

Ansoft HFSS Tutorial: Dipole Antenna

Dr. Ryan S. Adams

September 8, 2009

This tutorial introduces the interface of Ansoft's HFSS, and walks the student through an example problem of creating, simulating and evaluating the response of a standard stripline structure.

1 Starting HFSS

At UNCC, the HFSS application resides on a Linux based server by the name of "hertz" which is part of the Reconfigurable Computing System (rcs). The best way to access this server is through the Mosaic system.

1.1 Logging Into Mosaic Linux Server

When you are logged in to one of the Mosaic windows computers, you first need to log in to Exceed Linux. This is done by selecting the start menu => All Programs => MOSAIC XP => Unix Connectivity => Exceed Linux. In the box that comes up, lxs-sm1 and lxs-sm2 are identical linux servers that are maintained by Mosaic. Highlight whichever one you prefer and click OK. You should then be presented with a login screen for Red Hat Enterprise Linux 5. To login, first type your Mosaic user ID into the available block and press enter or tab. A new block should appear with the word password above it; type your Mosaic password into this block and press enter or tab. You should now be logged in to the linux server.

1.2 Logging into the Reconfigurable Computing System Network

At this stage, you should be looking at a linux desktop (probably with a red background). To log in to the RCS network, start from a command prompt. To do this, right click anywhere on the desktop and click "Open Terminal" in the pop-down menu that appears. In the terminal that appears type `ssh -C -Y rcs` you should now be prompted to log into the rcs system – do not use your mosaic password, use your rcs password instead. At this point, you need to setup your profile to recognize hfss. To do so, type

```
gedit .bash_profile
```

Remember to type this command exactly as it appears on this page. This command will bring up a text editor. Modify the file as necessary until it looks like the following:

```
# .bash_profile

# Get the aliases and functions
if [ -f ~/.bashrc ]; then
. ~/.bashrc
fi

# User specific environment and startup programs

PATH=$PATH:$HOME/bin:/opt/ansoft/hfss11

export PATH
```

To use the updated profile, you need to log out of hertz with the `exit` command. Then repeat step 2. above to log back into hertz and type `hfss`

Note: The first time you log into hfss, it will prompt you for certain file locations. Use the defaults and follow the prompts until the gui appears. You may find yourself back at a command prompt. If this happens, just retype `hfss` and the gui should come up.

2 The HFSS Interface

The main HFSS interface is shown in Figure 1, which illustrates the main components of the gui. They are summarized as follows:

- **3D Modeler Window** This is the area where you create the model geometry. This window consists of the model view area (or grid) and the history tree as shown in Figure 2. The history tree documents the actions that have been taken in the model view area, and provides an alternative way to select objects in the model view area.
- **Project Manager with Project Tree** The project manager window displays details about all open HFSS projects. Each project ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post processing information. An expanded view of the project manager is shown in Figure 3
- **Properties Window** The properties window consists of two tabs. The command tab displays information about an action selected in the history tree that was performed to either create an object or modify an object. The attribute tab displays information about the material and display properties of a selected object.
- **Progress Window** This window is used when a simulation is running to monitor the solution's progress.
- **Message Manager** This window displays messages associated with a project's development (such as error messages about the design's setup)

