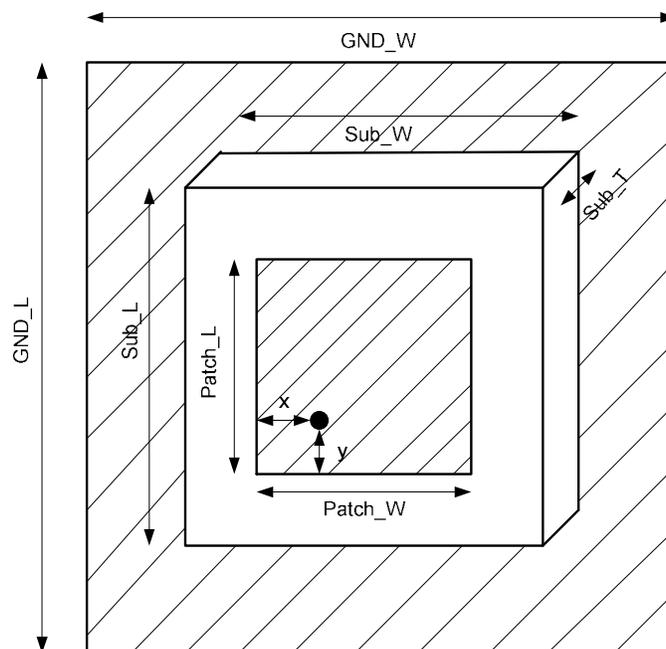


## FEKO Tutorial I

Mohammad S. Sharawi, Ph.D.  
Electrical Engineering Department

This tutorial will get you started with FEKO. FEKO is a full-wave electromagnetic field simulator that is based on the Method of Moments (MoM). It is a commercial software tool that can be used for antenna design, antenna placement analysis, RF structure performance prediction, EMC, as well as scattering problems and Bio-electromagnetics. A student version of FEKO can be obtained from [www.feko.info](http://www.feko.info), as well as the accompanying manuals. This is a quick step-by-step tutorial that is aimed to make learning and using the tool easier for new users.

1. We will design a Patch antenna with a Pin feed excitation (as opposed to Microstrip feed that is accomplished using a microstrip line with an Edge feed). Figure 1 shows the geometry of the antenna.



**Figure 1.**

2. Open your “CAD FEKO” program. You will have a window as in Figure 2.
3. From the “Model” pull down menu, choose “Model Unit” and select “(mm)”. Click “OK”.
4. If you will be performing some changes on the model geometry (usually the case), it is recommended to define variables and then use them as the dimensions in your design. For this exercise, you will define several variables.

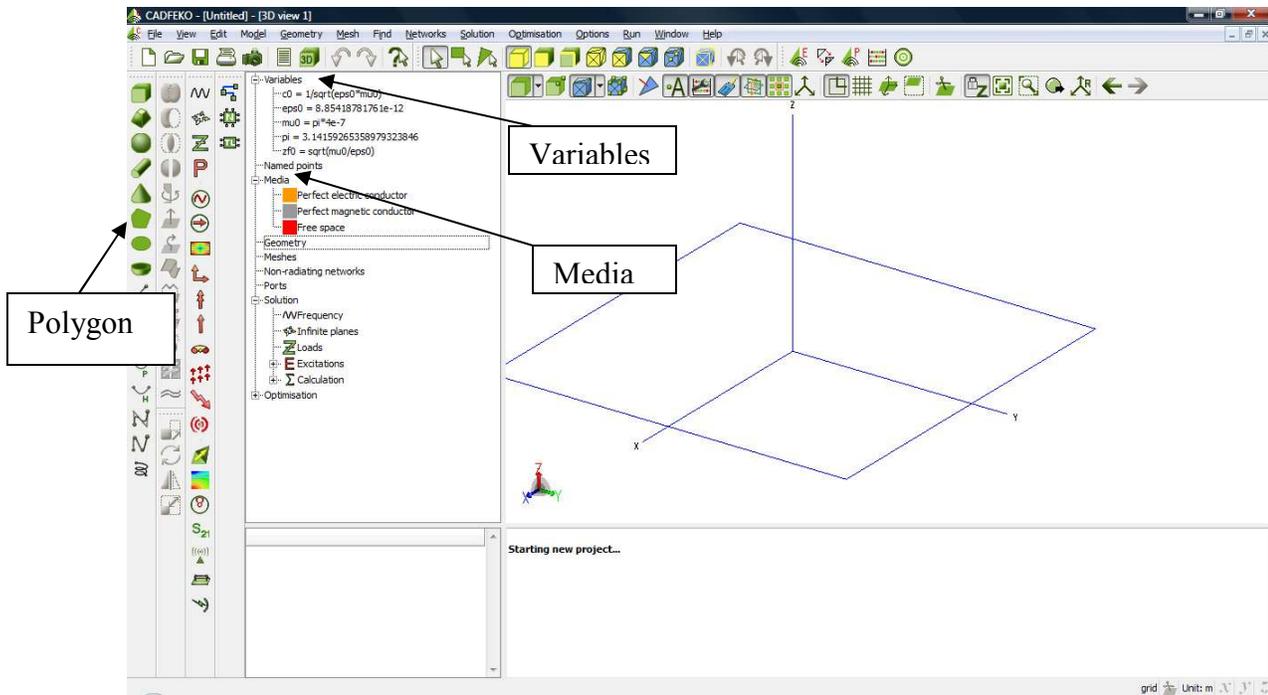


Figure 2.

- Right click the “Variables” tree on the left side of the screen, and click on “Add Variable” to be prompted with a window like Figure 3. Fill the variables one after the other for the geometry of interest.

The variables of interest as shown on Figure 1 are:

GND_L	=	90 mm
GND_W	=	90 mm
Sub_L	=	90 mm
Sub_W	=	90 mm
Sub_T	=	1.56 mm
Patch_L	=	45 mm
Patch_W	=	45 mm
X	=	16 mm
Y	=	12 mm

The feed point location (x,y) is not arbitrary, and several iterations are made to make sure the patch is at resonance in the location of the feed. Resonance is identified by having an  $|S_{11}|$  value of less than -10dB (remember Tutorial 0!).

