

High Frequency Structure Simulator (HFSS) Tutorial

Prepared by

Dr. Otman El Mrabet

IETR, UMR CNRS 6164, INSA, 20 avenue Butte des Coësmes 35043 Rennes, FRANCE

2005 - 2006

TABLE OF CONTENTS

INTRODUCTION	III
HIGH FREQUENCY STRUCTURE SIMULAOR (HFSS)	IV
CHAPTER ONE - THE DIPOLE ANTENNA	1
CHAPTER TWO - THE RECTANGULAR PATCH ANTENNA.....	22
CHAPTER THREE - PROBE FEED PATCH ANTENNA	34
CHAPTER FOUR - THE TRIANGULAR MICROSTRIP ANTENNA.....	48
PROJECTS	

Introduction

Wireless communications have progressed very rapidly in recent years, and many mobile units are becoming smaller and smaller. To meet the miniaturization requirement, the antennas employed in mobile terminals must have their dimensions reduced accordingly. Planar antennas, such as microstrip and printed antennas have the attractive features of low profile, small size, and conformability to mounting hosts and are very promising candidates for satisfying this design consideration. For this reason, compact, broadband and wideband design technique for planar antennas have been attracted much attention from antenna researchers. Very recently, especially after the year 2000, many novel planar antenna designs to satisfy specific bandwidth specifications of present day mobile cellular communication systems including the global system for mobile communication (GSM; 890 - 960 MHz), the digital communication system (DCS; 1710 - 1880 MHz), the personal communication system (PCS; 1850 - 1990 MHz), and the universal mobile telecommunication system (UMTS; 1920 - 2170 MHz), have been developed and published in the open literature.

Planar antennas are also very attractive for applications in communication devices for wireless local area network (WLAN) systems in the 2.4 GHz (2400 - 2484 MHz) and 5.2 GHz (5150 - 5350 MHz) bands.

The aim of this tutorial is to show you how to use HFSS to design planar antennas for wireless communications. Therefore, we have chosen four antennas types; dipole antenna, the rectangular patch antenna, probe feed patch antenna and triangular microstrip antenna. At the end, we will propose some projects.

High Frequency Simulator Structure (HFSS)

HFSS is a high performance full wave electromagnetic (EM) field simulator for arbitrary 3D volumetric passive device modelling that takes advantage of the familiar Microsoft Windows graphical user interface. It integrates simulation, visualization, solid modelling, and automation in an easy to learn environment where solutions to your 3D EM problems are quickly and accurately obtained. Ansoft HFSS employs the Finite Element Method (FEM), adaptive meshing, and brilliant graphics to give you unparalleled performance and insight to all of your 3D EM problems. Ansoft HFSS can be used to calculate parameters such as S-Parameters, Resonant Frequency, and Fields. Typical uses include:

- Package Modelling - BGA, QFP, Flip-Chip
- PCB Board Modelling - Power/ Ground planes, Mesh Grid Grounds, Backplanes
- Silicon/GaAs-Spiral Inductors, Transformers
- EMC/EMI - Mobile Communications - Patches, Dipoles, Horns, Conformal Cell Phone Antennas, Quadrafilar Helix, Specific Absorption Rate (SAR), Infinite Arrays, Radar Section (RCS), Frequency Selective Surface (FSS)
- Connectors - Coax, SFP/XFP, Backplane, Transitions
- Waveguide - Filters, Resonators, Transitions, Couplers
- Filters - Cavity Filters, Microstrip, Dielectric
- HFSS is an interactive simulation system whose basic mesh element is a tetrahedron. This allows you to solve any arbitrary 3D geometry, especially

those with complex curves and shapes, in a fraction of the time it would take using other techniques.

- The name HFSS stands for High Frequency Structure Simulator. Ansoft pioneered the use of the Finite Element Method (FEM) for EM simulation by developing / implementing technologies such as tangential vector finite elements, adaptive meshing, and Adaptive Lanczos - pade Sweep (ALPS). Today, HFSS continues to lead the industry with innovations such as Modes to Nodes and Full wave Spice.
- Ansoft HFSS has evolved over a period of years with input from many users and industries. In industry, Ansoft HFSS is the tool of choice for High productivity research, development, and virtual prototyping.

Chapter one - The Dipole Antenna

I – Introduction

The monopole and dipole antennas are commonly used for broadcasting, cellular phones, and wireless communications due to their omnidirectional property. Thus in this tutorial, a dipole antenna will be constructed and analyzed using the HFSS simulator. 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.

1- Starting HFSS

- Click the Microsoft **Démarrer** button, Select Programs, and select **Ansoft, HFSS 9.2** program group. Click **HFSS 9.2**.

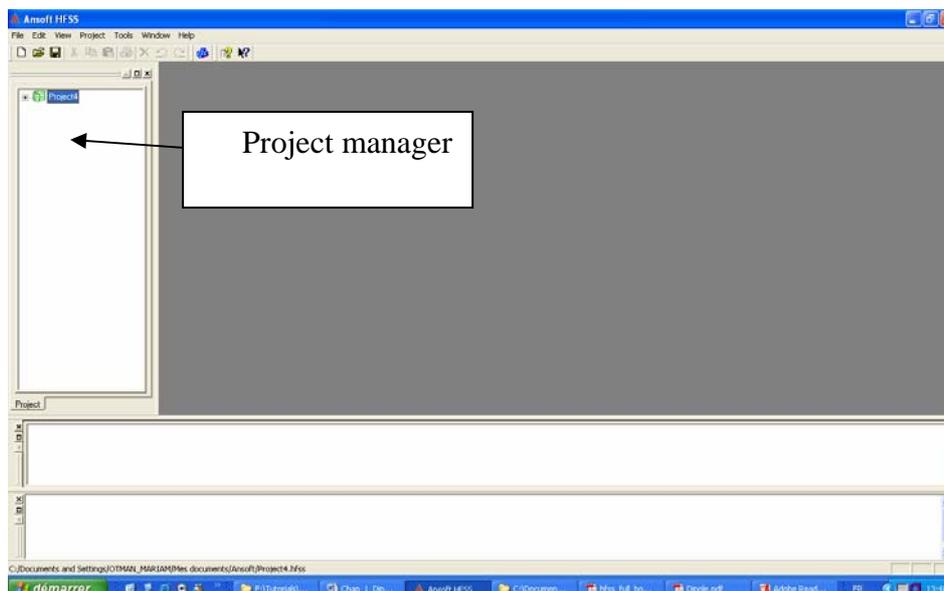
- Or Double click on the **HFSS 9.2** icon on the Windows Desktop.



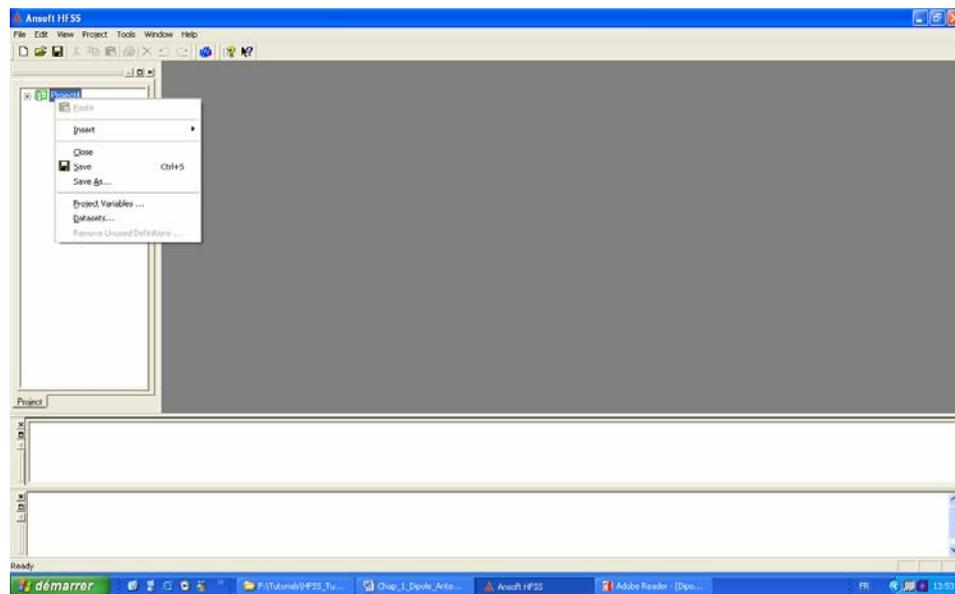
HFSS 9.Ink

2- Creating the Project

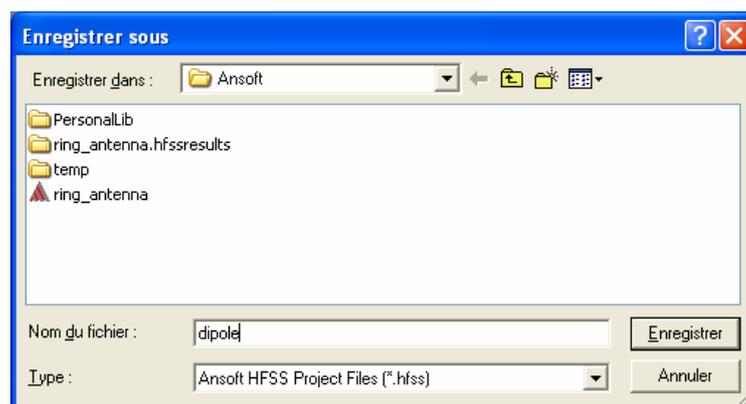
First launch the HFSS Simulator.



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**.

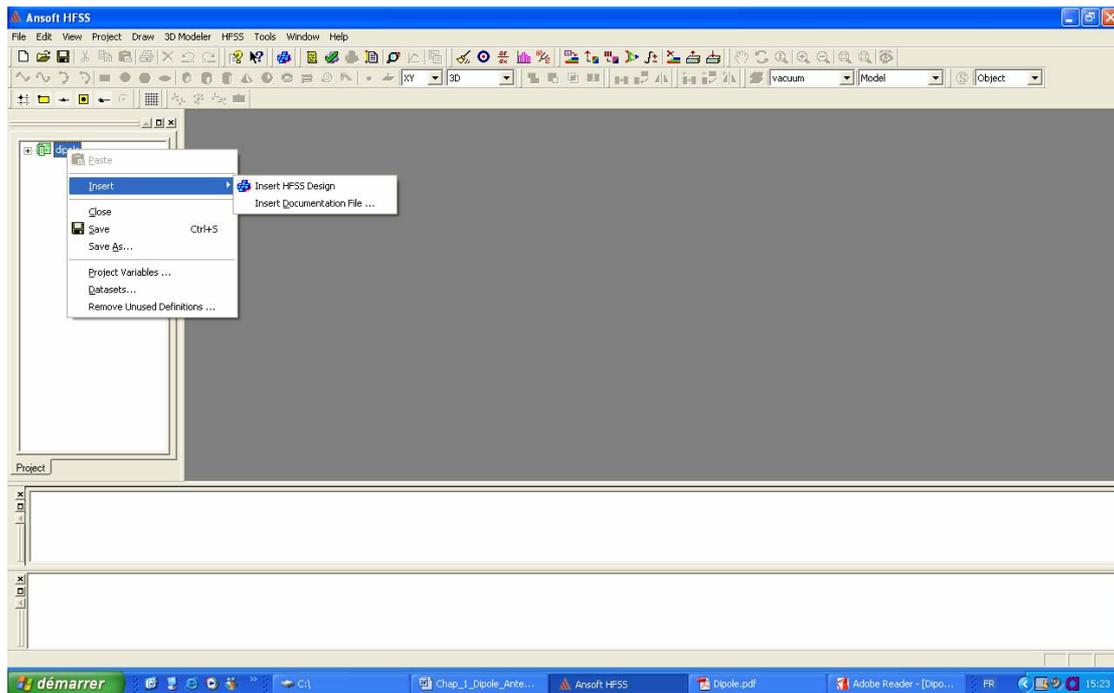


Note: Before click on “**Enregistrer**”, always create a personal folder to store all HFSS projects.

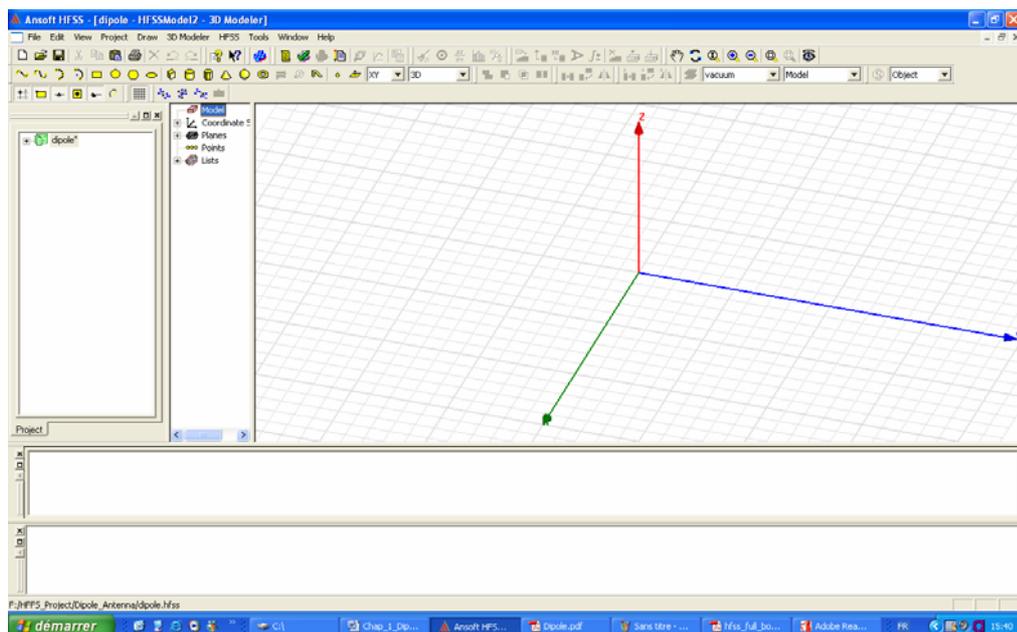
3- Working with geometries

To begin working with geometries.

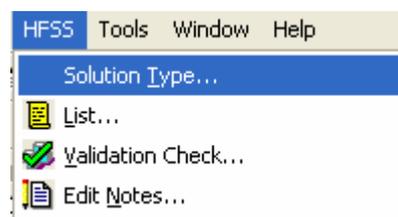
- you must insert an HFSS design. Right-Click the project file and select **Insert** > **Insert HFSS Design** from the menu.



- Or click on  from the toolbars.

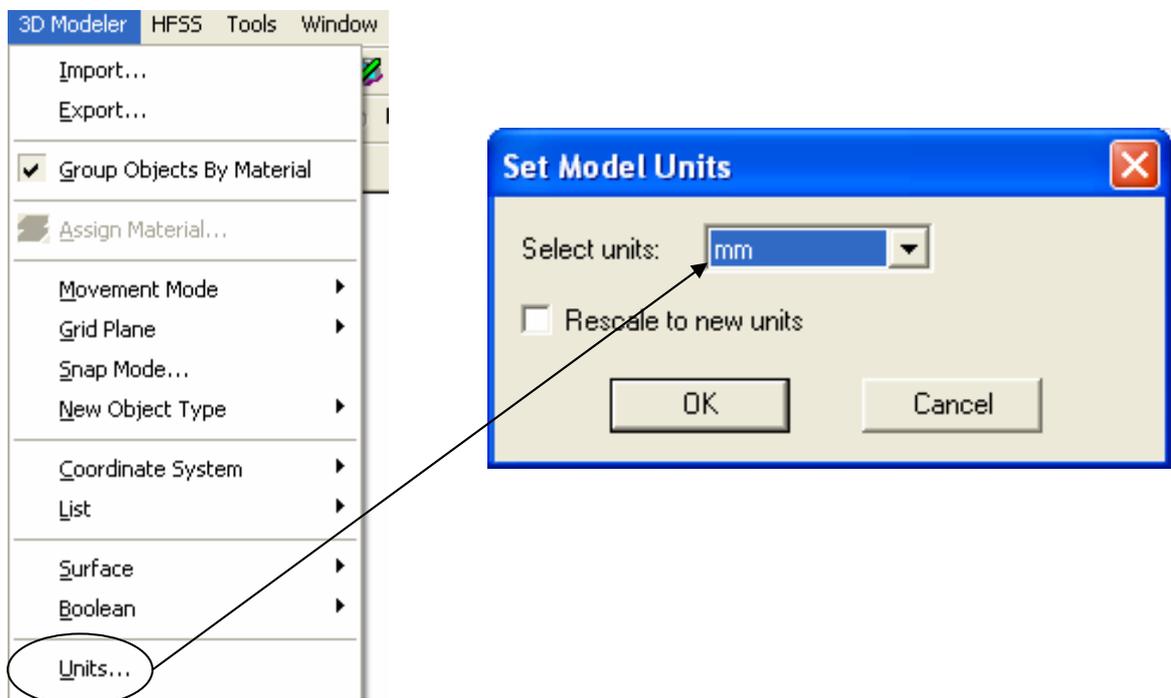


Due to the nature of this design we will use **Driven Modal** as the solution type. From the HFSS menu select **Solution Type** and **Driven Modal**.





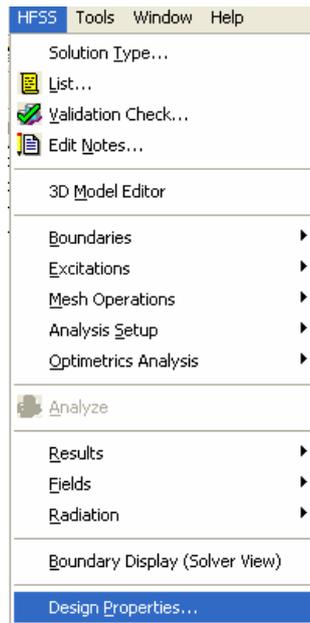
The units are chosen as mm by choosing the heading 3D modeler and Units from the menu.



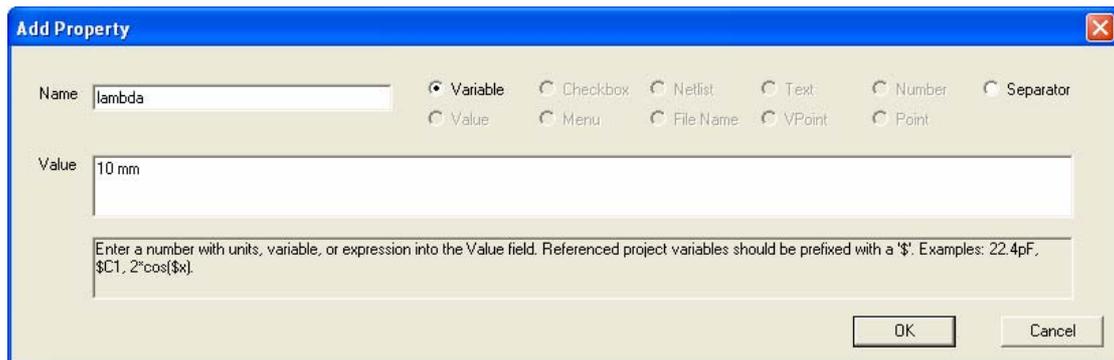
HFSS relies on variables for any parameterization / optimization within the project. Variables also hold many other benefits which will make them necessary for all projects.

- 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.

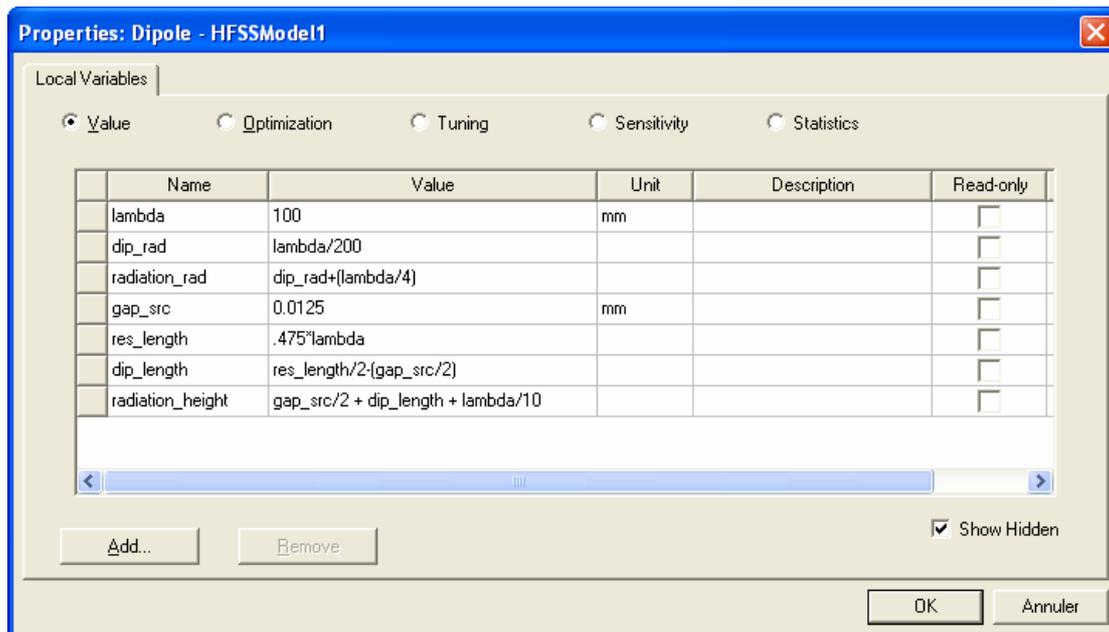
Click the **HFSS** heading and select **Design Properties** at the bottom of the menu.



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

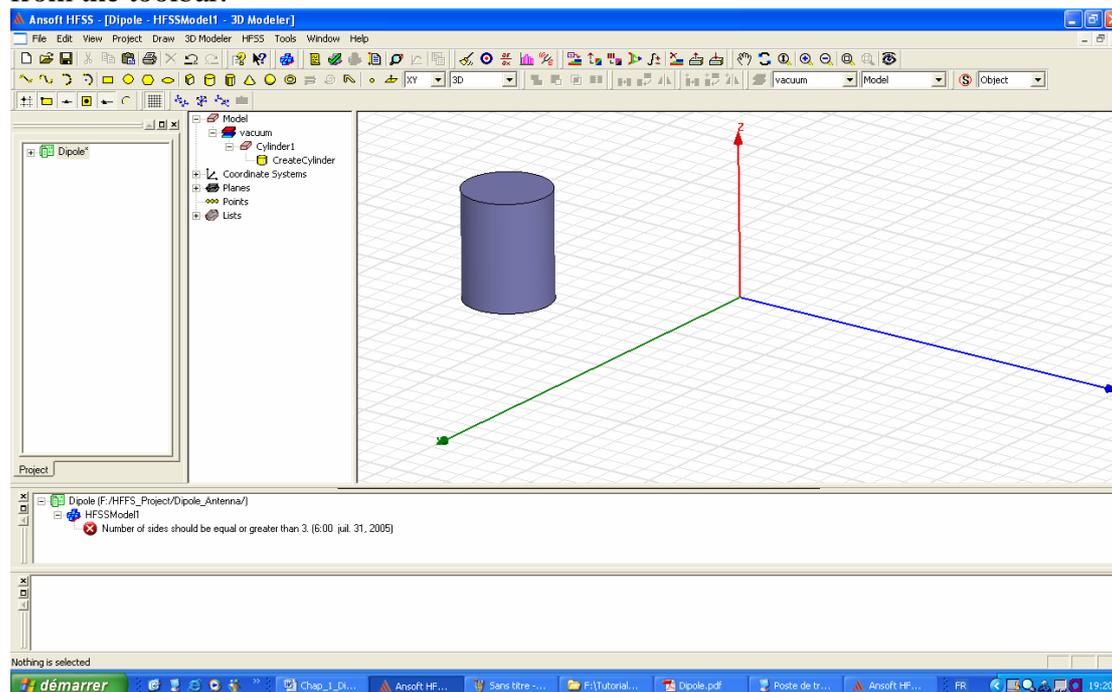


The final variable table should look like

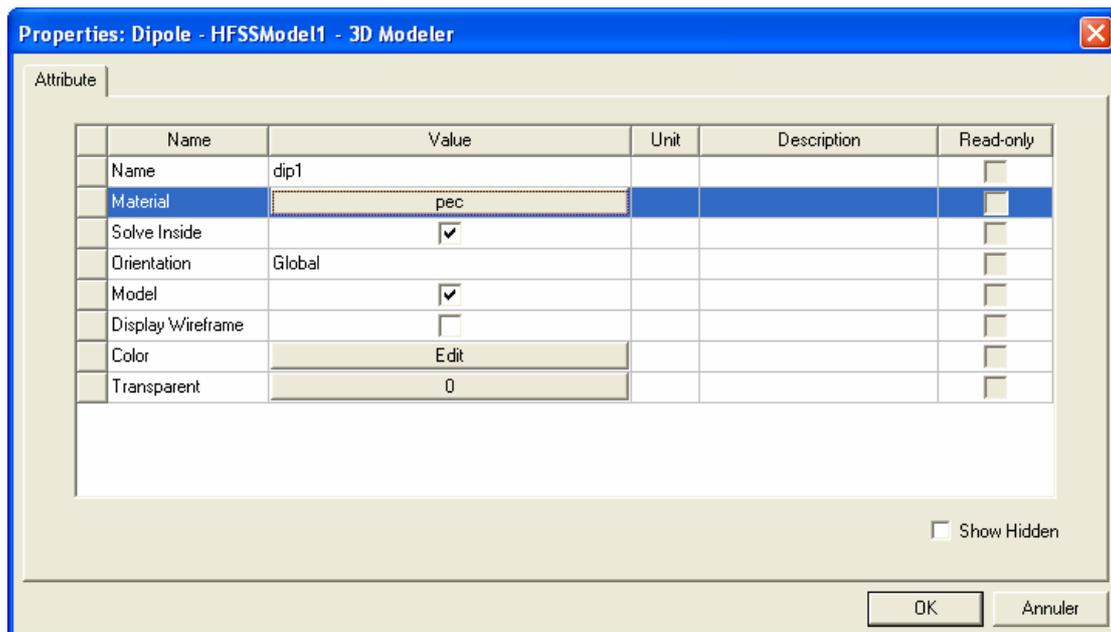
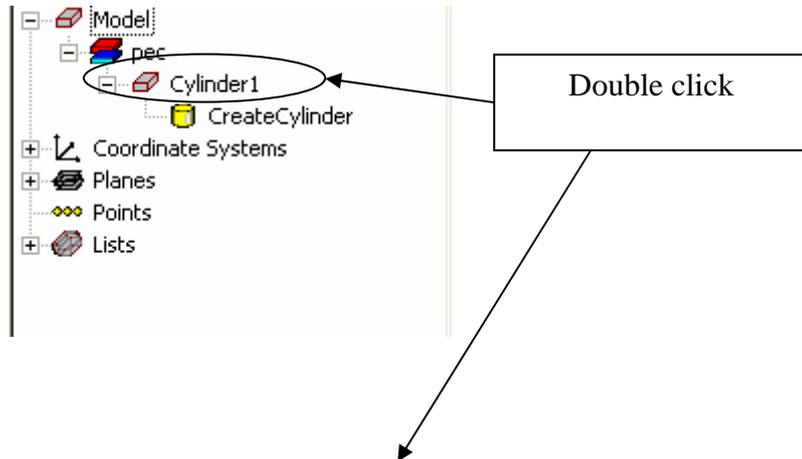
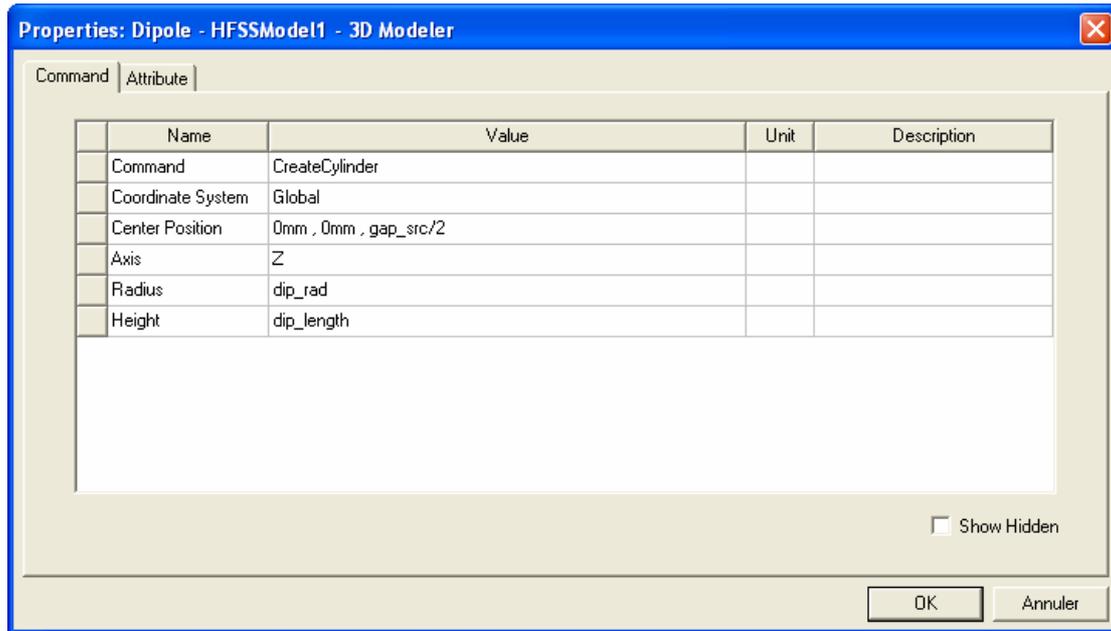


4- Drawing the Dipole

We will start to by creating the dipole element using the **Draw Cylinder** button  from the toolbar.

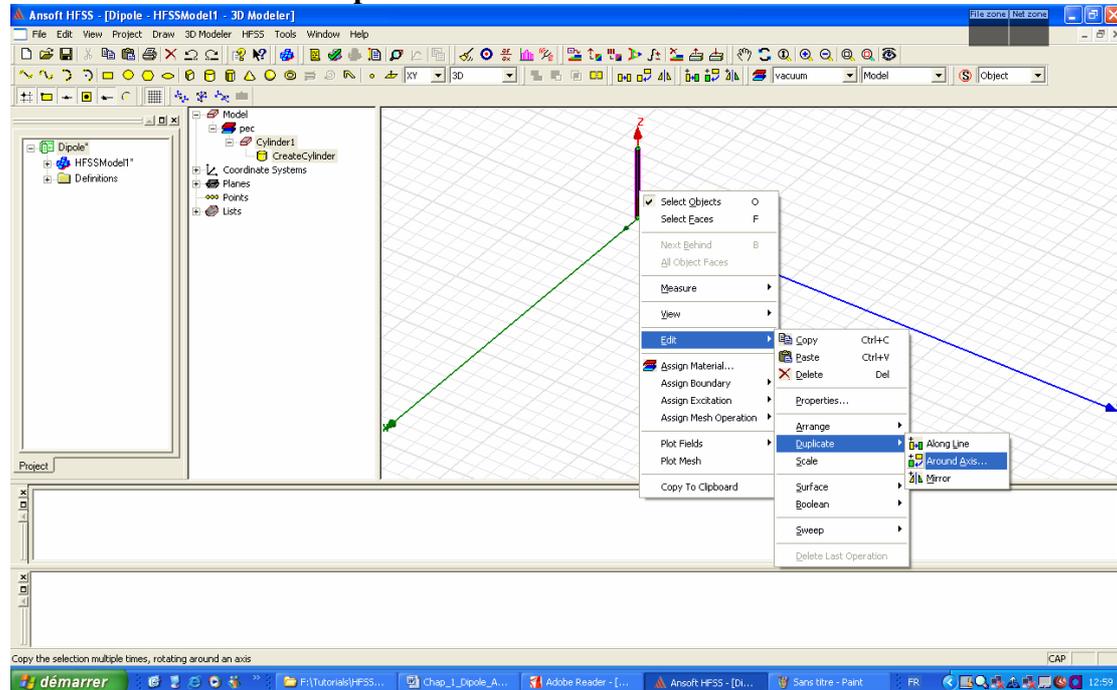


By default the properties dialog will appear after you have finished drawing an object. The position and size of objects can be modified from the dialog.

