Microstrip Patch Antenna

Introduction

The microstrip patch antenna is used in a wide range of applications because it is easy to design and fabricate. The antenna is attractive due to its low-profile conformal design, relatively low cost, and very narrow bandwidth. This model uses an inset feeding strategy that does not need any additional matching parts.
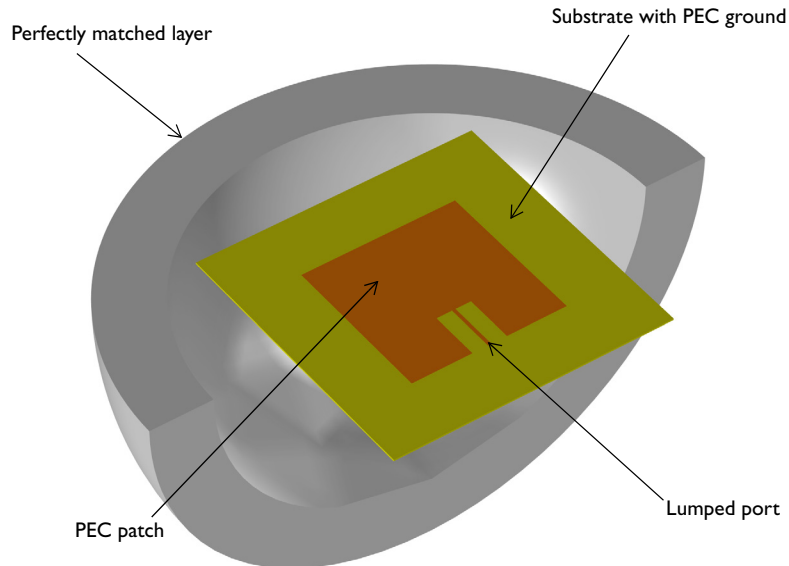

Figure 1: Microstrip patch antenna. The model consists of a PEC ground plane, a 50 Ω microstrip line fed by a lumped port, a region of free space, and a perfectly matched layer (PML) domain.

Model Definition

Feeding a patch antenna from the edge leads to a very high input impedance, causing an undesirable impedance mismatch if a conventional 50 Ω line is directly connected. One solution to this problem is to use a matching network of quarter-wave transformers between the feed point of the 50 Ω line and the patch. However, this approach has two drawbacks. First, the quarter-wave transformers would be realized as microstrip lines that would have to extend beyond the patch antenna, significantly

increasing the overall structure size. Second, these microstrip lines should have a high characteristic impedance and thus would have to be narrower than practical for fabrication. Therefore, a better approach is desired.

This example uses a different feed point for the patch antenna to improve matching between the 50 Ω feed and the antenna. It is known that the antenna impedance will be higher than 50 Ω if fed from the edge, and lower if fed from the center. Therefore, an optimum feed point exists between the center and the edge. The matching strategy is shown in Figure 2. A 50 Ω microstrip line, fed from the end, extends into the patch antenna structure. The width of the cutout region, W, is chosen to be large enough so that there is minimal coupling between the antenna and the microstrip, but not so large as to significantly affect the antenna characteristics. The length of the microstrip line, L, is chosen to minimize the reflected power, S_{11}. These optimal dimensions can be found via a parametric sweep; this example only treats the final design.
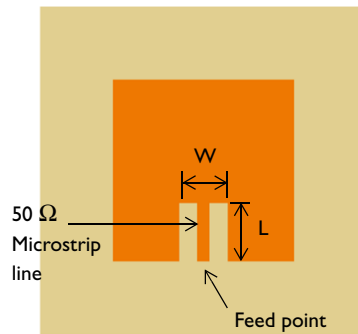

Figure 2: The matching strategy between a 50 Ω line and a patch antenna. A microstrip line of length L extends into a slot of with W cut into the patch antenna.

