

Laboratory Assignment: EM Numerical Modeling of a Stripline



Names: _____

Objective

This laboratory experiment provides a hands-on tutorial for drafting up an electromagnetic structure (a stripline transmission line on a circuit board) in the ANSYS HFSS modeling tool.

Preparation

Before coming to the laboratory to perform this assignment, the students should prepare the following:

- Identify a workstation in our school (or remote log-in portal) with an installation and/or license for running ANSYS HFSS electromagnetic modeling software.

Write-Up

The students performing this laboratory do *not* need to prepare a stand-alone laboratory report for this assignment. All work may be neatly shown on this document, with any supplemental answers to questions attached. Be sure to include all group member names on this sheet.

Equipment Guide



ANSYS HFSS software

Procedure

1. Starting HFSS

- a. Go to Start menu, and under all programs go to the following program:
ANSYS Electromagnetics\HFSS 15.0\Windows 64-bit\HFSS 15.0 (64-bit)
- b. Upon your first start it will state that the libraries have been reset. Select **OK**.
- c. It will give you options for your project directory and temporary directory. Select **OK**. The default which is in your “My Documents” folder has been adequate for myself. The temporary directory of C:\Temp is suggested as well.

2. Setting up HFSS

- a. For any new project, you will need to insert a design. Do this by going to Project -> Insert HFSS Design. Alternatively you can click the corresponding icon on the toolbar.

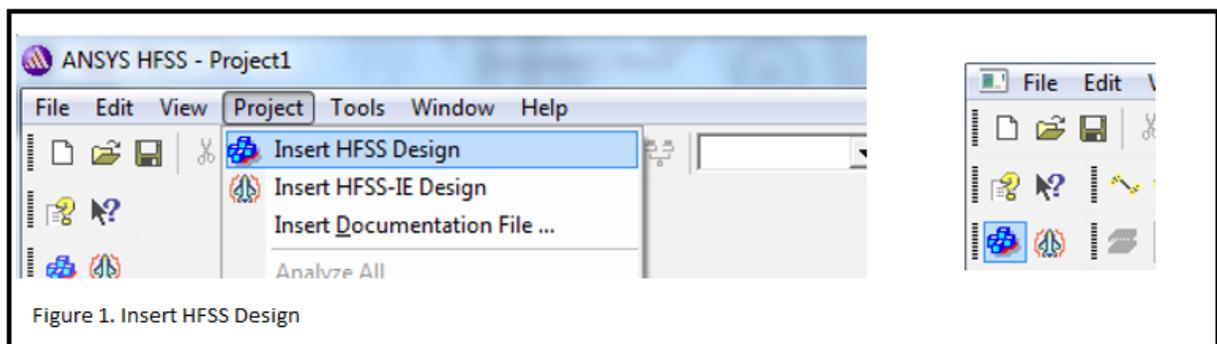


Figure 1. Insert HFSS Design

- b. Now let's take a minute to get familiar with the user interface of HFSS. In the following screen capture, you will notice a few additional objects that your screen will not have, such as the cube. This is only for demonstration, we will continue where we left off.

