Created in COMSOL Multiphysics 6.2

Circularly Polarized Antenna for GPS Applications

Introduction

One way to generate circular polarization from a microstrip patch antenna is to truncate the patch radiator. This example is tuned around the GPS frequency range. The axial ratios are calculated to show the degree of circular polarization.

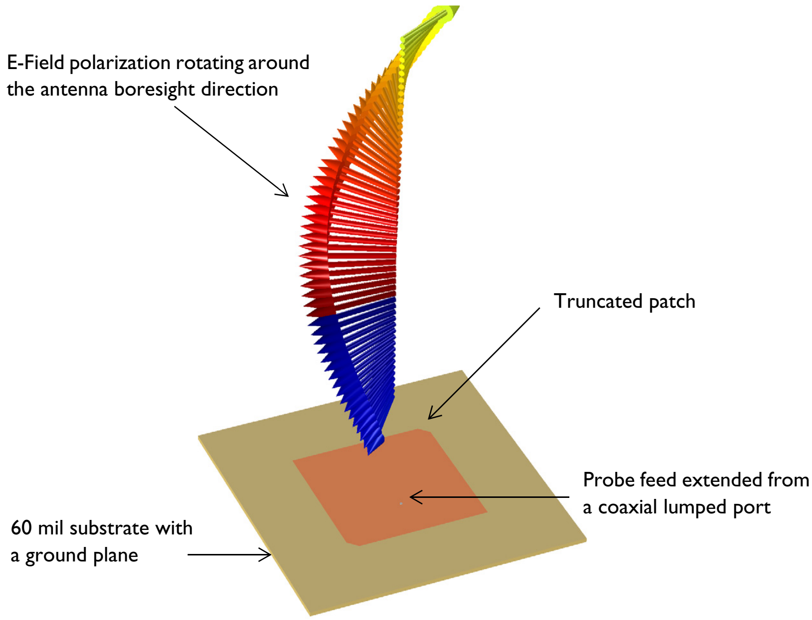


Figure 1: A truncated microstrip patch antenna fed by a probe generates circular polarization along the main radiation direction. Perfectly matched layers enclosing the model domain are not shown in this figure.

Model Definition

A circularly polarized microstrip patch antenna design begins by adding a square metallic patch on top of a 60 mil substrate with a ground plane. The patch size is approximately estimated by a half wavelength inside the substrate;

$$\frac{c_0}{f_0\sqrt{\varepsilon_r}}$$

where c_0 is the speed of light, f_0 is frequency, and ε_r is the relative permittivity of a substrate. This estimated value is only an initial guess number and the size needs to be tuned precisely for the intended frequency.

The basic square or rectangular patch radiator generates a linear polarization. By truncating two diagonally paired corners of the patch, the antenna can produce a circular polarization, electric fields with a fairly equal magnitude and ~90 degree phase difference between two orthogonal components, and x- and y-axis field components.

A rigid coaxial cable filled with Teflon ($\varepsilon_r = 2.1$) is added on the bottom of the substrate and the outer conductor of the coaxial cable is connected to the ground plane. The inner conductor pin of the cable is extended through the dielectric part of the substrate and shorted to the patch on the top surface. All metal parts including the patch, ground plane, and inner and outer conductors of the coaxial cable are modeled as perfect electric conductors.

A coaxial lumped port is used to excite the antenna. It is known that the input impedance on the edge center of a patch is very high while the input impedance around the center of the patch is quite low. The port location is optimized between these two points to get the best matching to the reference impedance 50 Ω with a coaxial probe feed.

The antenna is modeled in a spherical air domain. The air domain is truncated with Perfectly Matched Layers (PMLs) which absorb all outgoing radiation.

All domains except the PMLs are meshed by a tetrahedral mesh with maximum element size of five elements per wavelength so that the wave is well-resolved. The parts in the coaxial cable are meshed more finely to provide good resolution of the curved surfaces. The PMLs are swept with a total of five elements along the absorbing direction.

Results and Discussion

The default plot is modified to show the electric fields only on xy-plane (Figure 2). In general, a linearly polarized patch antenna provides strong fields on two parallel radiating edges. However, the truncated patch antenna shows the radiating fields confined at each corner of the patch. The antenna performs almost equally at every azimuthal angle in terms of the field intensity magnitude.

The axial ratio, which is a measure of the circularity of the polarization, is plotted in Figure 3. In the positive z direction, the evaluated value is less than 3 dB.

The 3D far-field radiation pattern is shown in Figure 4. Because the size of the ground plane is bigger than that of the radiating patch, it blocks the backward radiation and make the pattern directive in the positive z direction.

