Ansoft High Frequency Structure Simulator

Tutorial

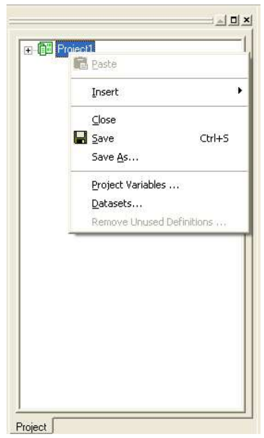
The Dipole Antenna

April 2004

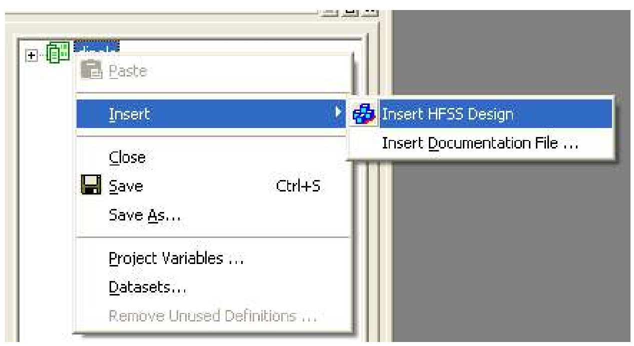
In this tutorial, a dipole antenna will be constructed and analyzed using the HFSS simulation software by Ansoft. The example will illustrate both the simplicity and power of HFSS through construction and simulation of this antenna structure. The following notes will provide a brief summary of goals.

- ✔ 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.

From the Project Manager window Right-Click the project file and select Save As from the sub menu. Name the file “dipole” and Click Save.



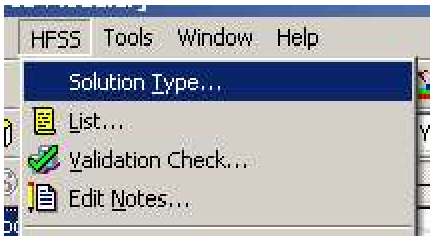
To begin working with geometries, you must insert an HFSS design. Right-Click the project file and select Insert -> Insert HFSS Design from the menu.

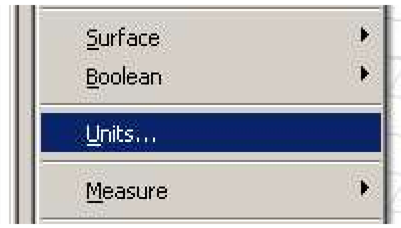


- Note:

  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.

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

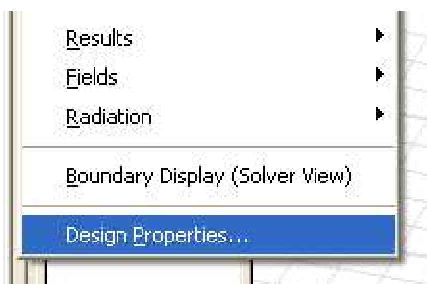




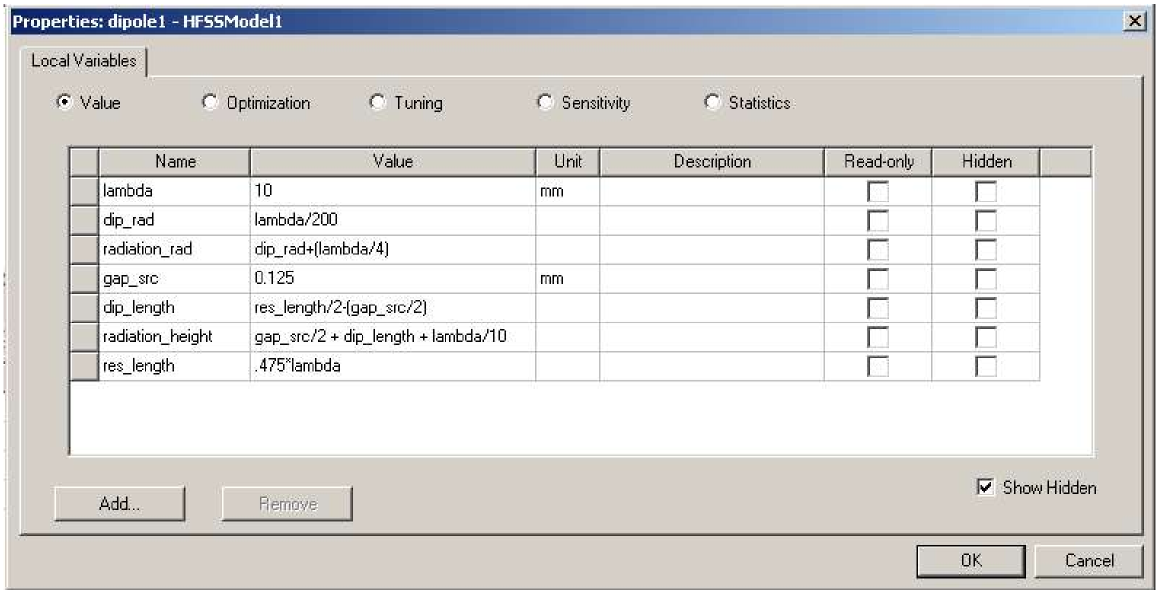
HFSS relies on variables for any parameterization / optimization within the project. Variables also hold many other benefits which will make them necessary for all projects.

- ✔ 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.



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

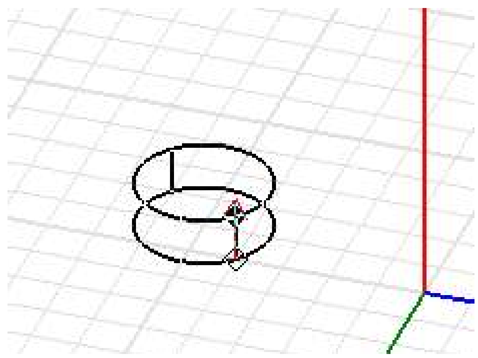


- **Note:**

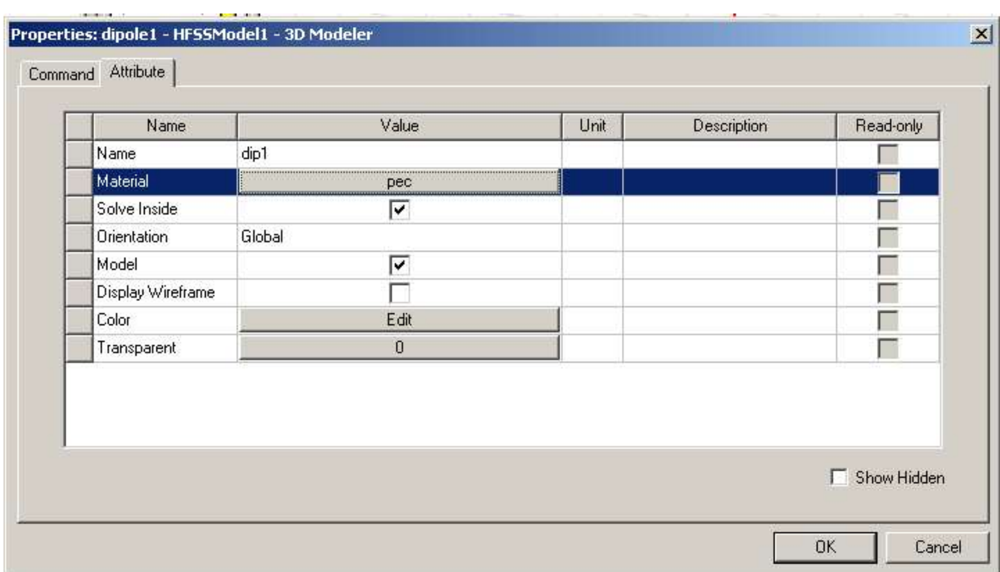
  Creating variables before defining the structure will allow the user to build the geometry much faster than using a fixed system.

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 drop down menu.