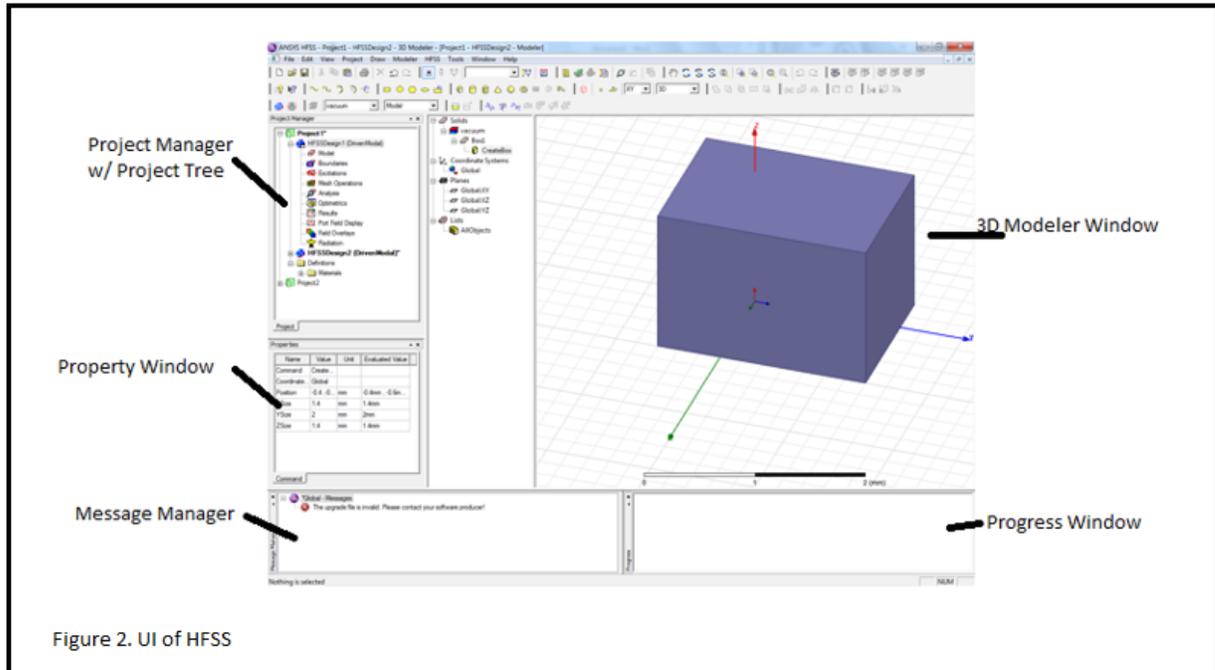
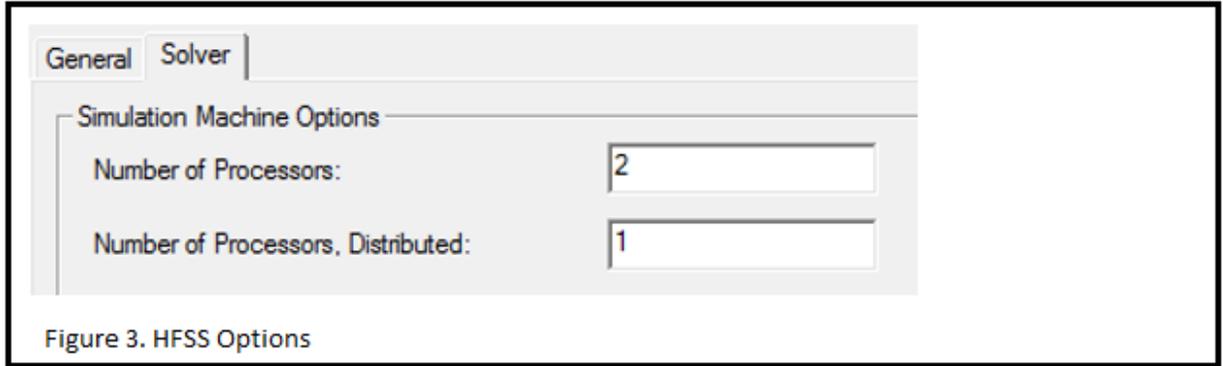


Figure 2. UI of HFSS

- **3D Modeler Window** This is the area where you create the model geometry. This window consists of the model view area. Directly to the left, but still part of the 3D Modeler Window, is an area where you can select objects and certain properties of the 3D model.
- **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. In this screen shot, one can see the hierarchy of multiple designs and multiple projects.
- **Properties Window** The properties window can contain attributes and variables. Depending on the object selected, it may list things such as properties of a 3D object, such as material. If the actions to create an object are selected, such as the screen shot, one can easily change the parameters used to create that object (X Y Z size of the box). Monitoring this window can save time by easily accessing and changing properties of your design.
- **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 or licensing issues)
- c. Setup HFSS. Go to Tools->Options->HFSS Options. Select solver, and type in 2 Processors.

