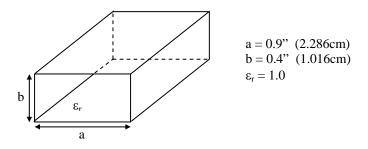
# **Project 1: Rectangular Waveguide (HFSS)**



# 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  $\beta$ ,  $\lambda$ ,  $\eta$  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  $\beta$ ,  $\lambda$ ,  $\eta$  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**

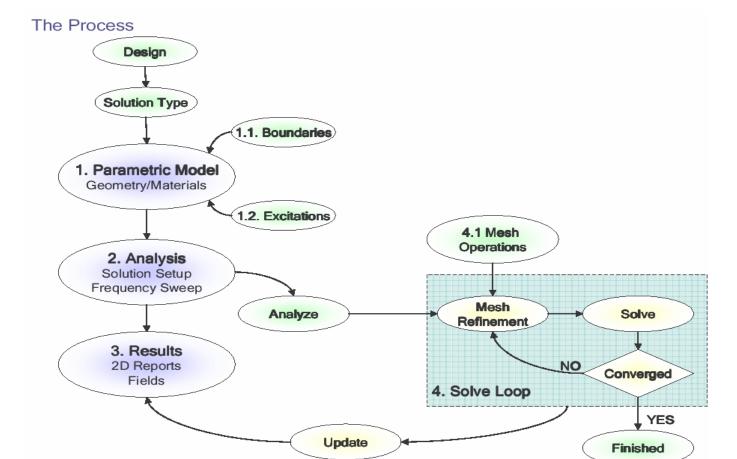
#### **Starting Ansoft HFSS**

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

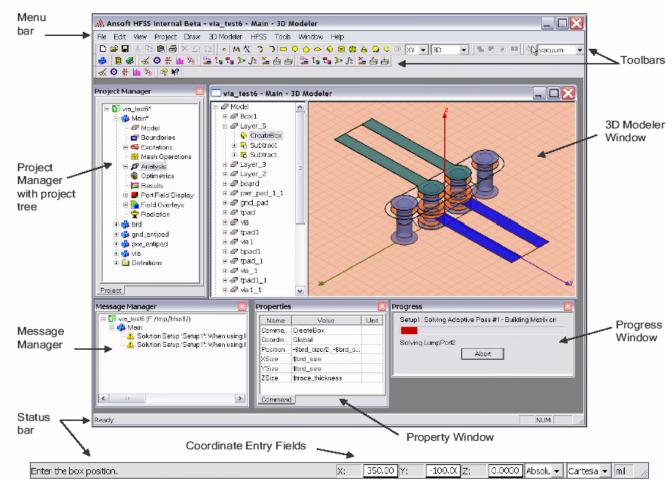
					1			
			Accessories	•				
		<b>••••</b>	Adobe	•			-	
			Ansoft	•			•	
			CNET	•			•	Ansoft on the Web
-2	AOL for Broadband		Games	•				🙏 HESS 9
			MATLAB 6.5	•		2		HFSS 9 Readme
	Set Program Access and Defaults		Microsoft Office Tools	•		Ansoft Designer		🜍 Uninstall HFSS 9
1000	Windows Catalog		Microsoft Visual Studio 6.0	•		Ansoft Home Page		
	windows catalog	<b>_</b>	Network Associates	•	1	· · · · · · · · · · · · · · · · · · ·		
	Windows Update	<b>1</b>	RealVNC	•				
		-	Serenade 8.71	•				
	WinZip		Sonnet 9.51	►	A	Maxwell Control Panel		
		-	Spybot - Search & Destroy	►		Optimetrics v2.5 Release Notes		
	Programs		Startup	•				
	Documents	_ 📼	Web Search Tools	►				
	Focautorica		WinZip	►				
2 🛂	Settings	•	WS_FTP	•				
	County 1		Zone Labs	•				
<del>8</del> 🧪	Search	* 💌	Acrobat Distiller 5.0					
Protessional	Help and Support	24	Adobe Acrobat 5.0					
		9	Internet Explorer					
	<u>R</u> un	-	MSN Explorer					
ž		- 🌤	MSN Messenger 6.0					
8 🕗	Log Off microwave	-	Outlook Express					
	Shut Down		Remote Assistance					
_		_ 🕑	Windows Media Player					
Start	🥯 🕑 🚾 🐣	- 28	Windows Messenger					

