometry 1>Work Plane 1>Plane Geometry>Rectangle 1** and choose **Rename**.
- 8 Go to the **Rename Rectangle** dialog box and type `Ring_cut` in the **New name** edit field.
- 9 Click **OK**.

Difference 1

- 1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Boolean Operations>Difference**.
- 2 Select the object **c1** only.
- 3 In the **Difference** settings window, locate the **Difference** section.
- 4 Under **Objects to subtract**, click **Activate Selection**.
- 5 Select the objects **r1** and **c2** only.

- 6 Click the **Build Selected** button.



Add a rectangle for the 100 ohm resistor.

Rectangle 2

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 2.
- 4 Locate the **Position** section. In the **xw** edit field, type -1.
- 5 In the **yw** edit field, type -8.
- 6 Right-click **Model 1>Geometry 1>Work Plane 1>Plane Geometry>Rectangle 2** and choose **Rename**.
- 7 Go to the **Rename Rectangle** dialog box and type Lumped element in the **New name** edit field.
- 8 Click **OK**.

Add rectangles for the 50 ohm microstrip feed lines.

Rectangle 3

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

- 3 In the **Width** edit field, type 3.2.
- 4 In the **Height** edit field, type 5.
- 5 Locate the **Position** section. From the **Base** list, choose **Center**.
- 6 In the **yw** edit field, type 10.5.

Rectangle 4

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 3.2.
- 4 In the **Height** edit field, type 2.
- 5 Locate the **Position** section. In the **xw** edit field, type -7.
- 6 From the **Base** list, choose **Center**.
- 7 In the **yw** edit field, type -12.

Rectangle 5

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 3.2.
- 4 In the **Height** edit field, type 6.
- 5 Locate the **Position** section. In the **xw** edit field, type -8.6.
- 6 In the **yw** edit field, type -11.
- 7 Locate the **Rotation Angle** section. In the **Rotation** edit field, type -28.

Mirror 1

- 1 Right-click **Plane Geometry** and choose **Transforms>Mirror**.
- 2 Select the objects **r4** and **r5** only.
- 3 In the **Mirror** settings window, locate the **Input** section.
- 4 Select the **Keep input objects** check box.

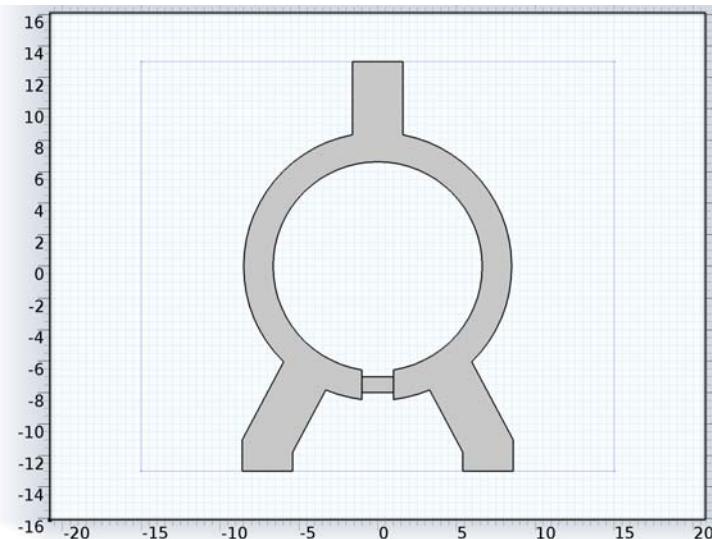
Create a union of all objects except the small rectangle for the resistor (r2) to remove unnecessary boundaries.

Union 1

- 1 Right-click **Plane Geometry** and choose **Boolean Operations>Union**.
- 2 Select the objects **r5**, **r4**, **r3**, **mir1(1)**, **dif1**, and **mir1(2)** only.
- 3 In the **Union** settings window, locate the **Union** section.

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

5 Click the **Build All** button.



The power divider layout drawn on the substrate.

Create the coaxial SMA receptacle composed of the coaxial inner and outer conductors, the SMA connector part, and the flange.

