Experiment No. 3
Simulations of Monopole Antennas using HFSS Software

<u>Introduction</u>:

A monopole antenna is a class of radio antenna consisting of a straight rod-shaped conductor, often mounted perpendicularly over some type of conductive surface, called a ground plane. The driving signal from the transmitter is applied, or for receiving antennas the output signal to the receiver is taken, between the lower end of the monopole and the ground plane. One side of the antenna feed line is attached to the lower end of the monopole, and the other side is attached to the ground plane, which is often the Earth. This contrasts with a dipole antenna which consists of two identical rod conductors, with the signal from the transmitter applied between the two halves of the antenna.

The monopole is a resonant antenna; the rod functions as an open resonator for radio waves, oscillating with standing waves of voltage and current along its length. Therefore, the length of the antenna is determined by the wavelength of the radio waves it is used with. The most common form is the quarter-wave monopole, in which the antenna is approximately one quarter of the wavelength of the radio waves.

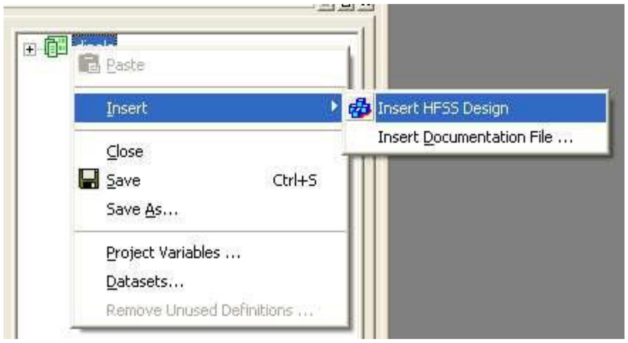
In this experiment, a monopole antenna will be constructed and analyzed using the HFSS simulation software by Ansoft. The following notes will provide a brief summary of goals.

✓ General navigation of software menus, toolbars, and quick keys.
✓ Variable assignment.
✓ Overview of commands used to create structures.
✓ Proper design and implementation of boundaries.
✓ Analysis Setup.
✓ Report Creation and options.

Creating the Project:

From the Project Manager Window Right-Click the project file and select Save As from the submenu. Name the file "monopole" and Click Save.

To begin working with geometries, you must insert an HFSS design. Right-Click the project file and select Insert > Insert HFSS Design from the menu.

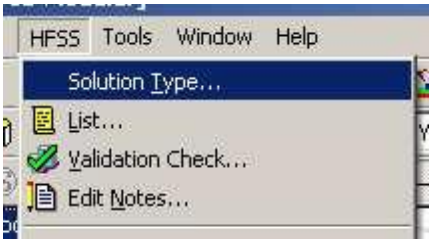


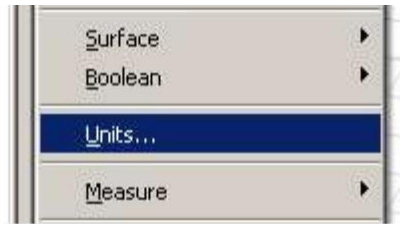
Note:

Always create a personal folder to store all HFSS projects. You may find that you do not have access rights to some portions of the hard drive. This will also allow the user to quickly backup/ copy data from projects.

Variable Definition:

Due to the nature of this design we will use DrivenModal as the solution type. From the HFSS menu select Solution Type and Driven Modal. The units are chosen as *meter* by choosing the heading 3D modeler and Units from the menu.

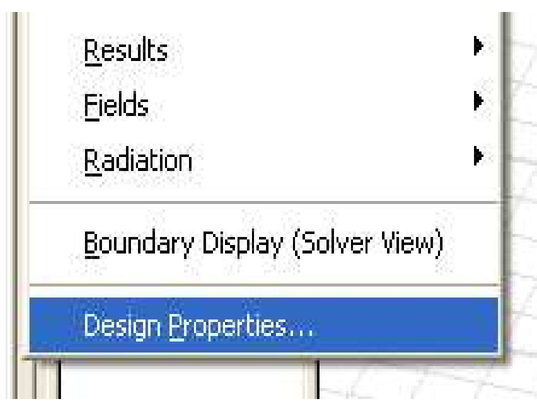




HFSS relies on variables for any parameterization/ optimization within the project. Variables also hold many other benefits which will make them necessary for all projects.

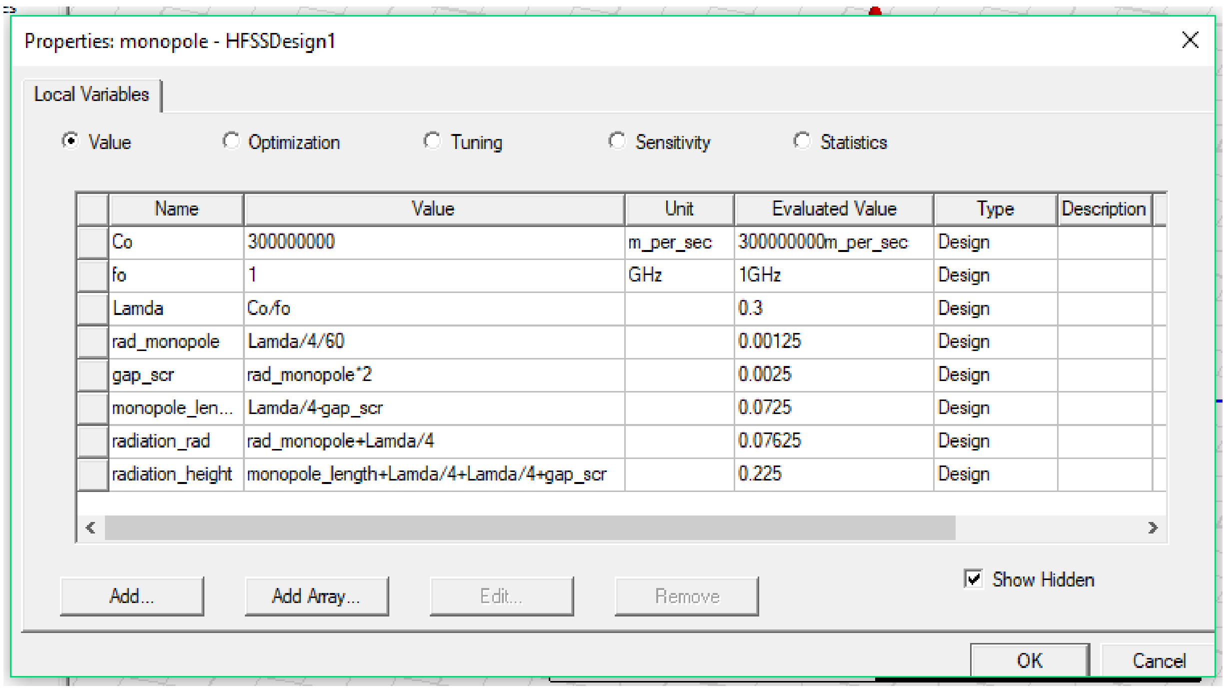
- √ Fixed Ratios (length, width, height) are easily maintained using variables.
- √ Optimetrics use variables to optimize the design according to user-defined criteria.
- √ All dimensions can be quickly changed in one window as opposed to altering each object individually.

Click the HFSS heading and select Design Properties at the bottom of the menu.



Variable Definition:

This will open the variable table. Add all variables shown below by selecting Add. Be sure to include units as needed.



Name	Value	Unit	Evaluated Value	Type	Description
Co	300000000	m_per_sec	300000000m_per_sec	Design	
fo	1	GHz	1GHz	Design	
Lamda	Co/fo		0.3	Design	
rad_monopole	Lamda/4/60		0.00125	Design	
gap_scr	rad_monopole*2		0.0025	Design	
monopole_len...	Lamda/4-gap_scr		0.0725	Design	
radiation_rad	rad_monopole+Lamda/4		0.07625	Design	
radiation_height	monopole_length+Lamda/4+Lamda/4+gap_scr		0.225	Design	

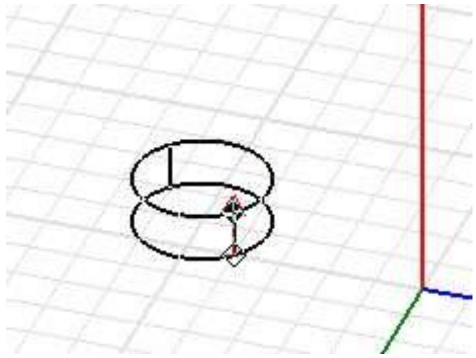
Note:

Creating variables before defining the structure will allow the user to build the geometry much faster than using a fixed system.

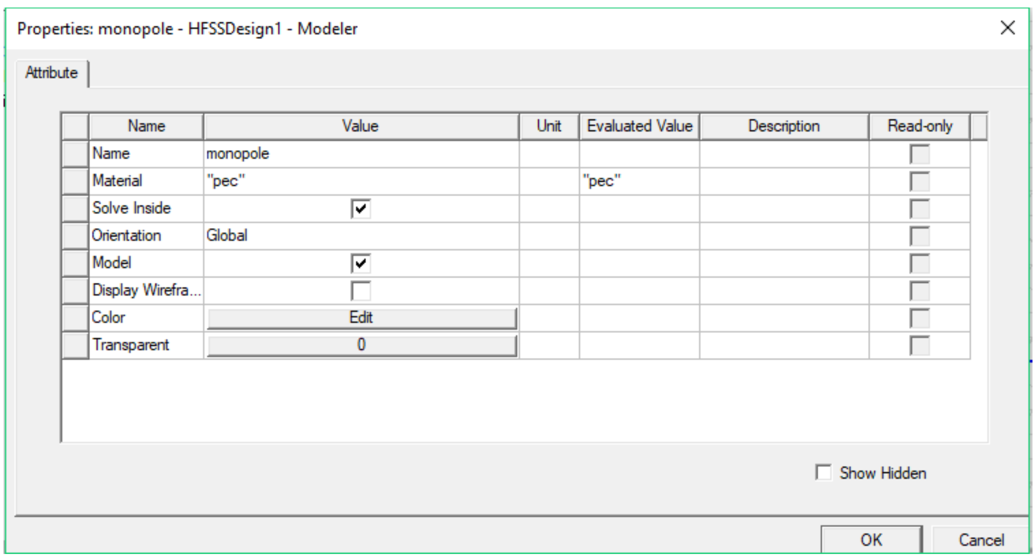
Creating the Model:

The 3D Modeler toolbars play a vital role in the creation of geometric structures within HFSS. By default, the 3D modeler toolbars should be visible on the screen. If you cannot locate the toolbar, then right-click on the upper border of the form and select them from the dropdown menu.

