Created in COMSOL Multiphysics 6.0

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 example uses an inset feeding strategy that does not need any additional matching parts.

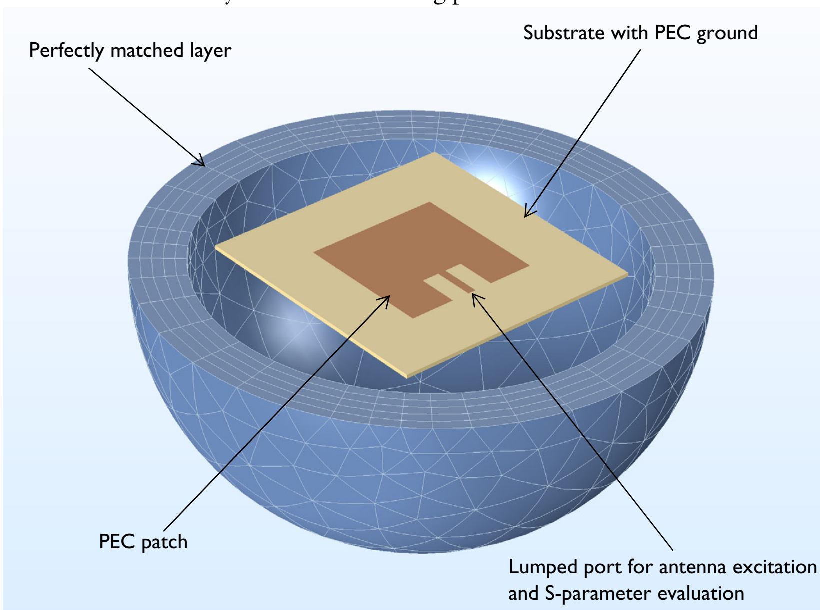


Figure 1: Microstrip patch antenna. The model consists of a PEC ground plane, a 50 Ω microstrip line fed by a lumped port, a region of free space, and a perfectly matched layer (PML) domain.

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 a possible width 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 is 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.

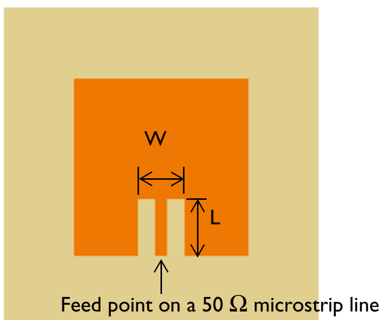


Figure 2: The matching strategy between a 50 Ω line and a patch antenna. A microstrip line of length L extends into a slot of with W cut into the patch antenna.

Results and Discussion

The norm of the electric field inside the antenna substrate is described in Figure 3 where an arrow plot of the electric field is included. The direction of the arrows indicate the dominant polarization in the direction of maximum radiation — the antenna boresight. Figure 4 shows the radiation pattern in the E-plane and H-plane. The E-plane is defined by the direction of the dominant antenna polarization and the H-plane is the plane the magnetic field is mainly polarized in. The 3D far-field radiation pattern is visualized in Figure 5 showing the directive beam pattern due to the ground plane that blocks the radiation toward the bottom side. The calculated antenna directivity is greater than 6.9 dB. 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. The frequency response evaluated with 100 kHz resolution is plotted in Figure 7. The –10 dB S_{11} bandwidth is wider than 10 MHz.

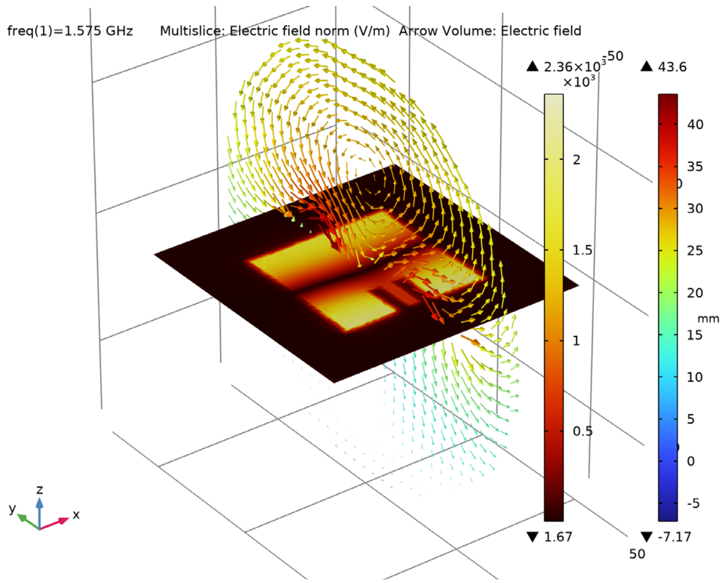


Figure 3: The norm of the electric field is stronger along the radiation edges. The arrow plot shows the dominant direction of polarization of the electric field at the antenna boresight.

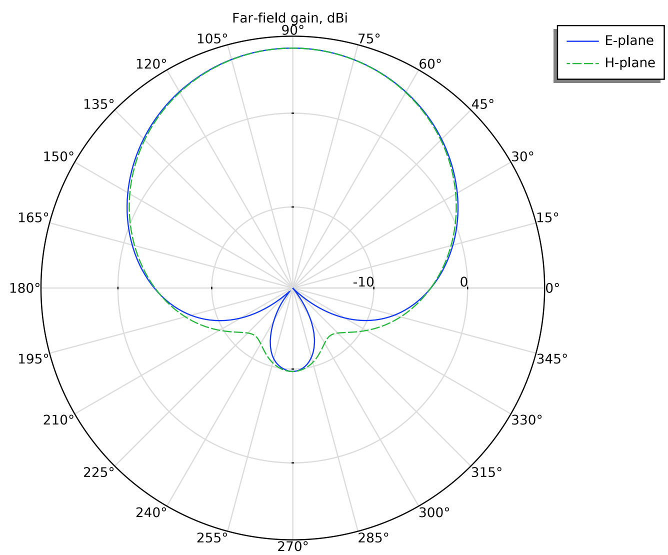


Figure 4: Far-field radiation pattern (gain in dBi) at E-plane and H-plane. Because of the bottom ground plane, the radiation pattern is directed toward the top.

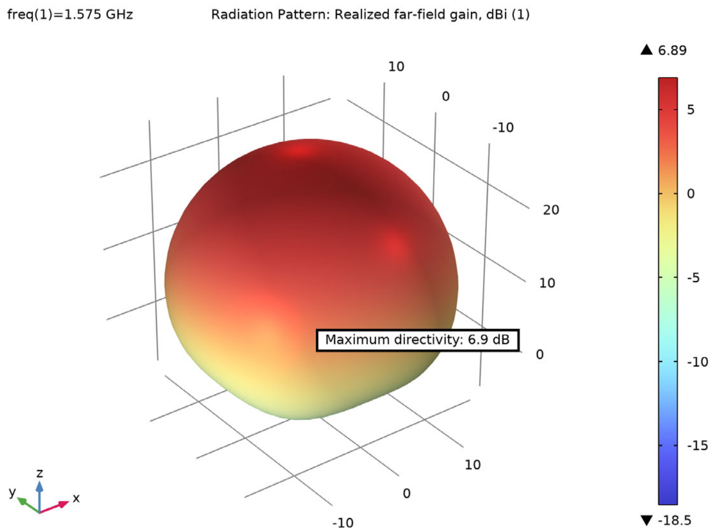


Figure 5: The 3D far-field radiation pattern is directed toward the top. The directivity can be evaluated when plotting the 3D far-field pattern.

