

Modeling of a Mobile Device Antenna

Introduction

Electrical components in wireless communication systems are designed to be small and light for portability and productivity while maintaining decent performance and efficiency. Antennas are essential components in mobile devices and are required to fit in the limited space allowed by industrial specifications. To fulfill this requirement, a planar inverted-F antenna (PIFA) is common and a popular choice for miniaturized antennas in cellphones. The PIFA design can be tuned and extended to cover multiple frequency bands from cellphones, Wi-Fi, and Bluetooth[®]. The antenna in this introductory example is tuned only for the Advanced Wireless Services (AWS) band downlink frequency range. The impedance matching properties of the antenna are calculated in terms of S-parameters and the far-field radiation pattern is simulated.



Figure 1: A simple PIFA on an FR4 circuit board is enclosed by an ABS package. The front part of the enclosure contains the combination of glass and a composite silicon substrate that form a touchscreen display. The surrounding air domain and perfectly matched layers, which are required for the simulation, are not included in this figure.

Model Definition

The AWS band downlink frequency range is from 2.11 GHz to 2.155 GHz. At this frequency range, the metal part of the antenna can be modeled using perfect electric

conductor (PEC) boundaries. The losses on the metal surfaces are negligible because of the high conductivity of the copper.

The FR4 circuit board with a ground plane is inserted inside an RF lossless acrylonitrile butadiene styrene (ABS) enclosure. The antenna with the cellphone mock-up case is modeled in a spherical air domain, which is enclosed by perfectly matched layers (PML) that absorb all outgoing radiation from the antenna.



Figure 2: Zoomed view of the PIFA. It consists of a radiating part, a feeding strip and an impedance matching strip shorted to the ground plane.

A lumped port with a reference impedance of 50 Ω is used to excite the antenna and evaluate the input impedance. The lumped port is assigned between two metallic boundaries: the ground plane of the FR4 board and a vertical feeding strip (Figure 2). Another strip shorted to the ground plane is added adjacent to the feeding strip for impedance matching. The distance Δ , the impedance matching gap, plays an important role for matching the antenna to the reference impedance.

Results and Discussion

Figure 3 shows the default E-field norm on the *xy*-plane where the height of the plane is adjusted to visualize the plot on the top surface of the PIFA. The field distribution plot indicates that the electric field is strong at one of the top metallic surfaces far from the

feeding point. This resembles the field distribution of a quarter-wave monopole antenna, which the PIFA evolved from.

The polar-formatted far-field radiation pattern of the antenna is shown in Figure 5. Because the antenna is miniaturized and placed on one corner of the ground plane, this azimuthal radiation pattern is not omnidirectional any more. The antenna gain on the *xy*-plane varies from -6 dBi to 2 dBi.



Figure 3: E-field norm distribution on the top of the PIFA.

The S-parameters in the given frequency range are plotted in Figure 4. All calculated S_{11} values are below -10 dB, which is a voltage standing wave ratio (VSWR) of less than 2:1. This ratio describes how well the antenna input impedance is matched to the 50 Ω reference impedance typical in common measurement systems such as network analyzers.



Figure 4: S-parameters (S_{11}) in the advanced wireless services (AWS) downlink frequency range.



Figure 5: Antenna gain pattern on the xy-plane.





The 2D far-field radiation pattern may not be sufficient to show the maximum radiation and nulls. Therefore, it is a good practice to review 3D radiation patterns as well shown in Figure 6.

Application Library path: RF_Module/Antennas/pifa_handheld

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Radio Frequency>Electromagnetic Waves, Frequency Domain (emw).
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Frequency Domain.
- 6 Click 🗹 Done.

STUDY I

Step 1: Frequency Domain

Set the simulation frequency to the range of the advanced wireless services (AWS) downlink.

- I In the Model Builder window, under Study I click Step I: Frequency Domain.
- 2 In the Settings window for Frequency Domain, locate the Study Settings section.
- 3 In the Frequencies text field, type range(2.11[GHz],5[MHz],2.155[GHz]).