We will start by creating the monopole element using the Draw Cylinder button from the toolbar. Choose 3 Arbitrary points inside the drawing area. These points will be defined using the variables created previously so there is no need to specify points.



Follow the format below for structure size. Give the name *monopole* to this object. Assign the material *PEC* and click OK. *PEC* (Perfect Electric Conductor) will create ideal conditions for the element.



Properties: monopole - HFSSDesign1 - Modeler

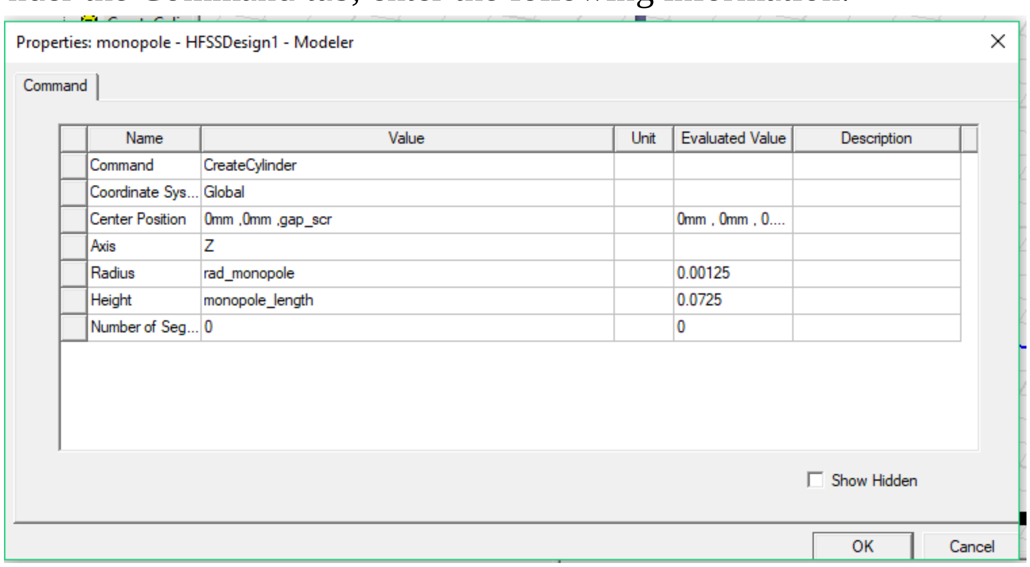
Attribute

Name	Value	Unit	Evaluated Value	Description	Read-only
Name	monopole				☐
Material	"pec"		"pec"		☐
Solve Inside	☑				☐
Orientation	Global				☐
Model	☑				☐
Display Wirefra...	☐				☐
Color	Edit				☐
Transparent	0				☐

☐ Show Hidden

OK Cancel

Under the Command tab, enter the following information:



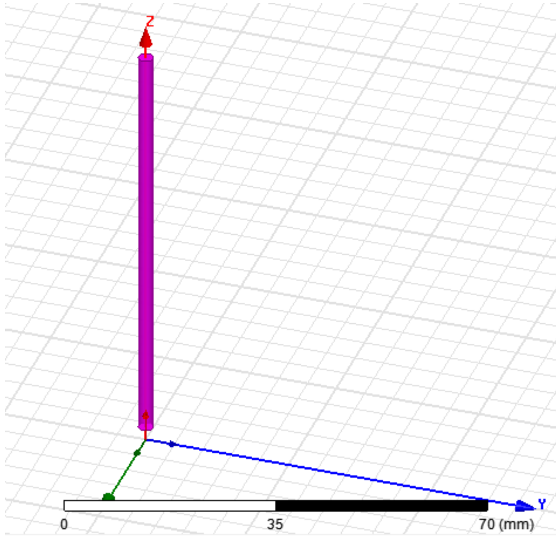
Properties: monopole - HFSSDesign1 - Modeler

Command

Name	Value	Unit	Evaluated Value	Description
Command	CreateCylinder			
Coordinate Sys...	Global			
Center Position	0mm ,0mm ,gap_scr		0mm , 0mm , 0....	
Axis	Z			
Radius	rad_monopole		0.00125	
Height	monopole_length		0.0725	
Number of Seg...	0		0	

☐ Show Hidden

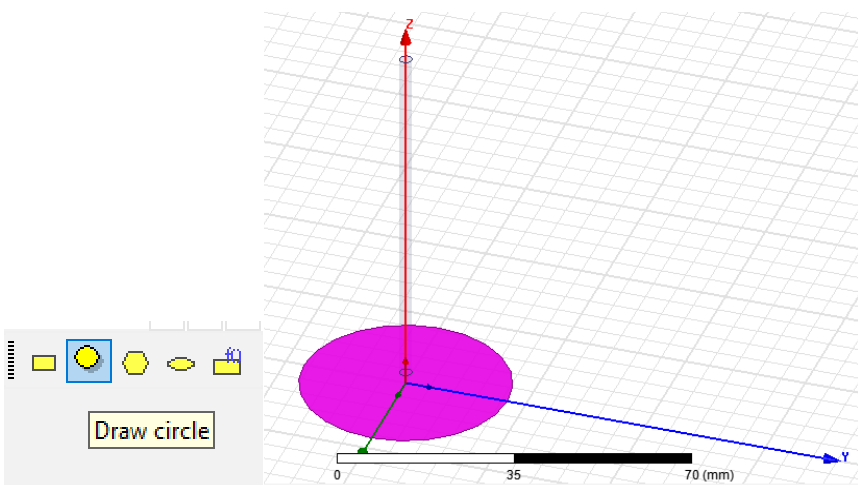
OK Cancel



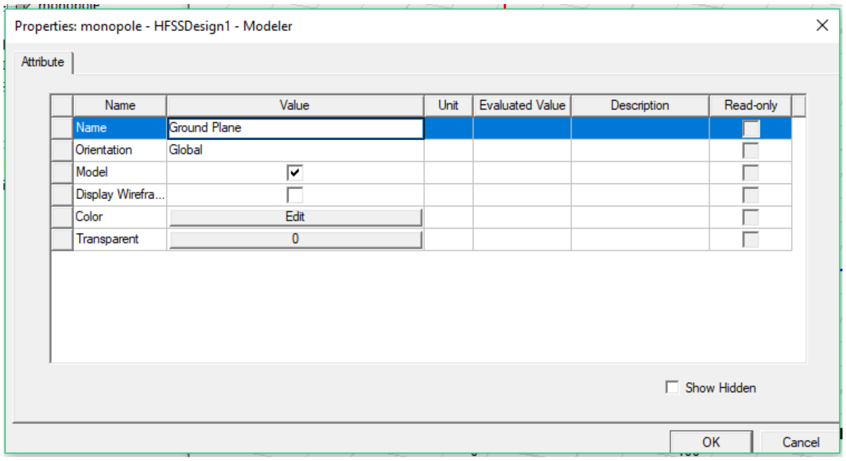
Ideally, the structure is one solid geometry. A slot has been created at the origin in this example. This will allow later placement of a source for excitation.

Creating the ground plane:

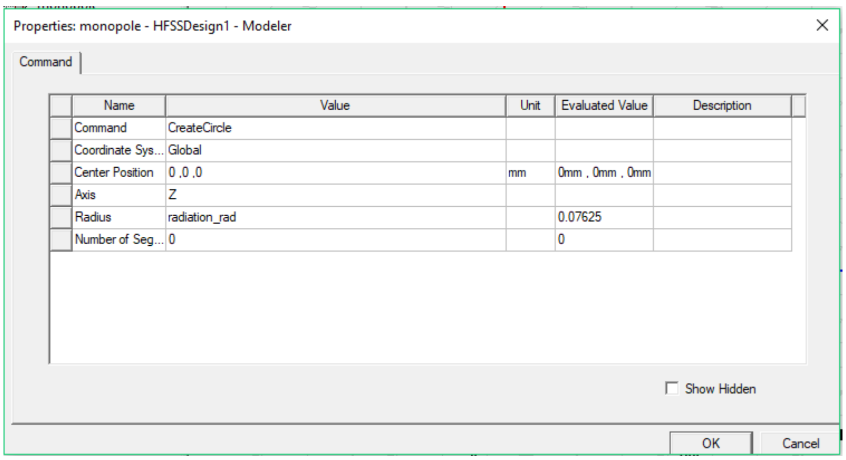
In this section the user will create a Ground Plane. This will provide a ground plane to the monopole structure. Using the 3D toolbar, click Draw Circle and place arbitrary point and arbitrary radius within the model area.



Enter the following:

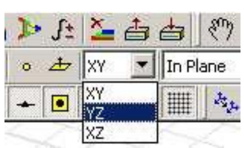


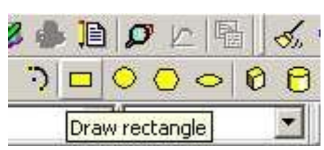
Click the Command tab and enter the following:



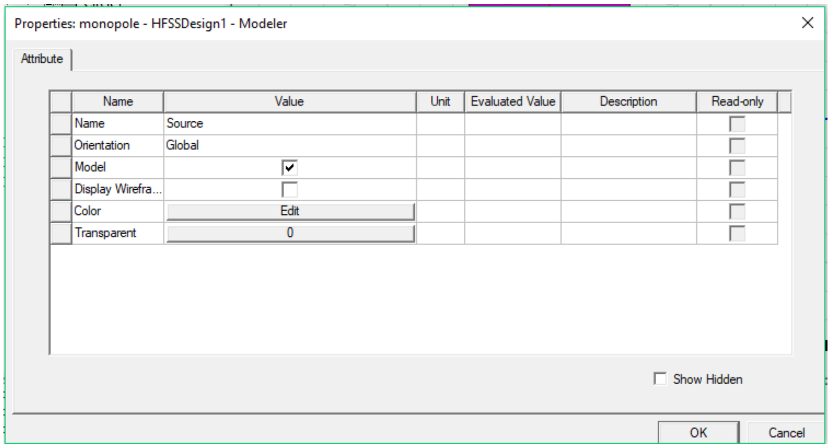
Creating the Port:

In this section the user will create a Lumped Gap Source. This will provide an excitation to the monopole structure. Begin by selecting the YZ plane from the toolbar. Using the 3D toolbar, click Draw Rectangle and place two arbitrary points within the model area.