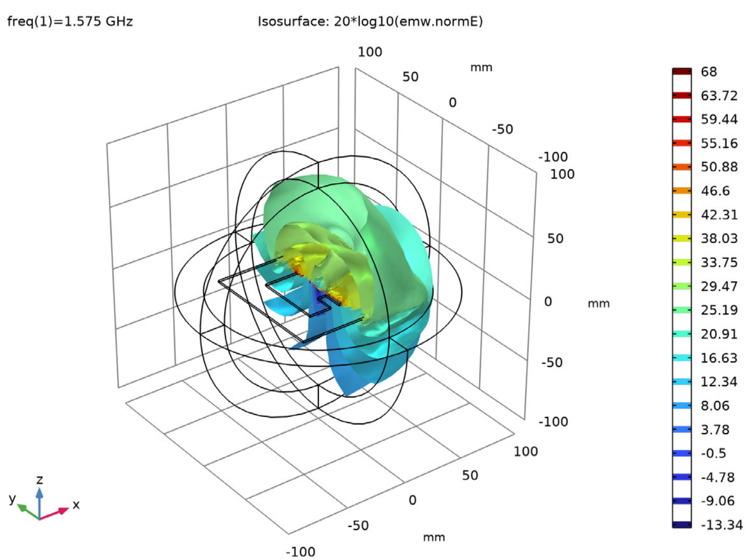


Figure 6: An isosurface plot visualizes the decay of the field amplitude.

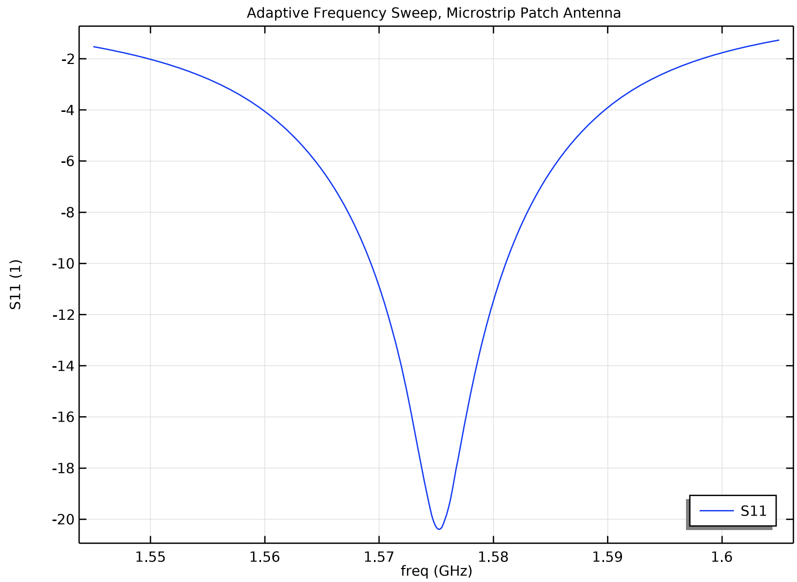


Figure 7: This S-parameter (S_{11}) plot shows that the antenna impedance is matched to 50 Ω around 1.575 GHz.

The 3D full-wave simulation for an antenna array is memory intensive. By using an asymptotic approach such as multiplying the far-field of a single antenna with a uniform array factor, the radiation pattern of an antenna array can be evaluated quickly. Note that this method does not include the coupling among array elements. The 3D uniform array factor function is available under **Definitions>Functions** from the postprocessing context menu when a Far-Field Calculation feature is defined in the physics interface:

```
af3(nx, ny, nz, dx, dy, dz, alphax, alphay, alphaz)
```

where `nx`, `ny`, and `nz` are the number of elements along the x-, y-, and z-axis, respectively. The arguments `dx`, `dy`, and `dz` are the distances between array elements in terms of wavelength. `alphax`, `alphay`, and `alphaz` are the phase progressions in radians. The gain evaluation of a virtual 8-by-8 antenna array in dB scale (Figure 8) uses the following expression:

```
emw.gaindBEfar+20*log10(emw.af3(8,8,1,0.48,0.48,0,0,0,0))+10*log10(1/64)
```

Since it is dB scale, the multiplication of the array factor represents a summation in the expression.

TABLE 1: INPUT ARGUMENTS OF ARRAY FACTOR OPERATOR FOR AN 8-BY-8 ARRAY.

PARAMETER	DESCRIPTION	ARGUMENT	UNIT
nx	Number of elements along *x*-axis	8	Dimensionless
ny	Number of elements along *y*-axis	8	Dimensionless
nz	Number of elements along *z*-axis	1	Dimensionless
dx	Distance between array elements along *x*-axis	0.48	Wavelength
dy	Distance between array elements along *y*-axis	0.48	Wavelength
dz	Distance between array elements along *z*-axis	0	Wavelength
alphax	Phase progression along *x*-axis	0	Radian
alphay	Phase progression along *y*-axis	0	Radian
alphaz	Phase progression along *z*-axis	0	Radian

It is assumed that the array is excited by a single input uniform distribution network, so the input power needs to be scaled by a factor `10*log10(1/total number of elements)`. The direction of the main beam can be steered by defining nonzero phase progression in the uniform array factor. The maximum radiation direction of the array factor along the x-axis is defined by the angle θ from the x-axis in the phase progression using

$$\alpha_x = -kd\cos\theta = -(2\pi d/\lambda)\cos\theta$$

Figure 9 demonstrates the use of the antenna array factor. It includes three plots to show the evolution of the antenna radiation pattern from a single antenna to a synthesized antenna array via the uniform array factor:

1. The gain of the single microstrip patch antenna.
2. The radiation pattern of the uniform array factor configured to have the maximum radiation at 60 degrees from the x-axis by setting the value of alphax as in Table 2.
3. The gain of the 8-by-8 microstrip patch antenna array, using the array factor defined above.

TABLE 2: INPUT ARGUMENTS OF ARRAY FACTOR OPERATOR TO STEER THE BEAM.

ARGUMENT	VALUE	UNIT
alphax	-2*pi*0.48*cos(pi/3)	Radian

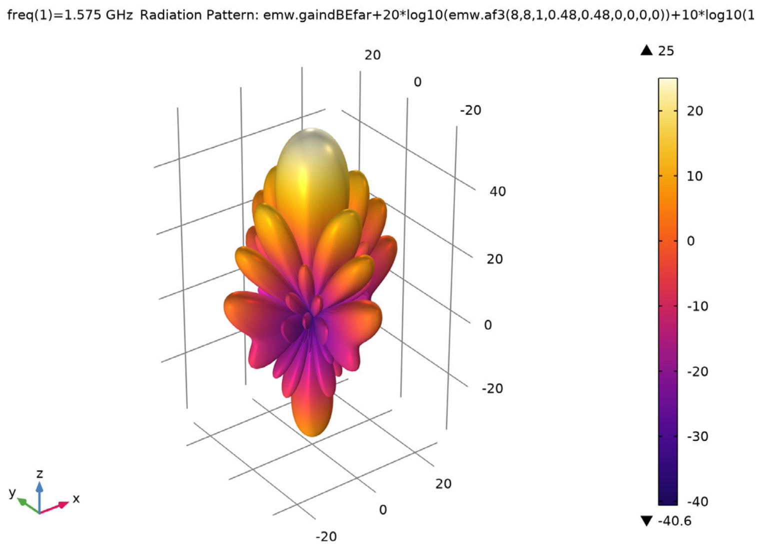


Figure 8: The far-field radiation pattern of a virtual 8-by-8 microstrip patch antenna array.

