APPLICATION NOTES

Design of a Microstrip Patch Antenna

JUNE 12, 2016

CHUONG KHONG

Abstract

This tutorial is to provide future students a way to design and simulate their microstrip patch antennas on HFSS for their senior design project. The guide begins with calculating pertinent design parameters and then proceeds to build the microstrip antenna for simulation. The type of antenna that will be designed in this short tutorial is an inset-fed microstrip patch antenna. The software used to design and simulate the antenna will be Ansoft HFSS.

Introduction

Being able to design and simulate microstrip antennas is a useful skill to have in this course. The benefits of microstrip antennas are that they are extremely lightweight compared to other antennas, such as horn antennas. Another benefit to microstrip antennas is that they can be easily fabricated on printed circuit boards (PCB). Thus, they are a relatively cheap alternative compared to other forms of antennas.

Tutorial

Calculations

Calculate the dimensions of the microstrip patch antenna. For the purpose of this tutorial, the resonant frequency will be 25 GHz, the relative dielectric constant, ε_r, is 3.66, and the dielectric thickness, 'h', is 0.254 mm. The relative dielectric constant and dielectric thickness can be found in the Rogers 4350B datasheet, which is the material that was used in my team's design. All calculations were made after converting lengths and widths into meters, and then converted back into millimeters for HFSS modeling.

$$f_r = 25\ GHz$$

$$\varepsilon_r = 3.66$$

$$h = 0.254\ mm$$

$$\varepsilon_{reff} = \frac{(\varepsilon_r + 1)}{2} + \frac{(\varepsilon_r - 1)}{2}\left(1 + \frac{12h}{W_{patch}}\right)^{-\frac{1}{2}} = \mathbf{3.3282}$$

$$W_{patch}(Width)| = \frac{c}{2f_r}\sqrt{\frac{2}{\varepsilon_r + 1}} = \mathbf{3.9307\ mm}$$

$$\Delta L = \frac{0.412h\left((\varepsilon_{reff} + 0.3)\left(\frac{W_{patch}}{h} + 0.264\right)\right)}{(\varepsilon_{reff} - 0.258)\left(\frac{W_{patch}}{h} + 0.8\right)} = \mathbf{0.1196\ mm}$$

$$L_{patch}(Length) = \frac{c}{2f_r\sqrt{\varepsilon_{reff}}} - 2\Delta L = \mathbf{3.0497\ mm}$$

$$R_{edge}(Edge\ impedance) = 90\ \frac{\varepsilon_r^2}{\varepsilon_r - 1}\left(\frac{L}{W}\right)^2$$

$$R_{edge} = \mathbf{272.8289\ \Omega}$$

$$d\ (Recessed\ Depth) = \frac{L * cos^{-1}\left(\sqrt{\frac{R_{in}}{R_{edge}}}\right)}{\pi} = \mathbf{0.8913\ mm}$$

Modeling the Microstrip Patch Antenna

After obtaining the dimensions of the microstrip patch antenna, the next step is to begin modeling the patch antenna on HFSS. In order you model the antenna, you have to create a new project by clicking the blue "blocks" button, refer to the red arrow that Fig. 1 points to. Another important feature that students should consider using is setting up design variables when modeling microstrip patch antennas. Setting up variables takes an extra couple minutes to complete, but it will end up saving the students hours when they begin to simulate the antennas because there will be a need to tune the antennas' dimensions in order to get the best results. See Fig. 2 of page 3 for the list of variables that was used in this tutorial. The advantages of using design variables is that they can be easily changed when you begin tuning the antenna to hit the correct resonant frequency.

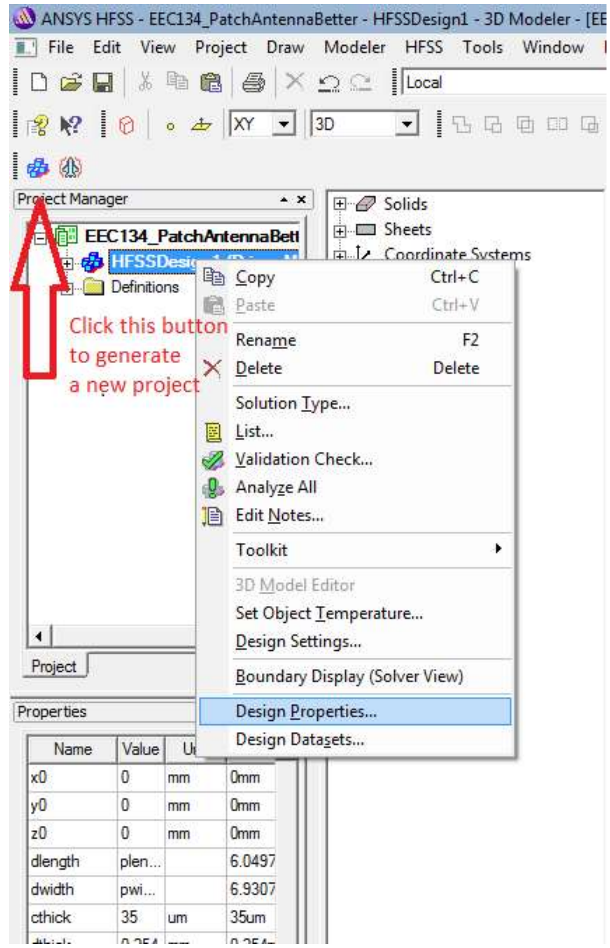
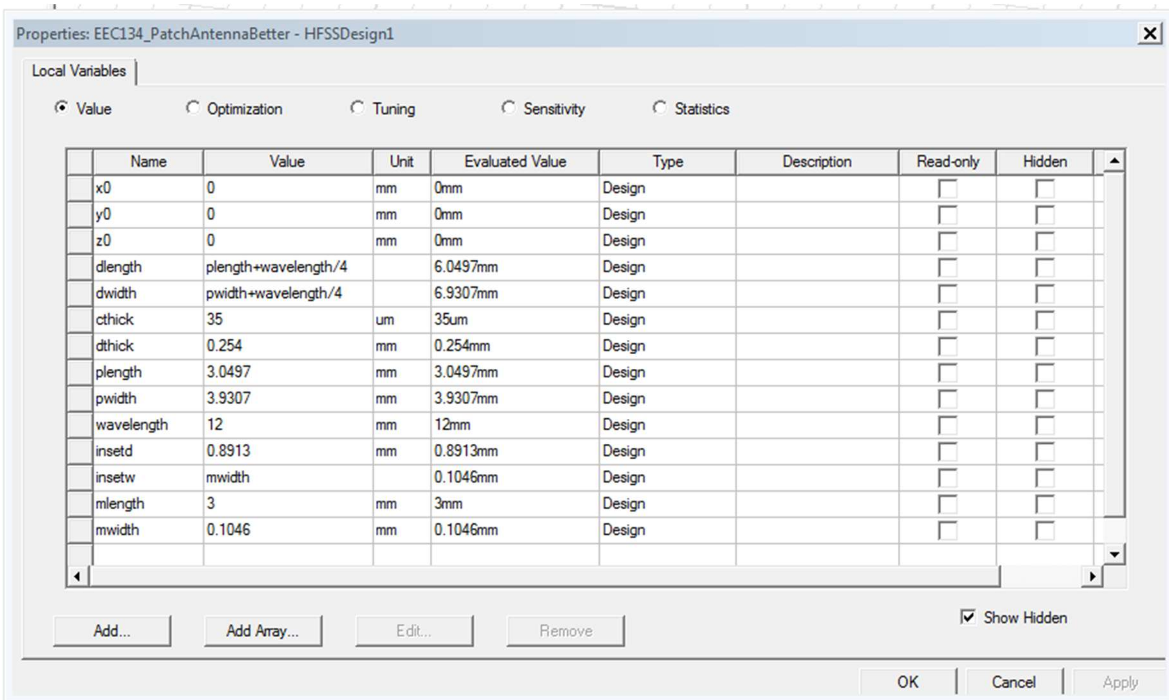


***Fig. 1** - Design Variables*



Name	Value	Unit	Evaluated Value	Type	Description	Read-only	Hidden
x0	0	mm	0mm	Design			
y0	0	mm	0mm	Design			
z0	0	mm	0mm	Design			
dlength	plength+wavelength/4		6.0497mm	Design			
dwidth	pwidth+wavelength/4		6.9307mm	Design			
cthick	35	um	35um	Design			
dthick	0.254	mm	0.254mm	Design			
plength	3.0497	mm	3.0497mm	Design			
pwidth	3.9307	mm	3.9307mm	Design			
wavelength	12	mm	12mm	Design			
insetd	0.8913	mm	0.8913mm	Design			
insetw	mwidth		0.1046mm	Design			
mlength	3	mm	3mm	Design			
mwidth	0.1046	mm	0.1046mm	Design			

Fig. 2 *- List of design variables that was used in modeling the patch antenna*

After the design parameters are set up, the next step is to begin modeling the patch antenna. We will begin with modeling the ground plane, dielectric plane, and the copper plane. To begin drawing the ground plane, select the “Draw Box” icon near the top right corner of the software. Click on the center of the XYZ axis and drag the box outward, the size of the box does not have to be exact since it can be easily changed. Refer to Fig. 6 for instructions on how to change the dimensions of the box. This is done by going to Properties, and changing XSize, YSize, and ZSize.