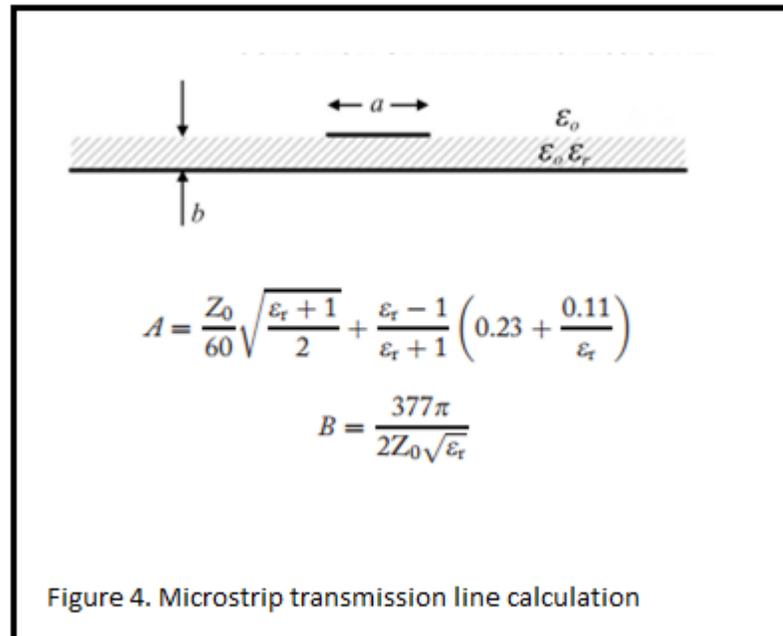


- d. It is wise to check your default units. Go to Modeler->Units and see that mm are selected.

3. What is a 50 Ohm Stripline.

We will model and demonstrate a 50 Ohm stripline. First we must know our material. We will be using the widely available FR4 substrate with a **thickness of 60 mils** and **1 Oz (1.34 mils thick) copper plating**, with a **relative permittivity ϵ_r of 4.4**. Though we setup the default units as millimeters earlier, it is straight-forward to input different units within the program without having to convert the numbers manually. There are many things to note about creating a stripline. First, ϵ_r may change with a variety of parameters, such as humidity, frequency, and board supplier. Second, there are several formulas used to calculate the size of a stripline.

One such calculation is provided by the material [Transient Signals on Transmission](#)