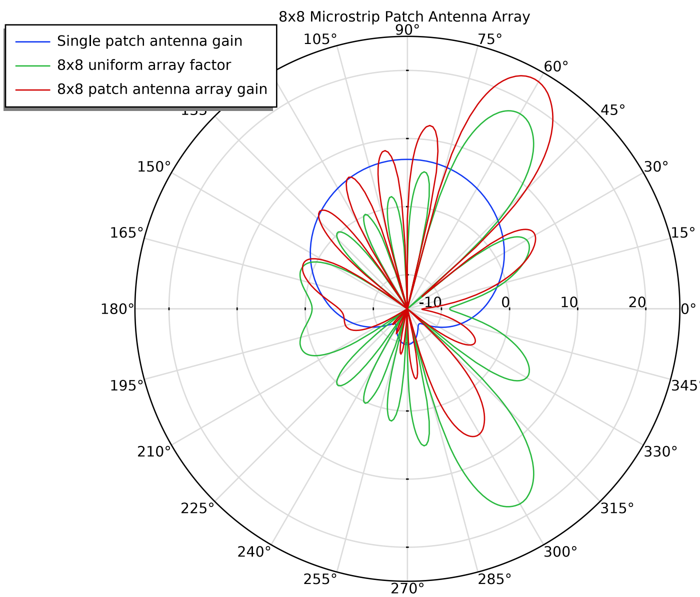


Figure 9: The single patch antenna gain, 8-by-8 uniform array factor, 8-by-8 microstrip patch antenna array gain plotted in dB scale.

The final part of this tutorial model demonstrates how mesh adaptation can be used to increase the result accuracy, by making the mesh finer in regions where it matters most for the results. Figure 10 shows the mesh after running the Frequency Domain, RF Adaptive Mesh study. Compared to the initial mesh, the adapted mesh is much finer around the patch edges, except where the field strength is low.

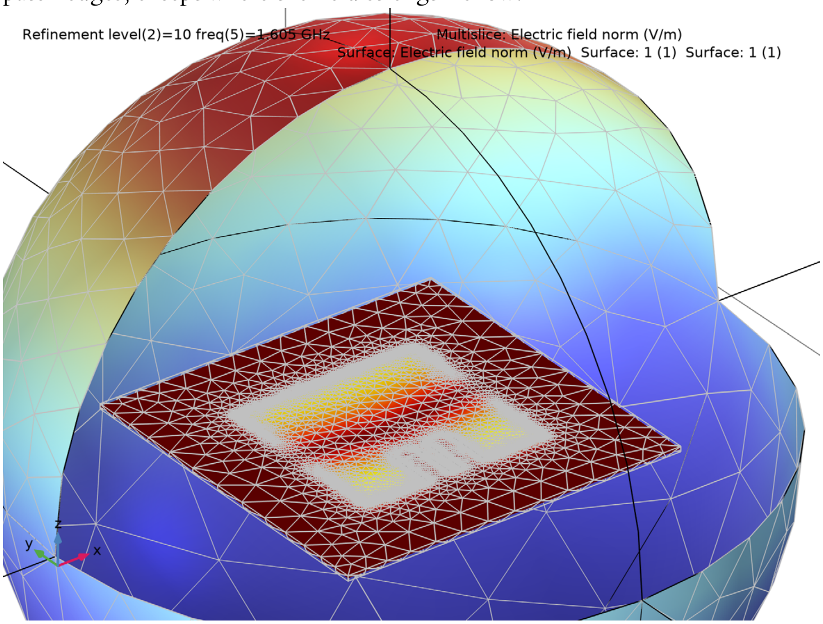


Figure 10: The mesh after running the Frequency Domain, RF Adaptive Mesh study.

Figure 11 shows how the mesh adaptation converges to reach the termination tolerance of 0.02.

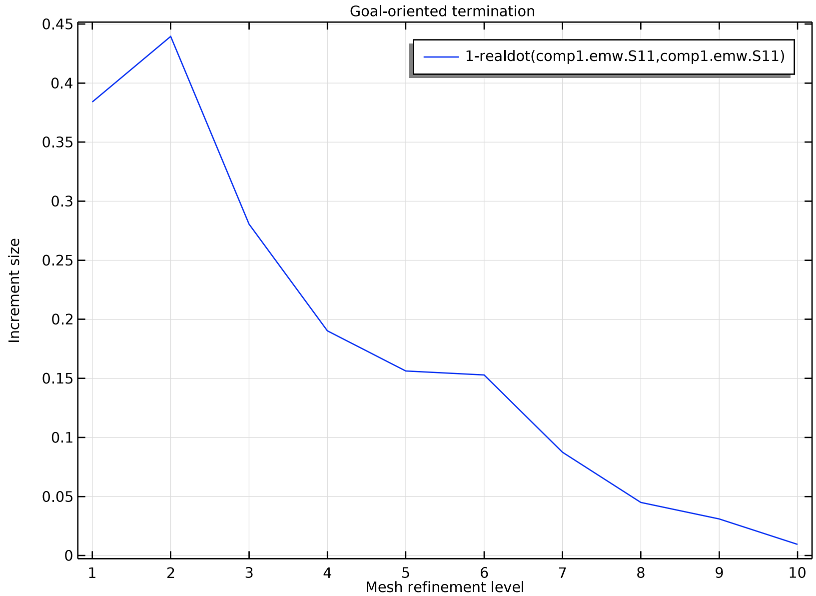


Figure 11: This plot shows the convergence of the mesh adaptation process. The process stops when the tolerance of 0.02 is reached.

Finally, Figure 12 shows that the resonance shifts to higher frequencies when using the finer adapted mesh.

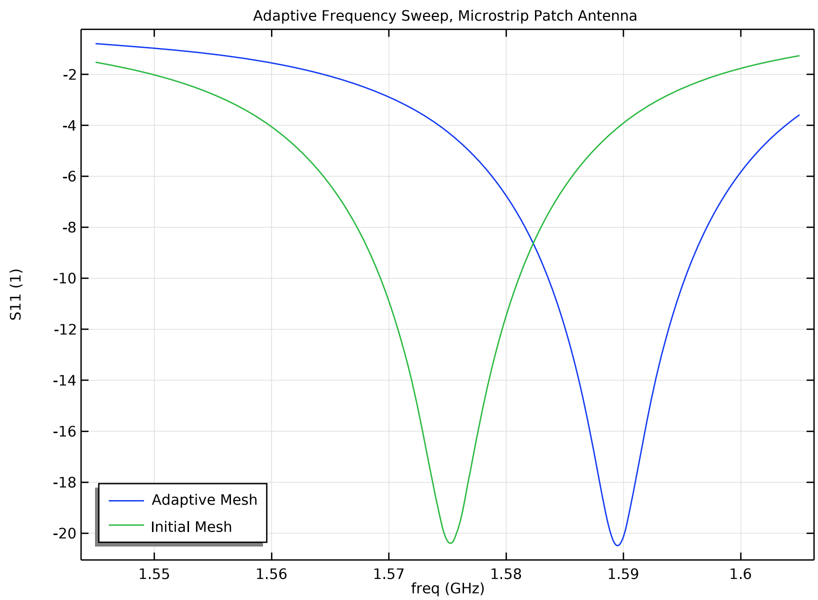


Figure 12: A comparison of the frequency spectra for the initial mesh and the adapted mesh.

