Created in COMSOL Multiphysics 5.2a

Balanced Patch Antenna for 6 GHz

Introduction

Patch antennas are becoming more common in wireless equipment, like wireless LAN access points, cellular phones, and GPS handheld devices. The antennas are small in size and can be manufactured with simple and cost-effective techniques. Due to the complicated relationship between the geometry of the antenna and the electromagnetic fields, it is difficult to estimate the properties of a certain antenna shape. At the early stages of antenna design the engineer can benefit a lot from using computer simulations. The changes in the shape of the patch are directly related to the changes in radiation pattern, antenna efficiency, and antenna impedance.

Balanced antennas are fed using two inputs, resulting in less disturbances on the total system through the ground. Balanced systems also provide a degree of freedom to alter antenna properties, by adjusting the phase and magnitude of the two input signals. Figure 1 shows the antenna that this example simulates.

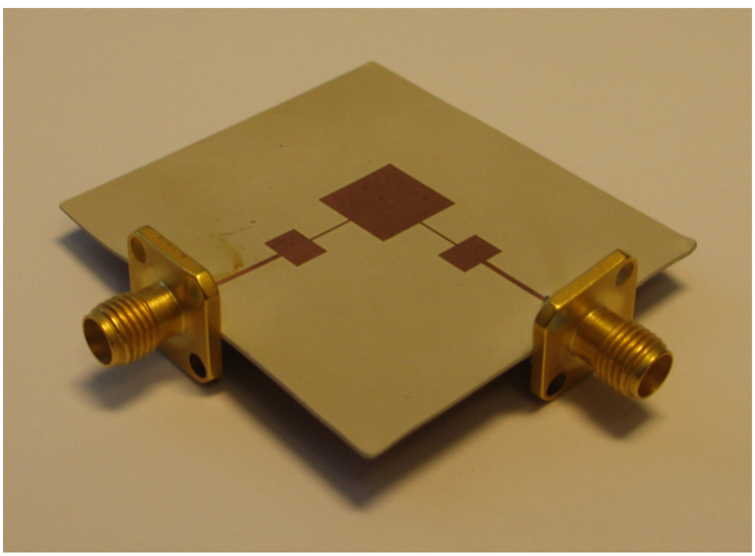


Figure 1: A photo of the real antenna that the model extracts the properties for.

Model Definition

The patch antenna is fabricated on a printed circuit board (PCB) with a relative dielectric constant of 5.23 (Ref. 1). The entire backside is covered with copper, and the front side has a pattern as shown in Figure 2 below.

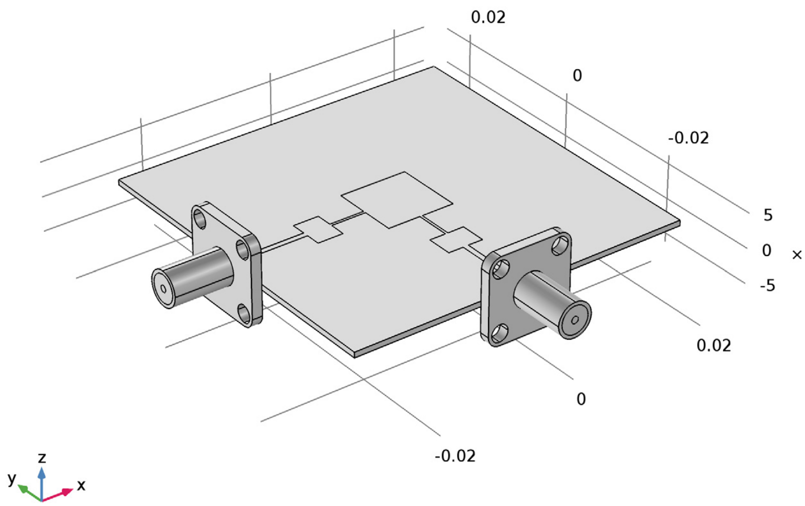


Figure 2: The patch antenna. The PCB is has a side length of 50 mm and a thickness of 0.7 mm. The centered printed square is 10 mm by 10 mm, the smaller rectangles are 5.2 mm by 3.8 mm, the thicker lines are 0.6 mm wide, and the thinner lines are 0.2 mm by 5.2 mm.