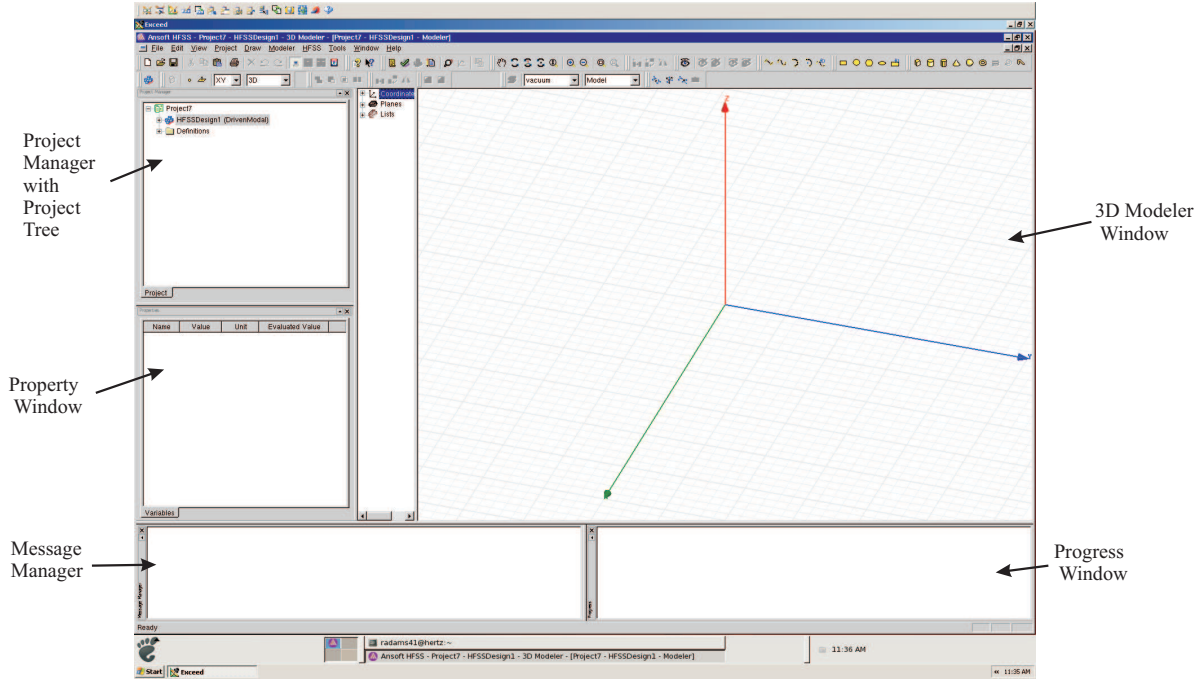


Figure 1: *Main screen of HFSS.*

3 Setting up HFSS

Before you can use HFSS for the first time, there are a couple of items that need to be configured for efficient and accurate operation.

1. On the Tools menu, select Options => General Options ..., click the Default Units tab and ensure that Length is set to mm. Click OK.
2. On the Tools menu, select Options => HFSS Options..., ensure the Include ferrite materials check box is checked. Click the Solver tab, set the number of Processors to 2 and Desired RAM Limit (MB) to 4000 (the Maximum RAM Limit (MB) should remain unchecked). Click OK.

You should now be ready to use HFSS.

4 An Example: Dipole Antenna

To begin to appreciate the functionality of this simulation tool, we will create and simulate a simple loop antenna. Before we can begin to work through the simulation though, we need to design the antenna on paper. This design will consist of a half-wave dipole antenna with center frequency of 300 MHz. At this frequency, the wavelength in free-space is

$$\lambda = \frac{c_0}{f} = \frac{3 \times 10^8}{300 \times 10^6} = 1m \quad (1)$$

