Created in COMSOL Multiphysics 6.0

Vivaldi Antenna

Introduction

A tapered slot antenna, also known as a Vivaldi antenna, is useful for wideband applications. Here, an exponential function is used for the taper profile. The objective of this example is to compute the far-field pattern and to compute the impedance of the structure. Good matching is observed over a wide frequency band.

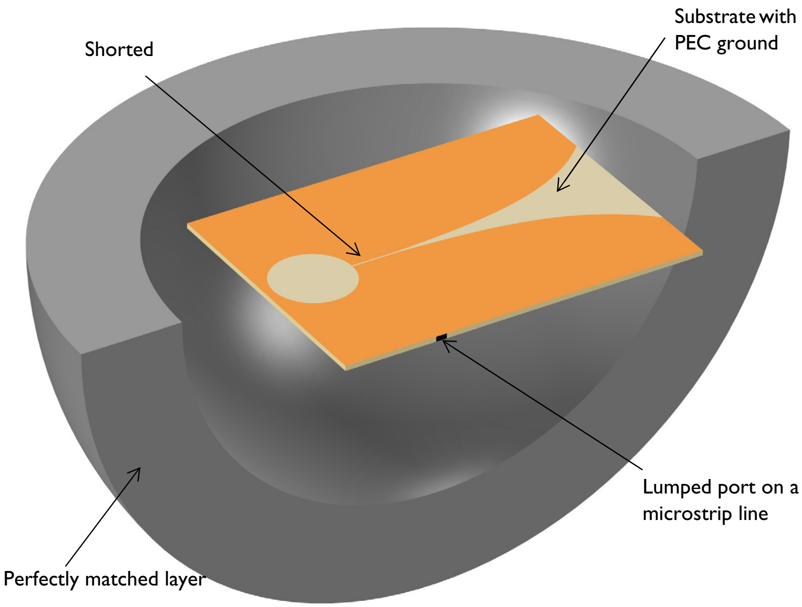


Figure 1: The Vivaldi antenna is realized on a thin dielectric substrate. The entire domain is bounded by a perfectly matched layer.

Model Definition

In this Vivaldi antenna model, the tapered slot is patterned with a perfect electric conductor (PEC) ground plane on the top of the dielectric substrate. A simple exponential function, $e^{0.044x}$ is used to create the tapered slot curves. One end of the slot is open to air and the other end is finished with a circular slot. On the bottom of the substrate, the shorted 50 Ω microstrip feed line is modeled as PEC surfaces. The entire modeling domain is bounded by a perfectly matched layer (PML) which acts like an anechoic chamber absorbing all radiated energy. To excite the antenna, a lumped port is used. The model is meshed using a tetrahedral mesh with approximately five elements per wavelength in each material and simulation frequency.

Results and Discussion

The simulated SWR plot, Figure 2, shows good wideband 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.