The coaxial cables have an outer conductor with an inner diameter of 4 mm and a center conductor with a diameter of 1 mm. The gap between the conductors is filled with a material with a dielectric constant of 2.07, giving a characteristic impedance close to 58 Ω. There are two coaxial cables feeding the patch antenna from two sides. In this example, the signals in the cables have the same magnitude but are shifted 180 degrees in phase. This results in a balanced feed.

The entire antenna is modeled in 3D. The time-harmonic nature of the signals makes it possible to solve the vector-Helmholtz equation for the electric field everywhere in the geometry,

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{E}) - k_0^2 \varepsilon_r \mathbf{E} = 0$$

where k_0 is the wave number for free space and is defined as

$$k_0 = \omega \sqrt{\varepsilon_0 \mu_0}$$

All metallic objects are defined as perfect electric conductors. The antenna is placed in a spherical air domain surrounded by a Perfectly Matched Layer (PML) serving to absorb the radiation from the antenna with a minimum of reflection. In addition, a scattering boundary condition is added outside the PML to further reduce the reflection.

The model is run through a range of frequencies surrounding the operational frequency of 6.28 GHz.

Results and Discussion

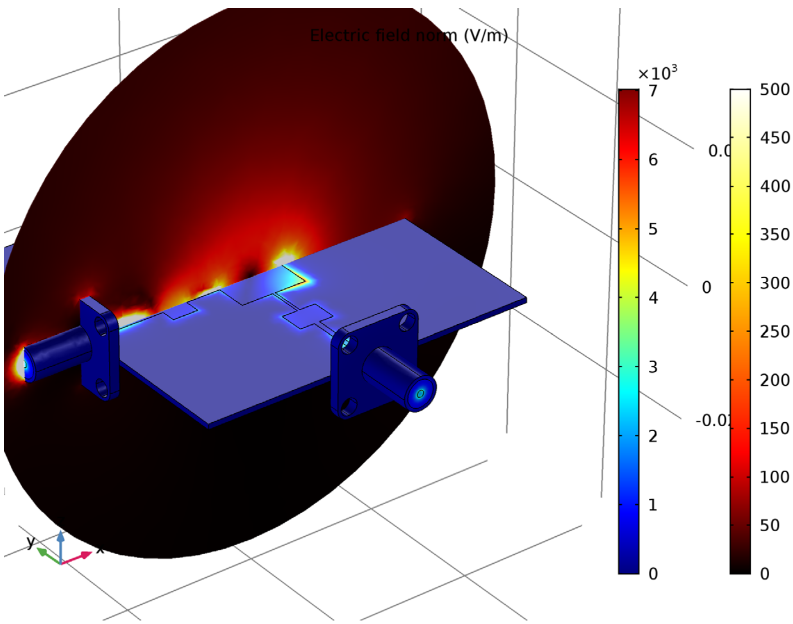


Figure 3: The patch antenna with the electric field plotted both on its surface and on a slice through the air domain. The surrounding PML is hidden from view.

Figure 3 shows the distribution of the electric field norm on the surface of the antenna and in the air, at 6.26 GHz. Most of the energy radiates out from the central patch.

The Lumped Port boundary condition, which is applied to the coaxial cables, is mimicking a connection to a transmission line feed with a characteristic impedance, Z_{ref}. The incident voltage wave from the transmission line has an amplitude equal to V_0, part of which is reflected directly at the port depending on how well Z_{ref} matches the characteristic impedance of the coaxial cable.

Under these circumstances and from each coaxial cable, the theoretical maximum power that can be produced in the antenna is achieved when the antenna impedance matches that of the coaxial cable. This power evaluates to

$$P_{\max} = \frac{V_0^2}{2Z_{\mathrm{ref}}}$$

where V_0 is the peak value of the time-harmonic applied voltage.