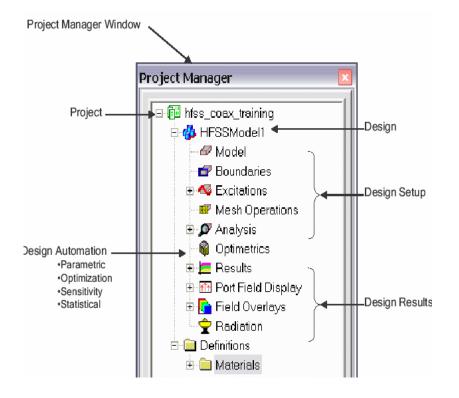
#### **Creating Projects**:

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



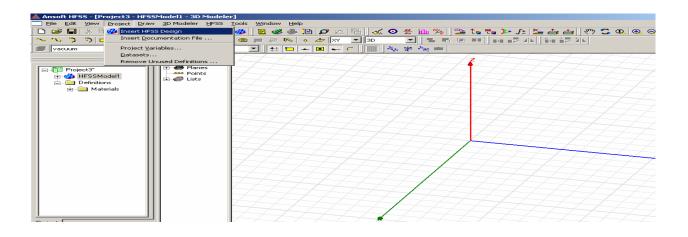


To set up an HFSS design, follow this general procedure. Note that after you insert a design, you do not need to perform the steps sequentially, but they must be completed before a solution can be generated.

#### 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 HFSSDesignn 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 -<u>Selecting the Solution Type</u>

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.

Solution Type: Project3 - HFSSModel1	×
Driven Modal	
Uliven Modal	
O Driven Terminal	
C Eigenmode	
OK Cancel	

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.

Set Model Units		×
Select units:		
Rescale to new u	inits	
ОК	Cancel	

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.

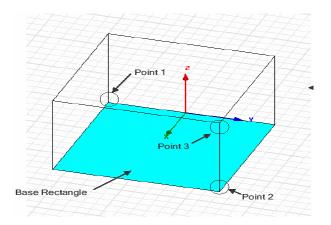
#### IV- Drawing a Model

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

Å Ansoft HFSS - [Project3 - H	IFSSModel1 - 3D Modele	eler]
File Edit View Project D	<u>)</u> raw <u>3</u> D Modeler <u>H</u> FSS	S <u>T</u> ools <u>W</u> indow <u>H</u> elp
🗋 🗅 🚅 🖬   X 🖻 🛍   🧉 🗥		- 👩 📱 🖉 🖶 🗎 🔎 🗠 🖼 🗹 📀 🐇 📶 🏂 🎦 🔭 🔈 🖈 🏝 🍰
$\sim \sim $ ) ) $\Box \circ < $	V. Spline	) 🕲 🚔 🖉 🔹 🛨 XY 💌 3D 💽 📲 🖷 💷 🖬 🖓 🕼 🖬
S vacuum Mod	Arc •	▶
	<u>R</u> ectangle	
······································	<ul> <li>Ellipse</li> </ul>	
	<mark>⊃</mark> ⊆ircle	
🗄 🍈 🥵 HFSSModel1 🤇	Regular Polygon	
🚊 🗋 Definitions 🛛	Box	
	 ] ⊆ylinder	
	 <u>R</u> egular Polyhedron	
	<mark>∆ ⊆</mark> one	
	🕑 Sphere	
	<u>T</u> orus	
	📑 Helix	
4	③ Spiral	
	<u>S</u> weep	



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



menas						
X:	0	Υ:	0	Z:	0	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.