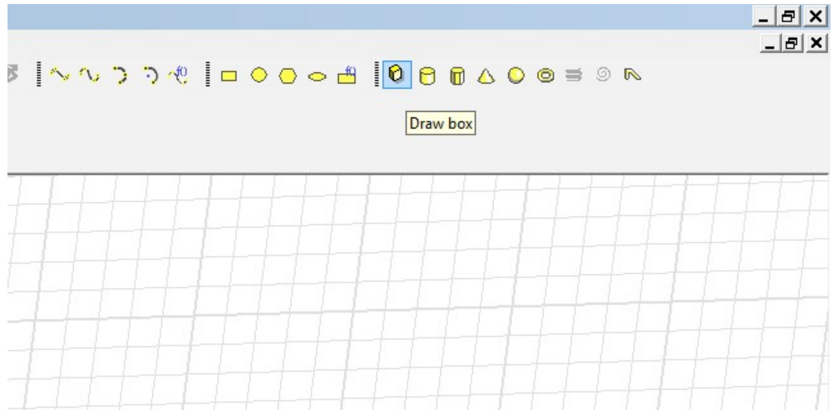


Figure 3 - "Draw Box"

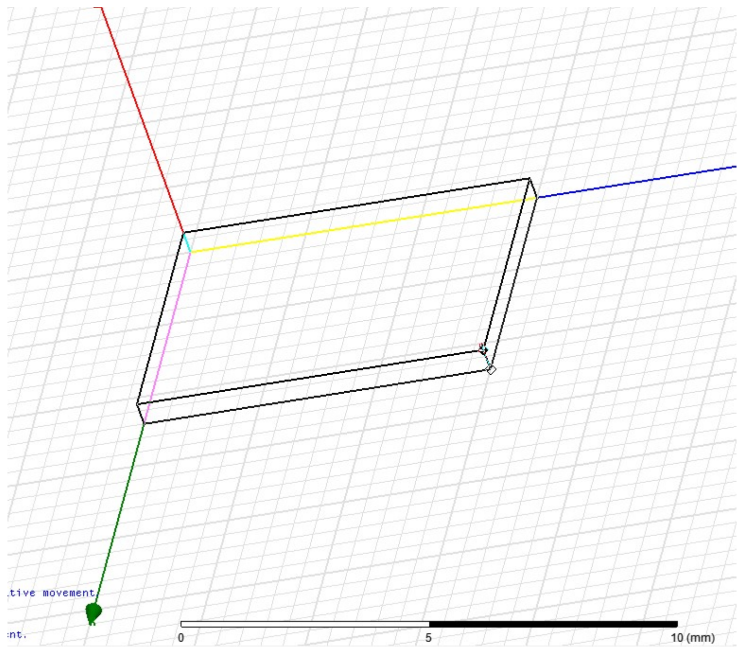


Figure 4 - Modeling the Ground Plane

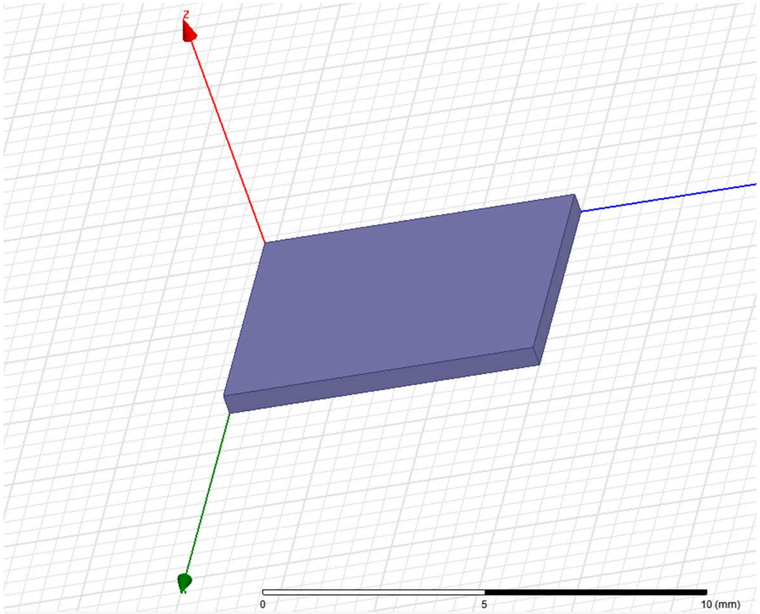


Figure 5

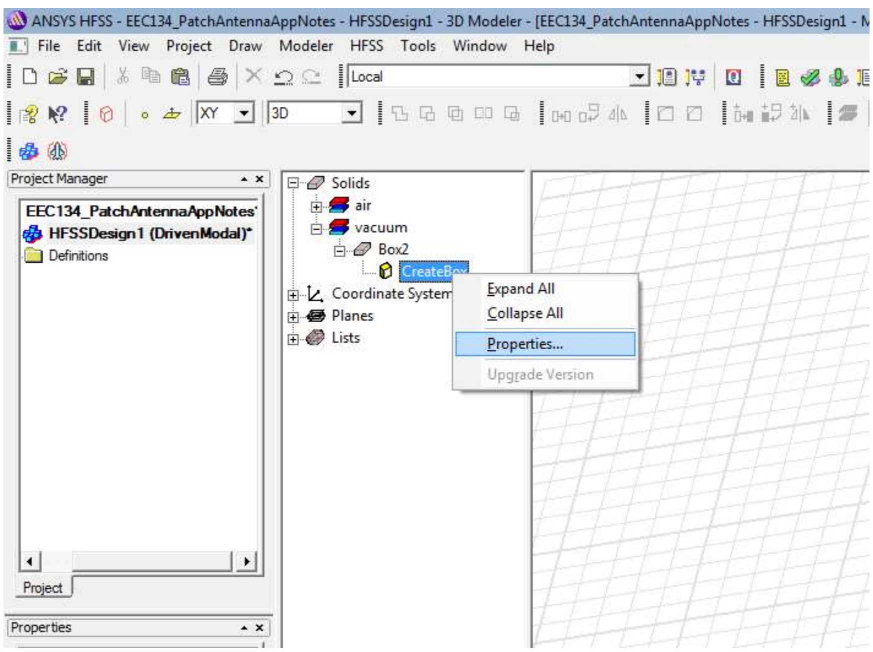
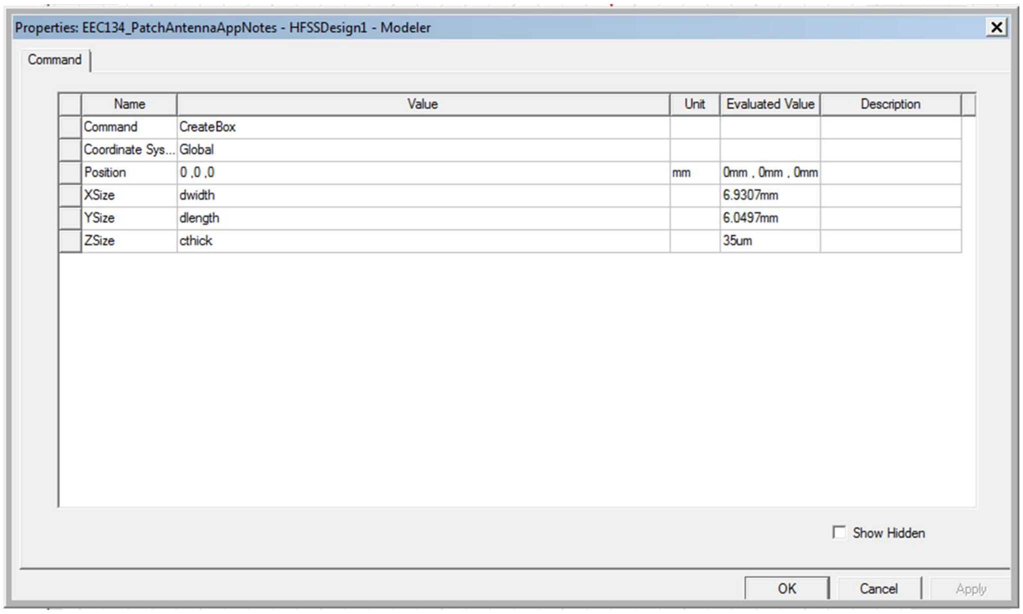


Figure 6 - After drawing the box, you have to right click "CreateBox" and then click on "Properties" in order to edit the dimensions.