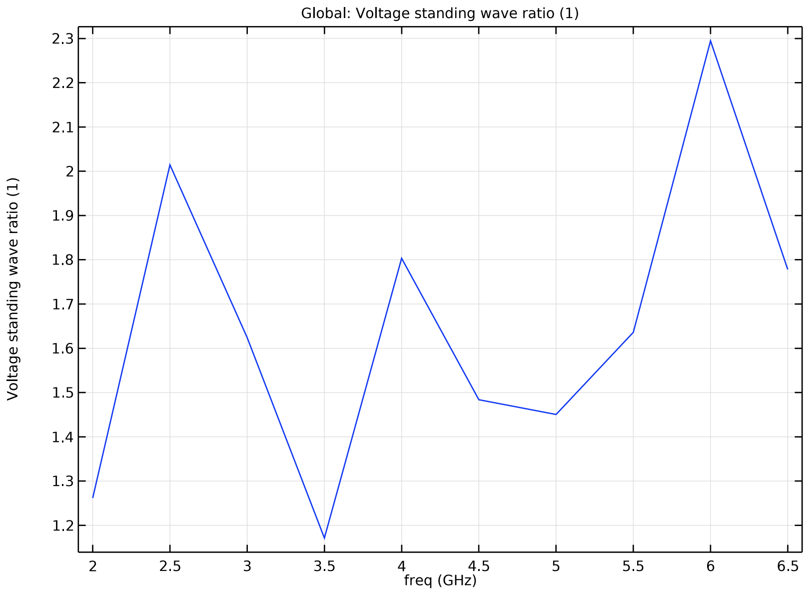


Figure 2: The frequency response SWR of the Vivaldi antenna shows wideband impedance matching, better than 2:1 in most of the simulated frequency range.

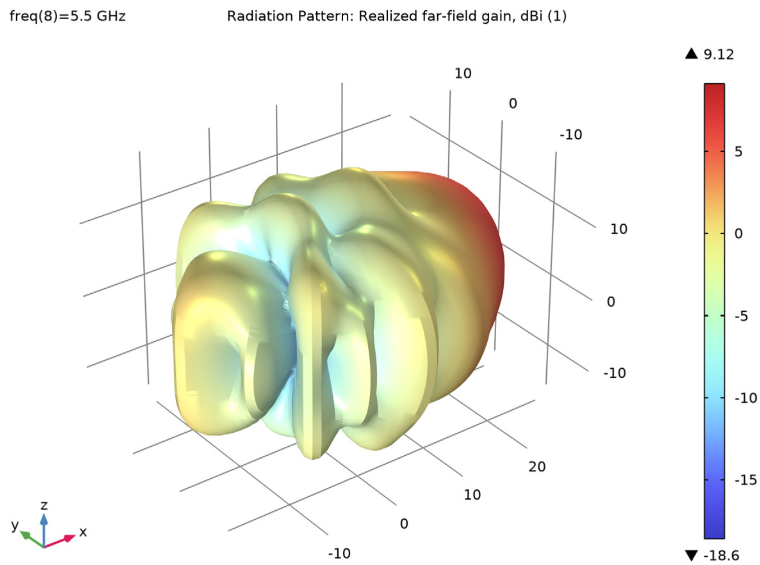


Figure 3: 3D far-field pattern at 5.5 GHz shows a directional radiation pattern.

Application Library path: `RF_Module/Antennas/vivaldi_antenna`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

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

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

STUDY 1

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `range(2[GHz],0.5[GHz],6.5[GHz])`.

GEOMETRY 1

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

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

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

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
thickness	60[mil]	0.001524 m	Substrate thickness
w_slot	0.5[mm]	5E-4 m	Slot with

Here, `mil` refers to the unit milliinch.

GEOMETRY 1

Create a block for the antenna substrate.

Substrate

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

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

3 Locate the **Size and Shape** section. In the **Width** text field, type `110`.

4 In the **Depth** text field, type `80`.

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

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

Next, add a block for the 50Ω microstrip feed line.

Feed line

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

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

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

4 In the **Depth** text field, type `40+w_slot/2`.

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

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

7 In the **x** text field, type `-26`.

8 In the **y** text field, type `-20+w_slot/4`.

Next, create a work plane where you will draw the Vivaldi antenna pattern. Use two parametric curves for the tapered slot.

Work Plane 1 (wp1)

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

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

3 In the **z-coordinate** text field, type `thickness/2`.

4 Click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

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

Add a parametric curve using the exponential profile.

Work Plane 1 (wp1)>Parametric Curve 1 (pc1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `70`.

4 Locate the **Expressions** section. In the **xw** text field, type `s-15`.

5 In the **yw** text field, type `exp(0.044*s)-1+w_slot/2`.

Generate the other parametric curve by mirroring the first one.

