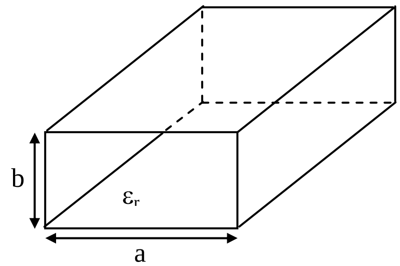
Project 1: Rectangular Waveguide (HFSS)



a = 0.9” (2.286cm)
b = 0.4” (1.016cm)
ε_r = 1.0

Objective

- Getting Started with HFSS (a tutorial)
- Using HFSS, simulate an air-filled WR-90 waveguide shown above.
- To obtain the Field patterns, intrinsic Impedance and wavelength for the first 4 modes.

Analysis

1.) Sweep from 4-20 GHz
2.) Analysis must include first three modes (TE_{10}, TE_{20}, TE_{01})
3.) Generate a graph for β, λ, η vs. frequency for each mode using HFSS

Report

1.) Format should include title, objective, analysis/discussion, results, and conclusion
2.) Include all relevant graphs and outputs from HFSS
3.) Explain and discuss results for each mode using relevant field expressions
4.) Compare results for β, λ, η with those obtained using corresponding theoretical expressions

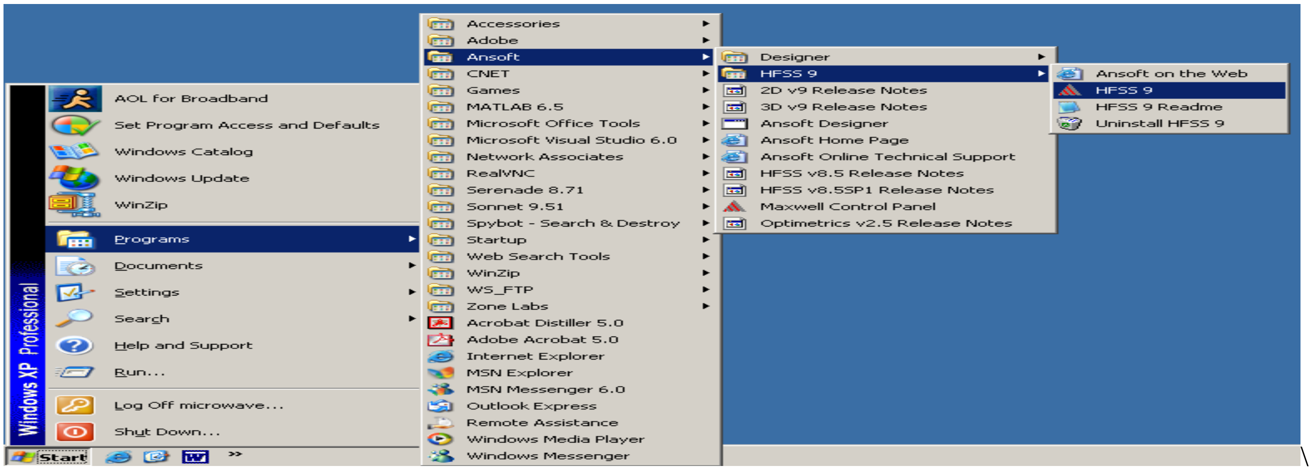
Hints and Tips:

1.) Use wave ports on the front and back faces of waveguide
2.) Length of waveguide is arbitrary, make it a few wavelengths
3.) Adapt the mesh first at 15 GHz, make delta S 0.025 at most, and use 5 passes (more may be needed if delta S criteria is not met)
4.) Set up simulation for Discrete, use 8 steps or so to generate a decent graph
5.) Perform a ‘Ports Only’ solution and use ‘Post Process, Matrix Data’ from the Executive Window to look at solution to determine propagation constants and wavelength
6.) Use ‘Post Process, Fields’ from the Executive Window to look at field distributions within the waveguide for the different modes

HFSS Tutorial

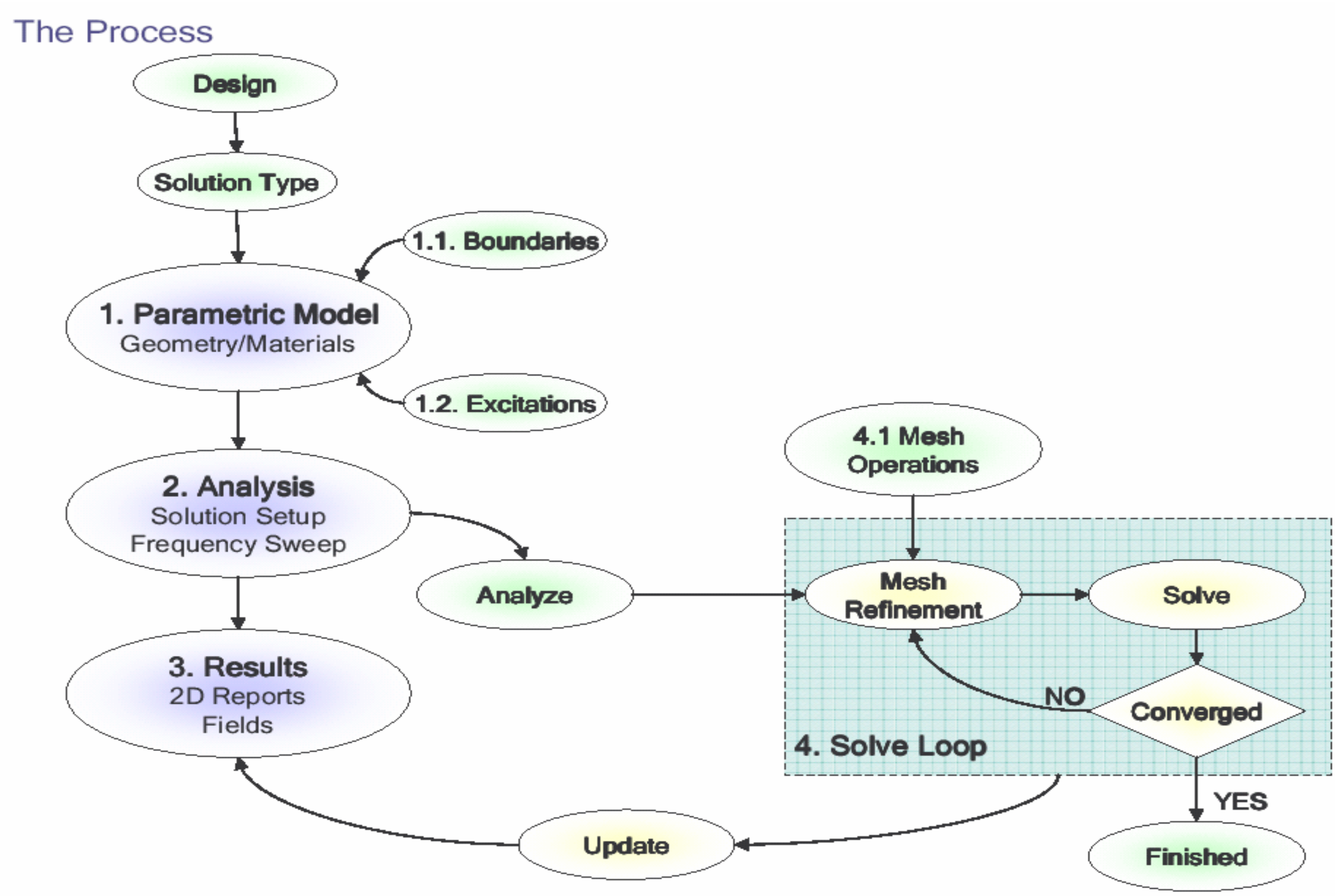
Starting Ansoft HFSS

Click theMicrosoft **Start** Button ,Select **Programs** and Select the **Ansoft>HFSS9>HFSS9**
or Double click the **HFSS9 icon** on the desktop

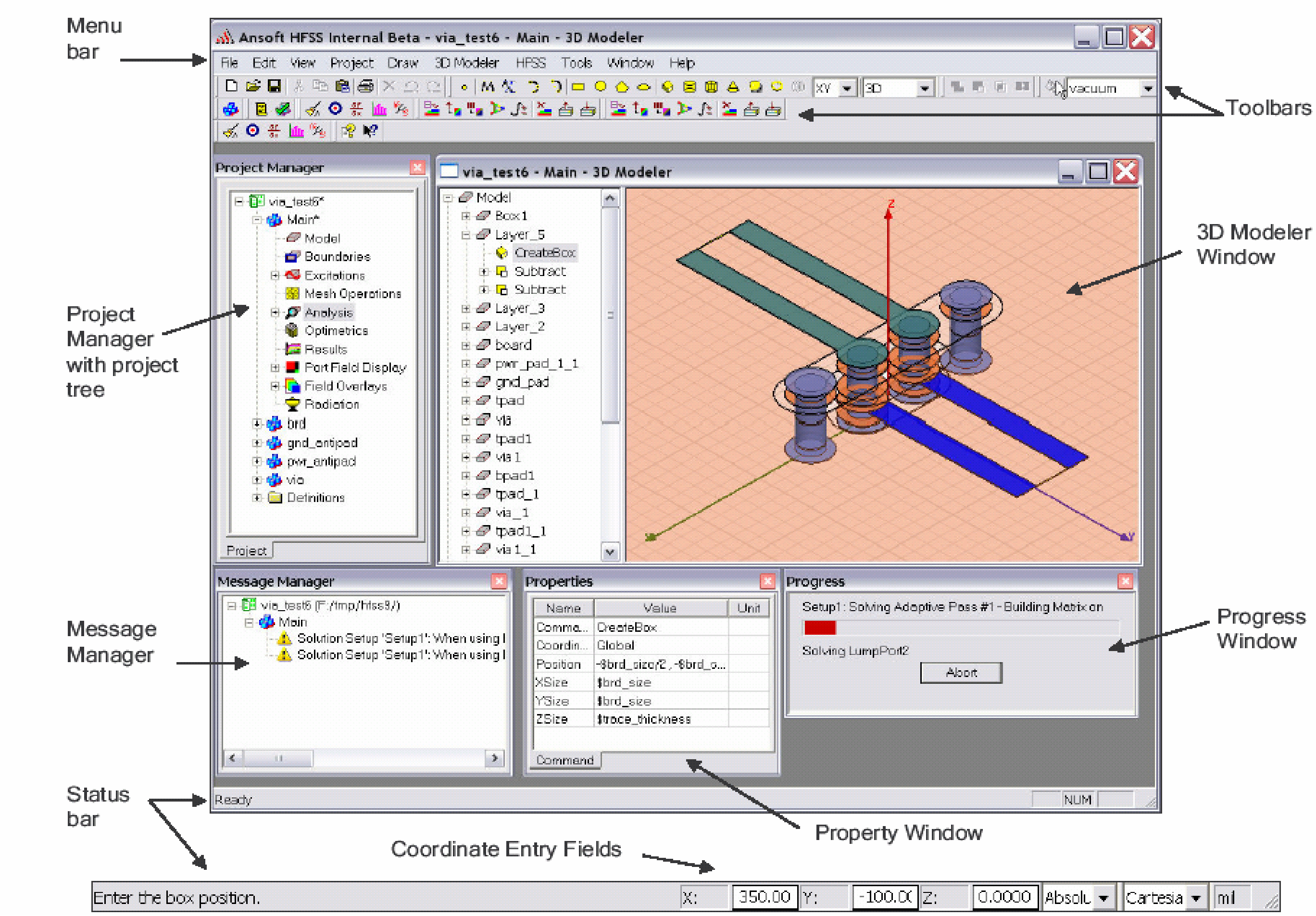


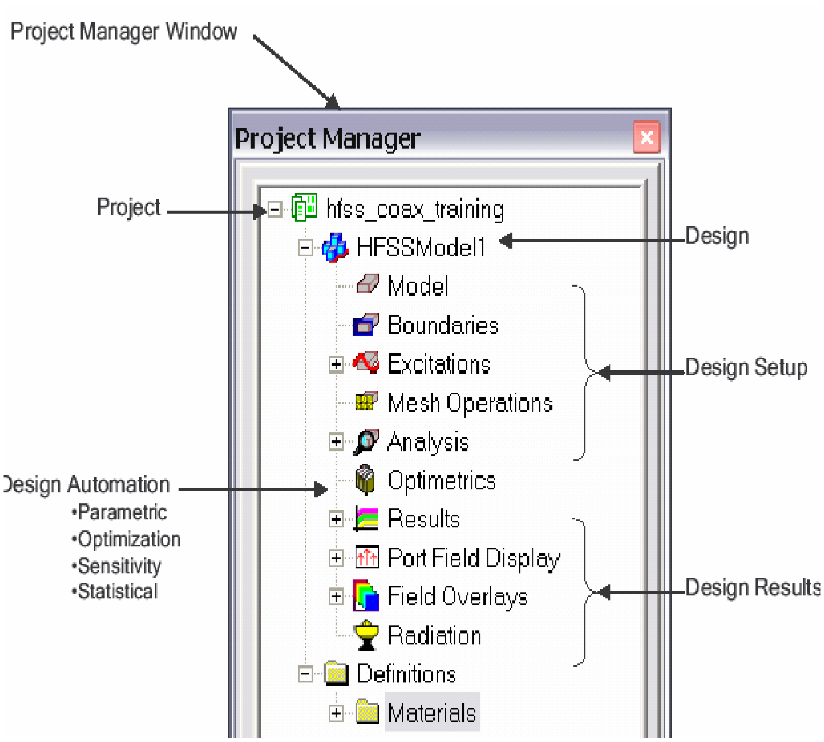
Creating Projects:

On the **File** menu, click **New.** You specify the name of the project when you save it using the **File>Save** or **File>Save As.**Open a previously saved project using the **File>Open** command



Ansoft Desktop



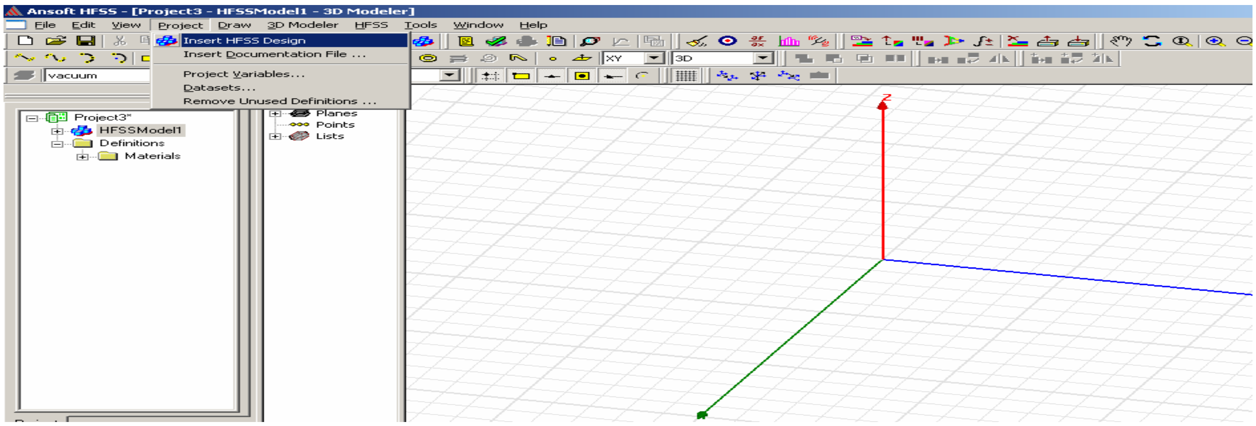


To set up an HFSS design, follow this general procedure. Note that after you insert a design, you do not need to perform the steps sequentially, but they must be completed before a solution can be generated.

I - Insert an HFSS design into a project.

1) On the **Project** menu, click **Insert HFSS Design**

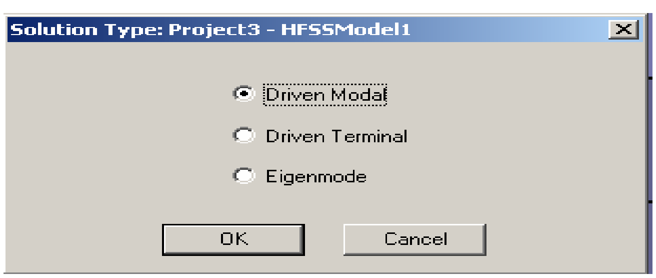
The new design is listed in the project tree. It is named HFSSDesign*n* by default, where *n* is the order in which the design was added to the project. The **3D Modeler** window appears to the right of the Project Manager. You can now create the model geometry



II -Selecting the Solution Type

Before you draw the model, specify the design's solution type.

1) On the **HFSS** menu, click **Solution Type**. The **Solution Type** dialog box appears.



2) Select Driven Modal in the solution types.

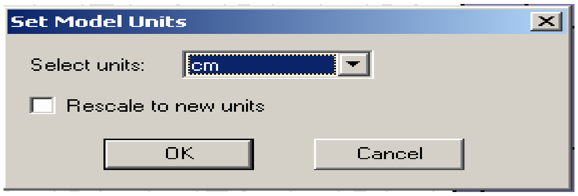
We select Driven Modal as our model is a rectangular waveguide and Driven modal is used for calculating the mode-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines, which are "driven" by a source

III- Setting the Model's Units of Measurement

You can then choose to display the model's dimensions in the new units, or rescale the model's dimensions to the new units.

To set the model's units of measurement:

1. On the **3D Modeler menu, click Units**. The **Set Model Units** dialog box appears.



2. Select the new units for the model from the **Select units** pull-down list.

You can select the **Rescale to new units** option to rescale the dimensions to the new units. Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale

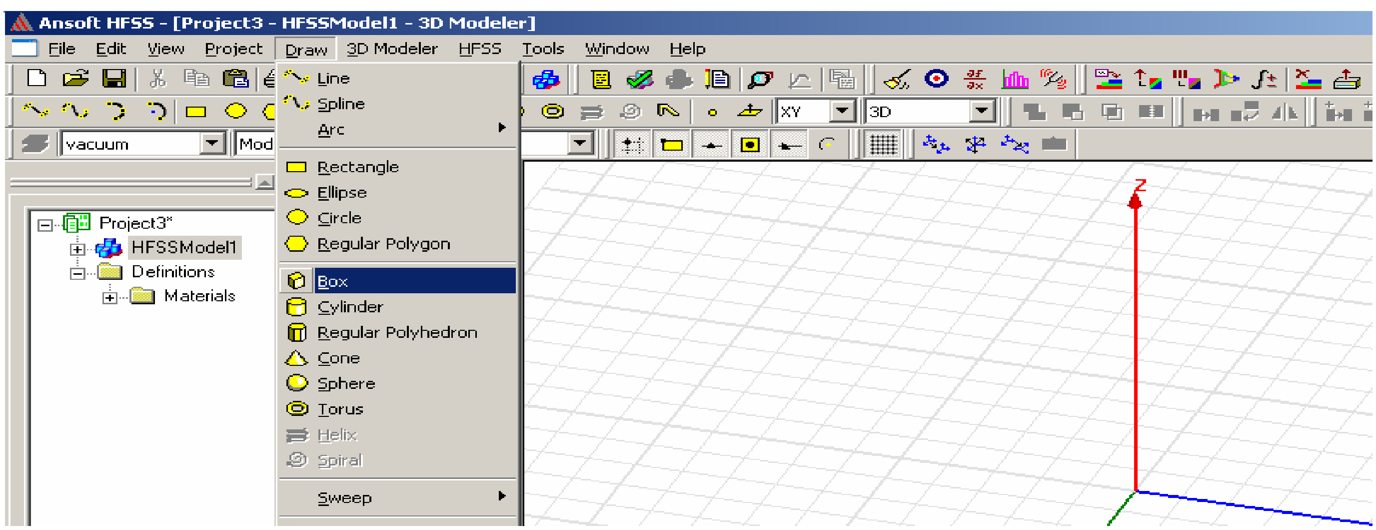
3. Click **OK** to apply the new units to the model.

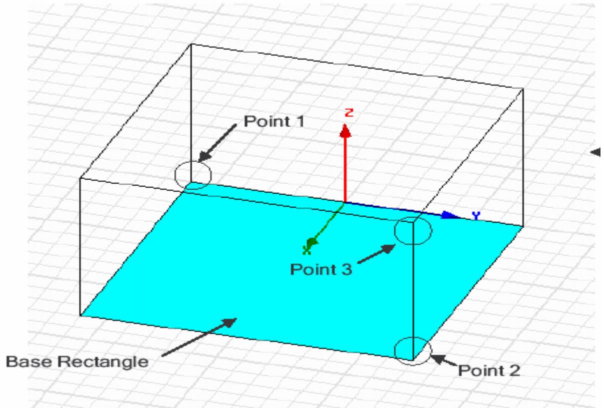
IV- Drawing a Model

You can create 3D objects by using HFSS's **Draw** commands. Objects are drawn in the **3D Modeler** window.

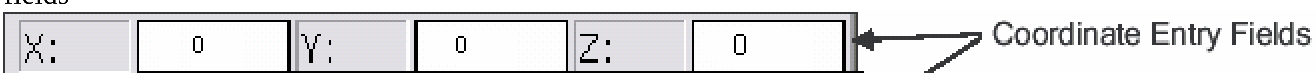
To draw a WR-90 Rectangular waveguide ,

1. On the **HFSS** menu, click **Draw**. The **Draw** dialog box appears.Select **Box**

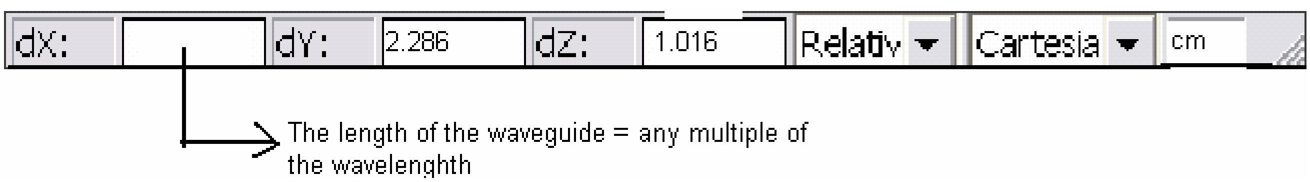




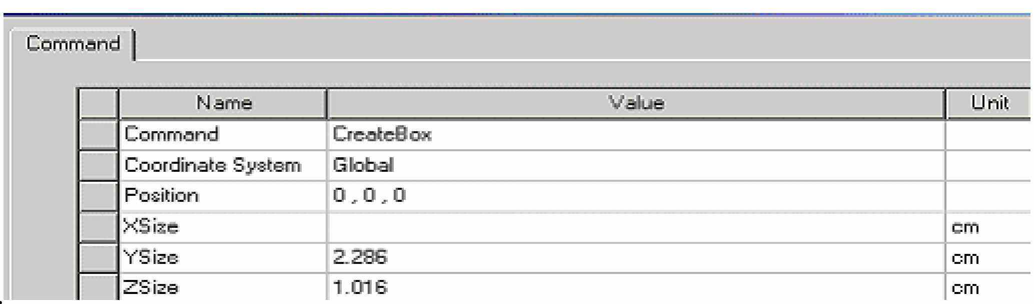
2. Dimensions of the box can be specified while drawing the box .At the lower end of the screen on the right is the Coordinate entry
fields



3. Enter the Initial XYZ coordinates and then enter the length in XY&Z direction in dX,dY&dZ .
For e.g to draw the box with initial point to be origin and propogation along X axis. Since it is a WR-90 rectangular waveguide the dimensions are **a=2.286cm , b=1.016cm**.