Notes About the COMSOL Implementation

This example uses the Adaptive Frequency Sweep study step, based on a model order reduction technique, asymptotic waveform evaluation (AWE), to compute the frequency response of the antenna with a fine frequency resolution. This approach is faster than a regular frequency sweep performed in a Frequency Domain study using the same fine frequency resolution, but the analysis is computationally intensive, and it may require more than 5 GB of RAM when running the Adaptive Frequency Sweep. The computed results could be sensitive to the relative tolerance value in the settings window. To enhance the accuracy, a finer value can be applied.

Application Library path: `RF_Module/Antenna_Arrays/microstrip_patch_antenna_inset`

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 `1.575[GHz]`.

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
d	60[mil]	0.001524 m	Substrate thickness
w_line	3.2[mm]	0.0032 m	50 ohm line width
w_patch	53[mm]	0.053 m	Patch width
l_patch	52[mm]	0.052 m	Patch length
w_stub	7[mm]	0.007 m	Tuning stub width
l_stub	15.5[mm]	0.0155 m	Tuning stub length
w_sub	100[mm]	0.1 m	Substrate width
l_sub	100[mm]	0.1 m	Substrate length
freq_min	1.545[GHz]	1.545E9 Hz	Minimum frequency
freq_max	1.605[GHz]	1.605E9 Hz	Maximum frequency
lda_min	c_const/freq_max	0.18679 m	Minium wavelength

Here mil refers to the unit milliinch, that is 1 mil = 0.0254 mm.

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 the substrate block.

Substrate

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

2 In the **Settings** window for **Block**, type Substrate in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type w_sub.

4 In the **Depth** text field, type l_sub.

5 In the **Height** text field, type d.

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

7 Click **Build Selected**.

Now add the patch antenna.

Patch

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

2 In the **Settings** window for **Block**, type `Patch` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_patch`.

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

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

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

7 Click **Build Selected**.

Create impedance matching parts and a 50Ω feed line.

Stub

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

2 In the **Settings** window for **Block**, type `Stub` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_stub`.

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

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

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

7 In the **x** text field, type `w_stub/2+w_line/2`.

8 In the **y** text field, type `l_stub/2-l_patch/2`.

9 Click **Build Selected**.

Copy 1 (copy1)

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

2 Select the object **blk3** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `-w_stub-w_line`.

5 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **blk2** only.

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

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

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

6 Click **Build Selected**.

Choose wireframe rendering to get a better view of the interior parts.

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

Continue with the surrounding air and the PML regions.

Sphere 1 (sph1)

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

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

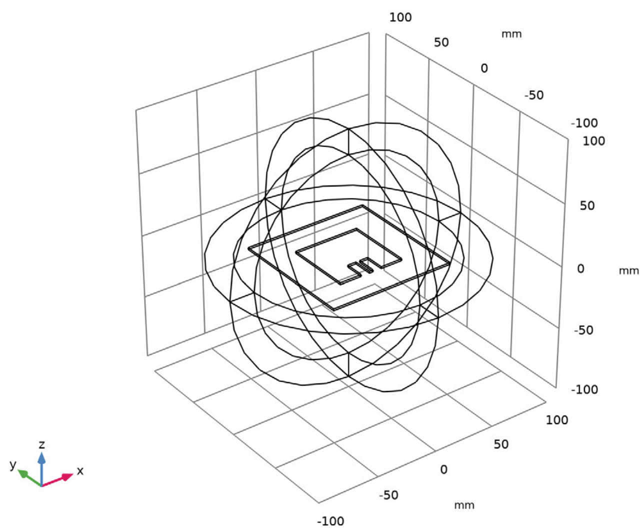
3 In the **Radius** text field, type `l_sub`.

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

Layer name	Thickness (mm)
`Layer 1`	`l_sub/5`

5 Click **Build All Objects**.

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



DEFINITIONS

Now, add some selections that later will be used when defining selections for materials, physics features, and plots.

Lumped Port

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

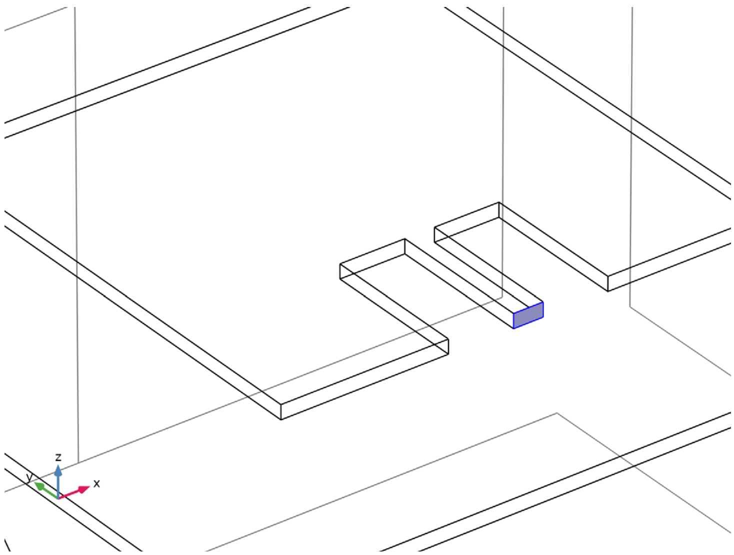
2 In the **Settings** window for **Explicit**, type `Lumped Port` in the **Label** text field.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `26` in the **Selection** text field.

