Spiral Slot Antenna

Introduction

Spiral slot antennas provide a conformal design and can be used for communication, sensing, tracking, positioning, and many applications in different microwave frequency bands due to their wideband frequency response. This example shows how to build a spiral geometry using parametric curves, and computes S-parameters and far-field patterns.

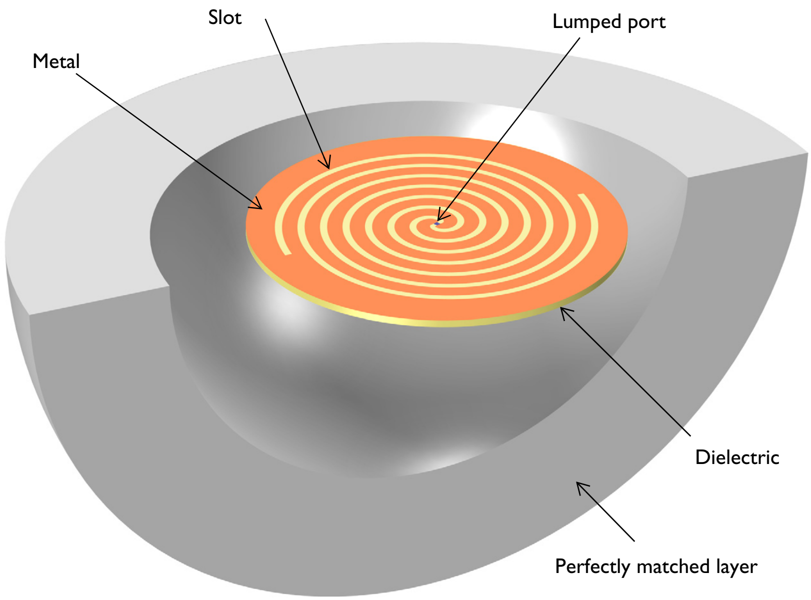


Figure 1: A spiral slot antenna patterned on a single-sided metal substrate is excited by a lumped port.

Model Definition

The spiral slot antenna is built with a two-arm Archimedean spiral slot, which is patterned on a thin single-sided metal substrate using parametric curves. The metal surface is modeled as a perfect electric conductor (PEC) assuming the conductivity is very high and the loss on the surface is negligible. A lumped port is placed at the center of the spiral slot to excite the antenna. The antenna structure and air region are enclosed by a perfectly matched layer (PML). All domains except the PML are meshed by a tetrahedral mesh with approximately five elements per wavelength and the slot boundary is meshed more finely. The PML is swept with a total of five elements along the radial direction.

Results and Discussion

Figure 2 shows the electric field norm on the top surface of the spiral slot antenna. The intensity of the fields along the slot is stronger than at the rest of the surface. The polar plot and 3D far-field visualization in Figure 3 and Figure 4 show bidirectional radiation pattern and maximum radiation along the *z*-axis. Figure 5 shows the calculated S-parameters. In particular, S_{11} over the simulated frequency range is better than −10 dB.