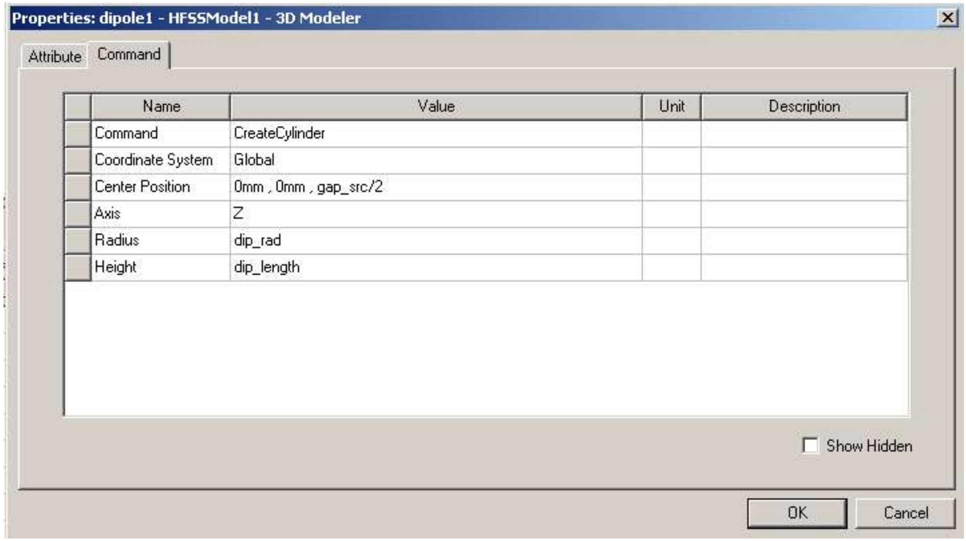
We will start by creating the dipole element using the Draw Cylinder button from the toolbar. Choose 3 Arbitrary points inside the drawing area. These points will be defined using the variables created previously so there is no need to specify points.



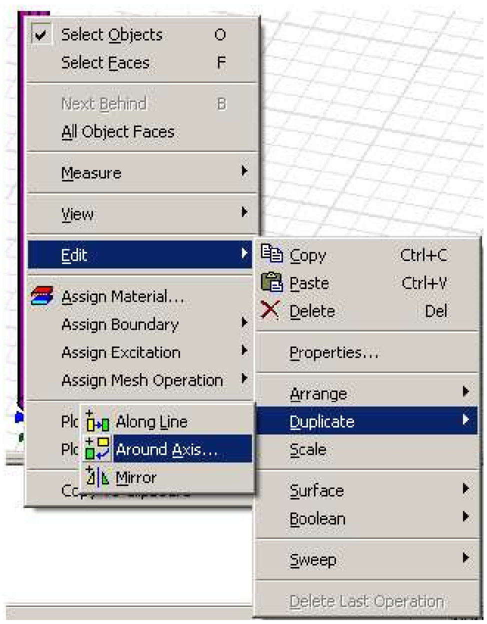
Follow the format below for structure size. Give the name *dip1* to this object. Assign the material *PEC* and click OK . *PEC* (Perfect Electric Conductor) will create ideal conditions for the element.



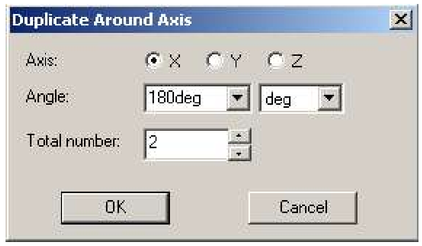
Under the Command tab, enter the following information:



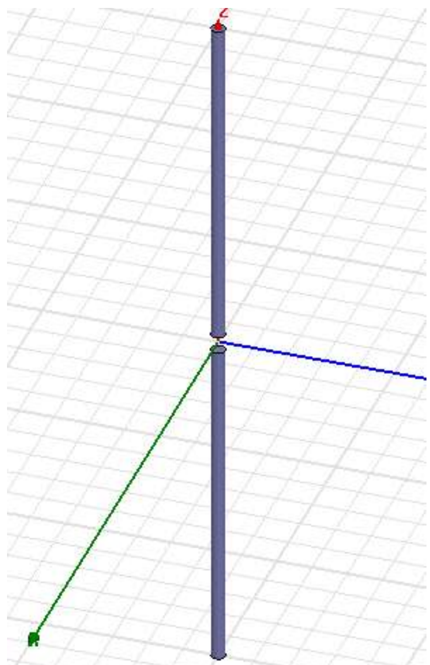
The next command is essential when building symmetric structures. Right-Click the drawing area and select Edit -> Duplicate -> Around Axis.



A mirror image is produced by enter the following:

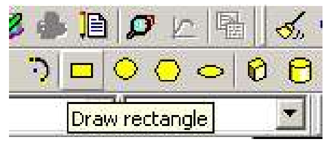


The dipole structure is illustrated below:

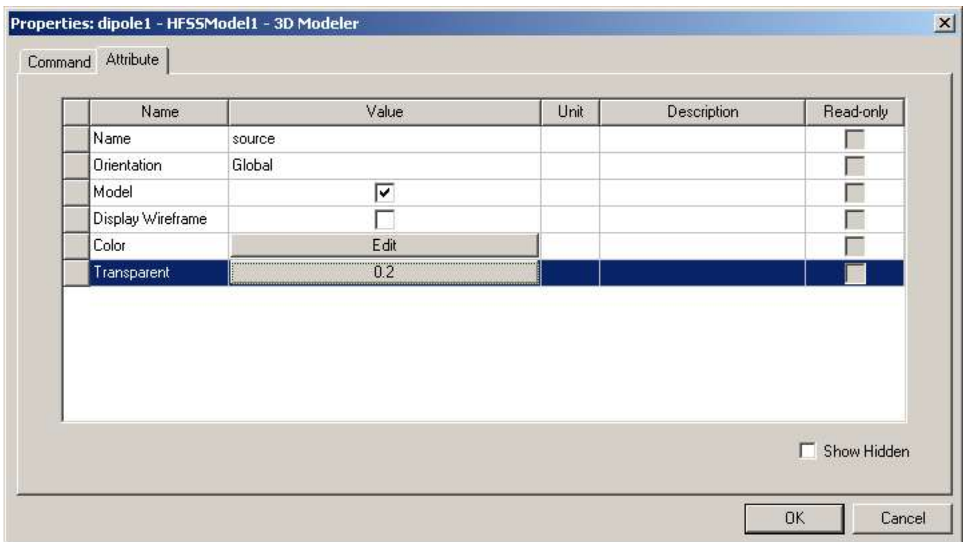


Ideally, the structure is one solid geometry. A slot has been created at the origin in this example. This will allow later placement of a source for excitation.

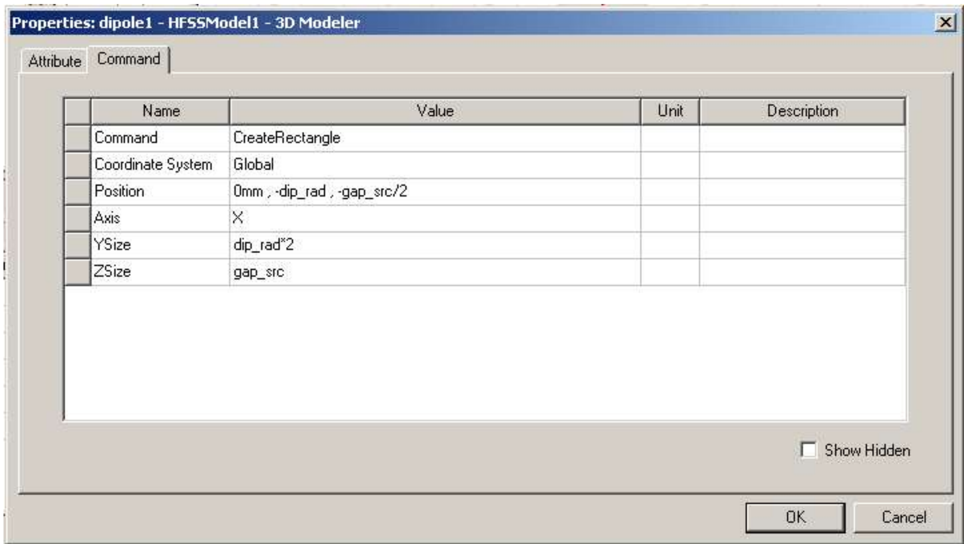
In the section the user will create a Lumped Gap Source. This will provide an excitation to the dipole structure. Begin by selecting the YZ plane from the toolbar. Using the 3D toolbar, click Draw Rectangle and place two arbitrary points within the model area.