Cylinder 1

- 1** In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.
- 2** In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3** In the **Radius** edit field, type `r_inner`.
- 4** In the **Height** edit field, type `l_sma+2`.
- 5** Locate the **Position** section. In the **x** edit field, type `-7`.
- 6** In the **y** edit field, type `-l_subs/2-l_sma`.
- 7** Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.
- 8** Right-click **Model 1>Geometry 1>Cylinder 1** and choose **Rename**.
- 9** Go to the **Rename Cylinder** dialog box and type **Coax_inner** in the **New name** edit field.
- 10** Click **OK**.

Cylinder 2

- 1 Right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type `r_outer+0.6`.
- 4 In the **Height** edit field, type `l_sma`.
- 5 Locate the **Position** section. In the **x** edit field, type `-7`.
- 6 In the **y** edit field, type `-l_subs/2-l_sma`.
- 7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.
- 8 Right-click **Model 1>Geometry 1>Cylinder 2** and choose **Rename**.
- 9 Go to the **Rename Cylinder** dialog box and type **SMA** in the **New name** edit field.
- 10 Click **OK**.

Block 3

- 1 Right-click **Geometry 1** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type `12.7`.
- 4 In the **Depth** edit field, type `12.7`.
- 5 In the **Height** edit field, type `1.65`.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **x** edit field, type `-7`.
- 8 In the **y** edit field, type `-(l_subs+1.65)/2`.
- 9 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.
- 10 Right-click **Model 1>Geometry 1>Block 3** and choose **Rename**.
- 11 Go to the **Rename Block** dialog box and type **Flange** in the **New name** edit field.
- 12 Click **OK**.

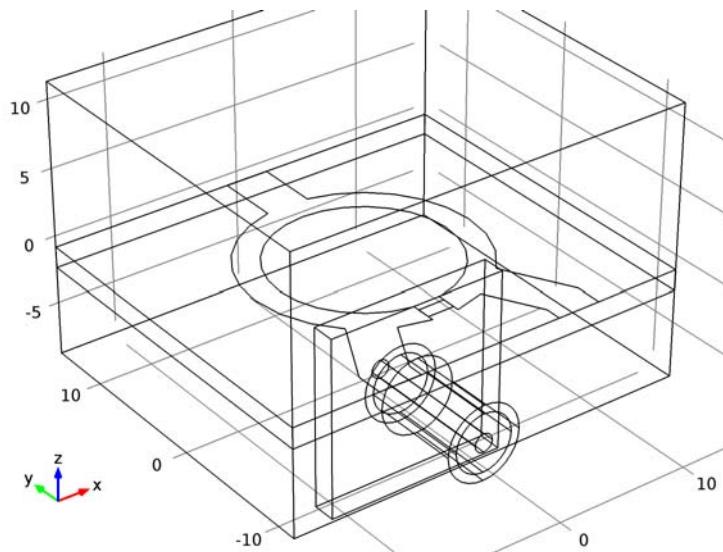
Create a union of a couple of objects, the SMA connector, and the flange to remove unnecessary boundaries.

Union 1

- 1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.
- 2 Select the objects **blk3** and **cyl2** only.
- 3 In the **Union** settings window, locate the **Union** section.
- 4 Clear the **Keep interior boundaries** check box.

Cylinder 3

- 1 Right-click **Geometry 1** and choose **Cylinder**.
- 2 In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type `r_outer`.
- 4 In the **Height** edit field, type `l_sma`.
- 5 Locate the **Position** section. In the **x** edit field, type `-7`.
- 6 In the **y** edit field, type `-1_subs/2-l_sma`.
- 7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.
- 8 Click the **Build Selected** button.



- 9 Right-click **Model 1>Geometry 1>Cylinder 3** and choose **Rename**.
- 10 Go to the **Rename Cylinder** dialog box and type **Coax dielectric** in the **New name** edit field.
- II Click **OK**.

Create two more SMA receptacles.

Copy 1

- 1 Right-click **Geometry 1** and choose **Transforms>Copy**.
- 2 Select the objects **uni1**, **cyl3**, and **cyl1** only.
- 3 In the **Copy** settings window, locate the **Displacement** section.

- 4** In the **x** edit field, type 7,14.