Results and Discussion

Figure 3 shows the radiation pattern in the E-plane; the E-plane is defined by the direction of the antenna polarization and may include the direction of maximum radiation. 3D far-field radiation pattern is visualized in Figure 4 showing the directive beam pattern due to the ground plane that blocks the radiation toward the bottom side. With the choice of feed point used in this example, the S_{11} parameter is better than −10 dB, and the front-to-back ratio in the radiation pattern is more than 15 dB.
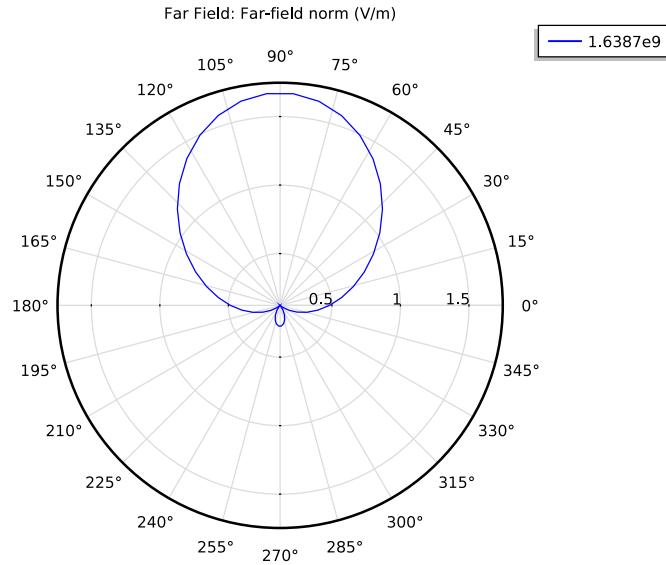
Far Field: Far-field norm (V/m)

Figure 3: Far-field radiation pattern at E-plane. Because of the bottom ground plane, the radiation pattern is directed toward the top.

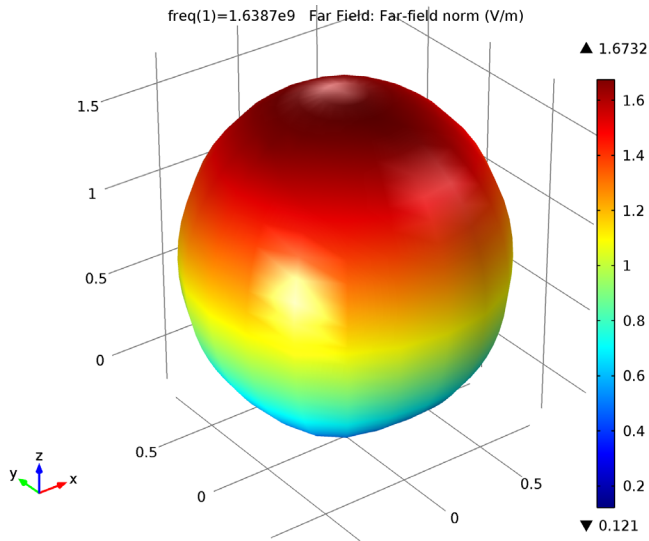freq(1)=1.6387e9   Far Field: Far-field norm (V/m)

Figure 4: 3D Far-field radiation pattern is directive toward the top.

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.

Model Library path: RF_Module/Antennas/microstrip_patch_antenna_inset

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

3 Click the **Add** button.

4 Click the **Study** button.

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

6 Click the **Done** button.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
D	20[mil]	5.0800E-4 m	Substrate thickness
W_line	1.13[mm]	0.0011300 m	50 ohm line width
W_patch	50[mm]	0.050000 m	Patch width
L_patch	50[mm]	0.050000 m	Patch length
W_stub	5[mm]	0.0050000 m	Tuning stub width
L_stub	16[mm]	0.016000 m	Tuning stub length
W_sub	90[mm]	0.090000 m	Substrate width
L_sub	90[mm]	0.090000 m	Substrate length

Here 'mil' refers to the unit milliinches, that is 1 mil = 0.0254×10^{-3} m.