Once you Draw the box the properties window opens up , you can also specify the coordinates and size of the box here

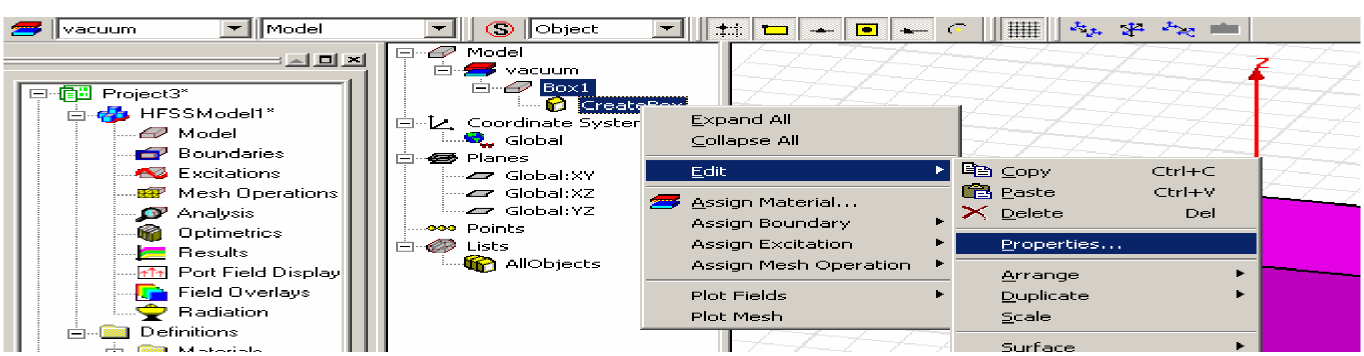


Command

Name	Value	Unit
Command	CreateBox	
Coordinate System	Global	
Position	0 , 0 , 0	
XSize		cm
YSize	2.286	cm
ZSize	1.016	cm

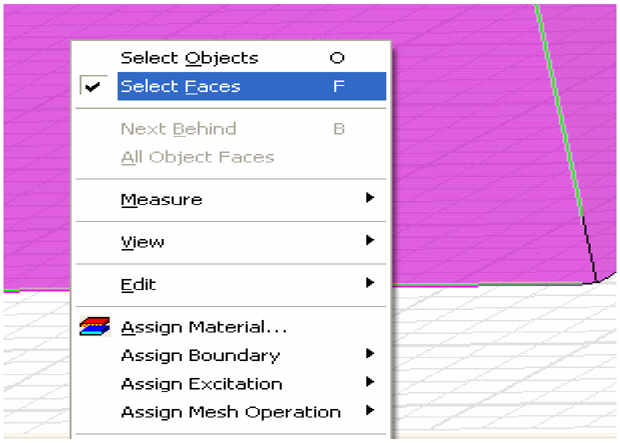
.

The properties window can also be obtained by

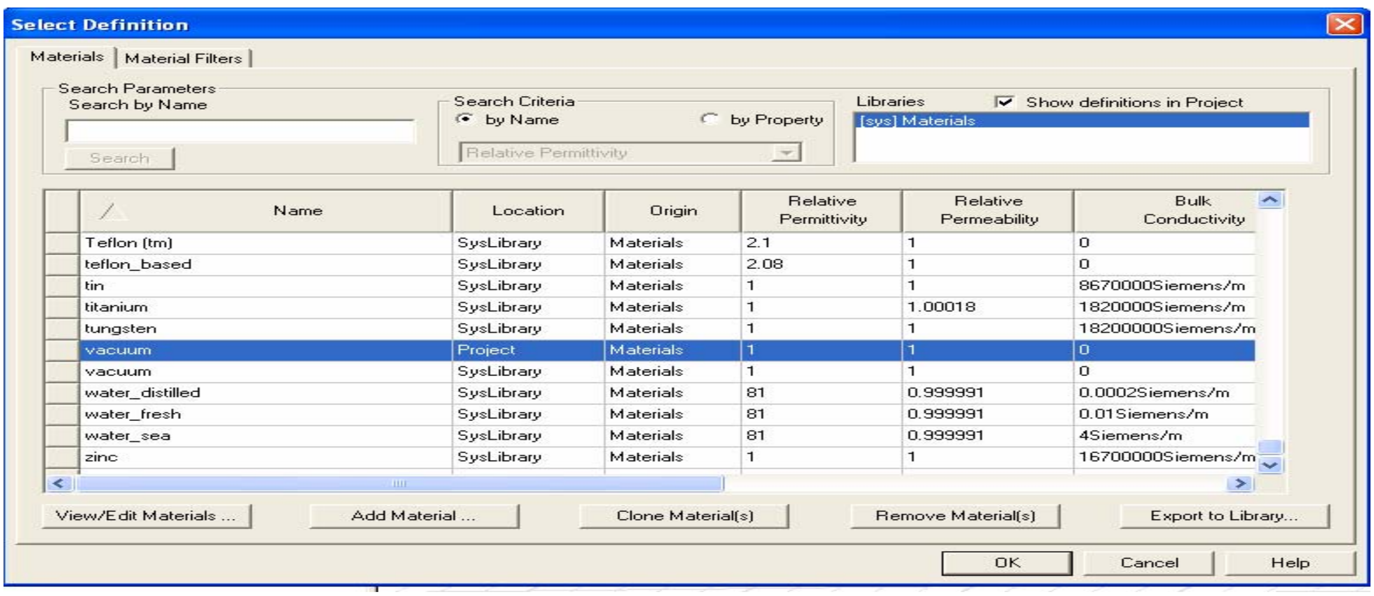


V- Assigning Materials

1) Right click on the 3D Modeler Window to get the 3D Modeler menu
2) On the 3D Modeler menu, click Assign Material.



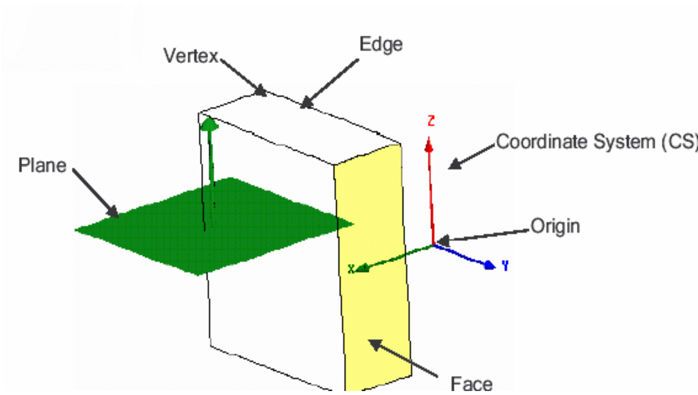
3) The **Select Definition** window appears. By default, it lists all of the materials in Ansoft's global material library as well as the project's local material library.



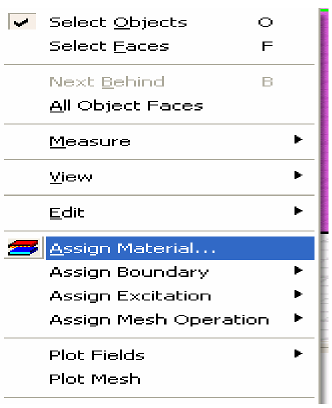
4.Select a material from the list. Select Air or vacuum for the whole box as our rectangular waveguide is not filled with any dielectric.
5.Click **OK**.
6. The material you chose is assigned to the object.

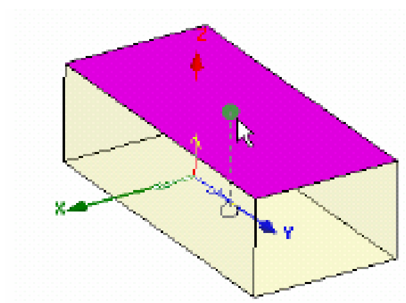
VI- Assigning Boundaries

Boundary conditions specify the field behavior at the edges of the problem region and object interfaces.



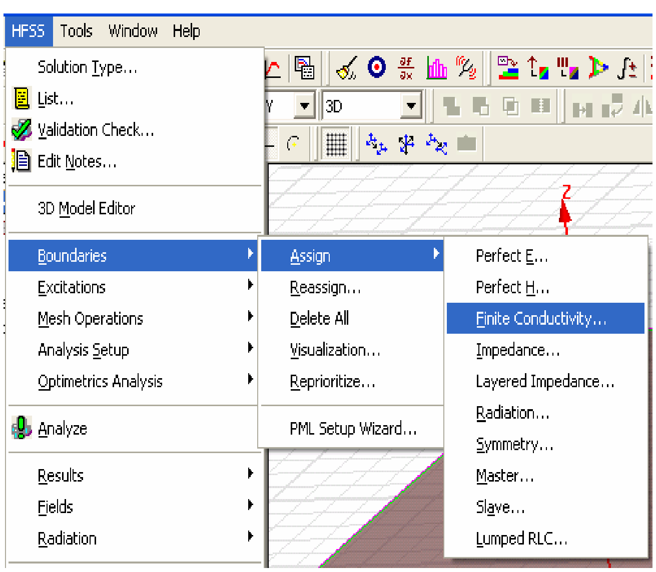
1) Right click on the 3D Modeler Window to select faces

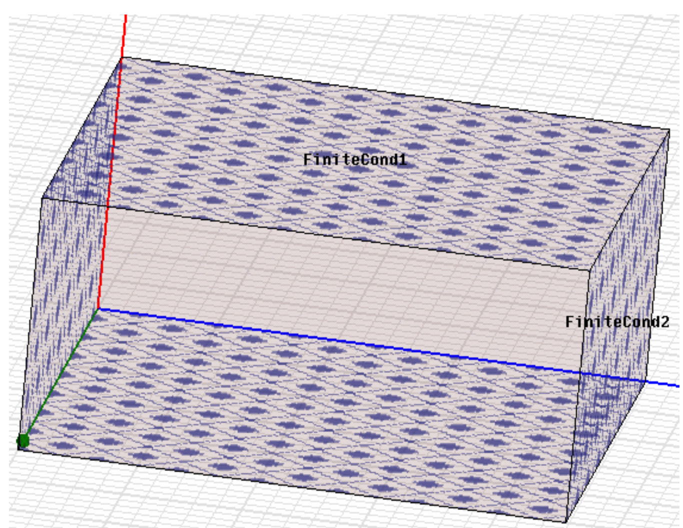




Select Face

2) Click on the faces to select the faces which are to be assigned to be a perfect conductor
3) On the **HFSS** menu, click **Boundaries**.Select **Assign** and choose **Finite conductivity**.





Assign **Finite conductivity** to 4 faces excluding the Port 1 and Port 2

VII- <u>Assigning Excitations</u>

Excitations in HFSS are used to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces in the design.

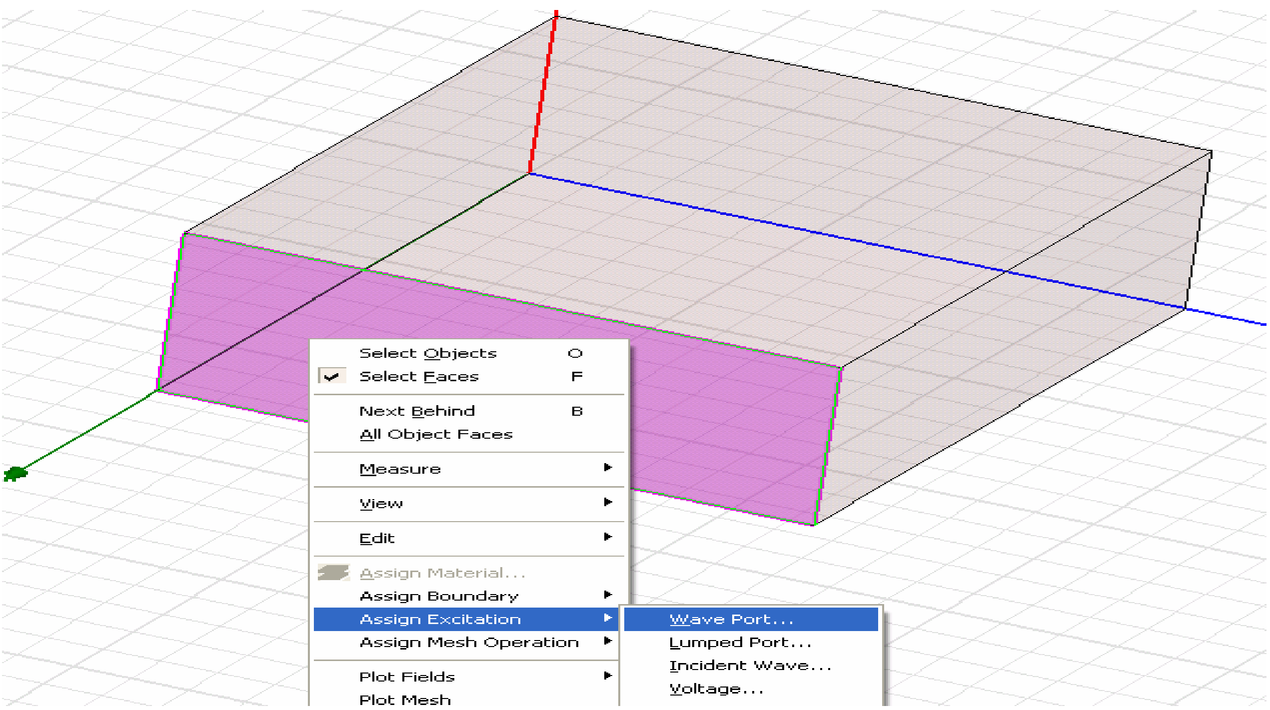
Assigining excitations is a two step process

VII. a) Assign Ports
b)Assign an Intergration Lines or Terminal lines separately for each modes

a) Assigning Ports

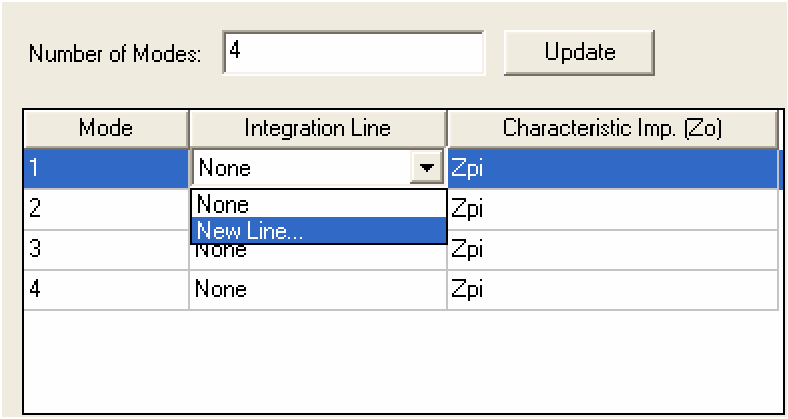
1. Select the object face to which you want to assign the port.
2. Click H FSS>Excitations>Assign>Wave Port.