Rotate 1

- 1** Right-click **Geometry 1** and choose **Transforms>Rotate**.
- 2** Select the objects **copy1(1)**, **copy1(3)**, and **copy1(5)** only.
- 3** In the **Rotate** settings window, locate the **Rotation Angle** section.
- 4** In the **Rotation** edit field, type 180.

Add a cylinder for the metal screw.

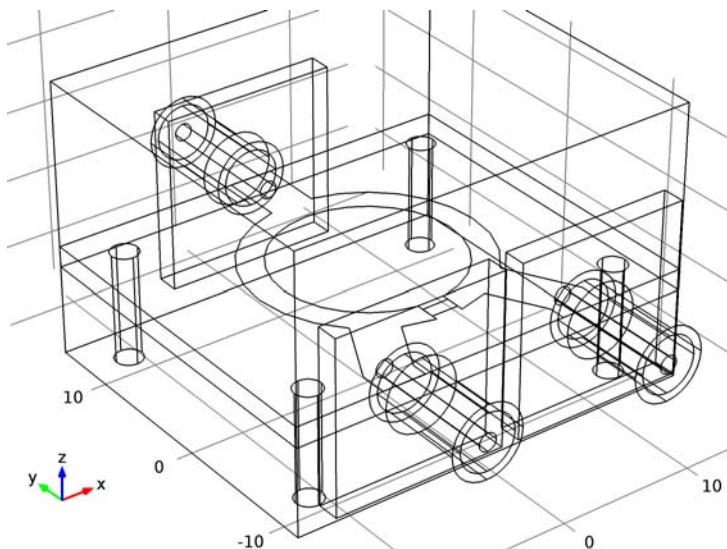
Cylinder 4

- 1** Right-click **Geometry 1** and choose **Cylinder**.
- 2** In the **Cylinder** settings window, locate the **Size and Shape** section.
- 3** In the **Height** edit field, type 8.
- 4** Locate the **Position** section. In the **x** edit field, type -12.
- 5** In the **y** edit field, type -10.
- 6** In the **z** edit field, type -8.
- 7** Right-click **Model 1>Geometry 1>Cylinder 4** and choose **Rename**.
- 8** Go to the **Rename Cylinder** dialog box and type **Screw** in the **New name** edit field.
- 9** Click **OK**.

Array 1

- 1** Right-click **Geometry 1** and choose **Transforms>Array**.
- 2** Select the object **cyl4** only.
- 3** In the **Array** settings window, locate the **Size** section.
- 4** In the **x size** edit field, type 2.
- 5** In the **y size** edit field, type 2.
- 6** Locate the **Displacement** section. In the **x** edit field, type 24.
- 7** In the **y** edit field, type 20.

- 8 Click the **Build All** button.



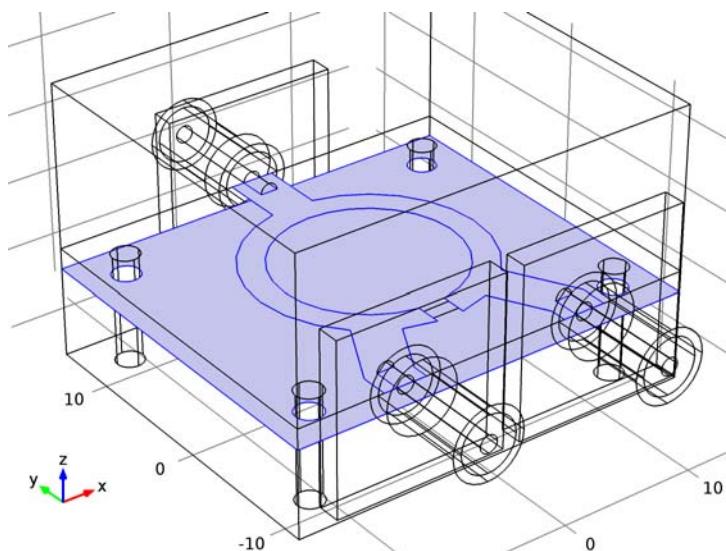
DEFINITIONS

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 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
- 2 In the **Explicit** settings window, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundaries 6, 54, 61, 102, and 147 only.