Work Plane 1 (wp1)>Mirror 1 (mir1)

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

2 In the **Settings** window for **Mirror**, locate the **Normal Vector to Line of Reflection** section.

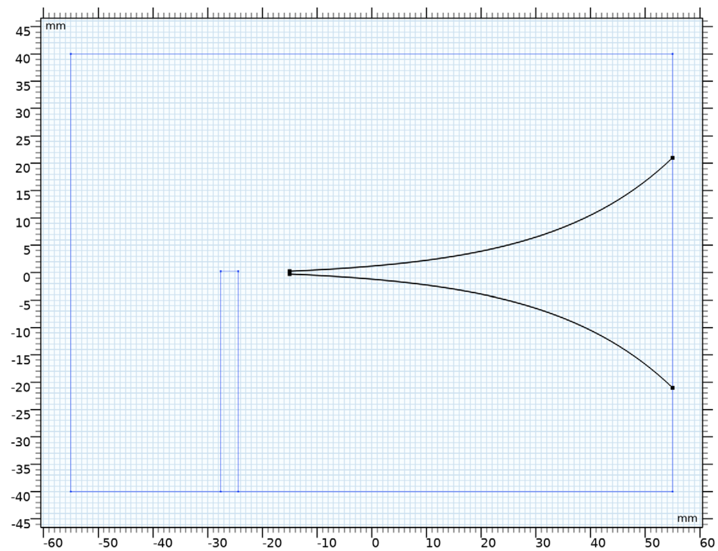
3 In the **yw** text field, type `1`.

4 In the **xw** text field, type `0`.

5 Locate the **Input** section. Select the **Keep input objects** check box.

6 Select the object **pc1** only.

7 Click **Build Selected**.



Add a rectangle describing the thin slot connected to the tapered slot.

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

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

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

3 In the **Width** text field, type `20`.

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

5 Locate the **Position** section. In the **xw** text field, type `-35`.

6 In the **yw** text field, type `-w_slot/2`.

Add a circle attached to the end of the slot.

Work Plane 1 (wp1)>Circle 1 (c1)

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

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

3 In the **Radius** text field, type `12`.

4 Locate the **Position** section. In the **xw** text field, type `-40.5`.

Create a union of the circle and the rectangle to remove unnecessary boundaries.

Work Plane 1 (wp1)>Union 1 (uni1)

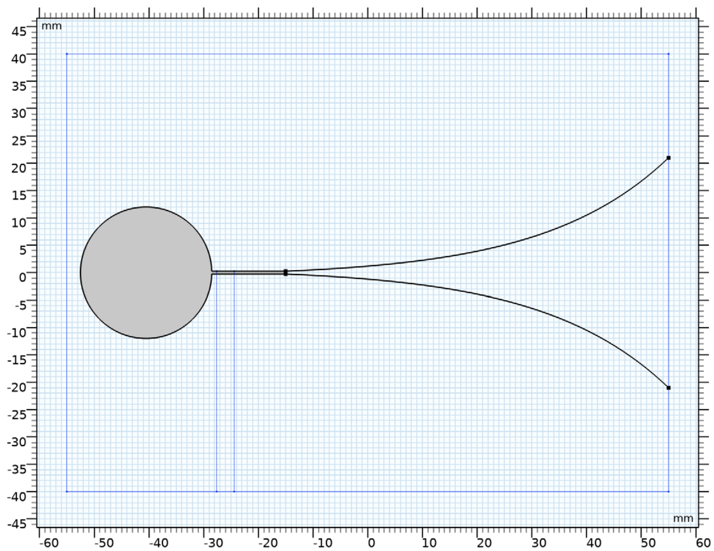
1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **c1** and **r1** only.

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

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

5 In the **Work Plane** toolbar, click **Build All**.



Add a sphere for the PMLs. Use a layer definition to create a shell-type structure.

