Created in COMSOL Multiphysics 5.6

Biconical Antenna

Introduction

The biconical antenna is a wideband antenna with an omnidirectional radiation pattern. This example models such an antenna, including its coaxial feed structure.

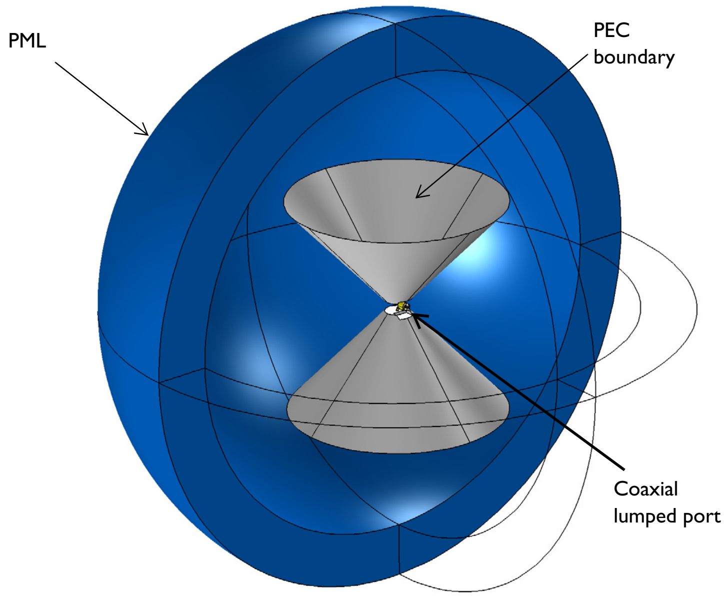


Figure 1: Biconical antenna fed by a coaxial lumped port. The region of free space around the antenna is truncated by a perfectly matched layer (PML).

Model Definition

The biconical antenna, shown in Figure 1, is composed of two conical, metallic, radiating elements. Figure 2 shows the details of the feed structure at the center. A short section of a dielectric-filled coaxial cable starts at a small cylindrical domain containing the power source, which is not part of the model domain. Instead, you model the source by applying a coaxial lumped port condition at the boundary facing the coaxial cable, which launches a wave down the coax. The inner and outer conductors of the coax are connected to the cone-shaped radiators via wires that you model as perfect electric conductors. A small symmetric cutout in each cone provides sufficient clearance for mounting and assembly.

The distance between radiators and the surface area of the cone end tips controls the reactance of the antenna's input port, and can be adjusted to alter antenna performance.

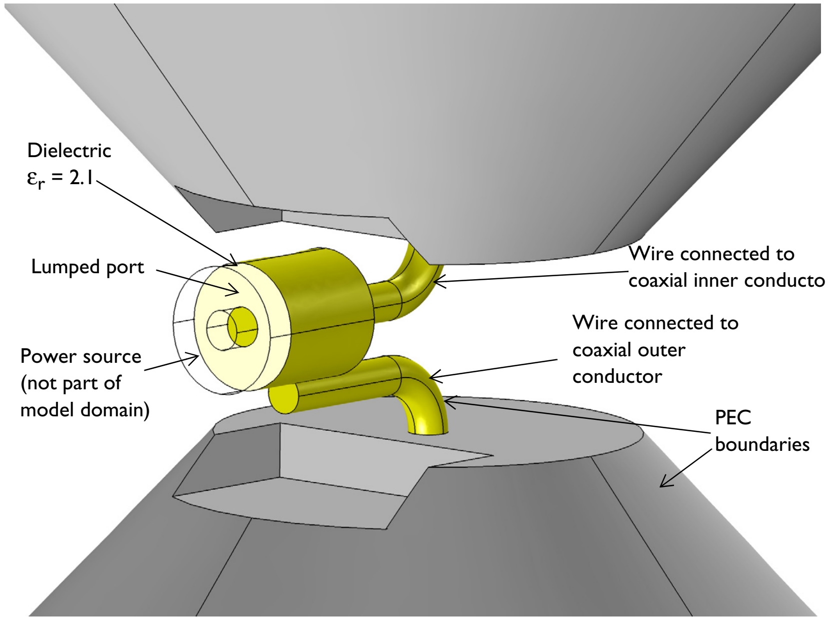


Figure 2: The zoomed view of the coaxial lumped port. The coaxial cable begins at a small cylindrical domain that is external to the model domain, as are the wire interiors.