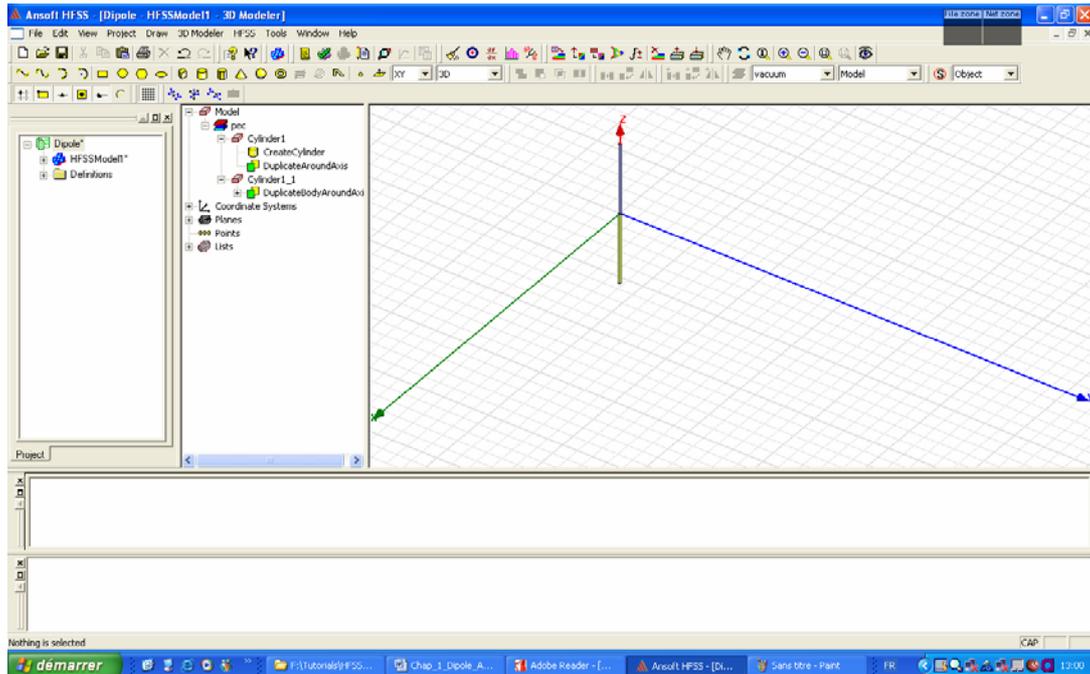


Follow the format above for structure size. Give the name dip1 to this object. Assign the material PEC and click OK. PEC (Perfect Electric Conductor) will create ideal conditions for the element.

The next step is to build the symmetric of dip1. To do that, Right -Click the drawing area and select **Edit -> Duplicate -> Around Axis**.

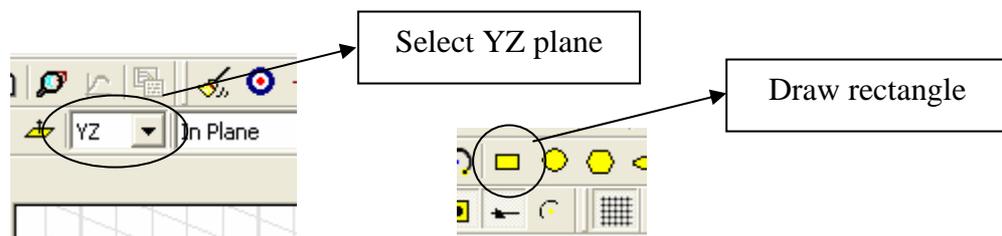


The dipole structure is illustrated below:

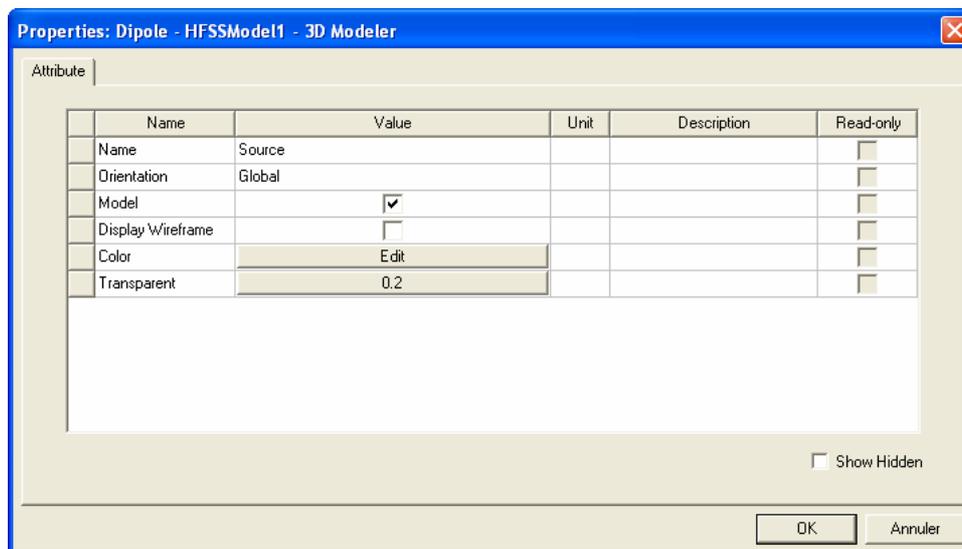


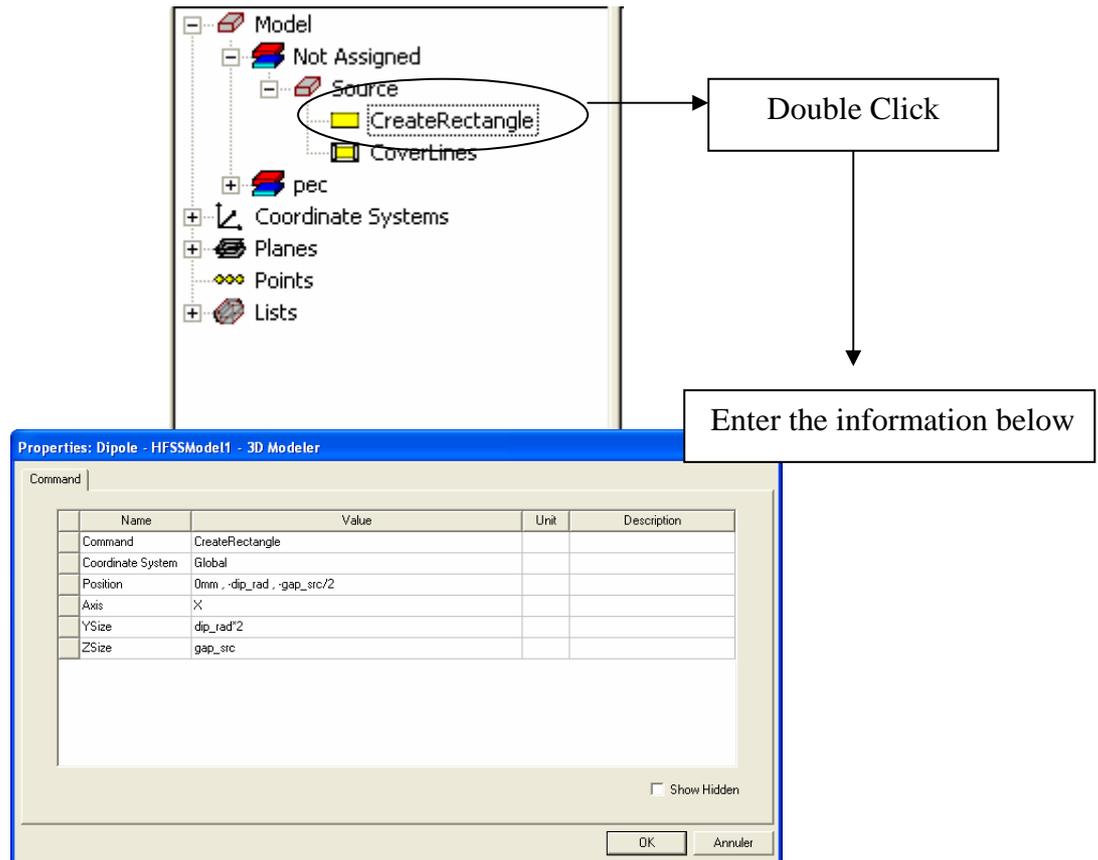
5- Creating the port

In the section you 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.

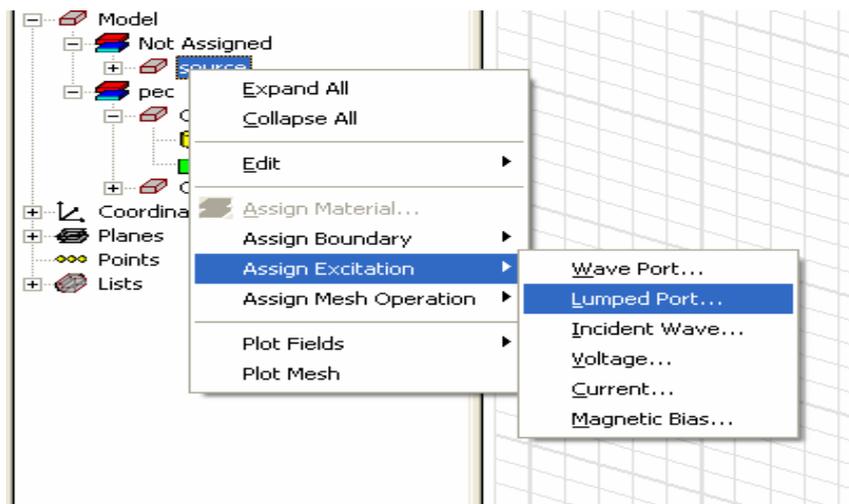


Enter the following information





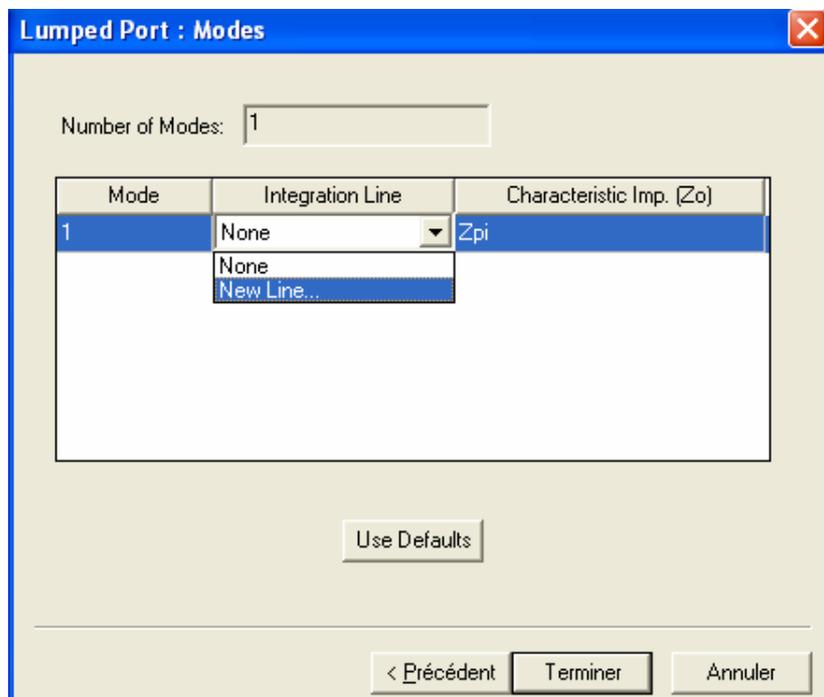
With the source geometry in place, the user must provide an excitation. A lumped port will be used for the dipole model. This excitation is commonly used when the far field region is of primary interest. In the project explorer, right-click **Excitation** -> **Assign** -> **Lumped Port**.



Name the port source and leave the default values for impedance.

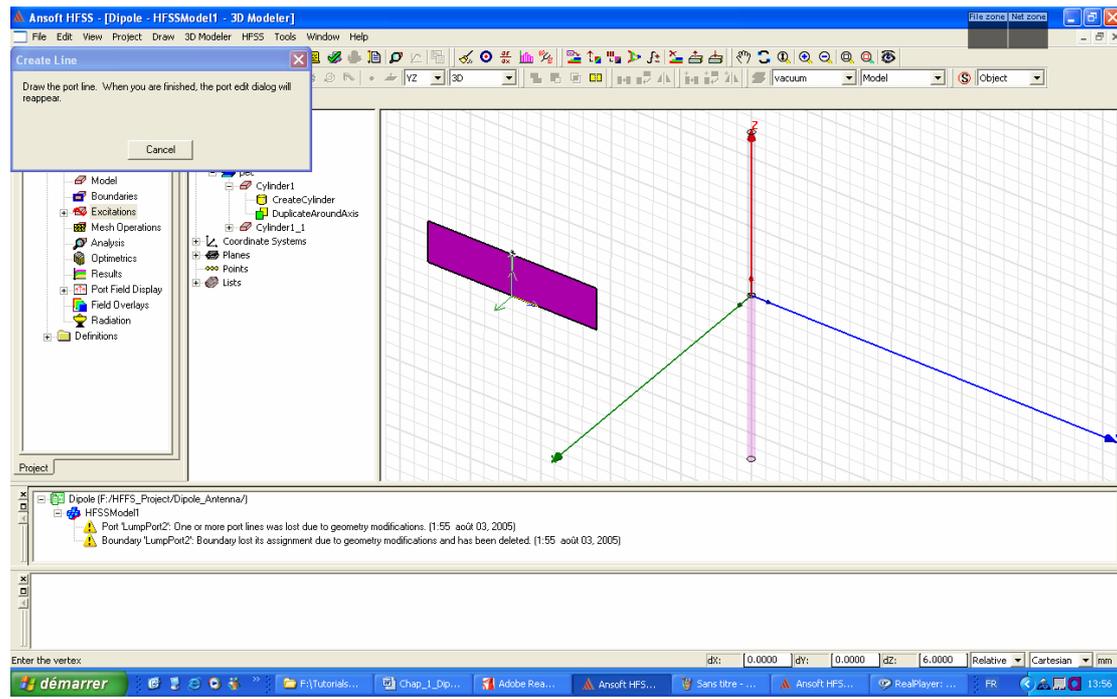


Click Next and enter the following:



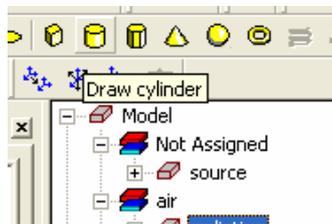
Using the mouse, position the cursor to the bottom-center of the port. Ansoft's snap feature should place the pointer when the user approaches the center of any object. Left-click to define the origin of the E-field vector. Move the cursor to the top-center of the port. Left-click to terminate the E-field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

Note: In case you find some difficulties for drawing the lumped port, you can redraw the rectangular plane, affect the lumped port, then resize the rectangular plane.

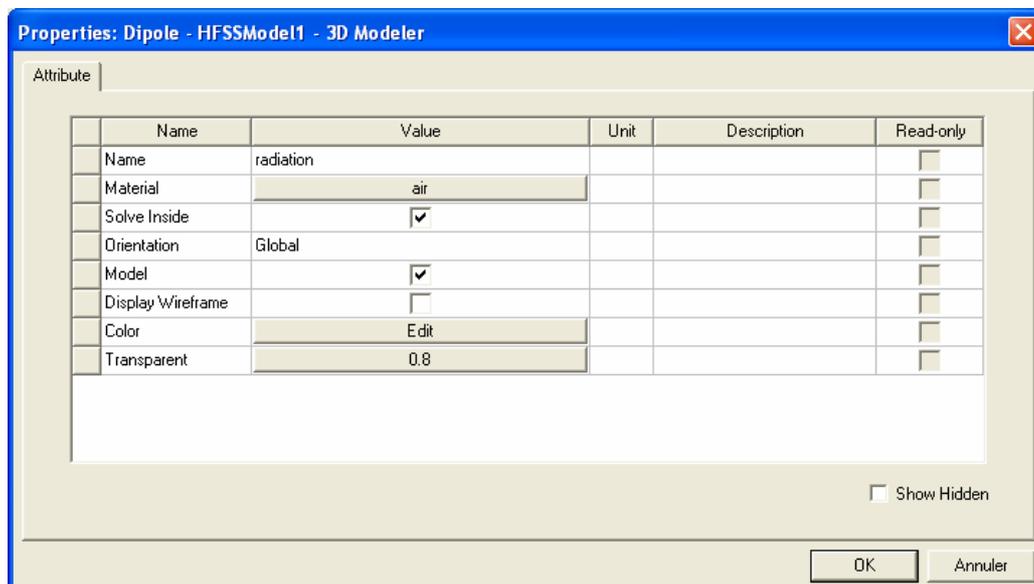


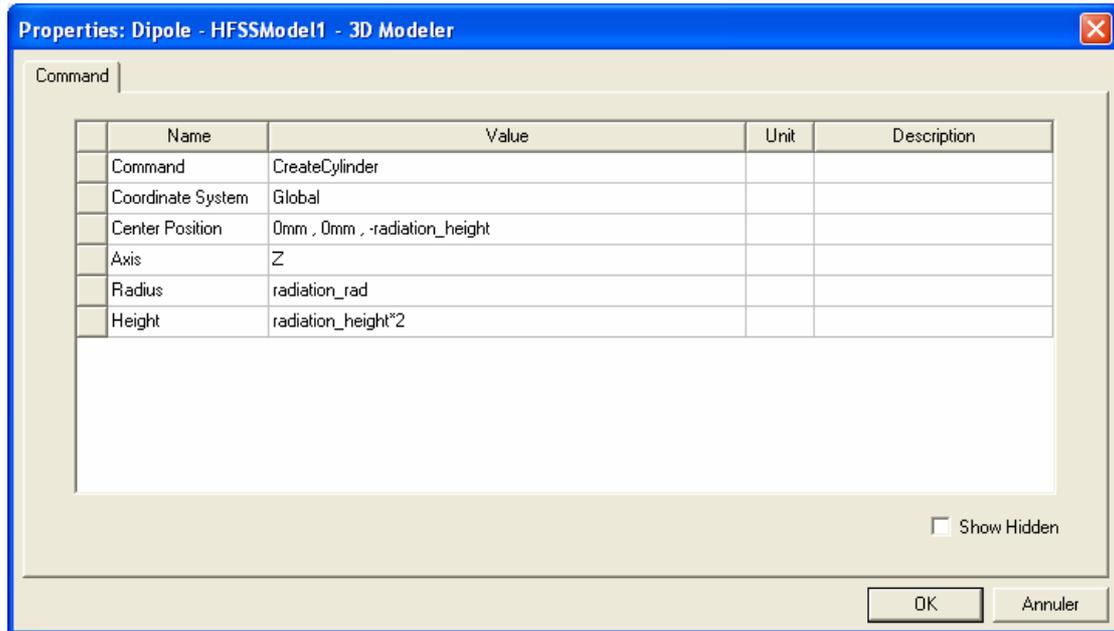
6- Radiation Boundary

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 $\lambda/4$. From the toolbar, select Draw Cylinder.



Enter the following information:





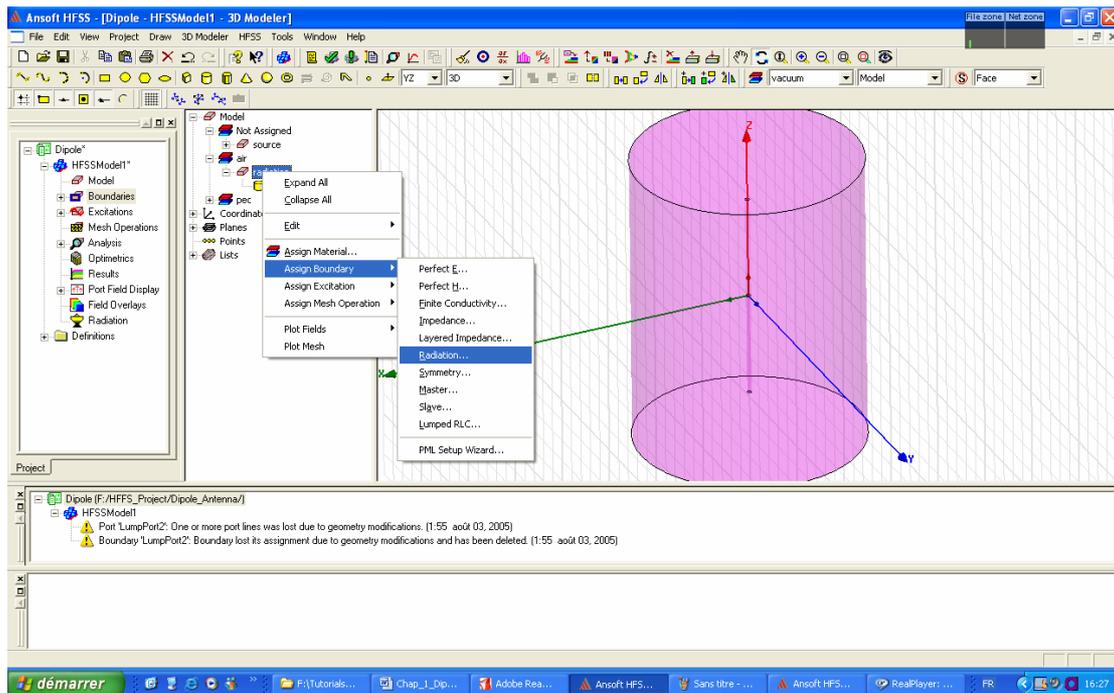
With the geometry complete, the actual radiation boundary may now be assigned. From the 3D toolbar select face from the drop down window as shown below.



Click  and select all faces as follow:



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

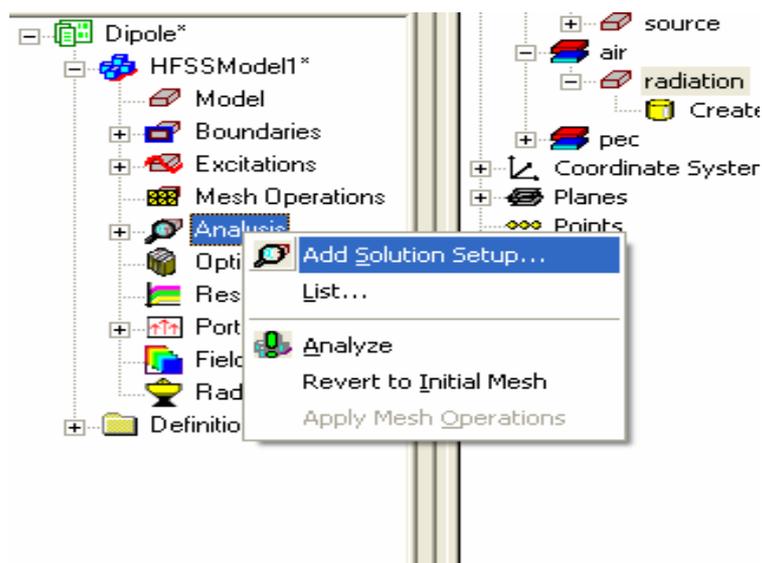


Leave the default name **Rad1** and click OK.

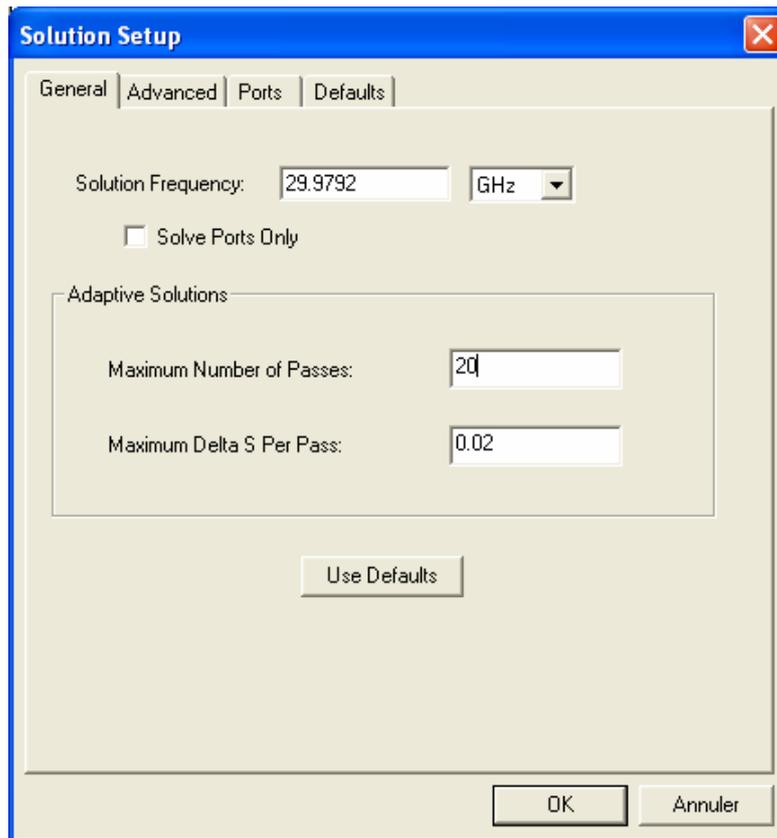
7- Solution Setup

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's ability to calculate general antenna parameters such as directivity, radiation resistance, radiation efficiency, etc... .

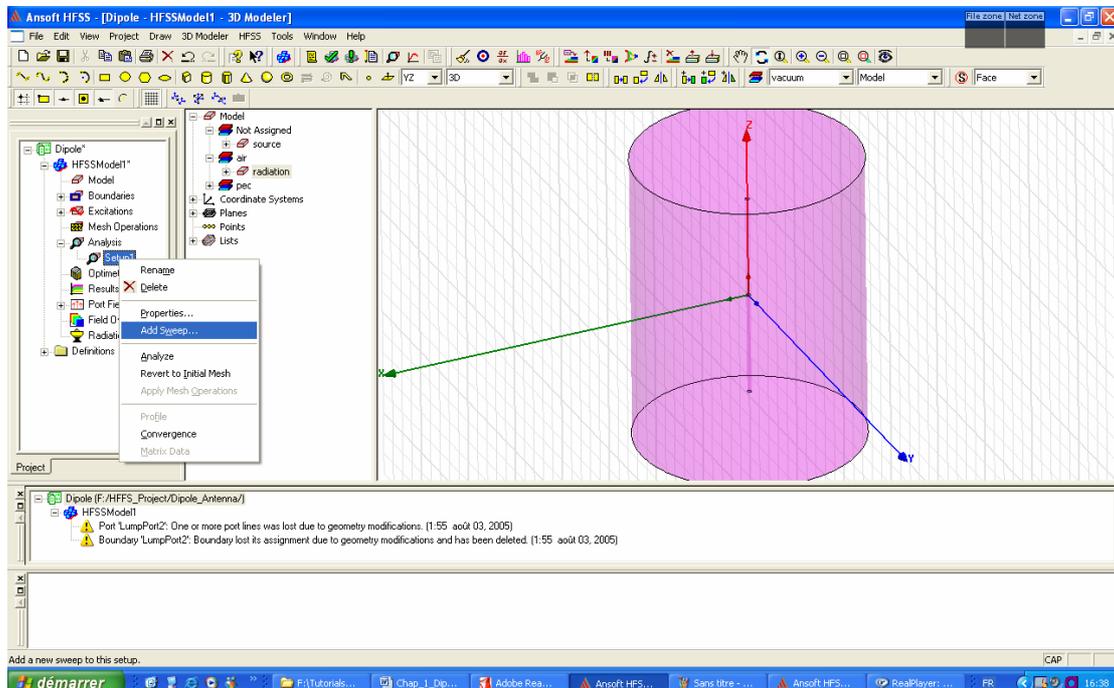
From the project explorer, select Analysis -> Add Solution Setup.



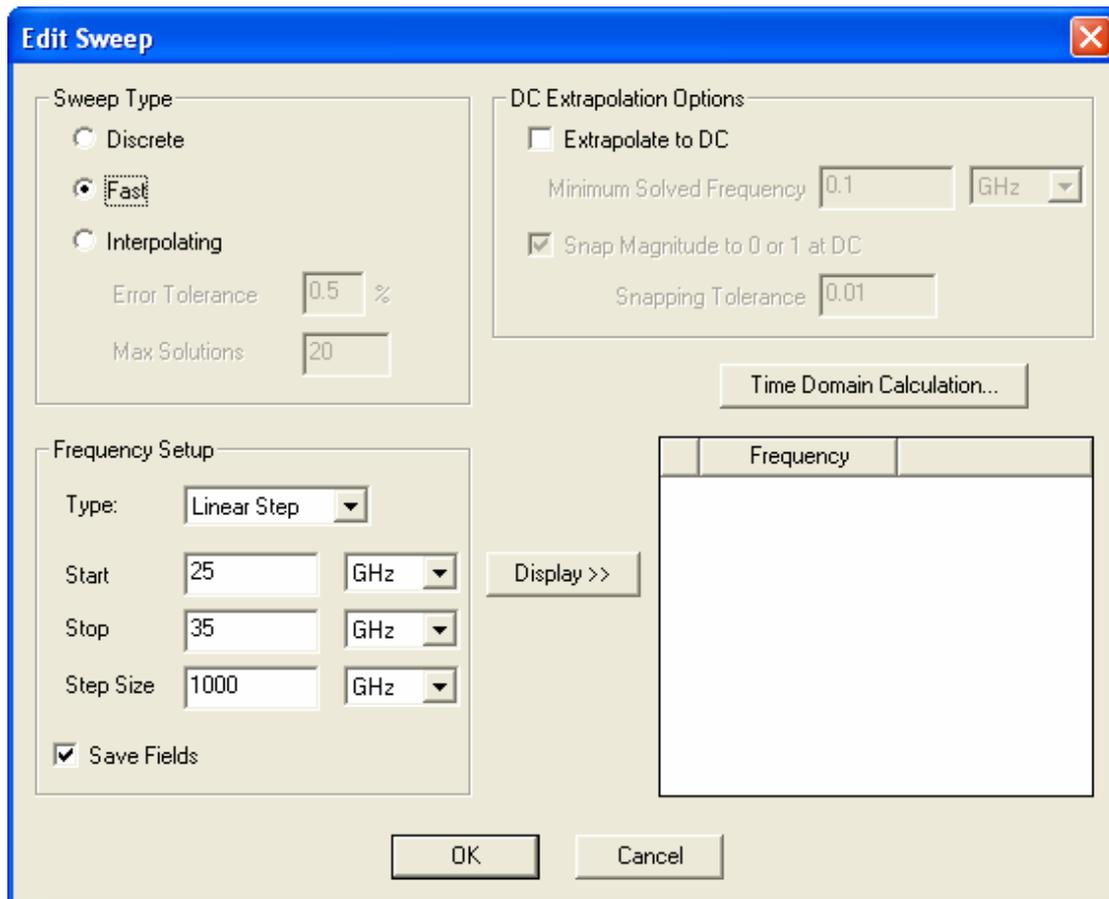
Enter the following. Click ok when complete.