PML

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

2 In the **Settings** window for **Sphere**, type `PML` in the **Label** text field.

3 Locate the **Size** section. In the **Radius** text field, type `110`.

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

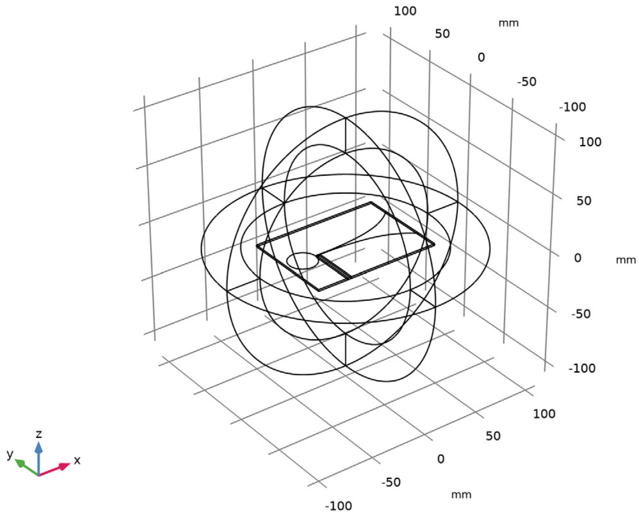
Layer name	Thickness (mm)
Layer 1	30

5 Click **Build All Objects**.

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

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

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



DEFINITIONS

Add a perfectly matched layer.

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1–4 and 8–11 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

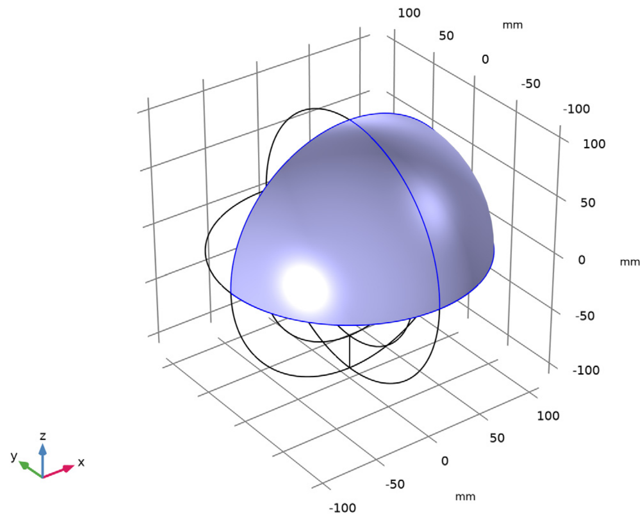
View 1

Hide some domains to get a better view of the interior parts when setting up the physics and reviewing the mesh.

Hide for Physics 1

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

2 Select Domains 2 and 9 only.



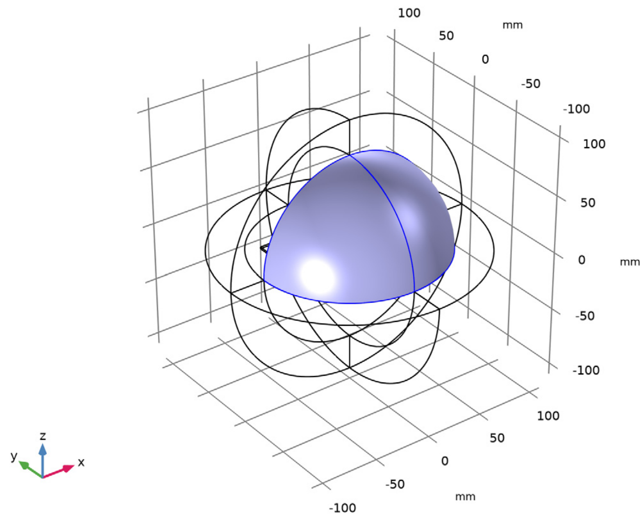
Hide for Physics 2

1 Right-click **View 1** and choose **Hide for Physics**.

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

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

4 Select Boundaries 10 and 36 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. Use the selections already defined when assigning boundary conditions.