Results and Discussion

The simulated results in Figure 3 shows that S_{11} is less than −10 dB from 1.5 GHz to 3.5 GHz. This is much wider than a typical dipole antenna bandwidth. The radiation pattern at the E-plane and the H-plane resembles that from a dipole antenna. The biconical antenna works well in applications requiring an omnidirectional radiation pattern and a wide bandwidth.

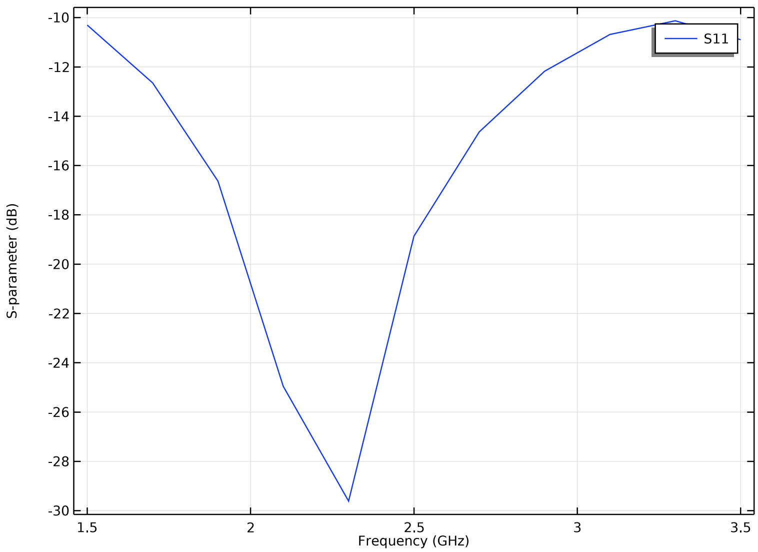


Figure 3: The frequency response of the biconical antenna shows wideband impedance matching.

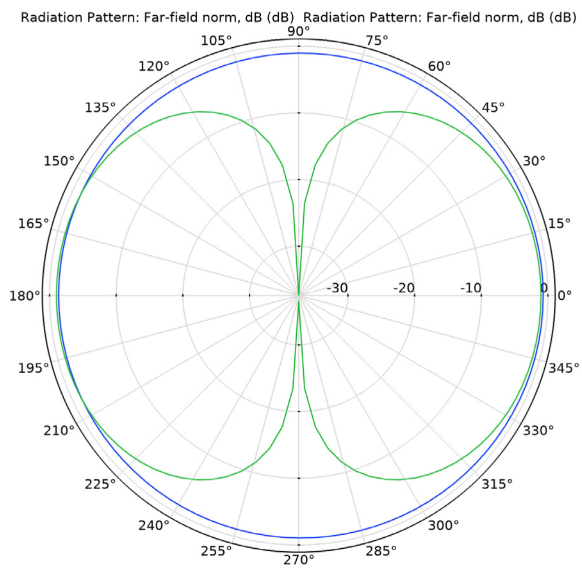


Figure 4: Far-field radiation pattern at the E-plane (blue) and the H-plane (green) at 1.9 GHz. It is similar to the radiation pattern of a dipole antenna.

References

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

2. C.A. Balanis, *Antenna Theory*, John Wiley & Sons, 1997.

3. R.E. Collin, *Antennas and Radiowave Propagation*, McGraw-Hill, 1985.

Application Library path: `RF_Module/Antennas/biconical_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],0.2[GHz],3.5[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**.

First, add a cylinder for the inner conductor.

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 `0.635`.

4 In the **Height** text field, type `7.95`.

5 Locate the **Position** section. In the **x** text field, type `-10`.

6 In the **z** text field, type `1.2`.

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

8 Click **Build Selected**.

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

Add a concentric cylinder to include the outer conductor.