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

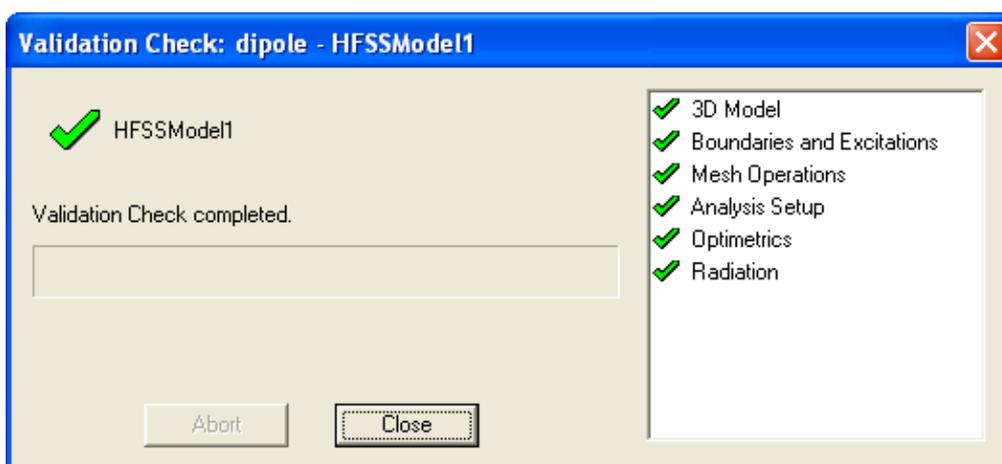


Enter the following



8- Structure Analysis

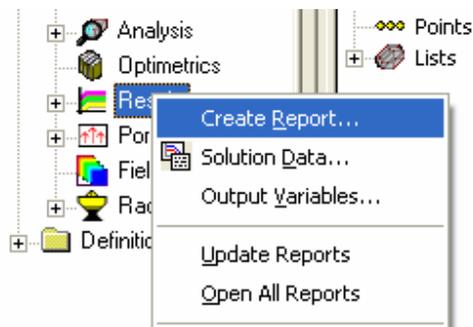
At this point, the user should be ready to analyze the structure. Before running the analysis, always verify the project by selecting  from the 3D toolbar. If everything is correct the user should see:



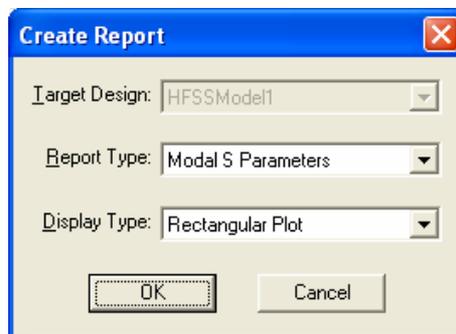
Analyze the structure by clicking 

9- Create Reports

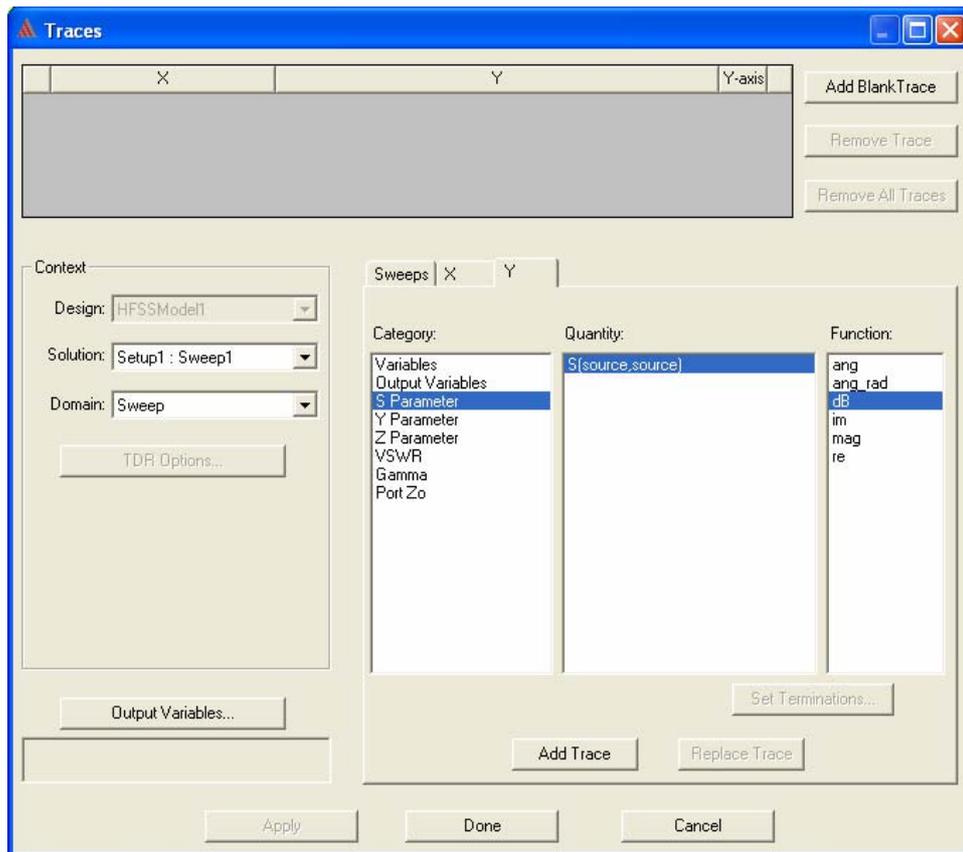
After completion of the analysis, we will create a report to display both the resonant frequency and also the radiation pattern. Click on the heading HFSS and select **Results -> Create Reports**.



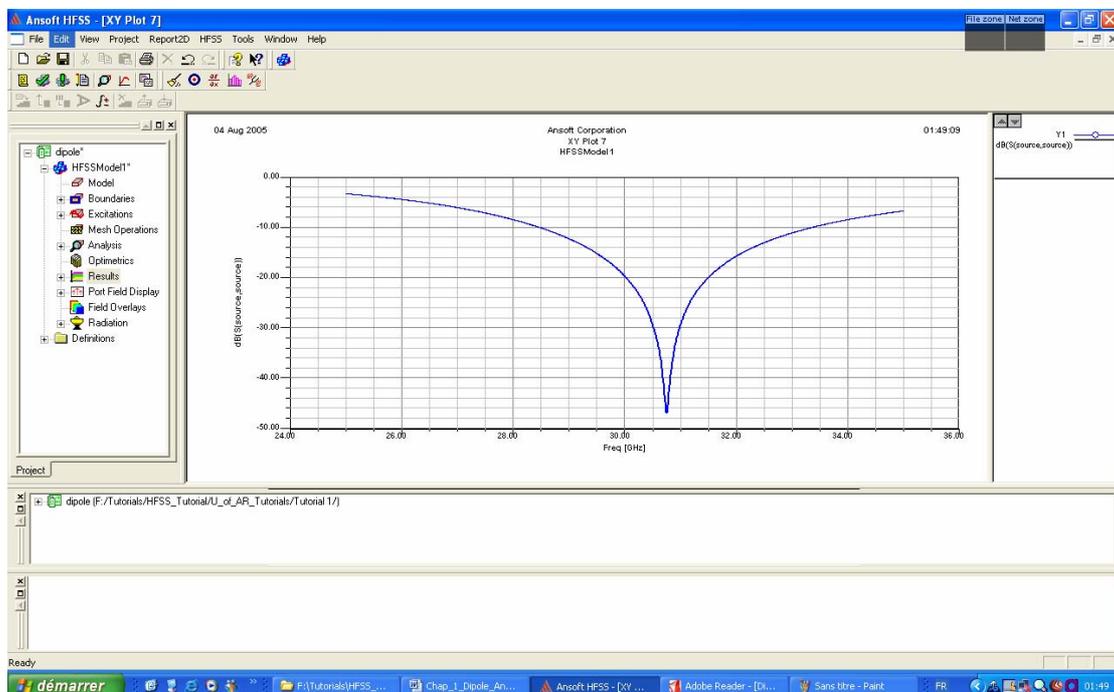
Choose the following in the Create Report window:



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



Click Done when complete.
The graph is displayed below:

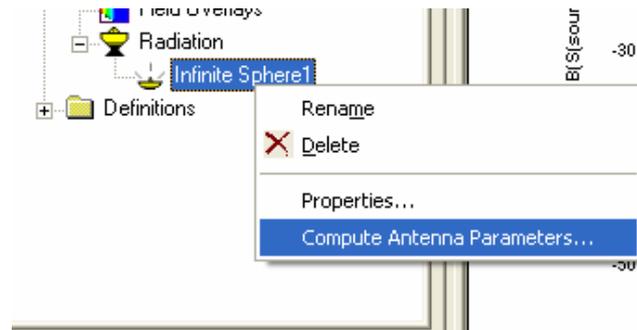


HFSS has the ability to compute antenna parameters automatically. In order to produce the calculations, the user must define an infinite sphere for far field calculations. Right-click the **Radiation** icon in the project manager window and

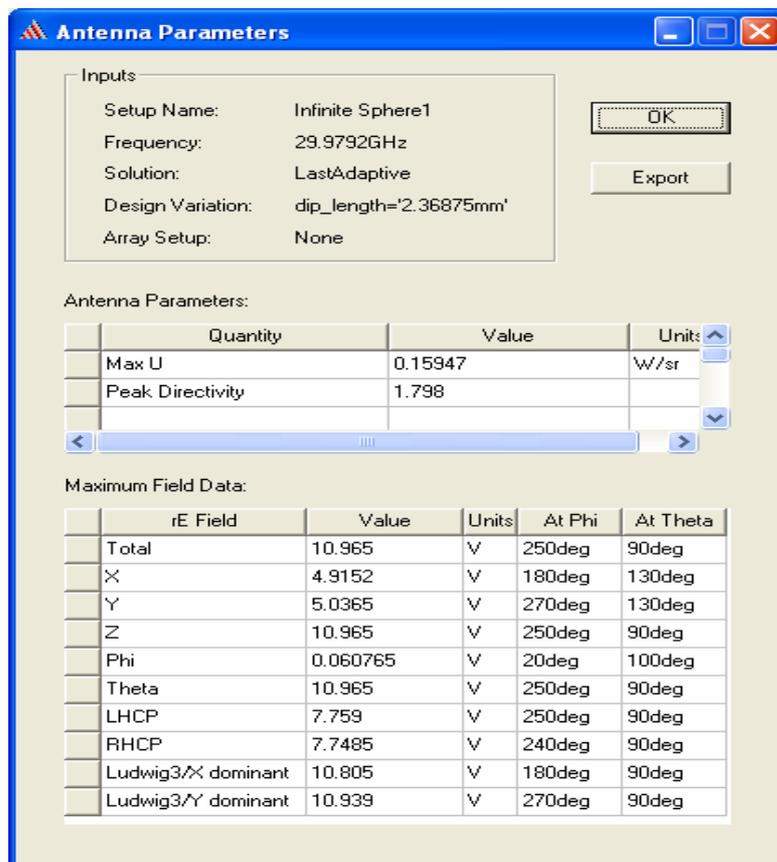
select **Insert Far Field Setup** -> **Infinite Sphere**.



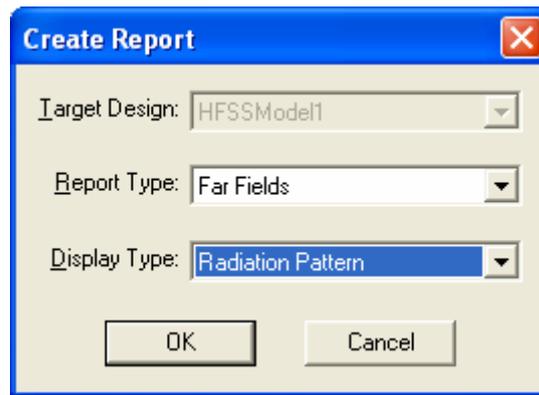
Accept all default parameters and click Done. Right-click Infinite Sphere1 -> Compute Antenna Parameters... from the project explorer as shown:



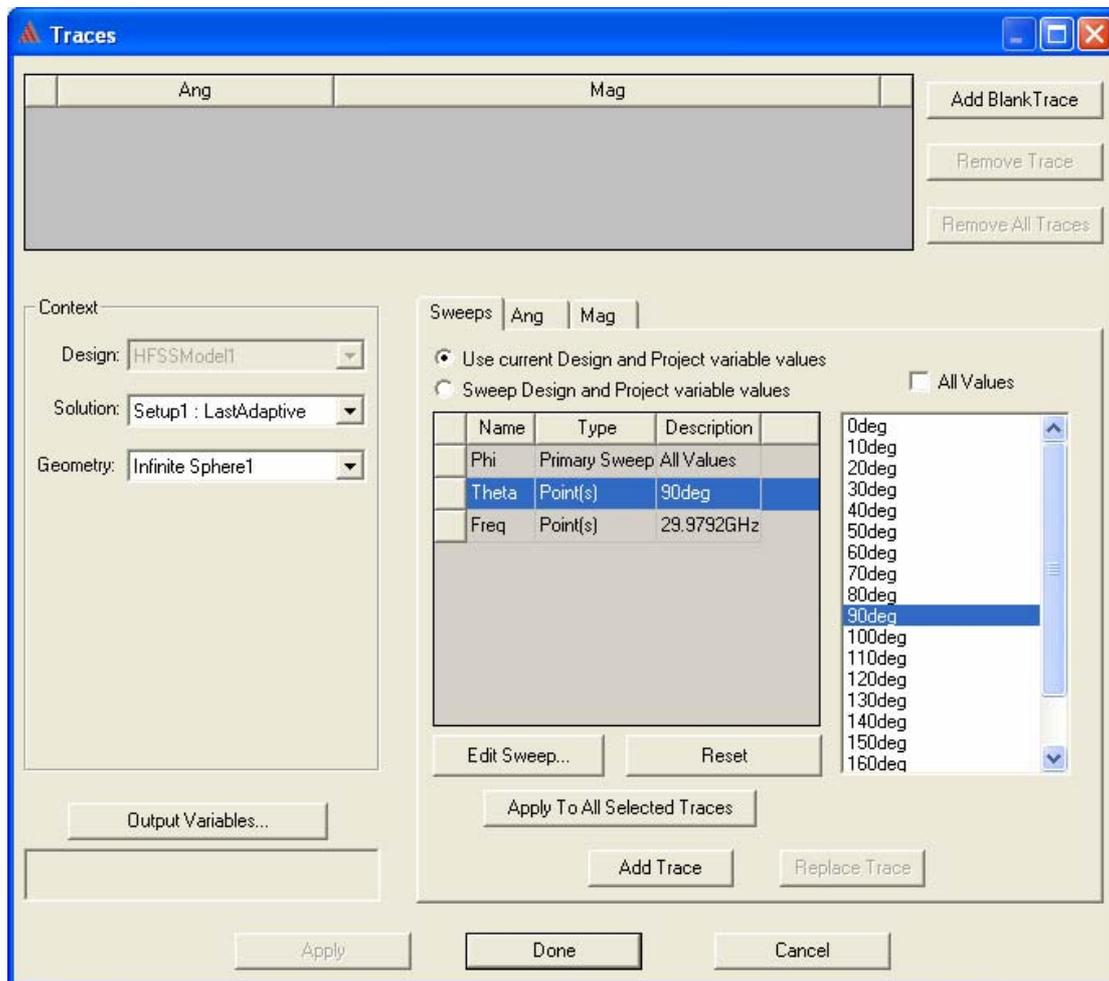
Select all defaults and results are displayed as follows:



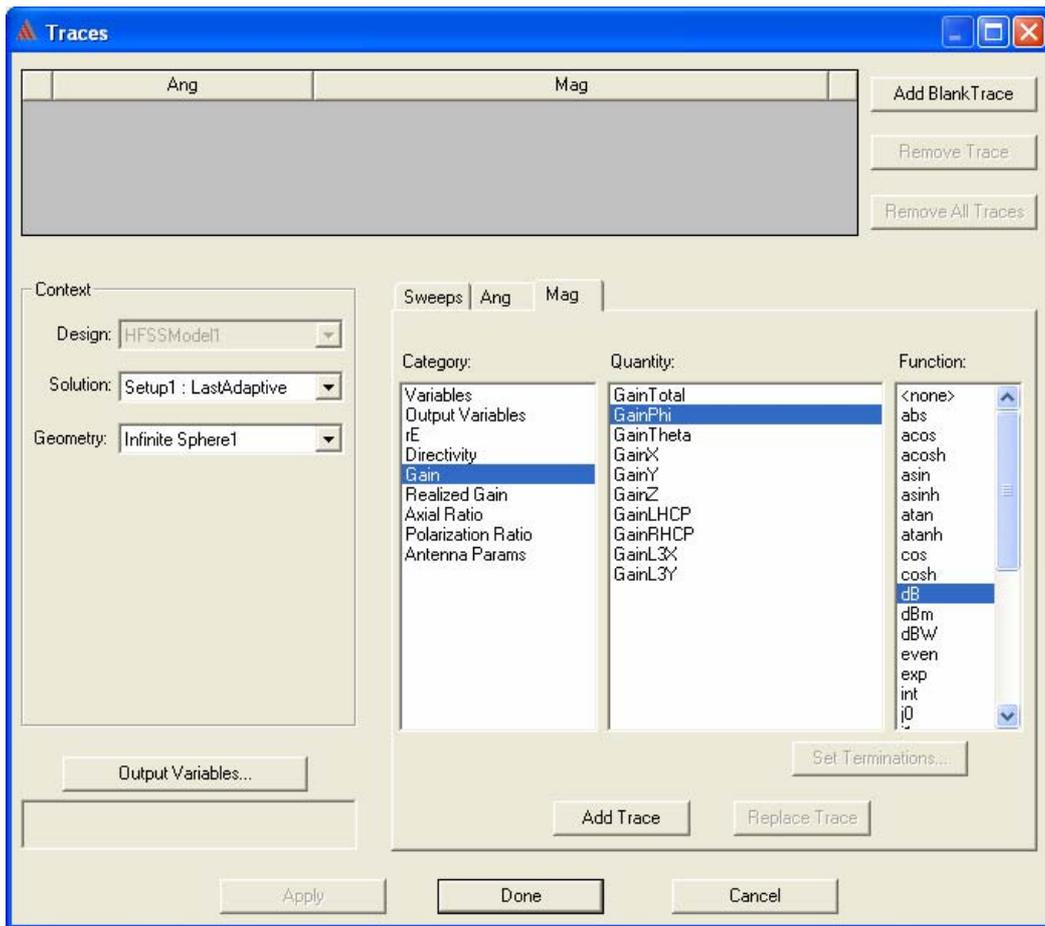
Next, the far field will be plotted. Create Reports as previously shown. Modify the following:



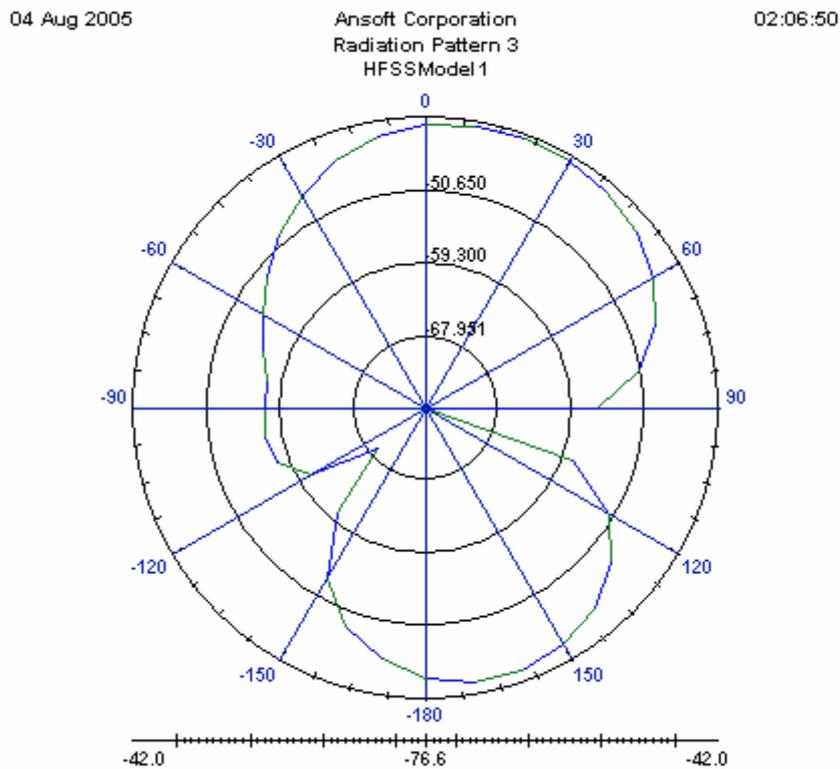
Enter the following:



Select the **Mag** and enter the following:



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



Chapter Two - The Rectangular Patch Antenna

I – Introduction

The objective of this chapter is to show you how to create, simulate and analyze a microstrip patch antenna resonating at a frequency of 7.5 GHz as shown in Fig.1.

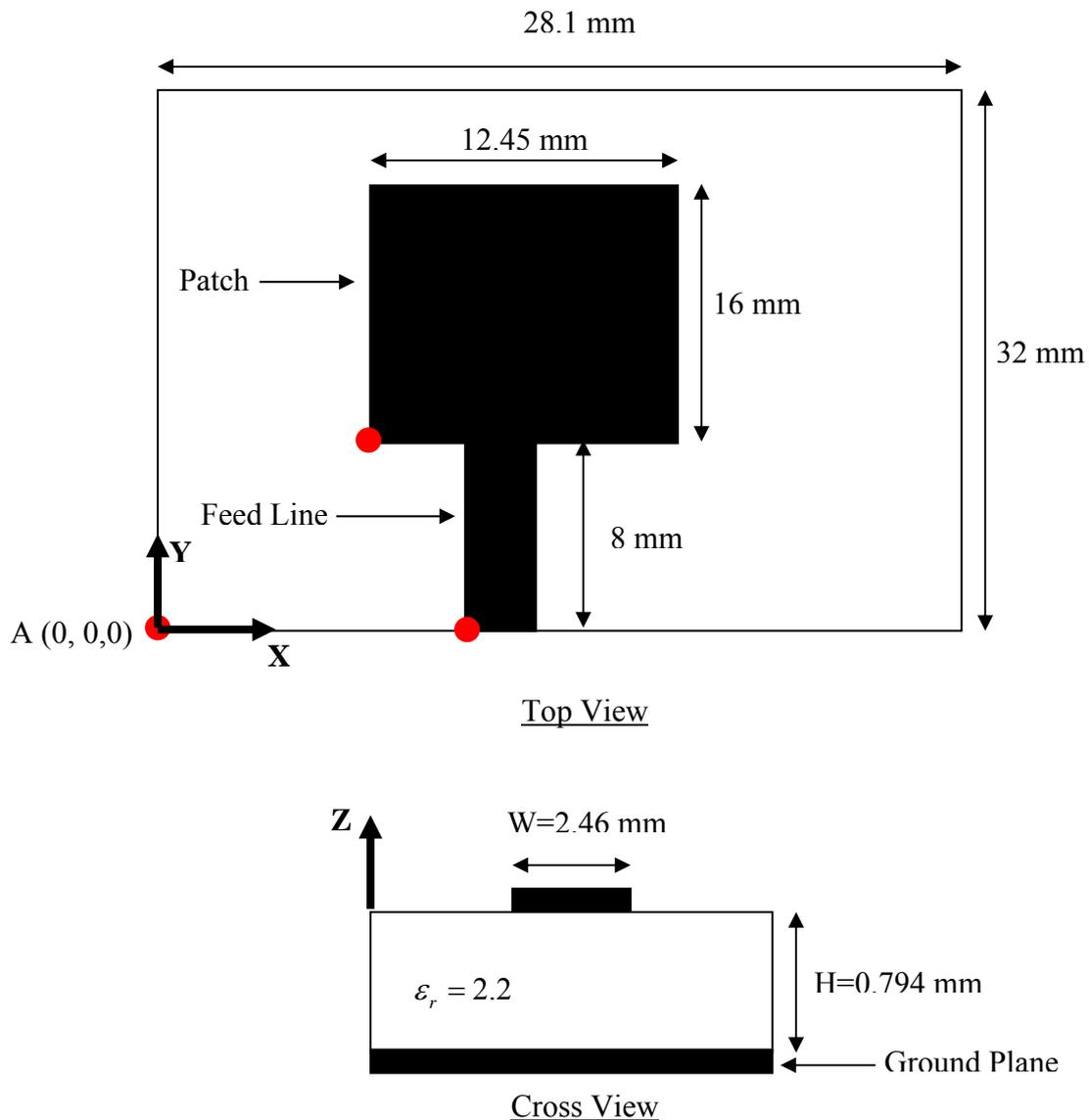


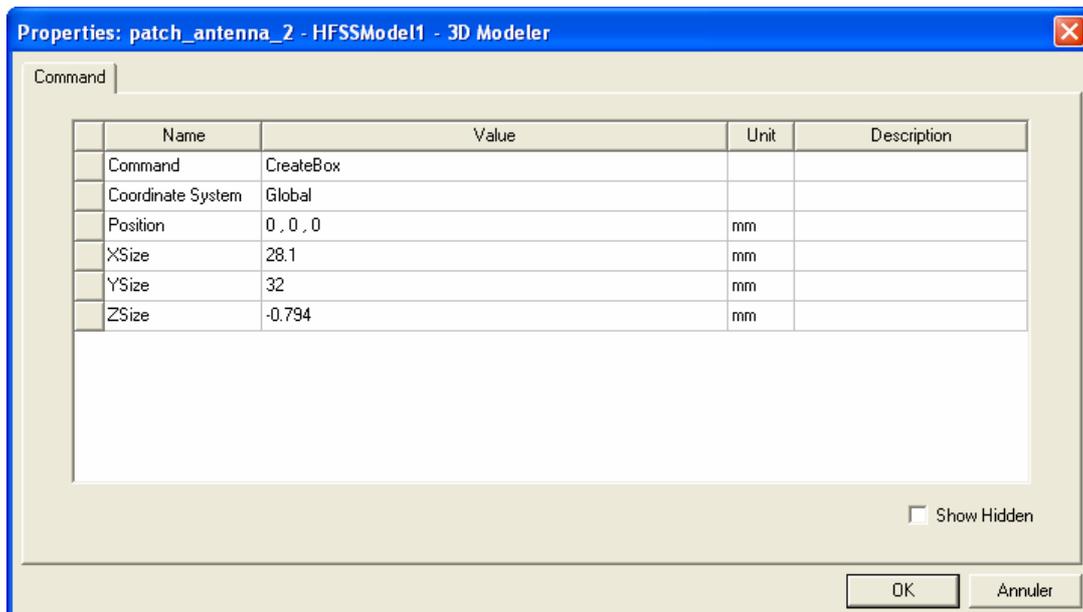
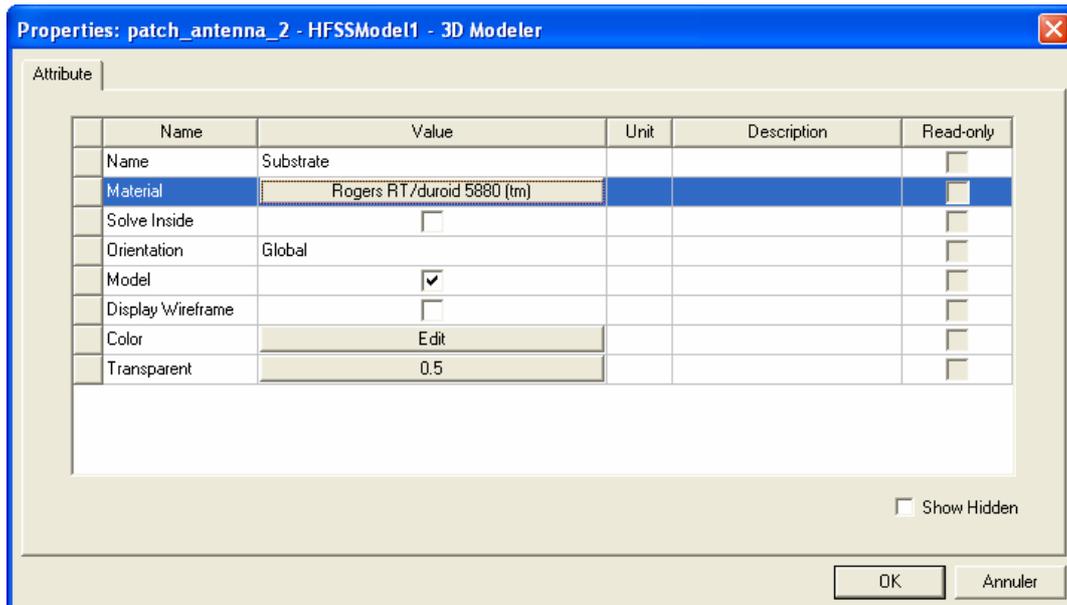
Fig.1 – Rectangular Patch Antenna

II – Creating the Rectangular Patch

According to the first chapter, you can create and save a project.

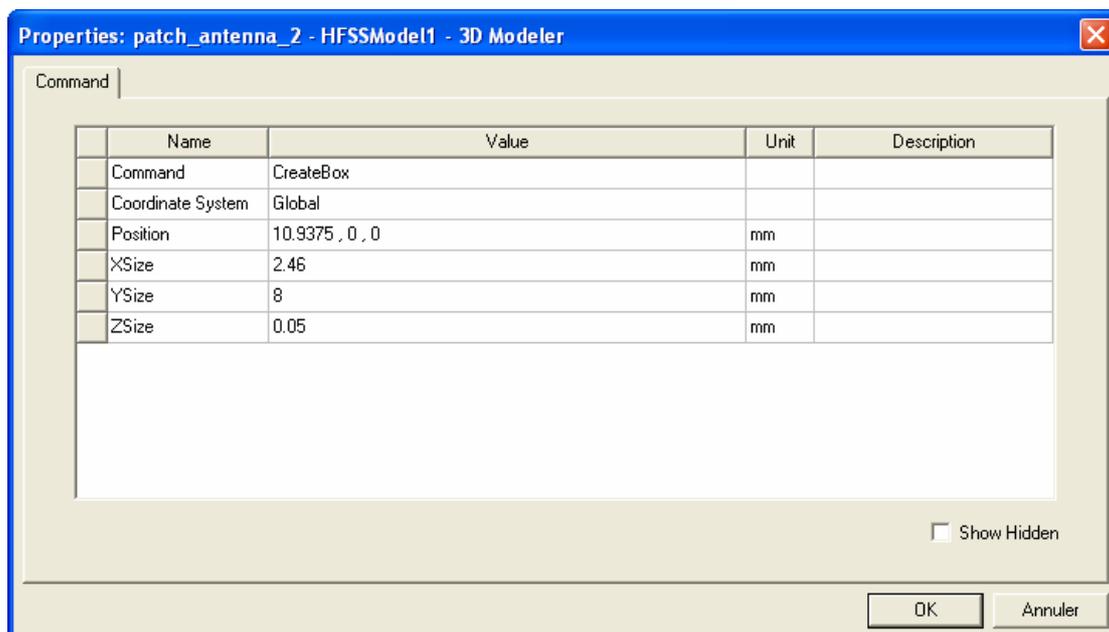
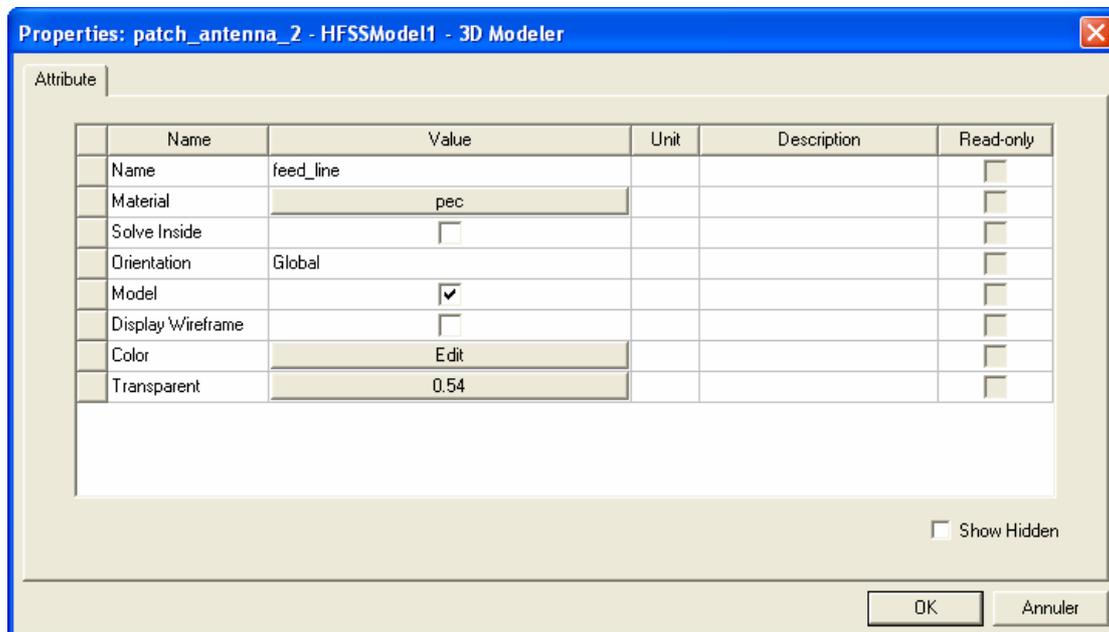
1 – Substrate

To draw the Substrate, click  on the toolbar. Then draw a box by filling the following data as shown below.