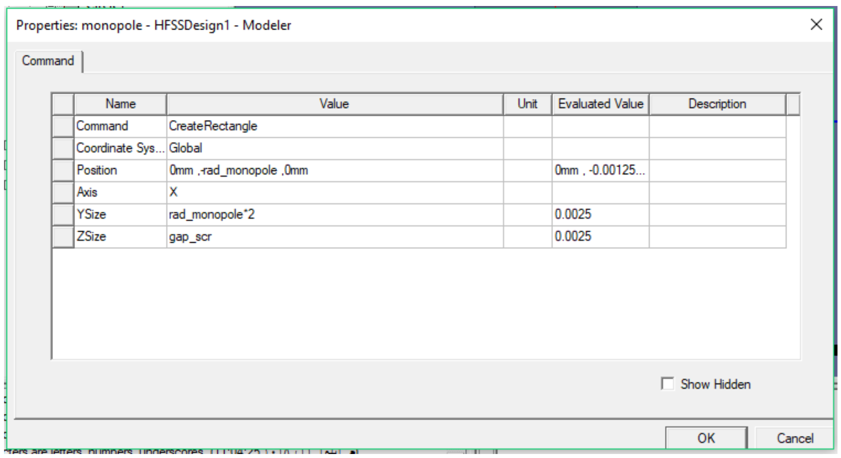




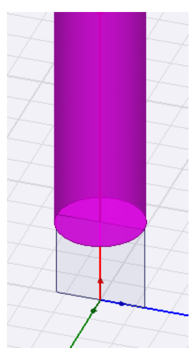
Enter the following:



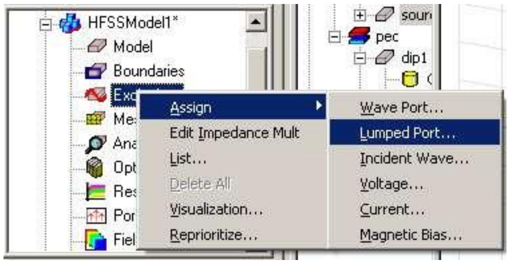
Click the Command tab and enter the following:



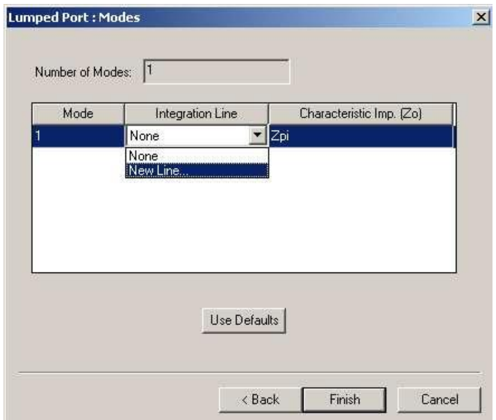
Please note that the variable *gap_src* was chosen relatively small in comparison to the monopole structure. This was done to minimize effects due to the source and place emphasis upon the structure. The source is depicted below.



With the source geometry in place, the user must provide an excitation. A lumped port will be used for the monopole model. This excitation is commonly used when the far field region is of primary interest. In the project explorer, right-click Excitation>Assign> Lumped Port.



Name the port *source* and leave the default values for impedance. Click Next and enter the following:

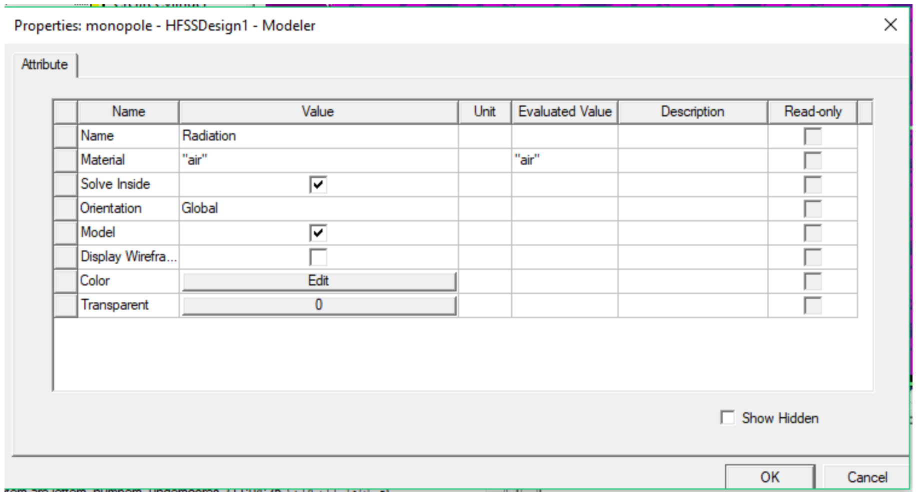


Using the mouse, position the cursor to the bottom-center of the port. Ansoft's snap feature should place the pointer when the user approaches the center of any object. Left-click to define the origin of the E-field vector. Move the cursor to the top-center of the port. Left-click to terminate the E-field vector. Click finish to complete the port excitation.

Radiation Boundary:

In this section, a radiation boundary is created so that far field information may be extracted from the structure. To obtain the best result, a cylindrical air boundary is defined with a distance of λ/4. From the toolbar, select Draw Cylinder and choose 3 arbitrary points within the model window.

Enter the following:



Properties: monopole - HFSSDesign1 - Modeler

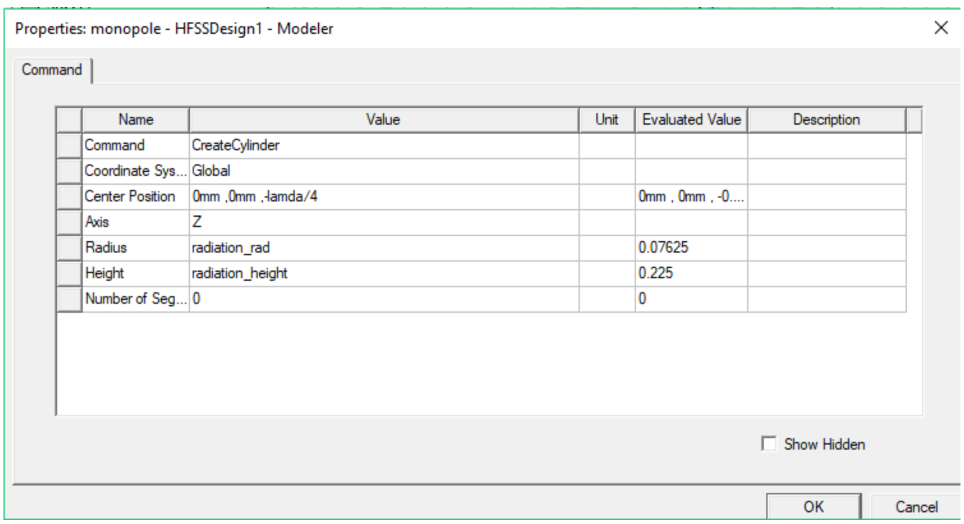
Attribute

Name	Value	Unit	Evaluated Value	Description	Read-only
Name	Radiation				☐
Material	"air"		"air"		☐
Solve Inside	☑				☐
Orientation	Global				☐
Model	☑				☐
Display Wirefra...	☐				☐
Color	Edit				☐
Transparent	0				☐

☐ Show Hidden

OK Cancel

Click the *Command* tab and enter the following:



Properties: monopole - HFSSDesign1 - Modeler

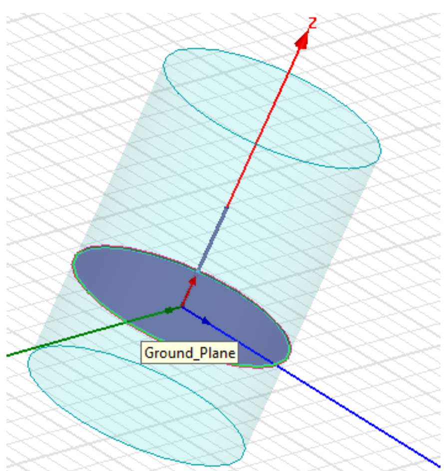
Command

Name	Value	Unit	Evaluated Value	Description
Command	CreateCylinder			
Coordinate Sys...	Global			
Center Position	0mm ,0mm ,-lamda/4		0mm , 0mm , -0....	
Axis	Z			
Radius	radiation_rad		0.07625	
Height	radiation_height		0.225	
Number of Seg...	0		0	

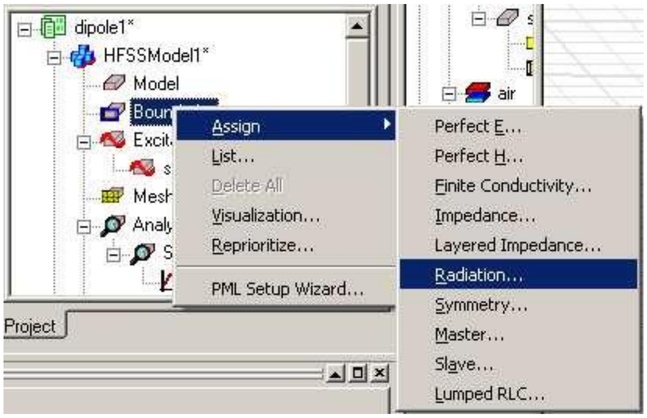
☐ Show Hidden

OK Cancel

Assuming all steps were properly completed, the boundary should resemble the illustration below:



With the geometry complete the actual radiation boundary may now be assigned. Right-click the Boundary icon in the object explorer and select Boundary > Assign> Radiation.