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

First, import a mobile device package design created with the Design Module.

Import I (imp1)

- I In the **Home** toolbar, click া Import.
- 2 In the Settings window for Import, locate the Import section.
- 3 Click 📂 Browse.
- 4 Browse to the model's Application Libraries folder and double-click the file pifa_handheld.mphbin.
- 5 Click া Import.
- 6 Click the Wireframe Rendering button in the Graphics toolbar.

Choose wireframe rendering in order to see the interior of the package.



Add a PTFE block where the planar inverted-F antenna (PIFA) will be placed.

PTFE antenna body

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, type PTFE antenna body in the Label text field.
- 3 Locate the Size and Shape section. In the Width text field, type 15.9.
- 4 In the **Depth** text field, type 10.
- 5 In the **Height** text field, type 4.
- 6 Locate the **Position** section. In the **x** text field, type -20.
- 7 In the z text field, type 2.

8 In the y text field, type 45.

Add two strips for the feeding and matching parts of the PIFA.

Work Plane I (wp1)

- I In the Geometry toolbar, click 📥 Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 From the Plane list, choose xz-plane.
- 4 In the **y-coordinate** text field, type 45.
- 5 Click 📥 Show Work Plane.

Work Plane I (wp1)>Rectangle I (r1)

- I In the Work Plane toolbar, click Rectangle.
- **2** Click the **Zoom Extents** button in the **Graphics** toolbar.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the **Height** text field, type 5.
- 5 Locate the Position section. In the xw text field, type -19.6.
- 6 In the **yw** text field, type 1.
- 7 Click 틤 Build Selected.

Work Plane 2 (wp2)

- I In the Model Builder window, right-click Geometry I and choose Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 From the Plane type list, choose Face parallel.

4 On the object **blk1**, select Boundary 2 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



5 Click Show Work Plane.

Work Plane 2 (wp2)>Rectangle 1 (r1)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 2.
- 4 In the **Height** text field, type 5.
- 5 Locate the **Position** section. In the **xw** text field, type -5.
- 6 In the **yw** text field, type -2.
- 7 Click 틤 Build Selected.

Work Plane 2 (wp2)

I In the Model Builder window, under Component I (compl)>Geometry I click Work Plane 2 (wp2). 2 In the Settings window for Work Plane, click 📳 Build All Objects.



Finish the geometry definition by adding a sphere for a surrounding air domain, where you will configure layer settings to define perfectly matched layers later on.

Sphere I (sph1)

- I In the Geometry toolbar, click \bigcirc Sphere.
- 2 In the Settings window for Sphere, locate the Size section.
- 3 In the Radius text field, type 100.
- 4 Click to expand the Layers section. In the table, enter the following settings:

Layer name	Thickness (mm)	
Layer 1	20	

5 In the Geometry toolbar, click 🟢 Build All.

6 Click the *Q* Zoom Out button in the Graphics toolbar.



DEFINITIONS

Perfectly Matched Layer 1 (pml1) I In the Definitions toolbar, click Matched Layer. 2 Select Domains 1–4 and 13–16 only.



These are all of the outermost domains of the sphere.

- 3 In the Settings window for Perfectly Matched Layer, locate the Geometry section.
- 4 From the Type list, choose Spherical.

Suppress some boundaries to get a better view when setting up materials, physics and mesh.

DEFINITIONS

Hide for Physics 1

- I In the Model Builder window, right-click View I 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 6, 10, 69, 72, and 74 only.



ADD MATERIAL

- I 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.

MATERIALS

Air (mat1)

The entire domain is set to air. Override this setting in certain parts with different materials one by one.

ADD MATERIAL

- I Go to the Add Material window.
- 2 In the tree, select Built-in>FR4 (Circuit Board).
- **3** Click **Add to Component** in the window toolbar.
- 4 In the tree, select Built-in>Glass (quartz).
- 5 Click Add to Component in the window toolbar.
- 6 In the tree, select Built-in>Silicon.