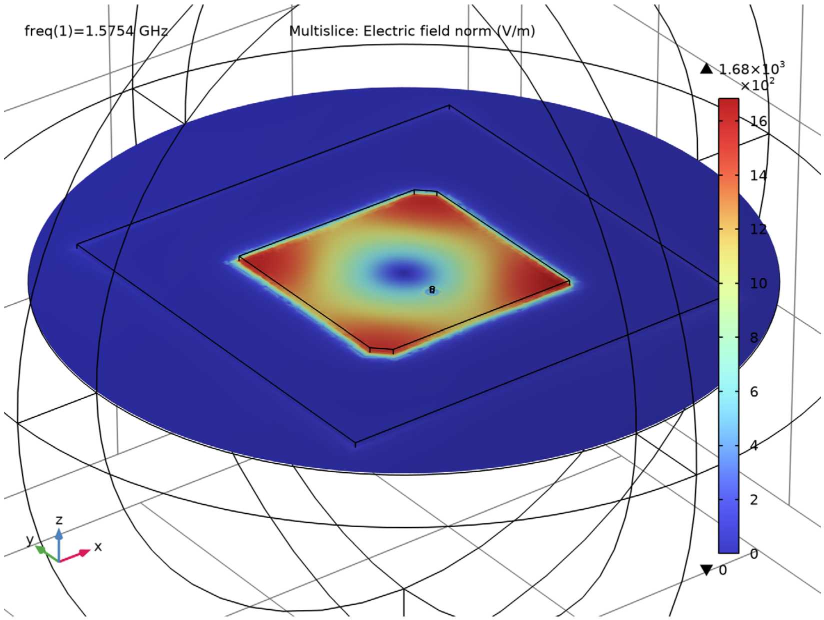


Figure 2: All corners are excited almost equally. This condition is necessary for the antenna to create a circular polarization.

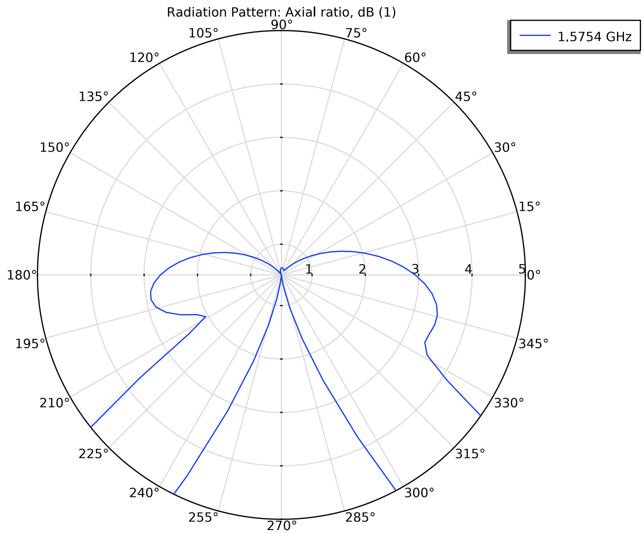


Figure 3: The minimum axial ratio is observed at the antenna boresight (the positive z direction).

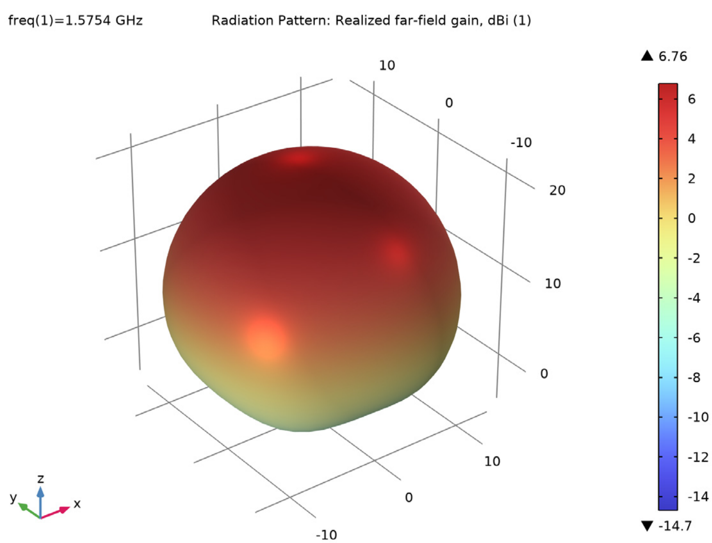


Figure 4: The 3D far-field pattern is directed to the positive z direction due to the ground plane.

Application Library path: `RF_Module/Antennas/circularly_polarized_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

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the specified frequency value.

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

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

3 In the **Frequencies** text field, type `1.57542[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
patch_w	50.28[mm]	0.05028 m	Patch width
probe_l	10[mm]	0.01 m	Probe location
ch_d	3.5[mm]	0.0035 m	Chamfer distance

Here, mil refers to the unit milliinch.