 $\rightarrow$  The length of the waveguide = any multiple of

the wavelenghth

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

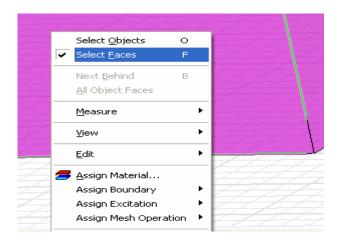
Name	Value	Uni
Command	CreateBox	
Coordinate System	Global	
Position	0,0,0	
×Size		cm
YSize	2.286	cm
ZSize	1.016	cm

The properties window can also be obtained by

📨 vacuum 💽 Model	💌 🔇 Object	I III III III IIII	C [] [] [] *2, 3	4 🔧 💼
Project3* HFSSModel1* Model Excitations Mesh Operations Mesh Operations Mesh Operations Mesh Operations	Model Vacuum Vac		Copy Easte Delete Properties Arrange Duplicate Scale Surface	Ctrl+C Ctrl+V Del

#### 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.

Search Parameters Search by Name Search	Search Criteria by Name Relative Permi			aries 🔽 Sho s] Materials	ow definitions in Project
Name	Location	Origin	Relative Permittivity	Relative Permeability	Bulk 📥 Conductivity
Teflon (tm)	SysLibrary	Materials	2.1	1	0
teflon_based	SysLibrary	Materials	2.08	1	0
tin	SysLibrary	Materials	1	1	8670000Siemens/m
titanium	SysLibrary	Materials	1	1.00018	1820000Siemens/m
tungsten	SysLibrary	Materials	1	1	18200000Siemens/m
Vacuum	Project	Materials	1	1	0
vacuum	SysLibrary	Materials	1	1	0
water_distilled	SysLibrary	Materials	81	0.999991	0.0002Siemens/m
water_fresh	SysLibrary	Materials	81	0.999991	0.01Siemens/m
water_sea	SysLibrary	Materials	81	0.999991	4Siemens/m
zinc	SysLibrary	Materials	1	1	16700000Siemens/m
c	- IIII		pa l		>

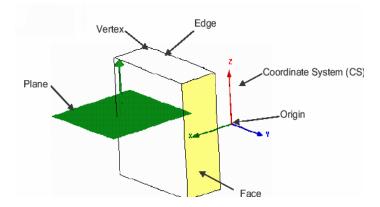
4.Select a material from the list. Select Air or vacuum for the whole box as our rectangular waveguide is not filled with any dielectric.

5.Click OK.

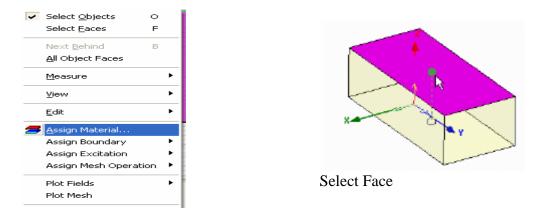
6. The material you chose is assigned to the object.

#### 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



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.

HFSS Tools Window Help					
Solution Type	🗠 🗟 💰 🗿 👯 🔟	🛯 🏂 🖍 🗤 🕨 🕂			
🧧 List		1590 B B B A			
💞 Validation Check	- C 🏢 🌮 🛠 🎝				
Edit Notes	- · · · · · · · · · · · · · · · · · · ·	ę —			
3D <u>M</u> odel Editor				FiniteGondi	
<u>B</u> oundaries	▶ <u>A</u> ssign ▶	Perfect <u>E</u>		TTTTTTTT	
<u>E</u> xcitations	• <u>R</u> eassign	Perfect <u>H</u>	7 時間 / / / /	+++++++++++++++++++++++++++++++++++++++	
Mesh Operations	Delete All	Einite Conductivity			
Analysis <u>S</u> etup	Visualization	Impedance			FiniteCor
Optimetrics Analysis	<u>R</u> eprioritize	Layered Impedance			
😓 Analyze	PML Setup Wizard	<u>R</u> adiation			
S Hudiyze		Symmetry			
<u>R</u> esults		Master			
Eields	•	Sl <u>a</u> ve			
<u>R</u> adiation		Lumped RLC			
	1 Martin to the				