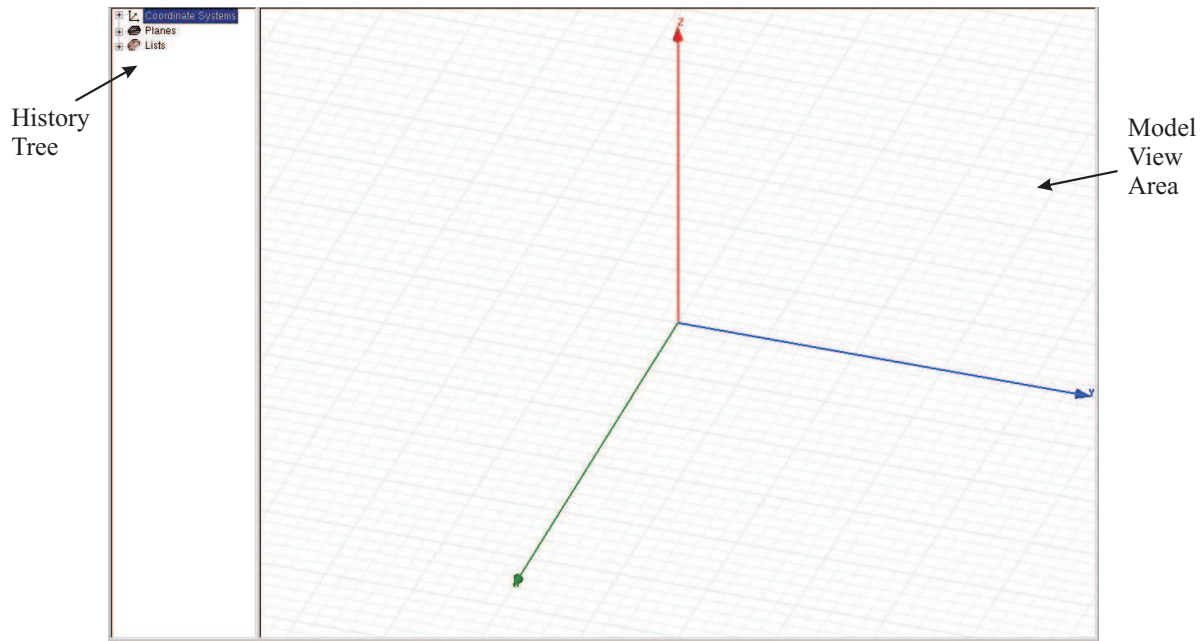


Figure 2: *3D Modeler Window, which consists of the model view area and the history tree.*

So, the total antenna length should be $\ell = \lambda/2 = 0.5$ m. Now, the only items remaining are the radius of the wires and the gap between the wires at the feed point. These items are difficult to determine analytically, so we will simply define them to be $r = 0.1$ mm, and $g = 20$ mm. With these choices, we expect to have an omnidirectional pattern in the far-field.

4.1 Creating the Antenna

Now it's time to build this model in HFSS. We'll begin with the radiating structure itself. To do so, we'll create a series of cylindrical shapes by selecting Draw => cylinder in the file menu. To make the cylinder the correct size, you can either

1. input the x , y , z coordinates of the center point, radius and height of the cylinder into the appropriate fields at the bottom of the 3D model window, or
2. click randomly in the 3D model window three times to create a cylinder and edit the size in the properties box. To do this, click the command tab in the properties box and input the correct center point, radius and height for our cylinder.

For this design, we'll choose the center point to be placed at the location 0,0,10mm, the radius should equal 0.1mm and the height should be 240mm. Next, we'll create the other half of the radiating element. This will be another cylinder with center point at 0,0,-10mm, radius 0.1mm and height of -0.240mm.

Next we need to create the feed structure. To do so, we need to create two more cylinders, and two spheres. The cylinders will occur in the zx -plane, so we need to change the desired plane accordingly in the pull-down menu. Then, we create two cylinders,

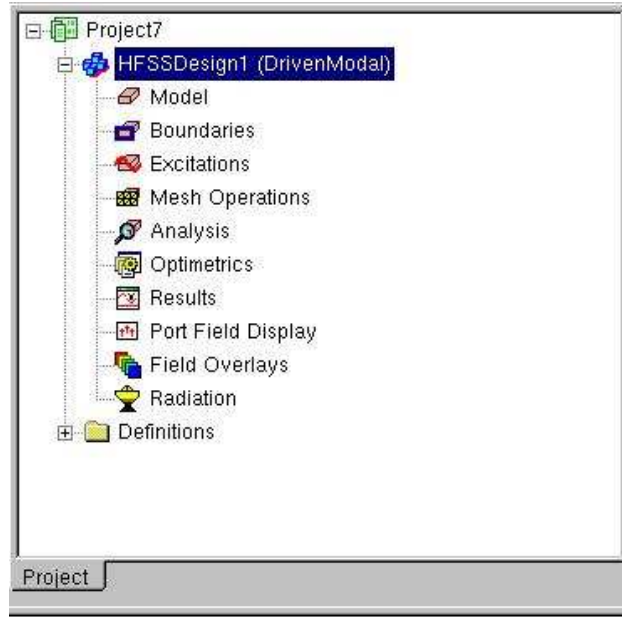


Figure 3: Project Manager window illustrating the boundary conditions, excitation, etc. of the current model.

- the first with center at 0,0,10mm, radius = 0.1mm, and height = 250mm
- the second with center at 0,0,-10mm, radius = 0.1mm, and height = 250mm

Now, we have an incomplete connection between the feed wires and the radiating wires. To complete this connection, we'll add two spheres at the junction.

- the first with center at 0,0,10mm, and radius = 0.1mm
- the second with center at 0,0,-10mm, and radius = 0.1mm

Now select all items that you have drawn so far and select Modeler => Boolean => Unite. Finally we will identify the resulting geometry as a perfect electric conductor (PEC) by right clicking in the geometry window and selecting Assign Material. In the resulting window, select PEC and click OK.

4.2 Creating the radiation box

Now, we need to establish a box around the antenna in which we will compute the fields, and from which we will calculate the far-field response. To assure good far-field calculations, the box should extend at least $\lambda/4$ away from all radiating surfaces. We will choose the box to be centered on the antenna and $\lambda/2$ wide, so that all surfaces are at an adequate distance from the antenna.