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 `6`.

5 Locate the **Position** section. In the **x** text field, type `-10`.

6 In the **z** text field, type `1.2`.

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

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

Layer name	Thickness (mm)
`Layer 1`	`1`

9 Clear the **Layers on side** check box.

10 Select the **Layers on bottom** check box.

11 Click **Build Selected**.

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

Next, create a structure connecting the coaxial cable and the radiator.

Torus 1 (tor1)

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

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

3 In the **Major radius** text field, type `2.05`.

4 In the **Minor radius** text field, type `0.635`.

5 In the **Revolution angle** text field, type `90`.

6 Locate the **Position** section. In the **x** text field, type `-2.05`.

7 In the **z** text field, type `2.25+1`.

8 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

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

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

11 Locate the **Rotation Angle** section. In the **Rotation** text field, type `270`.

12 Click **Build Selected**.

Create a domain backing the lumped port. This part is excluded from the model space later on.

Cylinder 3 (cyl3)

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 `0.635`.

4 In the **Height** text field, type `4.95`.

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

6 In the **z** text field, type `-1.2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

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

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

10 Click **Build Selected**.

Go on to add a radiator cone.

Cone 1 (cone1)

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

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

3 In the **Bottom radius** text field, type `51`.

4 In the **Height** text field, type `51`.

5 From the **Specify top size using** list, choose **Angle**.

6 In the **Semiangle** text field, type `40`.

7 Locate the **Position** section. In the **z** text field, type `54.25`.

8 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

9 In the **z** text field, type `-1`.

10 Click **Build Selected**.

Add a block representing the assembly and mounting cutout from the upper radiator.

Block 1 (blk1)

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

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

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

4 In the **Depth** text field, type `9`.

5 In the **Height** text field, type `3`.

6 Locate the **Position** section. In the **x** text field, type `-10`.

7 In the **y** text field, type `-4.5`.

8 In the **z** text field, type `2`.

9 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **cone1** only.

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

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

5 Select the object **blk1** only.

6 Click **Build Selected**.

Generate the second radiator by mirroring the first one.

Mirror 1 (mir1)

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

2 Select the objects **dif1** and **tor1** only.

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

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

5 Click **Build Selected**.

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 `150`.
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`40`

5 Click **Build All Objects**.
6 Click the **Go to Default View** button in the **Graphics** toolbar.

DEFINITIONS

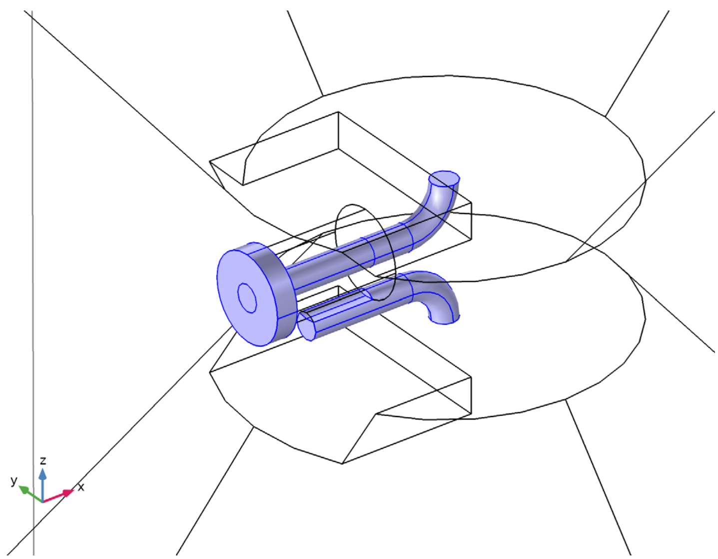
External Domains

1 In the **Definitions** toolbar, click **Explicit**.

Next, create a set of selections for use when setting up the physics.

2 In the **Settings** window for **Explicit**, type `External Domains` in the **Label** text field.
3 Select Domains 8, 9, 11, 12, and 14–16 only.

You can do this most easily by copying the text '8, 9, 11, 12, and 14-16', clicking in the selection box, and then pressing **Ctrl+V**, or by using the **Paste Selection** dialog box.