Name	Value	Unit	Evaluated Value	Description
Command	CreateBox			
Coordinate Sys...	Global			
Position	0 ,0 ,0	mm	0mm , 0mm , 0mm	
XSize	dwidth		6.9307mm	
YSize	dlength		6.0497mm	
ZSize	cthick		35um	

Figure 7 – Dimensions of the dielectric solid

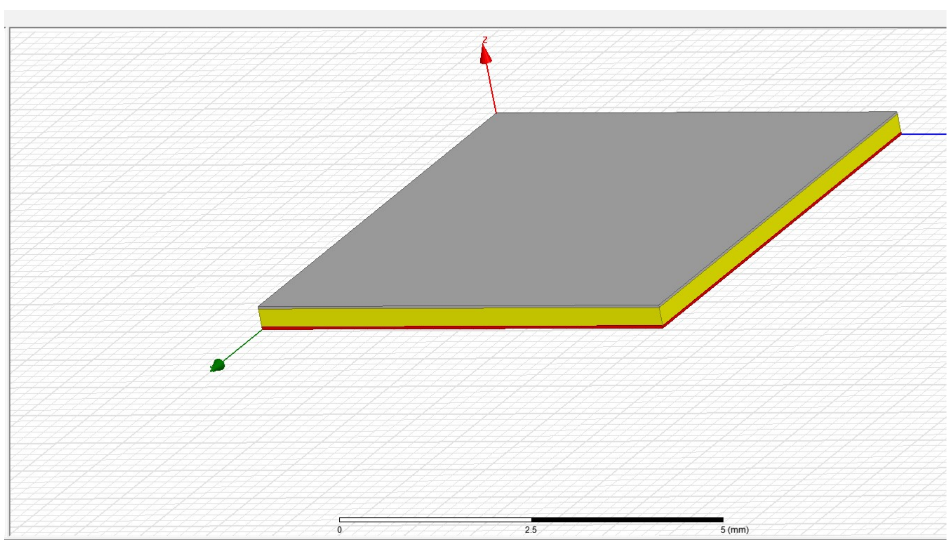
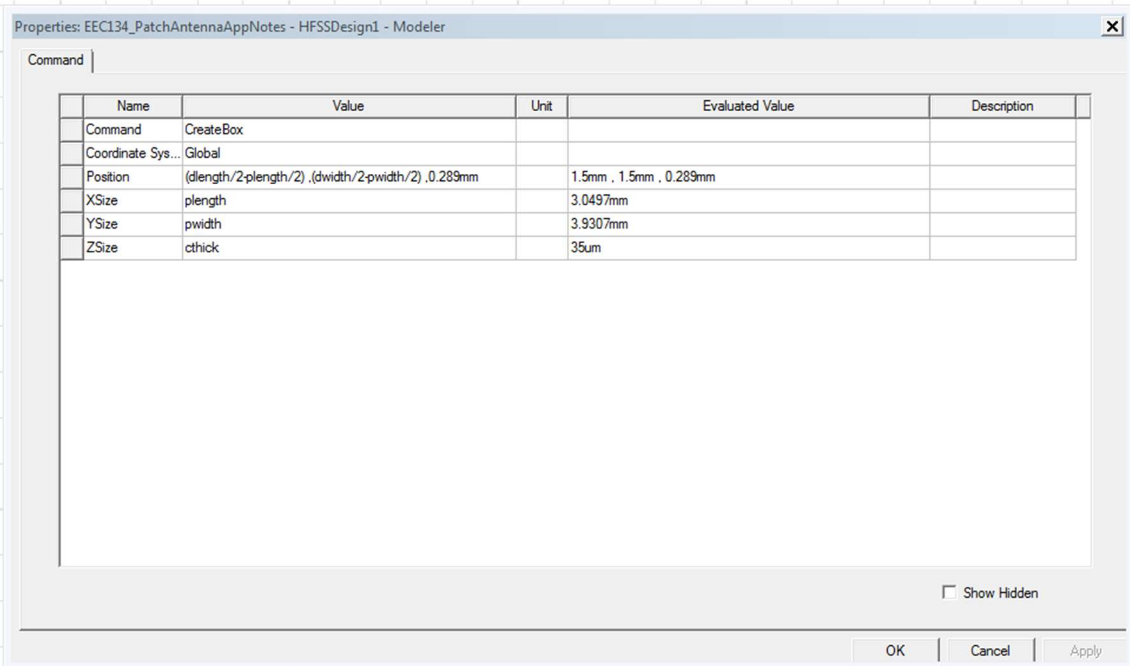


Figure 8 – From bottom to top: Ground plane, Dielectric plane, copper/antenna plane

The same steps are performed for the dielectric and the copper plane. Another step, for convenience, is to change the name of “Box2” (red), “Box3” (yellow), and “Box4” (grey) to ground, dielectric, and antenna, respectively. I prefer to have the patch antenna centered on the dielectric, so refer to the positions of the Antenna image. Here, it is easily noticeable that the way “Position” box contains an equation in reference to the position of the model. This is done so that the Antenna is directly centered on the dielectric surface.


Properties: EEC134_PatchAntennaAppNotes - HFSSDesign1 - Modeler

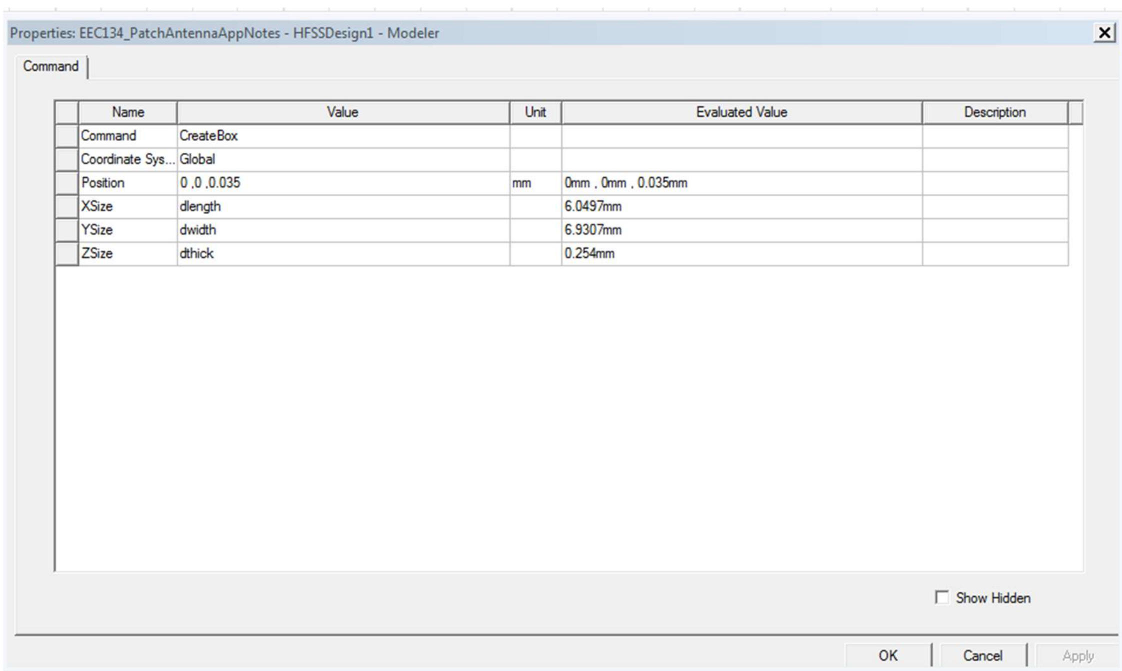
Command

Name	Value	Unit	Evaluated Value	Description
Command	CreateBox			
Coordinate Sys...	Global			
Position	(dlength/2-plength/2) ,(dwidth/2-pwidth/2) ,0.289mm		1.5mm , 1.5mm , 0.289mm	
XSize	plength		3.0497mm	
YSize	pwidth		3.9307mm	
ZSize	cthick		35um	

Show Hidden

OK Cancel Apply

Figure 9 Dimensions for the Antenna


Properties: EEC134_PatchAntennaAppNotes - HFSSDesign1 - Modeler

Command

Name	Value	Unit	Evaluated Value	Description
Command	CreateBox			
Coordinate Sys...	Global			
Position	0 ,0 ,0.035	mm	0mm , 0mm , 0.035mm	
XSize	dlength		6.0497mm	
YSize	dwidth		6.9307mm	
ZSize	dthick		0.254mm	