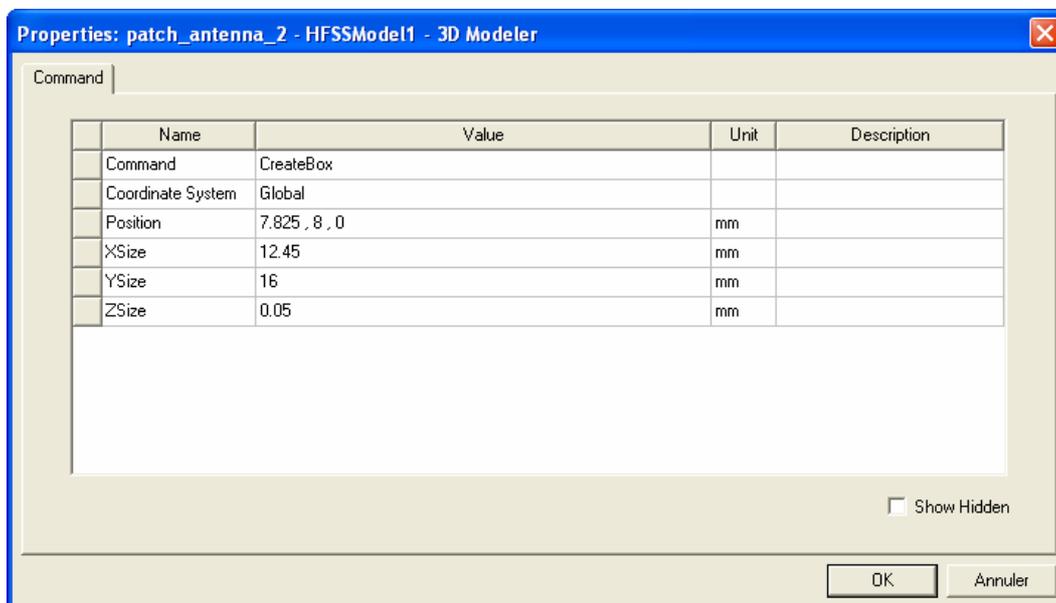
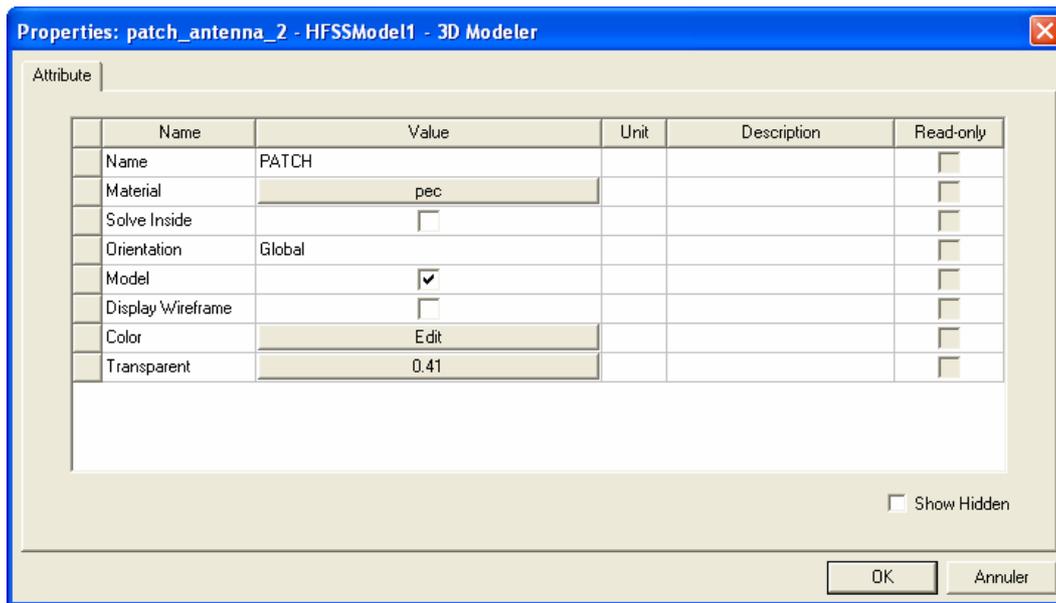
2 – Feed Line

To draw the Feed Line, click  on the toolbar. Then draw a box by filling the following data as shown below.

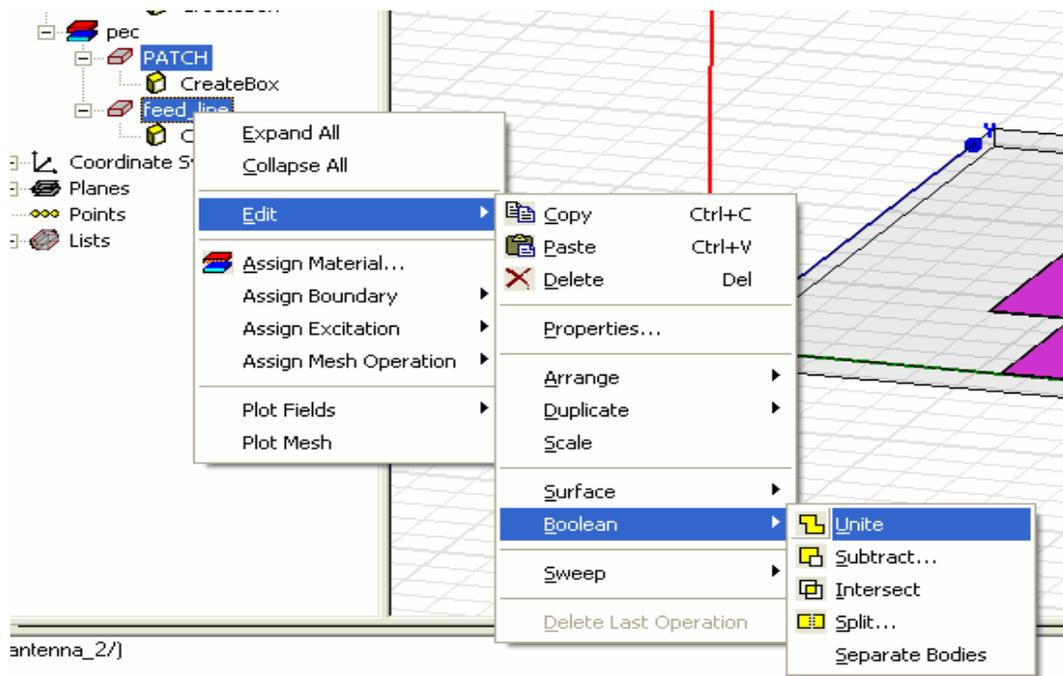


3 – Patch

To draw the Patch , click  on the toolbar. Then draw a box by filling the following data as shown below.

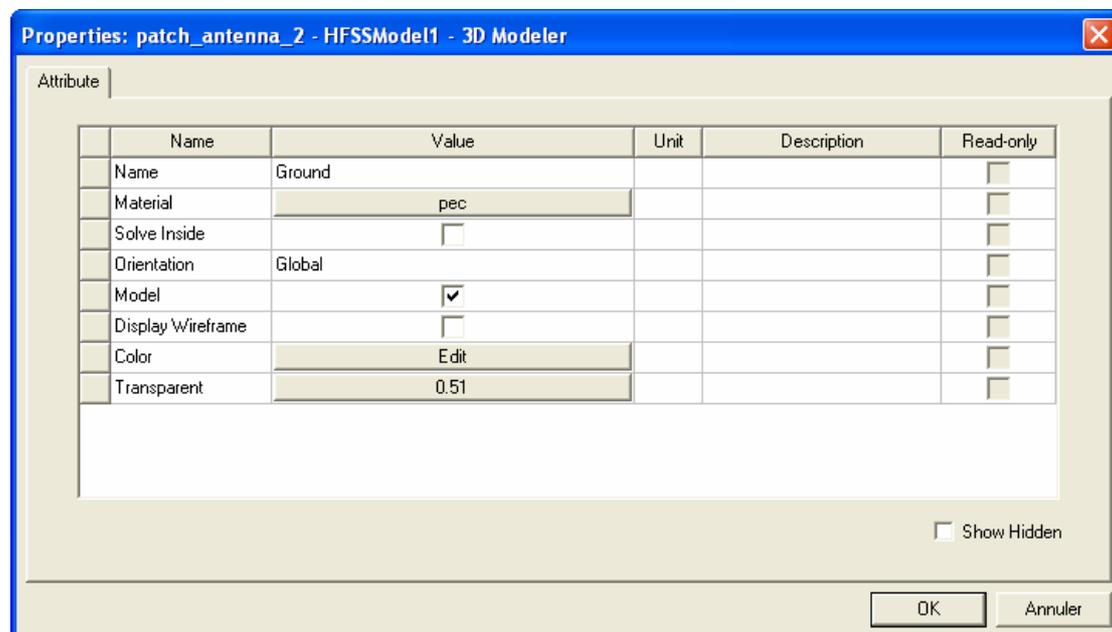


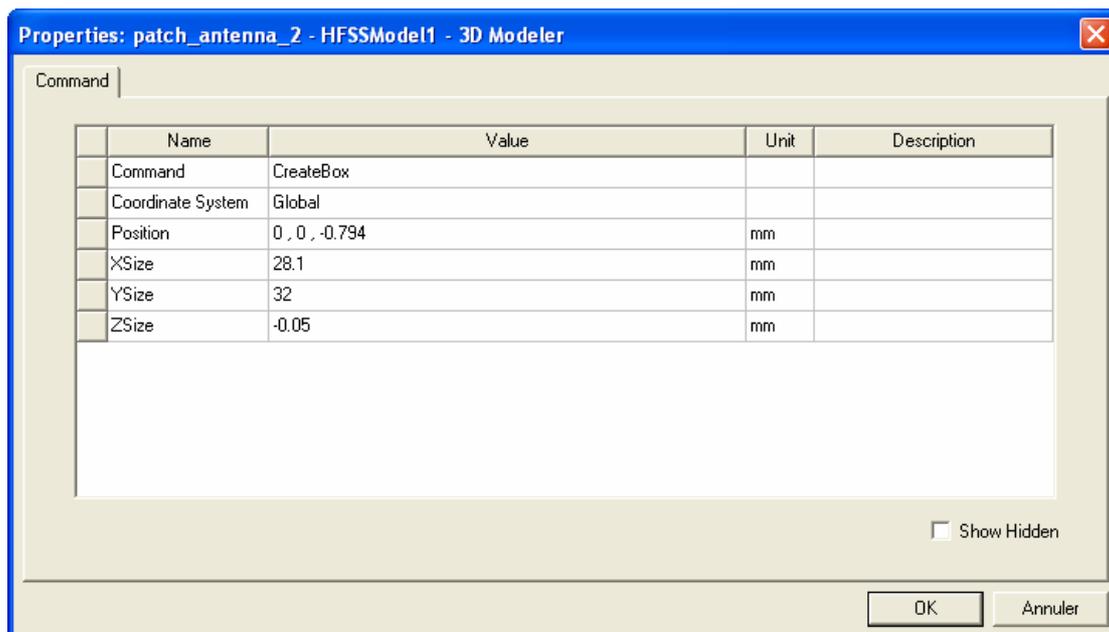
We know that the Patch and Feed line should be one object. So, we need to unite them. Note that both objects are of the same material. Click on both objects that you need to unite, i.e. Patch and Feed_line in the history tree. Click on one and hold the CTRL key and click on the other. Right Click **Edit** > **Boolean** > **Unite**. The two objects are united now.



4 – Ground Plane

To draw the Ground Plane, click  on the toolbar. Then draw a box by filling the following data as shown below.

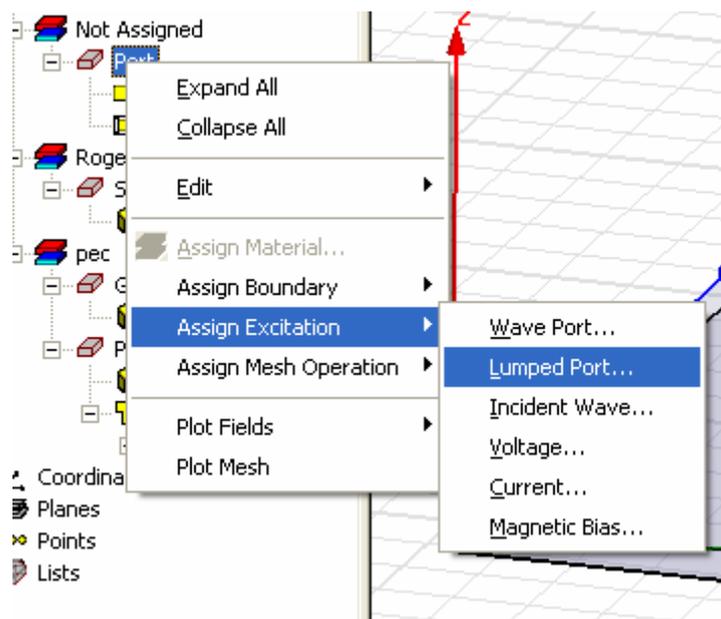




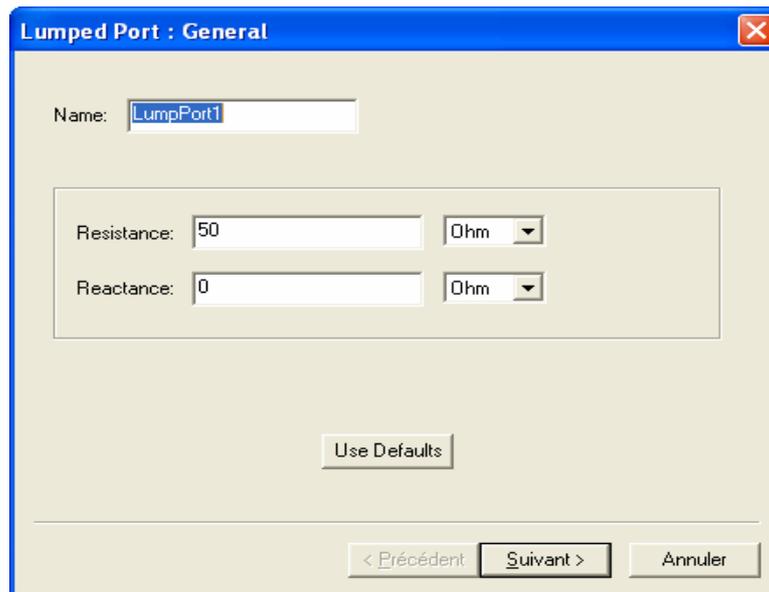
5 – Assign Excitation

The excitation is a waveguide port at the beginning of the microstrip line. The reference plane of this port is located directly at the beginning of the radiating plane. Antennas are excited through the port. To draw the Port, click  on the toolbar. Then draw a rectangle by filling the following data as shown below.

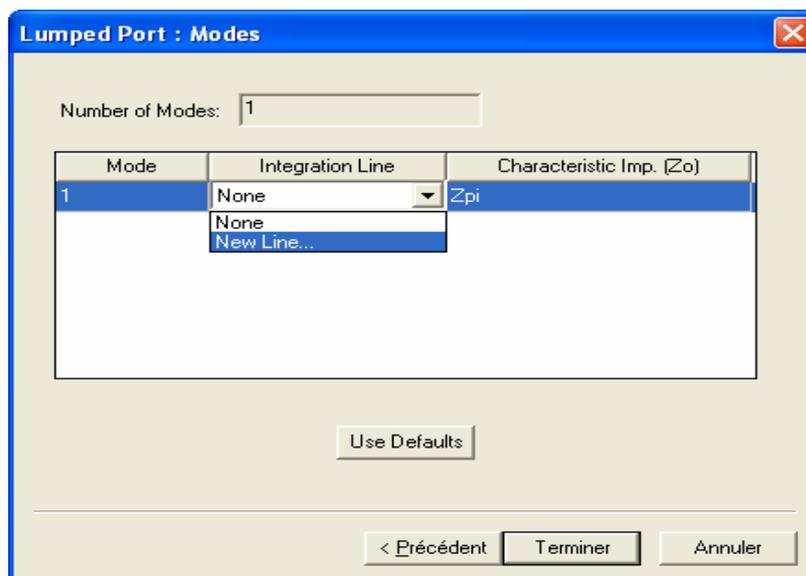
Choose the object Port from history tree, right-click and assign excitation. In our case, it is waveport. Click waveport, name it as your preference, then click Next, now define your integration line. Normally, integration line is defined from the bottom middle point to the upper middle point. Keep other values as default. Click Finish.



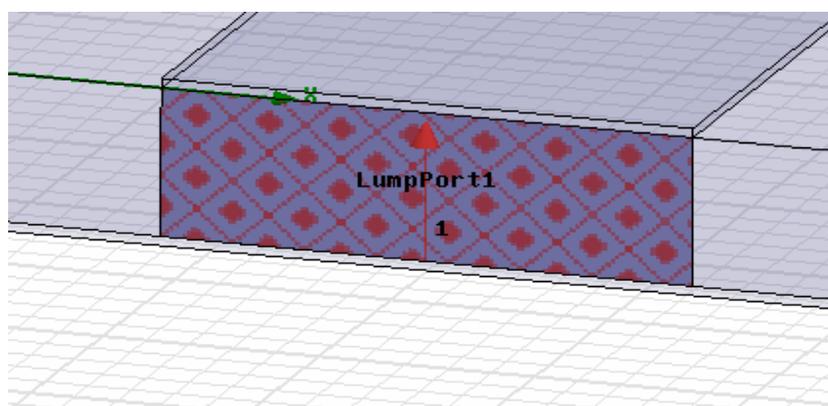
A pop up will come up



Then click 'suivant' and choose new line



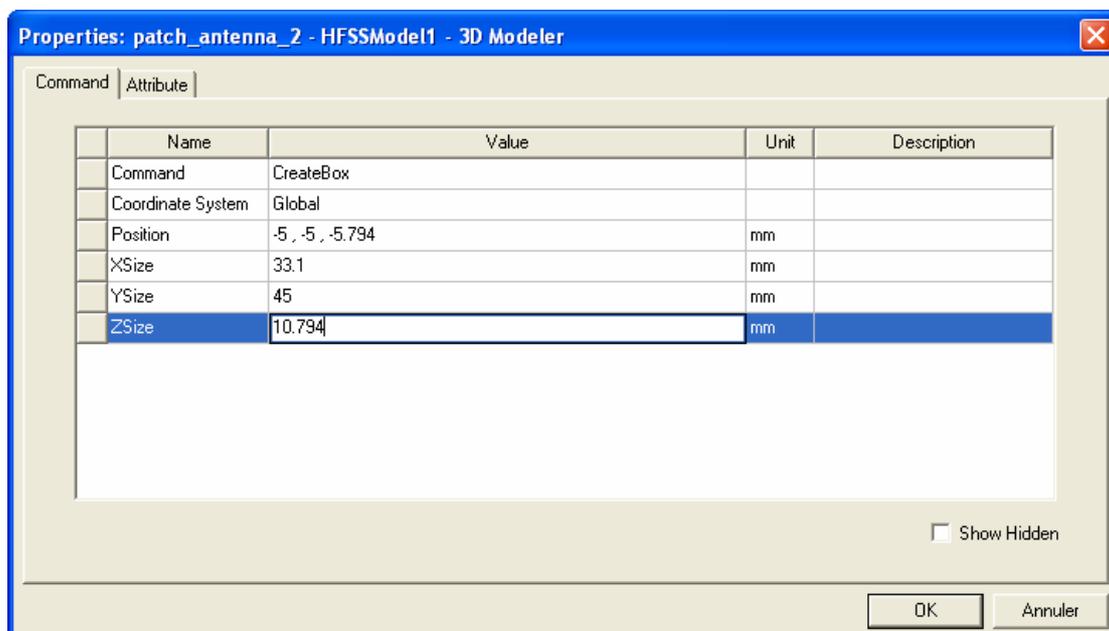
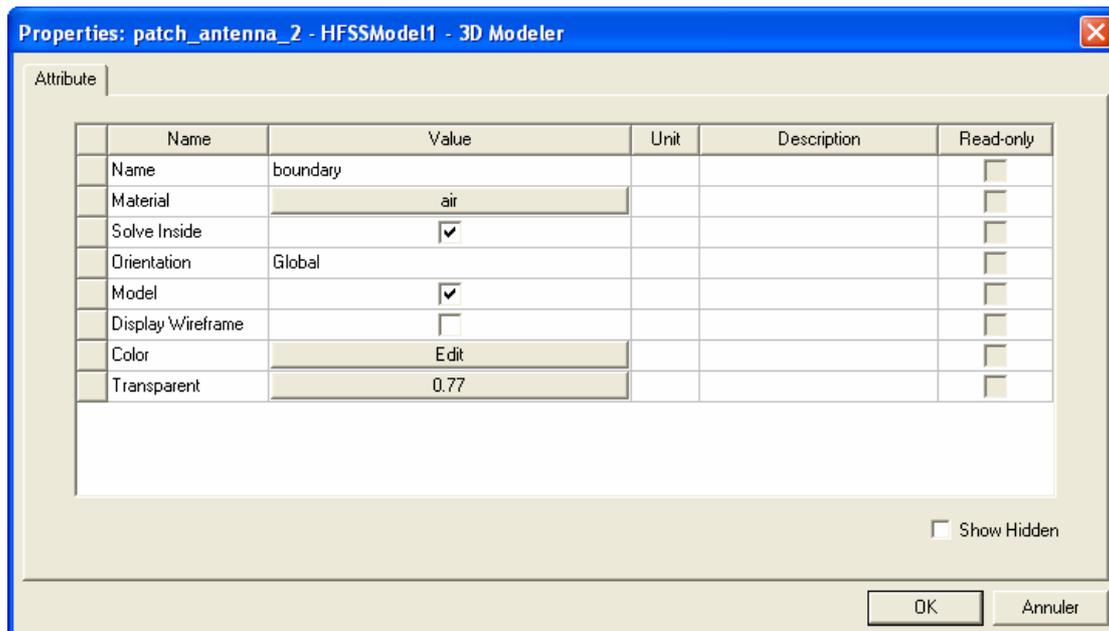
Draw the lumped port,



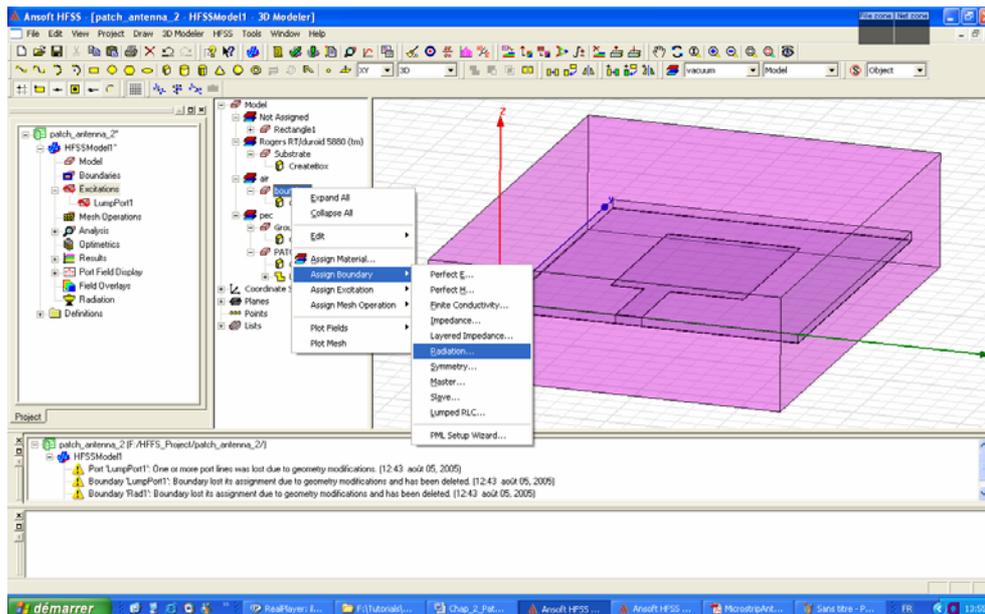
6 – Assign Boundary

Now the model has been created, we need to assign boundary conditions. In HFSS, radiation boundaries are used to simulate open problems that allow waves to radiate infinitely far into space. HFSS absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure. In our case, our ABC (Absorbing Boundary condition) is an air box.

To draw the Air Box, click  on the toolbar. Then draw a box by filling the following data as shown below.



Now select boundary, right click > Assign Boundary > radiation



7 – Analysis Setup

Finally, you have your model ready to run. Now you need to identify your analysis setup.

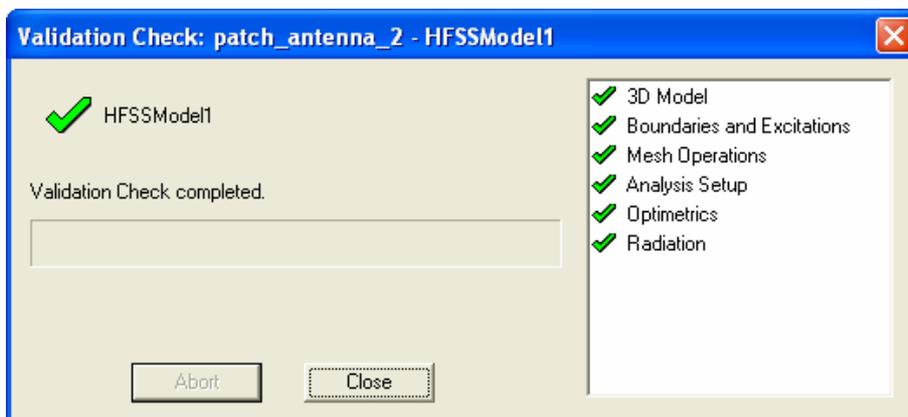
To create an analysis setup, select the menu item **HFSS > Analysis Setup > Add Solution Setup**. In the Solution Setup window, click the general tab, Solution frequency is 7.5 GHz, Maximum Number of Passes is 20 and Maximum Delta S per Pass is 0.02.

8 – ADD Frequency Sweep

To add a frequency sweep, select the menu item **HFSS > Analysis Setup > Add Sweep**. Select Solution Setup: Setup1. Click OK button. Then Edit Sweep Window. Sweep Type: Fast, Frequency Setup Type: Linear Count, Start: 5 GHz, Stop: 10 GHz, Count: 500. Click OK button.

9 – Model Validation

To validate the model, select the menu **HFSS > Validation Check**. Click the Close button. To view any errors or warnings messages, use the Message Manager.



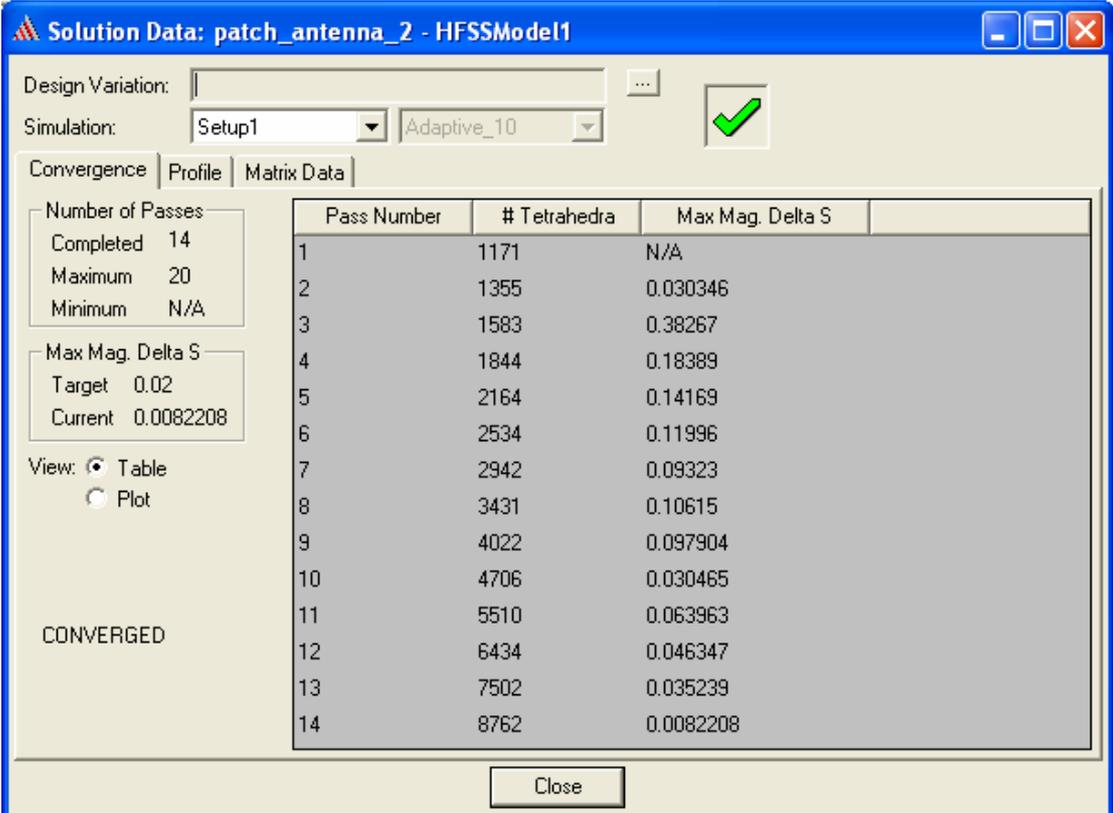
10 – Analyze

To start the solution process, select the menu item **HFSS > Analyze**.

Or click on the icon .

11- Solution Data

Note: The Solution Data window can be also displayed by right-click on the Setup1 under analysis on the HFSS design tree. Note also that the default view is Profile. Select the Convergence tab.



Design Variation: ...

Simulation: Setup1 Adaptive_10 

Convergence Profile Matrix Data

Number of Passes
Completed 14
Maximum 20
Minimum N/A

Max Mag. Delta S
Target 0.02
Current 0.0082208

View: Table
 Plot

CONVERGED

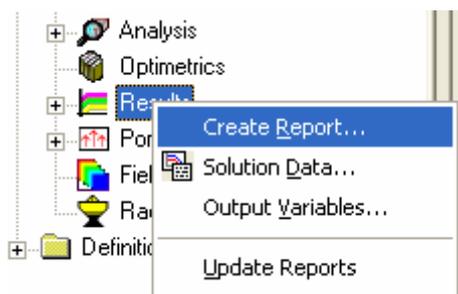
Pass Number	# Tetrahedra	Max Mag. Delta S
1	1171	N/A
2	1355	0.030346
3	1583	0.38267
4	1844	0.18389
5	2164	0.14169
6	2534	0.11996
7	2942	0.09323
8	3431	0.10615
9	4022	0.097904
10	4706	0.030465
11	5510	0.063963
12	6434	0.046347
13	7502	0.035239
14	8762	0.0082208

Close

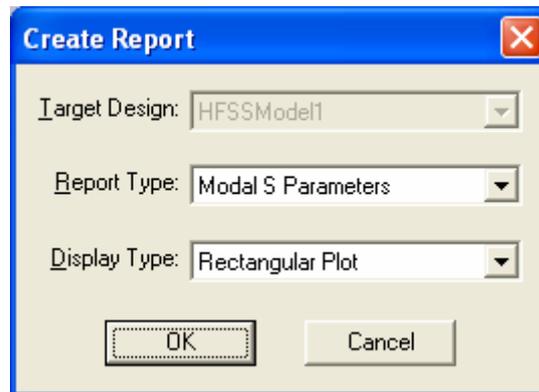
The simulation will stop as soon as the results converge, which is at pass 14.

12- Create Reports

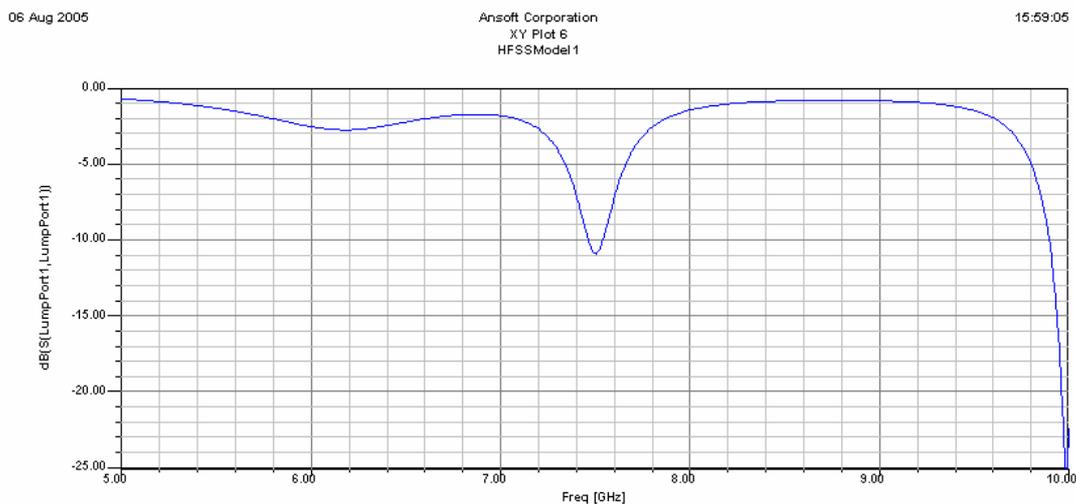
To create a report, select Results > Create Report.



Set Report Type to Modal S Parameters, Display Type to Rectangular then click OK button.