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

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

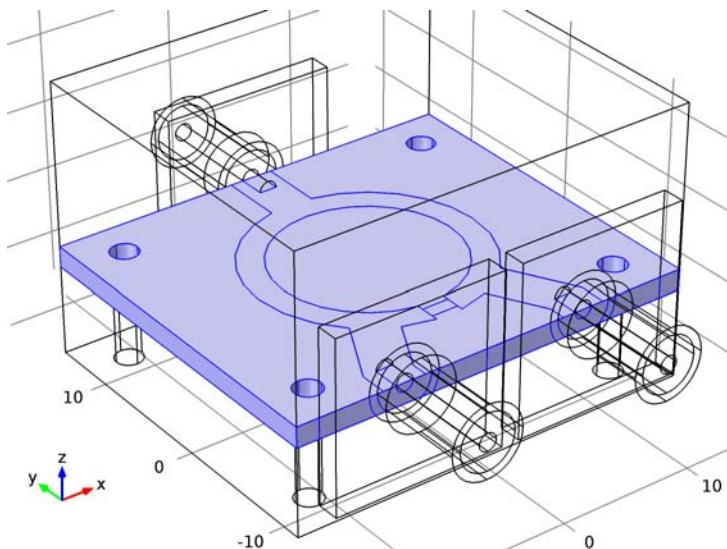
- 7 Click **OK**.

Add a selection for the substrate.

Explicit 2

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

- 2** Select Domains 2, 11, 15, and 21 only.



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

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

- 5** Click **OK**.

Add a selection for the coax dielectric (PTFE).

Explicit 3

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

- 2** Select Domains 9, 14, and 19 only.

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

- 4** Go to the **Rename Explicit** dialog box and type **Coax dielectric** in the **New name** edit field.

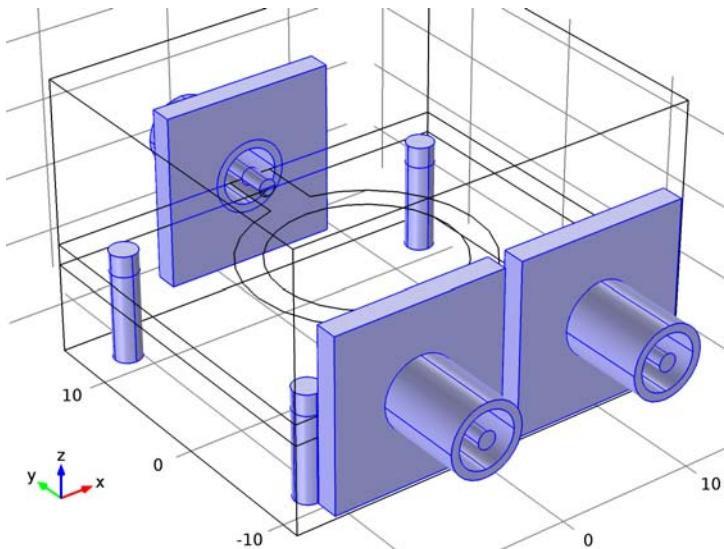
- 5** Click **OK**.

Add a selection for the domains consisting of metal. These domains are not part of the model analysis.

Explicit 4

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

- 2 Select Domains 4–8, 10, 12, 13, 16–18, 20, and 22–26 only.



- 3 Right-click **Model 1>Definitions>Explicit 4** and choose **Rename**.
4 Go to the **Rename Explicit** dialog box and type **Metal volume** in the **New name** edit field.
5 Click **OK**.

Define the model domain, which is the complement of the metal volume selection.