Show Hidden

OK Cancel Apply

Figure 10 - Dimensions for the dielectric solid

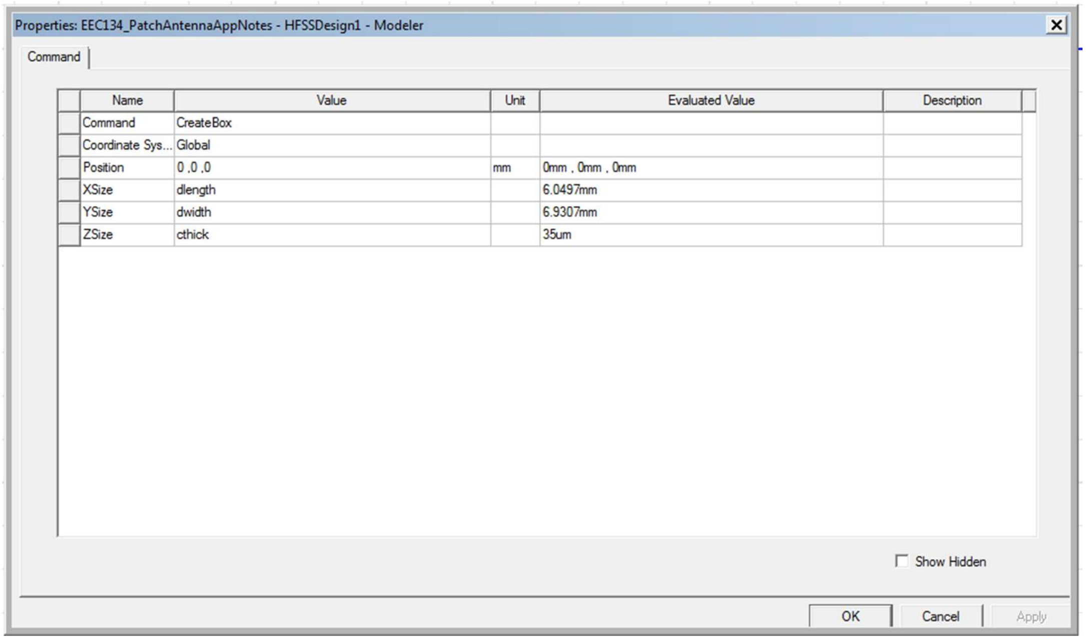


Figure 11 - Dimensions of the ground plane

The next step is to create the microstrip feedline that connects to the antenna and the model that will be used to create the inset depth. This can be created by using the "Draw Box" tool. You draw the microstrip line so that it lays on top of the dielectric plane and you can also use the corners of the antenna as a reference in order to ensure that the thickness of your box are correct. The lengths and widths of the microstrip line was obtained from using the LineCalc program that is found on Keysight's Advanced Design Systems. The dimensions of the box for the inset depth is approximately 0.8913 mm by 0.3138 mm. Note that 0.3138 is approximately 3 times the width of the microstrip feedline. After creating the microstrip feedline and the inset depth box, you can use the "Move" tool to center the feedline and the inset depth with the middle edge of the Antenna, see Fig. 14 for what the setup should look like.

The next step after creating and aligning the feedline and the inset depth is to "Subtract" the inset depth from the Antenna, and then to "Unite" the feedline and the Antenna. It must be done in this order to ensure the correct outcome. Refer to the following figures on how to subtract the two models.

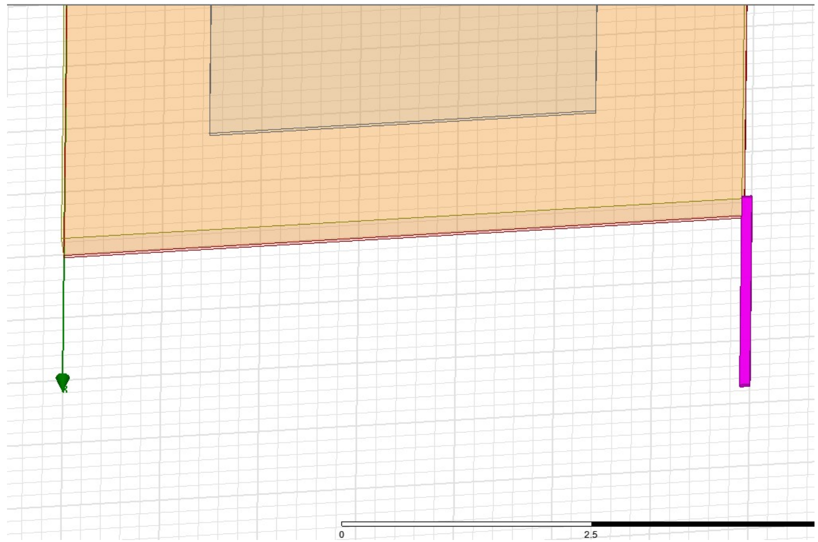


Figure 12 – Microstrip feedline

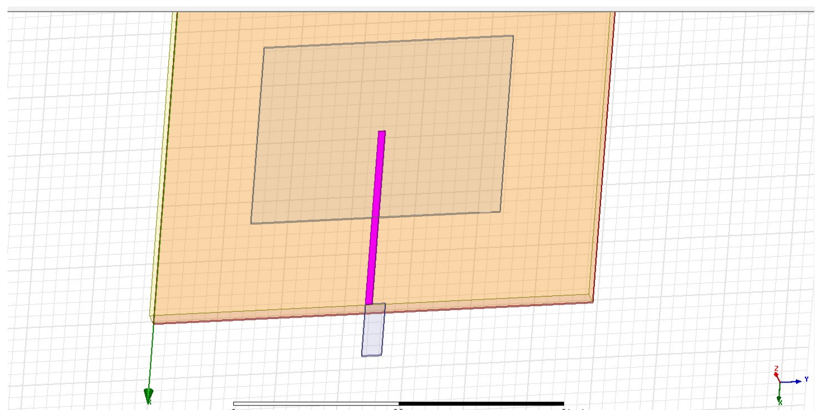


Figure 13

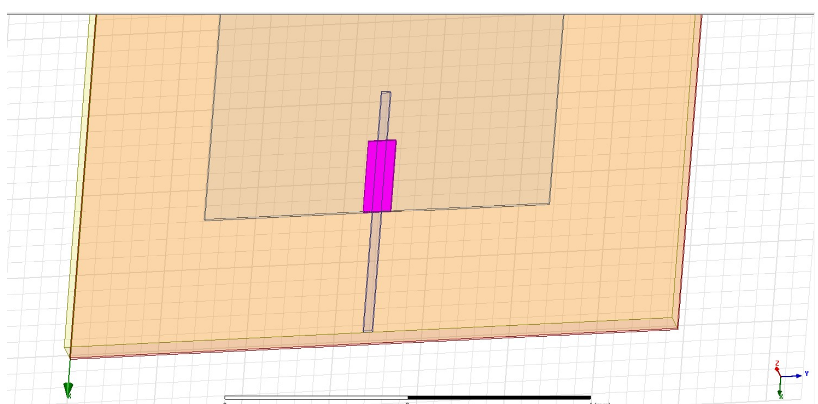


Figure 14

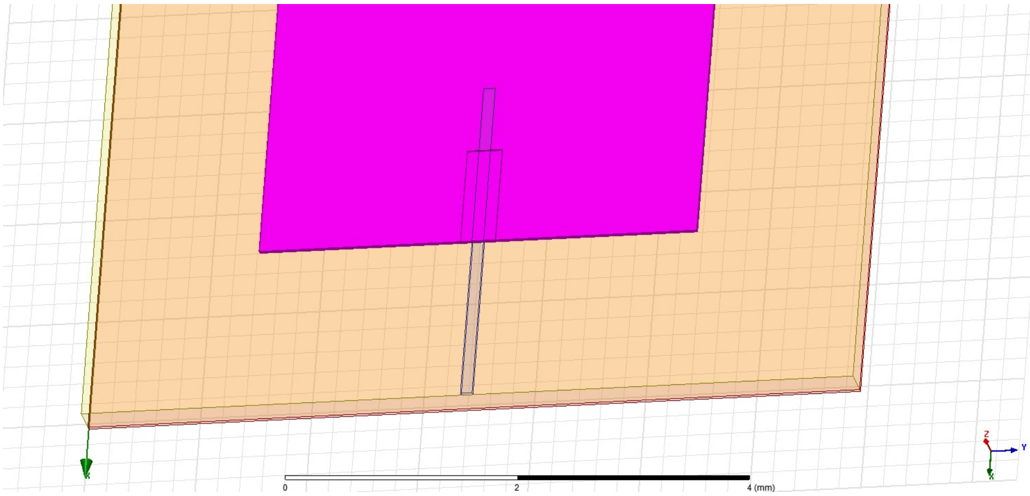


Figure 15 - Select the Inset Depth and the Antenna and then select the "Subtract" tool in the toolbar

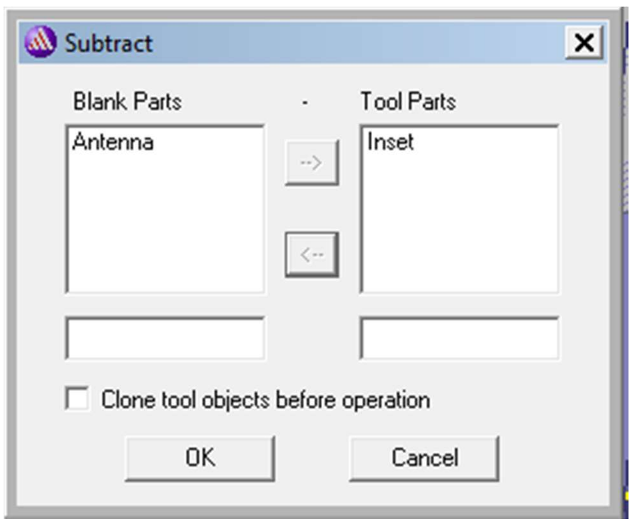


Figure 16 - A window looking like this should appear. Make sure the "Antenna" is on the left side while the inset is on the right side.