GEOMETRY 1

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type 100[mm].

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

Layer name	Thickness (mm)
Layer 1	20[mm]

5 Click **Build Selected**.

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

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

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

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

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

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

3 In the **Side length** text field, type `patch_w`.

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

Work Plane 1 (wp1)>Square 2 (sq2)

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

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

3 In the **Side length** text field, type `100[mm]`.

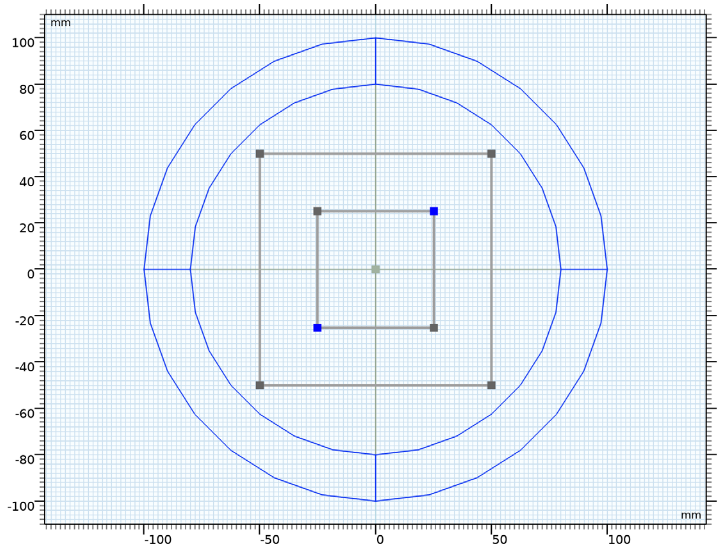
4 Locate the **Position** section. From the **Base** list, choose **Center**.

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

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

2 On the object **sq1**, select Points 1 and 3 only.

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



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

4 In the **Distance from vertex** text field, type `ch_d`.

Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (mm)
thickness

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 `0.5`.

4 In the **Height** text field, type `thickness+2`.

5 Locate the **Position** section. In the **y** text field, type `-probe_l`.

6 In the **z** text field, type `-2`.

Cylinder 2 (cyl2)

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

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

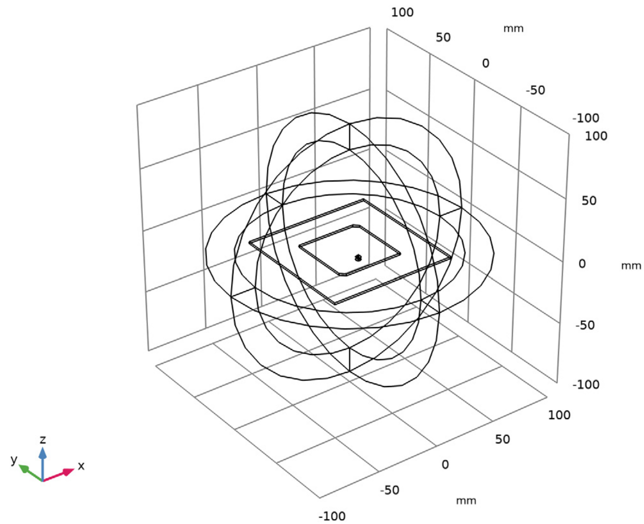
3 In the **Radius** text field, type `2.05`.

4 In the **Height** text field, type `2`.

5 Locate the **Position** section. In the **y** text field, type `-probe_l`.

6 In the **z** text field, type `-2`.

7 Click **Build All Objects**.



The finished geometry should look like this.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1–4 and 11–14 only.

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

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, Select all metal parts.

5 type `15, 20-21, 24-25, 28-32, 34, 42-47` in the **Selection** text field.

6 Click **OK**.

Lumped Port 1

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

2 Select Boundary 26 only.

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

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

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

Far-Field Domain 1

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

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

Material 2 (mat2)

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

2 Select Domains 6 and 7 only.

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

4 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

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

Material 3 (mat3)

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

2 Select Domain 8 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	2.1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