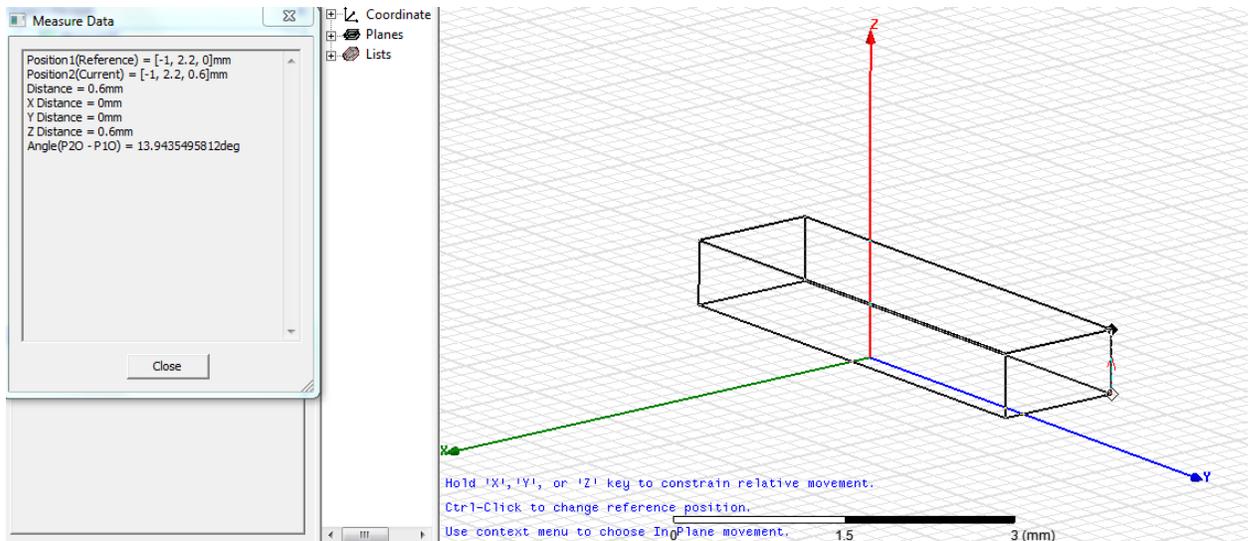
Lines by Peterson & Durgin. The formula can be seen in Figure 4. Some equations take into the thickness of the top layer of copper, as well as frequency of operation. The online calculator located at [http://www1.sphere.ne.jp/i-lab/ilab/tool/ms_line_e.htm] was used to calculate the parameters, and the results were verified with ADS LineCalc, a more professional tool used by the industry to calculate transmission lines. For a 50 ohm stripline on our material and frequency, we will need a **2.836 mm** wide stripline.

4. Your first HFSS Model: 50 Ohm Stripline.

We have now defined most of the parameters of our stripline. We will simulate one that is of **length 10 cm**. As a general rule of thumb, we want the substrate to be extend from both edges of the stripline by a minimum of three times the thickness of the substrate. We will assume the **board width to be 5 cm**.

A. First Copper Sheet.

1. Click on the 3D box icon() on the tool bar and click on the 3D workspace and move the mouse to create a flat square in the XY plane. Click again to set the point, and then move your mouse one more time and click to create an arbitrary 3D cube. When you are done, the object will fill itself in with purple.

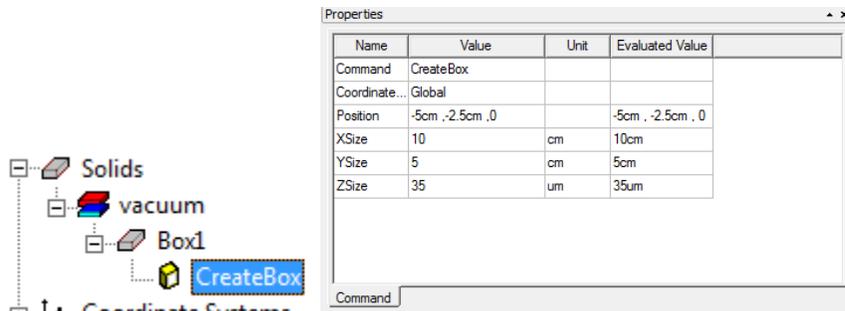


2. Rotate the object and get familiar.

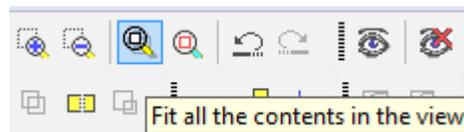
- o Friendly Tips:
 - Hold Alt while dragging to rotate the view
 - Hold shift while dragging to move the view
 - Hold shift+alt while dragging to zoom (similar to the scrolling wheel)

- Alt+Double click in the Top Right corner to get the default view as in the previous figure

3. Now that we have a rectangular box, we will manually insert the dimensions. Click on the **create box property** of the **Box1** within the 3D modeler. Once it is selected, its properties will become available as shown in following figures.