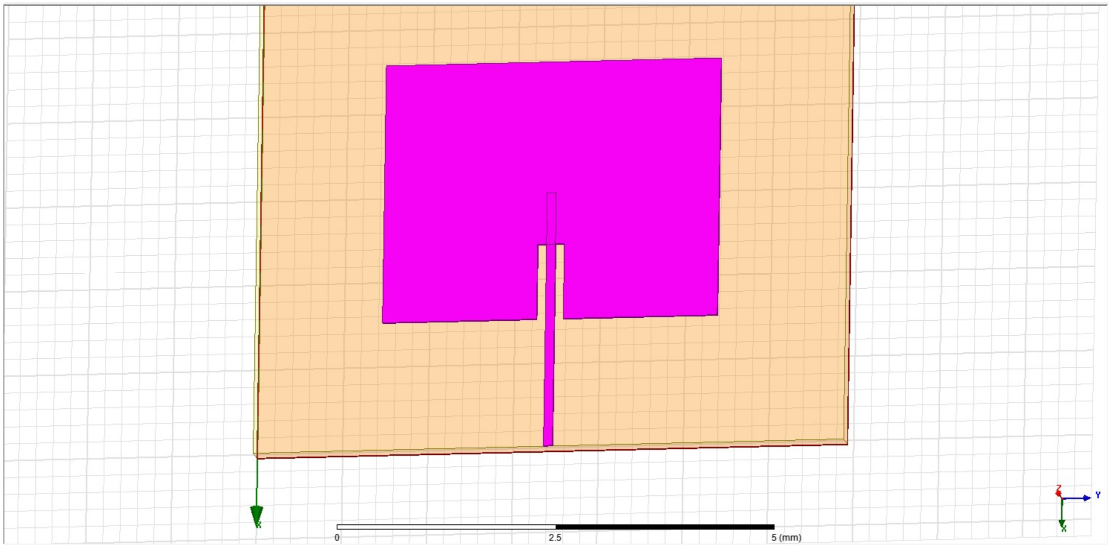


Figure 17 - After the subtraction is complete, you must unite the remaining two models. This is easily done by using the "Unite" tool in the toolbar

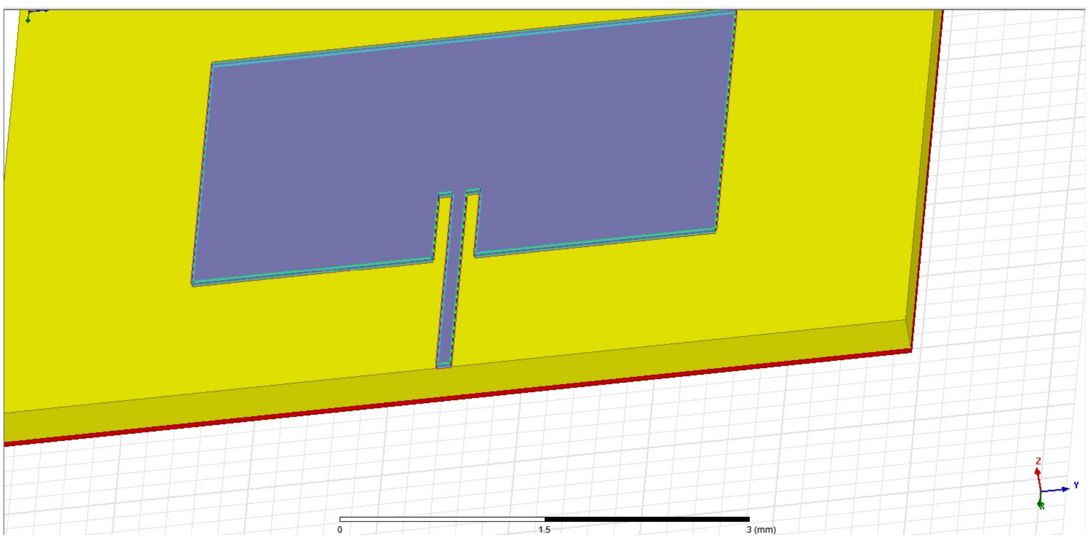


Figure 18 - The final inset fed microstrip patch antenna model.

Setting Up for Simulation

Solution Setup

Prior to being able to simulate the microstrip patch antenna, you have to set up the Analysis/Solution. To do this, right click "Analysis" and then "Add Solution Setup". After doing this, a window similar to "Driven Solution Setup" should appear. Change the "Solution Frequency" to the frequency that you have designed your antenna for. In this case, the patch antenna was design to be resonant at a frequency of 25 GHz, thus the "Solution Frequency" is set to 25 GHz. Another parameter that can be changed is the "Maximum Number of Passes". Increasing this will increase the amount of time it takes to simulate your design, however the results will be more accurate. .After this, you have to set up the "Frequency Sweep" of your simulation. The window of the frequency sweep should look like that shown in Fig. 21. Change the Start/Stop frequency to the upper and lower frequency that you are interested in looking at. For this case, the Start/Stop frequency is 24 GHz and 26 GHz because of the design of the radar system that this antenna is designed for. The antenna was designed for a 24 GHz transceiver, whose VCO operates between 24-26 GHz, hence the Start/Stop frequencies. Another parameter to change is to set the "Sweep Type" to Fast.