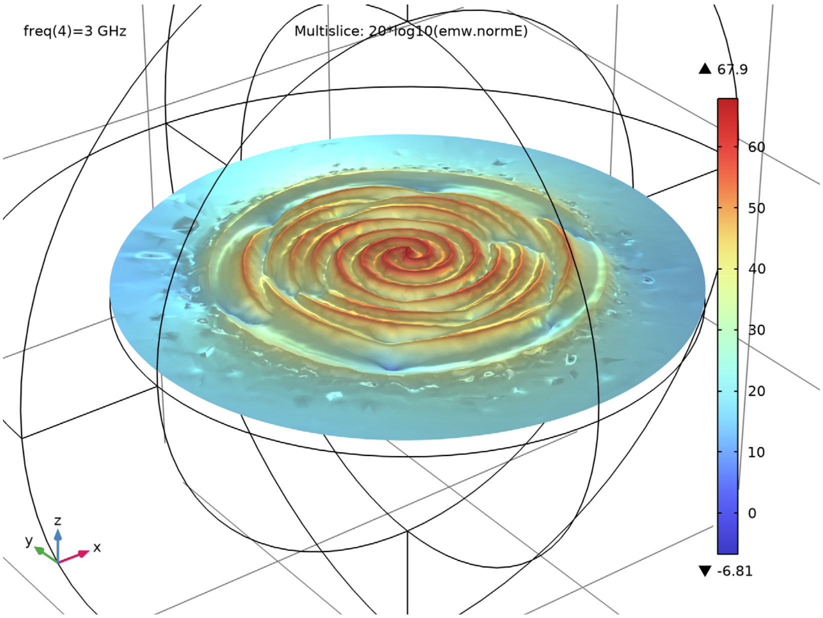


Figure 2: The log-scaled electric field norm on the xy-plane describes how the electric fields are confined on a slotted substrate.

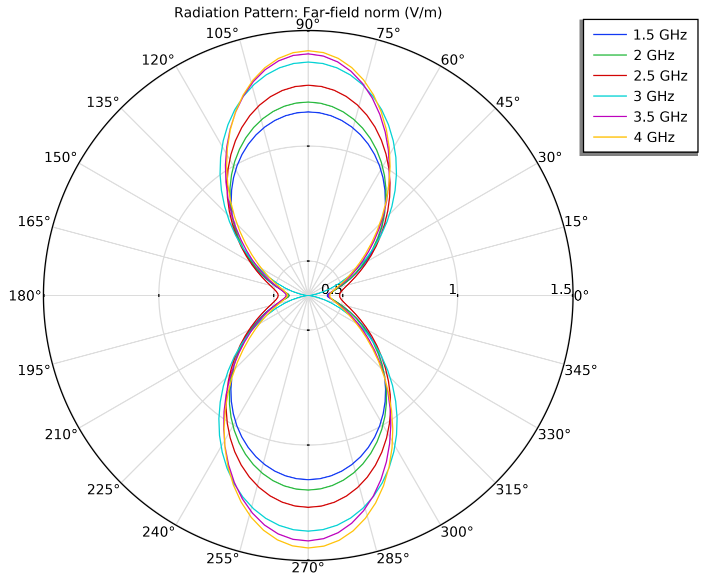


Figure 3: 2D polar plot on the yz-plane showing bidirectional radiation patterns.

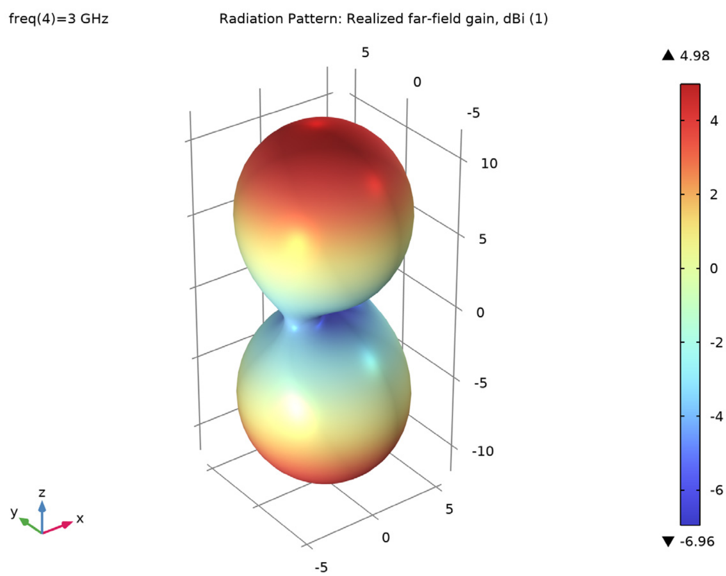


Figure 4: 3D far-field radiation pattern at 3 GHz. The direction of the maximum radiation is along the z-axis.