GEOMETRY 1

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

2 In the **Geometry** settings window, locate the **Units** section.

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

First, create the substrate block.

Block 1

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

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

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

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

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

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

7 Click the **Build Selected** button.

8 Right-click **Component 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**.

Now add the patch antenna.

Block 2

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

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

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

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

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

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

7 Click the **Build Selected** button.

8 Right-click **Component 1>Geometry 1>Block 2** and choose **Rename**.

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

10 Click **OK**.

Create impedance matching parts and a 50 Ω feed line.

Block 3

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

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

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

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

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

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

7 In the **x** edit field, type W_stub/2+W_line/2.

8 In the **y** edit field, type L_stub/2-W_patch/2.

9 Click the **Build Selected** button.

10 Right-click **Component 1>Geometry 1>Block 3** and choose **Rename**.

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

12 Click **OK**.

Copy 1

1 On the **Geometry** toolbar, click **Copy**.

2 Select the object **blk3** only.

3 In the **Copy** settings window, locate the **Displacement** section.

4 In the **x** edit field, type -W_stub-W_line.

5 Click the **Build Selected** button.

Difference 1

1 On the **Geometry** toolbar, click **Difference**.

2 Select the object **blk2** only.

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

4 Click the **Active** button next to the **Objects to subtract** selection list.

5 Select the objects **blk3** and **copy1** only.

6 Click the **Build Selected** button.

Continue with the surrounding air and the PML regions.

Sphere 1

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

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

3 In the **Radius** edit field, type 90[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 the **Build All Objects** button.

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

Now create selections of the geometry. You use them later when setting up the physics and meshes.

Explicit Selection 1

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 On the object **sph1**, select Domains 1–4 and 6–9 only. These are all eight domains that make up the outer spherical shell (the PML region). Rotate the geometry as needed.

3 Right-click **Component 1>Geometry 1>Explicit Selection 1** and choose **Rename**.

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

5 Click **OK**.
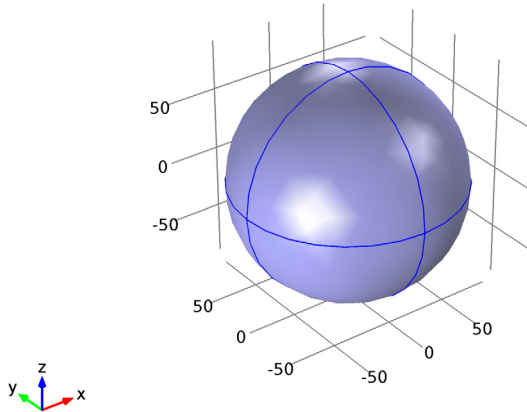
Explicit Selection 2

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Explicit Selection** settings window, locate the **Entities to Select** section.

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

4 On the object **sph1**, select Boundaries 5–8, 16, 17, 22, and 27 only. These are all eight boundary segments of the outer spherical shell. Again, rotate the geometry as needed.