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

#### VII- Assigning Excitations

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

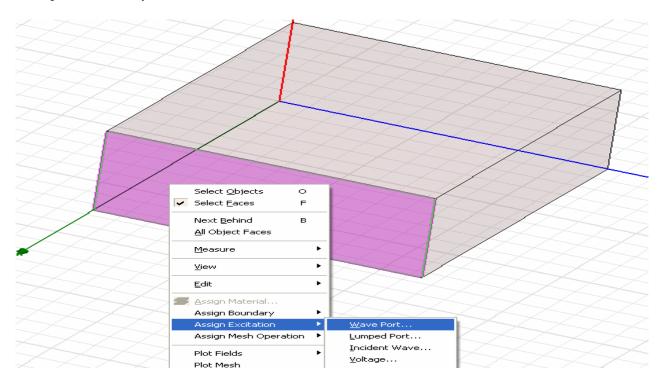
VII. a) Assign Ports

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

Number of Mode	es: 4		Update
Mode	Integration Line		Characteristic Imp. (Zo)
1	None	🗕 Zp	vi
2	None	Zp	oi i
3	New Line None	Zp	pi
4	None	Zp	pi

#### 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

	Create Line
	Draw the port line. When you are finished, the port edit dialog will reappear.
3.	Cancel

- 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 screen5. Select the endpoint of the vector using the mouse or the keyboard. The endpoint defines the direction and
- length of the integration line.

.

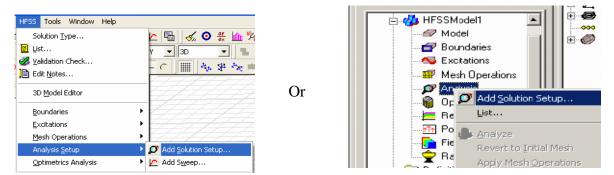
6. The Wave Port or Lumped Port dialog box reappears.

## 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



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  $2/3^{rd}$  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**

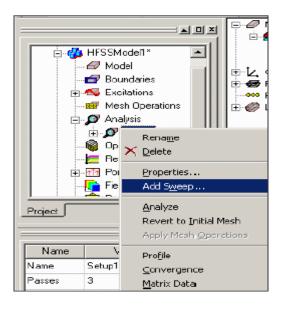
The delta S is the change in the magnitude of the S-parameters between two consecutive passes.

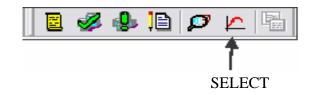
4.Click Ok

	Solution Setup
	General Advanced Ports Defaults
Adaptive Mesh_ Frequency	Solution Frequency: 20 GHz -
	Adaptive Solutions
	Maximum Number of Passes: 10
	Maximum Delta S Per Pass: 0.002
	Use Defaults
	OK Cancel

## 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** 

Edit Sweep				
Sweep Type © Discrete © Fast © Interpolating Error Tolerance 0.5 %	DC Extrapolation Options         ■ Extrapolate to DC         Minimum Solved Frequency       0.1         Image: Snap Magnitude to 0 or 1 at DC         Snapping Tolerance       0.01			
Max Solutions 20 Frequency Setup Type: Linear Step Start 1 GHz Stop 20 GHz Step Size 1 GHz Save Fields (All Frequencies)	Time Domain Calculation         Frequency         Display >>			
OK Cancel				

## VIII – <u>Running a Simulation</u>

To validate your model

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

HFSSMadefi Valdalion Check completed Errors: 0 Warnings: 3	<ul> <li>▲ Boundaries and Excitations</li> <li>✓ Meeh Operations</li> <li>✓ Andysis Schup</li> <li>✓ Optimetrics</li> <li>☑ Badiation</li> </ul>
l See Message Window for details.	