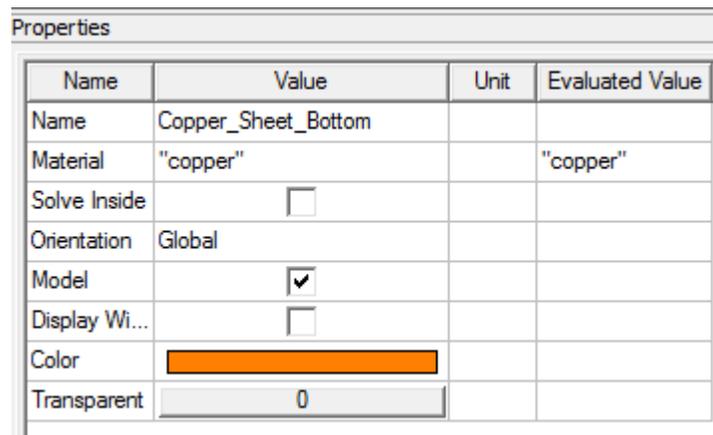
4. For the position, type in “ $-(L/2), -(W/2), 0$ ”. You will be prompted to enter the values of these variables. Enter “**10cm**” and “**5cm**” for the **L** and **W**, respectively.
 - **Note:** The sizes may make the object take up the whole screen. Click the following icon to fit to the screen.



- **Note:** You can append units to the end of a value, and HFSS will understand it. If it doesn't, you will get an error.
- You should realize you can put in any other character or word for these variables, as long as they don't have a space in them. This allows you to edit several objects at once that depend on the variable, and see the results real time. Makes dealing with multiple layers of a board very easy.
5. For XSize, YSize, and ZSize put in “L”, “W”, and “**Copper_T**”, and putting in “**35um**” for the value of the new variable “Copper_T”.
 - **Note: If you mistype**, you can change the values of these variables within HFSS by clicking on the Design and looking at the

properties tab, or by going to the toolbar and selecting HFSS->Design Properties.

6. Select Box1 to the left of the 3D modeler window and look at the Properties Window. Double click on **Box1** within the **properties window** and rename it to **Copper_Sheet_Bottom**.
7. Select “vacuum” that is next to Material, and under Value. Click and select edit. Start typing “cop”, and select copper. Last, choose a color that may help you visualize your object. I suggest you a color similar to copper. Your properties similar to the following figure.



Name	Value	Unit	Evaluated Value
Name	Copper_Sheet_Bottom		
Material	"copper"		"copper"
Solve Inside	<input type="checkbox"/>		
Orientation	Global		
Model	<input checked="" type="checkbox"/>		
Display Wi...	<input type="checkbox"/>		
Color			
Transparent	0		

- **Note:** You could use perfect electric conductor to speed up your simulations, though for these simulations the difference in time is minimal. Use copper for now.

B. FR4 Layer

1. We could create a new box from scratch and repeat the previous steps. As the bottom copper layer and FR4 substrate will have many similar dimensions, we will copy the copper layer and edit its properties as we see fit.
2. Click on “Copper_Sheet_Bottom” object within the left pane of the 3D Modeler. Copy and paste the object by typing Control+C and then Control+V. You will now have “**Copper_Sheet_Bottom1**”.
3. **Edit** the properties of Copper_Sheet_Bottom1 and its “Create Box” property to have the following values.

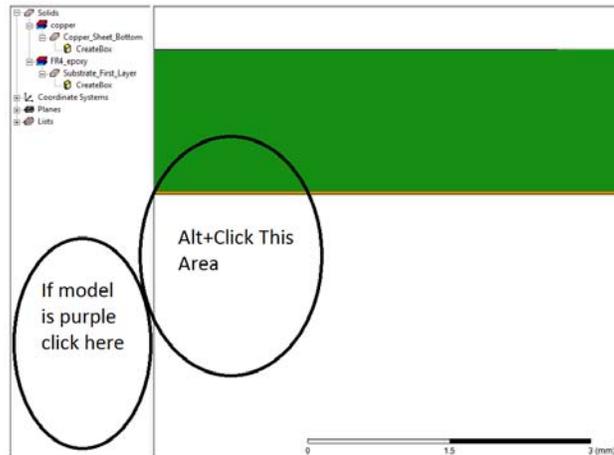
Properties			
Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-(L/2) , -(W/2) , Copper_T		-5cm , -2.5cm , ...
XSize	L		10cm
YSize	W		5cm
ZSize	FR4H		60mil

- **Note:** FR4H value is “60mils” This represents the substrate height as described in section 3.