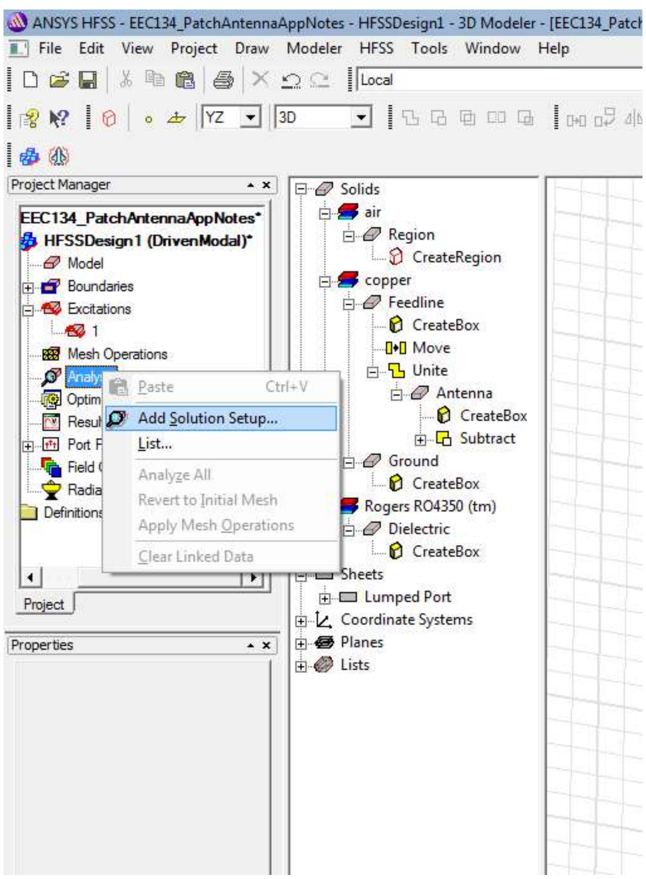


Figure 19

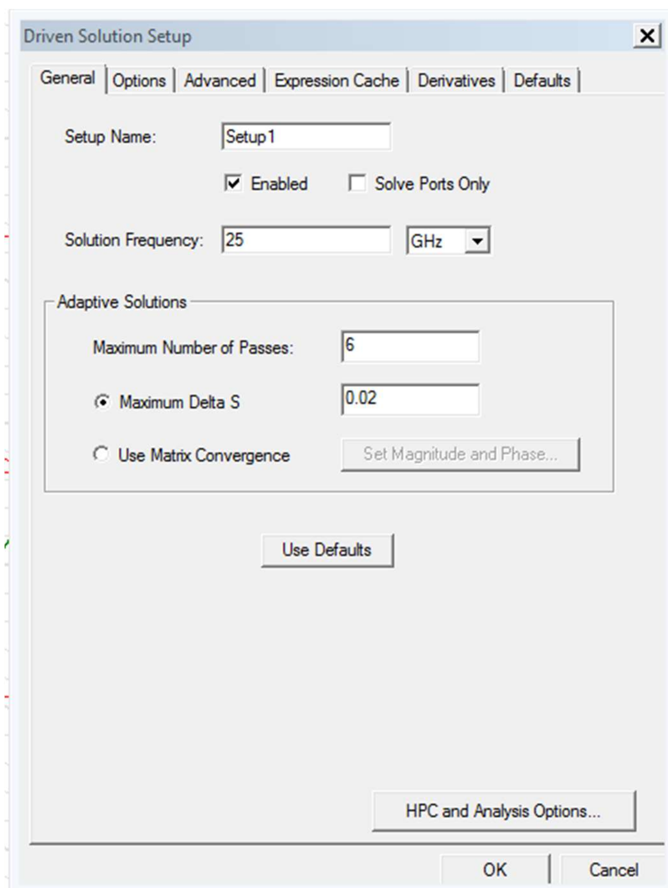


Figure 20

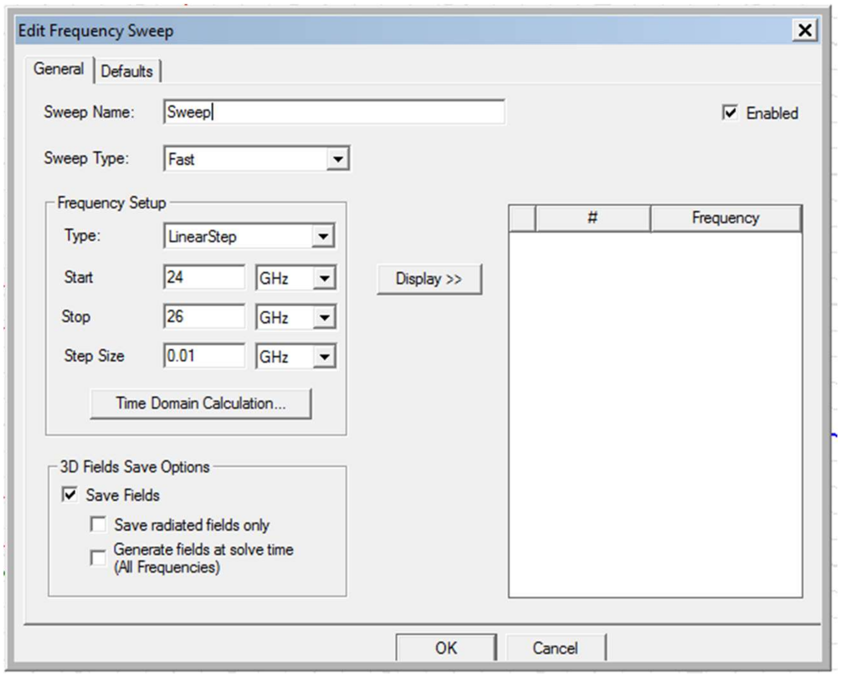

Edit Frequency Sweep
General
Defaults
Sweep Name:
Sweep
Enabled
Sweep Type:
Fast
Frequency Setup
Type:
LinearStep
Start
24
GHz
Stop
26
GHz
Step Size
0.01
GHz
Time Domain Calculation...
Display >>
#
Frequency
3D Fields Save Options
Save Fields
Save radiated fields only
Generate fields at solve time
(All Frequencies)
OK
Cancel

Figure 21

Excitation Setup

After setting up the analysis and the frequency sweep, the next step is to add an excitation over the port of the feedline, a radiation box, and set up the Far Field analysis. To do this, use the "Create Rectangle" tool and change the plane of reference from XY to YZ. For ease of drawing, I recommend to use the bottom corner of the Ground plane as a reference to draw the excitation rectangle and then use the "Move" tool to align the top middle edge of the excitation rectangle to the top middle edge of the microstrip feedline. Ideally, you want the excitation rectangle to be the same width as the feedline and the height to be the sum of the thickness of ground plane, dielectric plane, and antenna plane added together. After this process is complete, right-click the excitation rectangle and select "Assign Excitation" and then "Lumped Port".

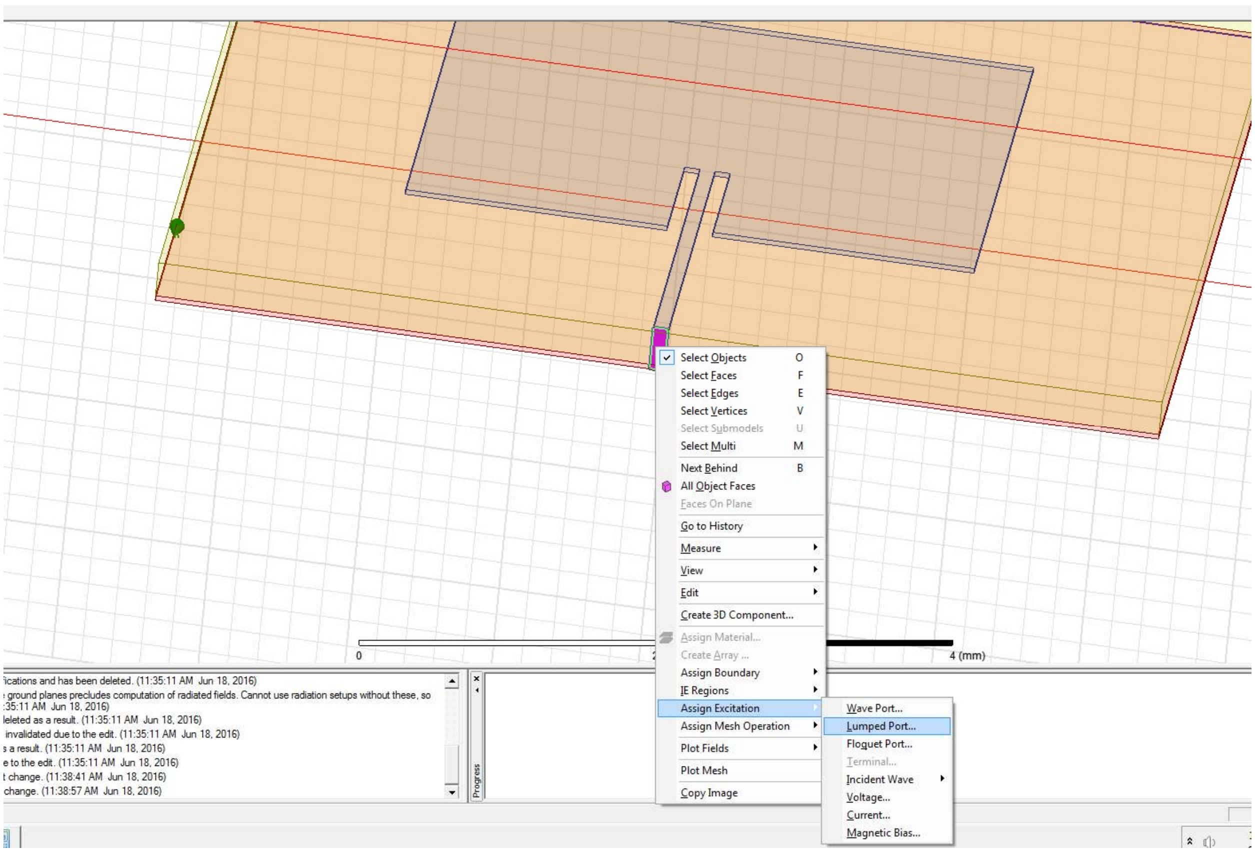


Figure 22

Afterwards, change the impedance values to the impedance of the microstrip feedline that you designed for. After that, click next until you get to the menu shown below. Click on the drop-down menu and select "New Line". Draw the excitation integration line from the bottom of the ground plane, over the excitation rectangle, to the top of the microstrip feedline. Setting up the excitation is now complete.

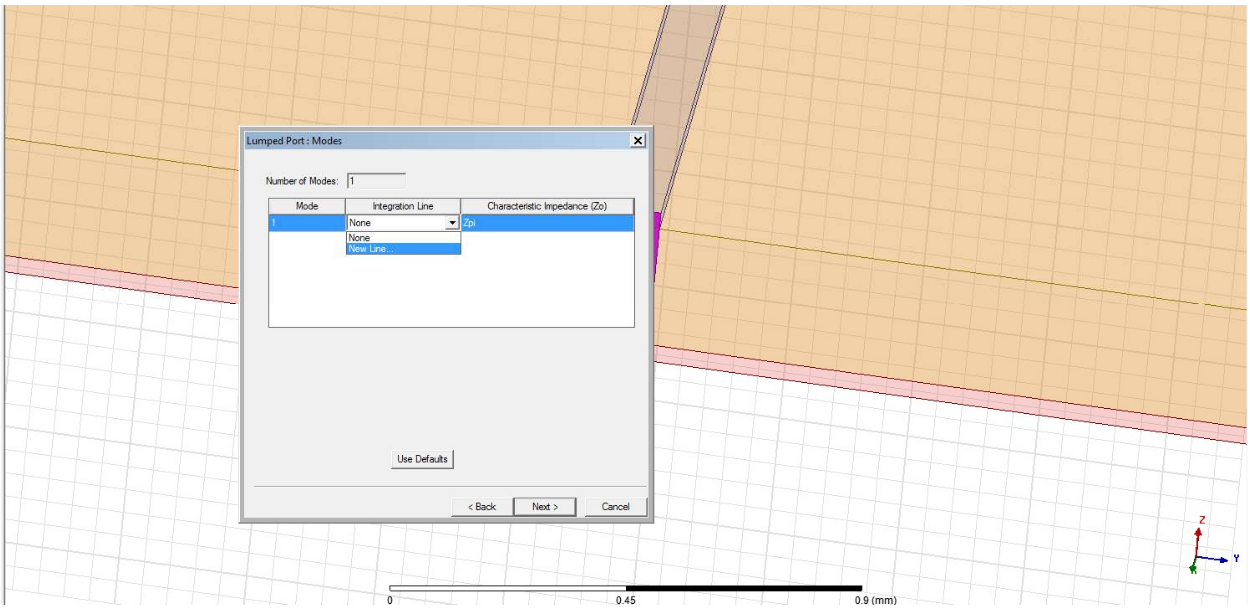


Figure 23

Radiation Boundary Setup & Assignment Material Type

Before being able to simulate, you must create an air box that surrounds the entire patch antenna. After this, the next step to assigning the boundary is to right-click the air box and then select "Assign Boundary" and then select "Radiation…". There are no parameters to change, so you can just accept default settings. Another thing to do after this is to change the material of all of the components. This is done by selecting "Assign Material" and this selecting the material of your model from the ones that are available. The material of the ground plane and the antenna is "Copper" while the material of the dielectric is "Rogers RO4350". The material assigned to the air box is "air".