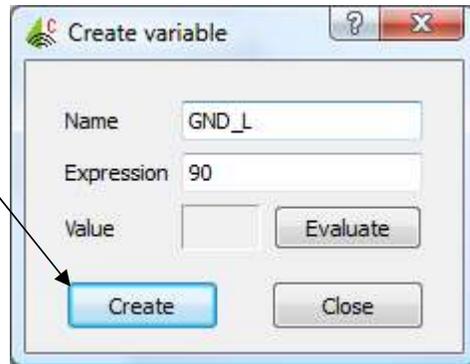


Figure 3.

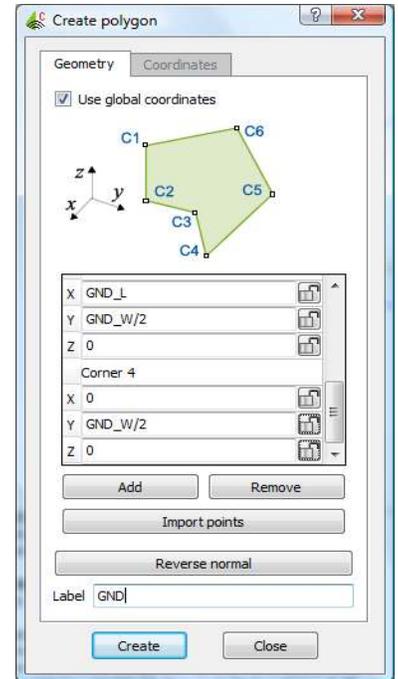


Figure 4.

6. Start from bottom to top. Thus, we will create the GND plane at the bottom. The dimensions of the GND plane are  $90 \times 90 \text{ mm}^2$ . The length is over the y-axis, while the width is over the x-axis. Click on the “Polygon” icon located at the far left of the program window. And fill in the coordinates of the 4-points as follows (you need to add an extra point, as the default for the number of points is 3 not 4):  
 $(0, -\text{GND\_W}/2, 0)$  ,  $(\text{GND\_L}, -\text{GND\_W}/2, 0)$  ,  $(\text{GND\_L}, \text{GND\_W}/2, 0)$  ,  
 $(0, \text{GND\_W}/2, 0)$

Name this polygon “GND”. This is illustrated in Figure 4.

7. After clicking “Create” in Figure 4 you will see the GND plane placed on the co-ordinate system as shown in Figure 5 (do not forget to Zoom To Extents).
8. By default, all the created geometries are assigned a “PEC” perfect element conductor material. If you want to use a dielectric material for example, you need to defined one, assign its properties and then use it for the geometry of interest. We will be using an FR-4 substrate above the GND plane that we created to print the antenna on. Thus, to create an FR-4 material, we need to right click in “Media” from the side menu, and select “Create Dielectric Medium”. Fill the window as shown in Figure 6. You have to know in advance the material properties to be able to predict the correct behavior of the material. Click “Create” after you fill in all the info of the material.

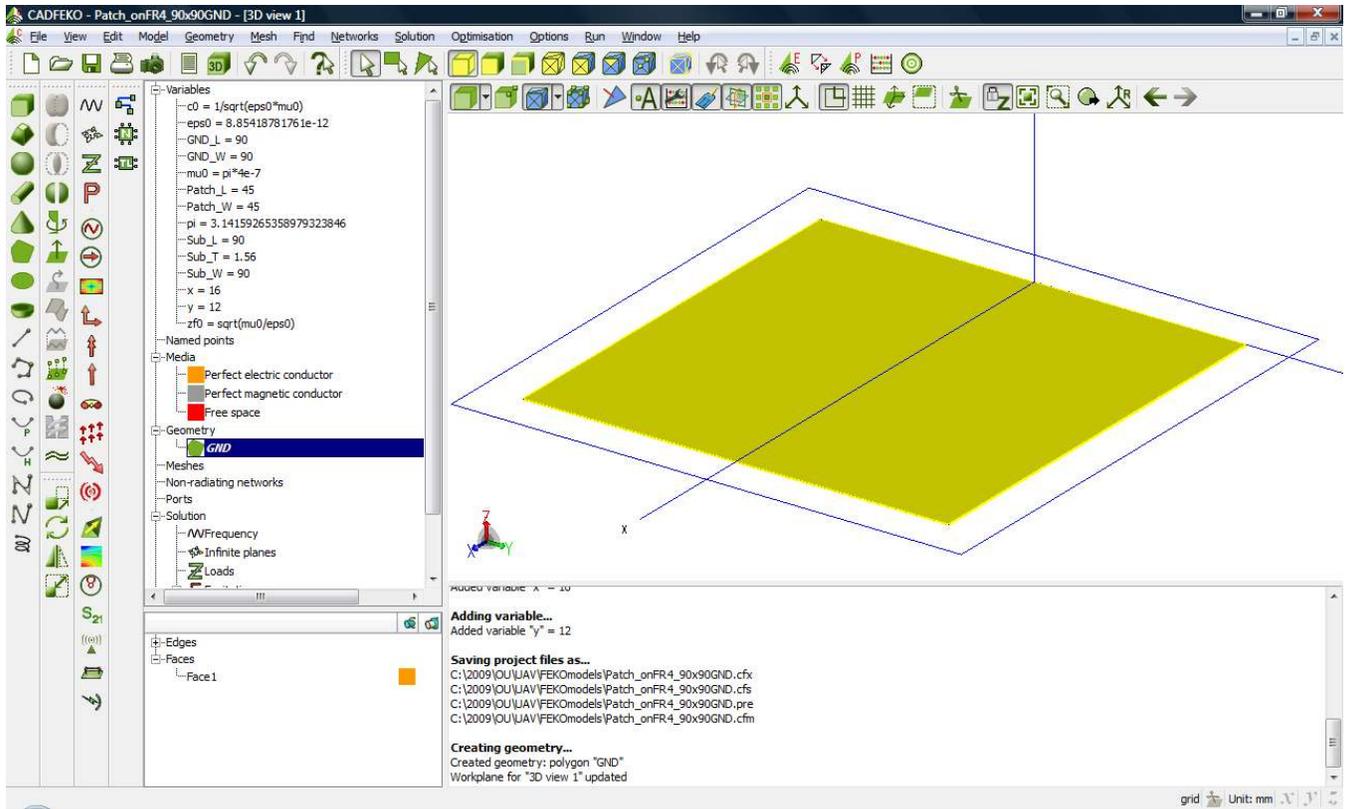


Figure 5.