Perfect Electric Conductor 2

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

2 Select Boundaries 16, 21, 22, 24, and 27 only.

Far-Field Domain 1

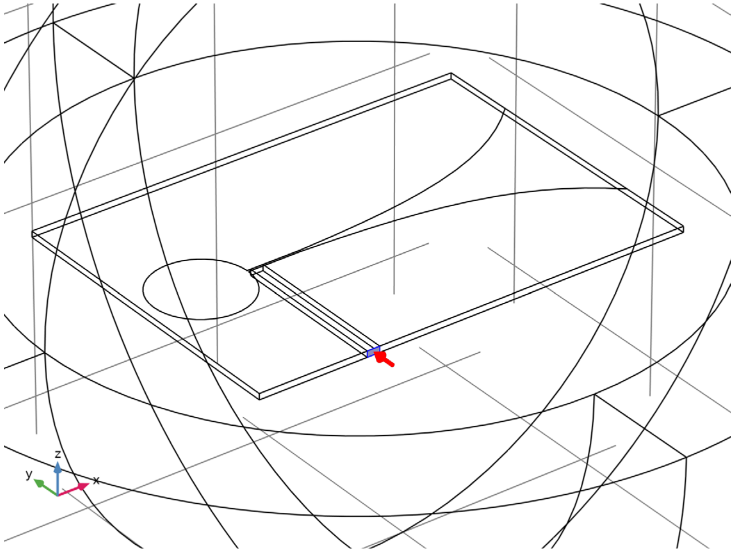
In the **Physics** toolbar, click **Domains** and choose **Far-Field Domain**.

Lumped Port 1

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

2 Click the **Zoom In** button in the **Graphics** toolbar, a couple of times to get a better view.

3 Select Boundary 20 only.



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

MATERIALS

Assign material properties for the model. First, use air for all domains.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in>Air**.

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

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

MATERIALS

Air (mat1)