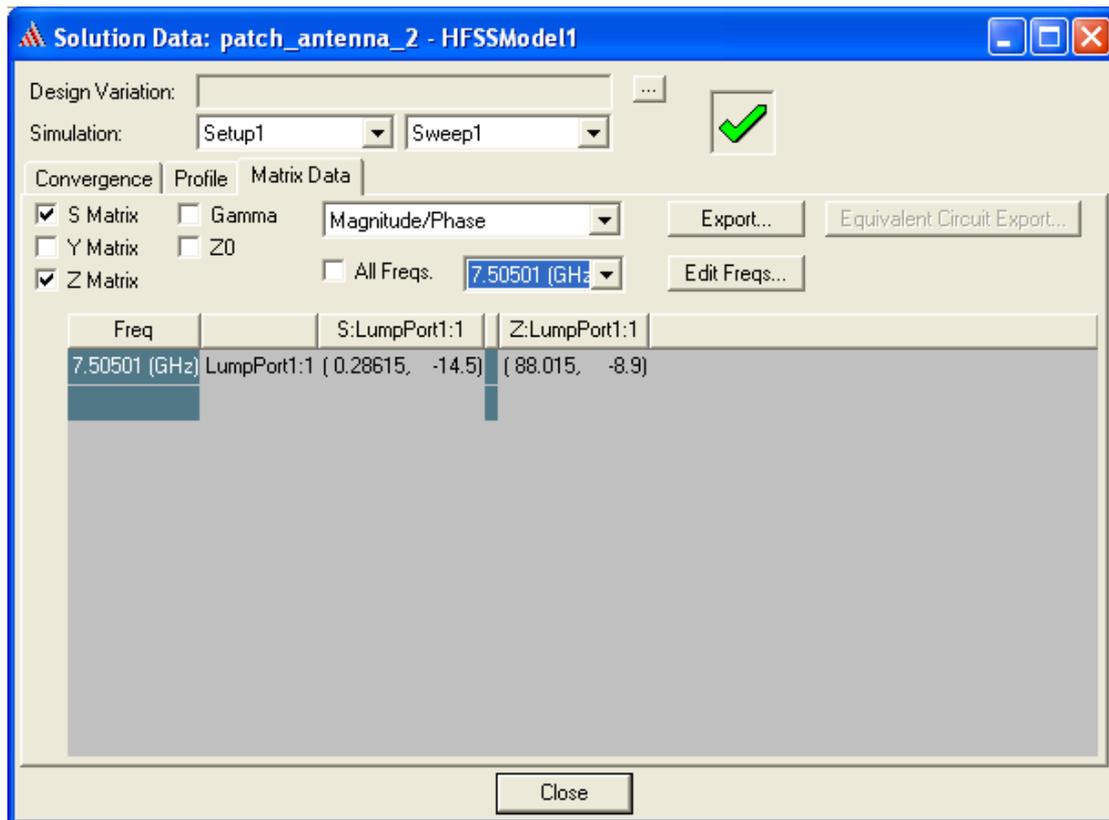
In the Traces Window, set Solution to Setup1: Adaptive1. In the Y tab, set Category to S Parameter, Quantity to S (waveport, waveport), Function to dB and click Add Trace button. Click Done button. Note that you can create any type of report it all depends on what you want to analyze specifically.



The antenna is resonating around 7.5 GHz.

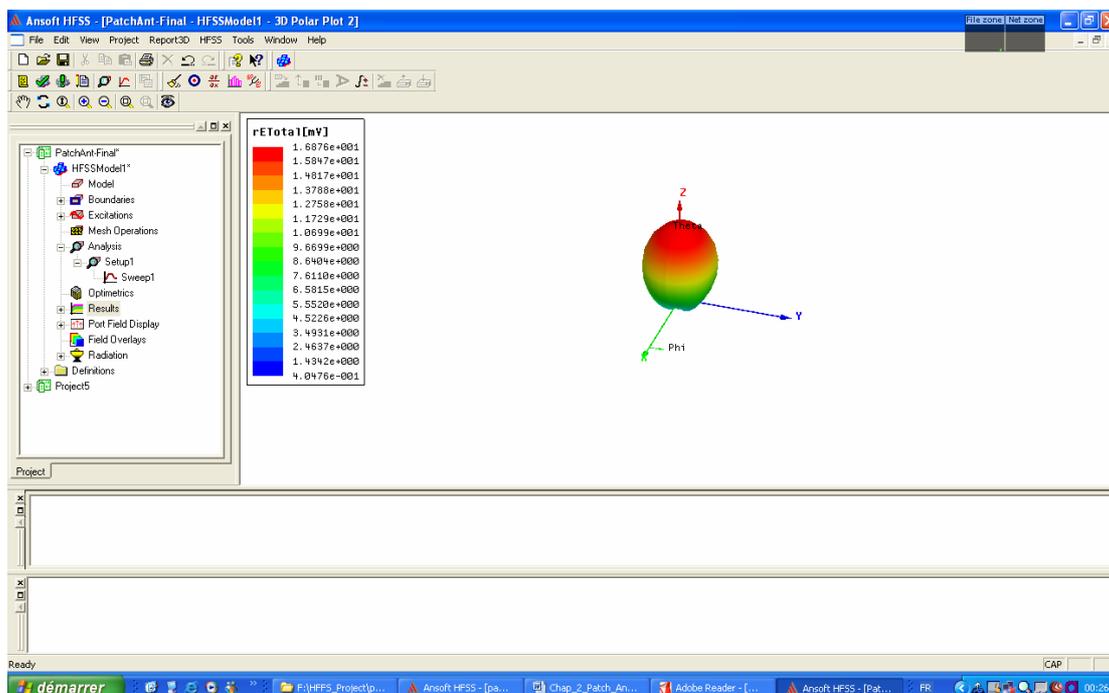
Note: More accurate results could be achieved by zooming in the simulation between 7.00 GHz and 8.00 GHz. (Change the Start and Stop values to 7 GHz and 8 GHz, respectively then run simulation again).

Moreover, we notice that Z_{in} at 7.5 GHz is 88.05Ω . To view Z_{in} , go to Results<Solution Data click on Z Matrix and drag the frequency menu to 7.5 GHz and read the Magnitude of the input impedance.



12- Radiation Pattern

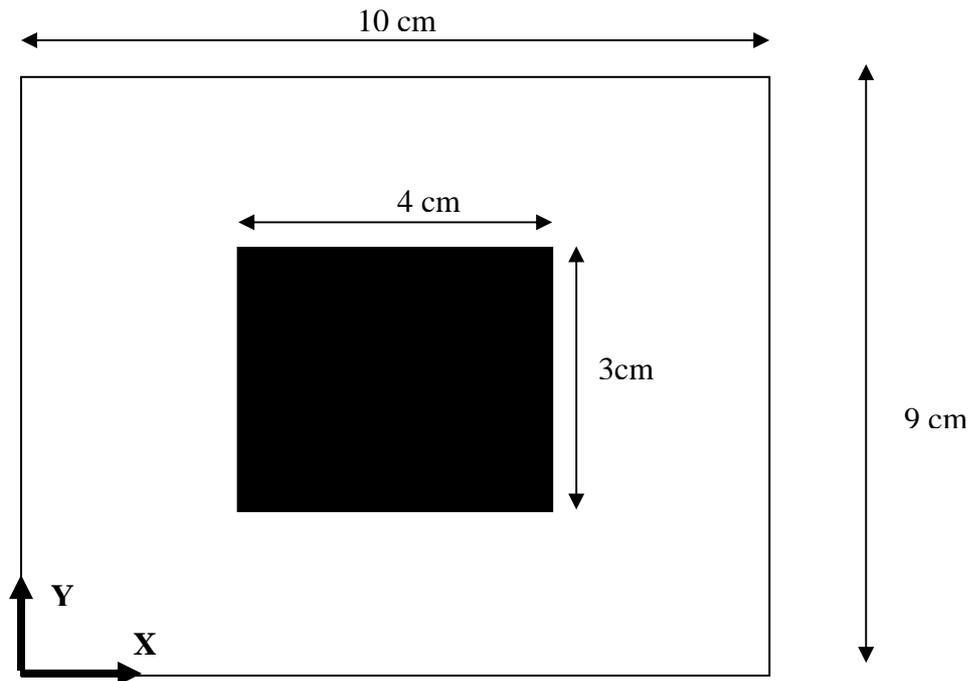
Create infinite sphere. Then go to Results < Create Report. When the new window pops up change the Report Type to **Far Field** and Display type to **3D Polar Plot**. Click Add Trace then Ok.



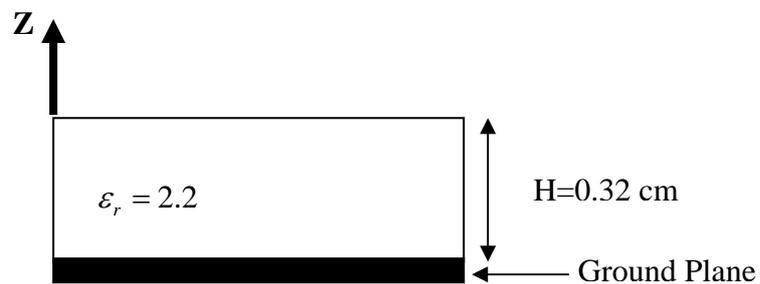
Chapter Three – Probe Feed Patch Antenna

I – Introduction

This third chapter is intended to show you how to create, simulate and analyze a Probe Feed Patch Antenna (Fig.1) using the Ansoft HFSS. The main aim of this chapter is to show how to create a coax cable probe.



Top View



Cross View

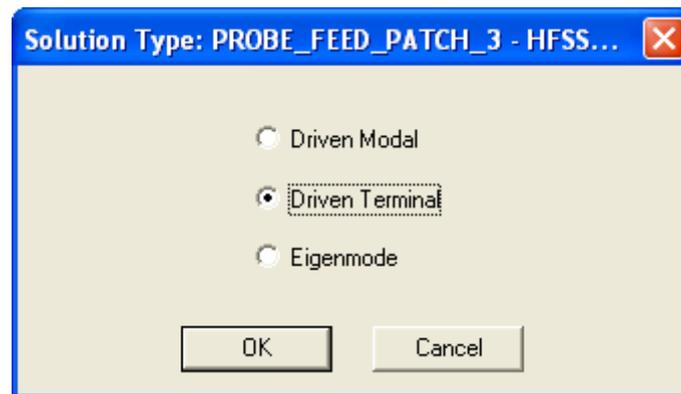
Fig.1 – Probe Feed Patch Antenna

II – Getting Started

By now, you can launch HFSS, opening a project and name it “probe_Feed_Patch_Antenna”.

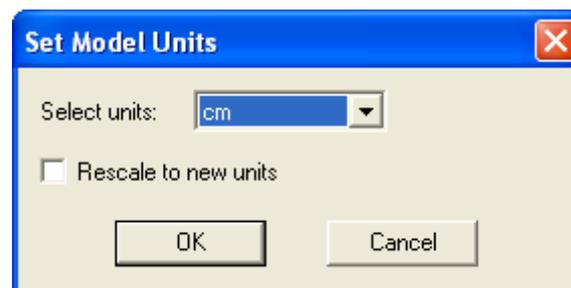
Then set the solution type:

- select the menu item **HFSS > Solution Type**
- choose **Driven Terminal**
- click **Ok** button



To set the units

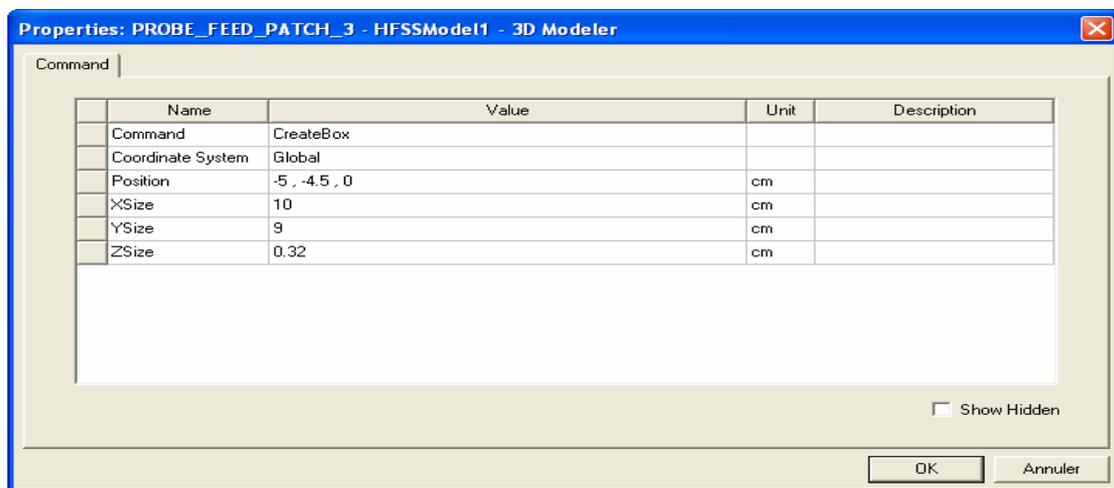
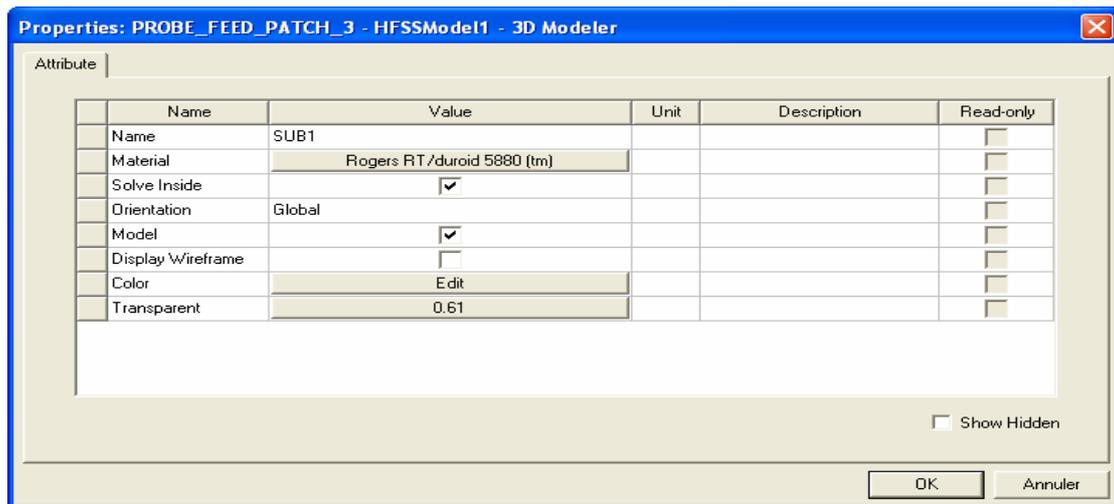
- select the menu item **3D Modeler > Units**
- select Units: **cm**
- click **ok** button



III – Creating the Probe Feed Patch Antenna

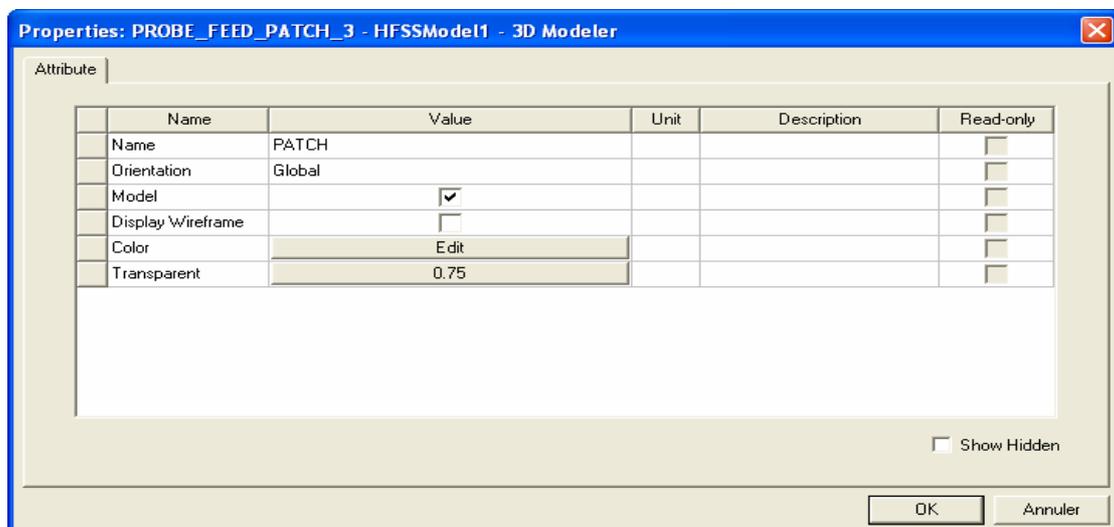
1 – Substrate

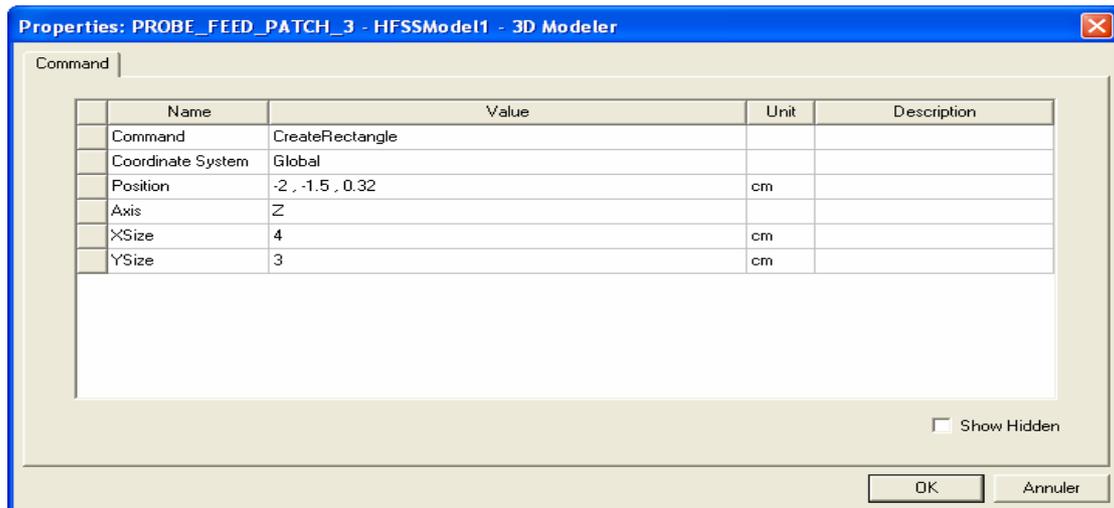
To draw the Substrate, click  on the toolbar. Then draw a box by filling the following data as shown below.



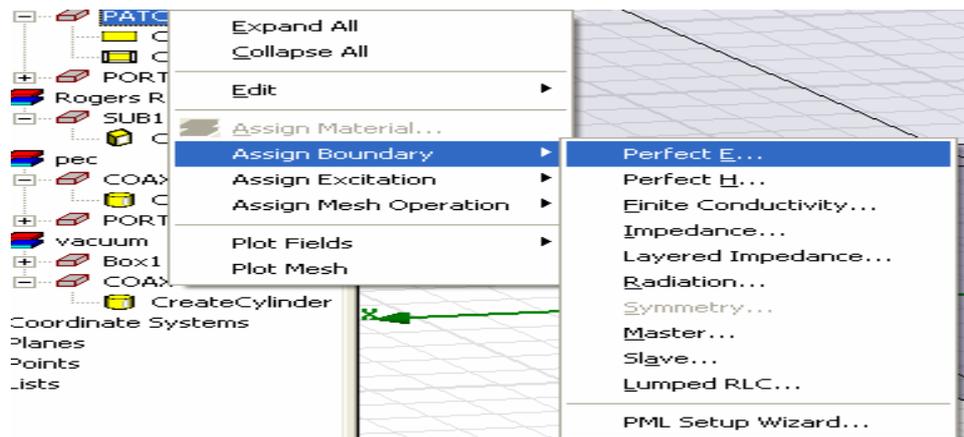
2 – Patch

To draw the Patch , click  on the toolbar. Then draw rectangle by filling the following data as shown below.



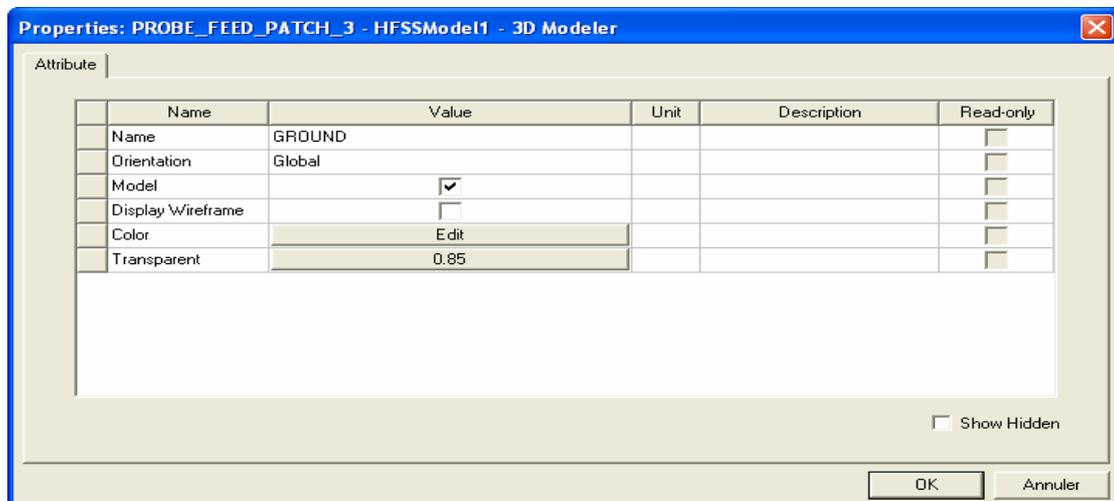


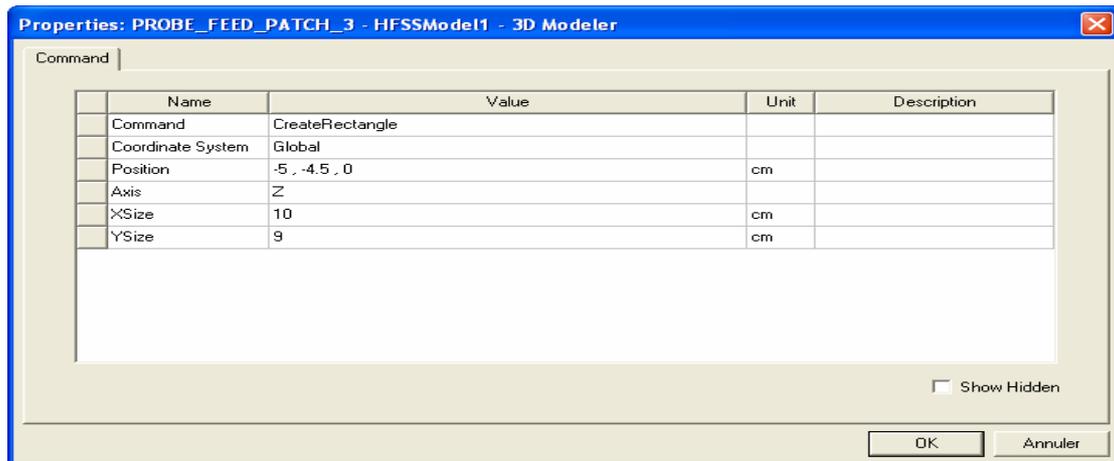
Then assign a perfect E boundary to the patch. Select **Patch**, double click, select **Assign Boundary > Perfect E...**



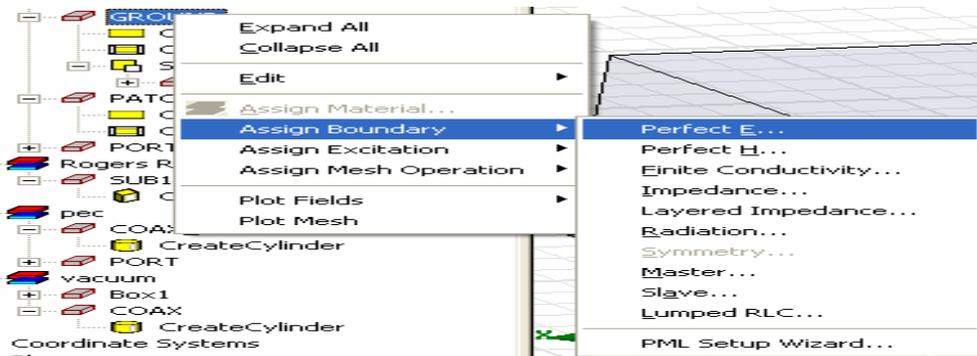
3 – Ground Plane

To draw the Ground Plane, click  on the toolbar. Then draw rectangle by filling the following data as shown below.





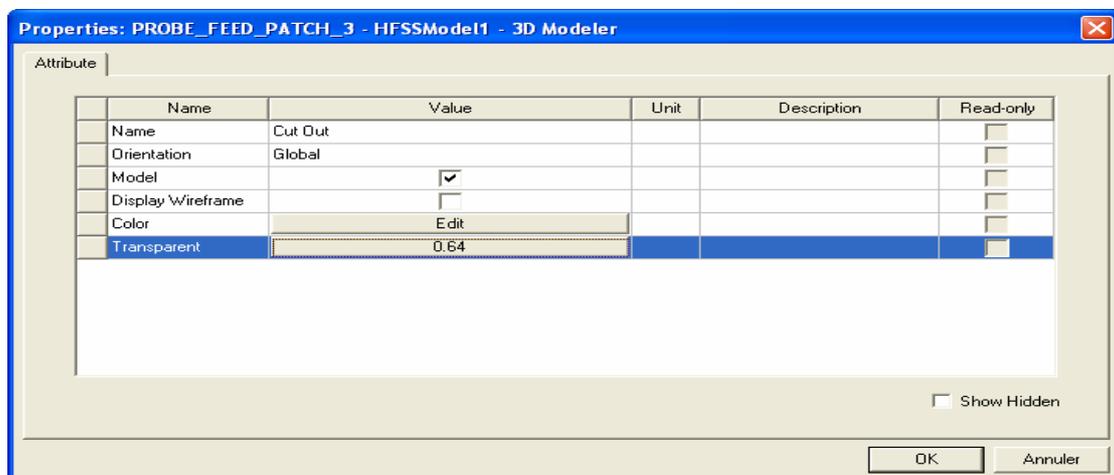
Then assign a perfect E boundary to the patch. Select **Ground**, double click, and select **Assign Boundary > Perfect E...**

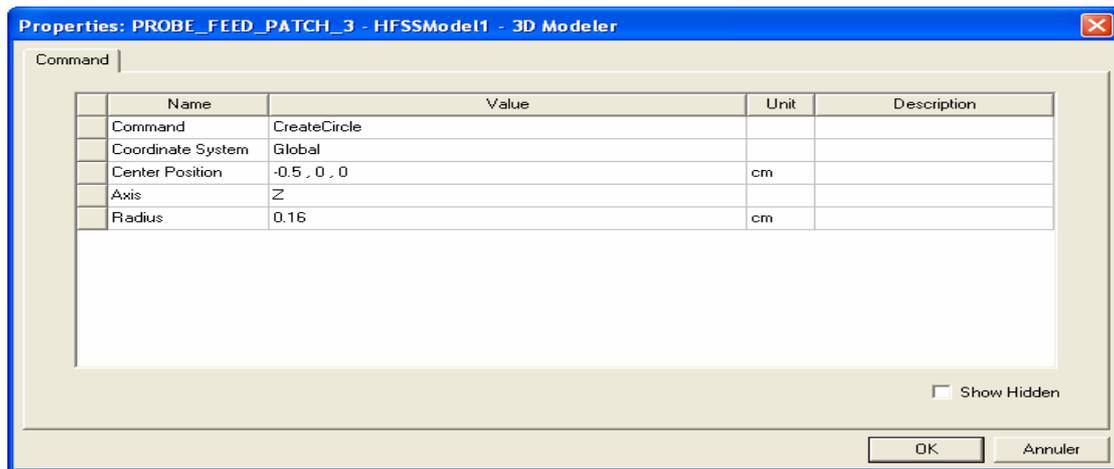


4 – Coax Cable

The antenna is excited using a coax cable port. This port is located under the patch.

- ✓ To draw the coax cable port, we start by drawing the infinite ground Cut Out as shown below.

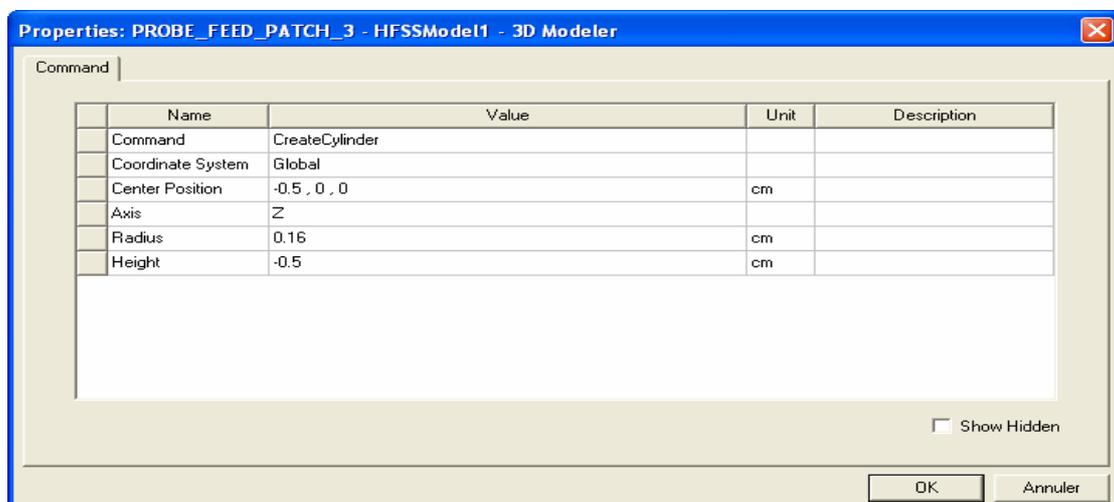
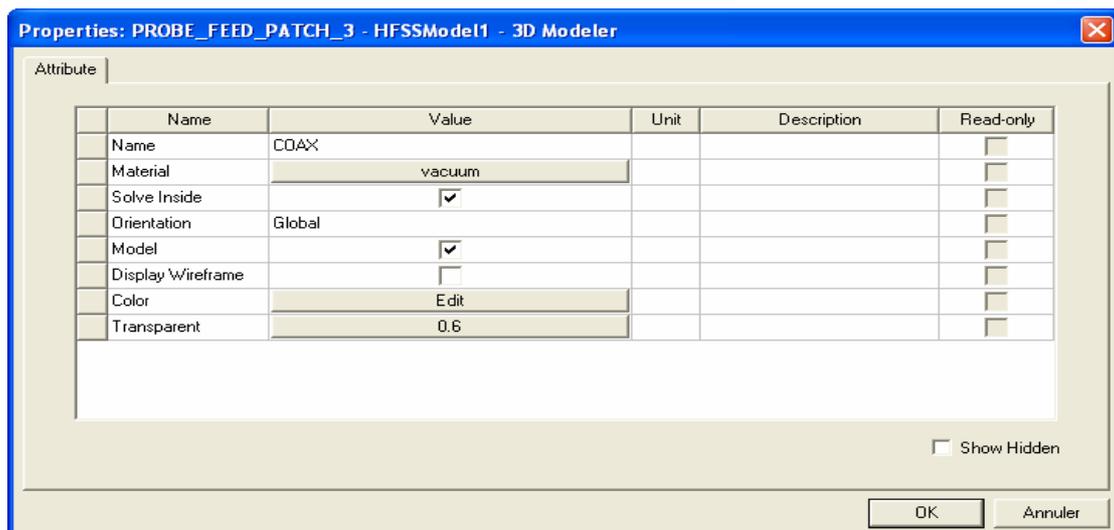




Then select the **Ground & Cut_Out**, right click, select **Edit > Boolean > Subtract**

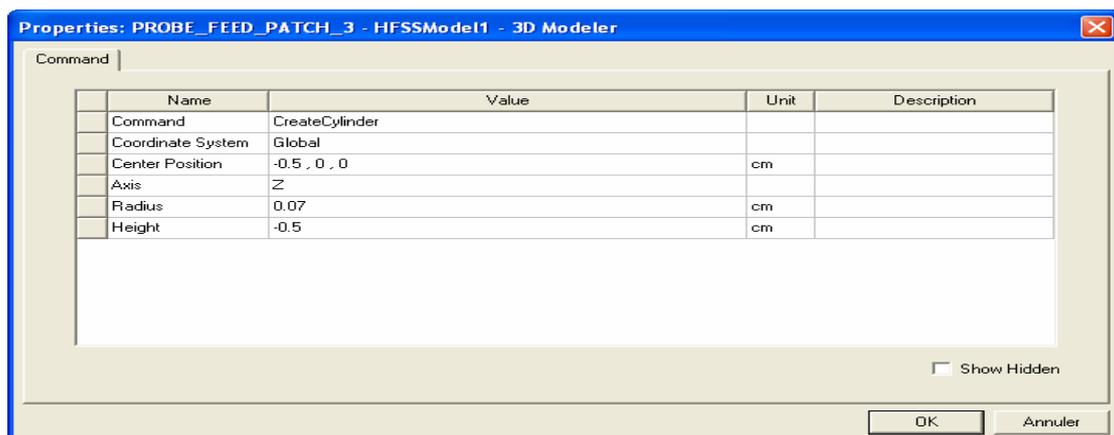
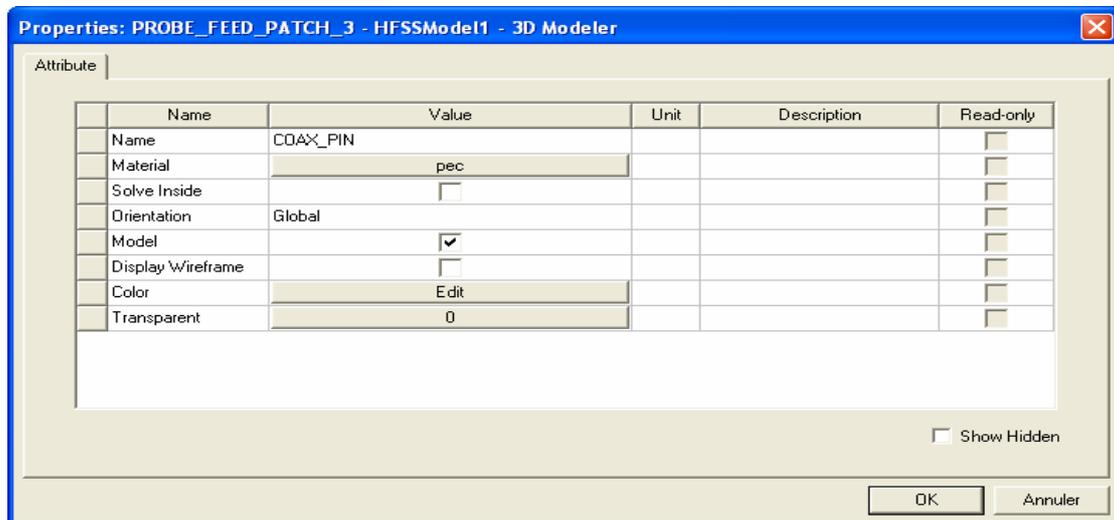
✓ Create the coax.