## 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.

Setup1: Solving	g Ports on Local Machine -	
Adapting p1, P	898 9	
	Abort	

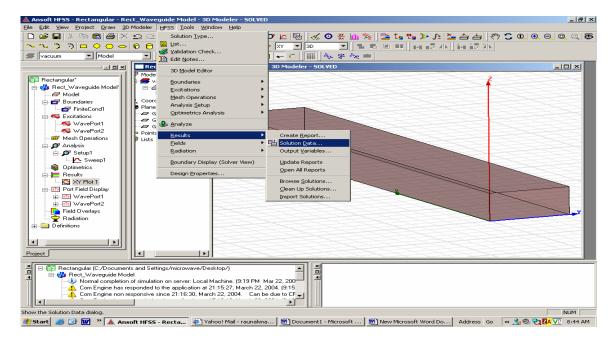
You can also view the following solution data at any time during or after the solution

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

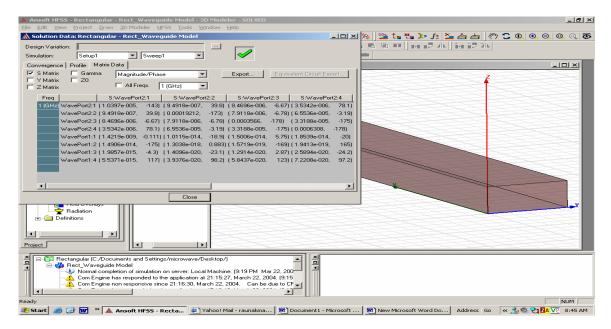
Once the simulation is completed HFSS Informs you in the message window.

# 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.

🝌 Ansoft HFSS - Rectangular - Rect_Waveguide Model - 3D Modeler - SOLVED 📃 🗖
Elle Edit View Project Draw 3D Modeler HFSS Tools Window Help
」 D 🖆 🖬 🖇 略 魯 × ユ ニ 😤 🧏 🐉 🗧 🥙 🌡 🍺 ビ 丘 🗟 🧹 🌜 籠 🏂 D 🖉 L 🗟 🖉 🦛 🧏 🔤 🕺 🖗 🖉 🔍
🖉 Vacuum 💌 Model 💌 🔇 Face 💌 🗱 🖿 🗕 🖛 C 🕮 🍇 🛠 🐜 🛑
A Comparison of the sect _ Waveguide Model - 3D Modeler - SOLVED
Image: Sectangular       Vacuum         Image: Sectangular       Image: Sectangular         Image: Sectangular
Rectangular (C:/Documents and Settings/microwave/Desktop/)     Arrow Rect. Waveguide Model     Normal completion of simulation on server: Local Machine. (9:19 PM. Mar 22, 200     Com Engine hors responded to the application at 21:15:27. March 22, 2004. Can be due to CF     Com Engine hor responsive since 21:16:30. March 22, 2004.
Ready NUM
🐮 Start 🧔 😳 👿 🐣 🛦 Ansoft HFSS - Recta 😰 Yahoo! Mail - raunakma 🕮 Document I - Microsoft ] 🕮 New Microsoft Word Do Address Go 🛛 « 🔧 👁 😋 🕼 🟹: 8:56 /

- 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

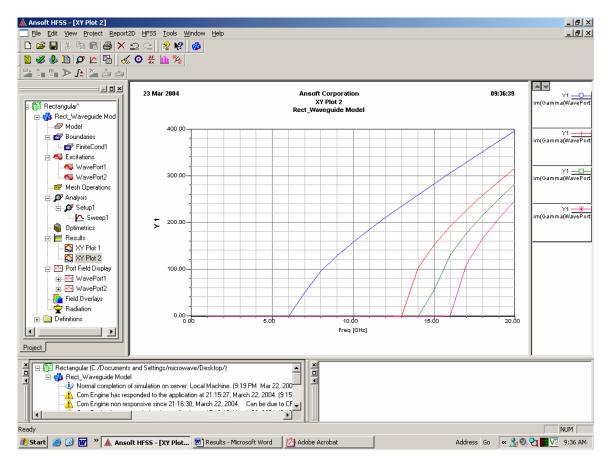
- 4) In the **Solution** list, click the solution containing the data you want to plot.
- 5) In the domain list, click a domain. For modal and terminal S- parameter reports, the domain can be frequency or time. *In this case we want frequency domain.*
- 6) Click on Add Trace, click Done