Wave port represents the surface through which a signal enters or exits the geometry. Hence 2 ports are required to be defined. HFSS assumes that each wave port you define is connected to a semi-infinitely long waveguide that has the same cross-section and material properties as the port. HFSS generates a solution by exciting each wave port individually.



3. The Wave Port wizard appears.
4. Type the port's name in the **Name** text box or accept the default name, and then click **Next**.

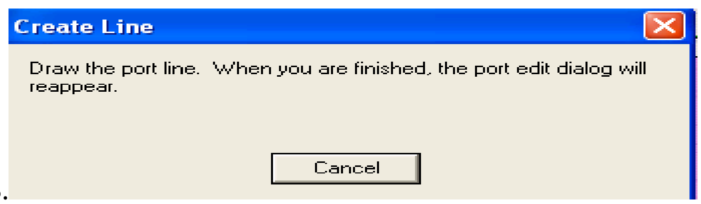
5. To specify more than one mode to analyze at the port, type a new value in the **Number of Modes** box, and then click **Update**. The mode spreadsheet is updated to include the total number of modes



b) Defining Integration Line

An integration line needs to be specified to define a port mode**.** Since we are analysing the WR-90 waveguide for the first 4 modes we need to specify 4 intergration lines

1. Select **New Line** from the mode's **Integration Line** list.
2.The dialog box disappears while you draw the vector

3.


4. Select the start point of the vector in one of the following ways
   4.a. Click the point. **Or** Type the point's coordinates in the **X**, **Y**, and **Z** boxes at the lower end of the screen
5. Select the endpoint of the vector using the mouse or the keyboard. The endpoint defines the direction and length of the integration line.
6. The **Wave Port** or **Lumped Port** dialog box reappears.

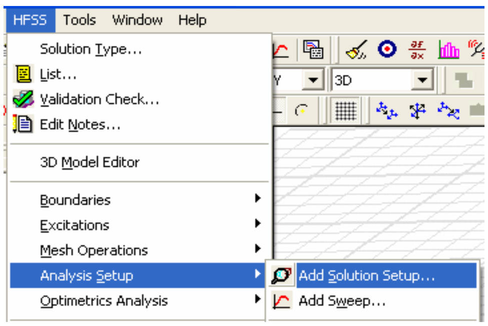
.

VIII- Solution Setup

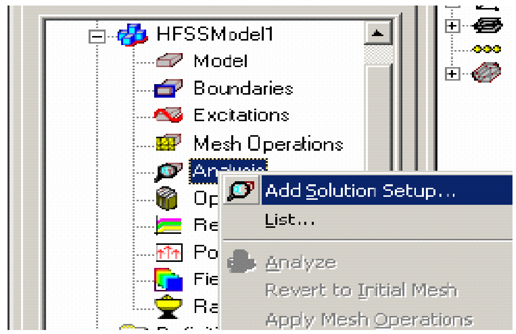
a) Adaptive solution setup
b) Frequency sweep setup

Adaptive solution setup

1. On the **HFSS** menu, point to **Analysis Setup**, and then click **Add Solution Setup**



Or



2. The **Solution Setup** dialog box appears. It is divided among the following tabs:

General - Includes general solution settings
Advanced - Includes advanced settings for initial mesh generation and adaptive analysis
Ports - (if a port was defined) Includes mesh generation options for model ports
Defaults - Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

3. Click the **General** tab.
3.a For Driven solution types, do the following:

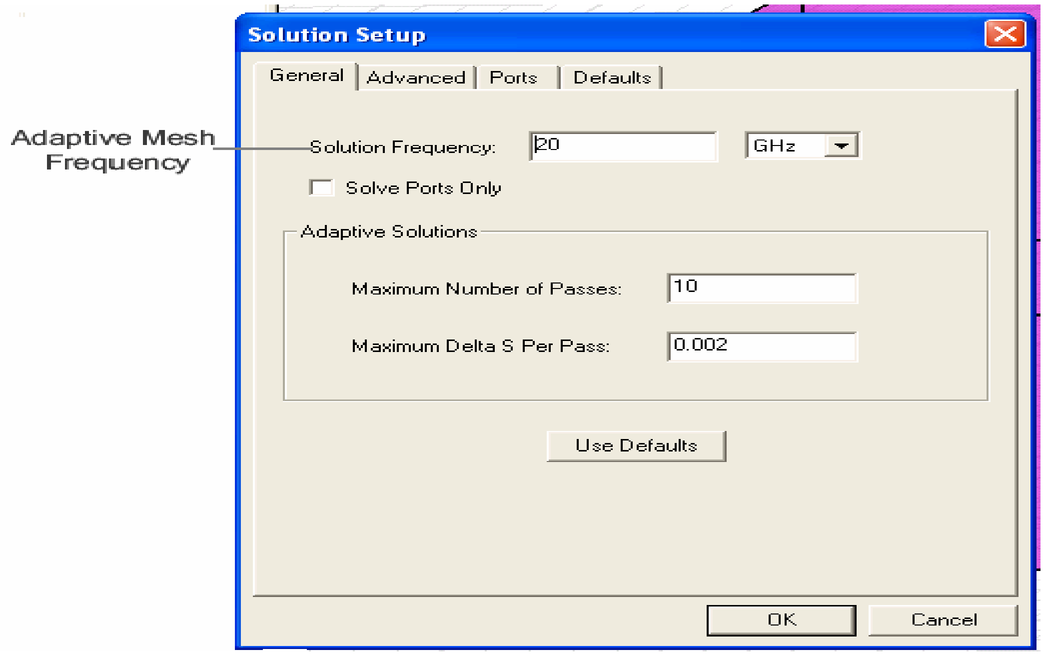
1. Enter the **Solution Frequency** in the frequency units.
The minimum value for adaptive Mesh Frequency is 2/3rd of the final frequency required .Although it is **recommended** to just adapt to the Final frequency.
Since we are analyzing the first 4 modes of the WR-90 waveguide the Cut-Off Frequency of the last mode is 16 Ghz. Hence the Final Frequency is a value, which is higher than that. For e.g. 20Ghz

2 .Enter the **Maximum Number of Passes = 10**

The Maximum Number of Passes value is the maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis will continue unless the convergence criteria are reached

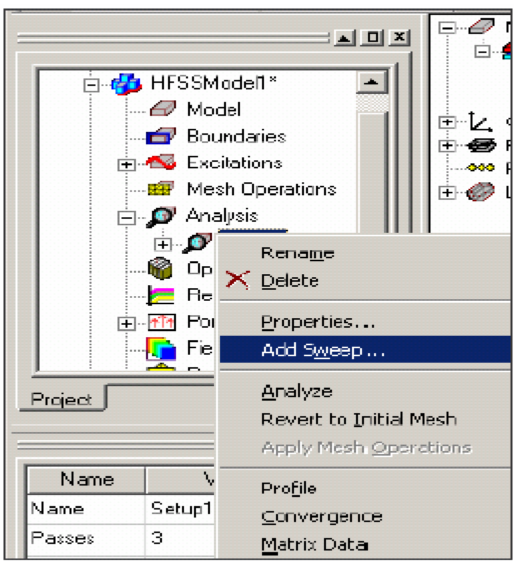
3. Enter the **Delta s =. 002**
The delta S is the change in the magnitude of the S-parameters between two consecutive passes.

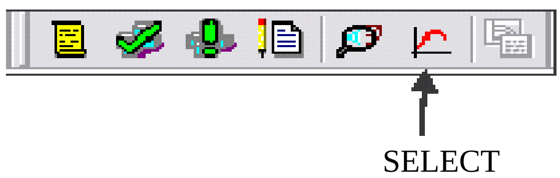
4.Click **Ok**



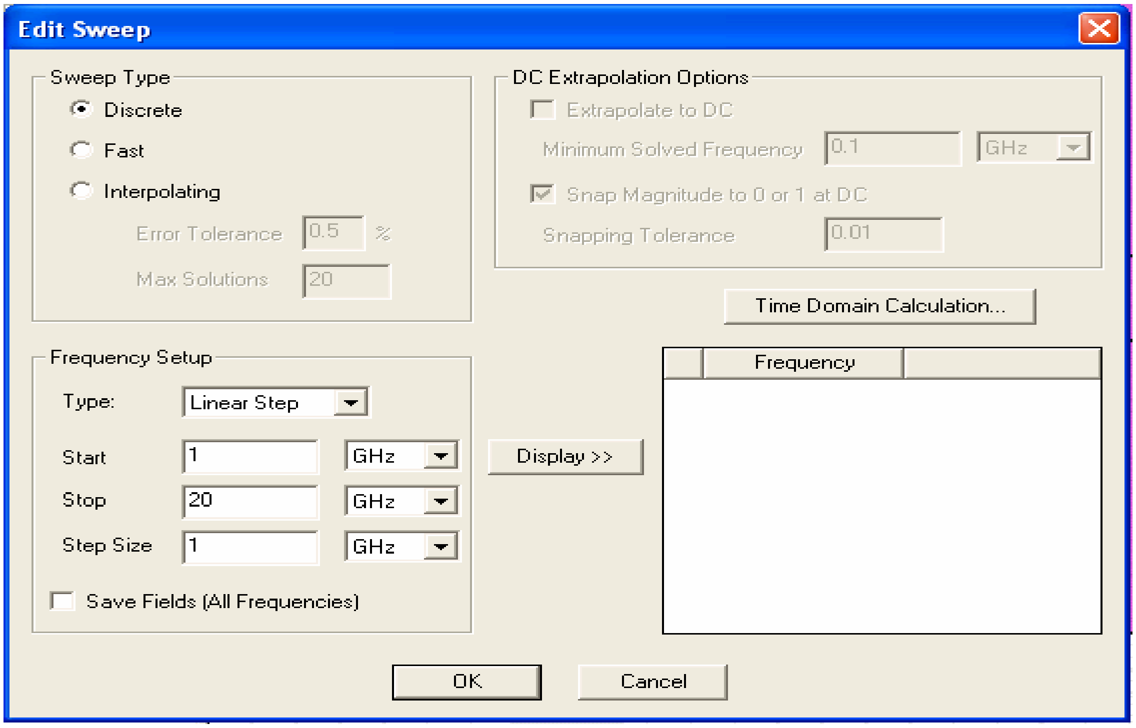
Frequency Sweep setup

1. In the **HFSS** menu Select **Analysis Setup** and then select **Add sweep**





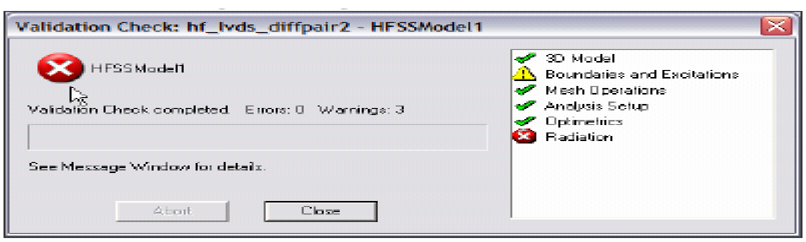
2.The **Edit Sweep** Dialog Box opens.
3.Select **Discrete** and enter the **Start** and **stop** Frequency.
Since we are analyzing the first 4 modes of the WR-90 waveguide the Cut-Off Frequency of the last mode is 16 Ghz. Hence the Stop Frequency is a value, which is higher than that. For e.g. 20Ghz
4.Click **OK**



VIII – Running a Simulation

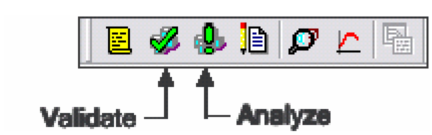
To validate your model

1. Select **HFSS** menu > **Validate Check**
2. Click **OK**



To **Analyze**

1. On the **HFSS** menu, click **Analyze**



While a simulation is running, you can monitor the solution's progress in the **Progress** window.