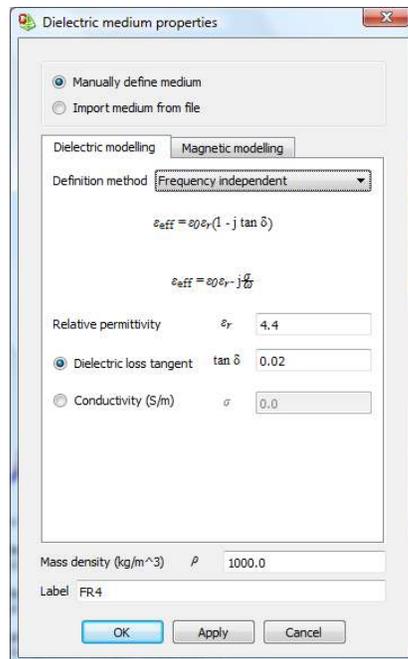


Figure 6.

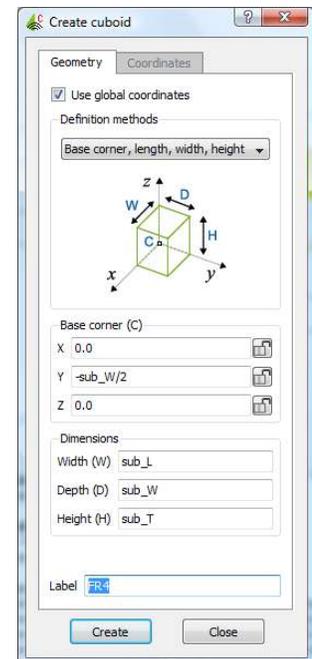


Figure 7.

9. Now, we can create the dielectric substrate. Choose “Cuboid” from the left side menu, and substitute the dimensions as shown in Figure 7, then hit “Create”. You will see the FR4 substrate on TOP of the GND plane you created. To make sure all is ok, rotate the view by clicking the left mouse button, and move the mouse around. Figure 8 shows one view that you might have (bottom view).

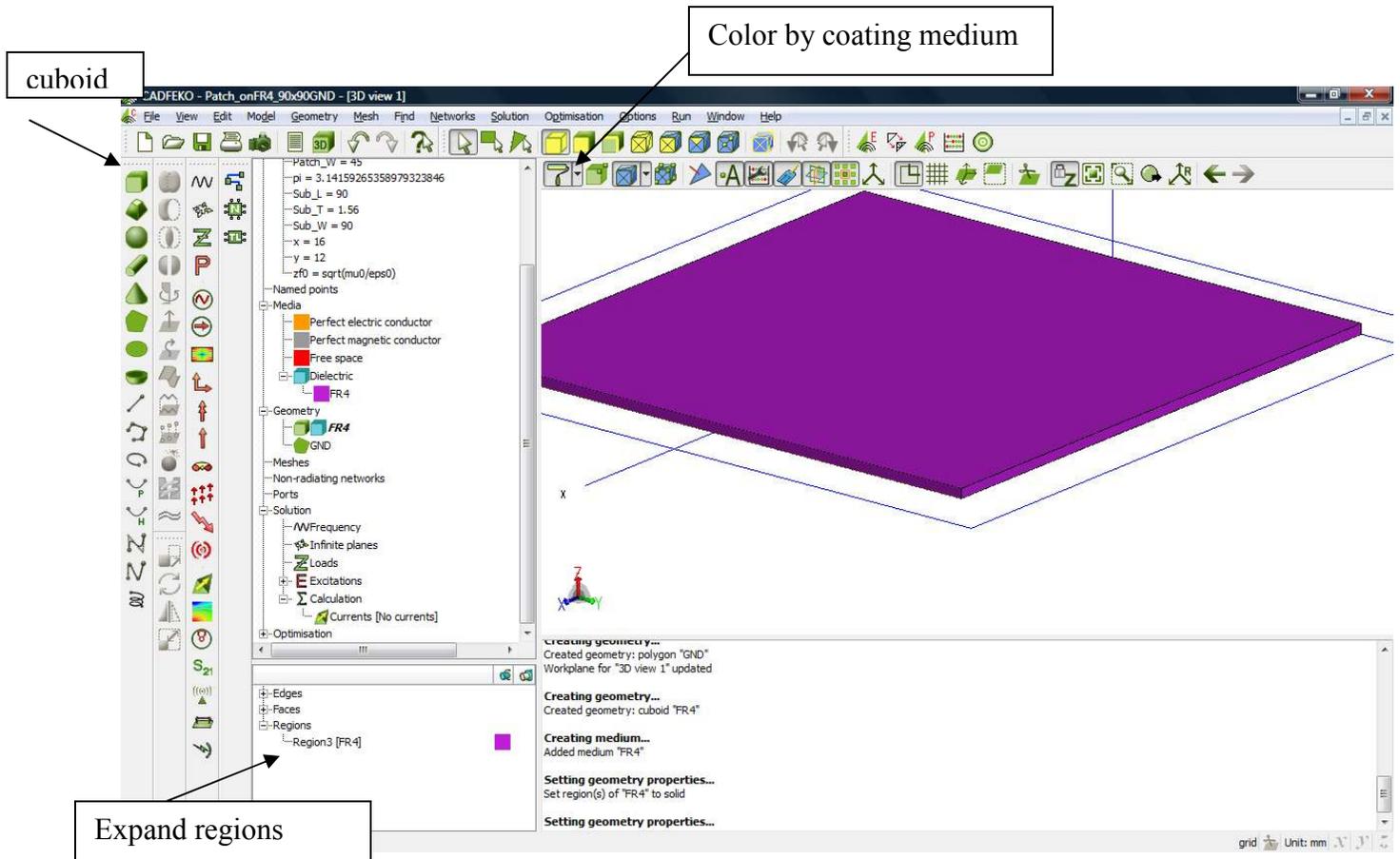


Figure 8.