The antenna efficiency η is defined as the fraction of the theoretical max power that actually radiates out of the antenna:

$$\eta = \frac{P_1 + P_2}{2P_{\max}}$$

where P_1 and P_2 are the net power flow through ports 1 and 2 respectively. In Figure 4 this efficiency is plotted against the frequency, showing that the optimum operating frequency is located at 6.26 GHz.

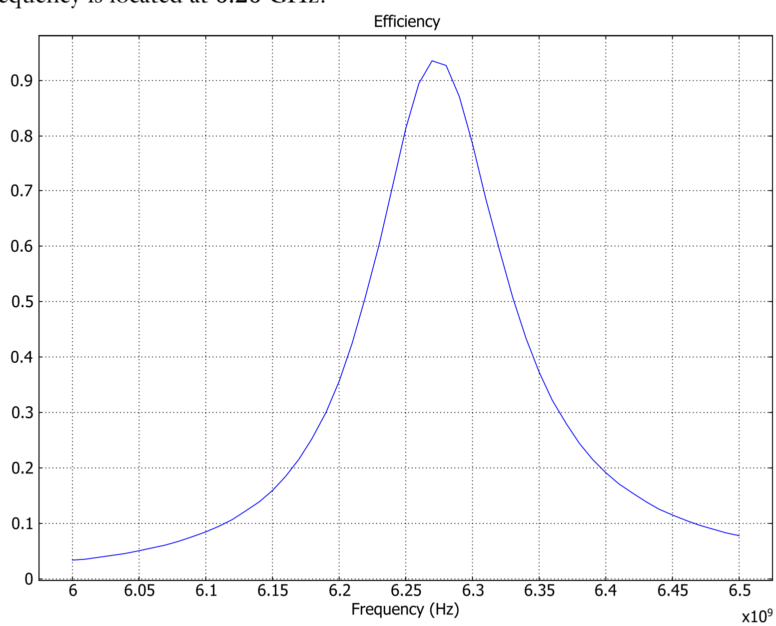


Figure 4: The antenna efficiency as a function of the frequency.

Application Library path: `RF_Module/Antennas/patch_antenna`

Notes About the COMSOL Implementation

This example uses a mesh resulting in almost 500,000 complex-valued degrees of freedom. It therefore needs a little bit more than 2 GB of memory and should be solved on a 64-bit platform. You can make it solve on a 32-bit computer with a coarser mesh, but the results are less accurate.

Possible model extensions include the addition of an external circuit or a far-field computation.

Reference

1. E. Recht and S. Shiran, "A Simple Model for Characteristic Impedance of Wide Microstrip Lines for Flexible PCB," *Proceedings of IEEE EMC Symposium 2000*, pp. 1010–1014, 2000.

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 **Preset Studies>Frequency Domain**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
V0	1[V]	1 V	Applied voltage
epsilonr_coax	2.07	2.07	Relative permittivity, coaxial cable
epsilonr_pcb	5.23	5.23	Relative permittivity, circuit board
a_coax	0.5[mm]	5E-4 m	Inner coax conductor radius
b_coax	2[mm]	0.002 m	Inner radius of outer coax conductor
Z_coax	sqrt(mu0_const/(epsilonr_coax*epsilon0_const))/(2*pi)*log(b_coax/a_coax)	57.77 Ω	Cable impedance
Pmax	V0^2/(2*Z_coax)	0.008655 W	Theoretical max power
f_min	6.2[GHz]	6.2E9 Hz	Minimum frequency
f_max	6.3[GHz]	6.3E9 Hz	Maximum frequency

GEOMETRY 1

Import 1 (imp1)

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

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

3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `patch_antenna.mphbin`.

5 Click **Import**.

The imported geometry consists of the patch antenna and its connectors. Add two concentric spheres, one for the air surrounding the antenna and one for the PML.

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type `0.06`.

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

Layer name	Thickness (m)
`Layer 1`	`0.02`

5 Click **Build All Objects**.

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

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

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

Activate the PML in the volume covered by the outer but not the inner sphere:

2 Select Domains 1–4 and 13–16 only.

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

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

ADD MATERIAL

1 On 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.

MATERIALS

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 2 (mat2)

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 `Coax Dielectric` in the **Label** text field.