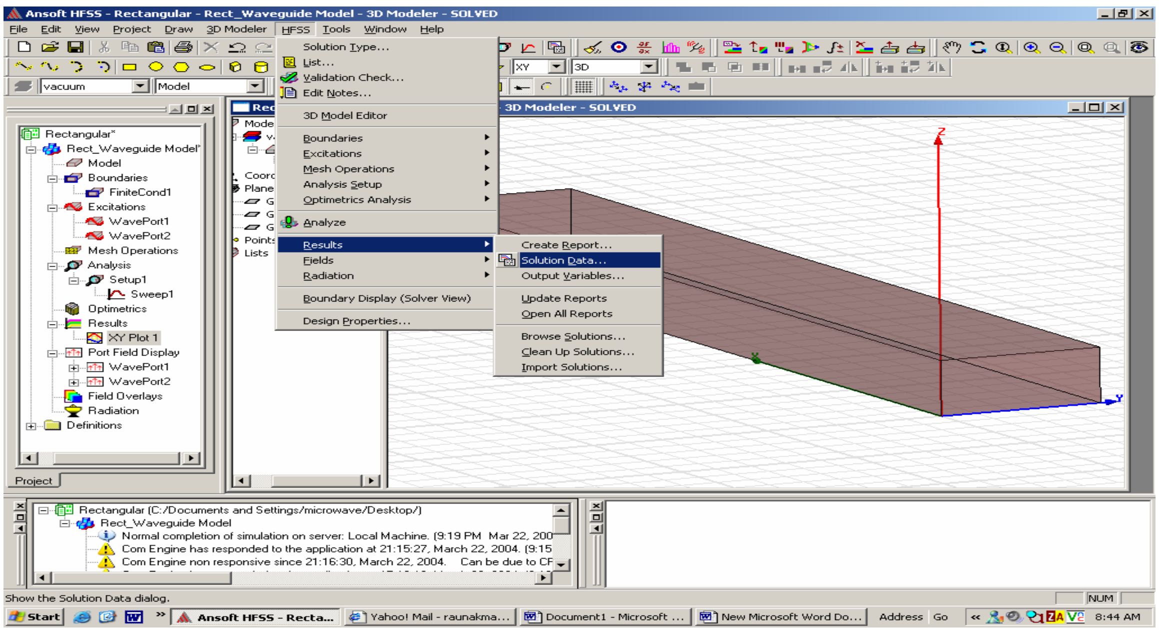
You can also view the following solution data at any time during or after the solution

- Convergence data-- by clicking **HFSS>Analysis Setup>Convergence**.
- Matrices computed for the S-parameters, impedances, and propagation constants by clicking **HFSS>Analysis Setup>Profile**.

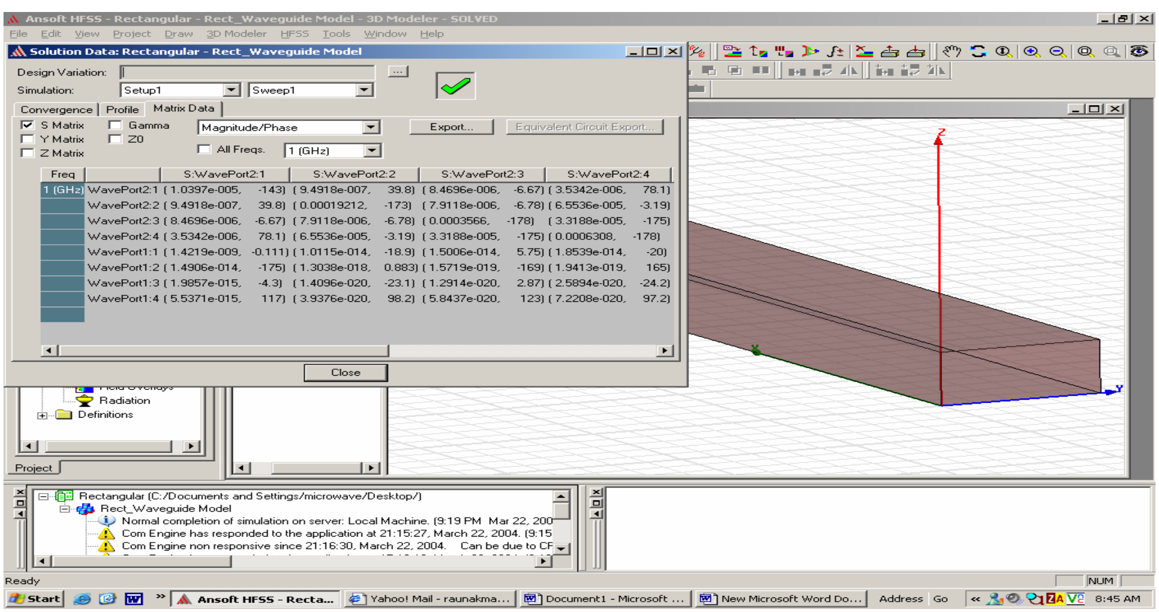
Once the simulation is completed HFSS Informs you in the message window.

Results

HFSS > Results > Solution Data



The solution data window appears



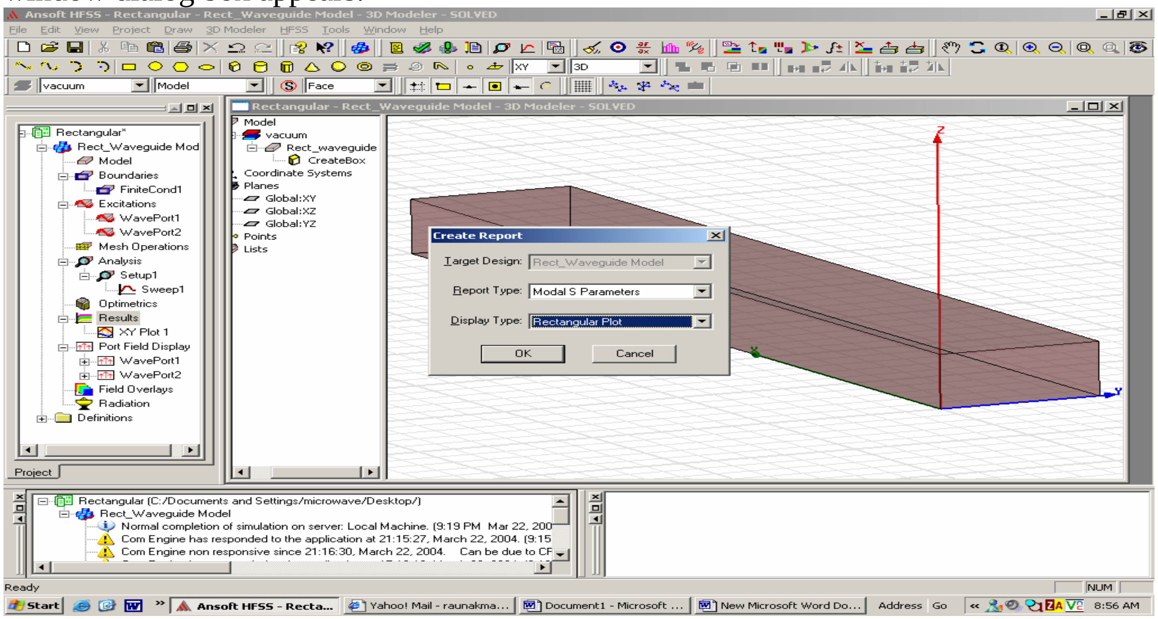
HFSS computes the following matrix data

- *S, Y, and Z Parameters*
- *VSWR*
- *Excitations: - Gamma and Zo*

Plotting the results

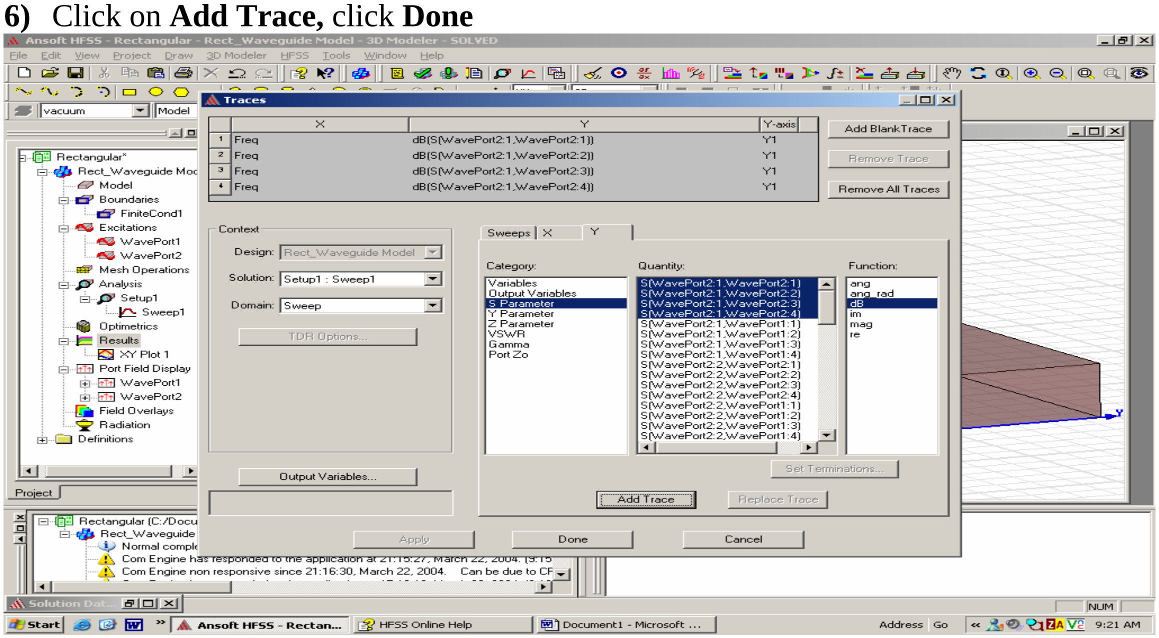
HFSS> Results> Create Report

Or you can also go to the project tree and right click on results and click create report. The Create report window dialog box appears.



1) Select the report type you want to view from the pull down list on the top of the dialog box
2) Select the type of plot you want to create, from the display type pull down list.
3) Click **OK**

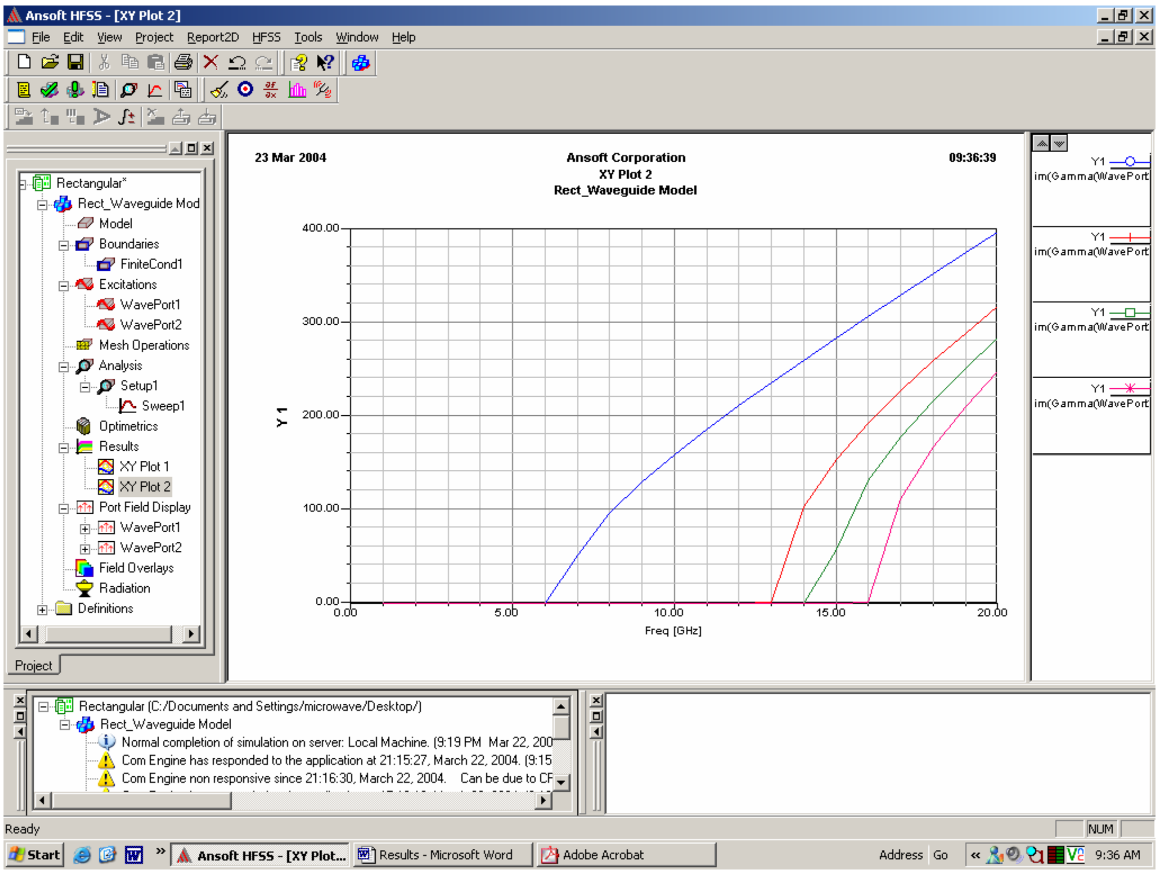
   The **Traces** dialog box appears
4) In the **Solution** list, click the solution containing the data you want to plot.
5) In the domain list, click a domain. For modal and terminal S- parameter reports, the domain can be frequency or time. *In this case we want frequency domain.*
6) Click on **Add Trace,** click **Done**