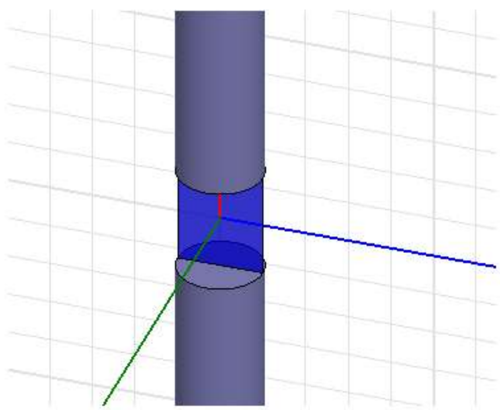
Enter the following:



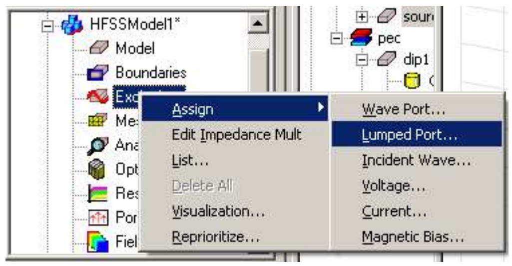
Click the Command tab and enter the following:



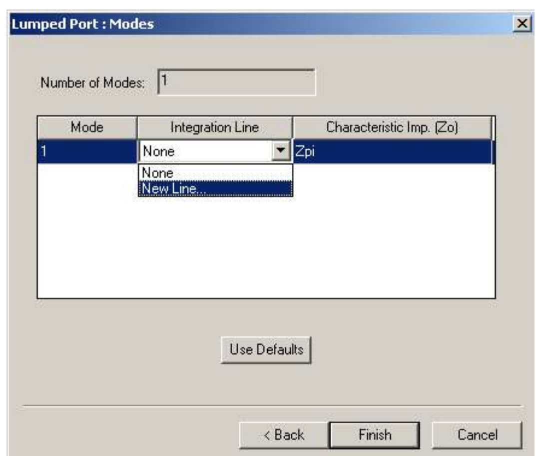
Please note that the variable *gap_src* was chosen relatively small in comparison to the dipole structure. This was done to minimize effects due to the source and place emphasis upon the structure. The source is depicted below.



With the source geometry in place, the user must provide an excitation. A lumped port will be used for the dipole 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.



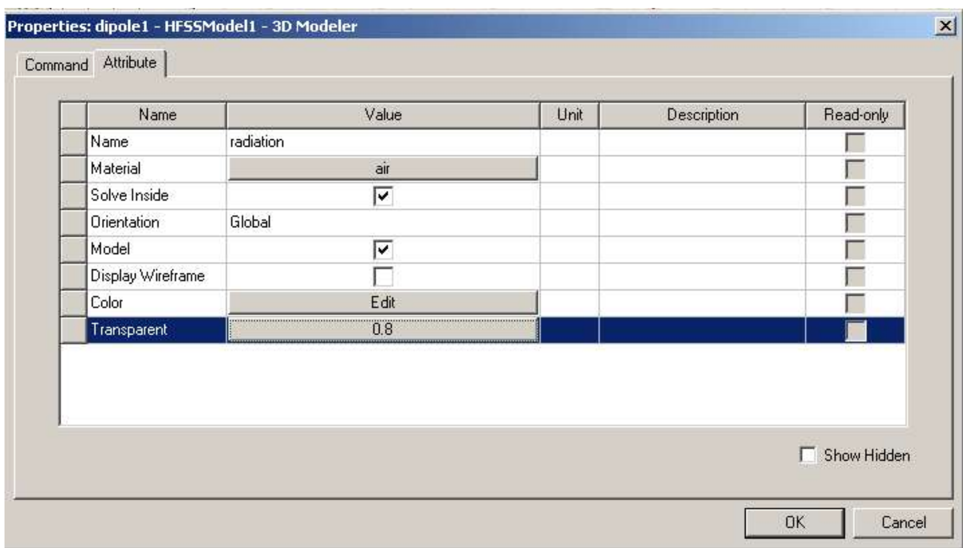
Name the port *source* and leave the default values for impedance. Click Next and enter the following:



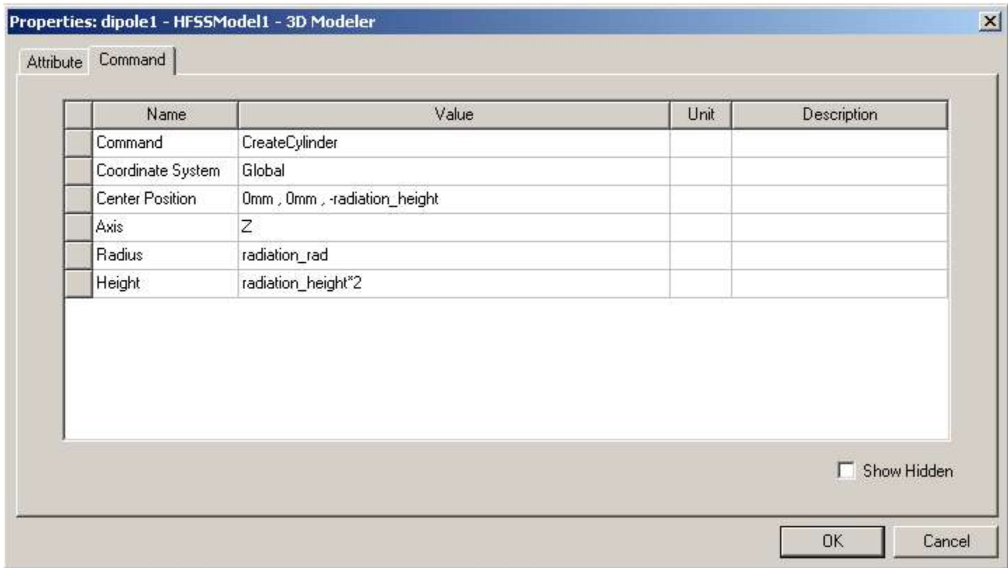
Using the mouse, position the cursor to the bottom-center of the port. Ansoft's snap feature should place the pointer when the user approaches the center of any object. Left-click to define the origin of the E-field vector. Move the cursor to the top-center of the port. Left-click to terminate the E-field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

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.

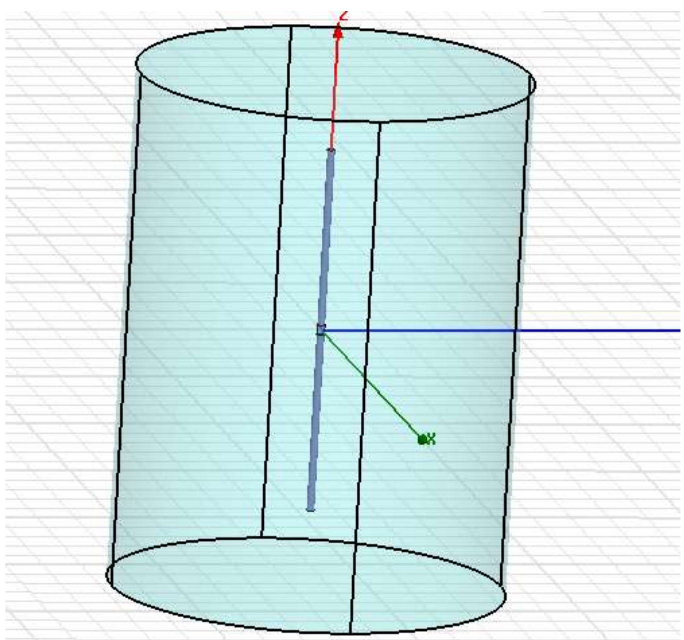
Enter the following:



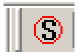
Click the *Command* tab and enter the following:



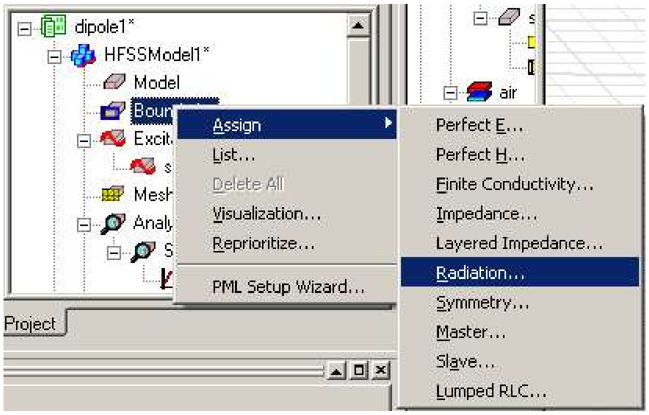
Assuming all steps were properly completed, the boundary should resemble the illustration below:



With the geometry complete, the actual radiation boundary may now be assigned. From the 3D toolbar select face from the drop down window as shown below.