Ele Edit View Projekt Draw 30 Modeler HFSS Tools Window Help D 문제 1/6 편 18 명 18 명 20 20 20 19 19 19 19 19 10 10 10 10 10 10 10 10 10 10 10 10 10	
≤	
	۲
Vacuum V Model	
Y Y-axis August 1	
Add BlankTrace	
EB Rectangular" 2 Freq dB(S(WavePort2:1),WavePort2:2)) Y1 Remove Trace	
Bect_Waveguide Moc Freq dB(S(WavePort2:1,WavePort2:3)) Y1	
Model • Freq dB(S(WavePort2:1,WavePort2:4)) Y1 Remove All Traces	
e 🗗 Boundaries	
TinteCond1	
Sweeps X Y	
S WavePott Design: Rect_Wavepuide Model	
Waver of Z     We provide      Category: Quantity: Function:	
Analysis Solution: Setup1 : Sweep1 Variables S(WayePort21) Ang	
Output Variables S(WavePort2:1, WavePort2:2) ang_rad	
Sweep1 Stroop YParameter StWavePort2:1.WavePort2:4 im	
Z Parameter     SWavePort21WavePort11     mag     re	
Gamma StWavePort21WavePort131	
PortZo SiWavePort21WavePort14)	
S(WavePort2:2)	
Image: Strategy of the state of the stat	
SfWavePort22WavePort111	
Radiation S(WavePort2.2WavePort1.2)	
S[WavePort2:2]WavePort1:4]	
Set Terminations	
Dutput Variables	
Project Replace Trace	
A B-B Rectangular (C/Docu	
Image: Section guider (C: / Docu       Image: Section	
A Constraine has responded to the application at 21 in 22. Note: 1 and 1	
A com Engine non responsive concertaince 21:16:27, march 22, 2004, [5:15]	
	_
🐉 Start 🧑 😳 👿 🐣 🔥 Ansoft HFS5 - Rectan 😭 HFS5 Online Help 🛛 Document 1 - Microsoft 🕴 Address Go 🤍 🔧 🔍 🚺 🗸 🖓 🖓	AM

# **Analysis and Results**

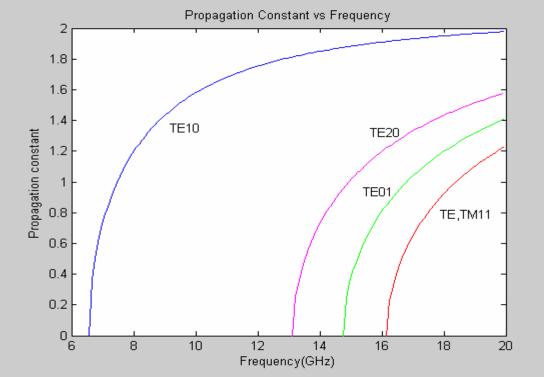
- 1) Analyze the propagation constant for the first four modes.
- 2) Determine the wavelength at different frequencies for the first four modes.
- 3) Determine the intrinsic impedance at different frequencies for the first four modes
- 4) Analyze E and H field patterns for the first four modes.

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  $\exp(-j\beta z)$ , has to be real and make  $\exp(-j\beta z)$  imaginary.

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



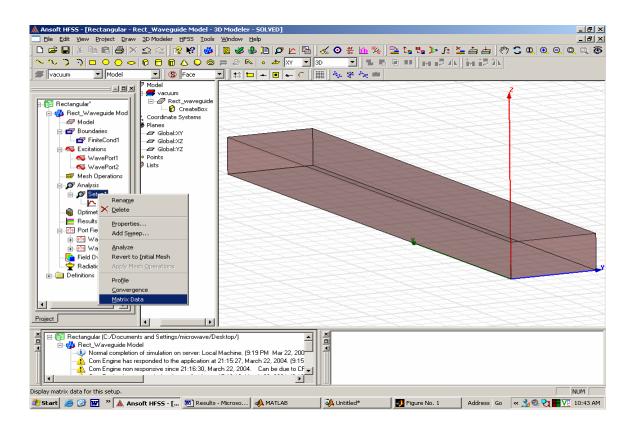
Similarly you can draw plots for wavelength vs. frequency and impedance vs. frequency from the data given in the solution data box.

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.