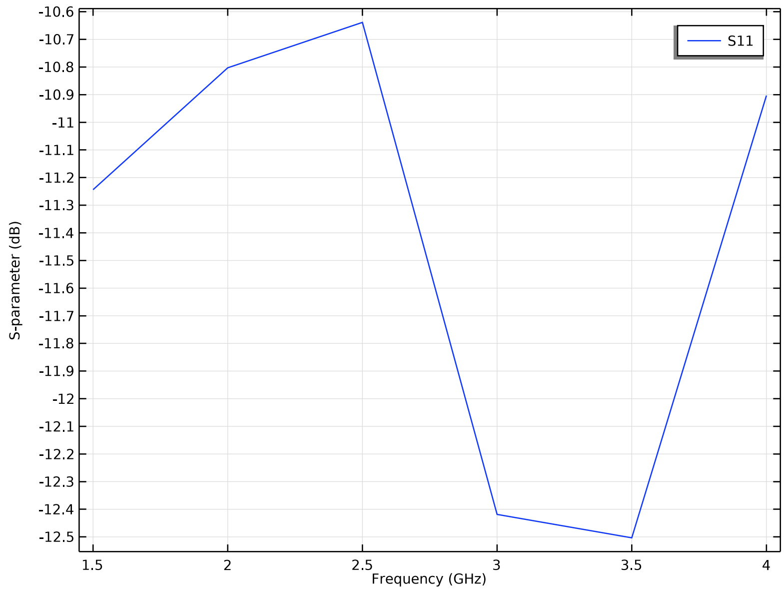


Figure 5: The S-parameter plot shows better than -10 dB S_{11} over the simulated frequency range.

Application Library path: `RF_Module/Antennas/spiral_slot_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(1.5[GHz],0.5[GHz],4[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**.

First, create a cylinder for the substrate.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `40`.

4 In the **Height** text field, type `1.524`.

5 Click **Build Selected**.

Add a work plane on the top surface of the substrate.

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

4 On the object **cyl1**, select Boundary 4 only.

5 Click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

Add a parametric curve to start building a spiral slot.

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.

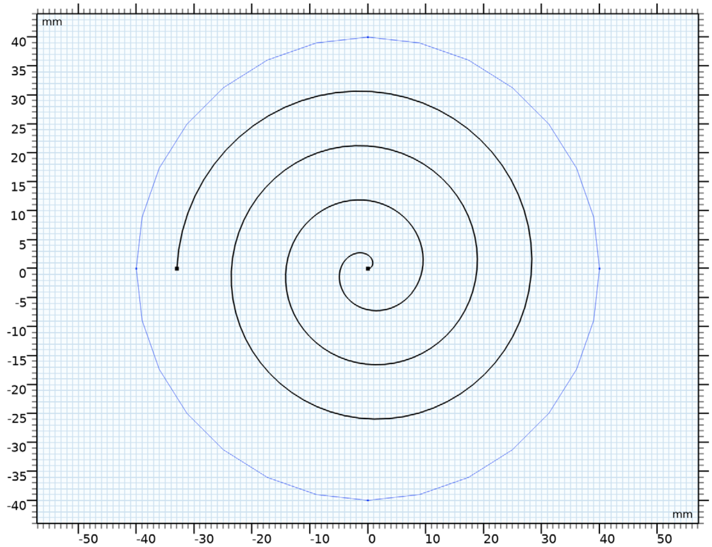
3 In the **Maximum** text field, type `7*pi`.

4 Locate the **Expressions** section. In the **xw** text field, type `1.5*s*cos(s)`.

5 In the **yw** text field, type `1.5*s*sin(s)`.

6 Click **Build Selected**.

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



Work Plane 1 (wp1)>Parametric Curve 2 (pc2)

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 `7*pi`.