So to create the coax, select the menu **item Draw > Cylinder**, then enter the data as described below



- ✓ Create the Coax Pin

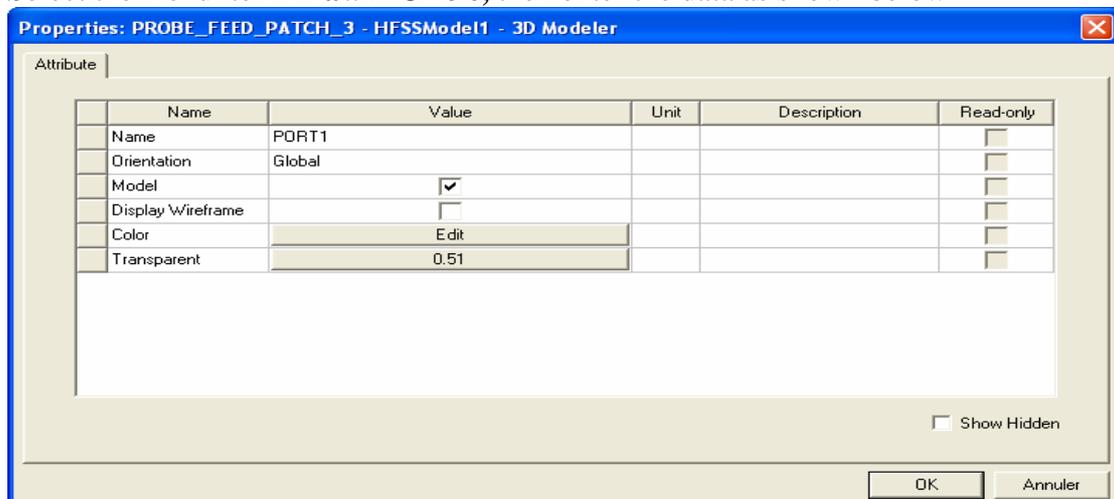
So Select the menu item **Draw > Cylinder**, then enter the data as described below

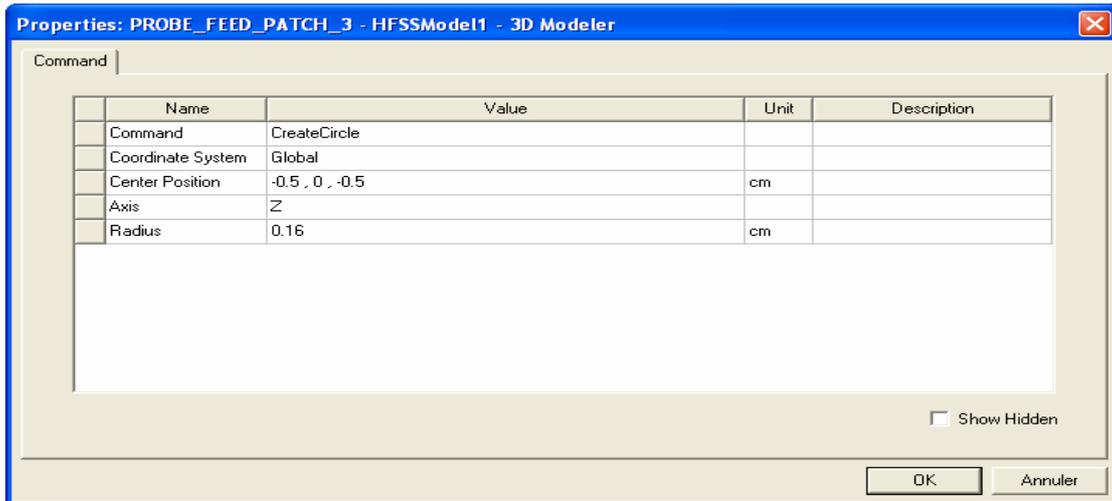


- ✓ Create the Wave port

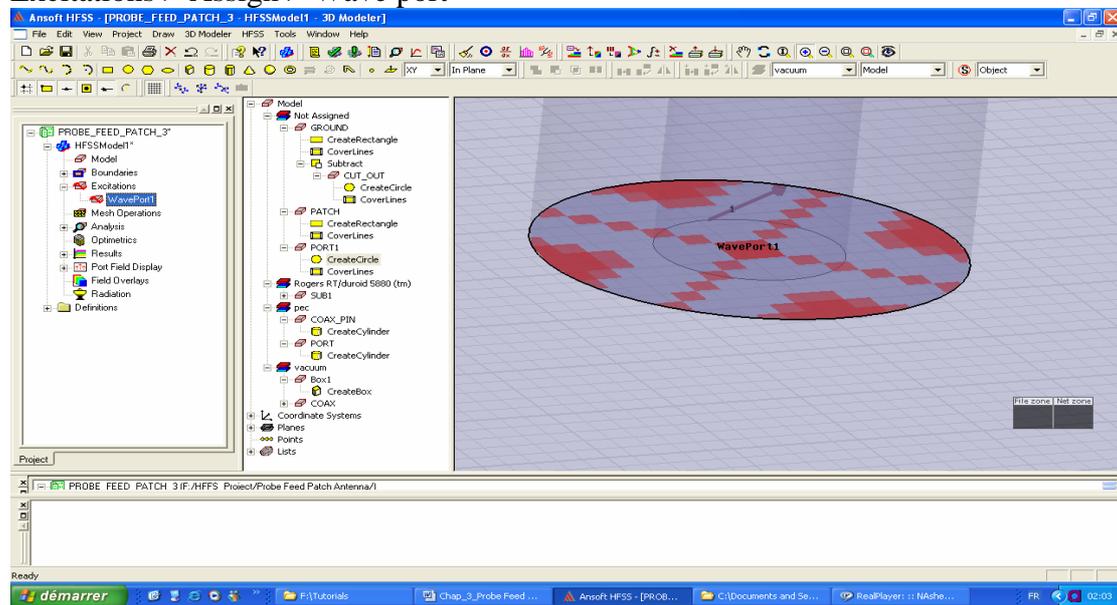
To create a circle that represents the port:

Select the menu item **Draw > Circle**, then enter the data as shown below



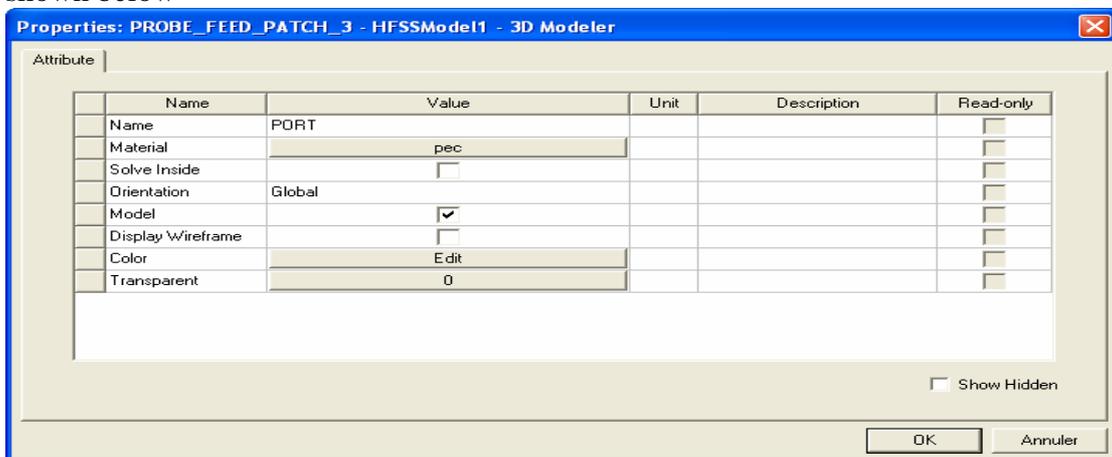


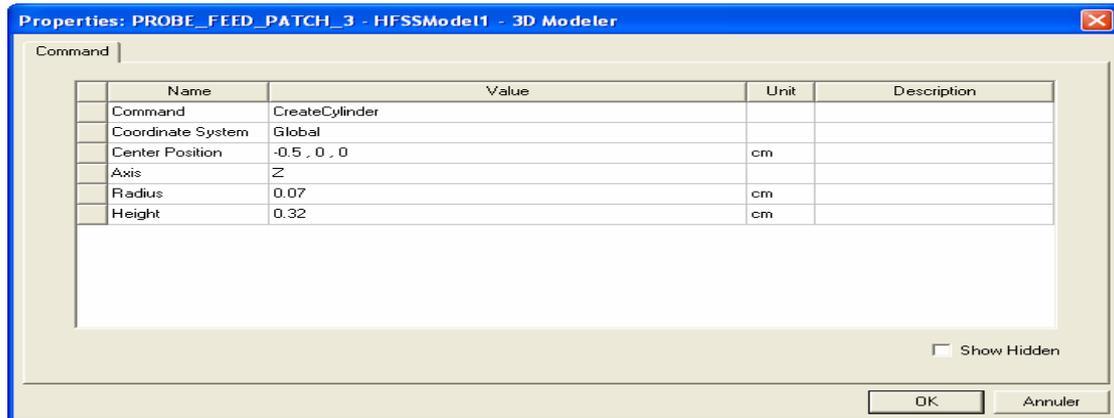
To assign wave port excitation, select **Port 1**, then go to menu item HFSS > Excitations > Assign > Wave port



✓ Create the Probe

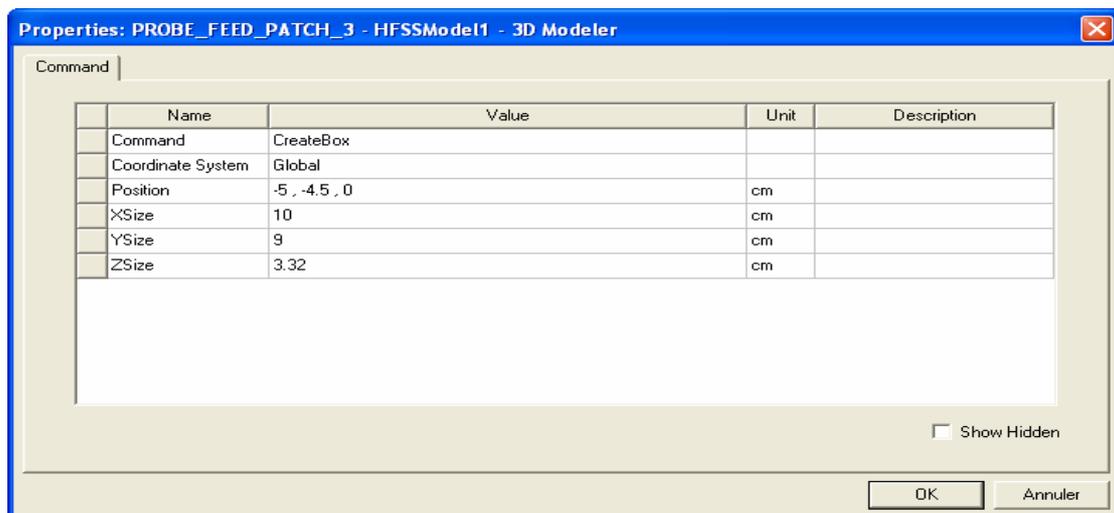
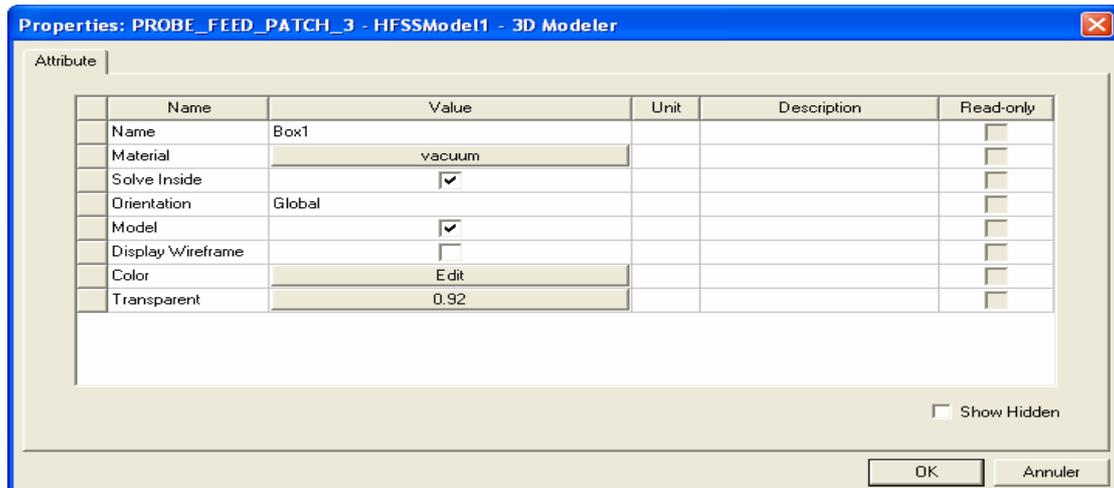
To create the probe, select the menu item **Draw > Cylinder**, then enter the data as shown below





5 – Assign Boundary

To draw the Air Box, click  on the toolbar. Then draw a box by filling the following data as shown below.



Now select Box 1, right click > Assign Boundary > radiation

6 – Analysis Setup

Finally, you have your model ready to run. Now you need to identify your analysis setup.

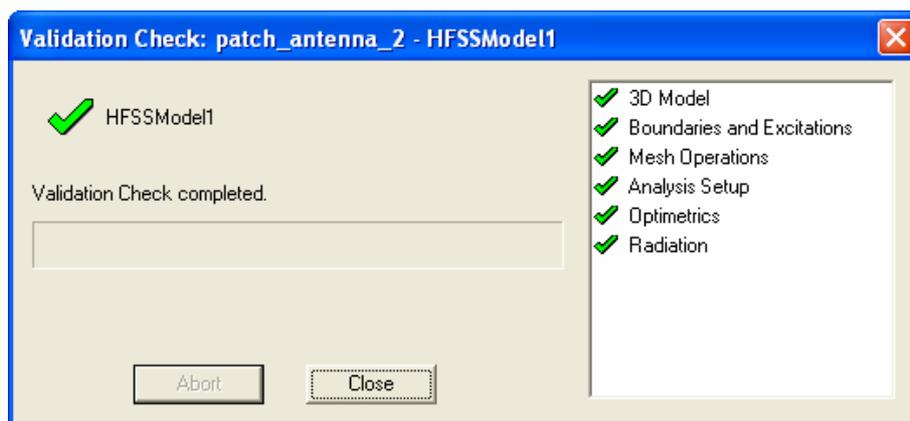
To create an analysis setup, select the menu item **HFSS > Analysis Setup > Add Solution Setup**. In the Solution Setup window, click the general tab, Solution frequency is 2.55 GHz, Maximum Number of Passes is 20 and Maximum Delta S per Pass is 0.02.

7 – ADD Frequency Sweep

To add a frequency sweep, select the menu item **HFSS > Analysis Setup > Add Sweep**. Select Solution Setup: Setup1. Click OK button. Then Edit Sweep Window. Sweep Type: Fast, Frequency Setup Type: Linear Count, Start: 1 GHz, Stop: 3 GHz, Count: 200. Click OK button.

8 – Model Validation

To validate the model, select the menu **HFSS > Validation Check**. Click the Close button. To view any errors or warnings messages, use the Message Manager.



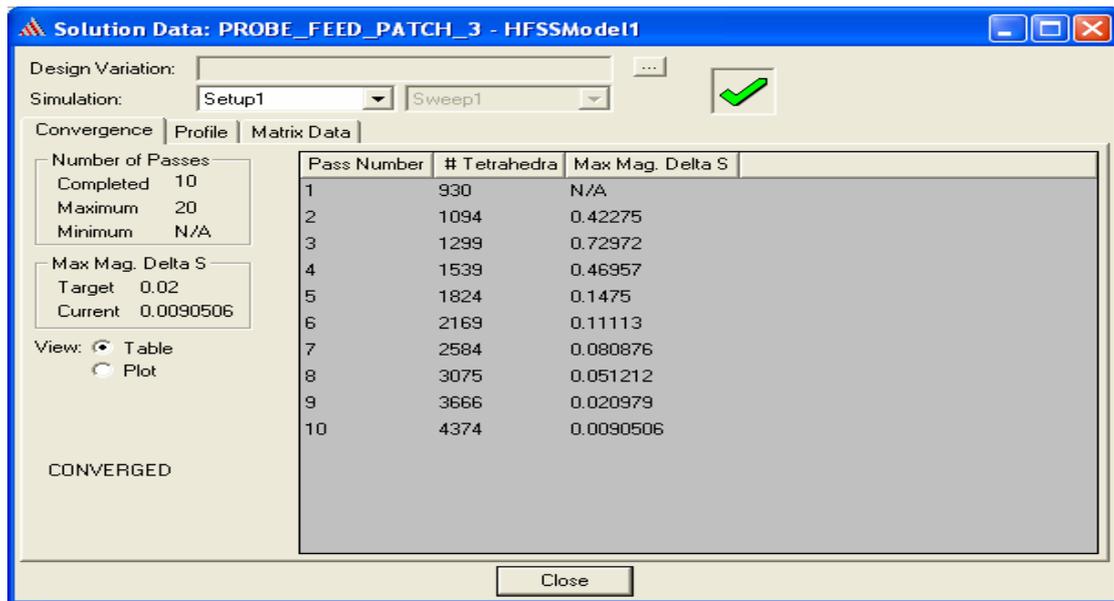
9 – Analyze

To start the solution process, select the menu item **HFSS > Analyze**.

Or click on the icon .

10- Solution Data

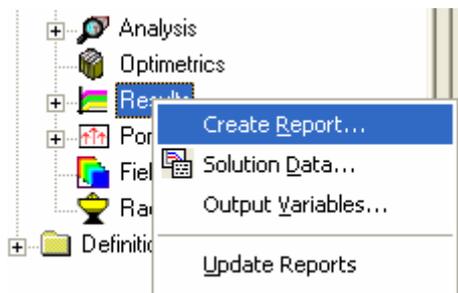
Note: The Solution Data window can be also displayed by right-click on the Setup1 under analysis on the HFSS design tree. Note also that the default view is Profile. Select the Convergence tab.



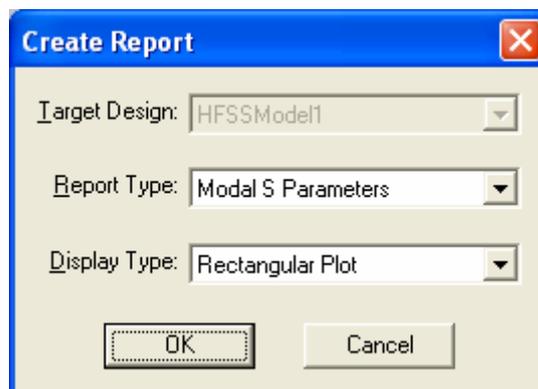
The simulation will stop as soon as the results converge, which is at pass 10.

11- Create Reports

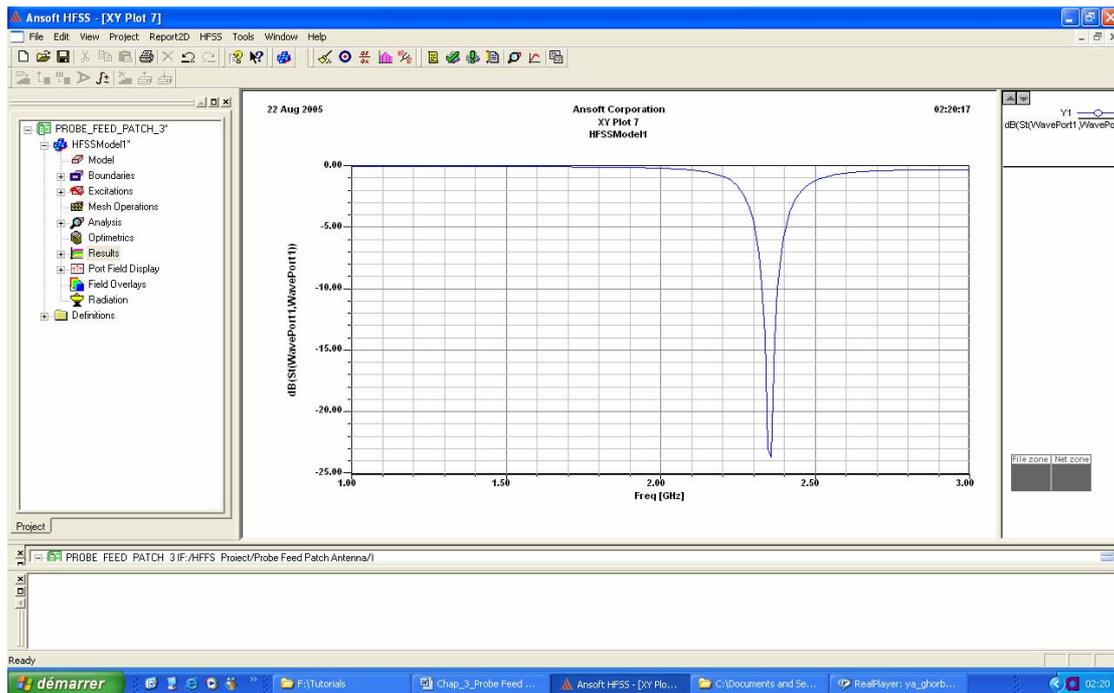
To create a report, select Results > Create Report.



Set Report Type to Modal S Parameters, Display Type to Rectangular then click OK button.

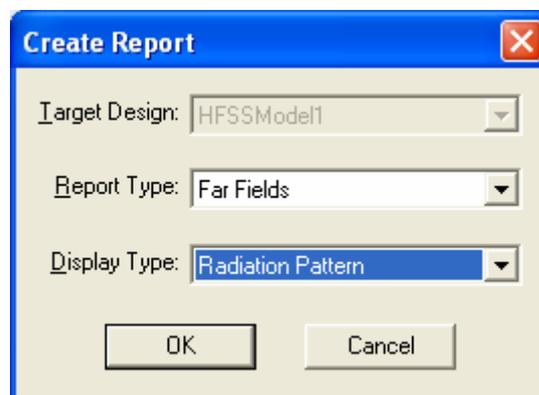


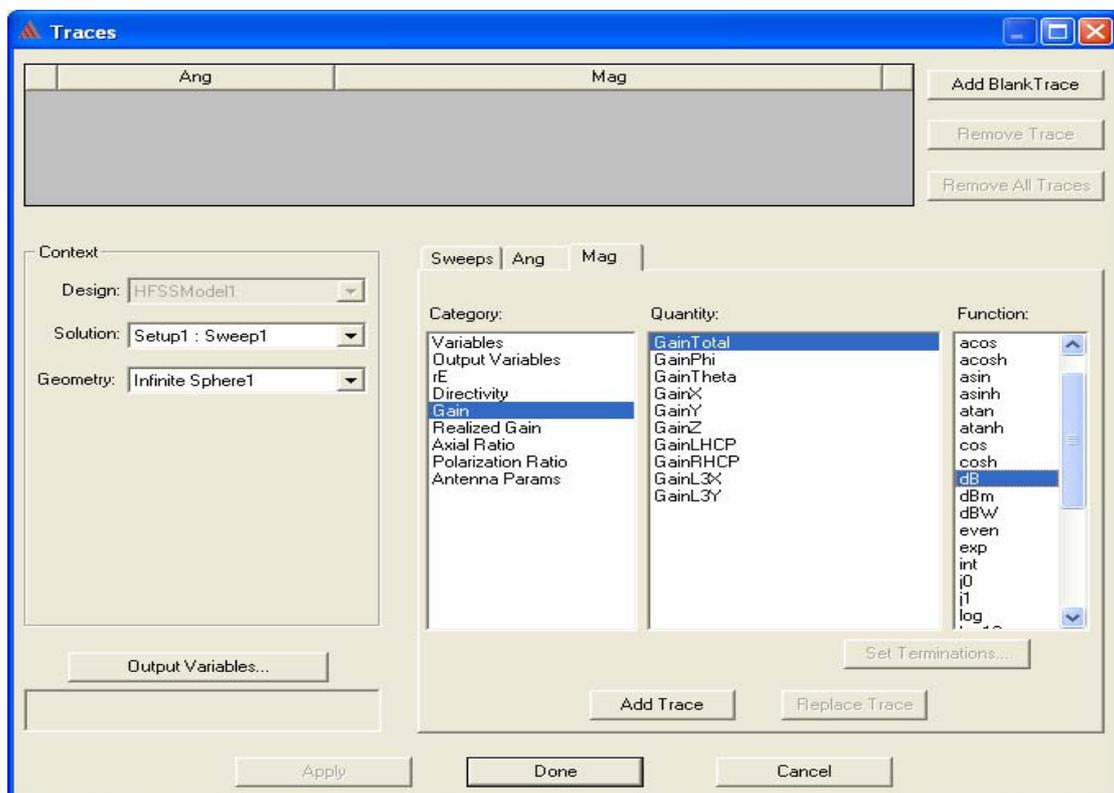
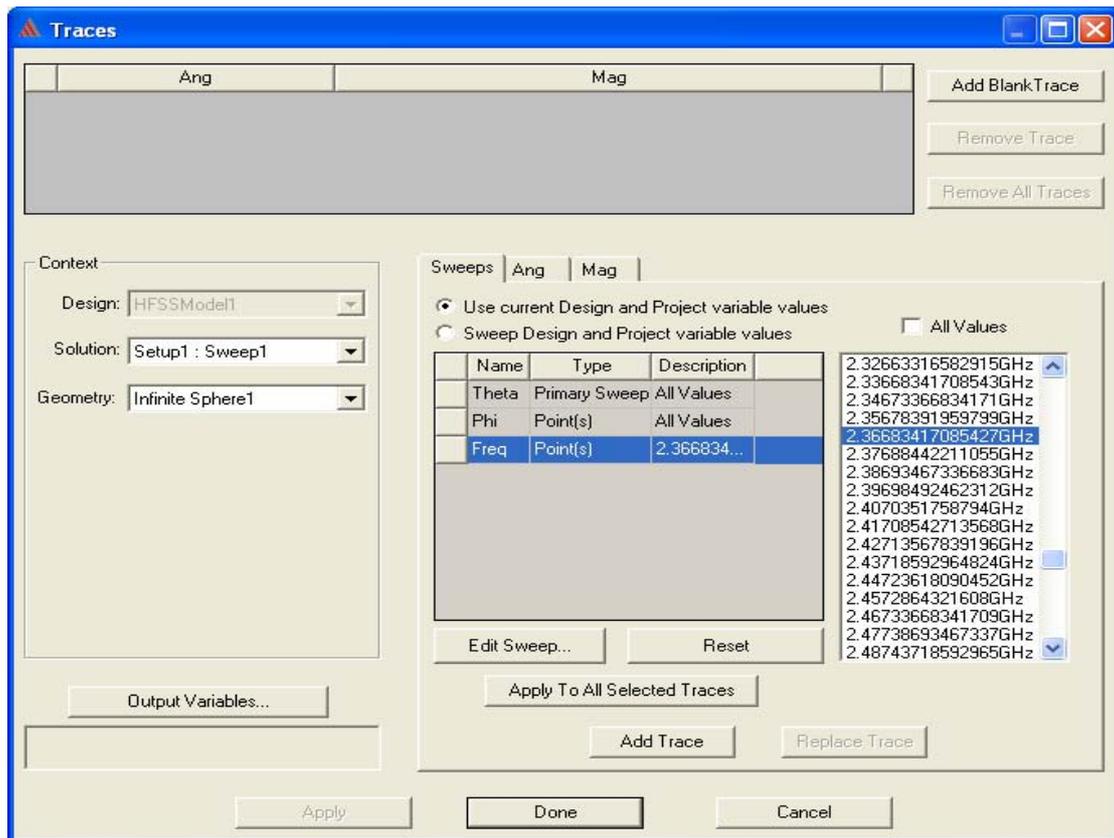
In the Traces Window, set Solution to Setup1: Adaptive1. In the Y tab, set Category to S Parameter, Quantity to S (waveport, waveport), Function to dB and click Add Trace button. Click Done button. Note that you can create any type of report it all depends on what you want to analyze specifically.

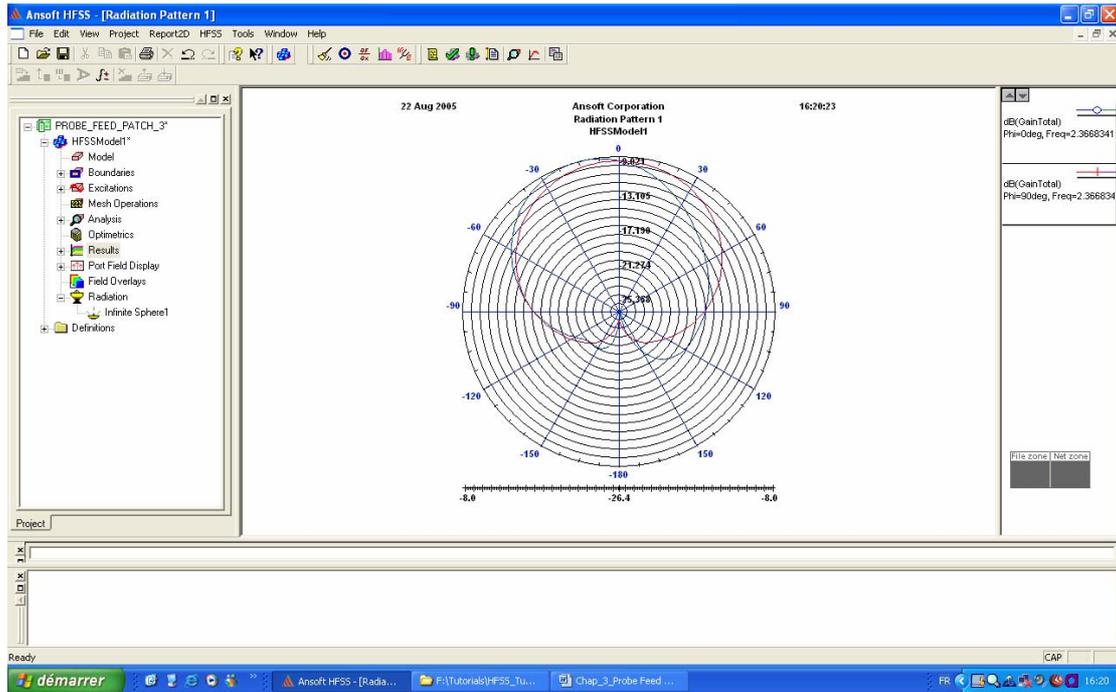


12- Radiation Pattern

To create a 2D polar far field plot go to Results > create Report. When the new window pops up change the Report Type to **Far Field** and Display type to **Radiation Pattern** then click OK.