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

4 Click **Clear Selection**.

Select the cylinders between the inner and outer conductors of the coaxial cables:

5 Select Domains 7 and 11 only.

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

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

Material 3 (mat3)

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

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

Select the PCB board:

3 Select Domain 9 only.

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

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

By default, the Electromagnetic Waves equation is active in all domains. However, because you represent the metal in this model as perfectly conductive boundaries, there is no need to model the interior of the contacts. Therefore, remove the metal domains from the domains selection.

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 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1-5, 7, 9, 11, 13-16` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for Electromagnetic Waves, Frequency Domain, locate the **Physics-Controlled Mesh** section.

8 Select the **Enable** check box.

Control the meshing by the maximum frequency. This will set the maximum mesh size to 0.2 wavelengths or smaller.

9 From the **Maximum mesh element size control parameter** list, choose **Frequency**.

10 In the **Maximum frequency** text field, type `f_max`.

11 Locate the **Analysis Methodology** section. From the **Methodology options** list, choose **Fast**.

Lumped Port 1

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

To define the first port, select the outer air/dielectric boundary on the cable facing the x direction:

2 Select Boundary 16 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**.

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

6 In the V_0 text field, type `V0`.

7 Locate the **Settings** section. In the Z_{ref} text field, type `Z_coax`.

Lumped Port 2

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

The second port is the outer air/dielectric boundary on the contact facing the y direction:

2 Select Boundary 96 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**.

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

6 In the V_0 text field, type `V0`.

7 In the θ_{in} text field, type `pi`.

8 Locate the **Settings** section. In the Z_{ref} text field, type `Z_coax`.

Although you have not yet specified any conducting boundaries, there is already a Perfect Electric Conductor condition in the model. By default, it applies to all boundaries that are exterior to the active domains. It then gets over-ridden by any other conditions that you are applying. If you click its node in the Model Builder, you can see that it still applies to the conductors.

Perfect Electric Conductor 1

The patch and the PCB ground plane are interior to the model domain (meaning they neighbor only to domains where the equation is active) and hence need to be explicitly assigned this same condition.

Perfect Electric Conductor 2

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

Select the patch and the ground plane (bottom surface) of the PCB:

2 Select Boundaries 59 and 69 only.

The settings that you have made until now completely define the physics of your model. To enable postprocessing of the antenna efficiency, you need to add integral operators on the port boundaries.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
P1	0.5*real(emw.Vport_1*conj(emw.Iport_1))	W	Power into Port 1
P2	0.5*real(emw.Vport_2*conj(emw.Iport_2))	W	Power into Port 2
eff	(P1+P2)/(2*Pmax)		Antenna efficiency

The power that goes through each of the ports is computed from port voltage and current. The last variable defines the efficiency as the ratio of the input power and the theoretical maximum for each port.

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.

STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `range(f_min, (f_max-f_min)/5, f_max)`.

This gives you 6 linearly spaced frequencies between 6.2 and 6.3 GHz. If you want to reproduce the plot in Figure 4 and are prepared to let the model run for a while, try range(6.0e9,0.01e9,6.5e9) instead.

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

RESULTS

Electric Field (emw)

The default plot shows a slice plot of the electric field norm at 6.3 GHz. It is dominated by the result near the antenna. Most of the remaining part of these model instructions will guide you towards an informative and nice-looking plot of the local electric field on and around the antenna. But first, take the following steps in order to plot the antenna efficiency versus the frequency.