4. **Edit** the properties of Copper_Sheet_Bottom1 to the following (and thus naming it now to “**Substrate_First_Layer**”). Colors can be helpful.

Properties			
Name	Value	Unit	Evaluated Value
Name	Substrate_First_Layer		
Material	"FR4_epoxy"		"FR4_epoxy"
Solve Inside	<input checked="" type="checkbox"/>		
Orientation	Global		
Model	<input checked="" type="checkbox"/>		
Display Wi...	<input type="checkbox"/>		
Color			
Transparent	0		

5. Hold ALT and double click on **the Left Center of the 3D Modeler window**. This will give us a view from the side. Zoom in until the model is larger. **If the model is purple**, click in the left pane on the blank space to unselect any objects.



6. Your modal should look similar from the side. A thin copper bottom layer, and a thick substrate. You can hold ALT and click the top right corner of the 3D modeler to restore the standard view.

C. Stripline

1. Create another box of copper. You may copy the first layer of copper, substrate, or start a new box entirely. As often you start building where one object ends, it may be easier to copy the last object to have the previous coordinates to edit. It may also be easier to copy a similar object for the material properties. Do what you see is fit. You should have similar properties of this new stripline layer as follows:

- **Note:** MS_W should be 2.836 mm

Properties			
Name	Value	Unit	Evaluated Value
Name	Copper_Sheet_Top		
Material	"copper"		"copper"
Solve Inside	<input type="checkbox"/>		
Orientation	Global		
Model	<input checked="" type="checkbox"/>		
Display Wi...	<input type="checkbox"/>		
Color			
Transparent	0		

Properties	
Name	Value
Command	CreateBox
Coordinate...	Global
Position	-(L/2) , -(MS_W/2) , Copper_T+FR4H
XSize	L
YSize	MS_W
ZSize	Copper_T

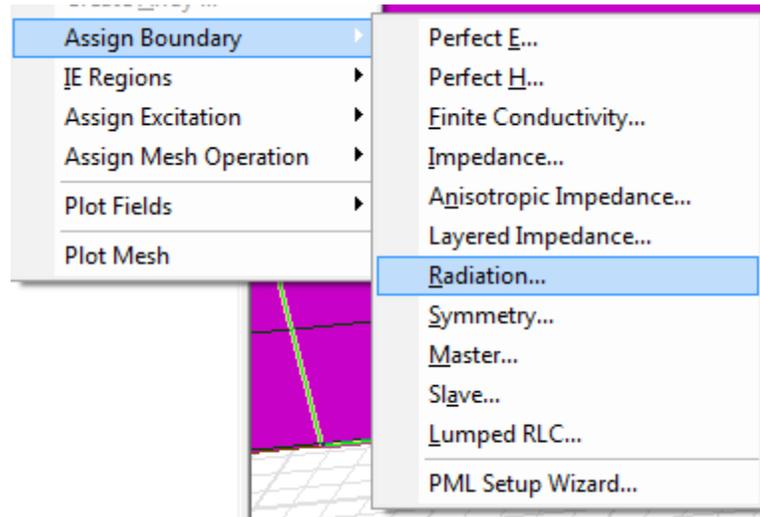
D. Radiation Boundary

1. Last we need to create a radiation boundary. This effects the radiation pattern and the fields. Though you would assume that it would create a vacuum and solve accordingly without it, it will show unexpected results if you forget this step.
2. Create a new box, with the following properties.