- 7 Click Add to Component in the window toolbar.
- 8 In the Home toolbar, click 👬 Add Material to close the Add Material window.

MATERIALS

- FR4 (Circuit Board) (mat2)
- I Click the 🔍 Zoom In button in the Graphics toolbar.
- 2 In the Model Builder window, under Component I (compl)>Materials click FR4 (Circuit Board) (mat2).
- **3** Select Domain 7 only.



Glass (quartz) (mat3)I In the Model Builder window, click Glass (quartz) (mat3).

2 Select Domain 10 only.



Silicon (mat4)

- I In the Model Builder window, click Silicon (mat4).
- **2** Select Domain 9 only.



PTFE antenna block

- I In the Model Builder window, right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type PTFE antenna block in the Label text field.
- **3** Select Domain 11 only.



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

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	2.1	I	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	I	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

ABS

- I Right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type ABS in the Label text field.

3 Select Domain 6 only.



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

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso; epsilonrii = epsilonr_iso, epsilonrij = 0	2.1	I	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

Now set up the physics. By assuming the losses on metal surfaces are negligible at the simulation frequency range, all metal parts can be modeled as perfect electric conductors (PEC).

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

- In the Model Builder window, under Component I (compl) right-click
 Electromagnetic Waves, Frequency Domain (emw) and choose the boundary condition
 Perfect Electric Conductor.
- **2** Select Boundaries 26, 42, 43, 46, and 50 only.



Lumped Port 1 I In the Physics toolbar, click 🔚 Boundaries and choose Lumped Port. 2 Select Boundary 49 only.



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.

MESH I

Choose the Coarse element size to reduce the size of the problem (degrees of freedom). This will generate a less dense mesh on curved parts while the maximum element size is still forced to be smaller than 0.2 wavelengths.

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- 3 From the Element size list, choose Coarse.

4 Click 📗 Build All.



STUDY I

Step 1: Frequency Domain In the Home toolbar, click **= Compute**.

RESULTS

Electric Field (emw) Adjust the default E-field norm multislice plot.

Multislice

- I In the Model Builder window, expand the 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 Find the Z-planes subsection. From the Entry method list, choose Coordinates.
- 6 In the Coordinates text field, type 5.

7 In the Electric Field (emw) toolbar, click 💽 Plot.

Compare the reproduced plot with Figure 3.

S-parameter (emw)

The default S-parameter plot shows the impedance matching properties. See Figure 4.

Smith Plot (emw)



2D Far Field (emw)

Adjust the default polar plot settings to show the antenna gain pattern on the xy-plane.

Radiation Pattern 1

- I In the Model Builder window, expand the 2D Far Field (emw) node, then click Radiation Pattern I.
- 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 I (compl)> Electromagnetic Waves, Frequency Domain>Far field>emw.gaindBEfar Far-field gain, dBi.

2D Far Field (emw)

- I In the Model Builder window, click 2D Far Field (emw).
- 2 In the Settings window for Polar Plot Group, locate the Axis section.
- **3** Select the **Manual axis limits** check box.

- 4 In the **r minimum** text field, type -20.
- 5 In the 2D Far Field (emw) toolbar, click 💿 Plot.

This reproduces Figure 5.

Radiation Pattern 1

I Click the $\stackrel{\times}{\downarrow}$ Go to XY View button in the Graphics toolbar.

The reproduced plots are shown in Figure 6.

Grid 3D I

- I In the Model Builder window, expand the 3D Far Field, Gain (emw) node.
- 2 Right-click Results>Datasets and choose More 3D Datasets>Grid 3D.
- 3 In the Settings window for Grid 3D, locate the Parameter Bounds section.
- 4 Find the First parameter subsection. In the Minimum text field, type -10.
- **5** In the **Maximum** text field, type 10.
- 6 Find the Second parameter subsection. In the Minimum text field, type 150.
- 7 In the Maximum text field, type 150.
- 8 Find the Third parameter subsection. In the Minimum text field, type 150.
- 9 In the Maximum text field, type 150.