5 Right-click **Component 1>Geometry 1>Explicit Selection 2** and choose **Rename**.

6 Go to the **Rename Explicit Selection** dialog box and type Scattering Boundaries in the **New name** edit field.

7 Click **OK**.
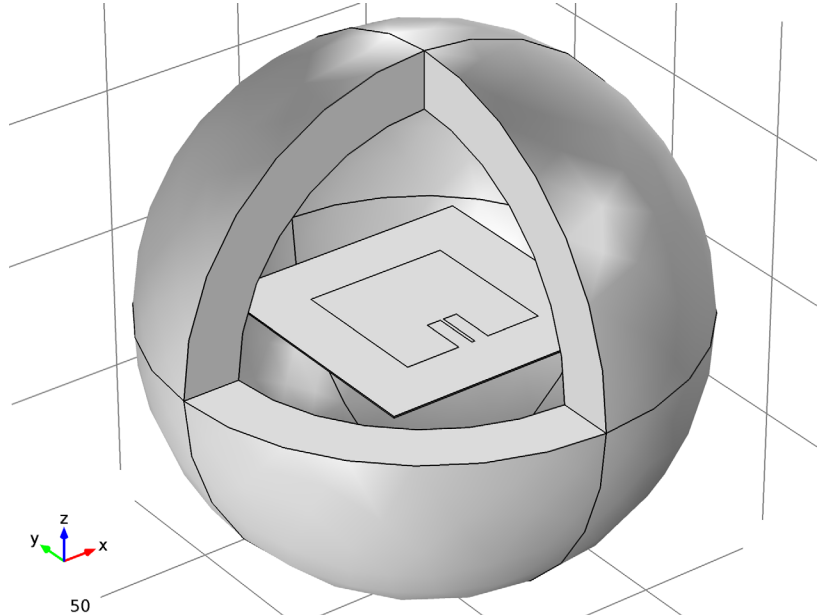
To get a better view, suppress some of the boundaries as shown in the above figure. Furthermore, by assigning the resulting settings to a View node, you can easily return to the same view later.

DEFINITIONS

View 2

1 In the **Model Builder** window, under **Component 1** right-click **Definitions** and choose **View**.

2 Right-click **View 2** and choose **Hide Geometric Entities**.

3 Click the **Show Objects in Selection** in the **Hide Geometric Entities** settings window.

4 Select Domains 2 and 5 only.

Click the **Zoom Box** button on the Graphics toolbar and then use the mouse to zoom in.

View 1

1 In the **Model Builder** window, under **Component 1>Definitions** click **View 1**.

To see the interior, you can also choose wireframe rendering:

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

Before creating the materials for the model, specify the physics. Using this information, the software can detect which material properties are needed.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Perfect Electric Conductor 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.

2 Select Boundaries 15, 20, and 21 only.

Lumped Port 1

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

2 Select Boundary 26 only.

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

4 From the **Wave excitation at this port** list, choose **On**.

Scattering Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 Select Boundaries 5–8, 30, 31, 36, and 41 only.

DEFINITIONS

Perfectly Matched Layer 1

1 On the **Definitions** toolbar, click **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**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Far-Field Domain 1

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

2 Select Domain 5 only.

Far-Field Calculation 1

1 In the **Model Builder** window, under **Component 1>Electromagnetic Waves, Frequency Domain>Far-Field Domain 1** click **Far-Field Calculation 1**.

2 Select Boundaries 9–12, 32, 33, 37, and 40 only.

MATERIALS

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Air**.

3 In the **Add material** window, click **Add to Component**.

4 On the **Home** toolbar, click **Add Material**.

This closes the **Add Material** window.

MATERIALS

Material 2

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

2 Select Domains 6 and 7 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	3.38	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

5 Right-click **Component 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. Moreover, scale the mesh size inside the substrate by the inverse of the square root of the relative dielectric constant.

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

2 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>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 37.

Free Tetrahedral 1

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

2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.

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

4 Select Domains 5–7 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.

2 Select Domains 6 and 7 only.

3 In the **Size** settings window, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated edit field, type 22.8.

Size 2

1 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 Select Boundaries 25, 26, and 43 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 0.5.

8 Select the **Minimum element size** check box.

9 In the associated edit field, type 0.2.

This increases the accuracy of the S-parameter calculations.

Use a swept mesh for the PML domains.