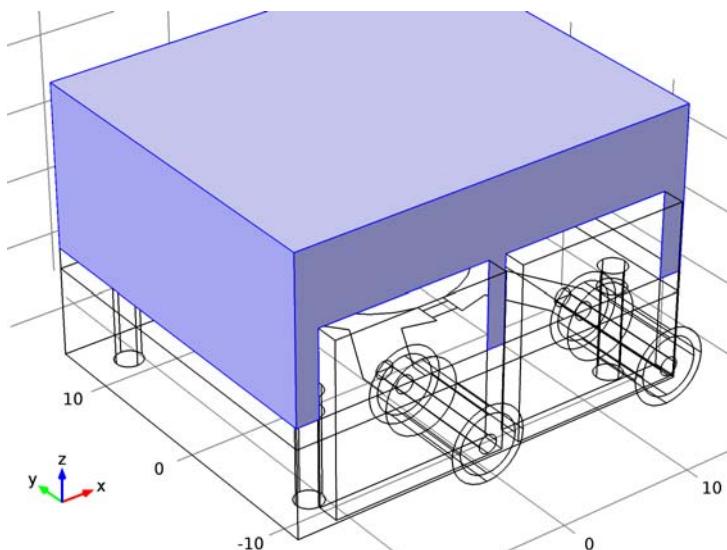
Complement

- 1 Right-click **Definitions** and choose **Selections>Complement**.
- 2 In the **Complement** settings window, locate the **Input Entities** section.
- 3 Under **Selections to invert**, click **Add**.
- 4 Go to the **Add** dialog box.
- 5 In the **Selections to invert** list, select **Metal volume**.
- 6 Click the **OK** button.
- 7 Right-click **Model 1>Definitions>Complement 1** and choose **Rename**.
- 8 Go to the **Rename Complement** dialog box and type **Model domain** in the **New name** edit field.
- 9 Click **OK**.

View I

Suppress some boundaries to get a view of the interior while setting the physics and mesh.

- 1 In the **Model Builder** window, under **Model I >Definitions** right-click **View I** and choose **Hide Geometric Entities**.
- 2 In the **Hide Geometric Entities** settings window, 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

- 1 In the **Model Builder** window, under **Model I** click **Electromagnetic Waves, Frequency Domain**.
- 2 In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Model domain**.

Perfect Electric Conductor I

The **Perfect Electric Conductor** applies by default to all exterior boundaries. After restricting the **Electromagnetic Waves, Frequency Domain** interface to the model

domain, these outer boundaries include the coaxial SMA receptacles and the metal screws. Add a **Perfect Electric Conductor** condition to the microstrip line and the substrate ground plane.

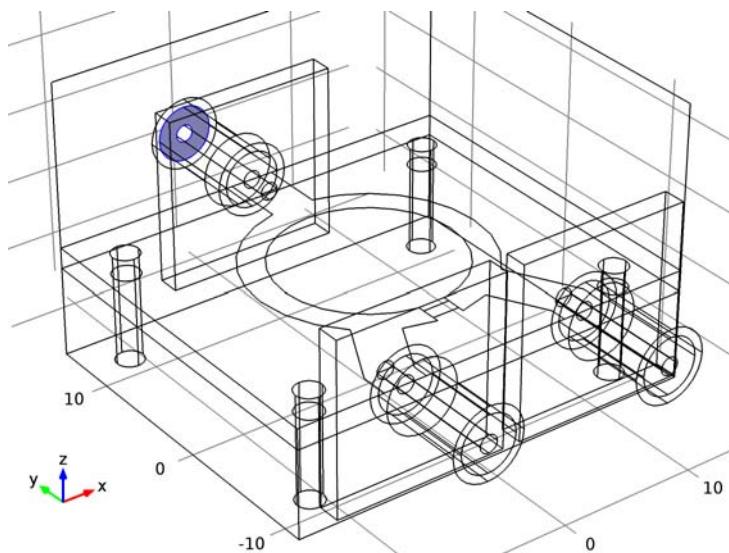
Perfect Electric Conductor 2

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.
- 2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Microstrip line**.

Proceed with the Lumped Port conditions.

Lumped Port 1

- 1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.
- 2 Select Boundary 95 only.