4 Locate the **Expressions** section. In the **xw** text field, type `(1.5+1.5*s)*cos(s)`.

5 In the **yw** text field, type `(1.5+1.5*s)*sin(s)`.

Work Plane 1 (wp1)>Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

xw (mm)	yw (mm)
-35	0
-32	0

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

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

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

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

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

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

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

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

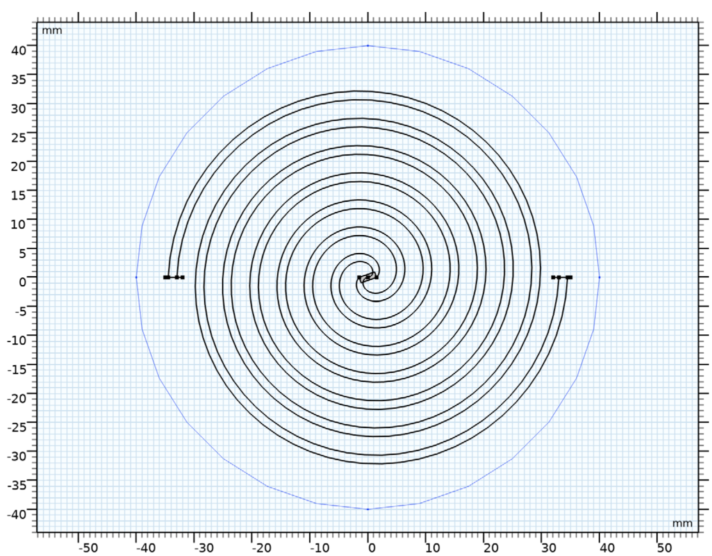
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `sqrt(8)`.

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

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `atan2(1,sqrt(8))/pi*180`.

6 Click **Build Selected**.



All parts to form a spiral slot are added.

Remove unnecessary geometry entities by converting the added parts to solid.

Work Plane 1 (wp1)>Convert to Solid 1 (csol1)

1 In the **Work Plane** toolbar, click **Conversions** and choose **Convert to Solid**.

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

Remove interior boundaries.

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

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

2 Select the object **csol1** only.

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

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

This is the boundary where the excitation port will be assigned.

Work Plane 1 (wp1)>Square 1 (sq1)

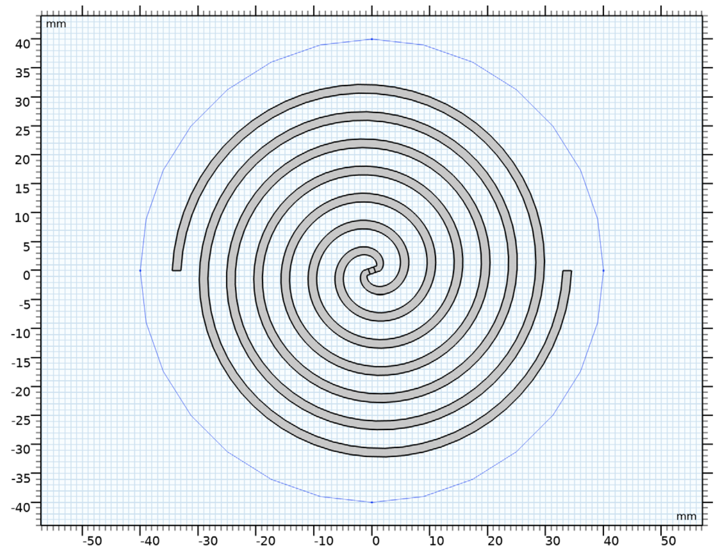
1 In the **Work Plane** toolbar, click **Square**.

2 In the **Settings** window for **Square**, locate the **Position** section.

3 From the **Base** list, choose **Center**.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `atan2(1,sqrt(8))/pi*180`.

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



The layout of the antenna is a two-arm Archimedean spiral.

Add a sphere with a layer definition for the PML.