Swept 1

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

2 In the **Swept** settings window, locate the **Domain Selection** section.

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

4 From the **Selection** list, choose **PML**.

Distribution 1

1 Right-click **Component 1>Mesh 1>Swept 1** and choose **Distribution**.

**2** Click the **Build All** button.
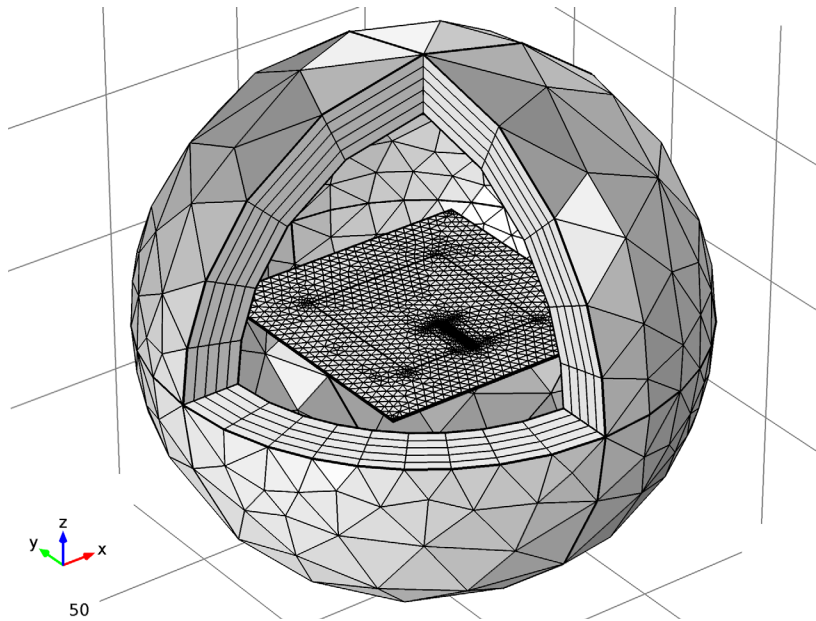
STUDY 1

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 `1.6387[GHz]`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

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

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 **y-planes** subsection. In the **Planes** edit field, type 0.

5 On the **Electric Field (emw)** toolbar, click **Plot**.

6 Click the **Zoom In** button on the Graphics toolbar.
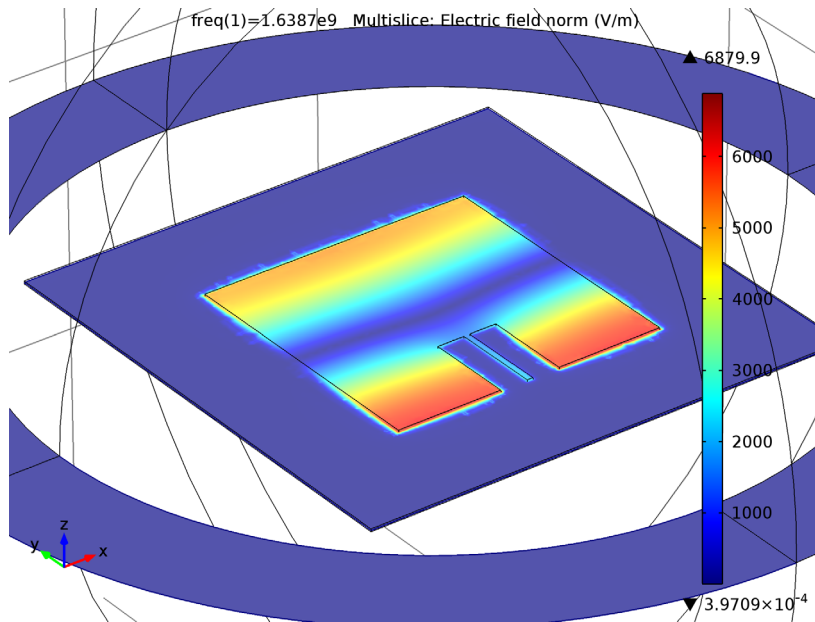
Strong electric fields are observed on the radiating edges.

Polar Plot Group 2

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

2 In the **Far Field** settings window, locate the **Evaluation** section.

3 Find the **Normal** subsection. In the **x** edit field, type 1.

4 In the **z** edit field, type 0.

5 On the **Polar Plot Group 2** toolbar, click **Plot**.

This is the far-field radiation patterns on the E-plane (Figure 3).

3D Plot Group 3

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

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

Derived Values

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

1 On the **Results** toolbar, click **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>S-parameter, dB (emw.S11dB)**.

3 Click the **Evaluate** button.