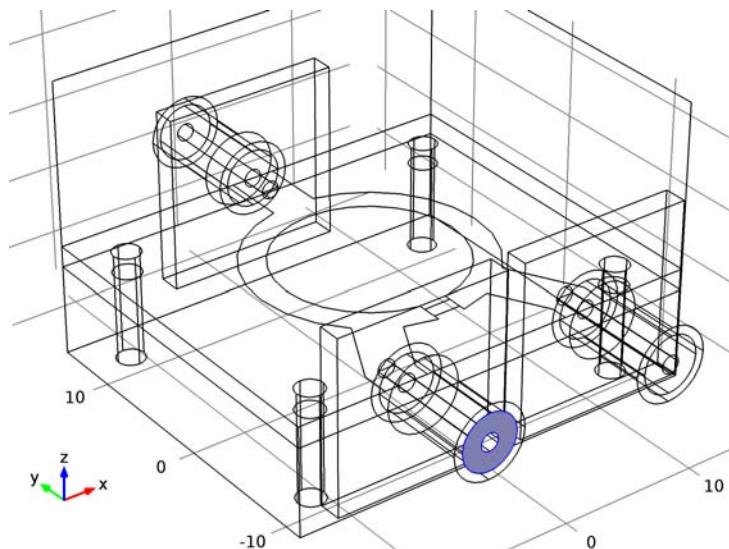


- 3 In the **Lumped Port** settings window, locate the **Port Properties** section.
- 4 From the **Type of port** list, choose **Coaxial**.
- 5 From the **Wave excitation at this port** list, choose **On**.

Lumped Port 2

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

2 Select Boundary 51 only.



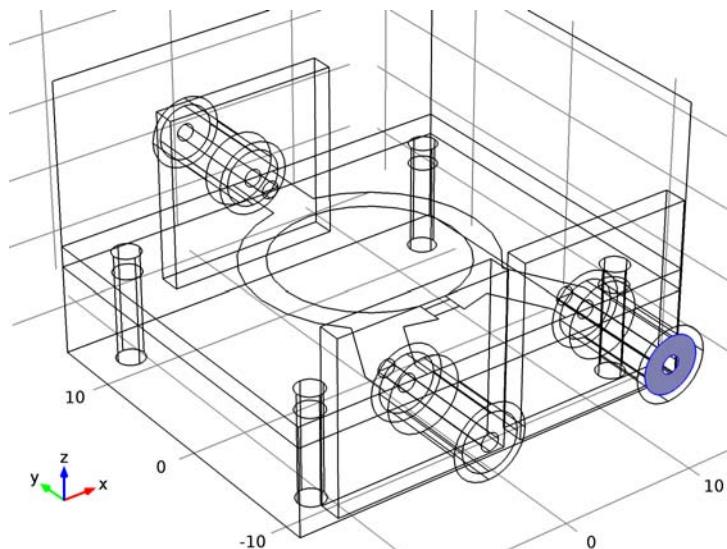
3 In the **Lumped Port** settings window, locate the **Port Properties** section.

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

Lumped Port 3

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

2 Select Boundary 136 only.



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

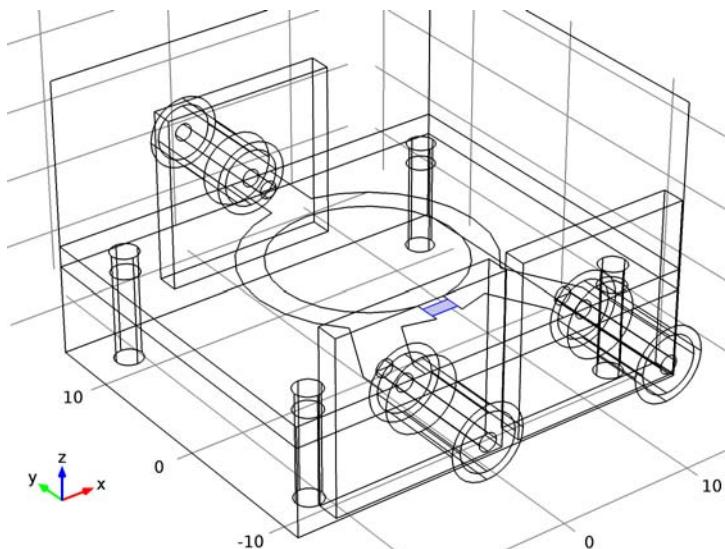
4 From the **Type of port** list, choose **Coaxial**.

Add a lumped element for the 100 ohm resistor.

Lumped Element 1

I Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Element**.

- 2** Select Boundary 97 only.



- 3** In the **Lumped Element** settings window, locate the **Settings** section.

- 4** In the Z_{element} edit field, type $100[\text{ohm}]$.

MATERIALS

Next, assign material properties. First, specify air for all domains.

Material Browser

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

Override the material for the substrate domains with a dielectric material of $\epsilon_r = 3.38$.