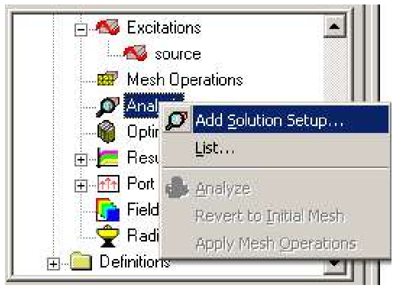
Click  and select all faces as follows:

With all faces selected, right-click the Boundary icon in the object explorer and select Boundary -> Assign -> Radiation.

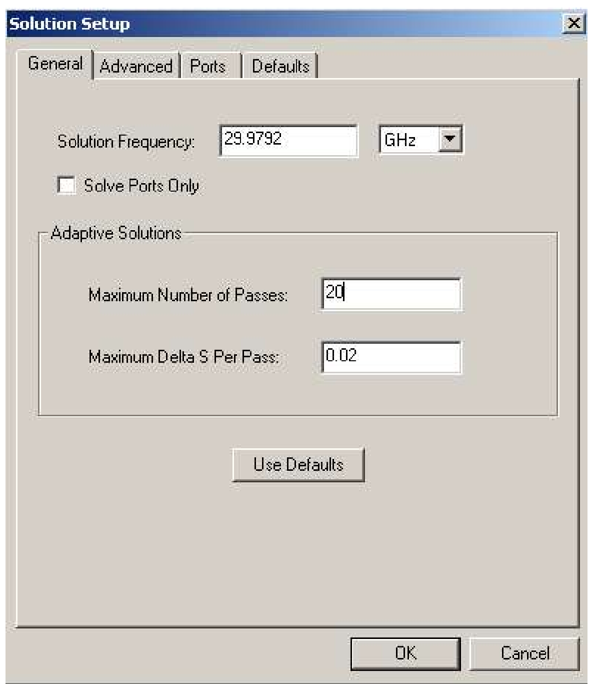


Leave the default name Rad1 and click OK.

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's ability to calculate general antenna parameters such as directivity, radiation resistance, radiation efficiency, etc... . From the project explorer, select Analysis -> Add Solution Setup.

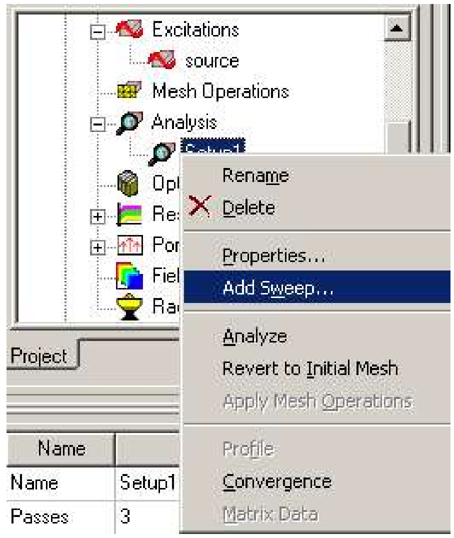


Enter the following:

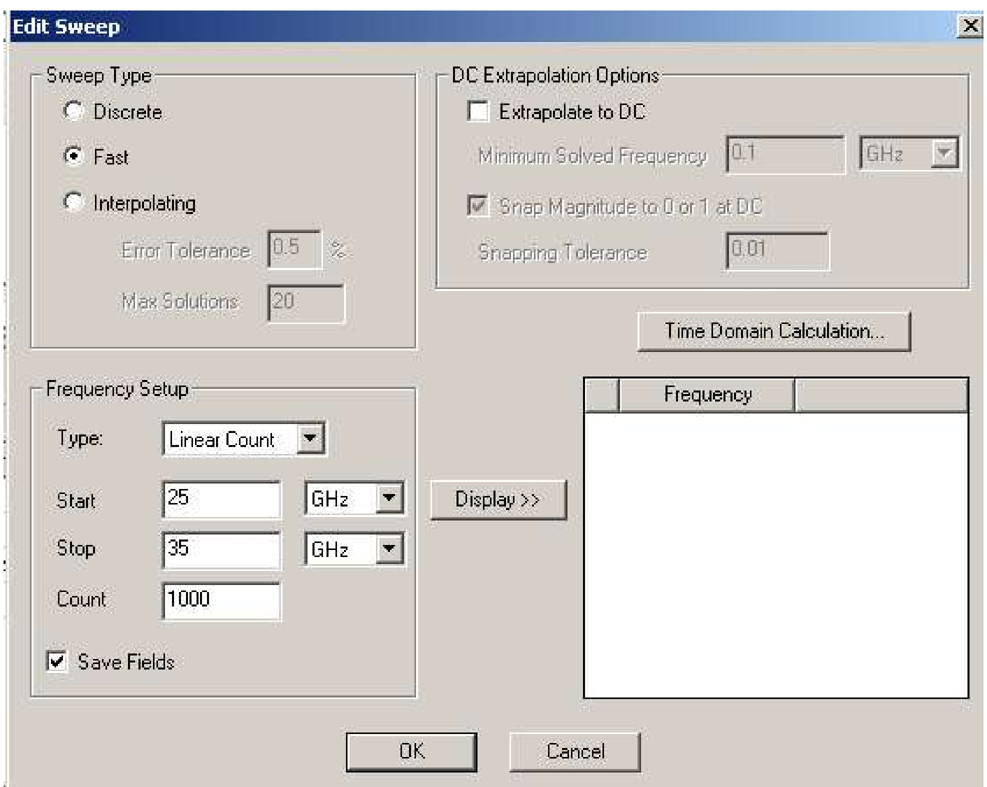


Leave all other settings as default. Click OK when complete.

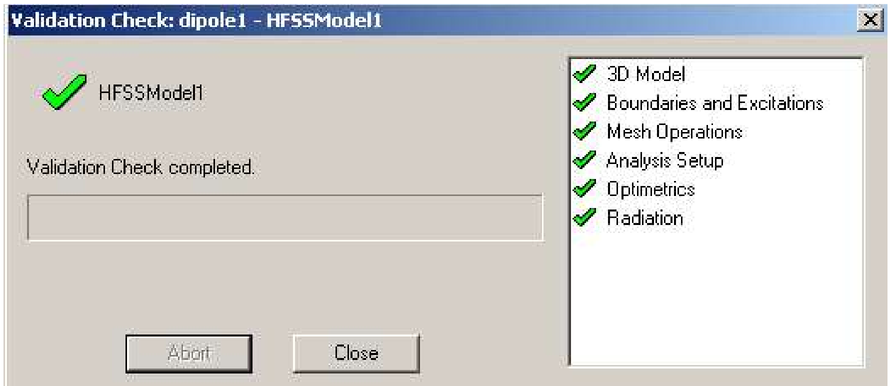
To view the frequency response of the structure, a frequency sweep must be defined. From the project explorer select Setup1 -> Add Sweep.



Enter the following:



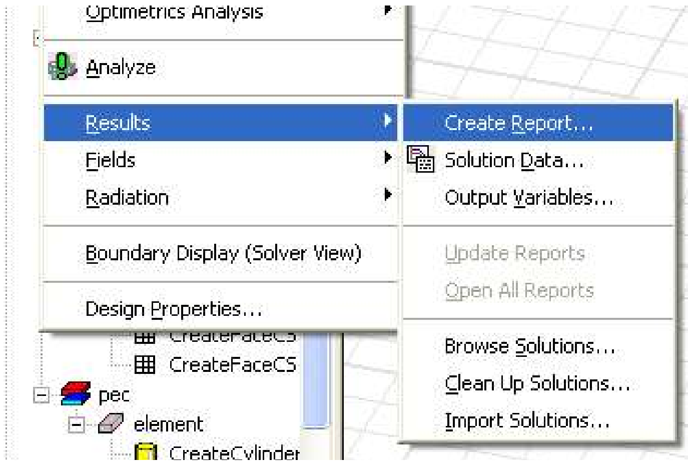
At this point, the user should be ready to analyze the structure. Before running the analysis, always verify the project by selecting  from the 3D toolbar. If everything is correct the user should see:



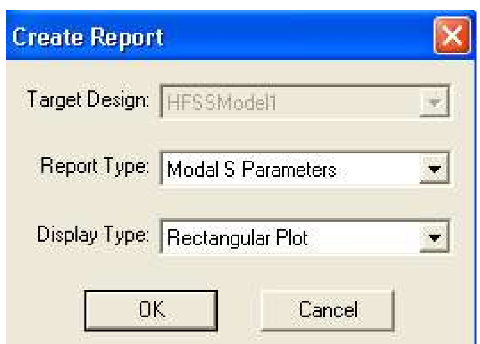
Analyze the structure by clicking 

Allow 5-20 minutes for the analysis, depending on the machine.