4 Click the **Go to Default View** button in the **Graphics** toolbar.

Model Domains

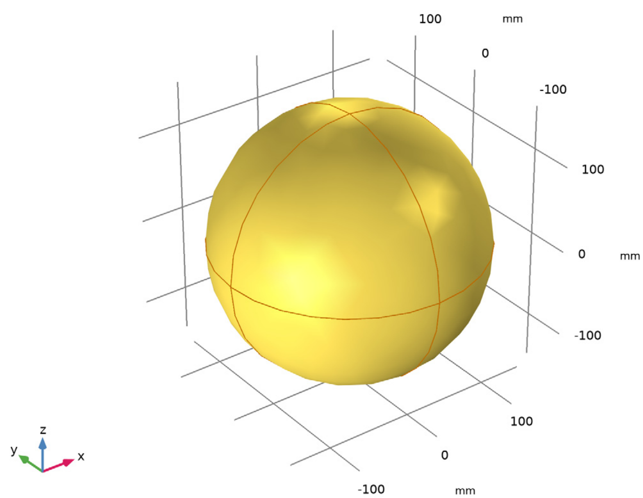
1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Model Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **External Domains** in the **Selections to invert** list.

5 Click **OK**.



Internal PEC Boundaries

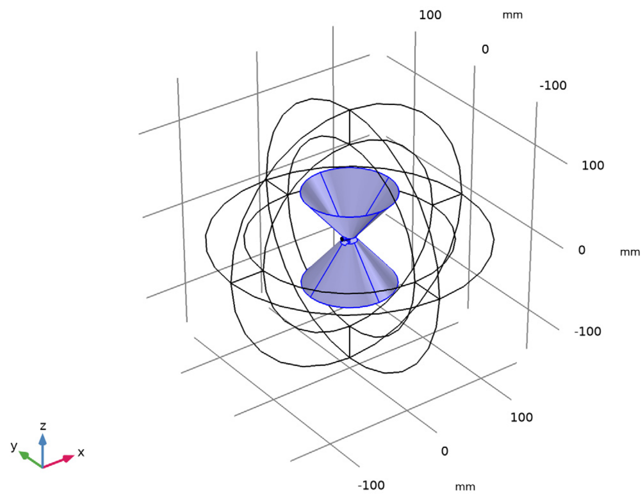
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Internal PEC Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 13, 14, 16, 17, 29, 30, 32, 33, 37, 40–44, 55, 56, 74, 75, 85, 86, 95, and 96 only.

5 Click the **Zoom In** button in the **Graphics** toolbar, a couple of times to see the selected boundaries clearly.



Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.
2 Select Domains 1–4 and 17–20 only.
3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.
4 From the **Type** list, choose **Spherical**.

Hide some domains to get a better view of the interior parts.

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.
2 Select Domains 1, 2, 5, 17, and 18 only.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Model Domains**.

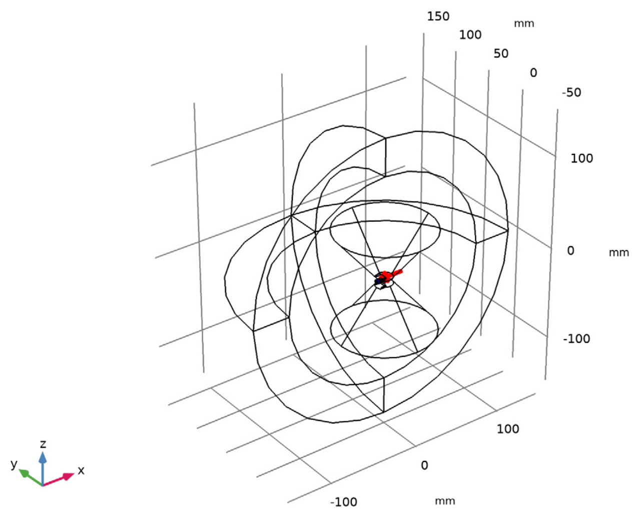
Perfect Electric Conductor 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.
2. In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Internal PEC Boundaries**.