Sphere 1 (sph1)

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

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

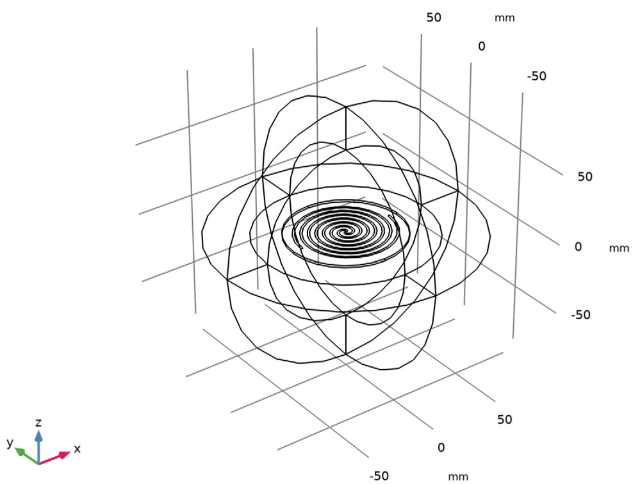
3 In the **Radius** text field, type `90`.

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



The antenna structure is enclosed by the spherical shell.

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 7–10 only.

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

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

DEFINITIONS

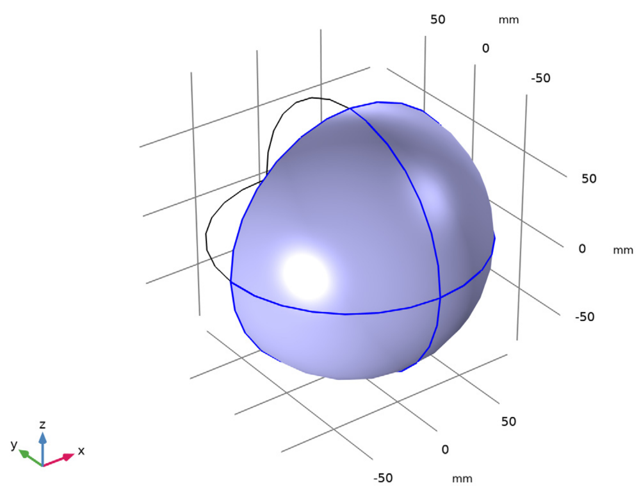
View 1

Suppress some domains and boundaries 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 1, 2, 7, and 8 only.



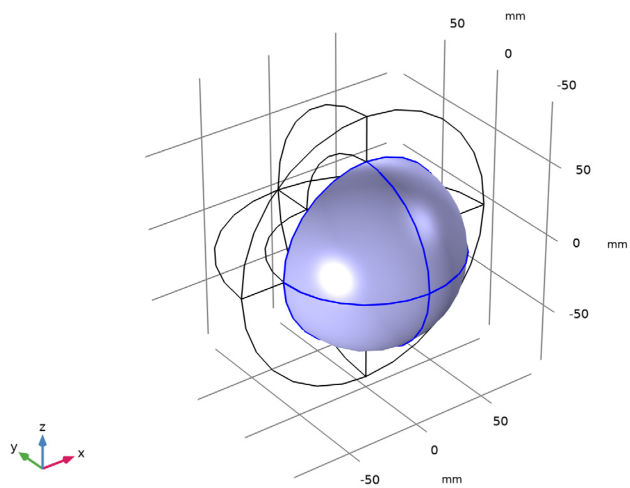
Hide for Physics 2

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

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

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

4 Select Boundaries 9, 10, 25, and 26 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Set up the physics. Start by assigning an additional PEC boundary on the metal surface.

Perfect Electric Conductor 2

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

2 Select Boundary 16 only.

Lumped Port 1

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

2 Select Boundary 19 only.

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

Far-Field Domain 1

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

MATERIALS

Now assign material properties. Use air for all domains and override the substrate with a dielectric material.