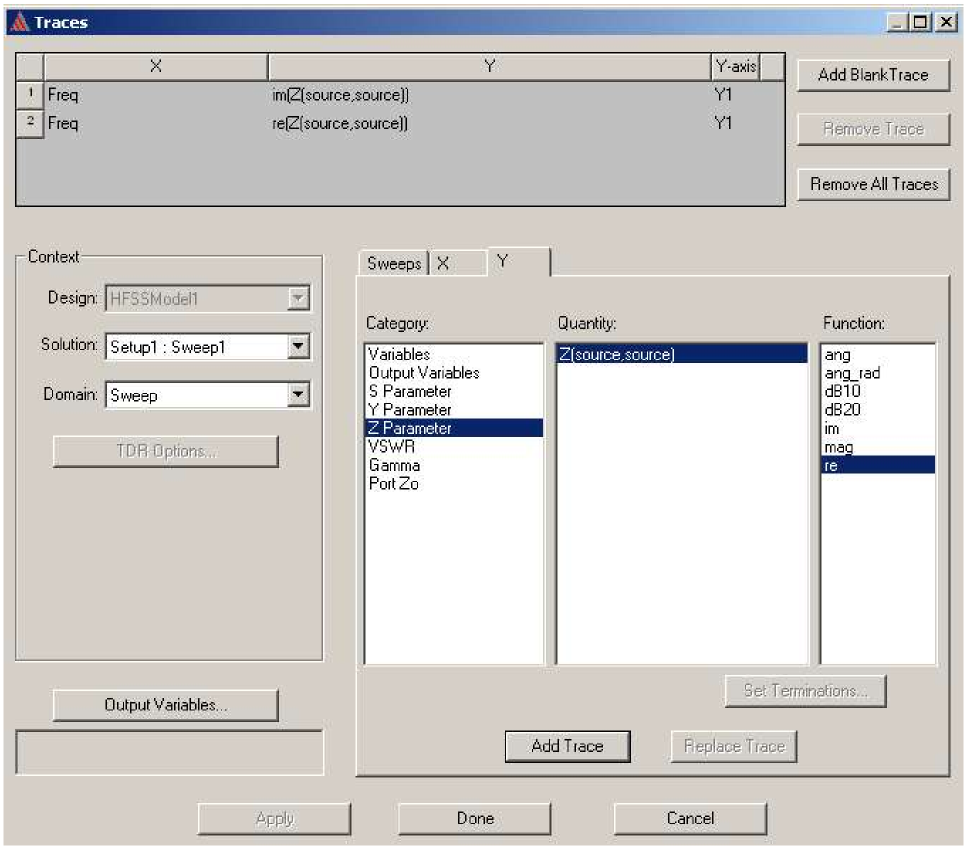
After completion of the analysis, we will create a report to display both the resonant frequency and also the radiation pattern. Click on the heading HFSS and select Results -> Create Reports.



Choose the following in the Create Report window:

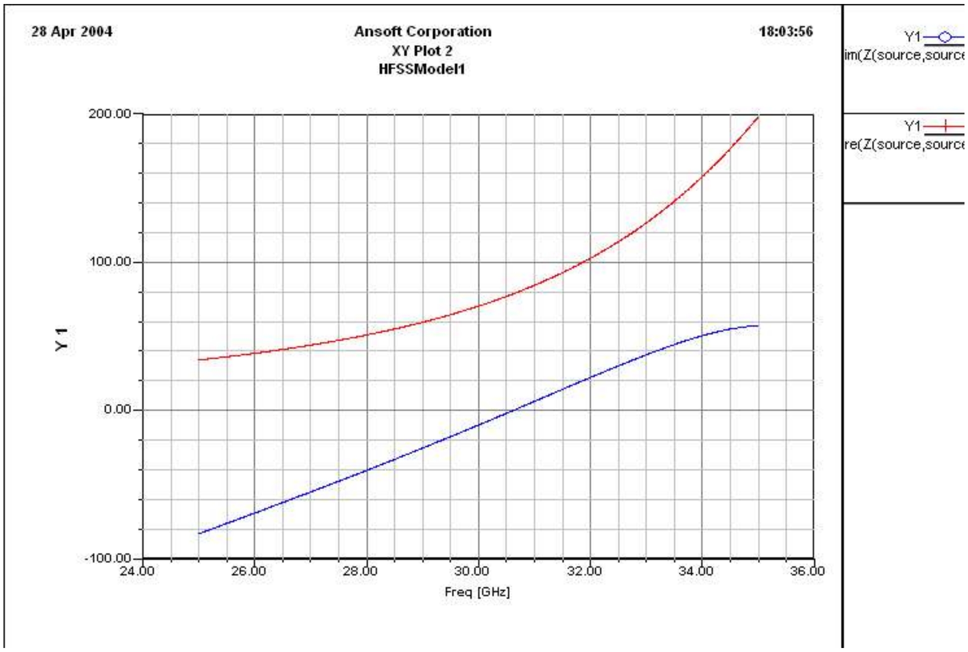


Select the following highlighted parameters and click Add Trace to load the options into the Trace window.

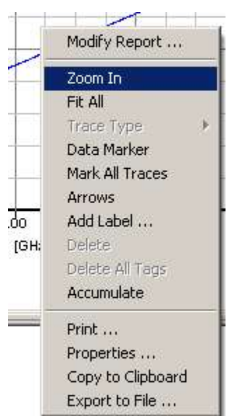


Click Done to display a graph of impedance vs. frequency .

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

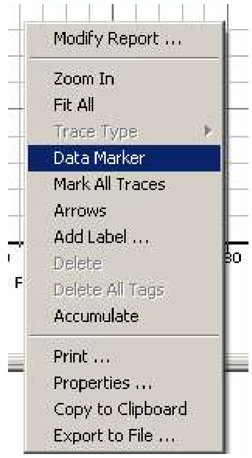


The input resistance can be directly determined from the graph. We will mark the point at which imaginary component crosses zero. This mark will allow the user to determine input impedance at the point of resonance. Right-Click the graph and select Zoom In.

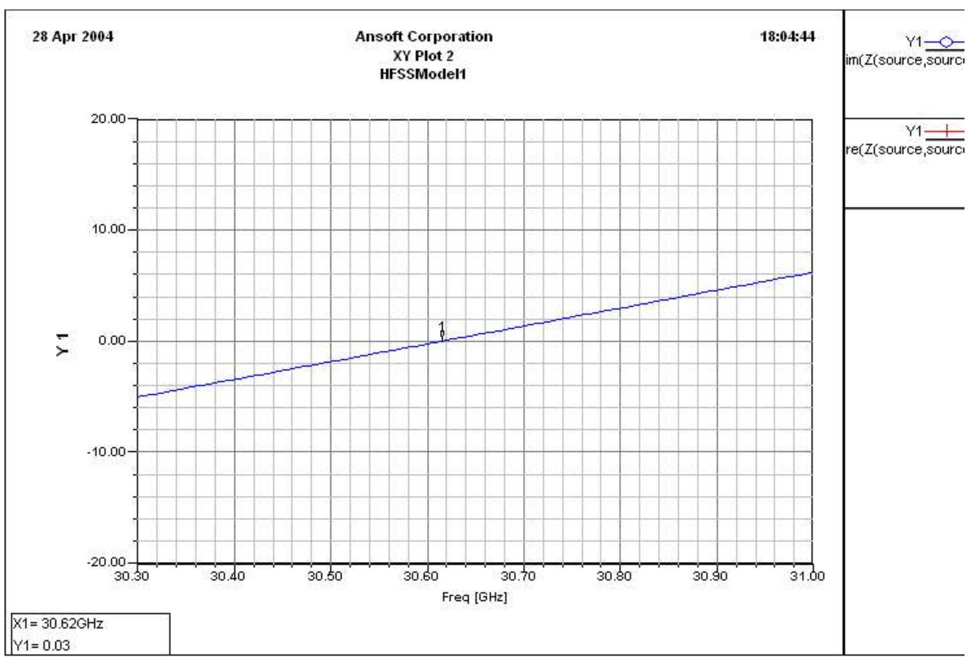


Using the mouse select a zoom window around the imaginary component as it crosses zero.