   While the perfect electric conductor is the default boundary condition for exterior boundaries, you need to apply this condition explicitly to interior boundaries.

Lumped Port 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.
2. Select Boundary 31 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

Air (mat1)

Override this material for the coaxial cable domain.

PTFE

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

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

3 Select Domain 10 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	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

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

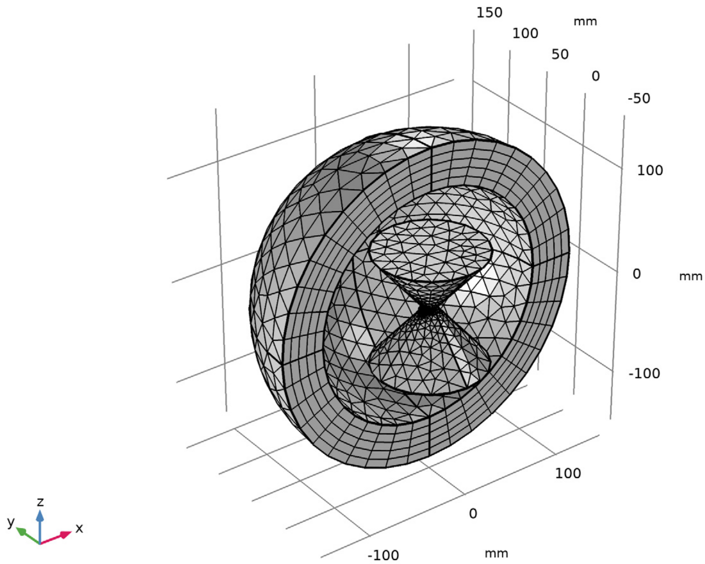
2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

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

4 Click **Build All**.

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

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



7 Click the **Reset Hiding** button in the **Graphics** toolbar, to reset the visibility state of the hidden domains in preparation of the results processing.

STUDY 1

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

RESULTS

Electric Field (emw)

Begin the results analysis and visualization by modifying the first default plot to show the E-field norm in the *xz*-plane.

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. Select the **Description** check box.
5. In the associated text field, type `Electric field norm (dB)`.

6 Locate the **Multiplane Data** section. Find the **Z-planes** subsection. In the **Planes** text field, type 0.

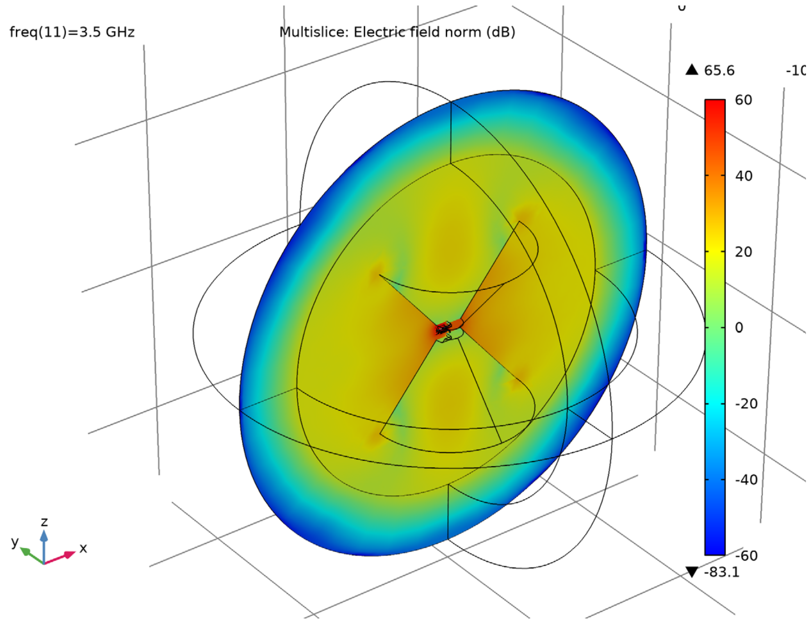
7 Find the **X-planes** subsection. In the **Planes** text field, type 0.