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

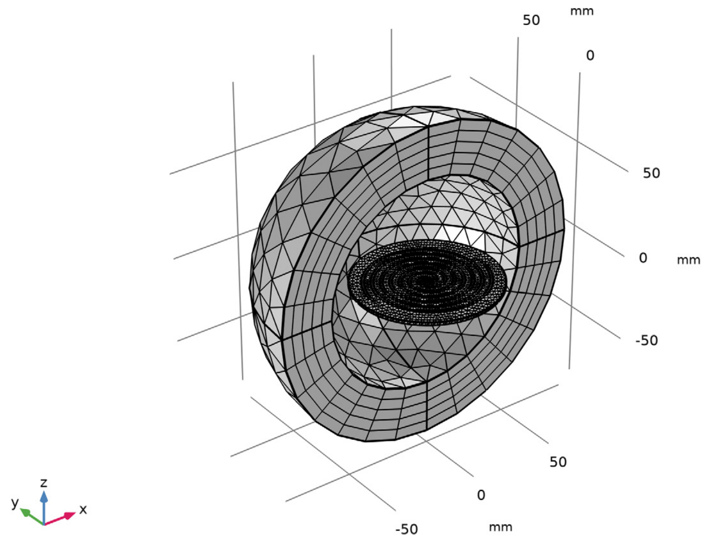
Dielectric material

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Dielectric material` in the **Label** text field.
3 Select Domain 6 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

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



STUDY 1

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

RESULTS

The default plot shows the E-field norm, a 2D far-field polar plot, and the 3D far-field radiation pattern. Adjust plot settings to reproduce the result figures.

Electric Field (emw)

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

Multislice

1. In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
2. In the **Settings** window for **Multislice**, locate the **Expression** section.
3. In the **Expression** text field, type `20*log10(emw.normE)`.

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

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

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

7 In the **Coordinates** text field, type `1.524`.

Deformation 1

1 Right-click **Multislice** and choose **Deformation**.

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

3 In the **Z-component** text field, type `20*log10(emw.normE)`.

4 In the **X-component** text field, type `0`.

5 In the **Y-component** text field, type `0`.

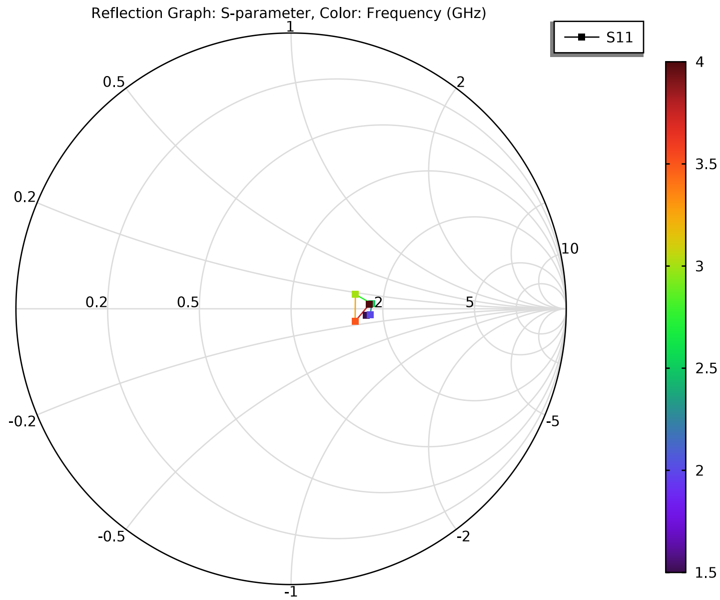
6 In the **Electric Field (emw)** toolbar, click **Plot**.

Compare the reproduced plot with that in Figure 2.

S-parameter (emw)

The calculated S-parameter plot should look like that shown in Figure 5.

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 Find the **Reference direction** subsection. In the **x** text field, type `0`.

5 In the **y** text field, type `1`.

6 Find the **Normal vector** subsection. In the **x** text field, type `1`.

7 In the **z** text field, type `0`.

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

The 2D far-field pattern shows bidirectional characteristics as plotted in Figure 3.

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 **3**.

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

Compare the 3D far-field pattern with the plot in Figure 4.