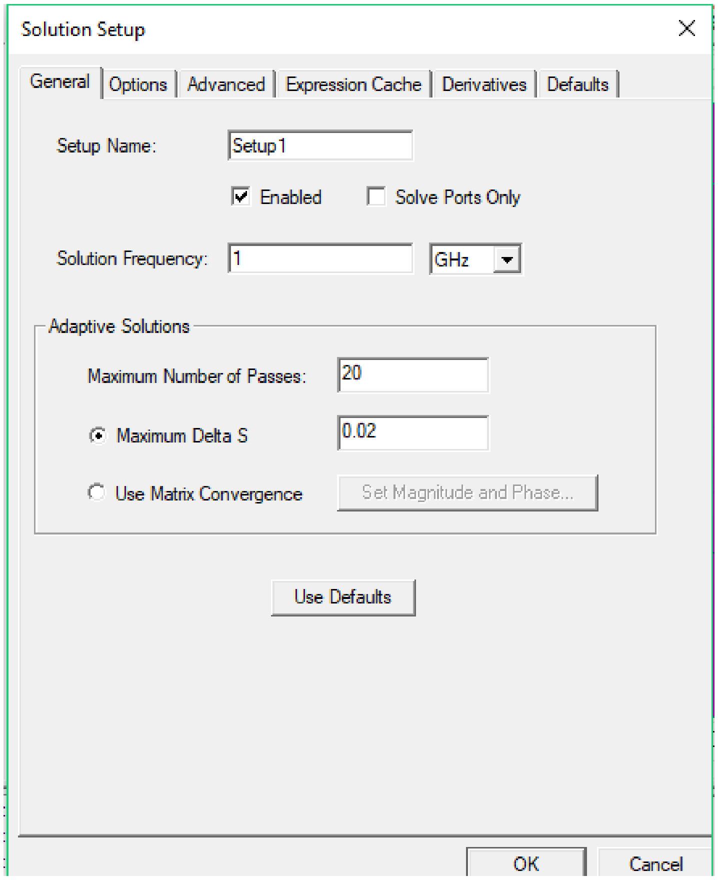


Leave the default name Rad1 and click OK.

Solution Setup:

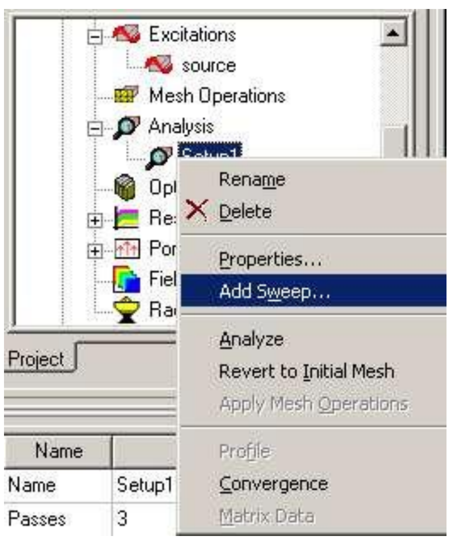
In this section a solution must be defined to display the desired data. We are primarily interested in the frequency response of the structure. We will also explore HFSS ability to calculate general antenna parameters such as directivity, radiation resistance, radiation efficiency, etc. From the project explorer, select Analysis >Add Solution Setup.

Enter the following:

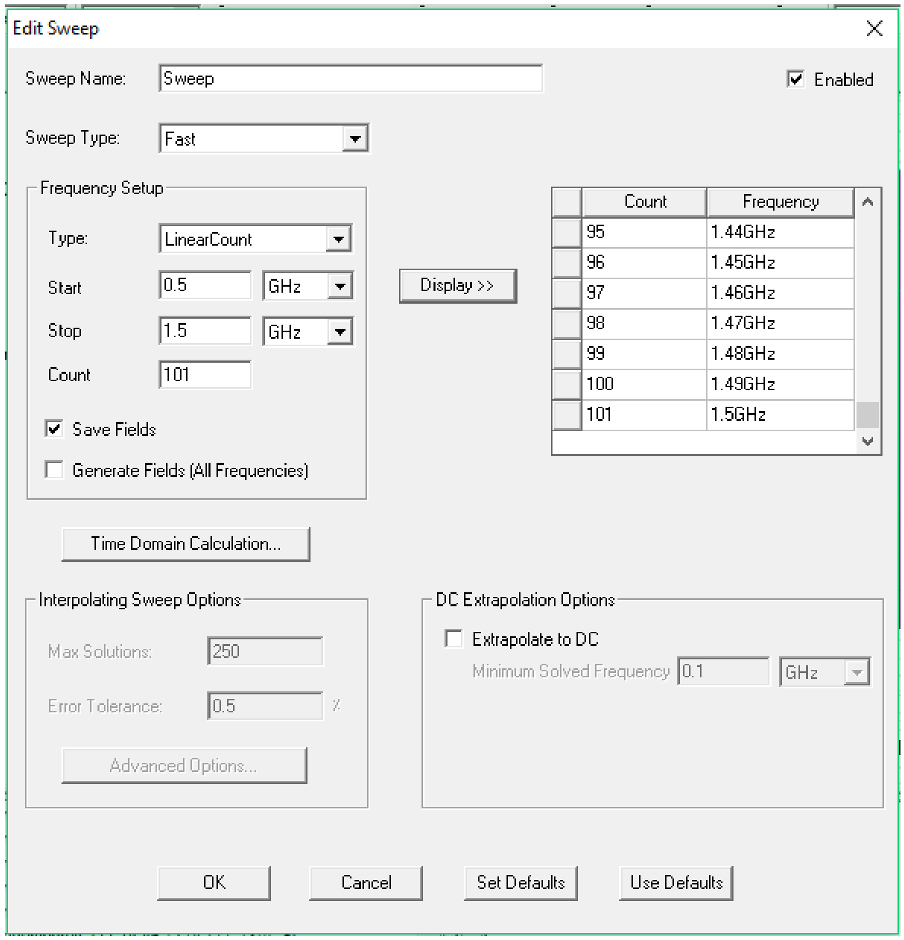


Leave all other settings as default. Click OK when complete.

To view the frequency response of the structure, a frequency sweep must be defined. From the project explorer select Setup1 > Add Sweep.

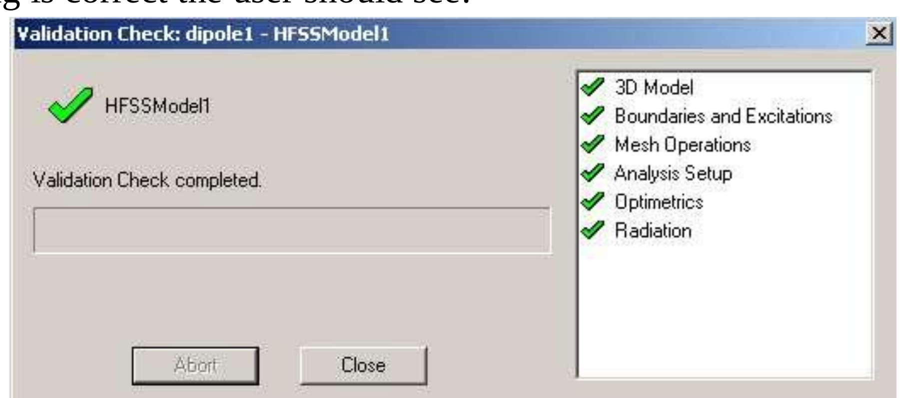


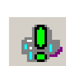
Enter the following:



Structure Analysis:

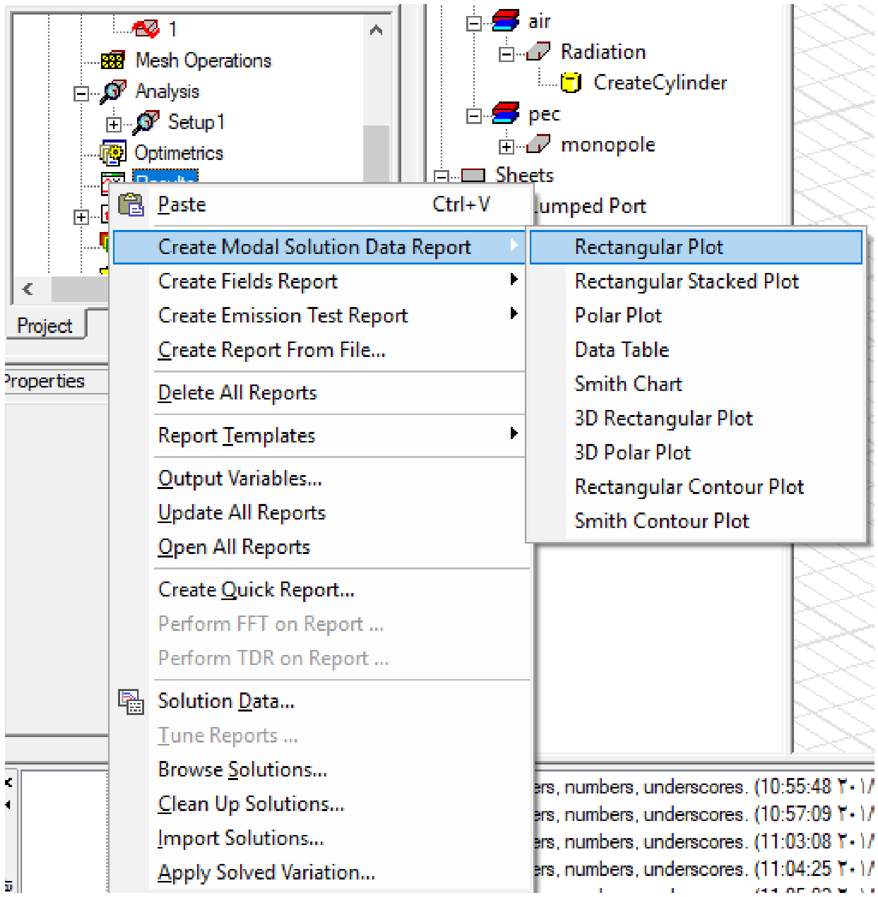
At this point, the user should be ready to analyze the structure. Before running the analysis, always verify the project by selecting  from the 3D toolbar. If everything is correct the user should see:



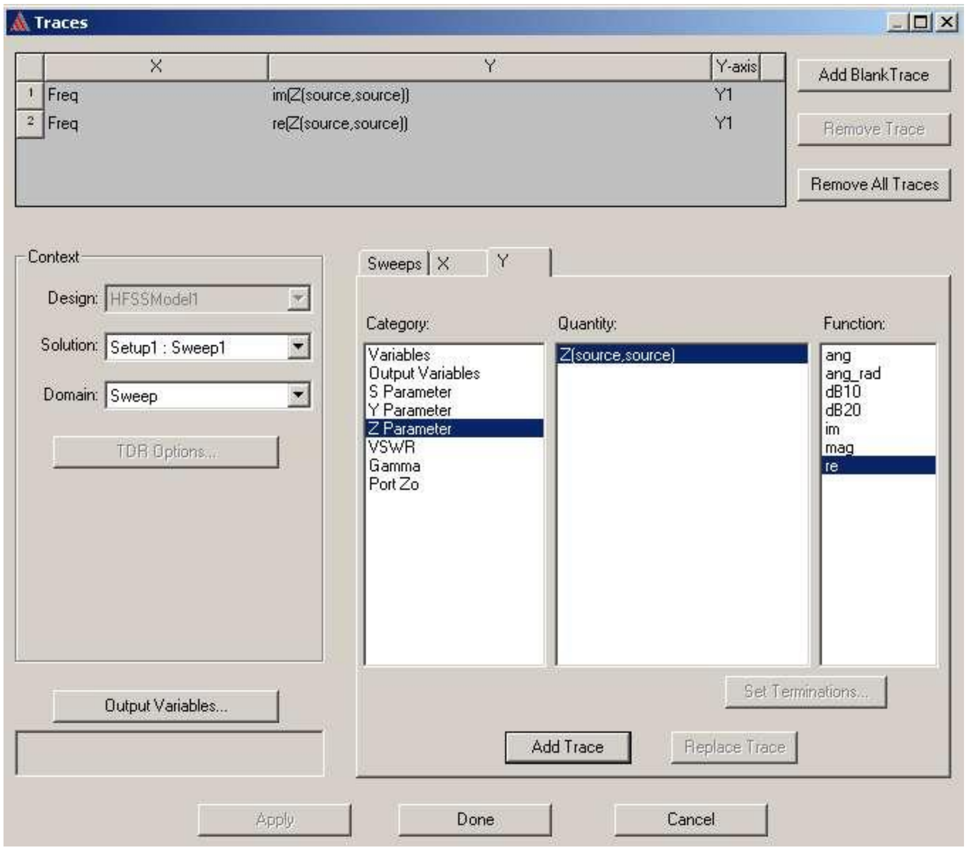
Analyze the structure by clicking , allow 5-20 minutes for the analysis, depending on the machine.

Create Reports:

After completion of the analysis, we will create a report to display both the reso nt frequency and also the radiation pattern. Click on the heading HFSS and select Resu > Create Reports.

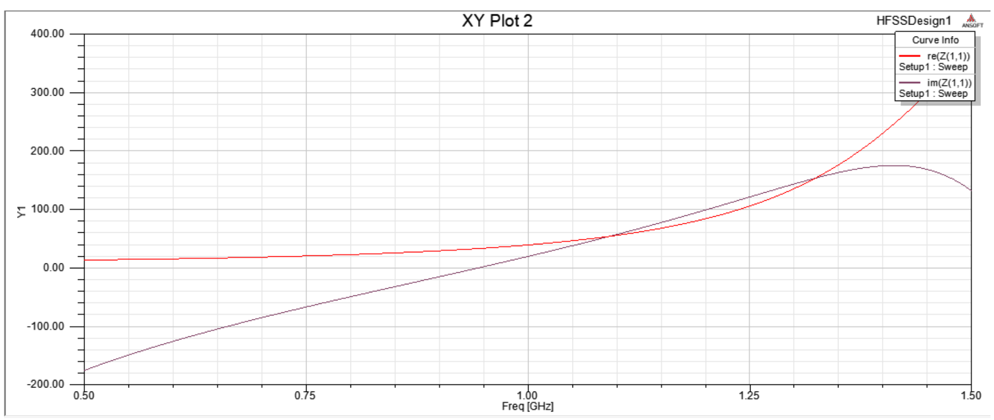


Select the following highlighted parameters and click Add Trace to load the opt ns into the Trace window.

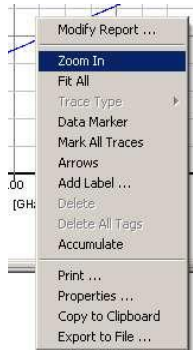


Click Done to display a graph of impedance vs. frequency.

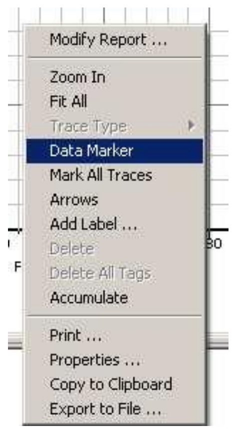
Looking at the graph below, both real and imaginary components of the impedance are displayed.



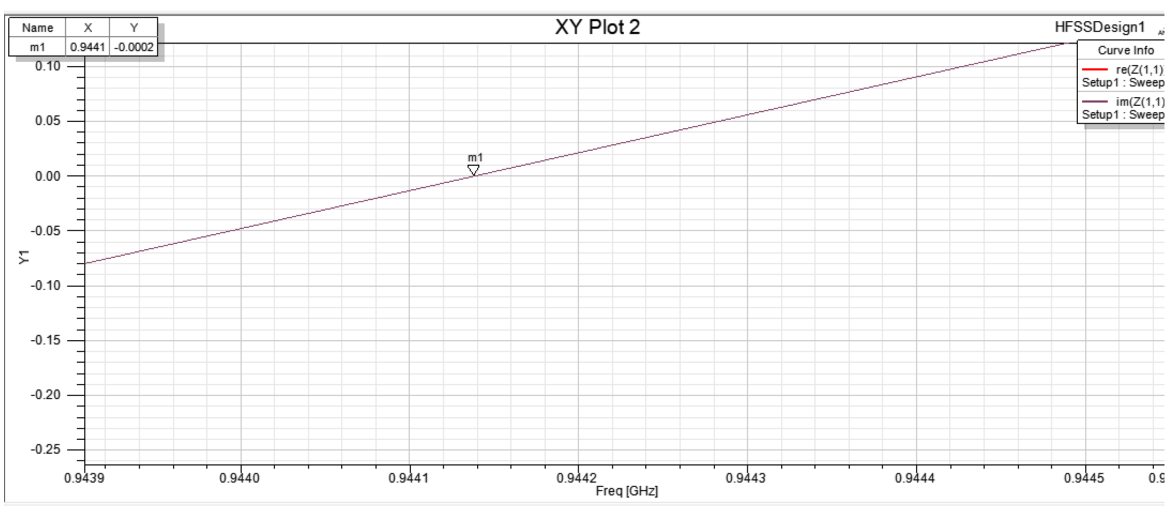
The input resistance can be directly determined from the graph. We will mark the point at which imaginary component crosses zero. This mark will allow the user to determine input impedance at the point of resonance. Right-Click the graph and select Zoom In.



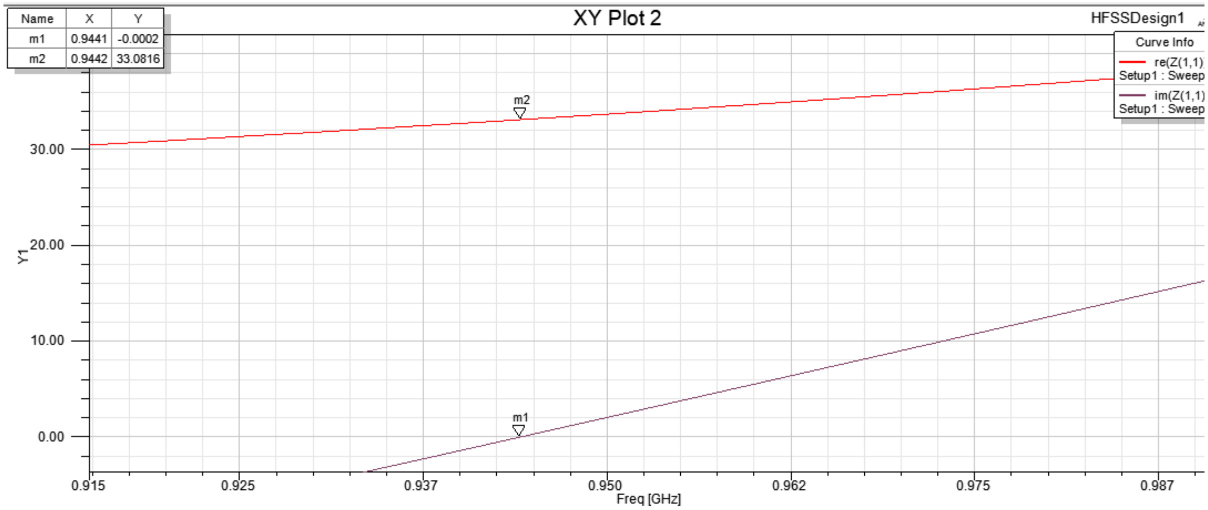
Using the mouse select a zoom window around the imaginary component as it crosses zero. Mark the zero point by right-clicking the plot window and selecting Data Marker



Select a point as close as possible to zero along the imaginary line. You will not be able to choose exactly zero due to the resolution chosen (101 points) in the solution setup. Left-click to mark the point as shown below:

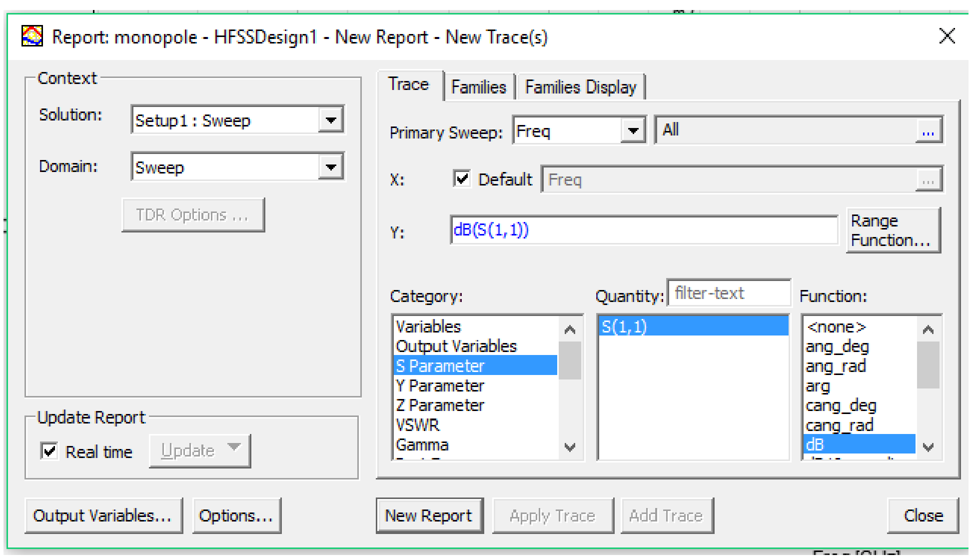


Right-click the plot window and select Fit All. Follow the same procedure to mark the real component at exactly the same frequency of the imaginary component. Both marked data can be seen in the graph below:

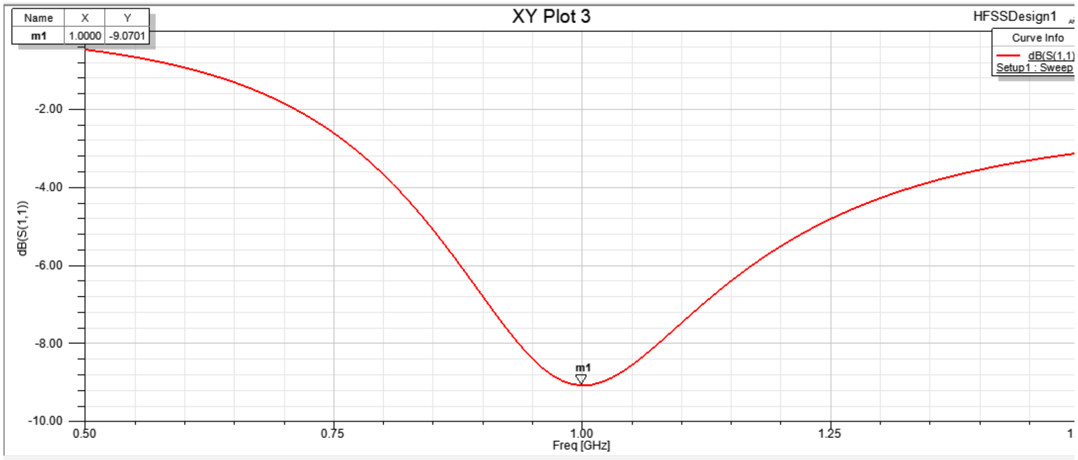


The input resistance of the antenna is 33.0816 ohms according to the gr h. Performing calculations from a text, the user should compute a resistance between 65 d 75 ohms. The port was previously defined with an impedance of 75 ohms. This ll produce sub-optimal results due to mismatched impedance. This will be corrected shc y.

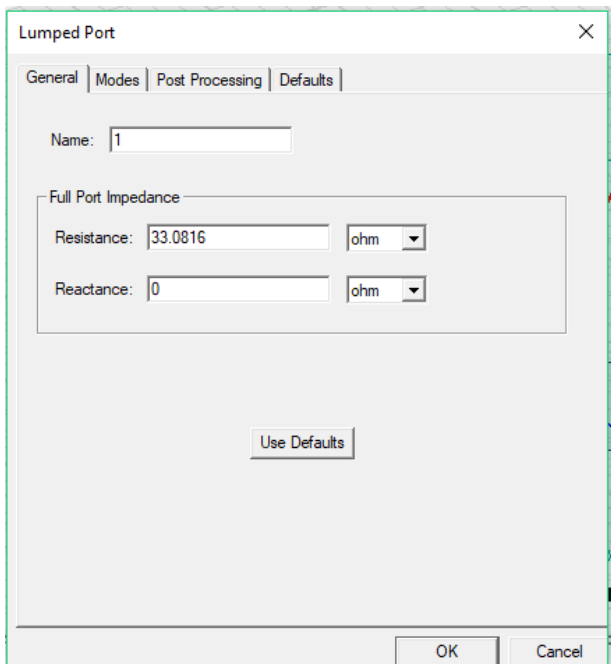
In the next step, we will plot S_{11}vs. frequency. Create a Report as previously sh n and add the following trace:



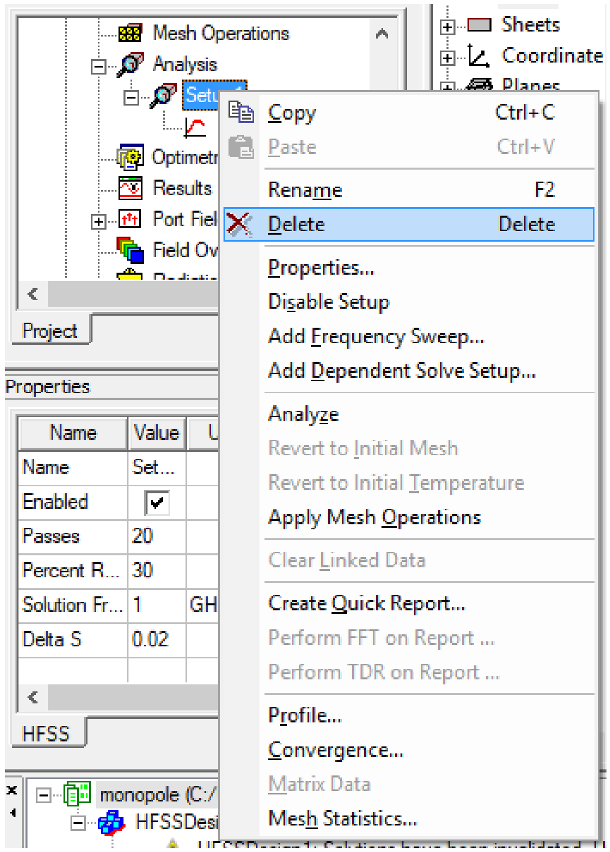
Click Done when complete. Then the graph is displayed below:



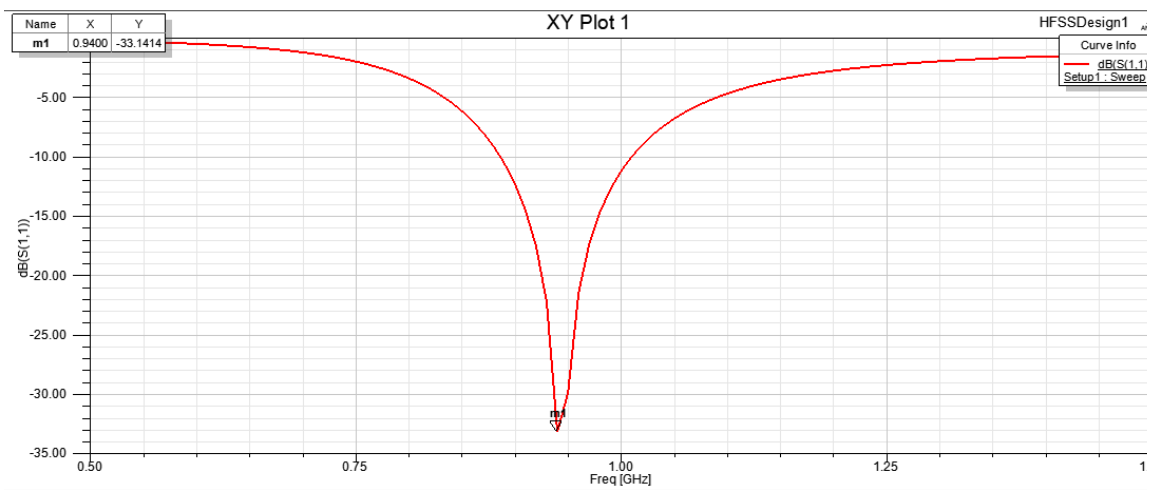
The point of resonance was marked at -9.0701dB. In order to compute accu e antenna parameters, the input must be matched. From the project explorer, right-c k *source* and select Properties. Adjust the port impedance as shown:



We will now re-analyze the structure with a properly matched port. In orde o preserve memory and calculation time, right-click Analysis > Delete in the pro ct explorer.



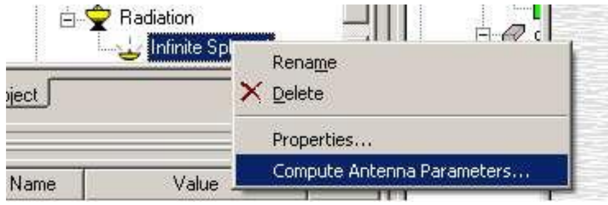
Re-analyze the structure. When complete create another plot of S_{11}vs. frequency as shown below:



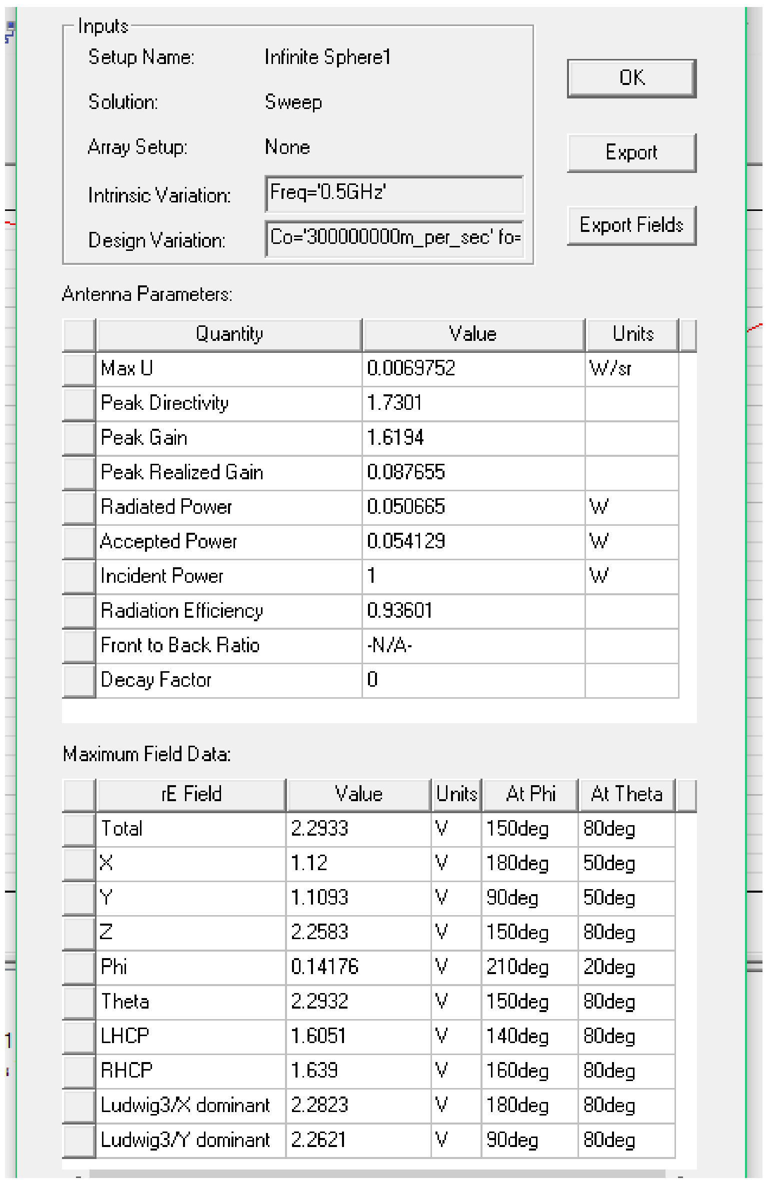
Note the improved responseof-33.1414 dB at resonance.

HFSS has the ability to compute antenna parameters automatically. In order to produce the calculations, the user must define an infinite sphere for far field calculations. Right-click the Radiation icon in the project manager window and select Insert Far Field Setup > Infinite Sphere.

Accept all default parameters and click Done. Right-click Infinite Sphere1 > Com e Antenna Parameters... from the project explorer as shown:



Select all defaults and results are displayed as follows:



Inputs

Setup Name: Infinite Sphere1

Solution: Sweep

Array Setup: None

Intrinsic Variation: Freq='0.5GHz'

Design Variation: Co='300000000m_per_sec' fo=

OK

Export

Export Fields

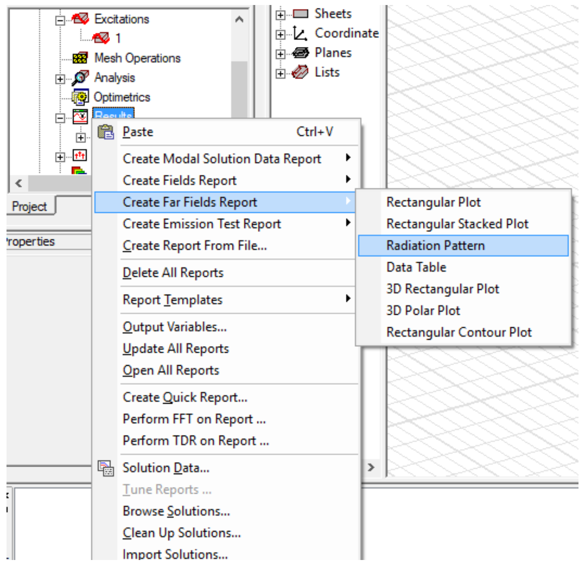
Antenna Parameters:

Quantity	Value	Units
Max U	0.0069752	W/sr
Peak Directivity	1.7301	
Peak Gain	1.6194	
Peak Realized Gain	0.087655	
Radiated Power	0.050665	W
Accepted Power	0.054129	W
Incident Power	1	W
Radiation Efficiency	0.93601	
Front to Back Ratio	-N/A-	
Decay Factor	0	

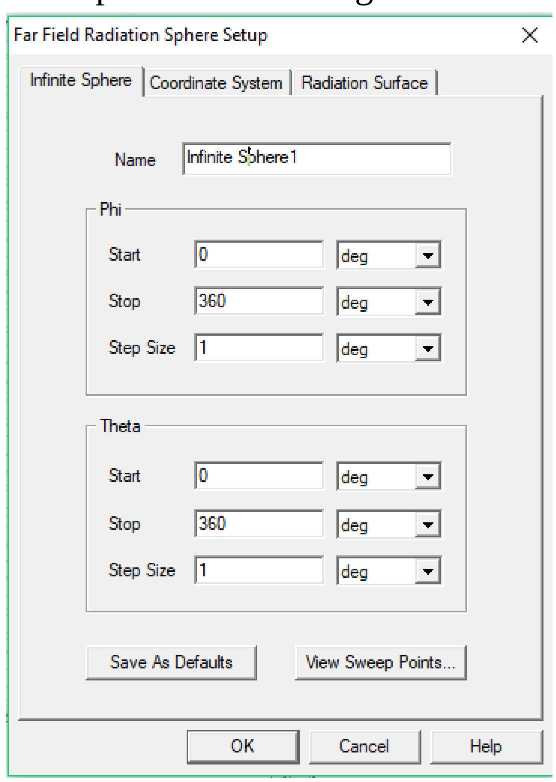
Maximum Field Data:

rE Field	Value	Units	At Phi	At Theta
Total	2.2933	V	150deg	80deg
X	1.12	V	180deg	50deg
Y	1.1093	V	90deg	50deg
Z	2.2583	V	150deg	80deg
Phi	0.14176	V	210deg	20deg
Theta	2.2932	V	150deg	80deg
LHCP	1.6051	V	140deg	80deg
RHCP	1.639	V	160deg	80deg
Ludwig3/X dominant	2.2823	V	180deg	80deg
Ludwig3/Y dominant	2.2621	V	90deg	80deg

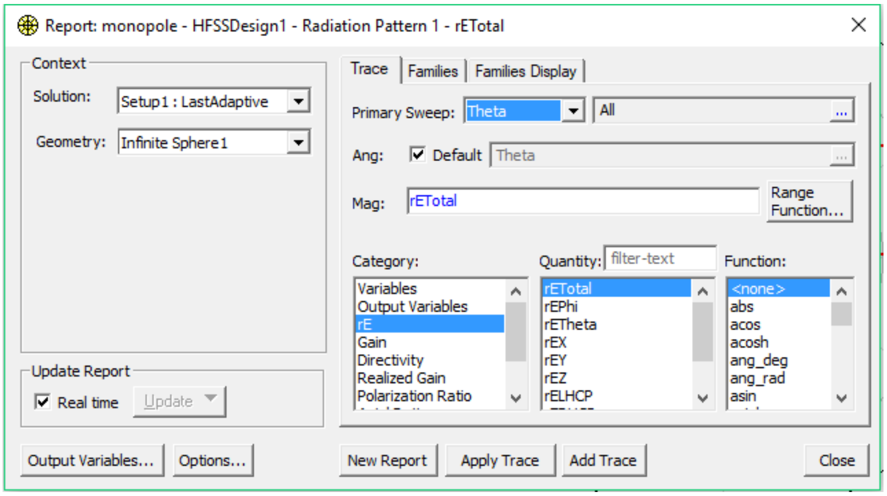
From the chart the Peak Directivity is 1.7301. All other parameters can be seen as slightly elevated above the expected. Adjustments to the radiation boundary might provide more accuracy. Next, the far field will be plotted. Create Reports as previously shown. Modify the following:



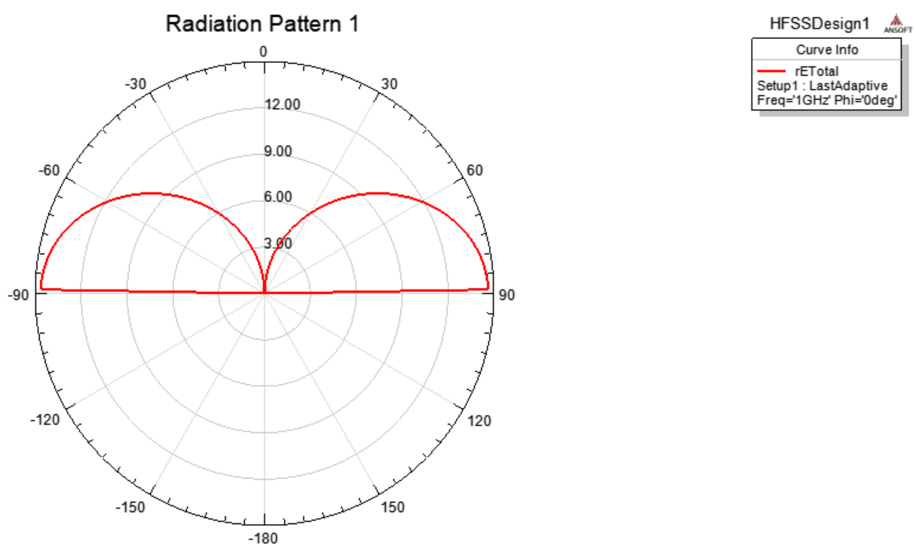
Changing the data of an infinite sphere as following



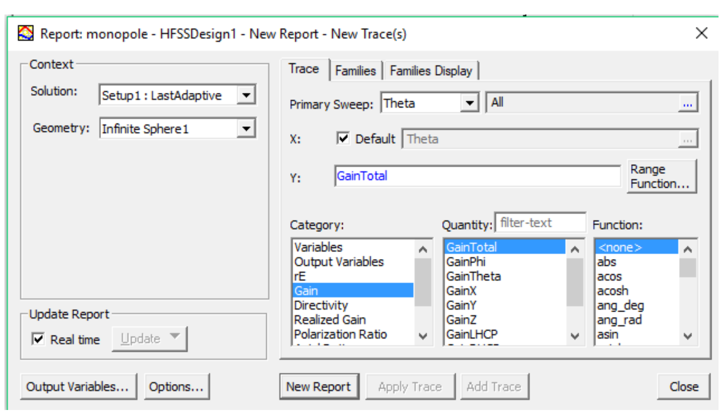
Enter the following:



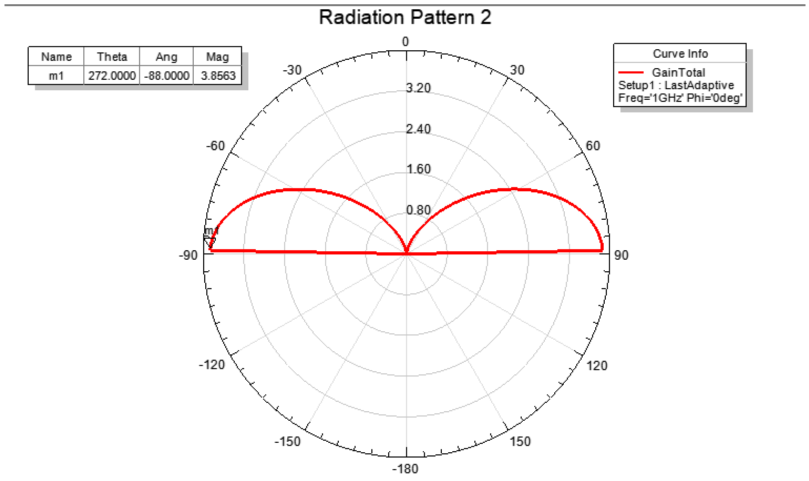
Select Add Trace and click Done when complete. The radiation pattern is displayed below:



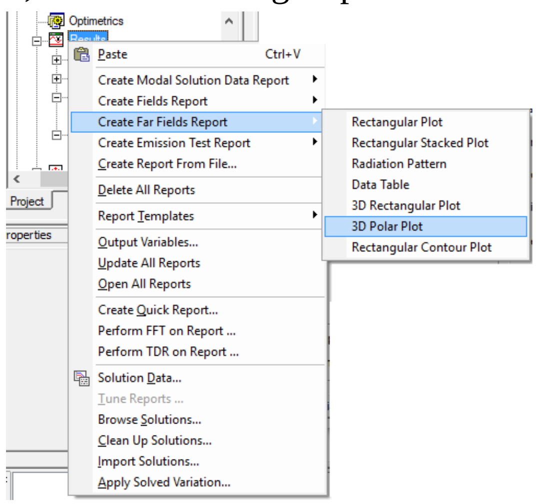
To find the total gain of antenna repeat the same previous steps and do the following changing

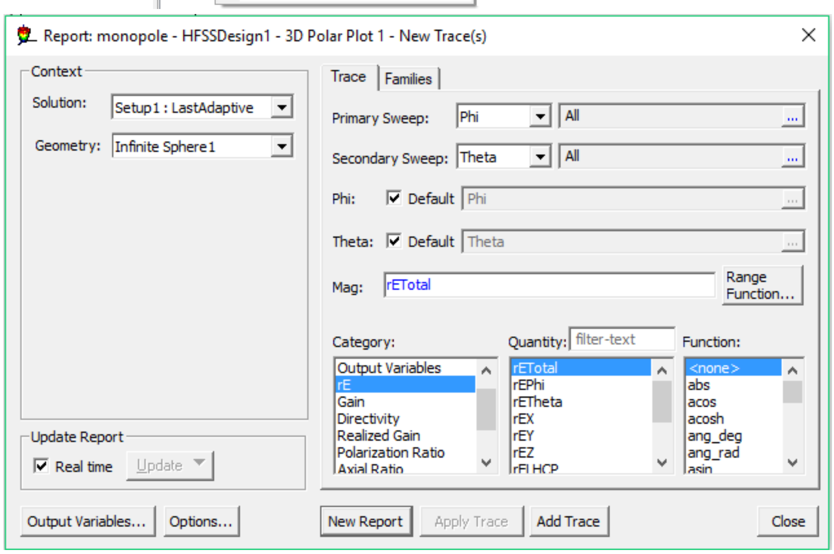


Choose the maximum point to determine the max gain and its position.

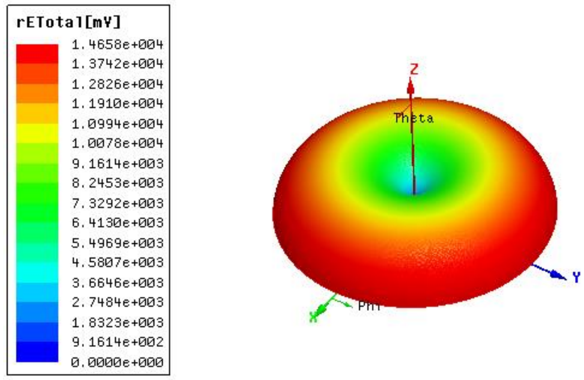


To draw 3D radiation pattern, do the following steps

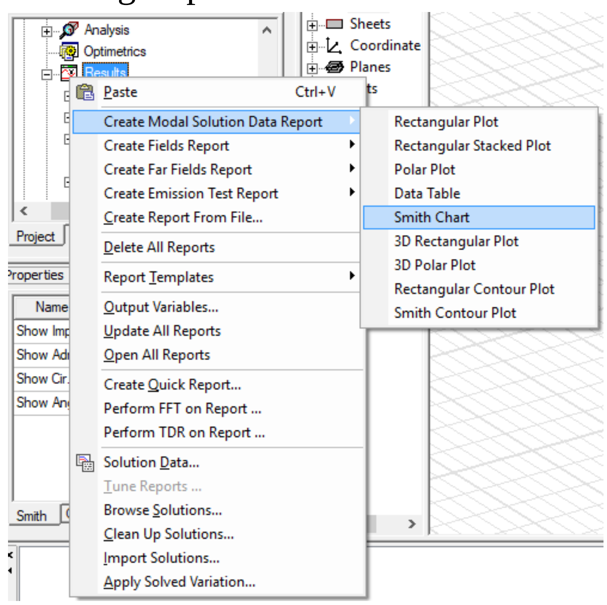


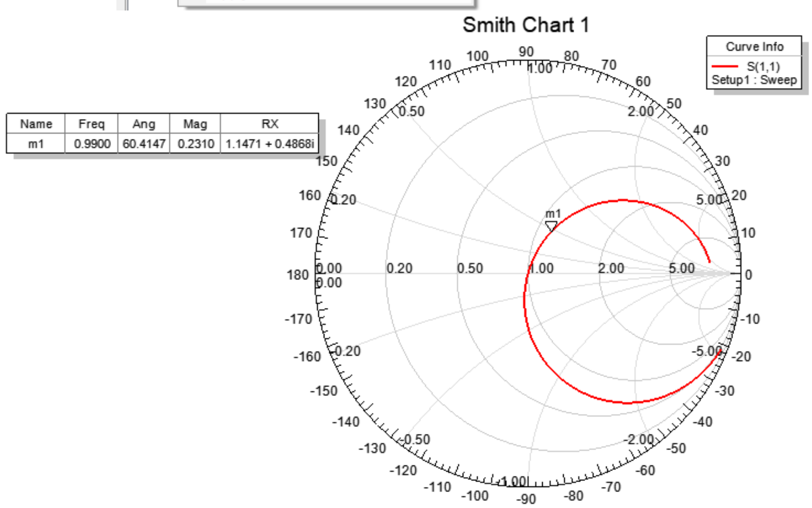


Select Add Trace and click Done when complete



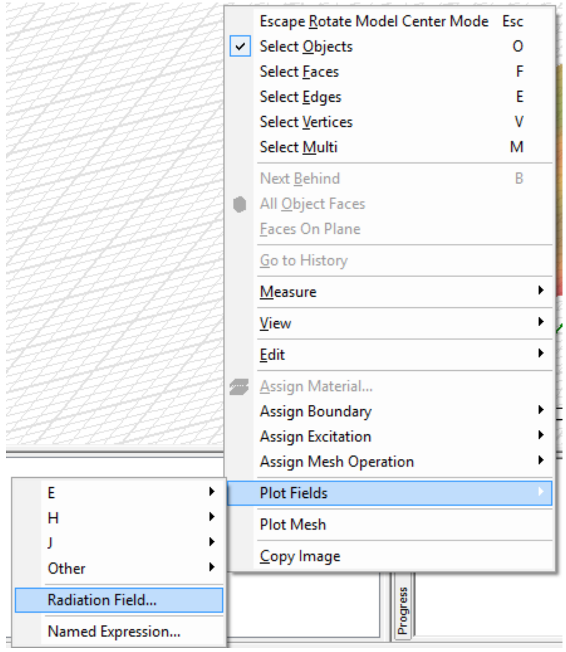
To draw smith chart the following steps must be followed



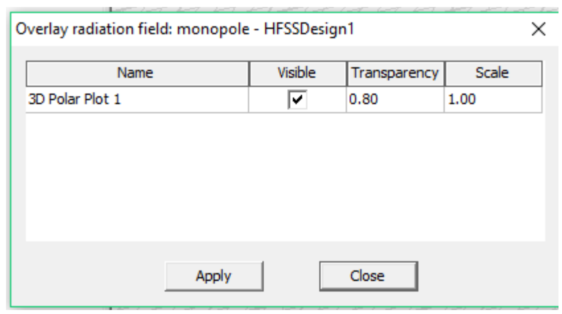


To show the field pattern with respect to the antenna structure:

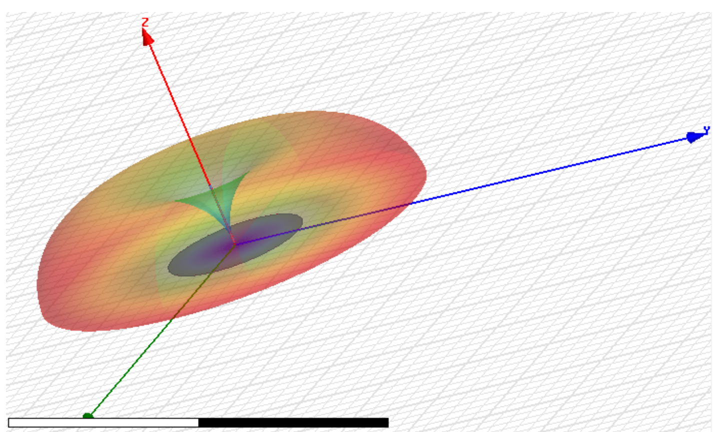
Firstly, right click and choice plot the fields as follows



Set the values of radiation field as shown below



Select Apply, the radiation field surrounding the antenna will appear as in figure below.



<u>Discussion:</u>

1- Why we use monopole antenna instead of dipole antenna.
2- Calculate directivity analytically.