Analysis and Results

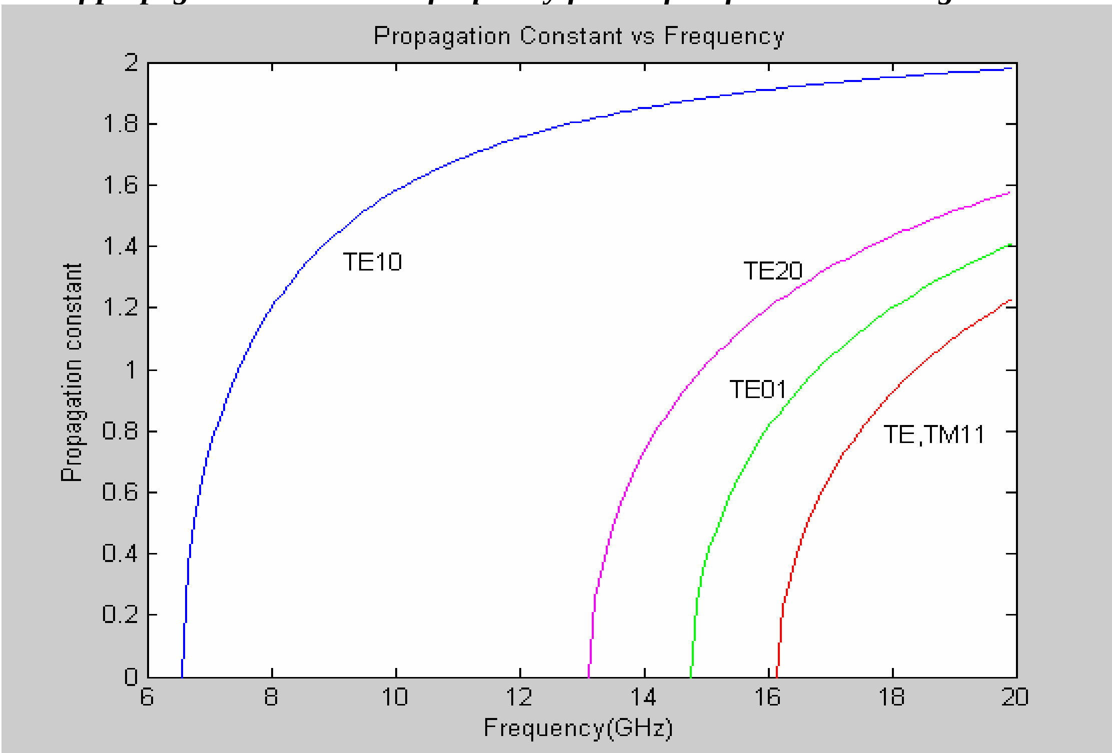
1) Analyze the propagation constant for the first four modes.
2) Determine the wavelength at different frequencies for the first four modes.
3) Determine the intrinsic impedance at different frequencies for the first four modes
4) Analyze E and H field patterns for the first four modes.

Plot of propagation constant vs. frequency for TE10, TE20, TE01, TE and TM11 using HFSS



Note that the mode propagates only when the propagation constant has is real and the operating frequency is greater than its cutoff frequency. As the traveling waves are functions of exp(-j β z), has to be real and make exp(-j β z) imaginary.

Plot of propagation constant vs. frequency for the first four modes using Theoretical values



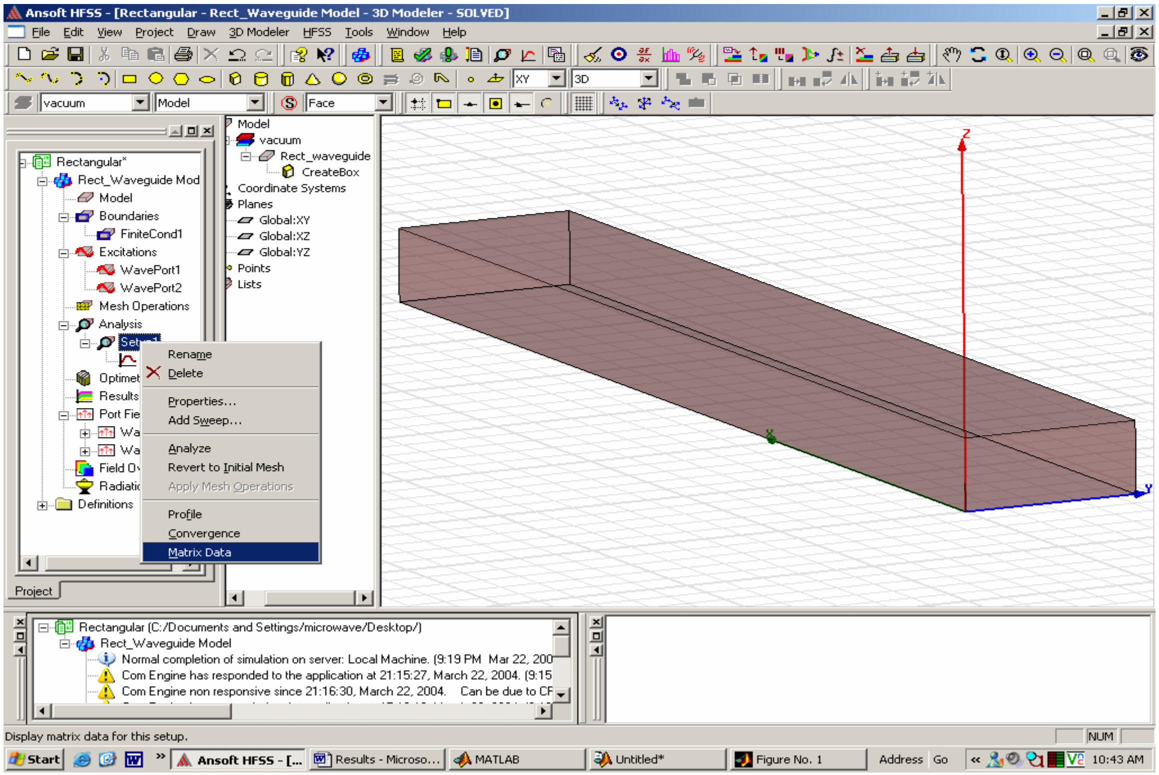
Similarly you can draw plots for wavelength vs. frequency and impedance vs. frequency from the data given in the solution data box.

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.The **Solution Data** dialog box appears. The **Matrix Data** tab is selected.

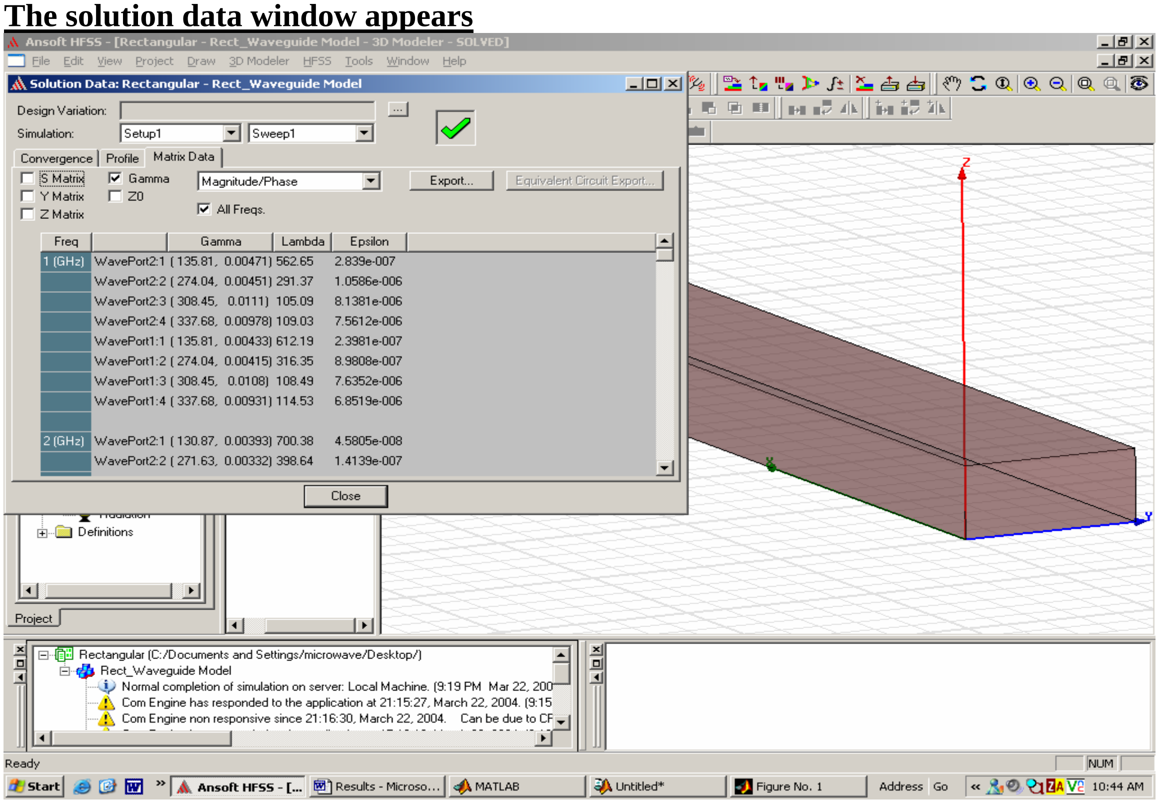
2. In the **Simulation** pull-down list, click the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.

3. Select the type of matrix you want to view: S,Y, and Z matrices or $\mathbf{Z}_o$ (characteristic impedance.). The wavelength data is displayed when you check the gamma box. The available types depend on the solution type.