10. To assign the dielectric material to the FR4 volume, you need to click on the FR4 geometry, and below the left side bar list, a new list shows up. Click on “Regions”, and then right click on the only region that shows up. Choose “Properties”, and choose “Medium” to be “FR4”. If you have defined more than one dielectric material, you need to choose which one you want to use for this region. After you choose the right material, click “Apply”. This is shown in Figure 9.

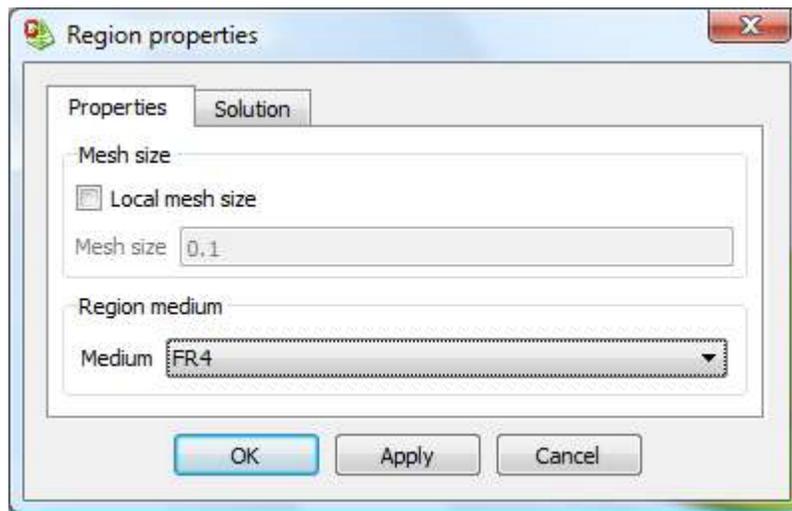


Figure 9.

11. To place the antenna patch, we need to place a polygon in the center of the substrate volume. To do this, select the “polygon” function from the left side menu, and choose the edges as:  $(Patch\_L/2, -Sub\_W/2+Patch\_W/2, Sub\_T)$ ,  $(Patch\_L/2, Sub\_W/2-Patch\_W/2, Sub\_T)$ ,  $(3*Patch\_L/2, Sub\_W/2-Patch\_W/2, Sub\_T)$ ,  $(3*Patch\_L/2, -Sub\_W/2+Patch\_W/2, Sub\_T)$ . Call this polygon “Patch”, and then hit “Create”.
12. The resulting geometry is shown in Figure 10.

