**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.

![Diagram of Microstrip Patch Antenna](image)

*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 width \( W \) cut into the patch antenna.](image)

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

Figure 4: 3D Far-field radiation pattern is directive toward the top.
Model Library path: RF_Module/Antennas/microstrip_patch_antenna_inset

References


Modeling Instructions

**MODEL WIZARD**
1. Go to the Model Wizard window.
2. Click Next.
4. Click Add Selected.
5. Click Next.
6. Find the Studies subsection. In the tree, select Preset Studies>Frequency Domain.
7. Click Finish.

**GLOBAL DEFINITIONS**

Parameters
1. In the Model Builder window, right-click Global Definitions and choose Parameters.
2. In the Parameters settings window, locate the Parameters section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>20[mil]</td>
<td>Substrate thickness</td>
</tr>
<tr>
<td>W_line</td>
<td>1.13[mm]</td>
<td>50 ohm line width</td>
</tr>
<tr>
<td>W_patch</td>
<td>50[mm]</td>
<td>Patch width</td>
</tr>
<tr>
<td>L_patch</td>
<td>50[mm]</td>
<td>Patch length</td>
</tr>
<tr>
<td>W_stub</td>
<td>5[mm]</td>
<td>Tuning stub width</td>
</tr>
<tr>
<td>L_stub</td>
<td>16[mm]</td>
<td>Tuning stub length</td>
</tr>
<tr>
<td>W_sub</td>
<td>90[mm]</td>
<td>Substrate width</td>
</tr>
<tr>
<td>L_sub</td>
<td>90[mm]</td>
<td>Substrate length</td>
</tr>
</tbody>
</table>

Here ‘mil’ refers to the unit milliinches, that is 1 mil = 0.00254 mm.

**GEOMETRY 1**

1 In the **Model Builder** window, under **Model 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 Right-click **Model 1>Geometry 1** and choose **Block**.
2 In the **Block** settings window, locate the **Size and Shape** 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 **Model 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 Right-click **Geometry 1** and choose **Block**.
2 In the **Block** settings window, locate the **Size and Shape** 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 **Model 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 Right-click **Geometry 1** and choose **Block**.
2 In the **Block** settings window, locate the **Size and Shape** 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 **Model 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 Right-click **Geometry 1** and choose **Transforms>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 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.
2 Select the object `blk2` only.
3 In the Difference settings window, locate the Difference section.
4 Under Objects to subtract, click Activate Selection.
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 Right-click Geometry 1 and choose Sphere.
2 In the Sphere settings window, locate the Size and Shape section.
3 In the Radius edit field, type 90[mm].
4 Click to expand the Layers section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer name</th>
<th>Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 1</td>
<td>20[mm]</td>
</tr>
</tbody>
</table>

5 Click the Build All 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 Right-click Geometry 1 and choose Selections>Explicit Selection.
2 Click and right-click to select all eight domains that make up the outer spherical shell (the PML region). Rotate the geometry as needed.
3 Right-click Model 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  Right-click Geometry 1 and choose Selections>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  Click and right-click to select all eight boundary segments of the outer spherical shell. Again, rotate the geometry as needed.
5  Right-click Model 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.
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 **Model 1** right-click **Definitions** and choose **View**.
2. Right-click **View 2** and choose **Hide Geometric Entities**.
3. Click the **Show Objects in Selection** button on the Graphics toolbar.
4. In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.
5. From the **Geometric entity level** list, choose **Domain**.
6. Select Domains 2 and 5 only.

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

1. In the **Model Builder** window, under **Model 1>Definitions** click **View 2**.
2 In the **View** settings window, locate the **View** section.

3 Select the **Lock camera** check box.

   This action locks the camera settings you just applied for this View node.

4 Click the **Go to View 1** button on the Graphics toolbar to return to the default state.

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

5 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 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

2 Select Boundaries 15, 20, and 21 only.

**Lumped Port 1**

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.

2 Select Boundary 26 only.

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

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

**Scattering Boundary Condition 1**

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Scattering Boundary Condition**.

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

**DEFINITIONS**

**Perfectly Matched Layer 1**

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **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 In the Model Builder window, under Model 1 right-click Electromagnetic Waves, Frequency Domain and choose Far-Field Domain.
2 Select Domain 5 only.

Far-Field Calculation 1
1 In the Model Builder window, under Model 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

Material Browser
1 In the Model Builder window, under Model 1 right-click Materials and choose Open Material Browser.
2 In the Material Browser settings window, In the tree, select Built-In > Air.
3 Click Add Material to Model.

Material 2
1 In the Model Builder window, right-click Materials and choose 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:

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relative permittivity</td>
<td>epsilonr</td>
<td>3.38</td>
</tr>
<tr>
<td>Relative permeability</td>
<td>mur</td>
<td>1</td>
</tr>
<tr>
<td>Electrical conductivity</td>
<td>sigma</td>
<td>0</td>
</tr>
</tbody>
</table>

5 In the Model Builder window, right-click 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.

Free Tetrahedral 1
1 In the Model Builder window, under Model 1 right-click Mesh 1 and choose Free Tetrahedral.
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 Model 1>Mesh 1>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 Domain.
4 Select Domains 6 and 7 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 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 calculation.
Size
1 In the Model Builder window, under Model 1>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.

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 Model 1>Mesh 1>Swept 1 and choose Distribution.
2 In the Settings window, click Build All.
3 Click the Go to View 2 button on the Graphics toolbar.
STUDY 1

Step 1: Frequency Domain
1. In the Model Builder window, expand the Study 1 node, then click Step 1: Frequency Domain.
2. In the Frequency Domain settings window, locate the Study Settings section.
3. In the Frequencies edit field, type $1.6387 \text{ [GHz]}$.
4. In the Model Builder window, right-click Study 1 and choose Compute.

RESULTS

Electric Field (emw)
1. In the Model Builder window, under Results>Electric Field (emw) click Multislice.
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. Click the Plot button.
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. Click the Plot button.
   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. In the Model Builder window, under Results right-click Derived Values and choose Global Evaluation.
3. Click the Evaluate button.