MESH 1

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

DEFINITIONS

Hide for Physics 1

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

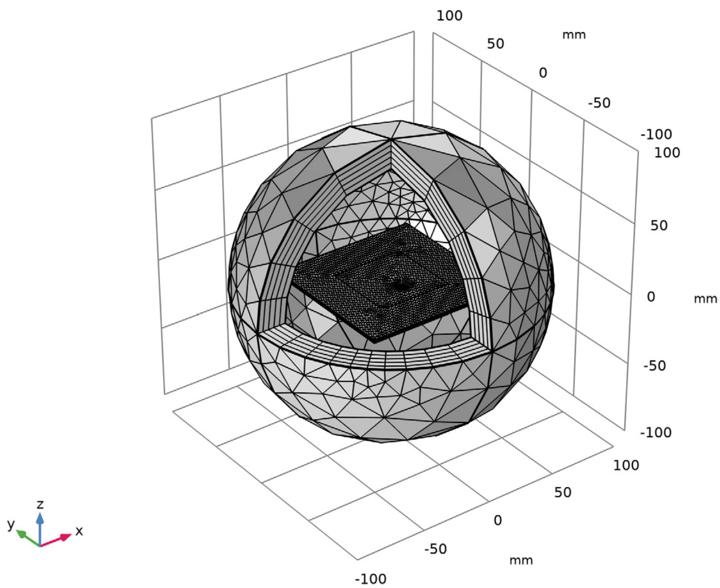
2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 6 and 10 only.

MESH 1

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



STUDY 1

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

RESULTS

Multislice

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

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

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

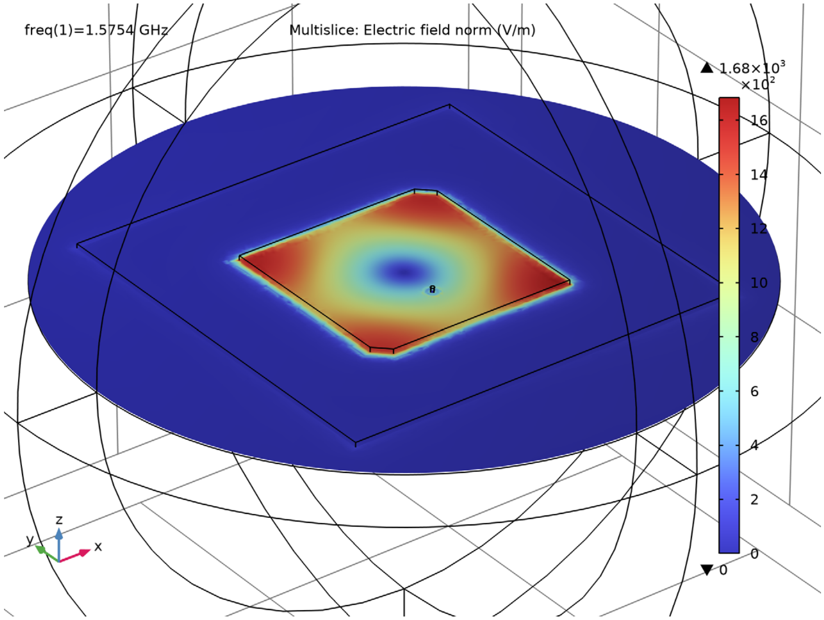
4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

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

6 In the **Coordinates** text field, type `thickness/4`.

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

8 Click the Zoom In button in the Graphics toolbar.



Compare the resulting plot with Figure 2.

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 **Expression** section.

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

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of angles** text field, type `100`.

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

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

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

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

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

2D Far Field (emw)

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

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

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

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

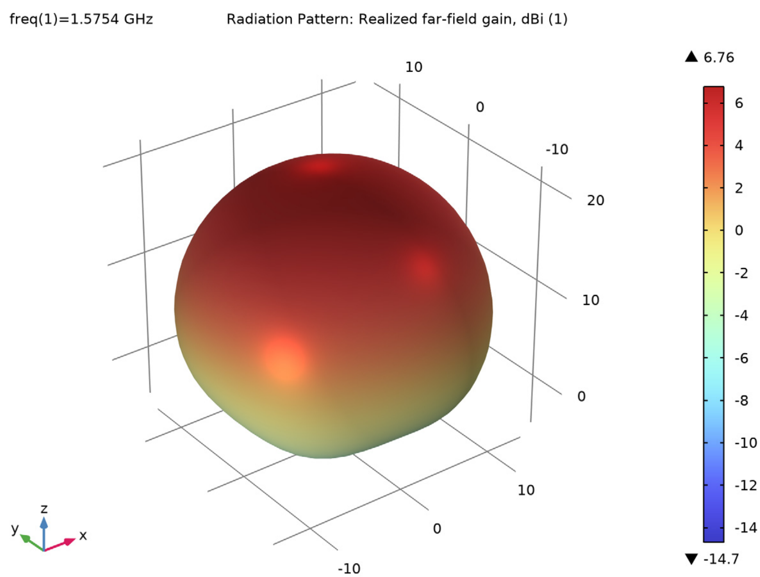
5 In the **2D Far Field (emw)** toolbar, click **Plot**.

3D Far Field, Gain (emw)

The axial ratio in a polar format is shown in Figure 3.

Radiation Pattern 1

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



See the 3D radiation pattern plotted in Figure 4.

Inspect the input matching (S_{11}) at the simulated frequency.