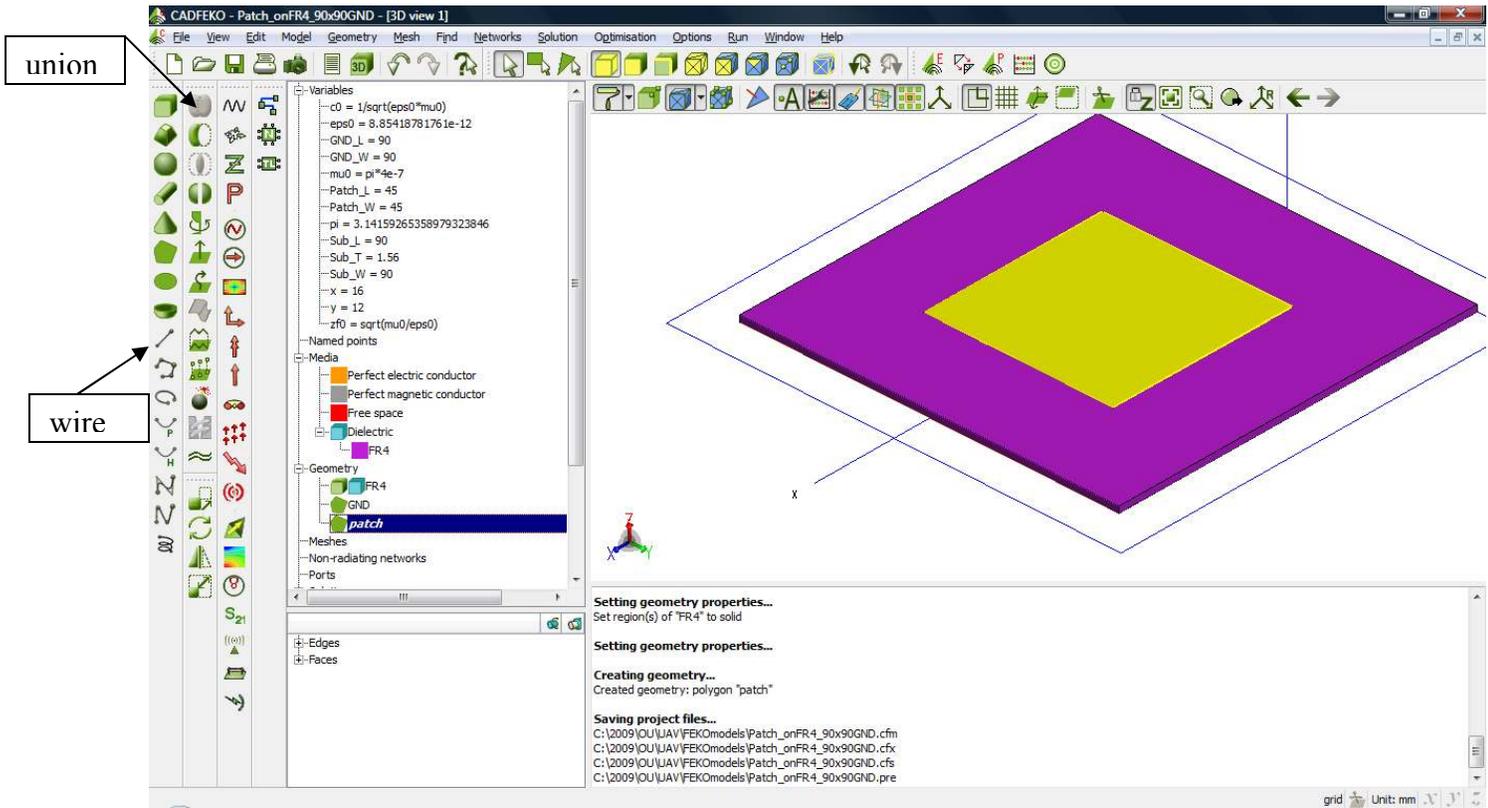


Figure 10.

13. To add the feeding pin, choose the “line” function. Choose the points as shown in Figure 11.
14. The geometry creation of the antenna is complete. Select all the geometry components and “Union” them using the union function in the side menu.
15. After the “Union” function, the properties of the metals get changed to that of the volume; i.e becomes a dielectric. Thus, we need to change the properties of the Patch and GND faces to “PEC” again. To do this, from the side menu, Click on “Union1”, and then from the bottom menu, expand the “Faces” tree. Start clicking on the faces one after the other to identify the patch and GND faces. This occurs by observing the highlighted portions on the 3D geometry. Once you find a desired face, i.e. Patch, right click on that face, and choose “Face Properties”, then change the “Face Medium” to “Perfect Element Conductor”, hit “Apply”. Repeat for the other face(s). This is shown in Figure 12.

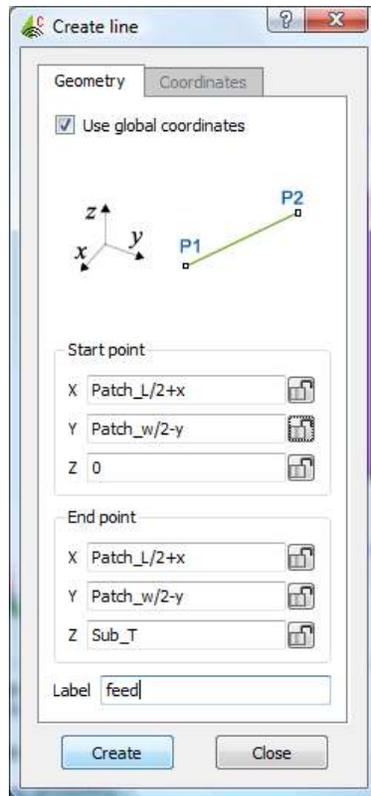


Figure 11.

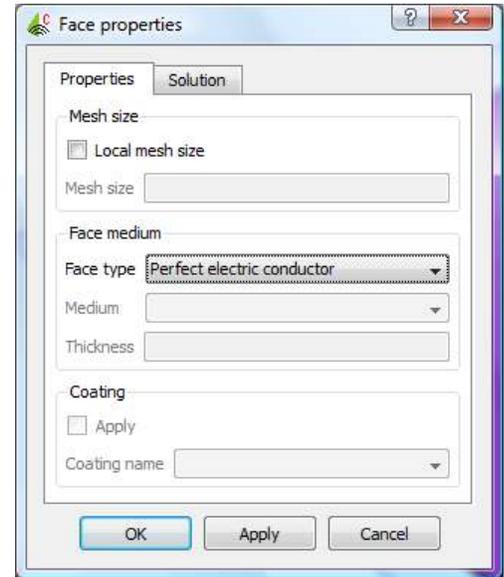


Figure 12.

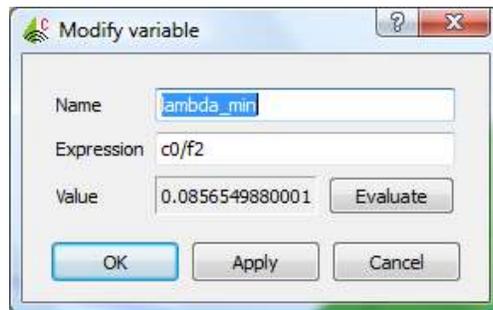


Figure 13.

16. We define  $F1=1.0e9$  and  $F2=2.0e9$  as the start and stop frequency variables. We will create the minimum lambda (wavelength) based on the highest frequency, thus  $\lambda_{min}=c0/F2$ . This variable is shown in Figure 13.
17. From the “Mesh” pull down menu, choose “Create Mesh”. Fill in the fields as shown in Figure 14. “Edge Length” and “Segment Length” are chosen by clicking the “suggest” button, then dividing by 4 (/4) [If you get an Error, you need to specify the Solution frequencies, so choose an interpolated analysis with f1 as the starting frequency and f2 and the end frequency]. (Usually for smaller structures we use lambda\_min variable for these two lengths, here if you use lambda\_min, the mesher will take a large amount of time to create an extremely fine grid).

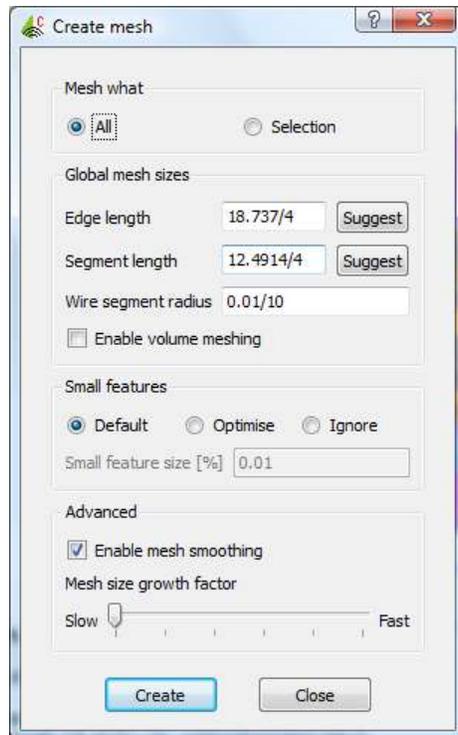


Figure 14.