To draw the radiation box, select Draw => box and input the following parameters for the box:

- Beginning point: -250mm, -250mm, -510mm

- The cartesian sizes are: Xsize = 500mm, Ysize = 500mm, and Zsize = 1020mm

Now select the attribute tab and do the following:

- ensure that the material is vacuum. If it is not, click the button next to material and select vacuum as the material.
- click the button next to *transparent* and set the transparency to 0.95.

Now, the model is complete.

4.3 Boundary Conditions

The following items need to be setup to assure a good simulation:

1. **Mesh Operations:** To ensure we receive a good quality far field calculation, we need to ensure that the mesh is generated with sufficient accuracy on the outer boundary of the domain. According to Ansoft, we can achieve sufficient accuracy if the following rules of thumb are followed:

- (a) The outer boundary must be at least $\lambda/4$ away from every radiating surface
- (b) The mesh length must be constrained to be no greater than $\lambda/6$ on the boundary

The wavelength at 300 MHz in free space is $\lambda = 1\text{m}$, so the maximum length of elements on the boundary is $\ell_{max} = 0.16\text{m}$.

To limit the mesh to this value, perform the following steps:

- (a) Highlight the vacuum box
 - (b) Right click in the 3D modeler window and select Assign Mesh Operation => On Selection => Length Based
 - (c) Set the maximum length of elements to 160mm
 - (d) Click OK
2. **Radiation Boundary:** To compute the radiated fields, we need to establish a radiation boundary on all surfaces of the vacuum box. To do this, we right click in the drawing window and click *select faces*. Then, click on each face of the vacuum box while holding the control key. Once all faces are selected, right click again in the drawing window and click *assign boundary* => Radiation.... Use all defaults on this boundary.
 3. **Far Field Calculation:** To compute the radiated fields in the far-field, we need to tell HFSS what level of granularity we want on the far-field sphere. To do so, right-click *radiation* in the project tree and select *Insert Far Field Setup* => *infinite sphere*; use the defaults for this sphere.

4.4 Excitations

We will create a “Waveport” excitation at the feed lines of the antenna. To do so, we perform the following steps:

1. Create a rectangle in the zx -plane with first point (-25mm, 250mm, -24.85mm) and axis = Y, Xsize = 50mm, Zsize = 50mm
2. Select the resulting rectangle
3. Right-click in the 3D modeler window and select Assign Excitation => Wave Port.
4. Click Next
5. Under “Integration Line,” click the word None, and select New Line...
6. In the 3D modeler window, click the center of the bottom wire
7. Click the center of the top wire
8. Click Next
9. Click Finish

4.5 Analysis

Perform the following steps to set up the analysis options:

1. Right click on Analysis in the Project Tree, and select “Add Solution Setup”
2. Under the General tab:
 - (a) Set the solution frequency to 300 MHz
 - (b) Set the maximum number of passes to 30
 - (c) Set maximum Delta S to 0.01
3. Under the Options tab:
 - (a) Set the Maximum Refinement per pass to 20 %
 - (b) Set the Order of Basis Functions to First Order
4. Under the Advanced Tab, check the box titled *Use Radiation Boundary on Ports*
5. Click OK

Perform the following steps to set up a frequency sweep (if desired):

1. Under the Analysis item in the Project Tree, right-click on Setup1
2. Select Add Frequency Sweep...

3. Set start frequency to 20 MHz
4. Set stop frequency to 300 MHz
5. Set step size to 20 MHz
6. Click OK

4.6 Final Checks and Running the Simulation

Select HFSS => Validation Check... to ensure the project is prepared for simulation (click close).

Save the project by clicking on the save icon at the top of the screen.

Right-click setup1 under Analysis in the project tree, select Analyze to begin the simulation. At this point the progress window should show the progress of the simulation, beginning with the mesh generation.

4.7 Simulation Results

To view the far-field results of the simulation, perform the following steps:

1. Right click on the results item in the Project Tree
2. Click Create Far-Fields Report => 3D Polar Plot
3. Under the trace tab, select Gain
4. Add trace by clicking the Add Trace button
5. Click Close

To view the impedance results of the simulation, perform the following steps:

1. Right click on the results item in the Project Tree
2. Click Create Modal Solution Data Report => Rectangular Plot
3. Under the trace tab, select Z-Parameter, Z(1,1), highlight re and im with the control key
4. Add traces by clicking the Add Trace button
5. Click Close