Name	Value	Unit	Evaluated Value
Name	RadiationBoundary		
Material	"air"		"air"
Solve Inside	<input checked="" type="checkbox"/>		
Orientation	Global		
Model	<input checked="" type="checkbox"/>		
Display Wi...	<input type="checkbox"/>		
Color			
Transparent	0		

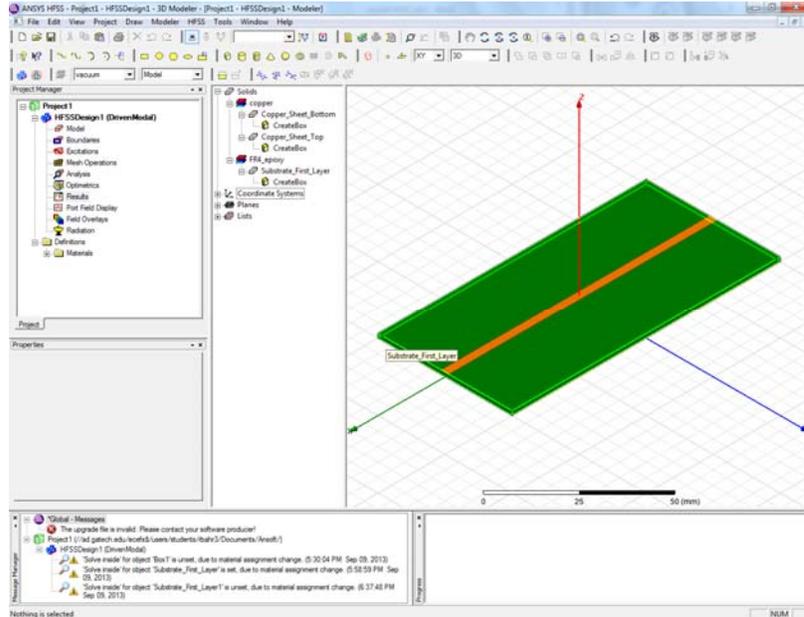
Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-L/2, -W/2, 0cm		-23.305mm, -2...
XSize	L		46.61mm
YSize	W		5cm
ZSize	5	cm	5cm

3. Now right click on the RadiationBoundary object that you just created in the left area of the 3D modeler, and select **Assign Boundary and Radiation**. Hit OK.



- Note: The radiation boundary encompasses the model. Depending on the port we will define in the future, the geometry of the radiation boundary you will design may be affected. This will be discussed later.
4. Let's hide the radiation boundary. You should still have the object selected, and click this hide view button (). We just need it there, we don't need to see it.

Congratulations! You have now created your first model. It should look as follows.



5. Setting up your first Simulation

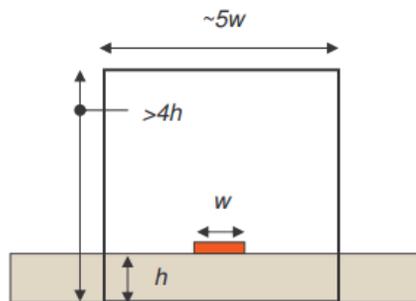
A. Defining Ports

- We must now create ports on our stripline to excite the object and to measure its properties at a given point. For this, we will create 2 wave ports.
- Create a 2D rectangle by clicking the icon  .
- Right click in the 3D modeler and select Grid Plane -> YZ.
- Draw an arbitrary rectangle.
- Edit the squares properties to the following numbers. These will be your waveguides.