6 Click **OK**.

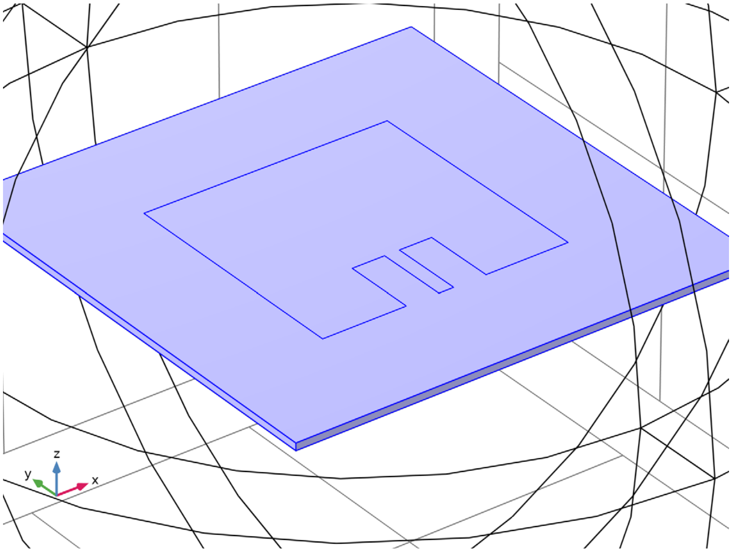


Substrate

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

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

3 Select Domains 6 and 7 only.

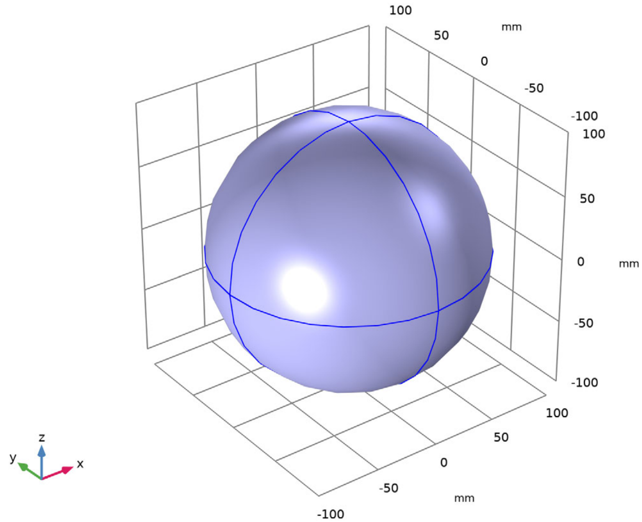


PML

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

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

3 Select Domains 1–4 and 8–11 only.



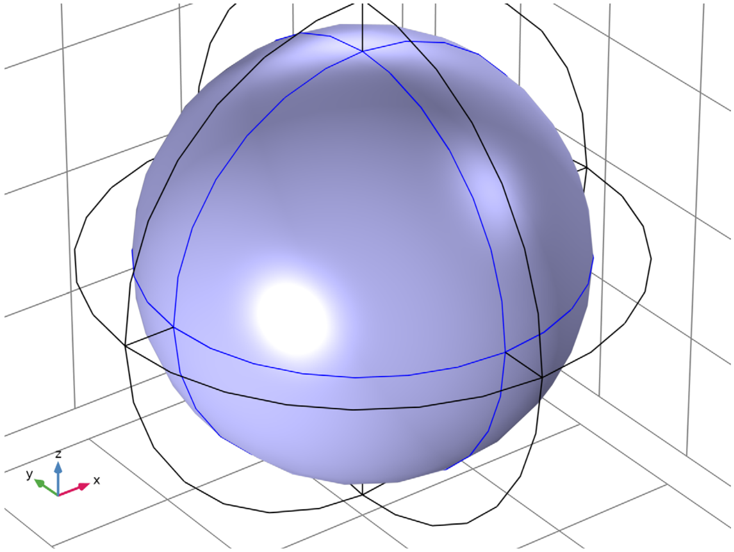
These are all of the outermost domains of the sphere.

Air

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

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

3 Select Domain 5 only.



PML, Exterior Boundaries

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

2 In the **Settings** window for **Adjacent**, type `PML, Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **PML** in the **Input selections** list.

5 Click **OK**.

Air, Exterior Boundaries

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

2 In the **Settings** window for **Adjacent**, type `Air, Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Air** in the **Input selections** list.

5 Click **OK**.

PML, Inside Boundaries

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

2 In the **Settings** window for **Intersection**, type `PML, Inside Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click + **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **PML, Exterior Boundaries** and **Air, Exterior Boundaries**.

6 Click **OK**.

Substrate Boundaries

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

2 In the **Settings** window for **Adjacent**, type `Substrate Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click + **Add**.

4 In the **Add** dialog box, select **Substrate** in the **Input selections** list.

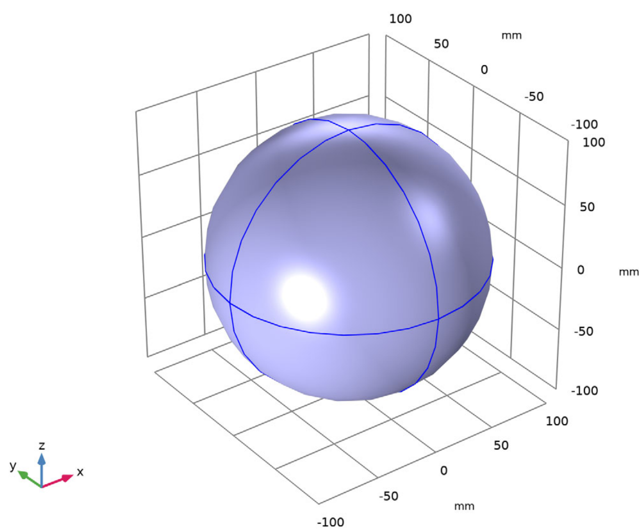
5 Click **OK**.

Perfectly Matched Layer 1 (pml1)

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

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

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



4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

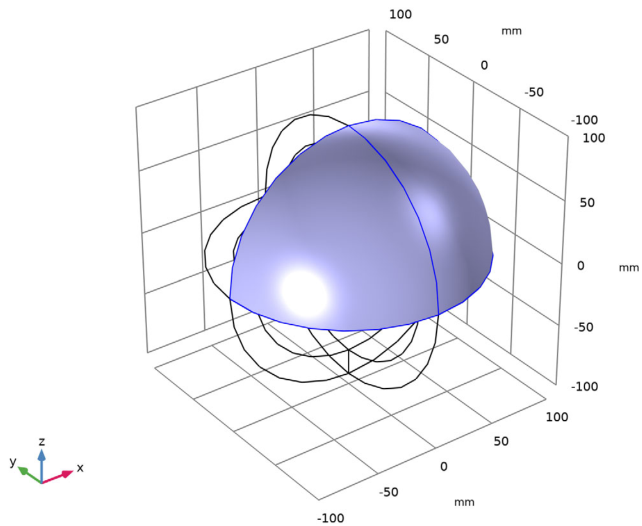
View 1

Suppress some domains and boundaries. This helps to see 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 2 and 9 only.



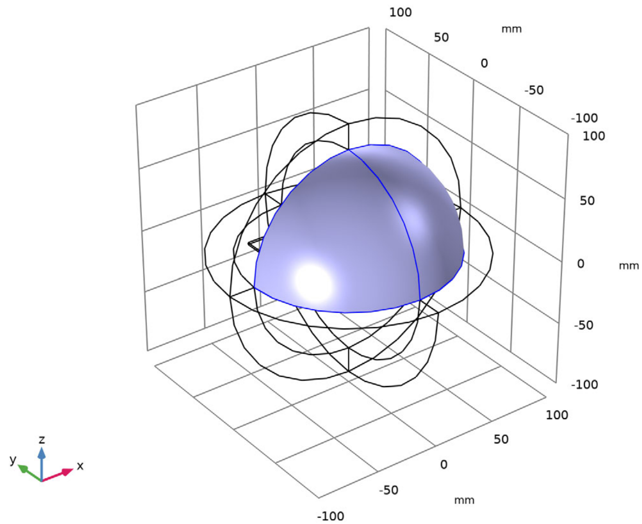
Hide for Physics 2

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 10 and 33 only.



Hidden domains and boundaries can be shown by pressing the **View All**, **View Hidden Only**, or **Reset Hiding** button in the Graphic Window 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 (EMW)

Perfect Electric Conductor 2

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

2 Select Boundaries 15, 20, and 21 only.

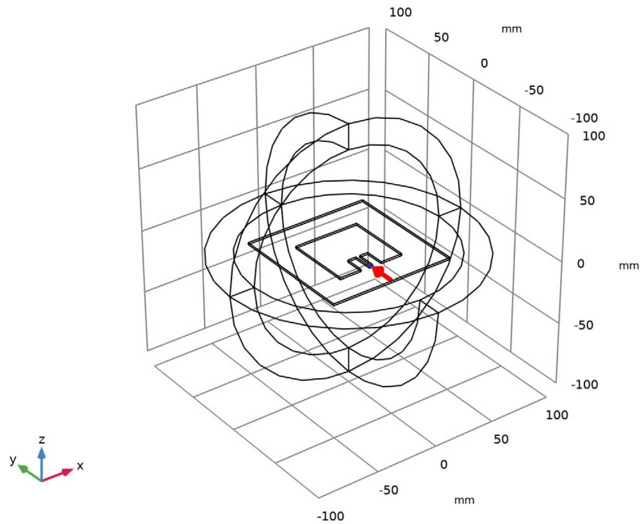
Lumped Port 1

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

2 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Lumped Port**.

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



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.