Using Grid 3D, you can evaluate the far-field outside the simulation domain.

3D Plot Group 6

- I In the **Results** toolbar, click 间 **3D Plot Group**.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Parameter value (freq (GHz)) list, choose 2.13.
- 4 Locate the Plot Settings section. Clear the Plot dataset edges check box.

Surface 1

- I Right-click **3D Plot Group 6** and choose **Surface**.
- 2 In the Settings window for Surface, locate the Data section.
- 3 From the Dataset list, choose Grid 3D I.
- 4 Locate the **Expression** section. In the **Expression** text field, type emw.gaindBEfar.
- 5 Locate the Coloring and Style section. Click Change Color Table.
- 6 In the Color Table dialog box, select Thermal>HeatCameraLight in the tree.
- 7 Click OK.

DEFINITIONS

Explicit I

- I In the **Definitions** toolbar, click http://www.explicit.
- 2 In the Settings window for Explicit, locate the Input Entities section.
- **3** From the **Geometric entity level** list, choose **Boundary**.
- 4 Select the Group by continuous tangent check box.

With **Group by continuous tangent**, you can select the surface of the phone easily. **Group by continuous tangent** allows you to select adjacent faces or edges that are continuously tangent with the angular tolerance you specified. Selecting any outer surface of the phone will automatically select all outer surfaces of the phone.

5 Select Boundaries 13–23, 39, 52–57, 65, 89–94, 97, 98, and 101 only.

RESULTS

Surface 2

- I In the Model Builder window, right-click 3D Plot Group 6 and choose Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the Expression text field, type 20*log10(emw.normE).

Selection I

- I Right-click Surface 2 and choose Selection.
- 2 In the Settings window for Selection, locate the Selection section.
- 3 From the Selection list, choose Explicit I.

Deformation I

- I In the Model Builder window, right-click Surface I and choose Deformation.
- 2 In the Settings window for Deformation, locate the Expression section.
- **3** In the **x-component** text field, type **0**.
- 4 In the y-component text field, type 55.
- 5 In the **z-component** text field, type 0.
- 6 Locate the Scale section.
- 7 Select the Scale factor check box. In the associated text field, type 1.
- 8 In the 3D Plot Group 6 toolbar, click 💽 Plot.
- **9** Click the *Q* **Zoom Out** button in the **Graphics** toolbar.





Surface: Far-field gain, dBi (1) Surface: 20*log10(emw.normE)



Analyze the same model with a much finer frequency resolution using **Adaptive Frequency Sweep** based on asymptotic waveform evaluation (AWE). When a device presents a slowly varying frequency response, the AWE provides a faster solution time when running the simulation on many frequency points. The following example with the AWE can be computed about twenty times faster than regular Frequency Domain sweeps with a same finer frequency resolution.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Lumped Port I

- In the Model Builder window, under Component I (compl)>Electromagnetic Waves, Frequency Domain (emw) click Lumped Port I.
- 2 In the Settings window for Lumped Port, locate the Boundary Selection section.
- 3 Click here are a create Selection.
- 4 In the Create Selection dialog box, type Lumped port 1 in the Selection name text field.
- 5 Click OK.

ADD STUDY

I In the Home toolbar, click $\stackrel{\text{res}}{\longrightarrow}$ 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 2 Add Study to close the Add Study window.

STUDY 2

- I 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

- I 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(2.11[GHz],0.5[MHz],2.155[GHz]).

A slowly varying scalar value curve works well for AWE expressions. 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 are 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 these 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 I in the Selections list.
- 7 Click OK.

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

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

RESULTS

ID Plot Group 7

In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.

Global I

- I Right-click ID Plot Group 7 and choose Global.
- 2 In the Settings window for Global, locate the y-Axis Data section.
- **3** In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	dB	S11 Regular Sweep

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Global 2

- I In the Model Builder window, right-click ID Plot Group 7 and choose 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 y-Axis Data section. In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	dB	S11 Adaptive Frequency Sweep



5 In the ID Plot Group 7 toolbar, click 💽 Plot.