Material 2

- 1** In the **Model Builder** window, right-click **Materials** and choose **Material**.
- 2** In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3** From the **Selection** list, choose **Substrate**.

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

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

- 5** Right-click **Model 1>Materials>Material 2** and choose **Rename**.

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

- 7** Click **OK**.

Similarly, override the coax dielectric domains with a material of $\epsilon_r = 2.1$.

Material 3

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

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

- 3** From the **Selection** list, choose **Coax dielectric**.

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

Property	Name	Value
Relative permittivity	epsilon_r	2.1
Relative permeability	mur	1
Electrical conductivity	sigma	0

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

- 6** Go to the **Rename Material** dialog box and type **PTFE** in the **New name** edit field.

- 7** Click **OK**.

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 **Model 1** right-click **Mesh 1** and choose **Size**.

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

- 3** From the **Geometric entity level** list, choose **Domain**.

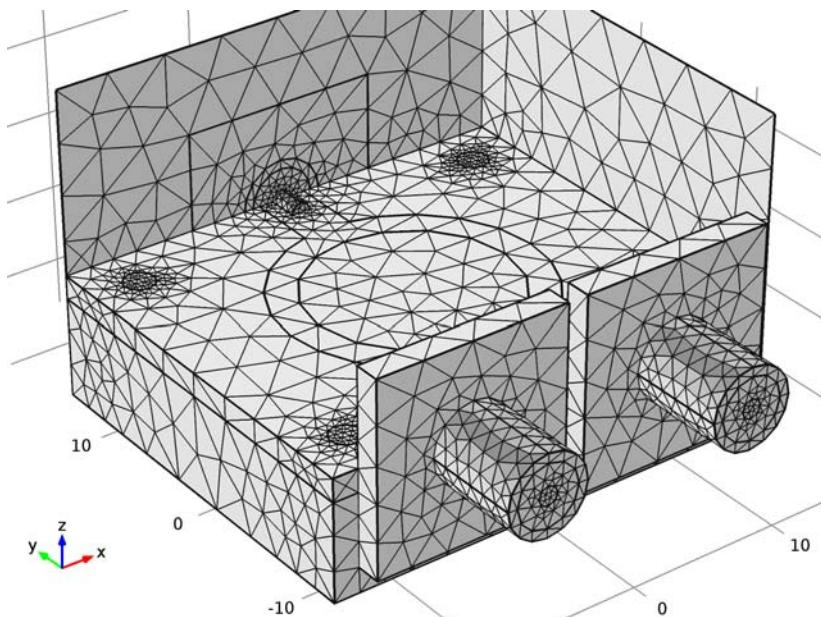
- 4 From the **Selection** list, choose **Substrate**.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type $h_{\max}/\sqrt{3.38}$.

Size 2

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
- 2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Coax 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 edit field, type $h_{\max}/\sqrt{2.1}$.

Free Tetrahedral 1

- 1 Right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Settings** window, click **Build All**.



STUDY 1

Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type `range(f_min,0.1[GHz],f_max)`.
- 4 In the **Model Builder** window, right-click **Study 1** and choose **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 **3D Plot Group** settings window, locate the **Data** section.
- 2 From the **Parameter value (freq)** list, choose **3e9**.
- 3 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
- 4 In the **Multislice** settings window, locate the **Multiplane Data** section.
- 5 Find the **x-planes** subsection. In the **Planes** edit field, type 0.
- 6 Find the **y-planes** subsection. In the **Planes** edit field, type 0.
- 7 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.
- 8 In the **Coordinates** edit field, type 0.
- 9 Click to expand the **Range** section. Select the **Manual color range** check box.
- 10 Set the **Maximum** value to **6**.

The resulting plot shows the E-field equally split between Port 2 and Port 3.

Compare with [Figure 2](#).

ID Plot Group 2

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Right-click **ID Plot Group 2** and choose **Global**.
- 3 In the **Global** settings window, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>\$-parameter, dB>\$-parameter, dB, 11 component (emw.S11dB)**.
- 4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>\$-parameter, dB>\$-parameter, dB, 21 component (emw.S21dB)**.

- 5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB, 3I component (emw.S31dB)**.

- 6 Click the **Plot** button.

The reproduced plot shows the calculated S-parameters. Compare with [Figure 3](#).