Chapter Four – Triangular Microstrip Antenna

I – Introduction

This fourth chapter is intended to show you how to create, simulate and analyze a complex antenna such as dual frequency equilateral triangular antenna with a pair of narrow slots (Figure 4.1) using the Ansoft HFSS. The main aim of this chapter is to show how to create complicated drawing.

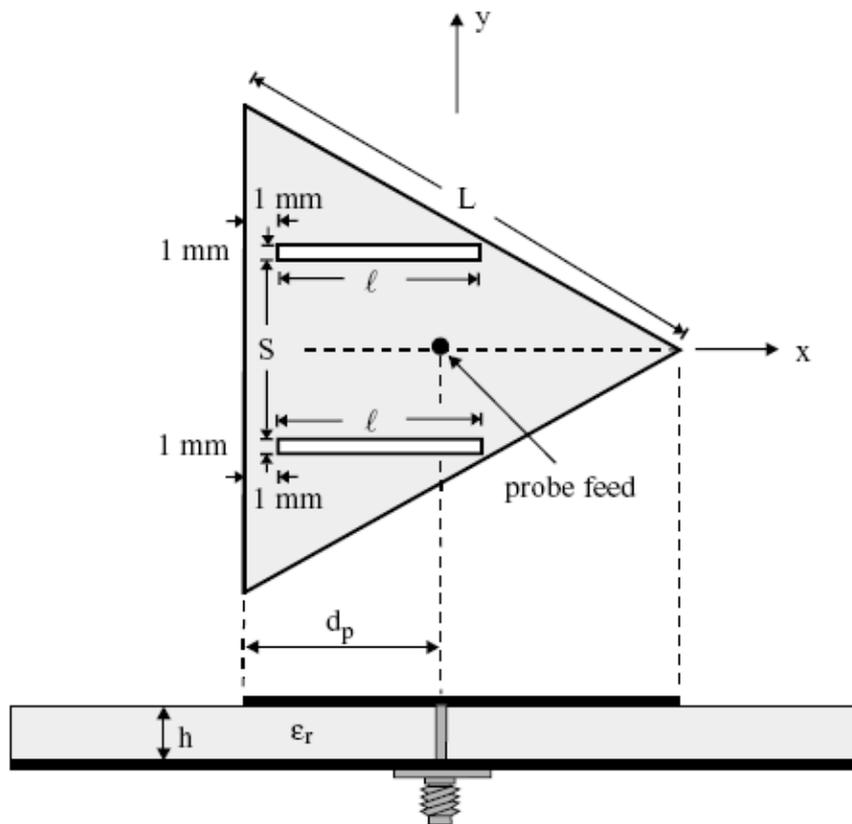


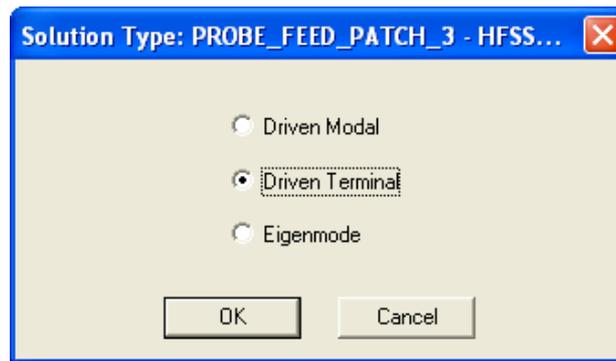
Figure 4.1 – Geometry of the dual frequency equilateral triangular antenna with a pair of narrow slots

II – Getting Started

By now, you can launch HFSS, opening a project and name it “**Triangular_Antenna**”.

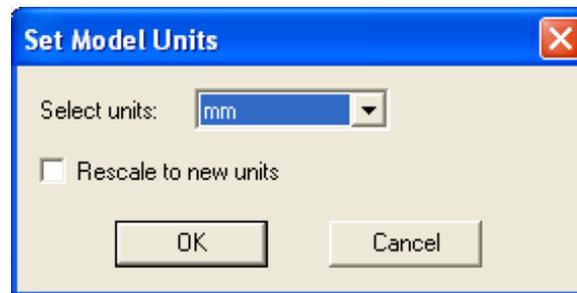
Then set the solution type:

- select the menu item **HFSS > Solution Type**
- choose **Driven Terminal**
- click **Ok** button



To set the units

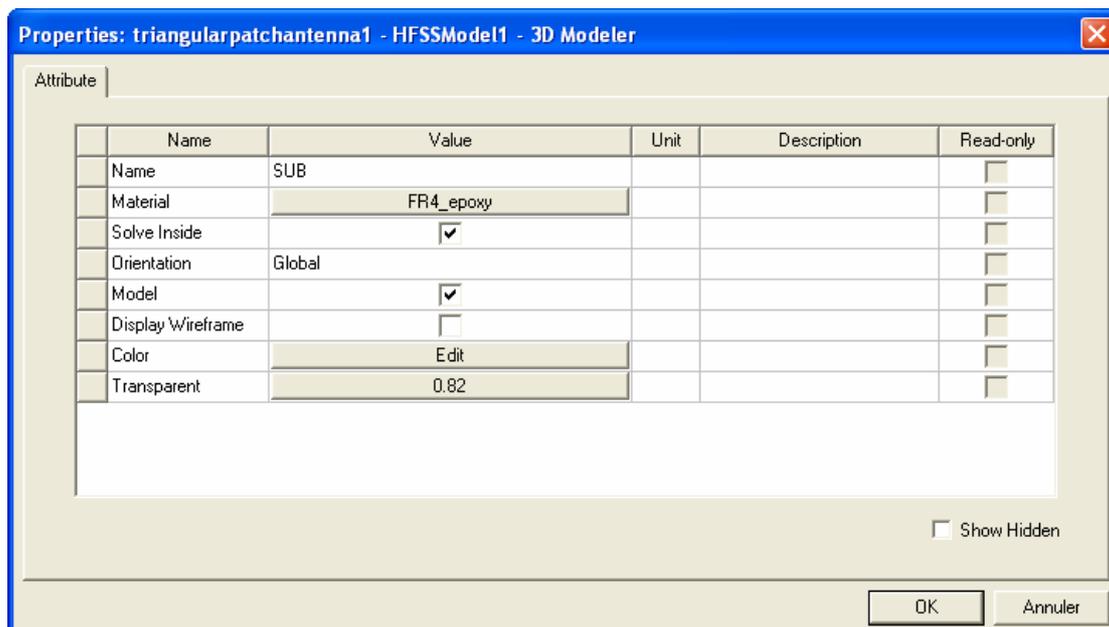
- select the menu item **3D Modeler > Units**
- select Units: **mm**
- click **ok** button

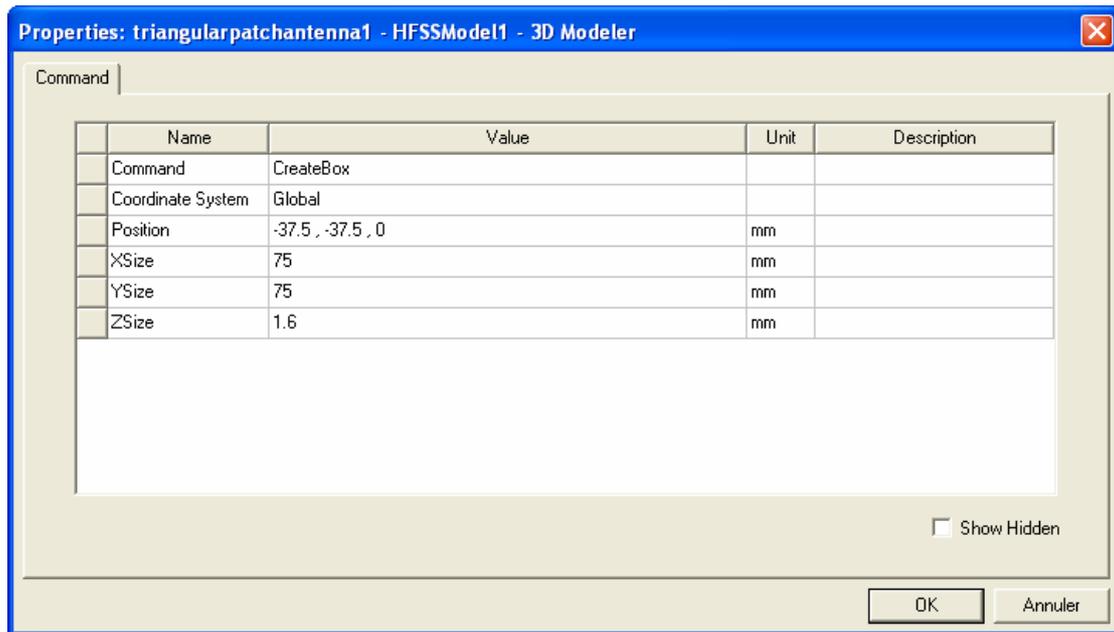


III – Dual frequency equilateral triangular antenna with a pair of narrow slots

1 – Substrate

To draw the Substrate, click  on the toolbar. Then draw a box by filling the following data as shown below.

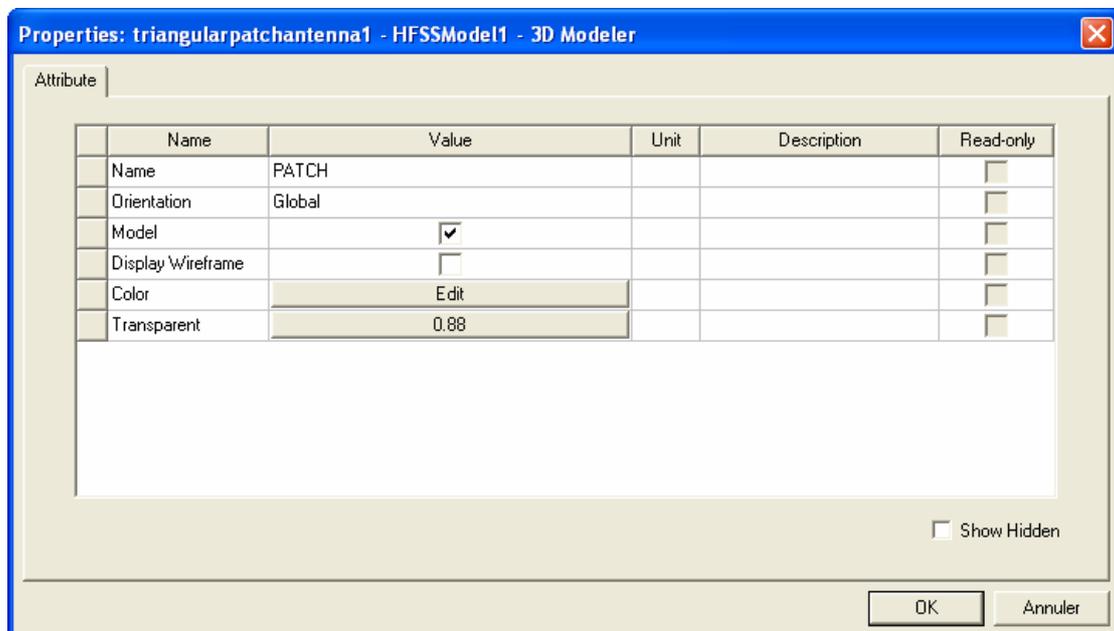


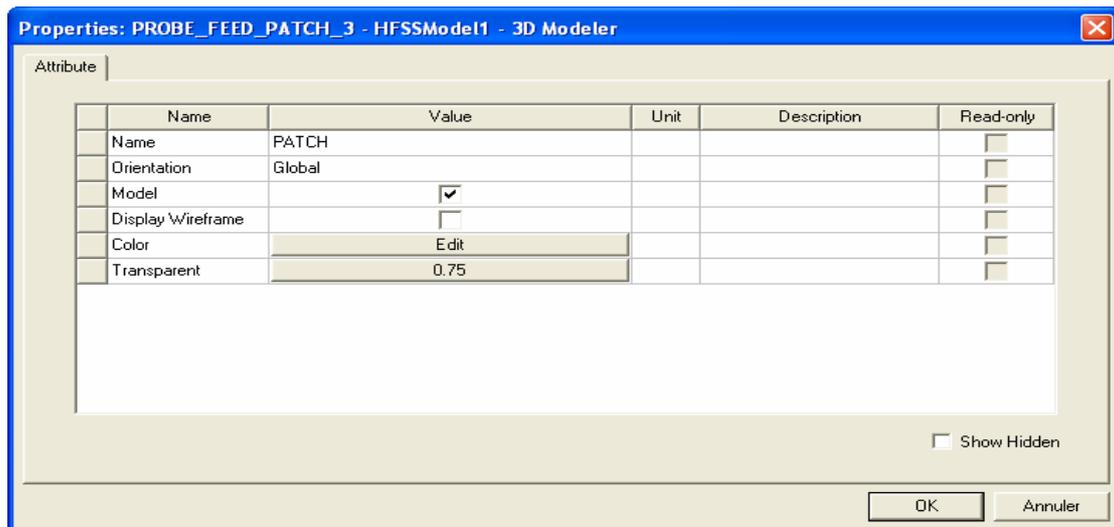


2 – Triangular Patch

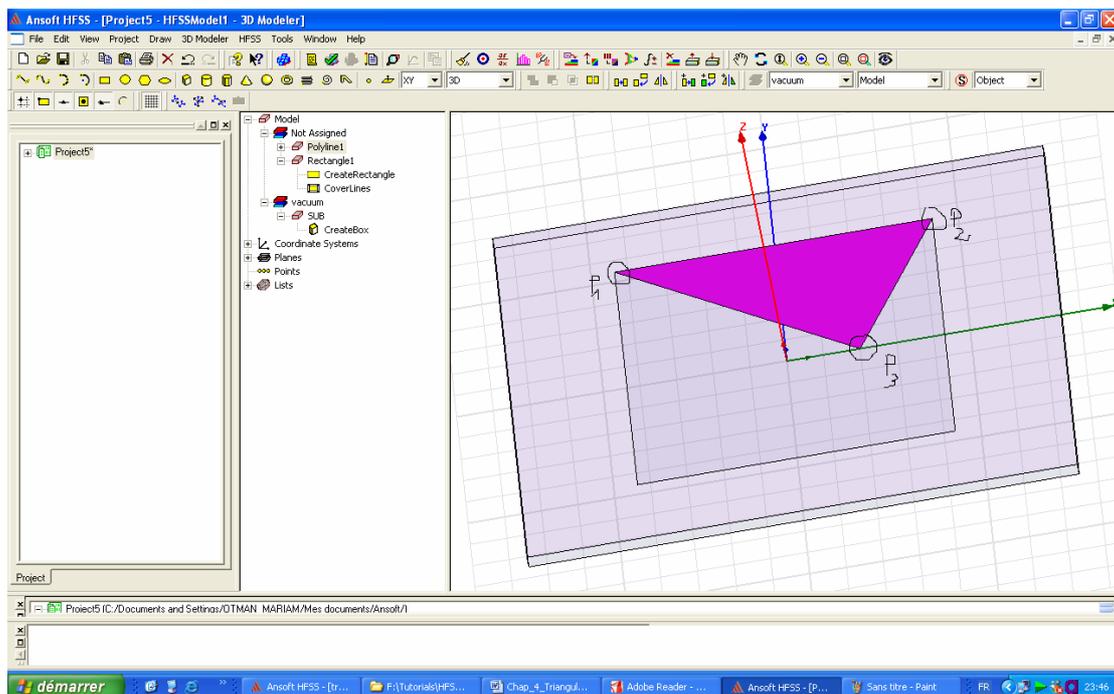
Since there isn't a triangular icon that allow us to draw triangular shapes. So to draw it, we must first begin by drawing a rectangular patch.

To draw the Patch, click  on the toolbar. Then draw rectangle by filling the following data as shown below.



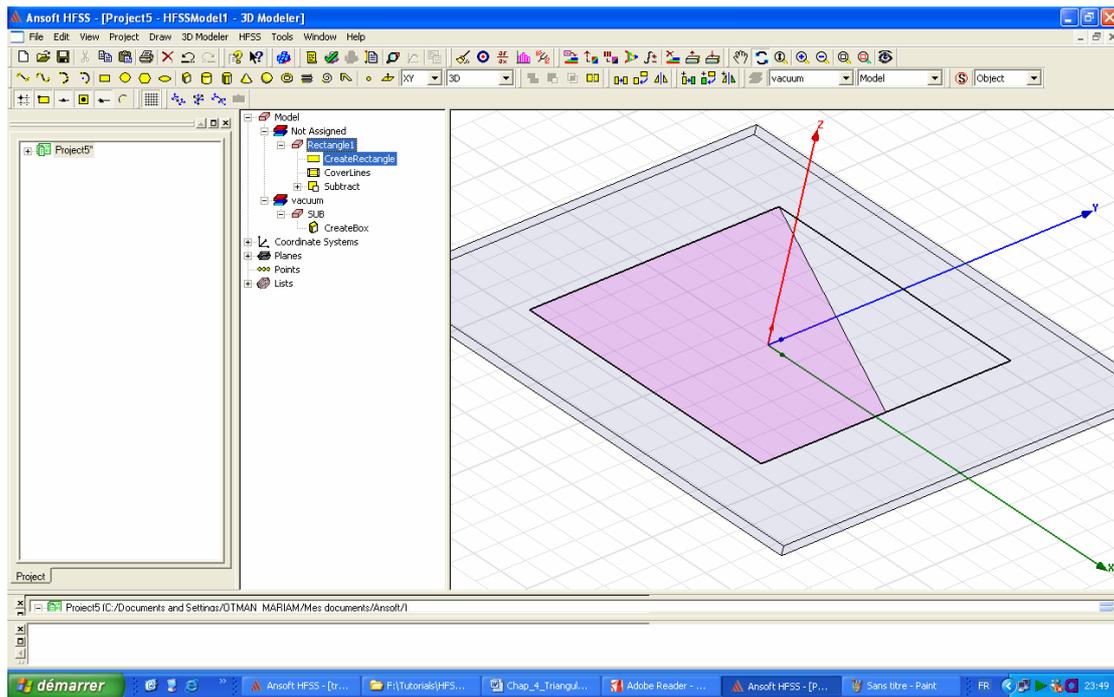


We start to draw a poly line to form a triangular shape as shown above by clicking on this icon  on the toolbar.

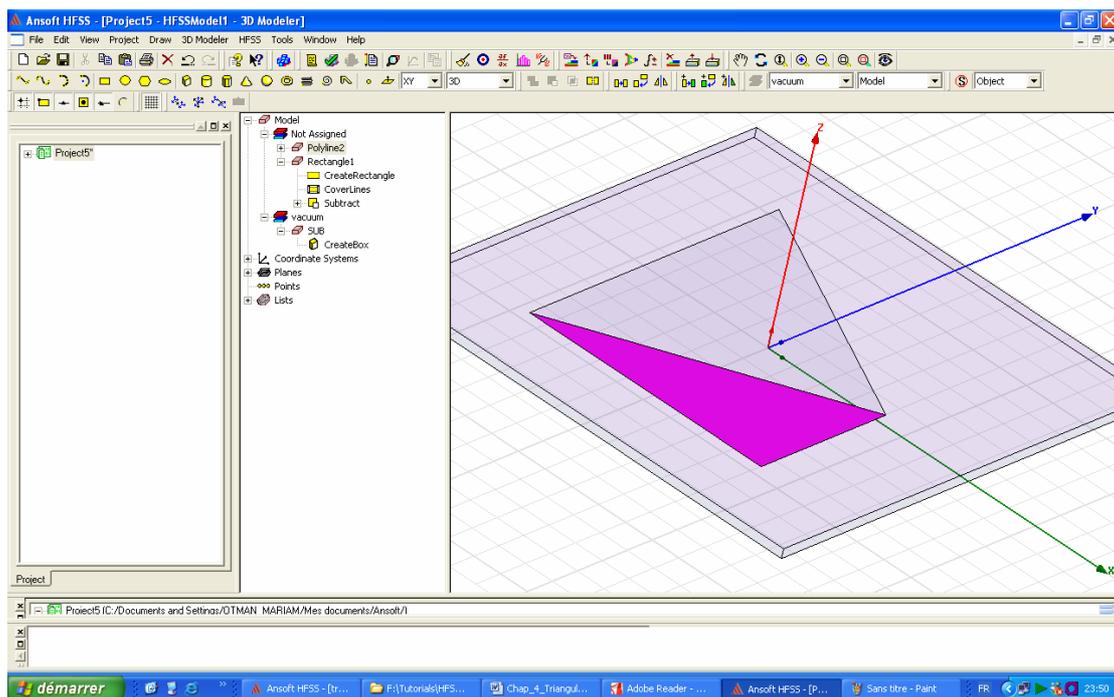


Then start to enter the point positions (P1, P2, and P3).

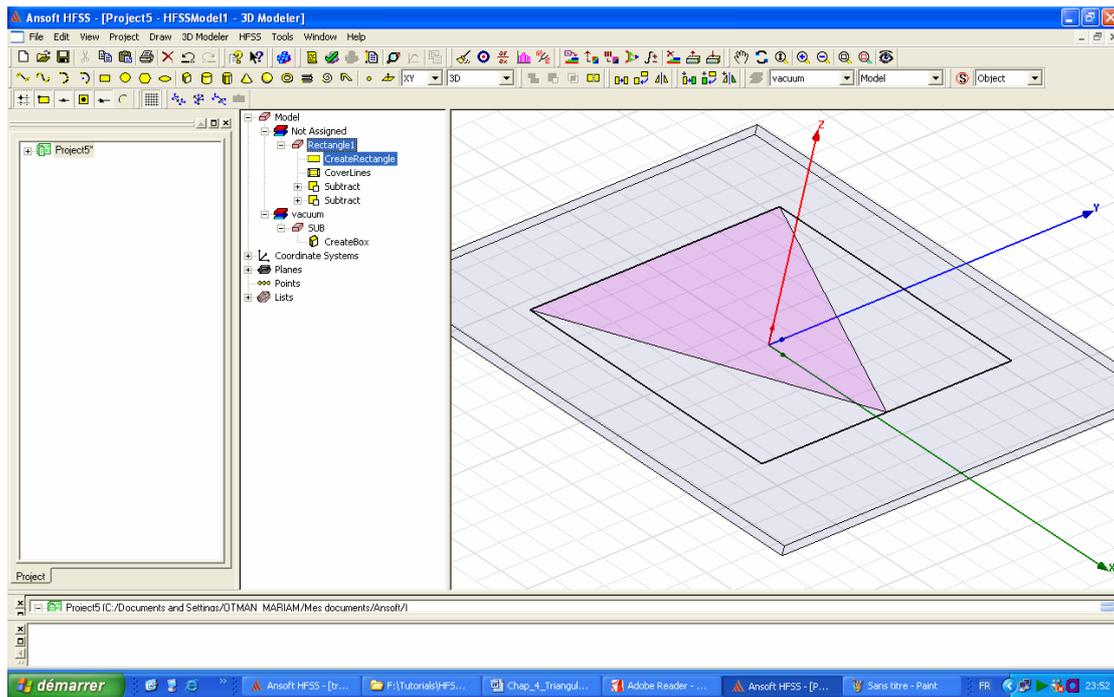
Then select the **Patch & Polyline1**, right click, select **Edit > Boolean > Subtract**



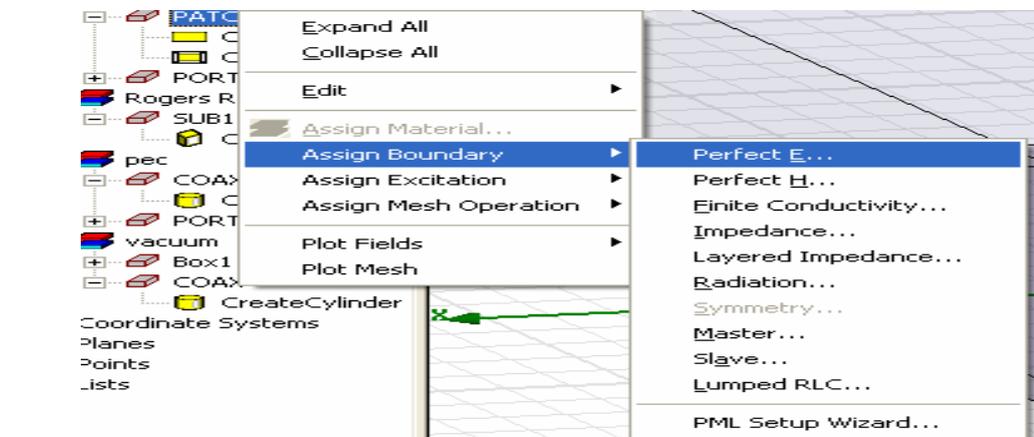
Now, we start to another a second Polyline



Then enter the appropriate point positions of the Polyline 2, when you finished select the **Patch & Polyline2**, right click, select **Edit > Boolean > Subtract**

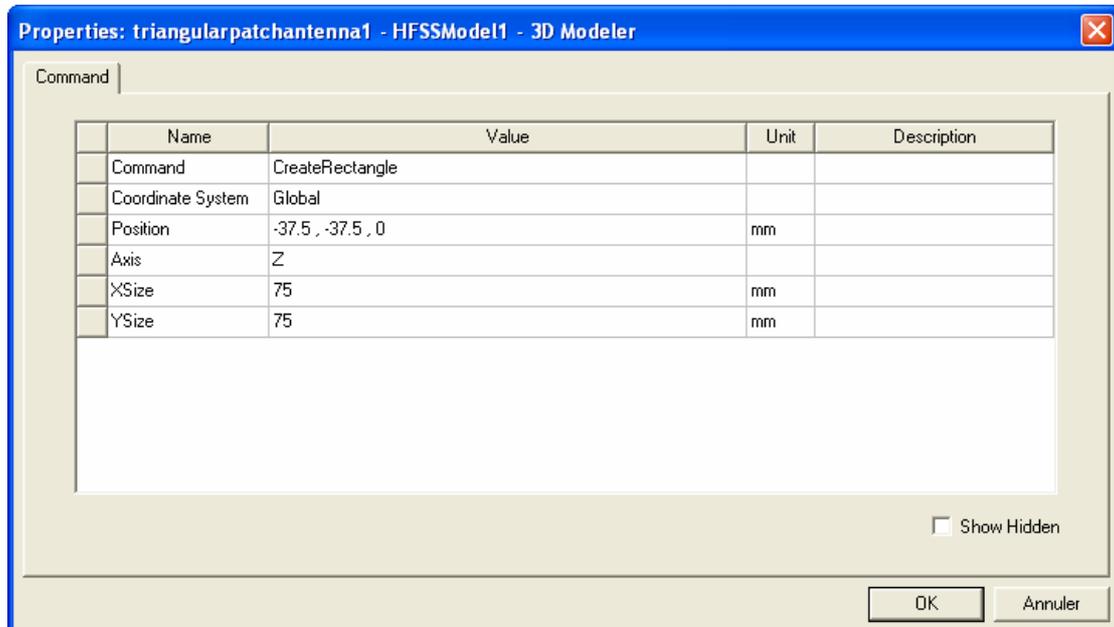
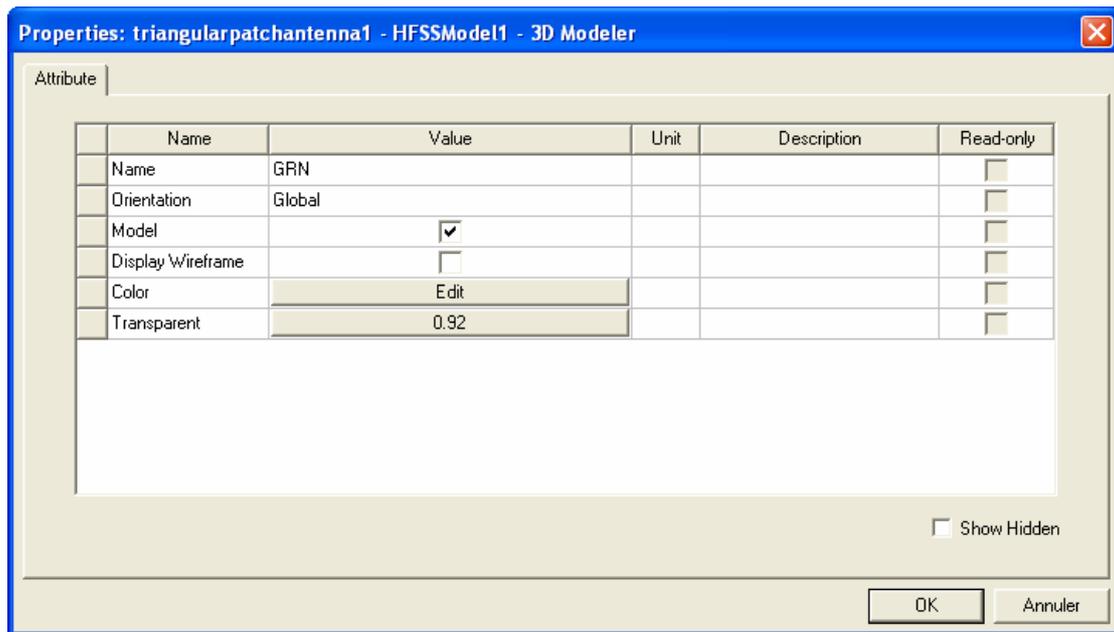


Assign a perfect E boundary to the patch. Select **Patch**, double click, select **Assign Boundary > Perfect E...**

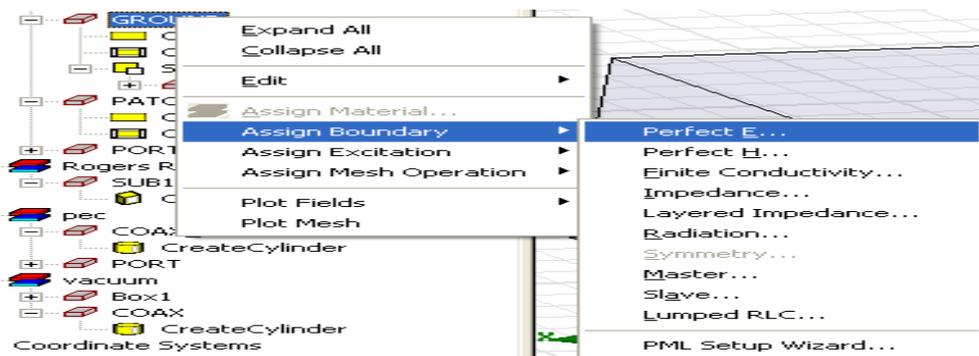


3 – Ground Plane

To draw the Ground Plane, click  on the toolbar. Then draw rectangle by filling the following data as shown below.



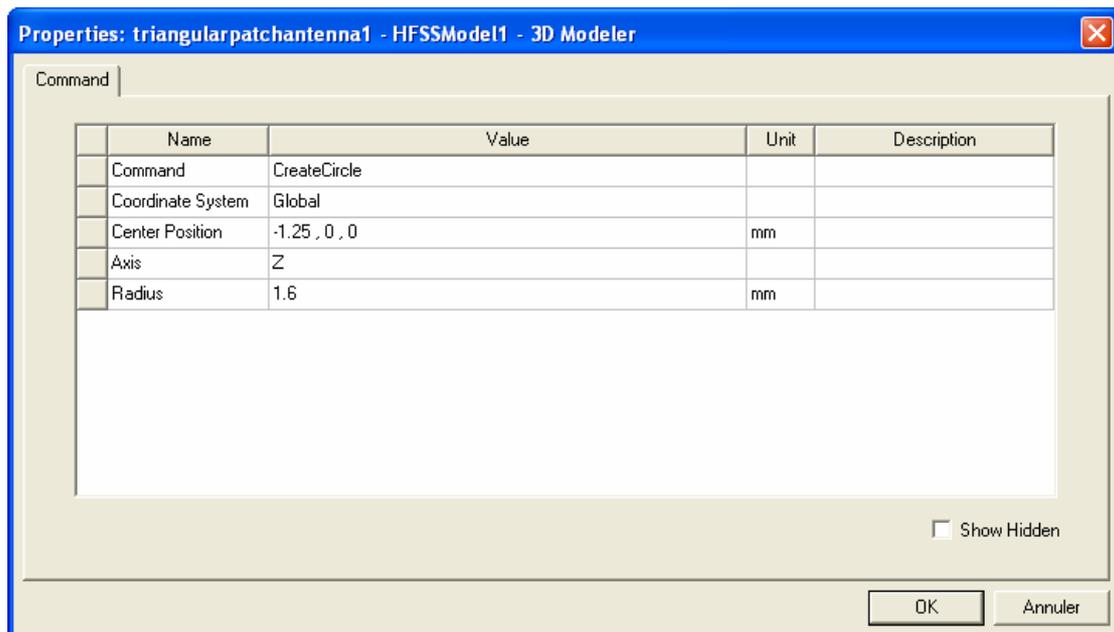
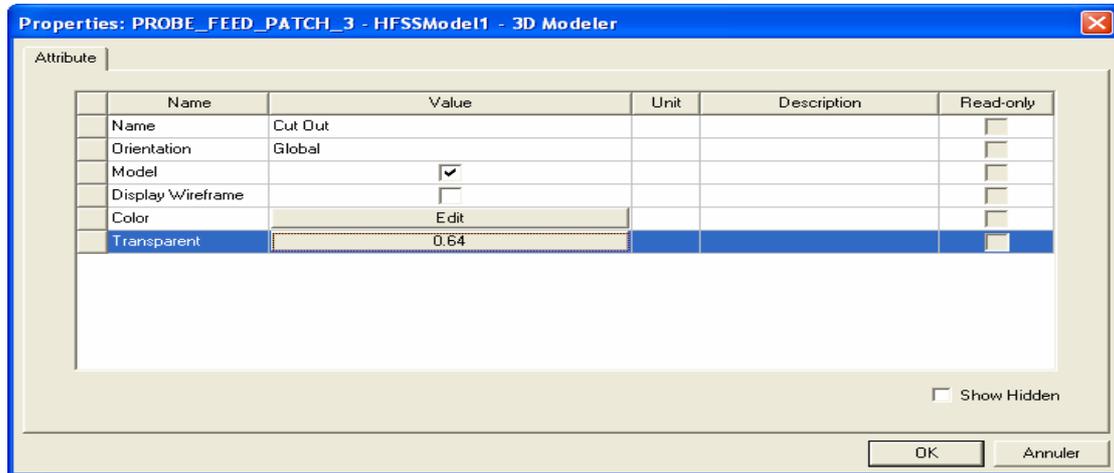
Then assign a perfect E boundary to the patch. Select **Ground**, double click, and select **Assign Boundary > Perfect E...**



4 – Coax Cable

The antenna is excited using a coax cable port. This port is located under the patch.