Mark the zero point by right-clicking the plot window and selecting Data Marker.

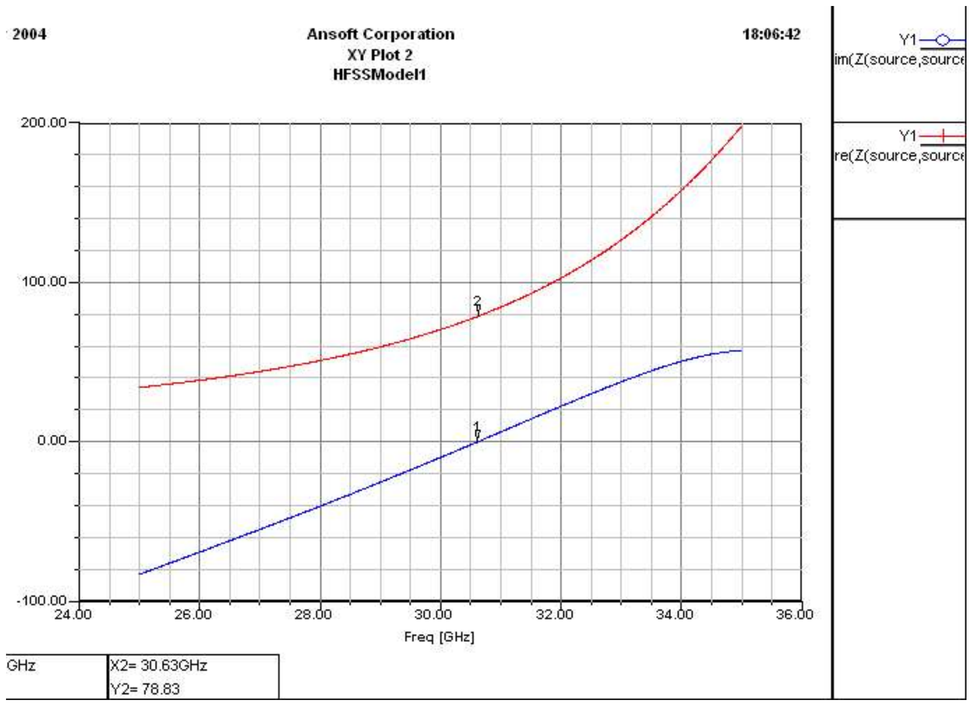


Select a point as close as possible to zero along the imaginary line. You will not be able to choose exactly zero due to the resolution chosen (1000 points) in the solution setup. Left-click to mark the point as shown below:



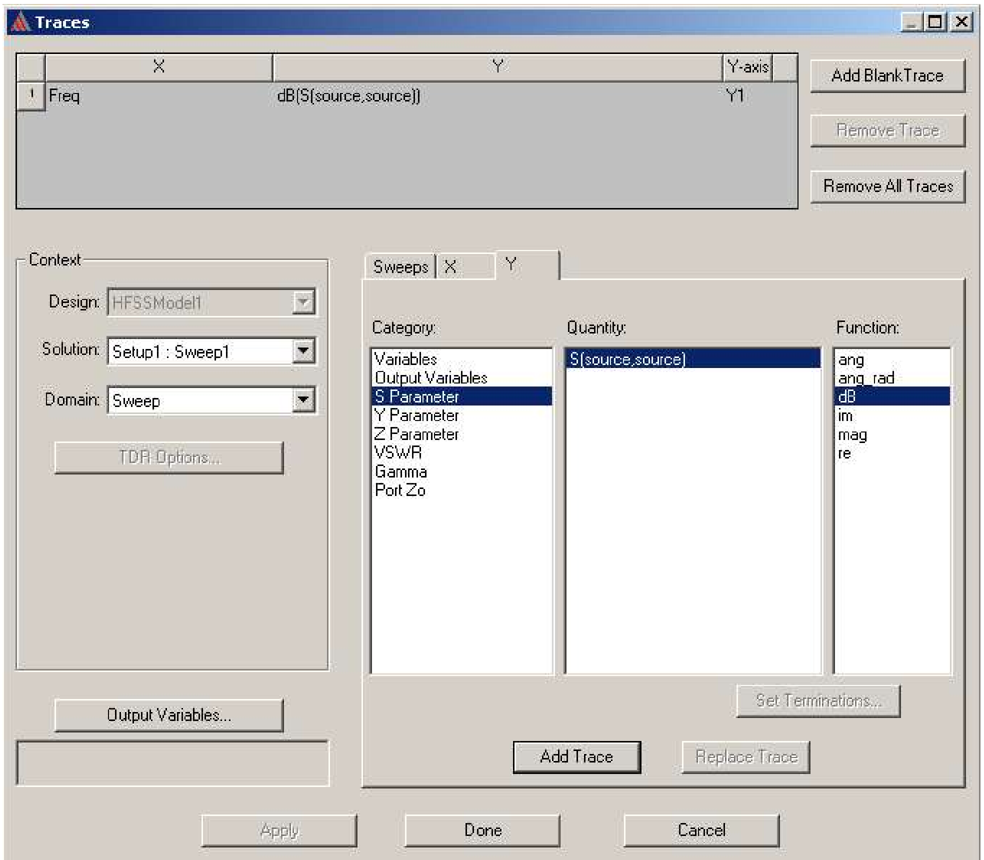
Right-click the plot window and select Fit All. Follow the same procedure to mark the real component at exactly the same frequency of the imaginary component.

Both marked data can be seen in the graph below:



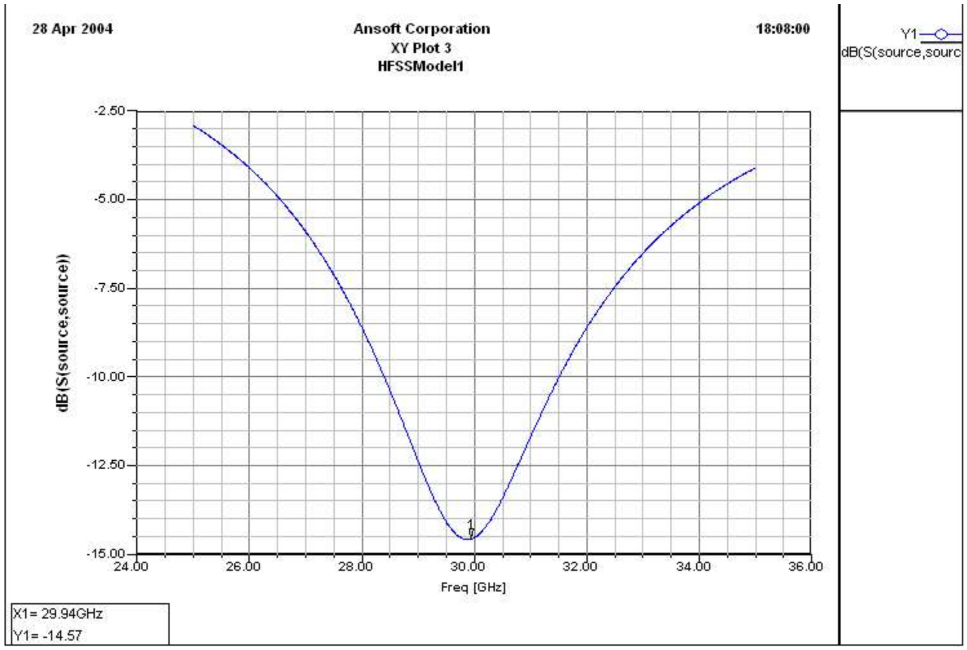
The input resistance of the antenna is 78.83 ohms according to the graph. Performing calculations from a text, the user should compute a resistance between 65 and 75 ohms. The port was previously defined with an impedance of 50 ohms. This will produce sub-optimal results due to mismatched impedance. This will be corrected shortly.