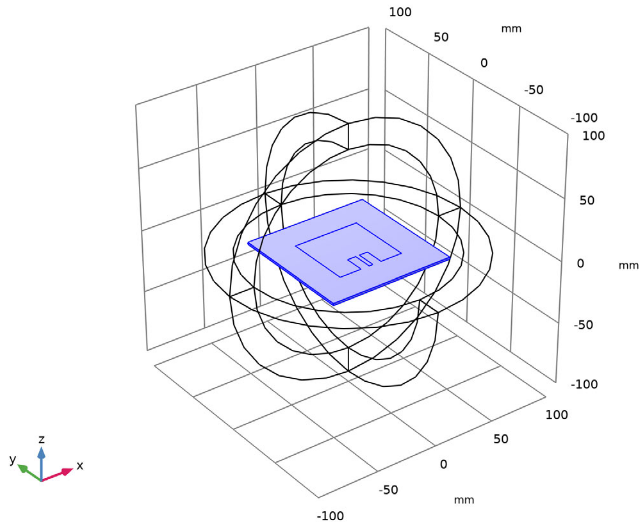
MATERIALS

Substrate

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 `Substrate` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Substrate**.

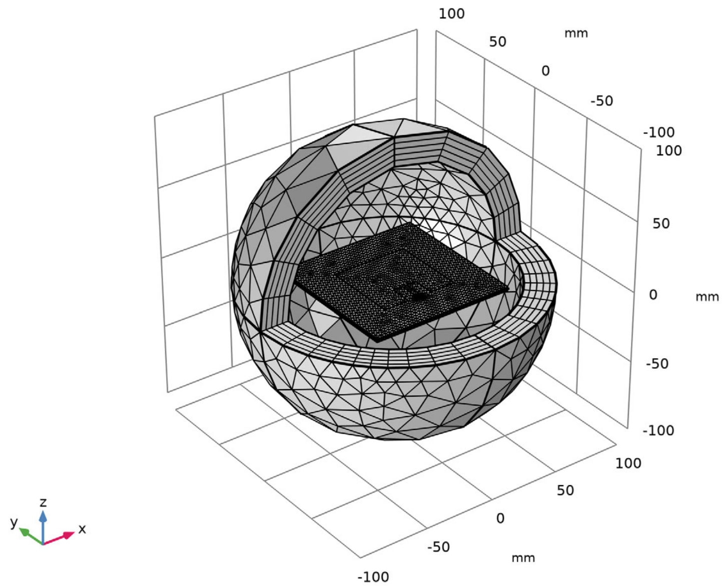


4 Locate the **Material Contents** section. In the table, enter the following settings:

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

Multislice

1 In the **Model Builder** window, expand the **Results>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 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalDark**.

Selection 1

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

2 Select Domains 6 and 7 only.

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 **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Arrow Volume 1

1 Right-click **Electric Field (emw)** and choose **Arrow Volume**.

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

3 In the **Model Builder** window, click **Arrow Volume 1**.

4 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.

5 Find the **X grid points** subsection. In the **Points** text field, type `1`.

6 Find the **Y grid points** subsection. In the **Points** text field, type `31`.

7 Find the **Z grid points** subsection. In the **Points** text field, type `31`.

8 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

Selection 1

1 Right-click **Arrow Volume 1** and choose **Selection**.

2 Select Domain 5 only.

Color Expression 1

1 In the **Model Builder** window, right-click **Arrow Volume 1** and choose **Color Expression**.

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

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

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

Strong electric fields are observed on the radiating edges. See Figure 3 and check the dominant polarization at the boresight.

2D Far Field (emw)

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

2 In the **Settings** window for **Polar Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Far-field gain, dBi`.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **2D Far Field (emw)** node, then click **Radiation Pattern 1**.

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.gaindBEfar - Far-field gain, dBi**.

3 Locate the **Evaluation** section. 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 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`E-plane`

Radiation Pattern 2

1 Right-click **Results>2D Far Field (emw)>Radiation Pattern 1** and choose **Duplicate**.

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

3 Find the **Normal vector** subsection. In the **x** text field, type `0`.

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

5 Find the **Reference direction** subsection. In the **x** text field, type `1`.

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

7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
`H-plane`

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

This is the far-field gain patterns on the E- and H-plane (Figure 4). The E- and H-plane of a linearly polarized antenna are defined by the dominant polarization at the boresight. The E-plane includes the main polarization, E_y in this model, while the H-plane is perpendicular to the main polarization.

Radiation Pattern 1

The default 3D far-field plot evaluates the norm of the electric far field that is calculated from the near field using the Stratton-Chu formula. When the 3D far-field is visualized and

Compute directivity is on, it also calculates the maximum directivity of an antenna. The default **Directivity expression** for **Electromagnetic Waves, Frequency Domain** is set to `emw.normEfar^2`, since antenna directivity is defined by the maximum radiation intensity, that is, the maximum power density per unit solid angle. The directivity calculation is used not only for electromagnetics but also for other physics such as acoustics, where the **Directivity expression** has a different input expression. The calculated maximum directivity value for the microstrip patch antenna is around 6.9 dB.

Annotation 1

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

2 Right-click **3D Far Field, Gain (emw)** and choose **Annotation**.

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

4 In the **Text** text field, type `Maximum directivity: 6.9 dB`.

5 Locate the **Position** section. In the **Z** text field, type `1.71`.

The location is set based on the maximum value of `normEfar` at the antenna boresight.

6 Locate the **Coloring and Style** section. From the **Background color** list, choose **White**.

7 Select the **Show frame** check box.

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

Compare the 3D far-field radiation pattern plot with Figure 5.

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

3D Plot Group 4

1 In the **Model Builder** window, expand the **Results>Derived Values** node.

2 Right-click **Results** and choose **3D Plot Group**.

Isosurface 1

1 In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Isosurface**.

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

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

4 Locate the **Levels** section. In the **Total levels** text field, type `20`.

Selection 1

1 Right-click **Isosurface 1** and choose **Selection**.

2 Select Domains 5–7 only.

Filter 1

1 In the **Model Builder** window, right-click **Isosurface 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `x>0`.

4 In the **3D Plot Group 4** toolbar, click **Plot**.

Figure 6 shows the above isosurface plot.

Note that the following simulation requires more than 5 GB RAM.

In order to have the S-parameter plot of the microstrip patch antenna with a fine frequency resolution, analyze the model using **Adaptive Frequency Sweep** based on asymptotic waveform evaluation (AWE). When a device presents a slowly varying frequency response, the AWE solver is much faster than the regular frequency domain solver, when running the simulation for many frequency points.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Adaptive Frequency Sweep**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

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

3 Clear the **Generate default plots** check box.

Step 1: Adaptive Frequency Sweep

1 In the **Model Builder** window, under **Study 2** click **Step 1: Adaptive Frequency Sweep**.

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

3 In the **Frequencies** text field, type `range(freq_min,100[kHz],freq_max)`.

A slowly varying scalar value curve works well for AWE expressions. For one-port devices like antennas, a trivial AWE expression is `S11`. However, if the frequency response of the AWE expression contains an infinite gradient — the case for the S_{11} value of an antenna, with excellent impedance matching at a single frequency point — the simulation will take longer to complete. If the loss from the antenna is negligible, an alternative expression such as `sqrt(1-abs(comp1.emw.S11)^2)` may work well and reduce the computation time. When **AWE expression type** is set to **Physics controlled** in the **Adaptive Frequency Sweep** study settings, `sqrt(1-abs(comp1.emw.S11)^2)` is used automatically for one-port devices.