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

Substrate

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

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

3 Select Domains 6 and 7 only.

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

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

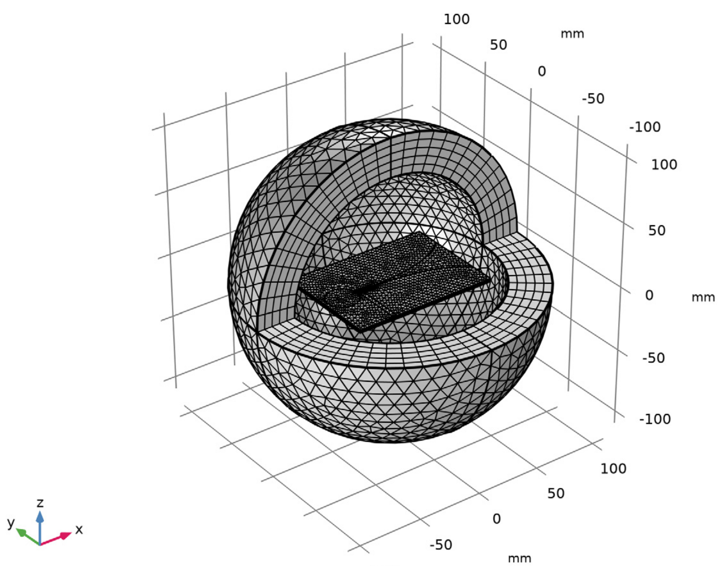
MESH 1

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

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

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

4 Click **Build All**.



STUDY 1

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

RESULTS

Multislice

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

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

3 In the **Expression** text field, type `emw.Ey`.

4 Locate the **Multiplane Data** section. Find the **X-planes** subsection. In the **Planes** text field, type `0`.

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

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

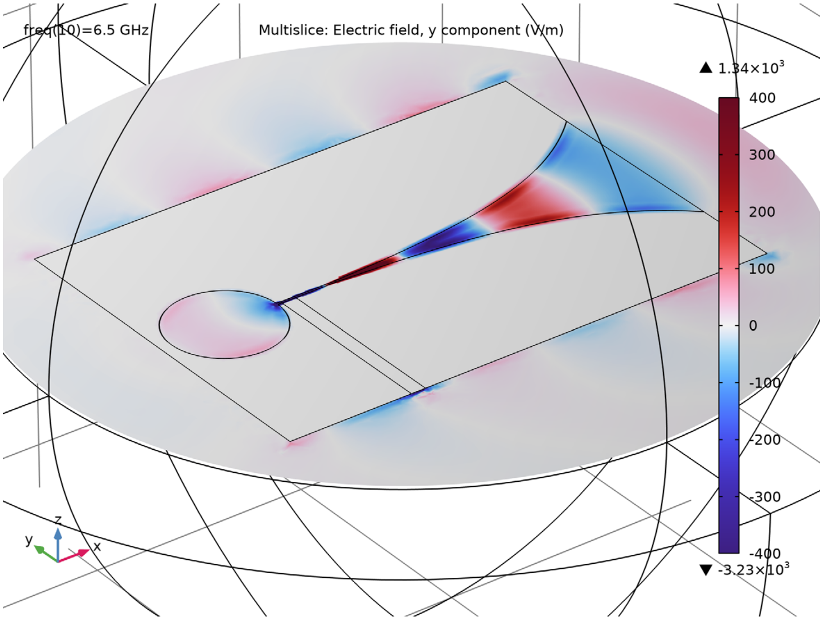
7 In the **Coordinates** text field, type `thickness/2`.

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

9 In the **Minimum** text field, type `-400`.

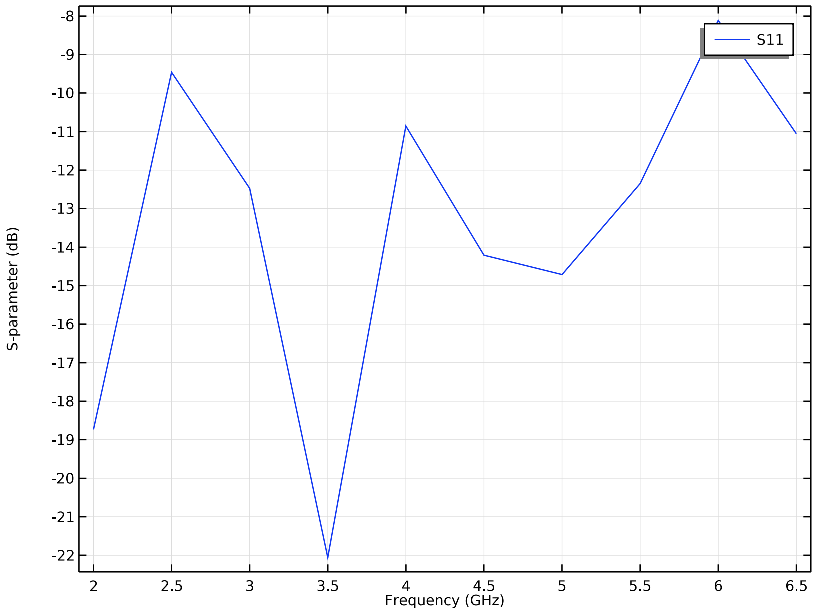
10 In the **Maximum** text field, type `400`.