Name	Value	Unit
Name	WavePort1	
Orientation	Global	
Model	<input checked="" type="checkbox"/>	
Display Wi...	<input type="checkbox"/>	
Color		
Transparent	0	

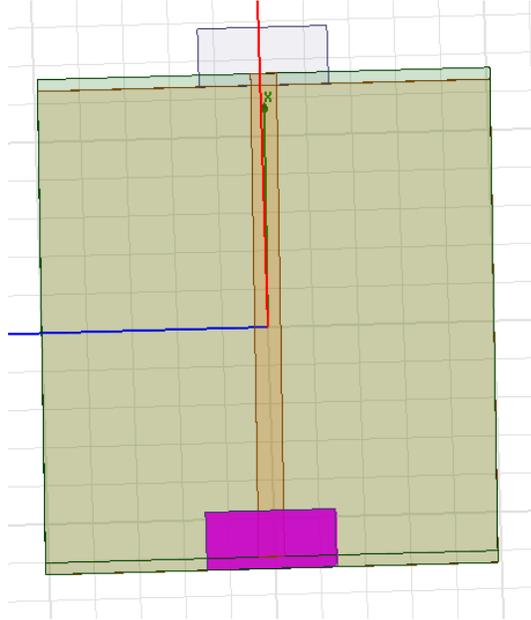
Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate...	Global		
Position	$-L/2, -(5*MS_W/2), 0\text{mm}$		-5cm, -7.15mm ...
Axis	X		
YSize	$5*MS_W$		14.3mm
ZSize	$5*FR4H$		300mil

- Note: The multiple of 5 comes from *guidelines* provided by HFSS.



- Copy this WavePort1 and paste it. It will be WavePort2. Edit the position so that instead of $-L/2$, it is now $L/2$ for the x coordinate of the position.
- You now have 2 squares. Press “f” on your keyboard, which allows you to select faces. Select WavePort1. It should look similar to the following figure.

- Note, to go back to selecting objects, type “o” after completion of



these steps

- h. Right click on it and select Assign Excitation-> Waveport. Hit next through all the options.
- i. Do the same for WavePort2
 - **Note:** As discussed earlier, your port choice affects your radiation boundary. For a waveguide to function within HFSS, the Wave Port must be on the edge of the radiation boundary. If you were simulating a device where the port was inside a device, such as full models of a cellular phone, you would use a lumped port. The radiation boundary generally encompasses your entire device.
- j. Go to HFSS -> Analysis Setup -> **Add Solution Setup. Put in a Solution frequency of 2.4 GHz.** Leave delta as .02, and passes as 6.
 - **Note:** The model re-simulates simulations with more refinement until the difference of the s parameters between two simulations is less than .02 (2%), or 6 passes, whichever comes first. The delta can essentially represent your estimated error, and the passes can be a hard limit set so it doesn't simulate for days while you wait for an accurate result. If it doesn't meet this error criterion, you will get a warning that the adaptive passes did not converge.

- k. In the left pane, right click on Setup1 and select **Add Frequency Sweep. Select .1 to 10 GHz.**
- l. Right click on **Setup1** and select **Analyze**. Give it some time to solve.
- m. Right click on **Results**. We will now choose some graphs of interest.
 - i. Select Create Modal Solution Data Report -> Smith Chart. Here we will select S Parameters-> **S(1,1)**. Select New Report.
 - ii. Now select **S(2,2)**, and thereafter click New Report.
 - iii. Now select Port Zo, and control click functions of re and img so both are highlighted. Then click New Report.
 - iv. Exit that out.
 - v. Select Create Modal Solution Data Report-> **Rectangular Chart**. Here we will select S-Parameters-> **S(1,1)**. The function should be of dB. Select New Report.
 - vi. Now select **VSWR**, and select **VSWR(1)**. Select New Report.
- n. You should now have a bunch of graphs of data that you can easily switch between. Look at these basic results, and analyze them.

Analysis

1. What do you see in the two smith charts? Do they look similar? Why? Right click and select Add marker, and notice what changes as you move around the data plotted.
2. What is the port impedance Z_0 ?
3. What do you see in the Rectangular graph of the S(1,1)? Look at how unique points on this chart relates to the same point on the Smith charts.
4. Now look at the VSWR graph. Compare once more how it looks on this near ideal transmission line.

Acknowledgment

This laboratory experiment was designed by Ryan Bahr.