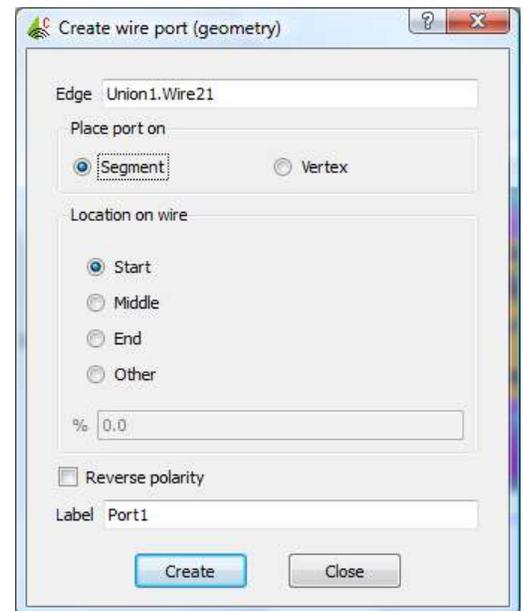


Figure 15.

18. After meshing is complete, you need to assign a port to feed the antenna with the excitation needed. From the side menu, right click “Ports” → “Create Port” → “Wire Port”. A window like Figure 15 pops up. From the geometry tree on the left side menu, click on the “Union1” geometry, expand the “Edges” tree in the bottom side menu, and choose the “Wire xx”. Automatically, the name of the wire is substituted in the port window as shown in Figure 15. Keep the label as “Port 1”, hit “Create”.
19. Observe the created port in the geometry. After port creation, we need to specify the frequency of simulation. We will first examine the resonance of the antenna, and thus we are interested in an  $|S_{11}|$  measurement. We need to sweep the frequency between F1 and F2 as shown in Figure 16.
20. Adding the excitation source is the step that follows. We will use a voltage source of amplitude 1 and phase 0. Figure 17 shows this window after right clicking the “Excitations” → “Voltage source”. Since we have only one port, then the excitation source will be automatically assigned to the single port. Hit “Create”. If you have multiple ports, then choose the port label from the “Port” menu in Figure 17.

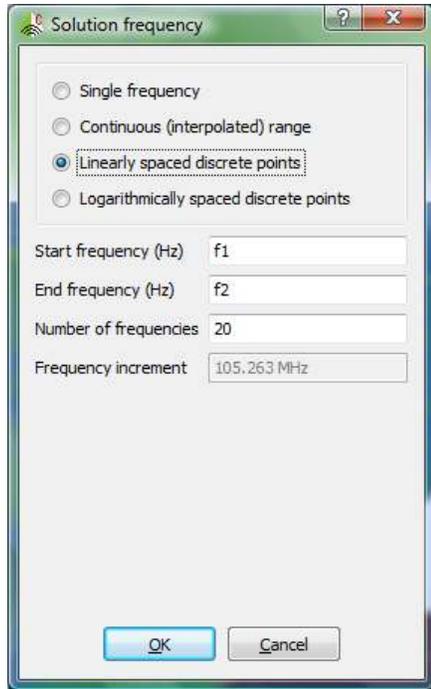


Figure 16.

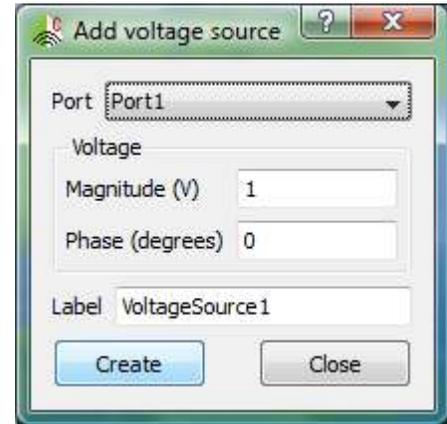


Figure 17.

21. Now, you can request the type of simulation/analysis you want to conduct. For this particular example, we are interested in the  $|S_{11}|$  measurement. Thus right click “Calculation” → “Request S-parameters”. You will get Figure 18. Click “Restore loads after calculation” then click “Create” and then “Close”.

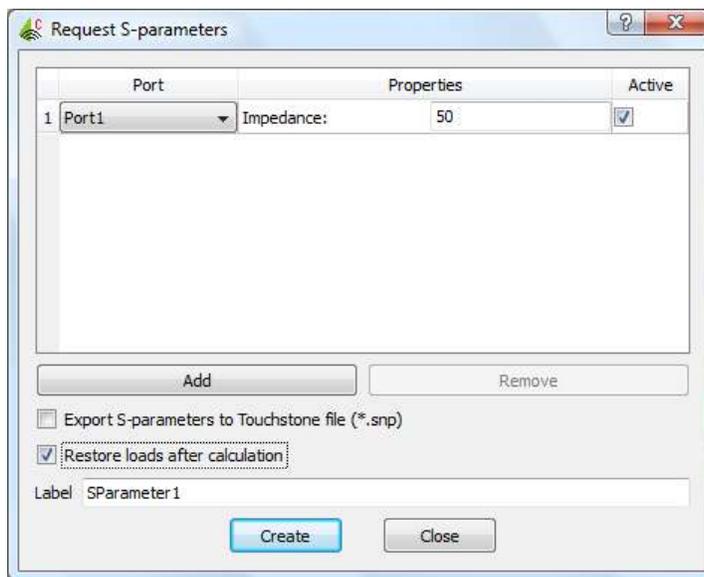


Figure 18.

22. You have to RE-MESH one more time before running the simulation to take into account the new port you created. Thus go to the “Mesh” pull down menu, and with the same values as before, just mesh again, and over-write the previous results.