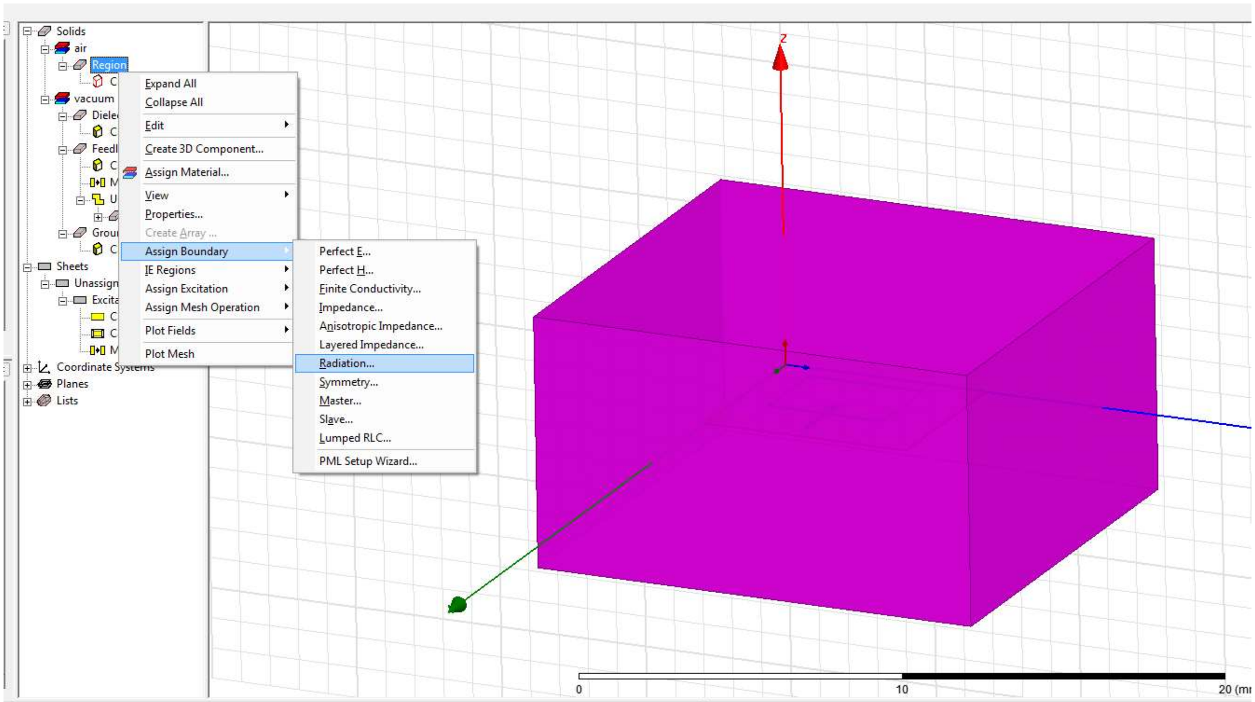


Figure 24

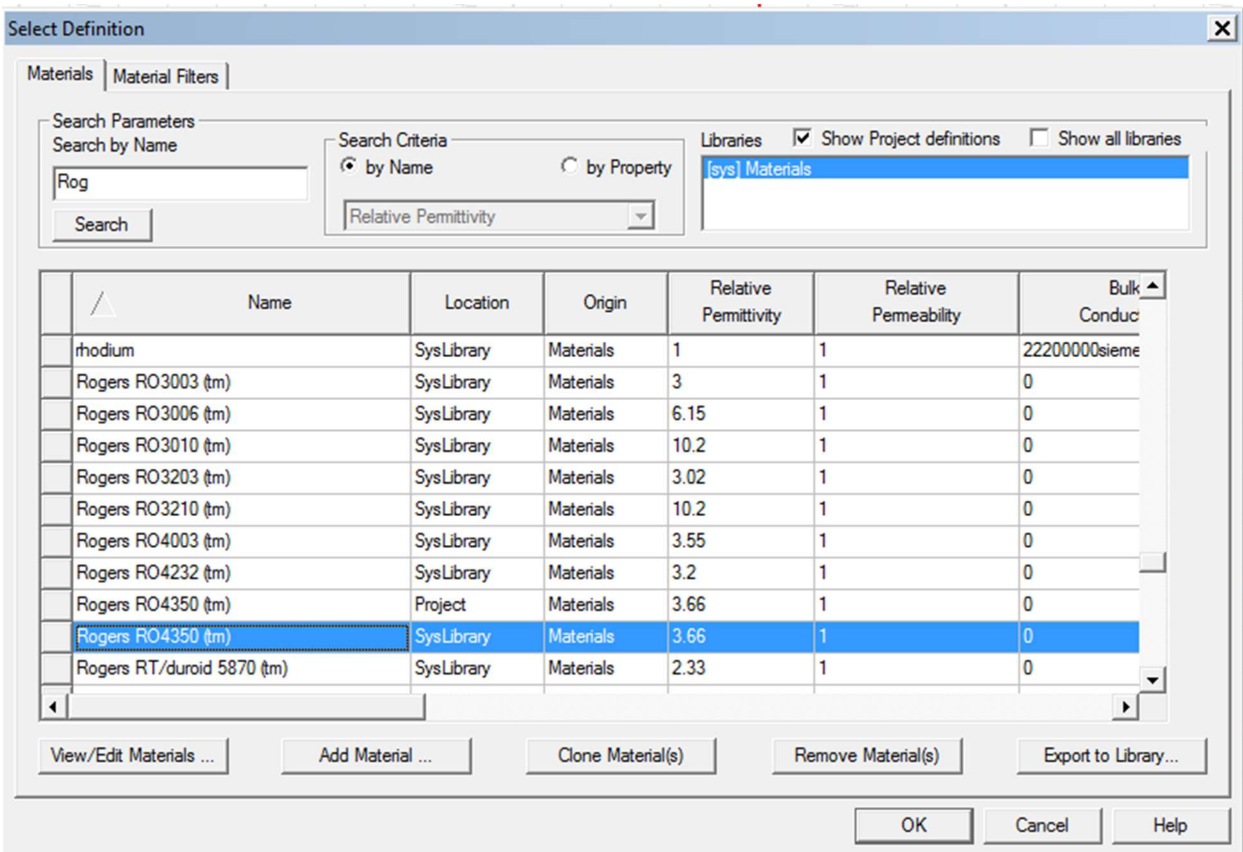

Select Definition
Materials
Material Filters
Search Parameters
Search by Name
Rog
Search
Search Criteria
by Name
by Property
Relative Permittivity
Libraries
Show Project definitions
Show all libraries
[sys] Materials
Name
Location
Origin
Relative Permittivity
Relative Permeability
Bulk Conduc
rhodium SysLibrary Materials 1 1 22200000sieme
Rogers RO3003 (tm) SysLibrary Materials 3 1 0
Rogers RO3006 (tm) SysLibrary Materials 6.15 1 0
Rogers RO3010 (tm) SysLibrary Materials 10.2 1 0
Rogers RO3203 (tm) SysLibrary Materials 3.02 1 0
Rogers RO3210 (tm) SysLibrary Materials 10.2 1 0
Rogers RO4003 (tm) SysLibrary Materials 3.55 1 0
Rogers RO4232 (tm) SysLibrary Materials 3.2 1 0
Rogers RO4350 (tm) Project Materials 3.66 1 0
Rogers RO4350 (tm) SysLibrary Materials 3.66 1 0
Rogers RT/duroid 5870 (tm) SysLibrary Materials 2.33 1 0
View/Edit Materials ...
Add Material ...
Clone Material(s)
Remove Material(s)
Export to Library...
OK
Cancel
Help

Figure 25

Far-Field Setup

The final step before simulating the design is to perform a “Far Field Setup” so that you can measure the gain of the antenna. This is done by selecting “Radiation”, “Insert Far Field Setup”, and then “Infinite Sphere”. The following window should have the following setup. Change the Start/Stop of Phi to the ones shown below. You can also change the “Step Size” of the setup also. Changing the “Step Size” gives you a better resolution of the response when it is plotted.