Because such a fine frequency step generates a memory-intensive solution, the model file size will increase tremendously when it is saved. When only the frequency response of port related variables is of interest, it is not necessary to store all of the field solutions. By selecting the **Store fields in output** check box in the **Values of Dependent Variables** section, we can control the part of the model on which the computed solution is saved. We only add the selection containing the boundaries where the port variables are calculated. The lumped port size is typically very small compared to the entire modeling domain, and the saved file size with the fine frequency step is more or less that of the regular discrete frequency sweep model when only the solutions on the lumped port boundaries are stored.

4 Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, select **Lumped Port** in the **Selections** list.

7 Click **OK**.

It is necessary to include the lumped port boundaries to calculate the S-parameters. By choosing only the lumped port boundaries for **Store fields in output** settings, it is possible to significantly reduce the size of the model file.

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

RESULTS

S-parameter, Asymptotic Waveform Evaluation

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

2 In the **Settings** window for **1D Plot Group**, type `S-parameter, Asymptotic Waveform Evaluation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Adaptive Frequency Sweep, Microstrip Patch Antenna`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **S-parameter, Asymptotic Waveform Evaluation** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Ports>emw.S11dB - S11**.

3 In the **S-parameter, Asymptotic Waveform Evaluation** toolbar, click **Plot**.

Review the S-parameter plot in Figure 7.

The following instructions show how to perform a quick evaluation of the far-field radiation pattern of an antenna array using the uniform array factor operator.

3D Far Field, Virtual Array

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `3D Far Field, Virtual Array` in the **Label** text field.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Radiation Pattern 1

1 In the **3D Far Field, Virtual Array** toolbar, click **More Plots** and choose **Radiation Pattern**.

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

3 In the **Expression** text field, type `emw.gaindBEfar+20*log10(emw.af3(8,8,1,0.48,0.48,0,0,0,0))+10*log10(1/64)`.

See the Results and Discussion section for the usage of the uniform array factor operator `af3`.

4 Select the **Threshold** check box.

5 In the associated text field, type `-30`.

6 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of elevation angles** text field, type `180`.

7 In the **Number of azimuth angles** text field, type `180`.

8 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCamera**.

9 In the **3D Far Field, Virtual Array** toolbar, click **Plot**.

The far-field radiation pattern of a virtual 8x8 microstrip patch antenna array is plotted in Figure 8.

2D Far Field Gain (dB), Virtual Array

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

2 In the **Settings** window for **Polar Plot Group**, type `2D Far Field Gain (dB), Virtual Array` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `8x8 Microstrip Patch Antenna Array`.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **r minimum** text field, type `-15`.

7 In the **r maximum** text field, type `25`.

8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Radiation Pattern 1

1 In the **2D Far Field Gain (dB), Virtual Array** toolbar, click **More Plots** and choose **Radiation Pattern**.

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

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

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of angles** text field, type `360`.

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

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

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Single patch antenna gain`

10 In the **2D Far Field Gain (dB), Virtual Array** toolbar, click **Plot**.

Radiation Pattern 2

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

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

3 In the **Expression** text field, type `20*log10(emw.af3(8,8,1,0.48,0.48,0,-2*pi*0.48*cos(pi/3),0,0))+10*log10(1/64)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`8x8 uniform array factor`

5 In the **2D Far Field Gain (dB), Virtual Array** toolbar, click **Plot**.

Radiation Pattern 3

1 Right-click **Radiation Pattern 2** and choose **Duplicate**.

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

3 In the **Expression** text field, type `emw.gaindBEfar+20*log10(emw.af3(8,8,1,0.48,0.48,0,-2*pi*0.48*cos(pi/3),0,0))+10*log10(1/64)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`8x8 patch antenna array gain`

5 In the **2D Far Field Gain (dB), Virtual Array** toolbar, click **Plot**.

See Figure 9 for the dB-scaled gain of the virtual 8x8 microstrip patch antenna array. It is plotted with the uniform array factor which has the maximum radiation at 60 degrees from the *x*-axis.

GEOMETRY 1

Next, perform a Frequency Domain, RF Adaptive Mesh study. To improve the mesh adaptation process, the thickness of the PML should be of the same order as the wavelength. Thus, start by changing the thickness of the sphere layer that represents the PML.

Sphere 1 (sph1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Sphere 1 (sph1)**.

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

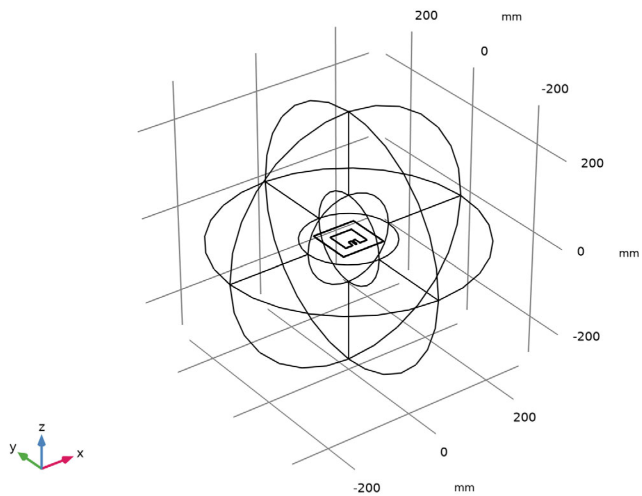
3 In the **Radius** text field, type `l_sub+lda_min`.

4 Locate the **Layers** section. In the table, enter the following settings:

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

5 Click **Build All Objects**.

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



ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Frequency Domain, RF Adaptive Mesh**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain, RF Adaptive Mesh

1 In the **Settings** window for **Frequency Domain, RF Adaptive Mesh**, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

4 In the **Start** text field, type `freq_min`.

5 In the **Stop** text field, type `freq_max`.

6 In the **Number of values** text field, type `5`.

7 Click **Replace**.

8 In the **Model Builder** window, click **Study 3**.

9 In the **Settings** window for **Study**, locate the **Study Settings** section.

10 Clear the **Generate default plots** check box, as the default S-parameter and far-field plots are not of interest for the mesh adaptation study. However, in the following steps a field plot is built, so the mesh adaptation progress can be followed while solving.

Solution 3 (sol3)

In the **Study** toolbar, click **Show Default Solver**, to create the dataset for this study. The field plot that will be created will refer to this dataset.

RESULTS

Electric Field (emw), Mesh Adaptation

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Field (emw), Mesh Adaptation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3/ Adaptive Mesh Refinement Solutions 1 (sol4)**.

Multislice 1

1 In the **Electric Field (emw), Mesh Adaptation** toolbar, click **More Plots** and choose **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 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

Selection 1

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

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Substrate**.

Electric Field (emw), Mesh Adaptation

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

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

3 Clear the **Show legends** check box.

Surface 1

1 Right-click **Electric Field (emw), Mesh Adaptation** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 From the **Color table** list, choose **RainbowLight**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **PML, Inside Boundaries**.
4 In the list, choose **10 (hidden)** and **33 (hidden)**.
5 Click **Remove from Selection**, to avoid that the mesh is visualized on the boundaries that otherwise are hidden in the physics view.
6 Select Boundaries 9, 11, 12, 32, 37, and 40 only.

Surface 2

1 In the **Model Builder** window, under **Results>Electric Field (emw), Mesh Adaptation** right-click **Surface 1** and choose **Duplicate**, to add the first of two surface plots to visualize the mesh.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type 1.
4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5 From the **Color** list, choose **Gray**.
6 Select the **Wireframe** check box.

Surface 3

Right-click **Surface 2** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Selection 1**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Substrate Boundaries**.