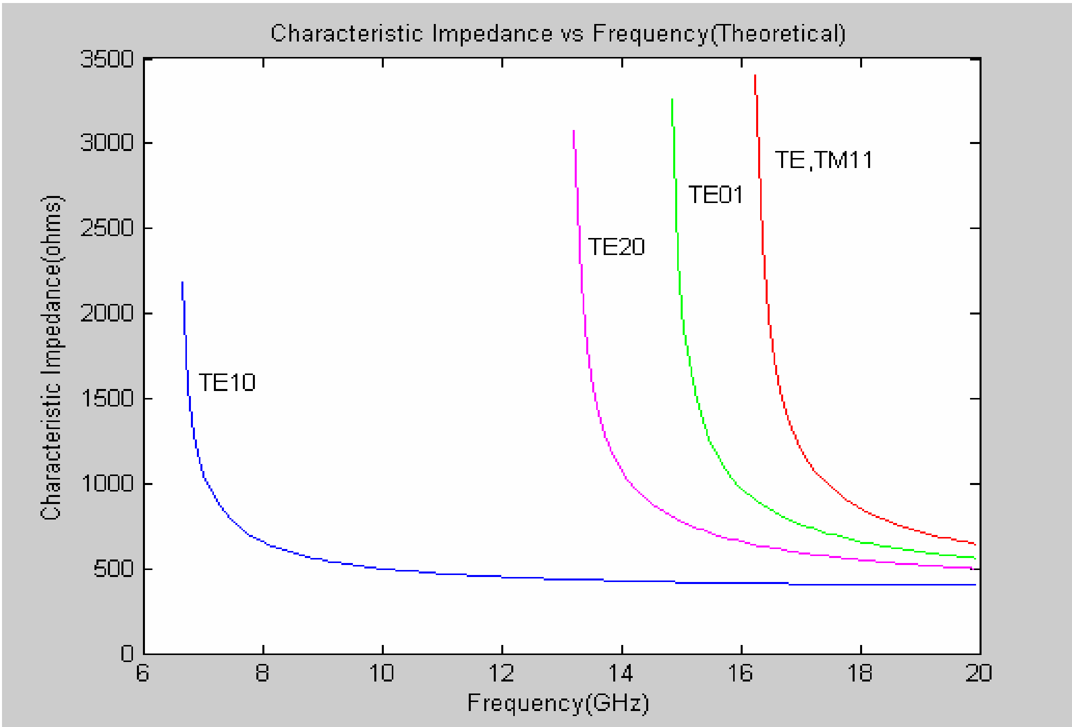
4. Select the format — **Magnitude/ Phase**, **Real/ Imaginary**, **dB/ Phase**, **Magnitude**, **Phase**, **Real**, **Imaginary**, or **dB** — in which to display the matrix information. The available formats depend on the matrix type being displayed.



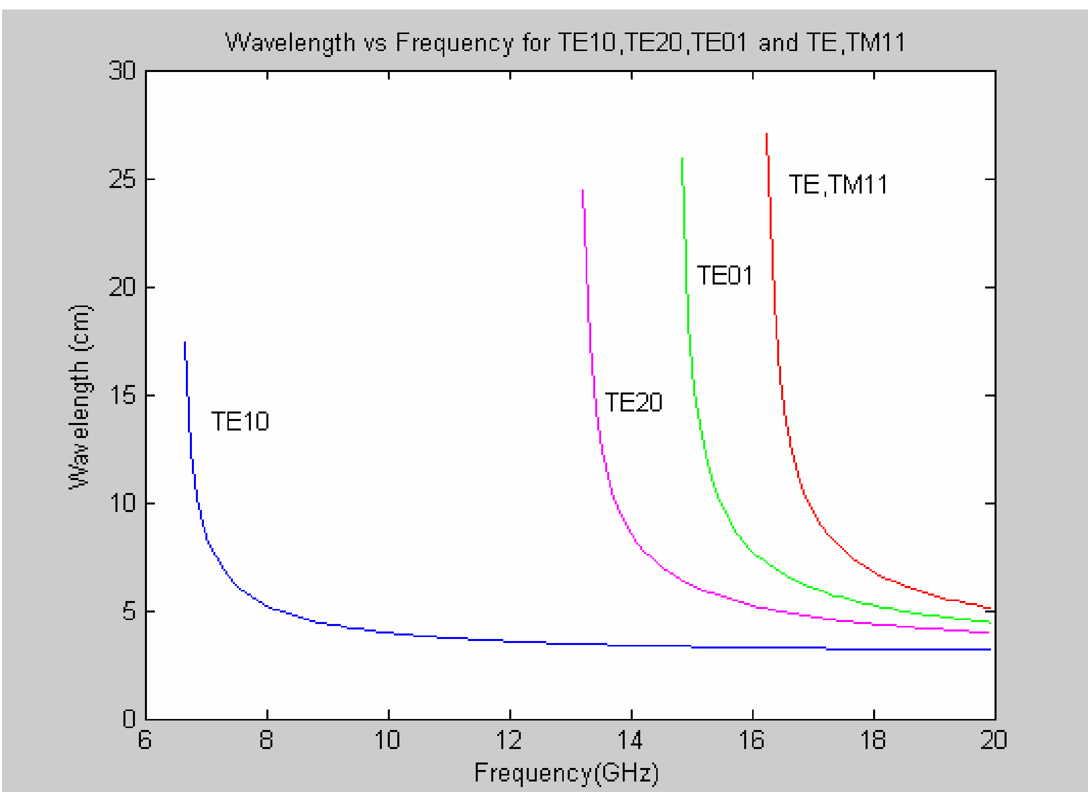
<u>The solution data window appears</u>



Impedance vs. Frequency for the first four modes

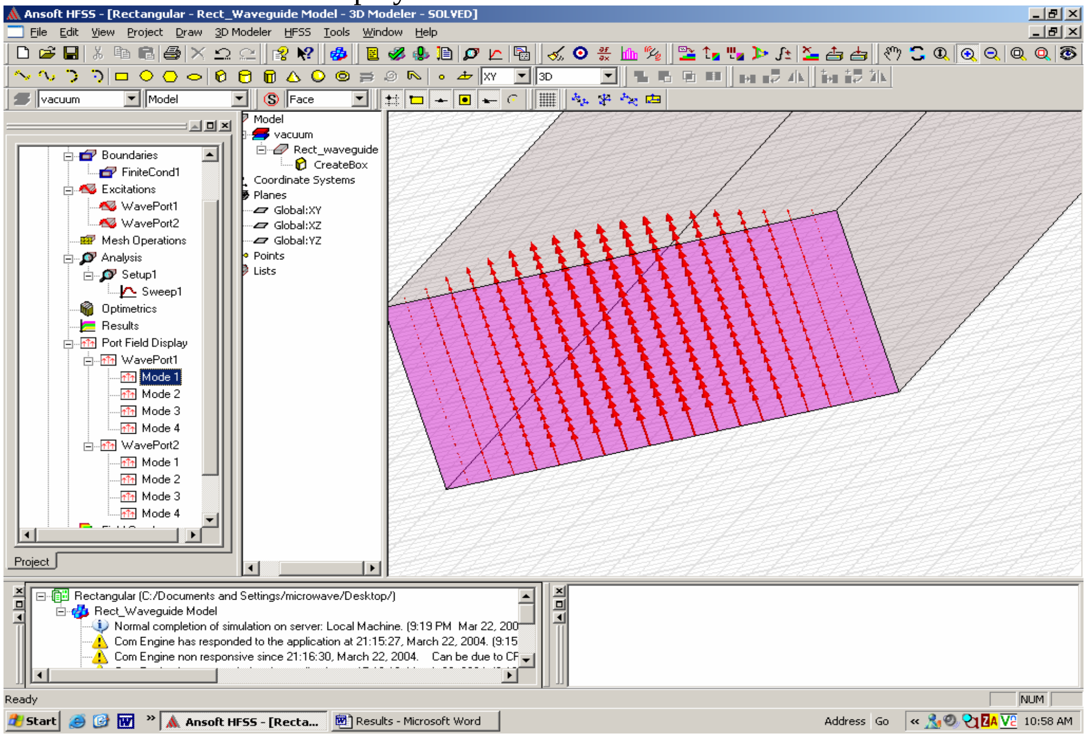


Wavelength vs. frequency for the first four modes

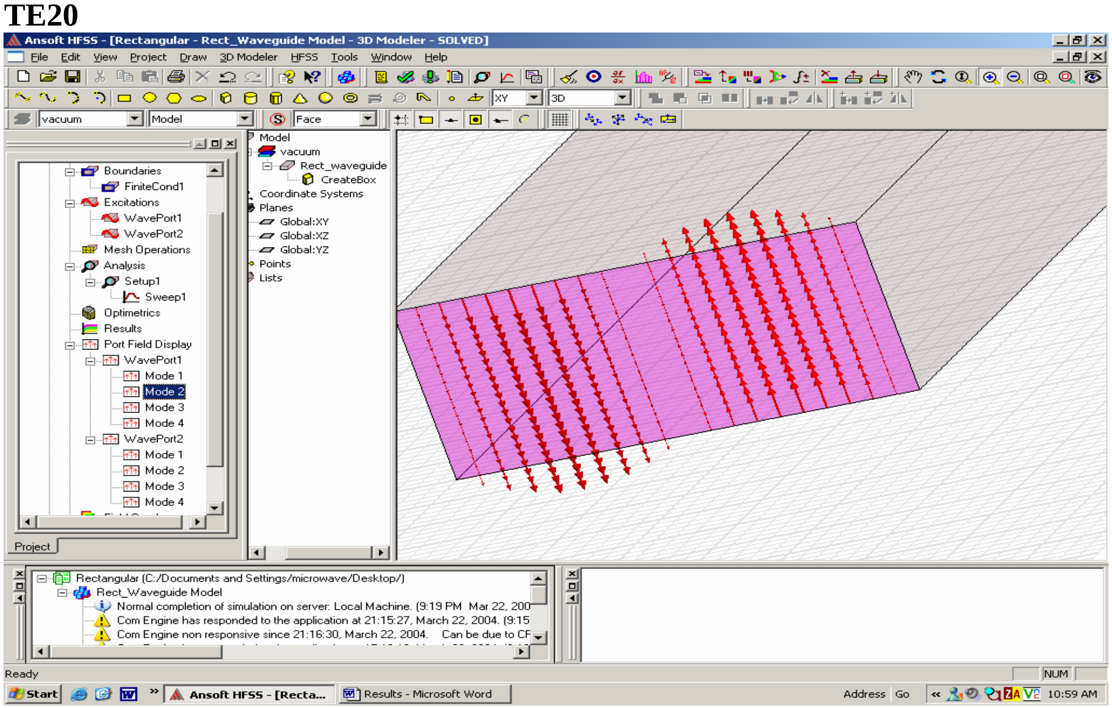


Field Patterns for E and H fields

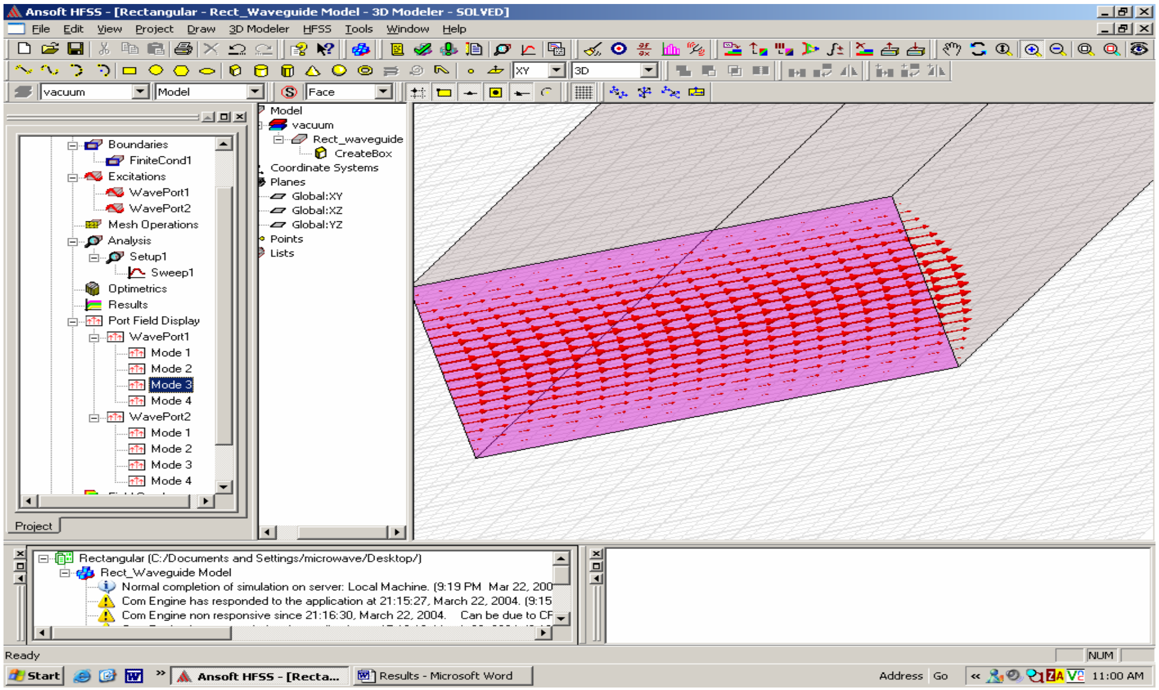
On the project tree click on **Port Field Display>wave port 1>mode1**
The TE10 mode will be displayed in the model