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

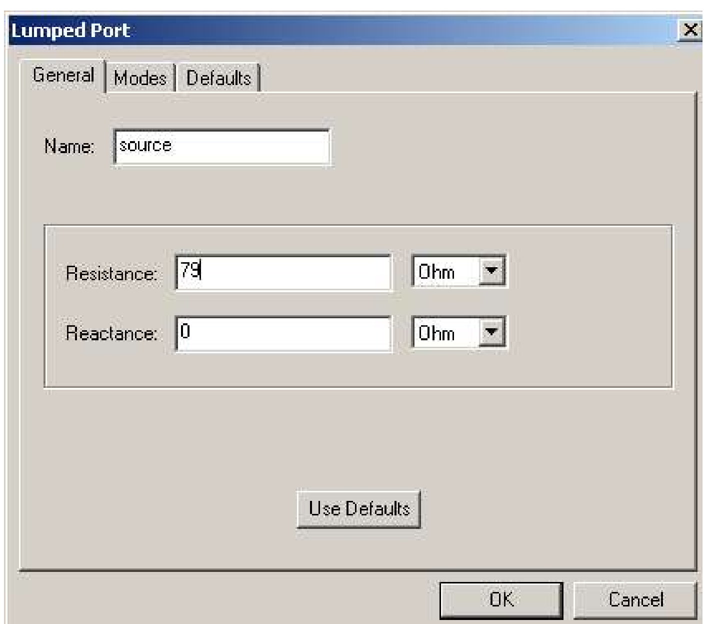


Click Done when complete.

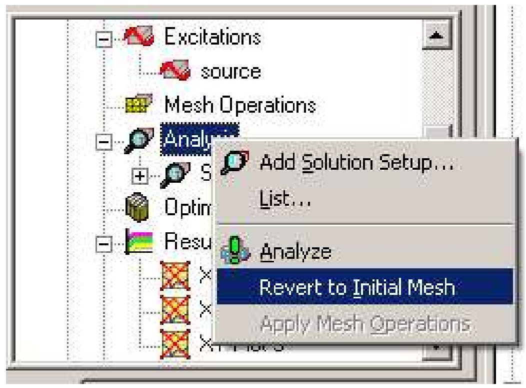
The graph is displayed below:



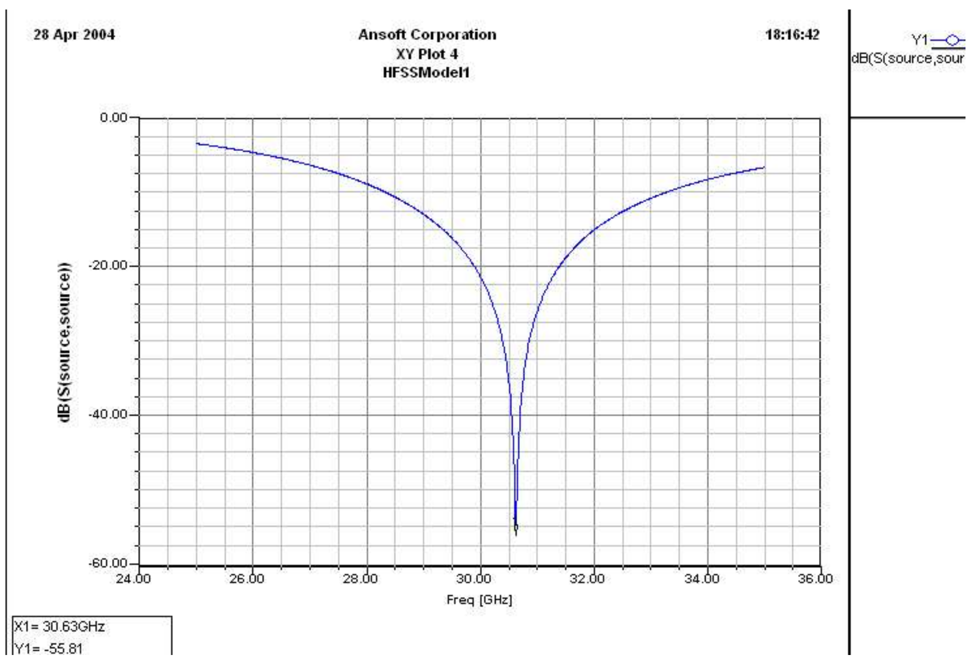
The point of resonance was marked at -14.57dB. In order to compute accurate antenna parameters, the input must be matched. From the project explorer, right-click *source* and select Properties. Adjust the port impedance as shown:



We will now re-analyze the structure with a properly matched port. In order to preserve memory and calculation time, right-click Analysis -> Revert to Initial Mesh in the project explorer.



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

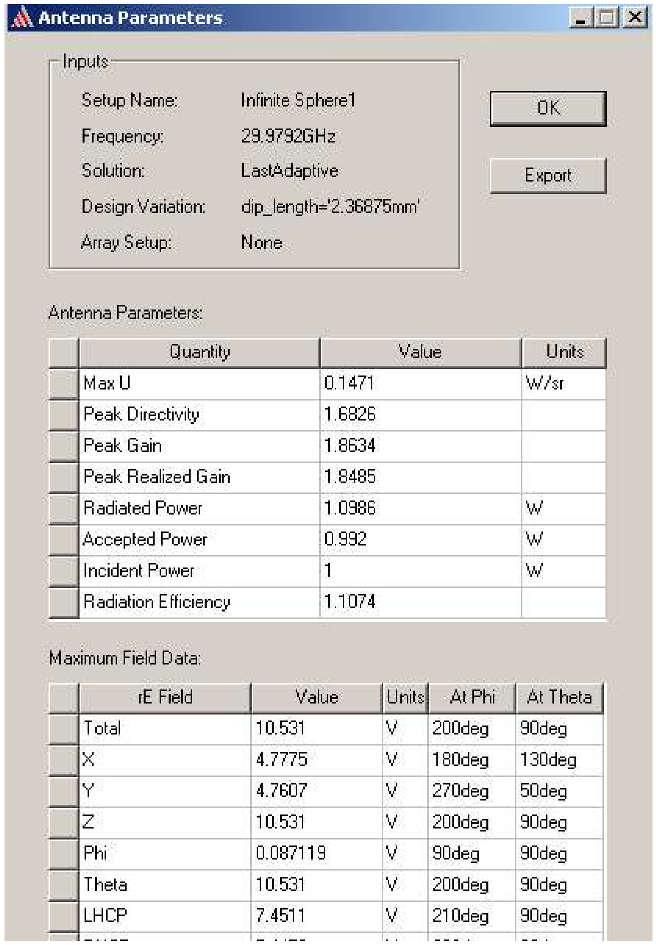


Note the improved response of -55.81dB at resonance.

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 -> Compute Antenna Parameters... from the project explorer as shown:

Select all defaults and results are displayed as follows:



Antenna Parameters

Inputs

Setup Name:	Infinite Sphere1
Frequency:	29.9792GHz
Solution:	LastAdaptive
Design Variation:	dip_length='2.36875mm'
Array Setup:	None

OK

Export

Antenna Parameters:

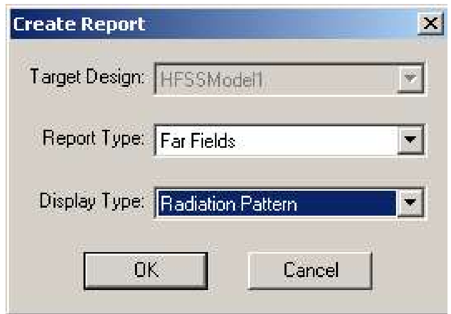
Quantity	Value	Units
Max U	0.1471	W/sr
Peak Directivity	1.6826	
Peak Gain	1.8634	
Peak Realized Gain	1.8485	
Radiated Power	1.0986	W
Accepted Power	0.992	W
Incident Power	1	W
Radiation Efficiency	1.1074	

Maximum Field Data:

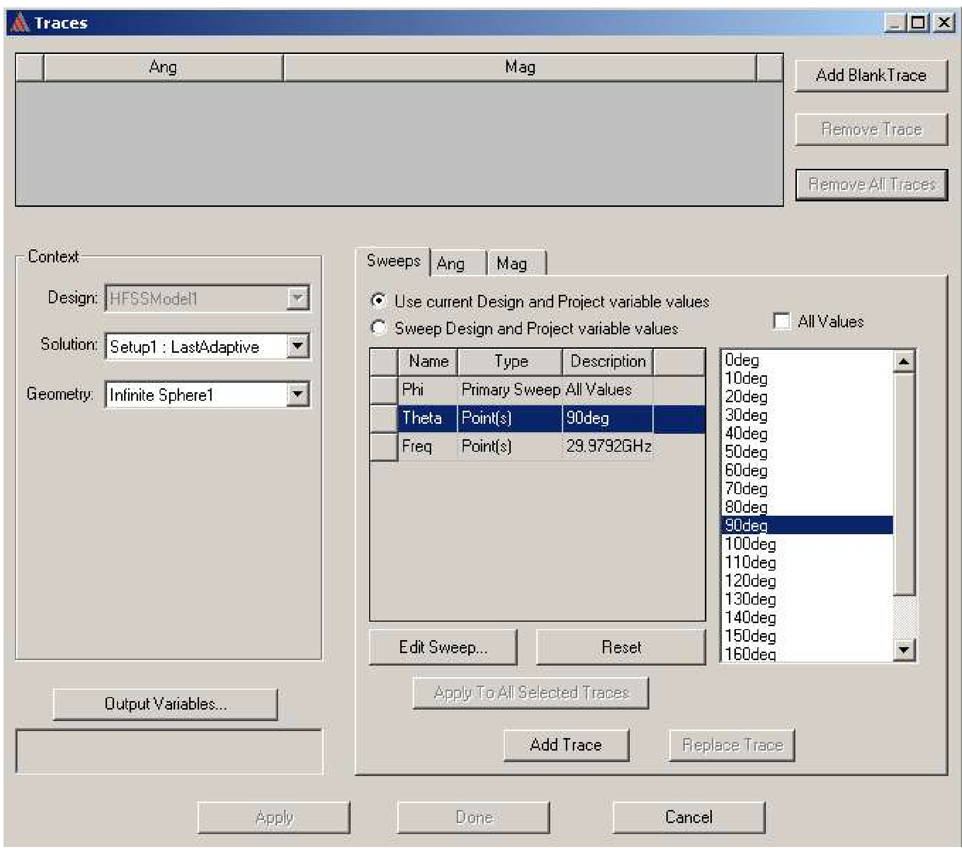
rE Field	Value	Units	At Phi	At Theta
Total	10.531	V	200deg	90deg
X	4.7775	V	180deg	130deg
Y	4.7607	V	270deg	50deg
Z	10.531	V	200deg	90deg
Phi	0.087119	V	90deg	90deg
Theta	10.531	V	200deg	90deg
LHCP	7.4511	V	210deg	90deg

From the chart the Peak Directivity is 1.68. Calculations from standard antenna texts will show this model to be approximately 1.63. All other parameters can be seen as slightly elevated above the expected. Adjustments to the radiation boundary might provide more accuracy.

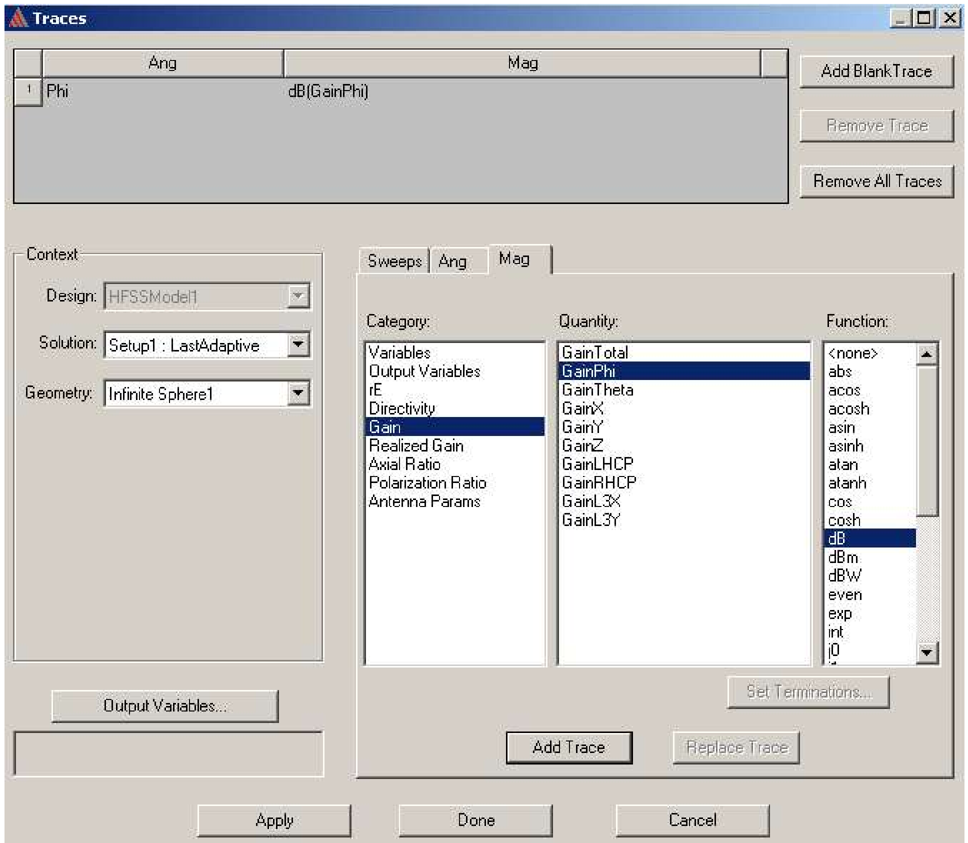
Next, the far field will be plotted. Create Reports as previously shown. Modify the following:



Enter the following:

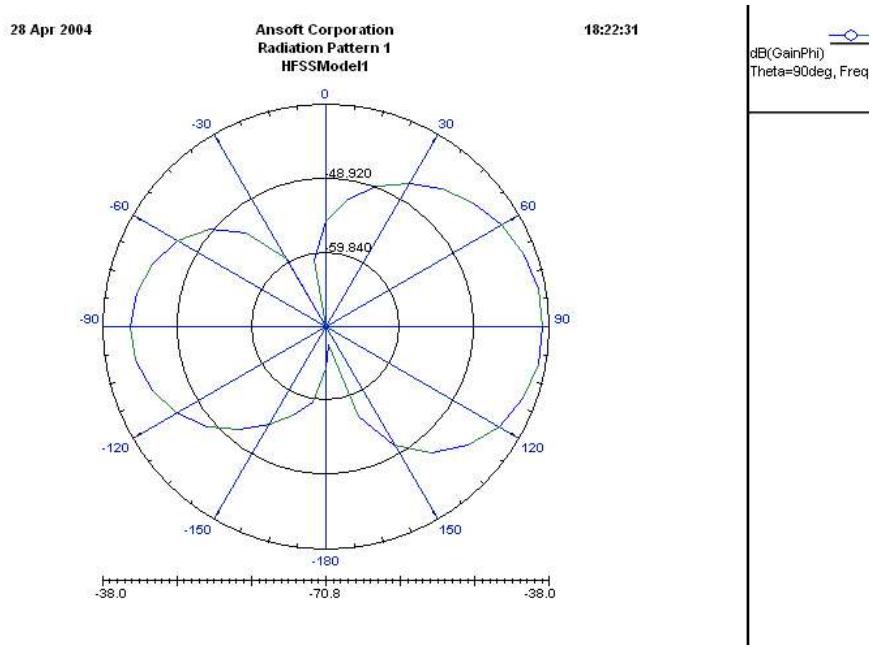


Select the Mag and enter the following:



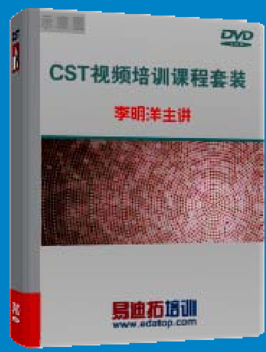
Select Add Trace and click Done when complete.

The radiation pattern is displayed below:



This tutorial was intended to introduce the user to the basic commands and the general procedure for simulation. The software is complex in nature and requires extensive knowledge Electromagnetic Theory to fully utilize its capabilities. The dipole antenna was intended to accelerate the software learning curve and generate interest in the subject.

CST 学习培训课程套装



该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com