11 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

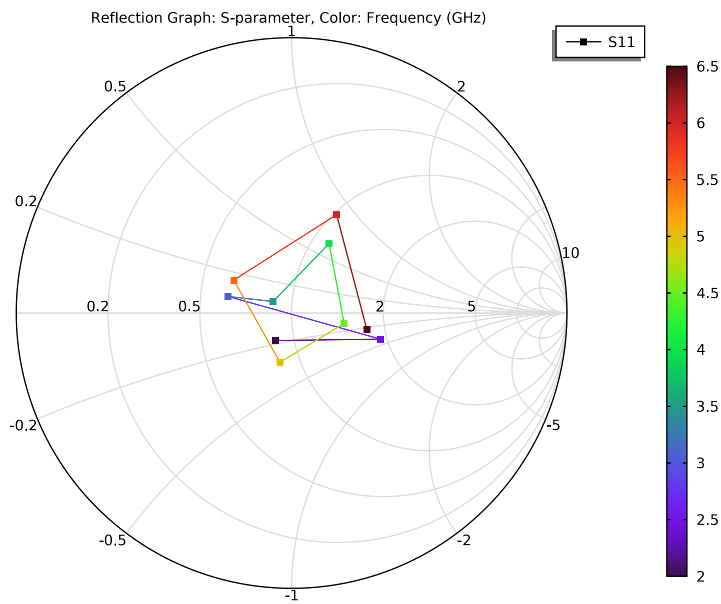


Electric fields are guided along the tapered slot.

S-parameter (emw)



Smith Plot (emw)



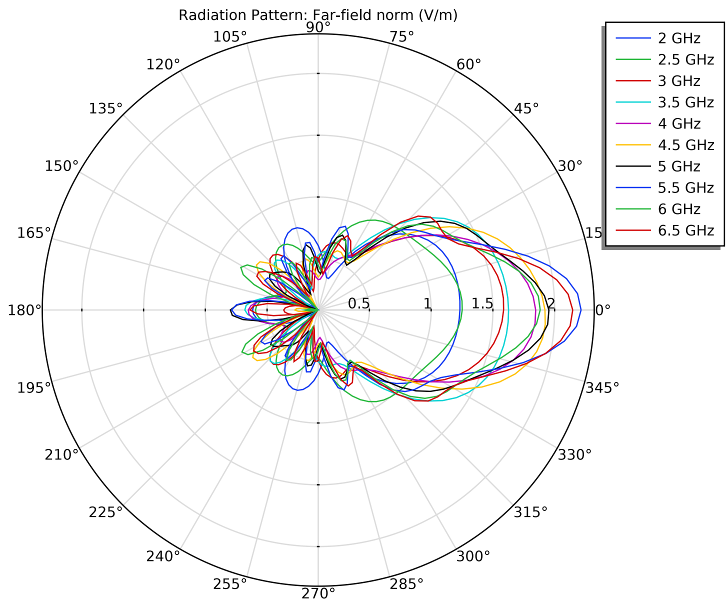
Radiation Pattern 1

1 In the **Model Builder** window, expand the **Results>2D Far Field (emw)** node, then click **Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type 100.

4 In the **2D Far Field (emw)** toolbar, click **Plot**.



2D far-field radiation patterns in the /[xy/]-plane plotted for all frequencies.

3D Far Field, Gain (emw)

1 In the **Model Builder** window, under **Results** click **3D Far Field, Gain (emw)**.

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

3 From the **Parameter value (freq (GHz))** list, choose **5.5**.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **3D Far Field, Gain (emw)** node, then click **Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of elevation angles** text field, type 90.

4 In the **Number of azimuth angles** text field, type 90.

5 In the **3D Far Field, Gain (emw)** toolbar, click **Plot**.

TABLE

1 Go to the **Table** window.

Compare the resulting 3D radiation pattern plot with Figure 3.

RESULTS

1D Plot Group 6

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

Global 1

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

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Ports>emw.VSWR_1 - Voltage standing wave ratio**.

3 Click to expand the **Legends** section. Clear the **Show legends** check box.

4 In the **1D Plot Group 6** toolbar, click **Plot**.

This VSWR plot replicates the wideband frequency response shown in Figure 2.

3D Plot Group 7

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

Isosurface 1

1 Right-click **3D Plot Group 7** and choose **Isosurface**.

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

3 In the **Expression** text field, type `20*log10(emw.normE+0.1)`.

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

Selection 1

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

2 Select Domains 5 and 6 only.

Filter 1

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

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

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

4 In the **3D Plot Group 7** toolbar, click **Plot**.