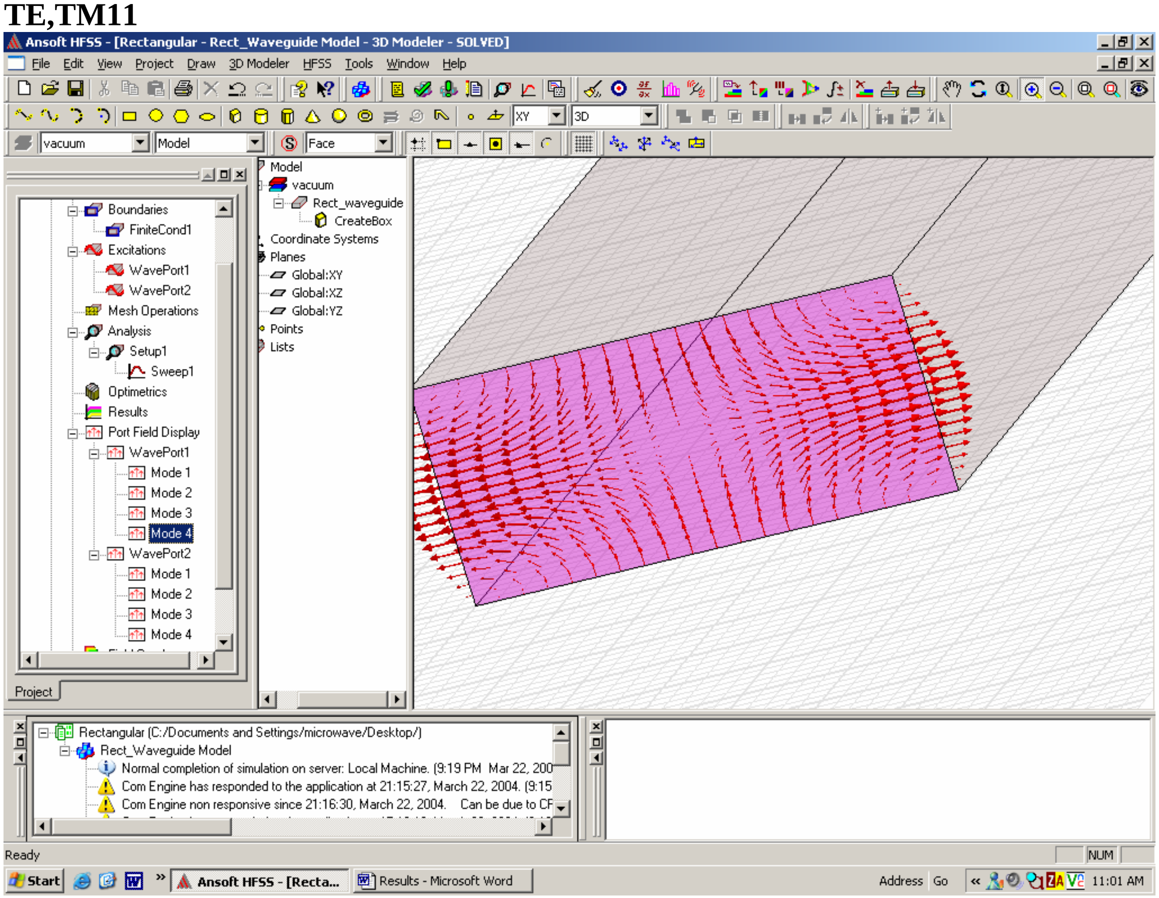
Similarly click on mode2, mode3 and mode 4
TE20



TE01

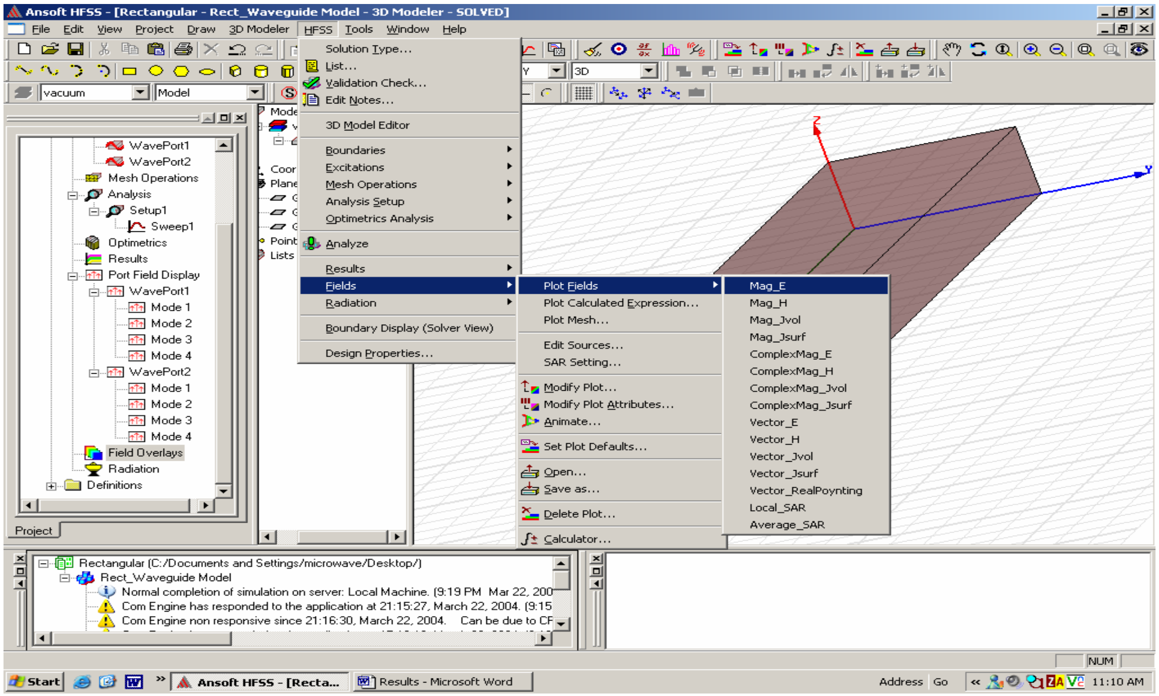

Ansoft HFSS - [Rectangular - Rect_Waveguide Model - 3D Modeler - SOLVED]
File Edit View Project Draw 3D Modeler HFSS Tools Window Help
Boundaries
FiniteCond1
Excitations
WavePort1
WavePort2
Mesh Operations
Analysis
Setup1
Sweep1
Optimetrics
Results
Port Field Display
Mode 1
Mode 2
Mode 3
Mode 4
Model
vacuum
Rect_waveguide
CreateBox
Coordinate Systems
Planes
Global:XY
Global:XZ
Global:YZ
Points
Lists
Rectangular (C:/Documents and Settings/microwave/Desktop/)
Rect_Waveguide Model
Normal completion of simulation on server: Local Machine. (9:19 PM Mar 22, 200
Com Engine has responded to the application at 21:15:27, March 22, 2004. (9:15
Com Engine non responsive since 21:16:30, March 22, 2004. Can be due to CF
Ready
NUM
Start
Ansoft HFSS - [Recta...
Results - Microsoft Word
11:00 AM

TE,TM11

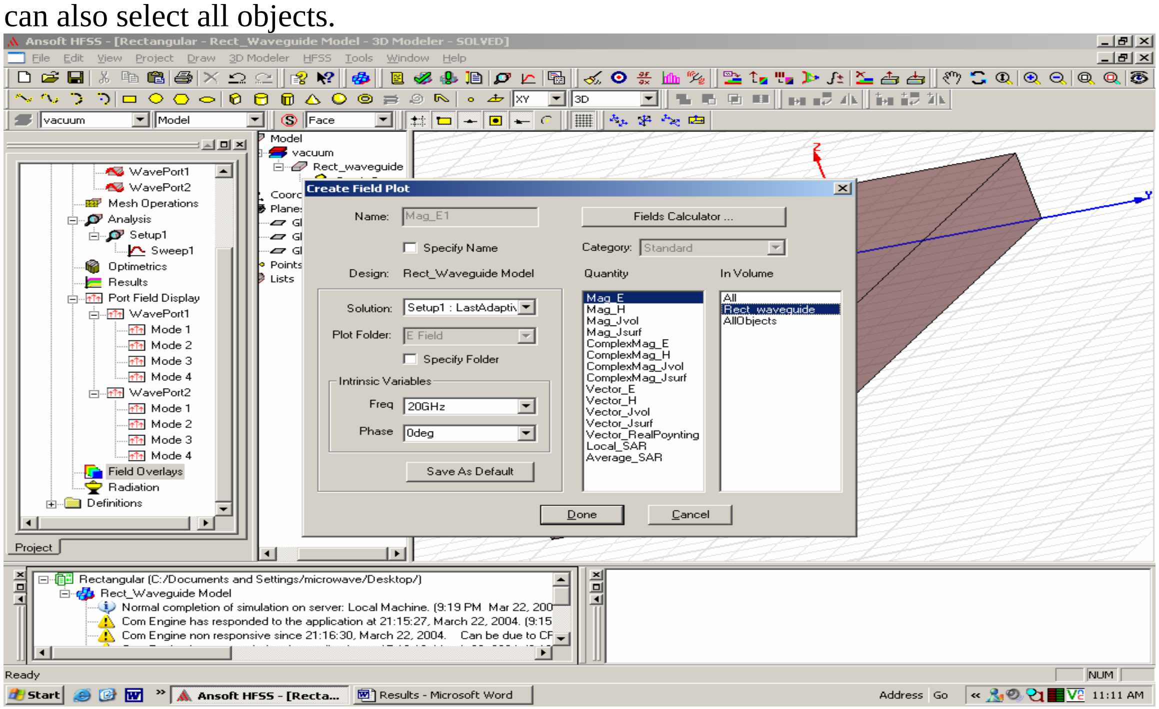

Ansoft HFSS - [Rectangular - Rect_Waveguide Model - 3D Modeler - SOLVED]
File Edit View Project Draw 3D Modeler HFSS Tools Window Help
Port Field Display
Mode 4
Rectangular (C:/Documents and Settings/microwave/Desktop/)
Rect_Waveguide Model
Ready
11:01 AM

To view E and H field patterns

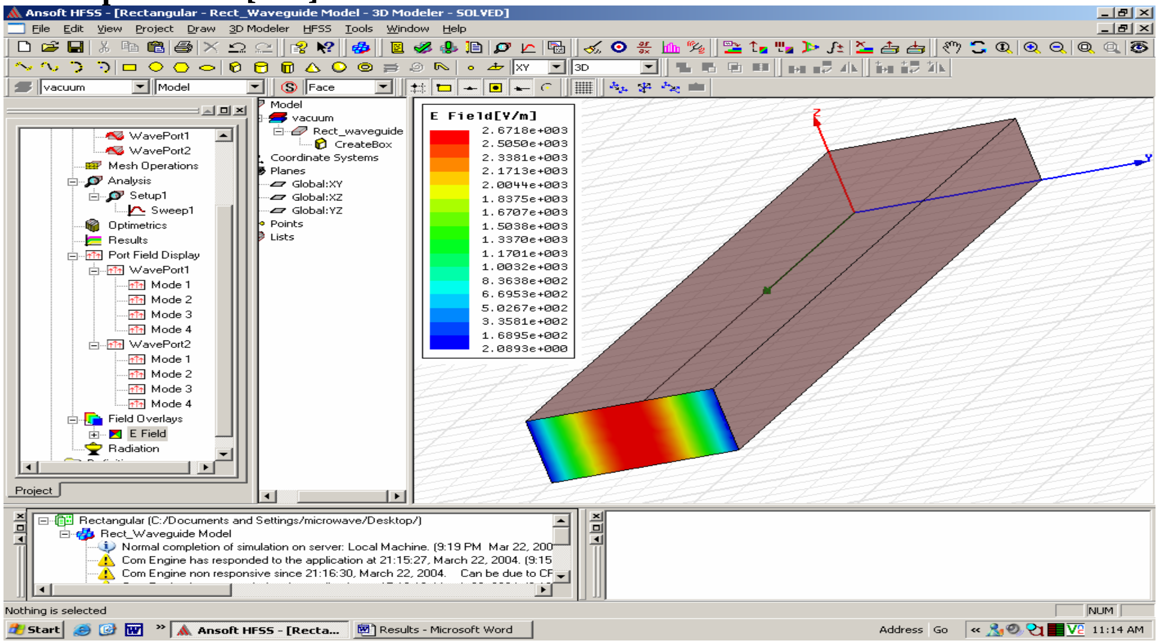
Select the face for which you want to view the field pattern
On the **HFSS** menu click on **Fields>plot fields>mag E**
For H field pattern click on **Fields>plot fields>mag H**



Create field dialog box appears: select mag E under Quantity and the model name under volume or you can also select all objects.



E field pattern in [v/m]



The colors indicate the intensity of the field decreasing from top to bottom.

H Field Pattern in [v/m]