1D Plot Group 2

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

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

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

4 In the **Title** text area, type `Efficiency`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Frequency (Hz)`.

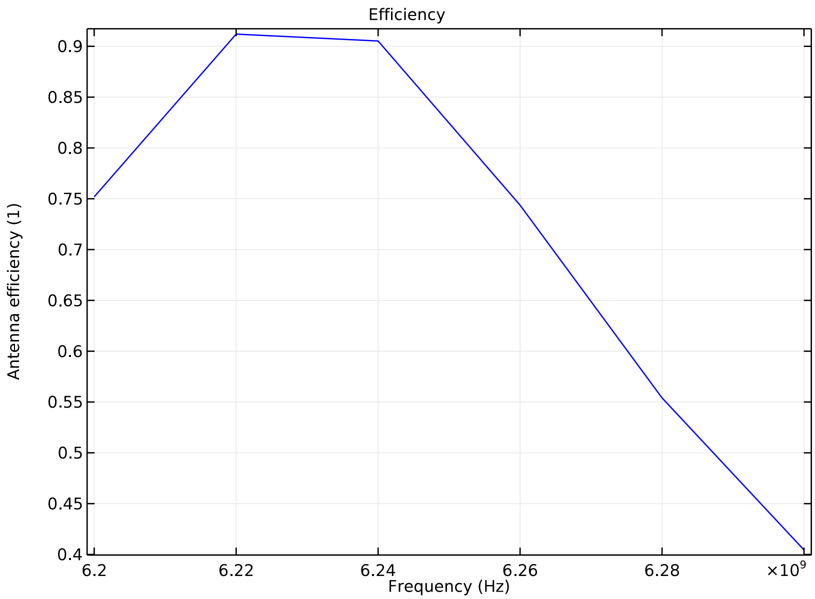
Global 1

1 On the **1D Plot Group 2** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>eff - Antenna efficiency**.

3 Click to expand the **Legends** section. Clear the **Show legends** check box.

4 On the **1D Plot Group 2** toolbar, click **Plot**.



The plot has sharp edges because you solved only for 6 frequencies. See Figure 4 for a smoother version over a wider frequency range.

Data Sets

In order to prepare for the 3D plot, define selections of the domains, boundaries, and edges that you want the plot to include. As everything is included per default, you will make these selections with the purpose of hiding what you do not select.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for Selection, type `Physical Domain` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5-12` in the **Selection** text field.

6 Click **OK**.

Study 1/Solution 1 (2) (sol1)

On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `13-106, 114-121, 131-144` in the **Selection** text field.

6 Click **OK**.

Study 1/Solution 1 (3) (sol1)

On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for Selection, type `Physical Edges` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Edge**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `10-235, 243-264, 278-335` in the **Selection** text field.

6 Click **OK**.

Electric Field (emw)

The plot group you just selected already contains a slice plot of the electric field norm. Note that because it uses the data set for which you defined the domain selection, the plot does not show up in the PML.

Multislice

Delete the multislice plot and add a single slice.

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

2 Right-click **Multislice** and choose **Delete**.

Electric Field (emw)

In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Slice**.

Slice 1

1 In the **Settings** window for Slice, locate the **Plane Data** section.

2 From the **Plane** list, choose **zx-planes**.

3 In the **Planes** text field, type 1.

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

5 In the **Minimum** text field, type 0.

6 In the **Maximum** text field, type 500.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

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

   You are now looking at a nicely scaled plot of the electric field norm on a slice of your geometry, excluding the PML where it does not have any physical relevance.

Electric Field (emw)

Right-click **Electric Field (emw)** and choose **Surface**.

Surface 1

1 In the **Settings** window for Surface, locate the **Data** section.

2 From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

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

   The electric field norm now also shows up on the surface of the antenna. All exterior surfaces are hidden from view, but the edges defining the contour of the PML are still visible.

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 data set edges** check box.

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

   Now all edges are gone. This makes the contours of the PCB and the contacts less prominent. To retain a sharper-looking geometry, draw your selected edges in black with the help of a line plot.

Line 1

1 Right-click **Results>Electric Field (emw)** and choose **Line**.

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

3 From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.
5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
6 From the **Color** list, choose **Black**.
7 On the **Electric Field (emw)** toolbar, click **Plot**.

Electric Field (emw)

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.
2 In the **Settings** window for 3D Plot Group, click to expand the **Title** section.
3 From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Electric field norm (V/m)`.
5 On the **Electric Field (emw)** toolbar, click **Plot**.
6 Click the **Go to Default 3D View** button on the **Graphics** toolbar.
7 Click the **Zoom In** button on the **Graphics** toolbar.

Your plot should now look like that in Figure 3.