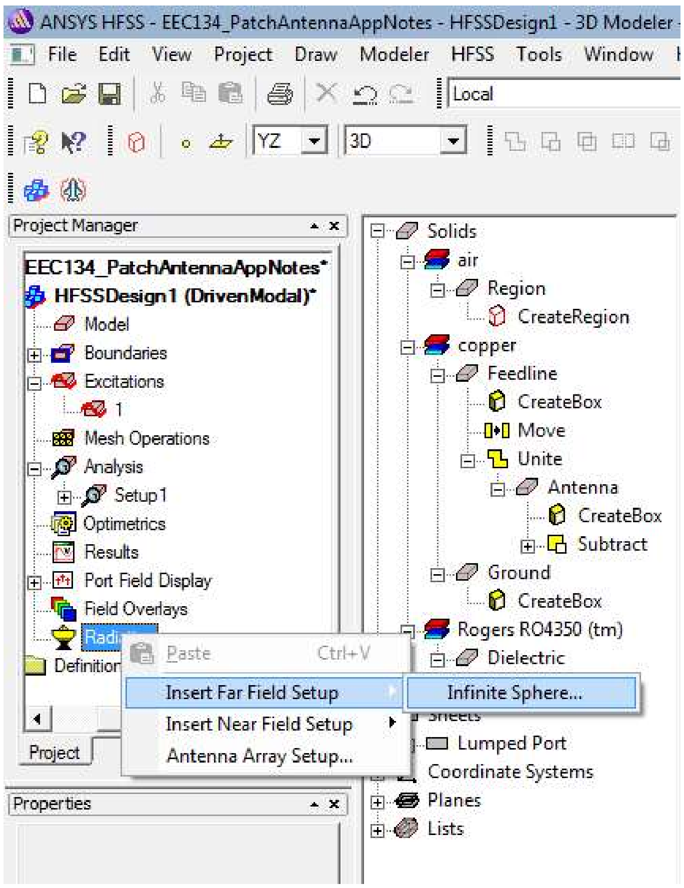


Figure 26

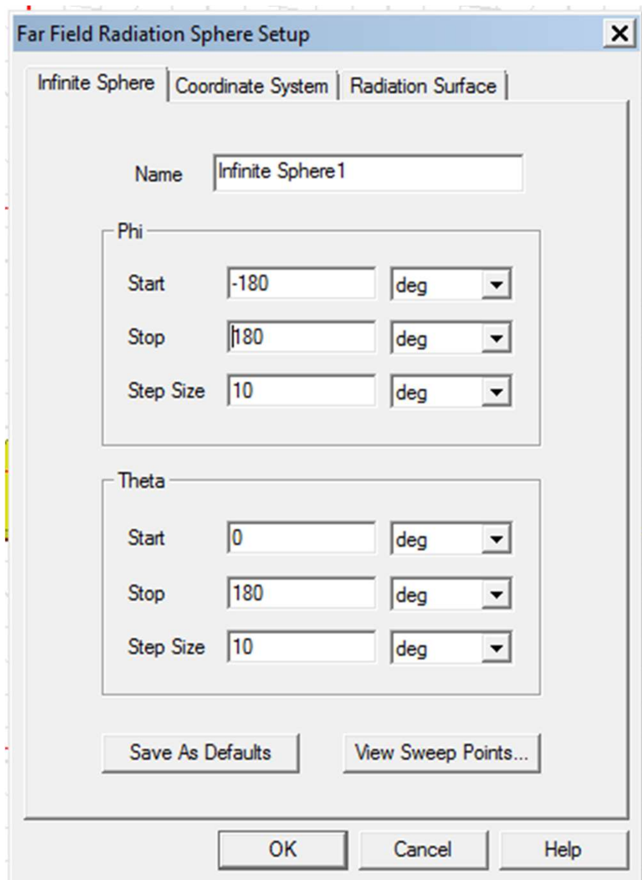


Figure 27

Simulation

After all of the above steps are complete, the final things to do are to perform a Validation Check and then proceed to performing the EM Analysis of the microstrip patch antenna.

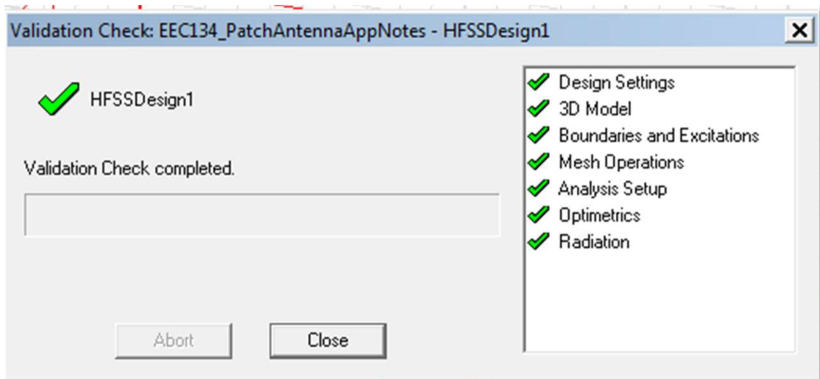


Figure 28

Most likely what will happen is that you will get an S11 (dB) response looking like something below. This is because there are some things that was not taken into consideration in designing the patch antennas, such as fringe fields, which would alter the results seen. This can be overcome by tuning some dimensions of the patch antennas such as: the lengths, widths, feedline dimensions, inset depth of the patch. This is where the design variables will make the tuning process much more manageable.

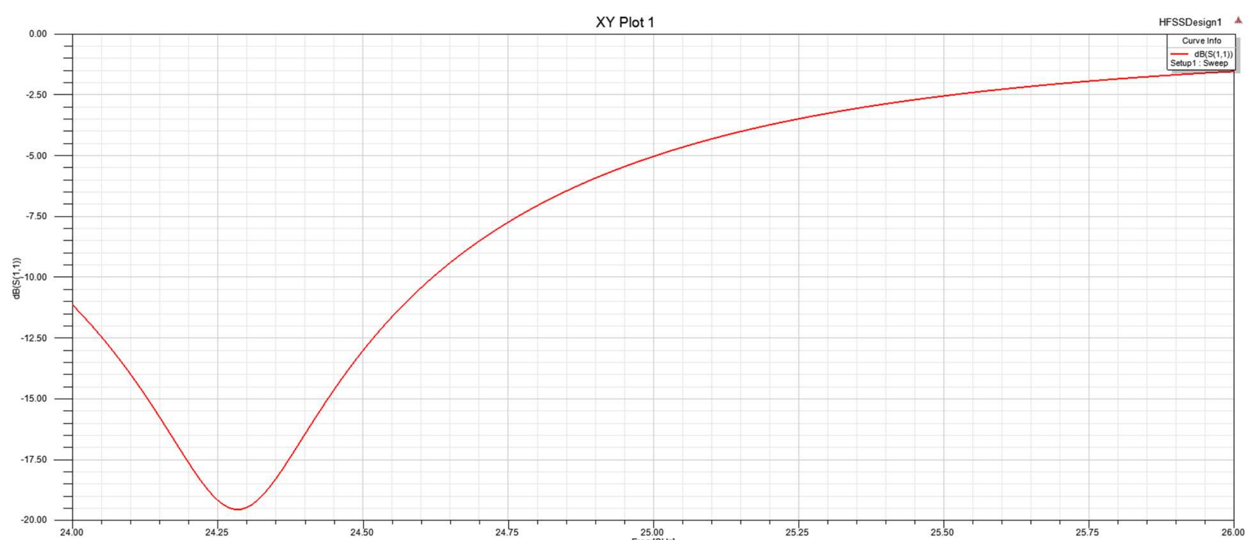


Figure 29

The tuning process is extremely tedious, but eventually you will get the results that you desire. The S11 (dB), which is return loss, shown below reveals that the antenna is resonant at 25 GHz, which was what it was designed for and it has a bandwidth of approximately 620 MHz.

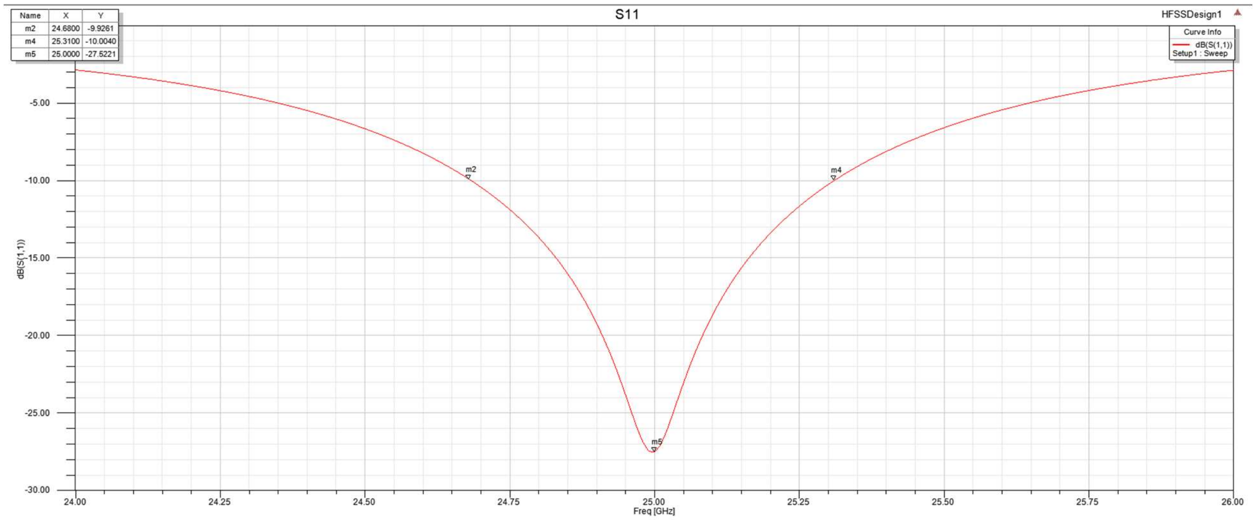


Figure 30

Conclusion

Overall, this short tutorial is meant to provide future students with an insight into designing a microstrip patch antenna for their radars. The modeling process for the antenna is relatively quick, however the tuning process will be the most time consuming and tedious portion of the design. Ultimately, the goal is to get a working patch antenna and then move onto incorporating that antenna into an array using a corporate feed network.