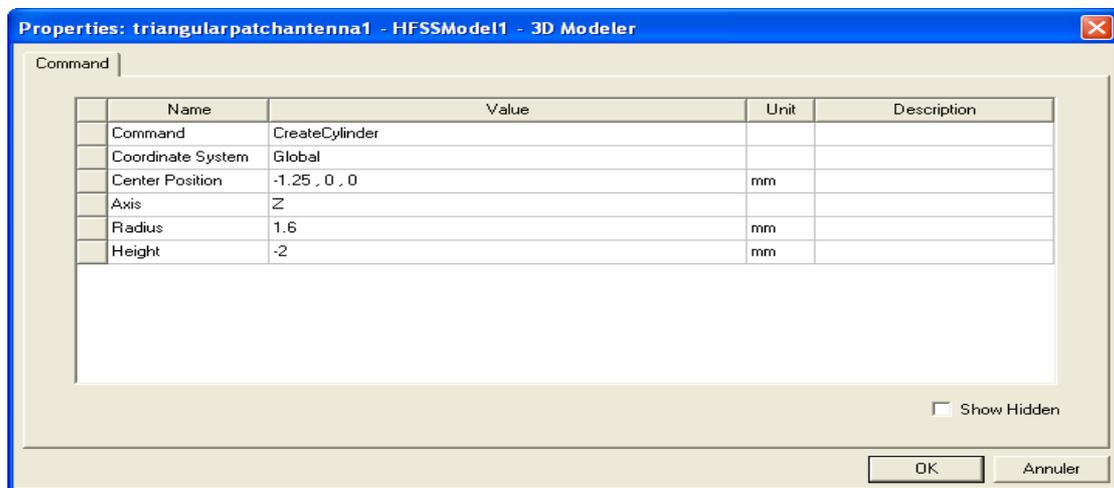
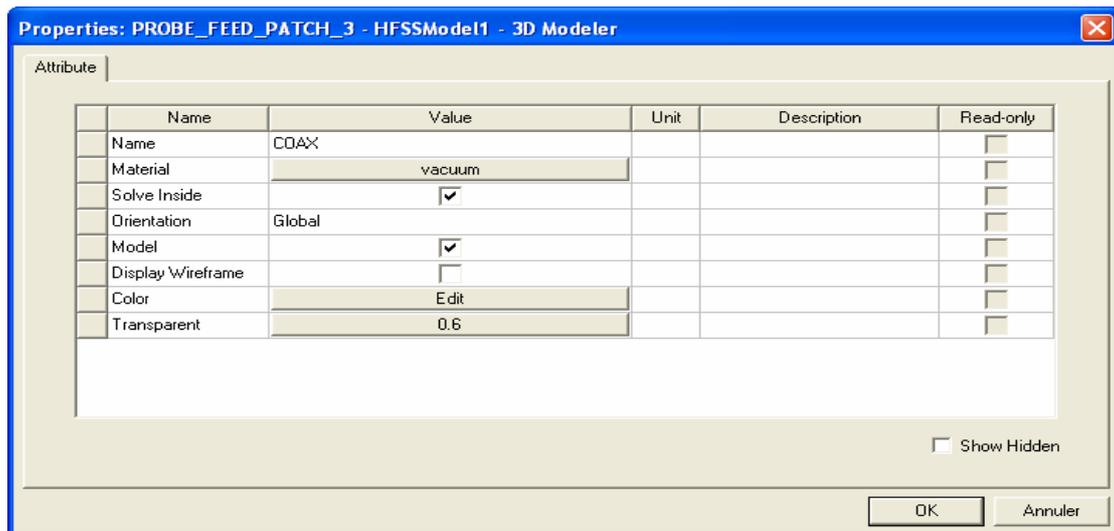
- ✓ To draw the coax cable port, we start by drawing the infinite ground Cut Out as shown below.



Then select the **Ground & Cut_Out**, right click, select **Edit > Boolean > Subtract**

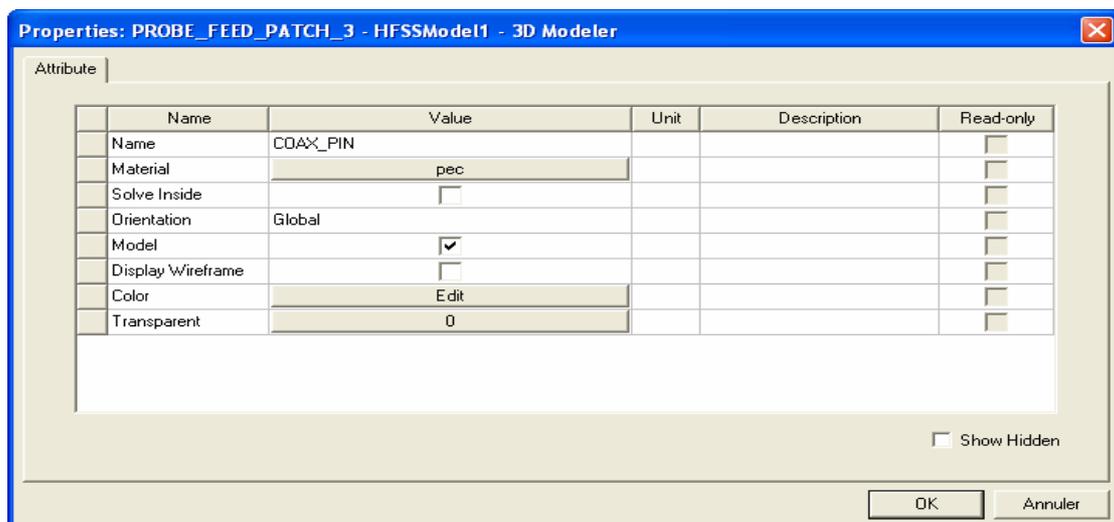
- ✓ Create the coax.

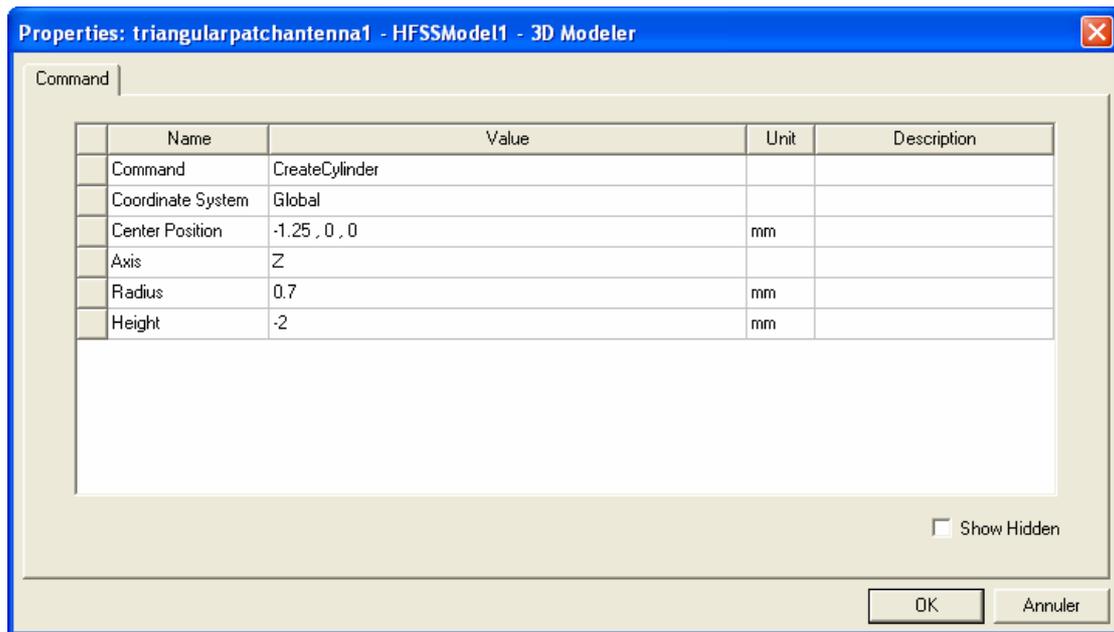
So to create the coax, select the menu item **Draw > Cylinder**, then enter the data as described below



✓ Create the Coax Pin

So Select the menu item **Draw > Cylinder**, then enter the data as described below

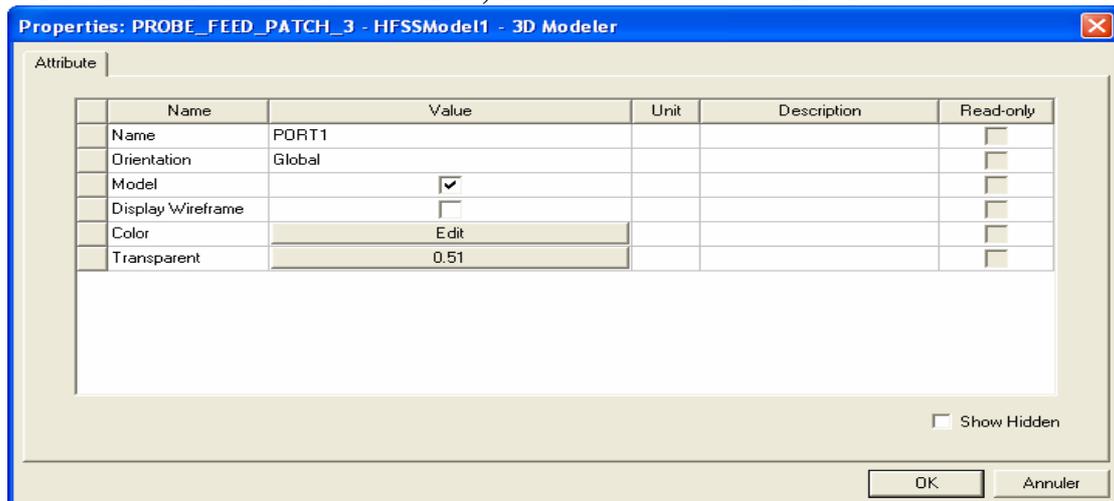


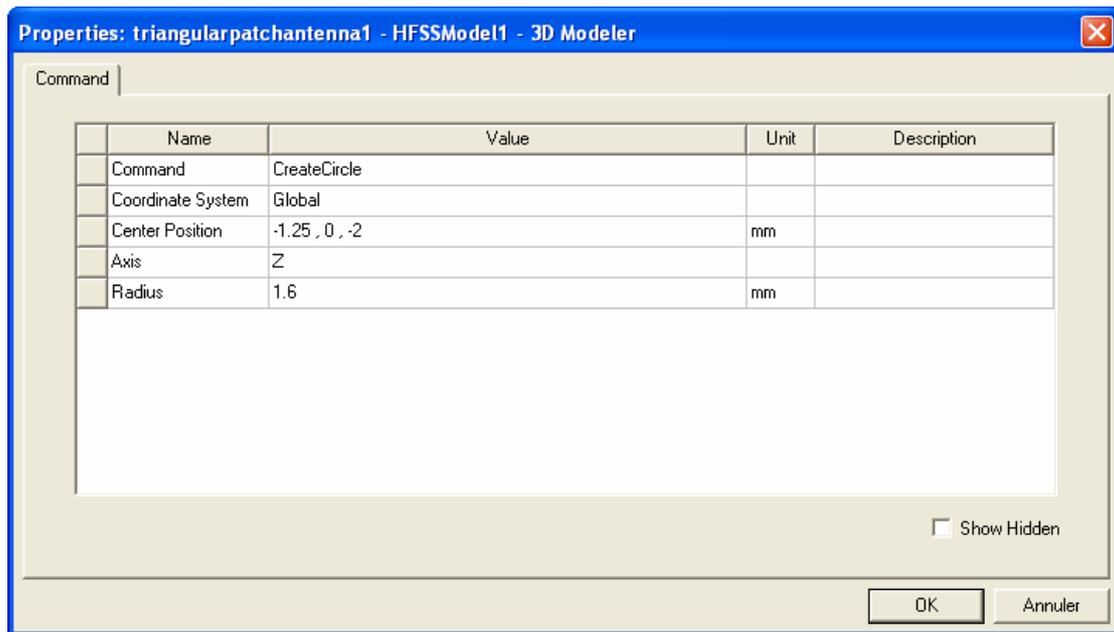


✓ Create the Wave port

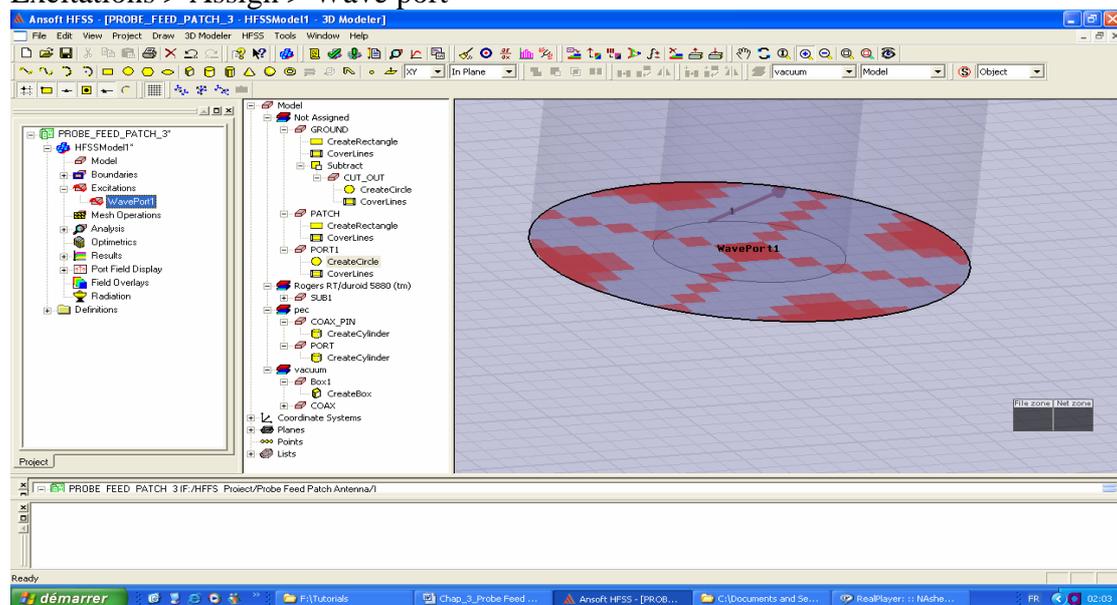
To create a circle that represents the port:

Select the menu item **Draw > Circle**, then enter the data as shown below



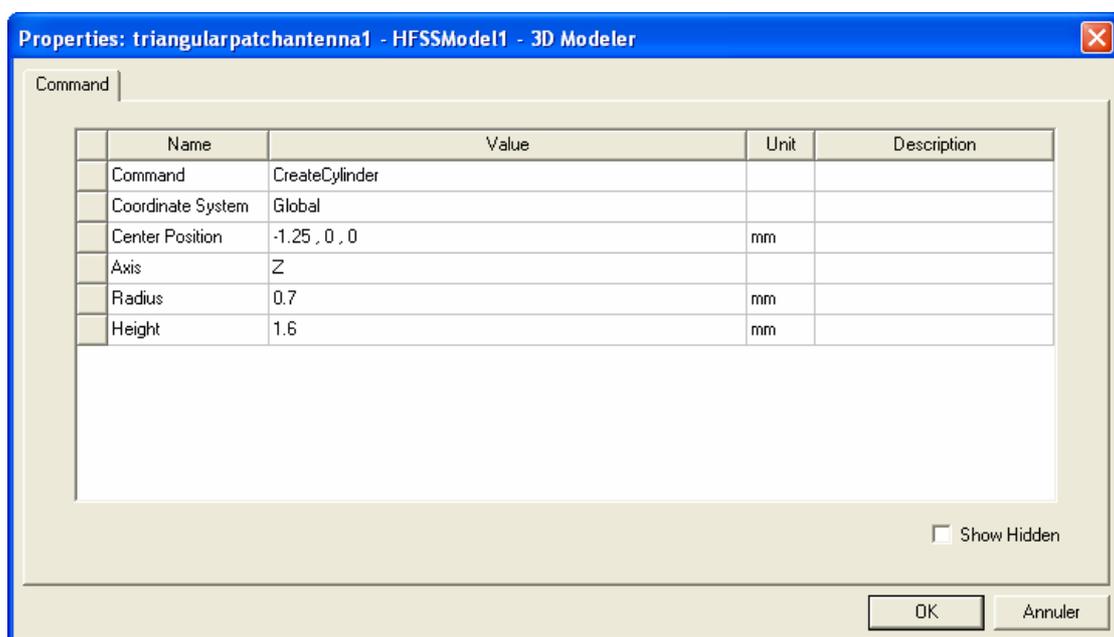
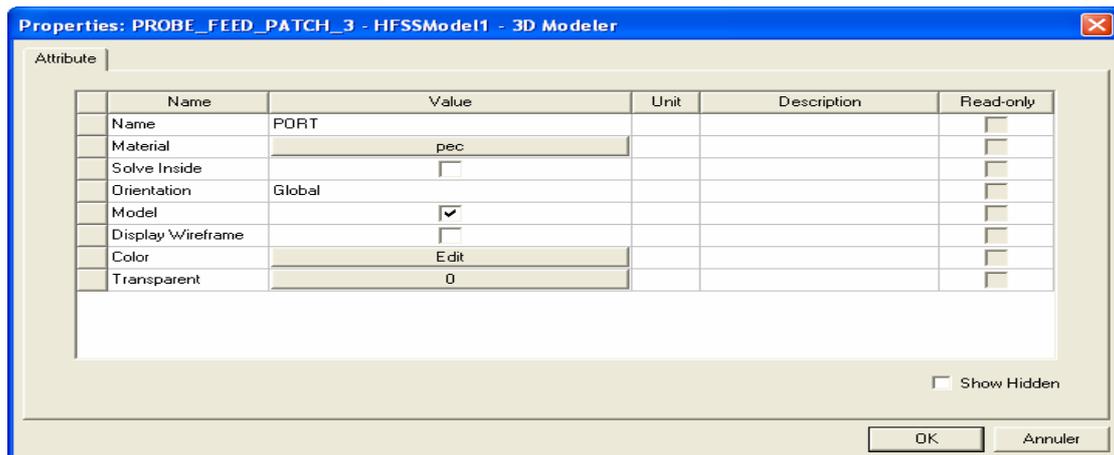


To assign wave port excitation, select **Port 1**, then go to menu item HFSS > Excitations > Assign > Wave port



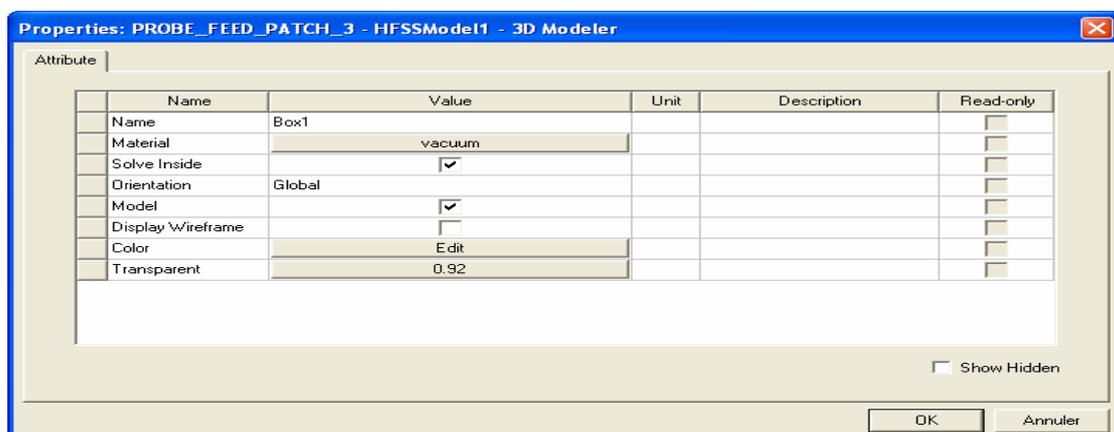
✓ Create the Probe

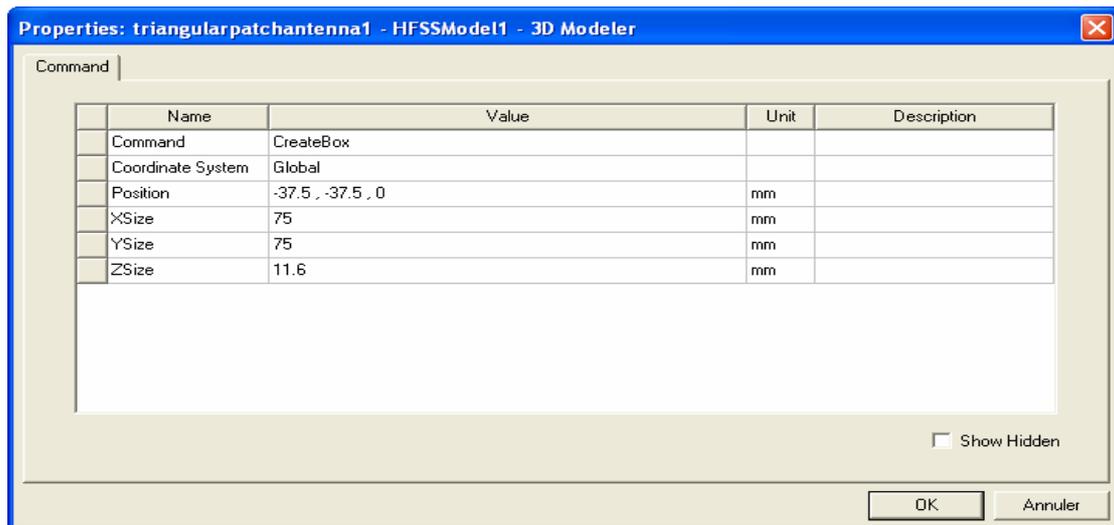
To create the probe, select the menu item **Draw > Cylinder**, then enter the data as shown below



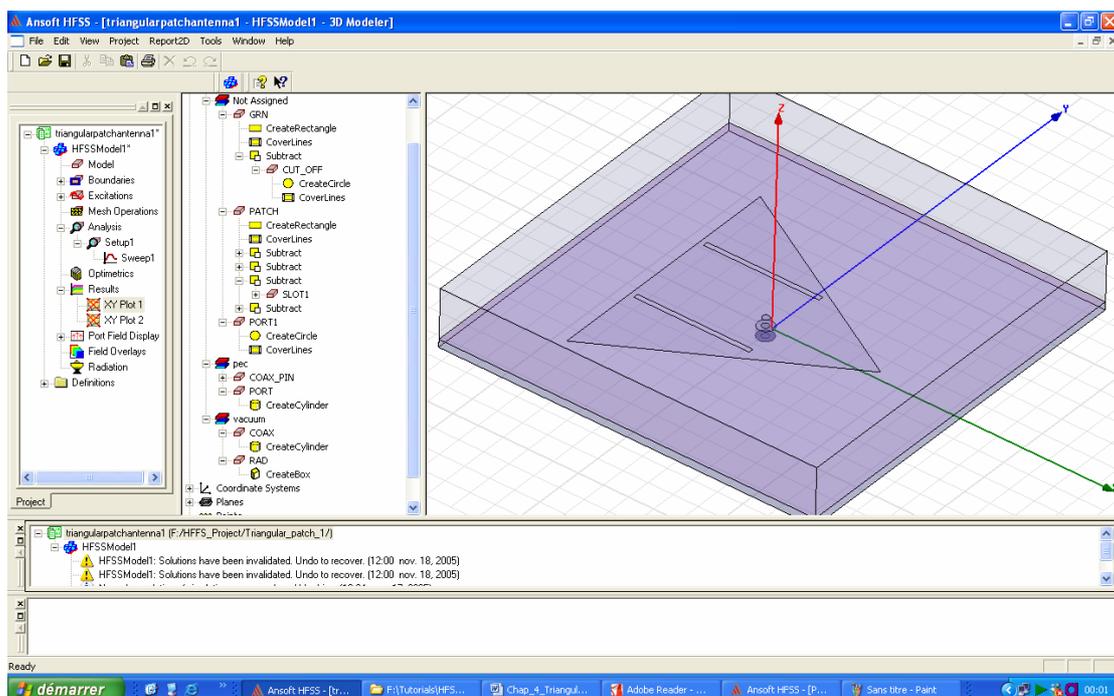
5 – Assign Boundary

To draw the Air Box, click  on the toolbar. Then draw a box by filling the following data as shown below.





Now select Box 1, right click > Assign Boundary > radiation
The final antenna should like as follow



6 – Analysis Setup

Finally, you have your model ready to run. Now you need to identify your analysis setup.

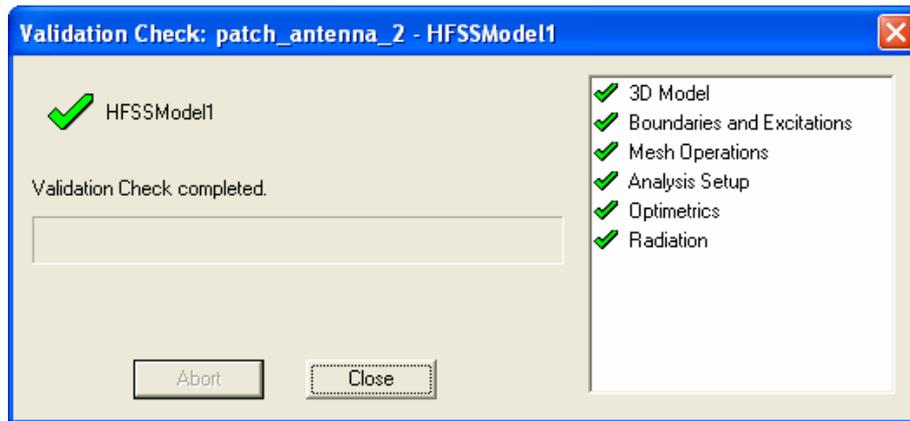
To create an analysis setup, select the menu item **HFSS > Analysis Setup > Add Solution Setup**. In the Solution Setup window, click the general tab, Solution frequency is 1.8 GHz, Maximum Number of Passes is 20 and Maximum Delta S per Pass is 0.02.

7 – ADD Frequency Sweep

To add a frequency sweep, select the menu item **HFSS > Analysis Setup > Add Sweep**. Select Solution Setup: Setup1. Click OK button. Then Edit Sweep Window. Sweep Type: Fast, Frequency Setup Type: Linear Count, Start: 1 GHz, Stop: 3 GHz, Count: 200. Click OK button.

8 – Model Validation

To validate the model, select the menu **HFSS > Validation Check**. Click the Close button. To view any errors or warnings messages, use the Message Manager.



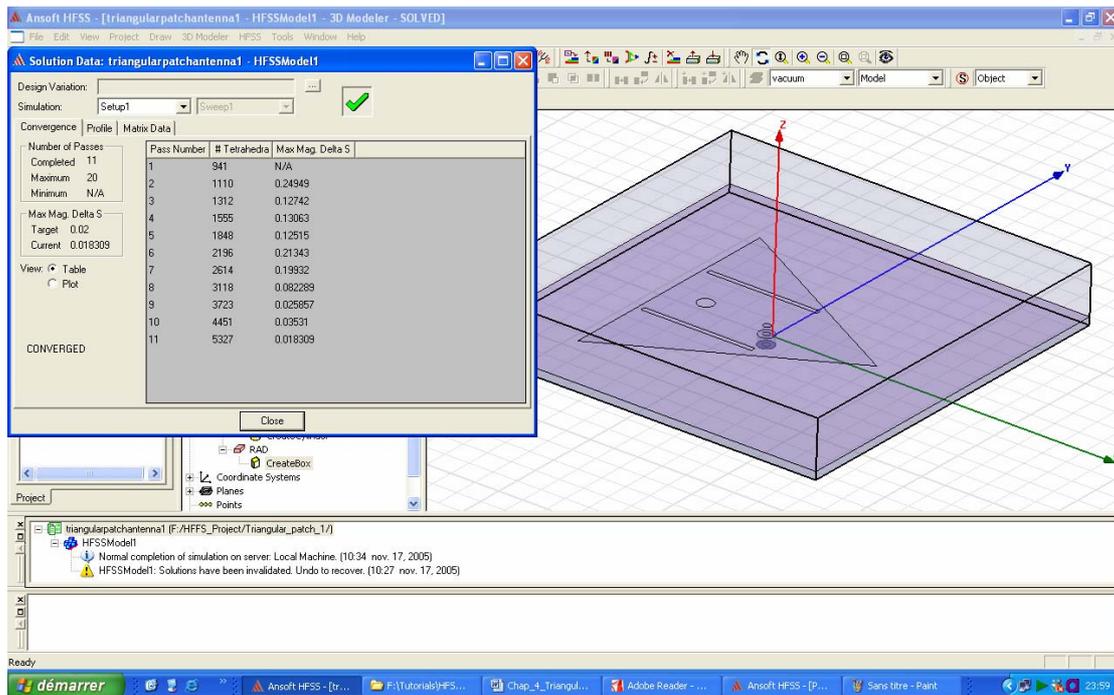
9 – Analyze

To start the solution process, select the menu item **HFSS > Analyze**.

Or click on the icon .

10- Solution Data

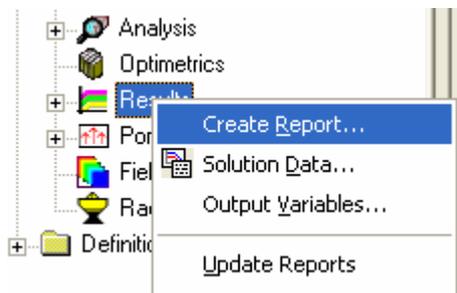
Note: The Solution Data window can be also displayed by right-click on the Setup1 under analysis on the HFSS design tree. Note also that the default view is Profile. Select the Convergence tab.



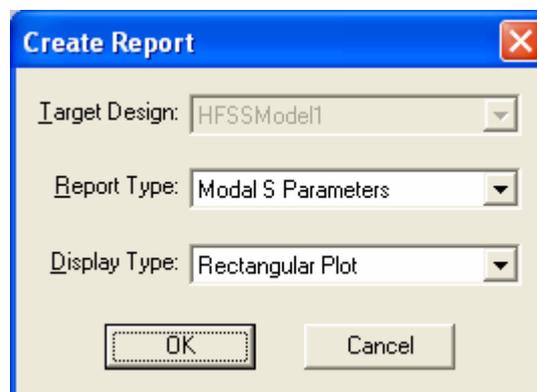
The simulation will stop as soon as the results converge, which is at pass 10.

11- Create Reports

To create a report, select Results > Create Report.



Set Report Type to Modal S Parameters, Display Type to Rectangular then click OK button.



In the Traces Window, set Solution to Setup1: Adaptive1. In the Y tab, set Category to S Parameter, Quantity to S (waveport, waveport), Function to dB and click Add

Trace button. Click Done button. Note that you can create any type of report it all depends on what you want to analyze specifically.