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

9 In the **Minimum** text field, type -60.

10 In the **Maximum** text field, type 60.

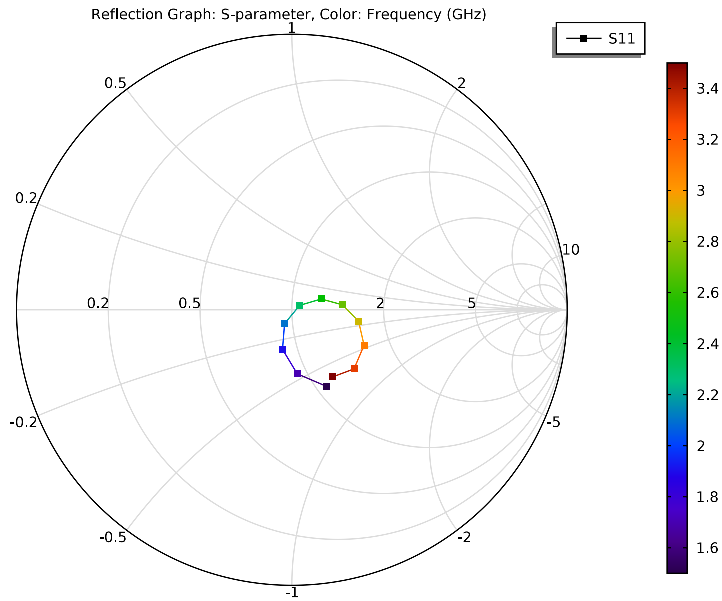
11 In the **Electric Field (emw)** toolbar, click **Plot**.



Use the zoom controls in the **Graphics** toolbar to explore the plot further.

The following instructions reproduce the frequency-response plot shown in Figure 3.

Smith Plot (emw)



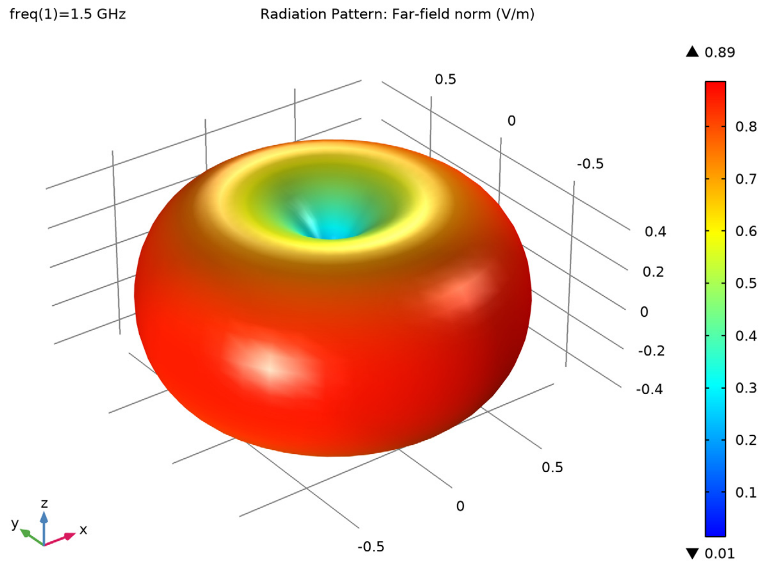
3D Far Field (emw)

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

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

3 From the **Parameter value (freq (GHz))** list, choose **1.5**.

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



3D far-field pattern is isotropic on the *xy*-plane.

Finally, reproduce the polar plot of the far-field on the E- and H-plane.

Polar Plot Group 6

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

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

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

4 In the **Parameter values (freq (GHz))** list, select **1.9**.

Radiation Pattern 1

1 In the **Polar Plot Group 6** toolbar, click **More Plots** and choose **Radiation Pattern**.

2 In the **Settings** window for **Radiation Pattern**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Far field>emw.normdBEfar - Far-field norm, dB - dB**.

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

Radiation Pattern 2

1 Right-click **Radiation Pattern 1** and choose **Duplicate**.

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

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

4 In the **y** text field, type 1.

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

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

7 In the **Polar Plot Group 6** toolbar, click **Plot**.

Compare the resulting plot with that shown in Figure 4.