23. Now you should have something that looks like Figure 19. Click on “Run FEKO” and wait for the simulation to finish\*.

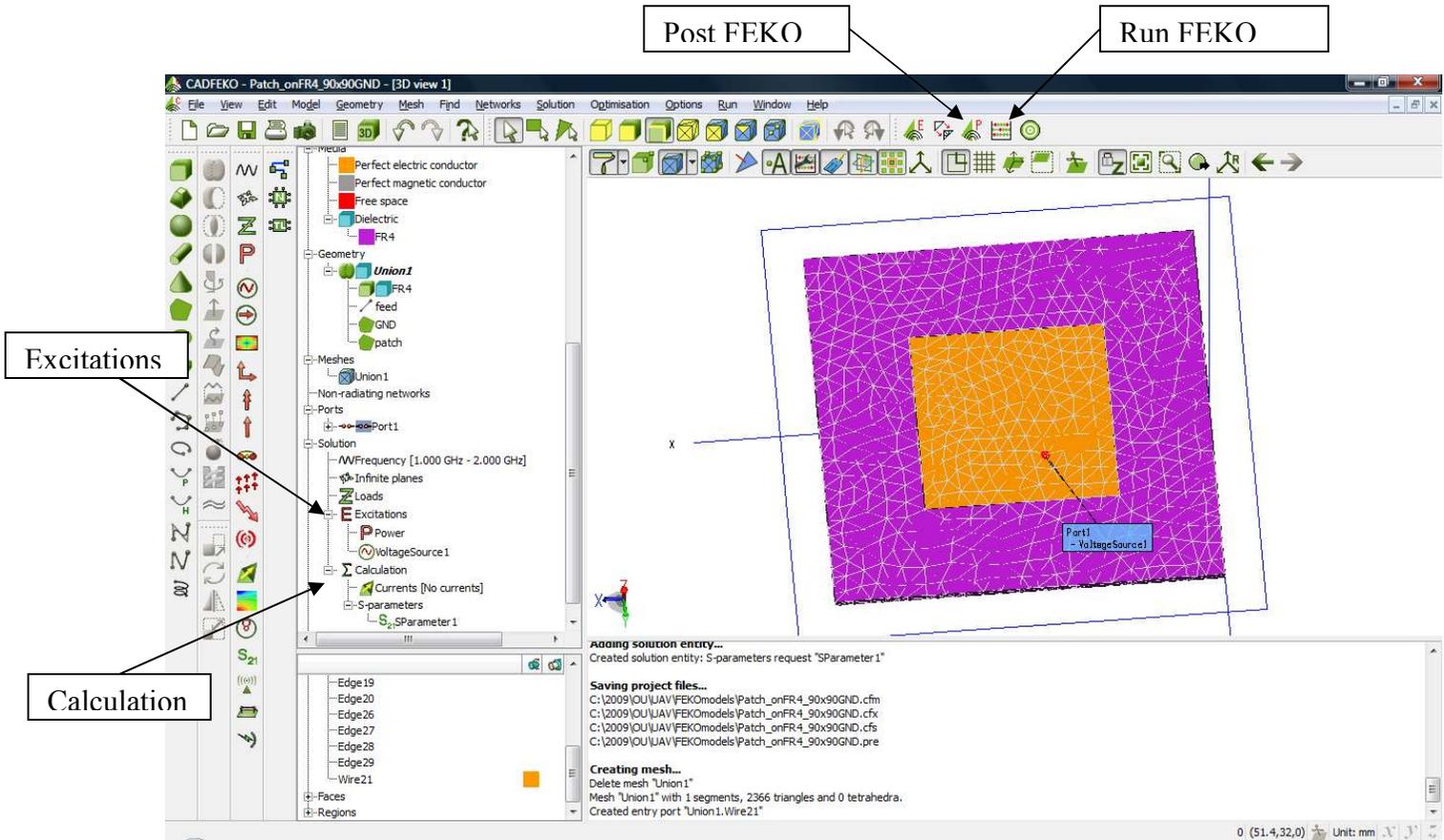


Figure 19.

24. After the simulation finishes, click on “Post FEKO” to plot/view the results.
25. After running “Post FEKO”, you will obtain a window as shown in Figure 14.
26. Click on the “Cartesian” icon, and a graph with Cartesian coordinates appears. Select the “S-Matrix” icon as shown in Figure 20. You will get the  $|S_{11}|$  measurement as a function of frequency. Choose the “dB” display.
27. Note that the Patch resonance is around 1.55 GHz (Figure 21). What is the -10 dB bandwidth for this antenna? Use the cursors to identify the bandwidth of this antenna. Which wireless service falls within the bandwidth of this antenna?
28. Other functionalities can be done from the menus provided.
29. Vary the pin location by +/- 1 or +/- 2 mm, and notice the effect on the resonance frequency and the BW. Choose 4 arbitrary combinations and plot the resonance frequency,  $|S_{11}|$  at resonance and the -10 dB bandwidth versus the feed location. Plot using Excel or MATLAB.

\* In case “Run FEKO” complains that the geometry is too large, or your license does not support the memory requirement of this geometry, you can always shrink a patch by 2, 4, 8, etc. But, your resonance frequency will shift, and you need to use a higher dielectric constant material. Please refer to your “Antenna” test book to take a look at the governing equations of creating a patch antenna. This is just an exercise. If you need more computing power out of FEKO, the full version is needed.