STUDY 3

Step 1: Frequency Domain, RF Adaptive Mesh

1 In the **Model Builder** window, under **Study 3** click **Step 1: Frequency Domain, RF Adaptive Mesh**.

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

3 In the **Damping factor** text field, type `0.05`.

4 In the **Maximum number of adaptations** text field, type `20`.

5 Click to expand the **Results While Solving** section. Select the **Plot** check box.

6 From the **Plot group** list, choose **Electric Field (emw), Mesh Adaptation**.

Adjust the view, to clearly see the mesh on both the substrate and the boundary towards the PML.

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

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

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

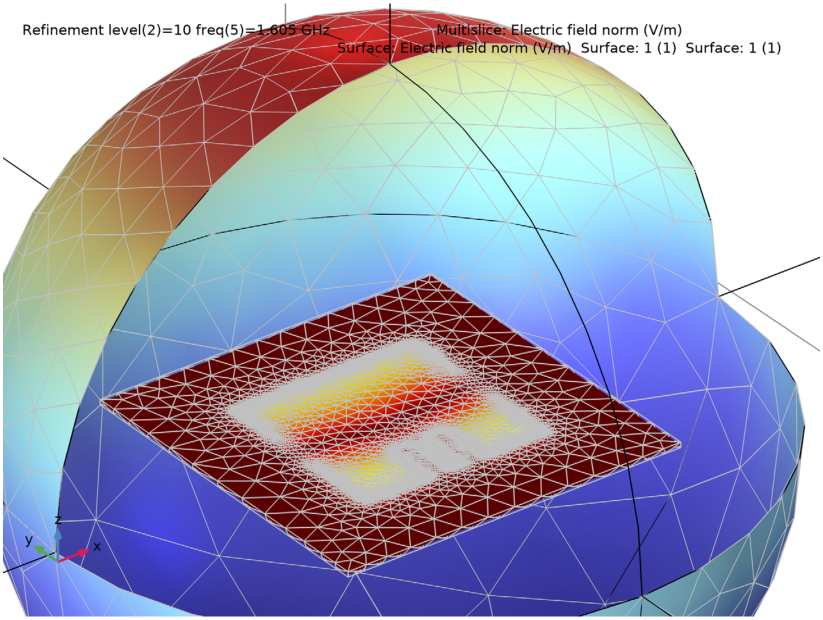
10 Click the **Zoom In** button in the **Graphics** toolbar.

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

In the Graphics window, follow how the mesh is adapted while solving.

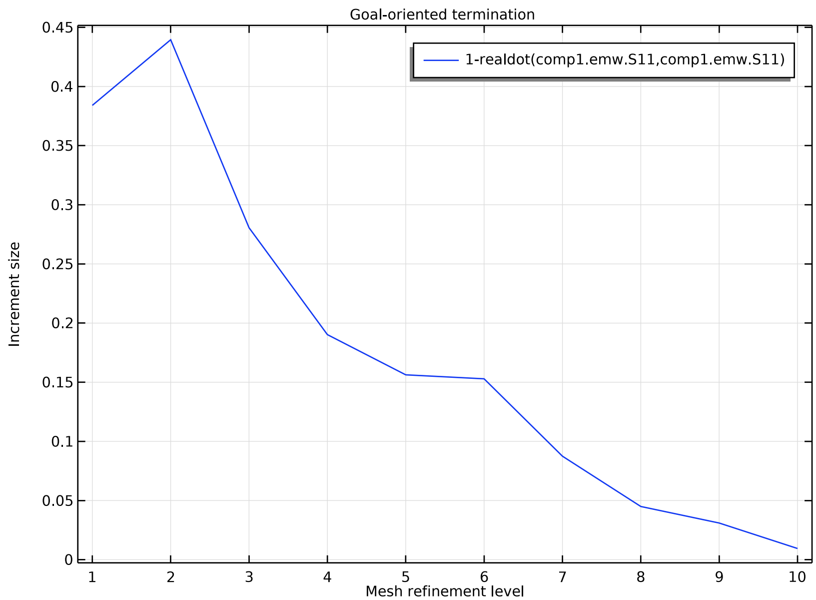
RESULTS

Electric Field (emw), Mesh Adaptation



Notice that the mesh is much denser in the adapted mesh around the patch edges, compared to the original mesh, except where the field strength is low.

Goal-oriented termination expression 1D Plot Group 9



This plot shows how the mesh adaptation converges and stops when the increment size is less than the tolerance of 0.02.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Adaptive Frequency Sweep**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

1 In the **Model Builder** window, click **Study 4**.

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

3 Clear the **Generate default plots** check box, as we again will not be saving the field in this study. Thereby, it doesn't make sense generating the default field plots.

Step 1: Adaptive Frequency Sweep

1 In the **Model Builder** window, under **Study 4** click **Step 1: Adaptive Frequency Sweep**.

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

3 In the **Frequencies** text field, type `range(freq_min,100[kHz],freq_max)`.

4 Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, select **Lumped Port** in the **Selections** list.

7 Click **OK**.

Again, only include the lumped port boundaries to calculate S-parameters, to reduce the size of the model.

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

RESULTS

S-parameter, Asymptotic Waveform Evaluation on Adaptive Mesh

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

2 In the **Settings** window for **1D Plot Group**, type `S-parameter, Asymptotic Waveform Evaluation on Adaptive Mesh` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4/Solution 16 (sol16)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Adaptive Frequency Sweep, Microstrip Patch Antenna`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **S-parameter, Asymptotic Waveform Evaluation on Adaptive Mesh** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Ports>emw.S11dB - S11**.

3 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Adaptive Mesh`

5 Right-click **Global 1** and choose **Copy**.

Global 2

1 In the **Model Builder** window, right-click **S-parameter, Asymptotic Waveform Evaluation on Adaptive Mesh** and choose **Paste Global**.

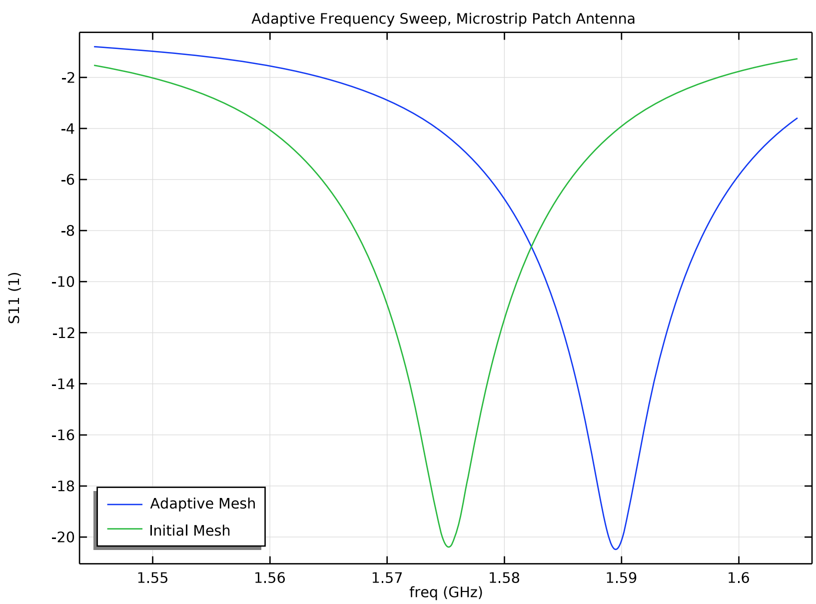
2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Initial Mesh`

5 In the **S-parameter, Asymptotic Waveform Evaluation on Adaptive Mesh** toolbar, click **Plot**.



For the denser, adapted mesh, the resonance has shifted to a slightly higher frequency.