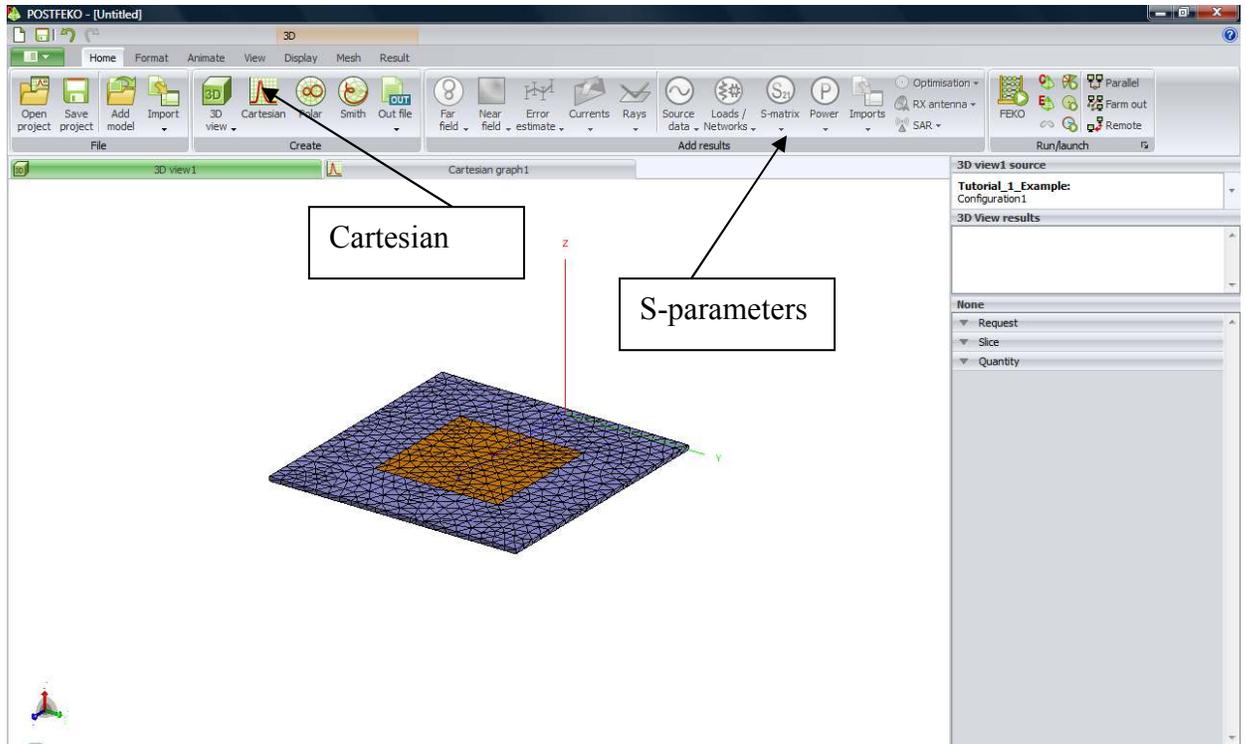


Figure 20.

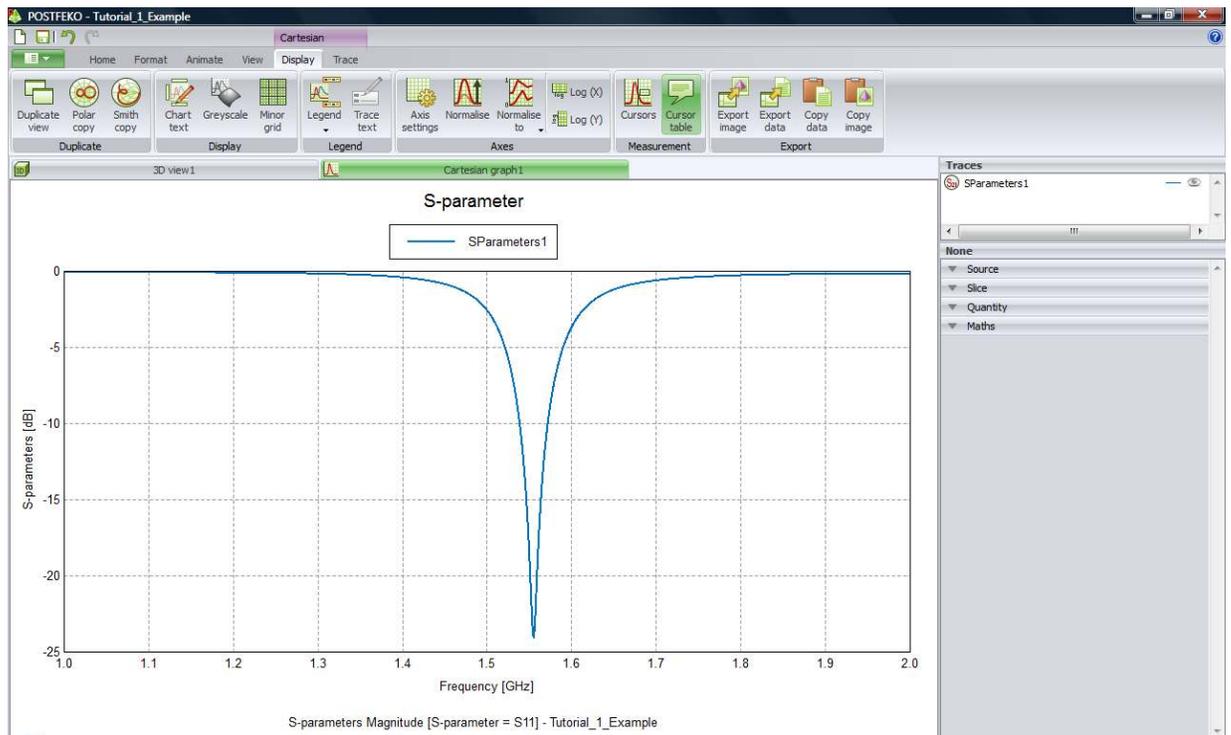


Figure 21.

30. Follow the procedure given to you in Tutorial 0 to get the gain pattern of the antenna at the center and two corner frequencies that you obtained from step 27. Record the Maximum gain

values and their directions, and also, get snap shots of the 3D gain patterns for each of the 3 frequencies examined.

31. Obtain the polar plots of the Azimuth plane (gain vs. phi when theta=90°) and the Elevation plane for this antenna (gain vs. theta when phi=90° and phi=0°) at the resonance frequency.
32. Submit your results in a short report.

THE END.