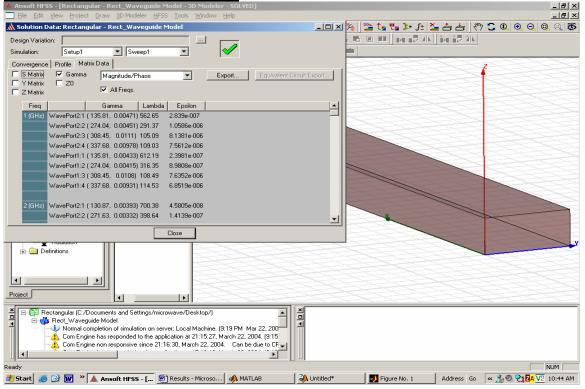
2. In the **Simulation** pull-down list, click the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.

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.

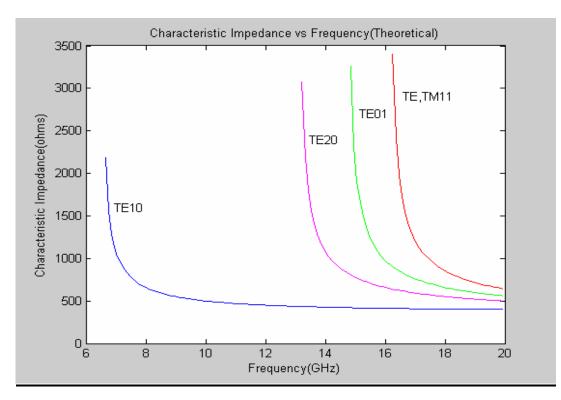
4. Select the format — Magnitude/ Phase, Real/ Imaginary, dB/ Phase, Magnitude, Phase, Real, Imaginary, or dB — in which to display the matrix information. The available formats depend on the matrix type being displayed.



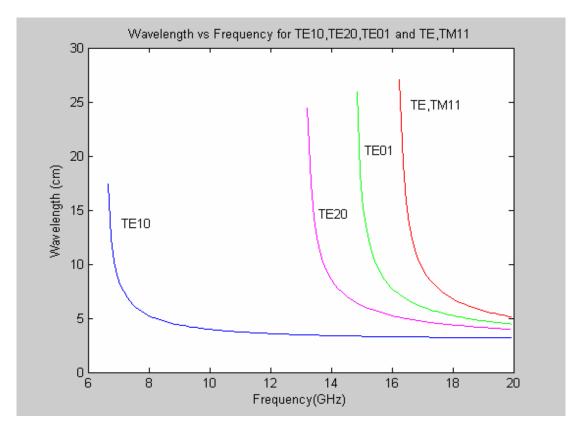
#### The solution data window appears



# **Impedance vs. Frequency for the first four modes**

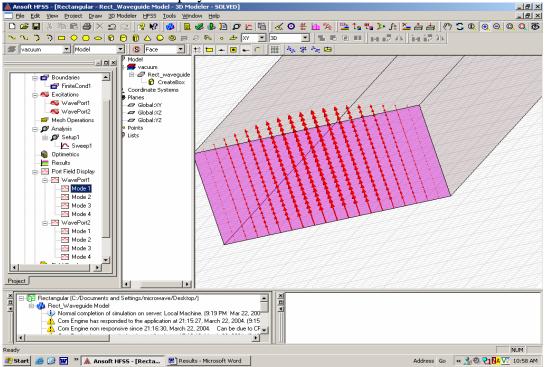


# Wavelength vs. frequency for the first four modes

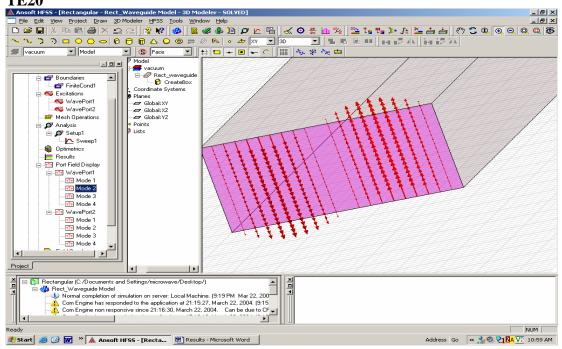


# Field Patterns for E and H fields

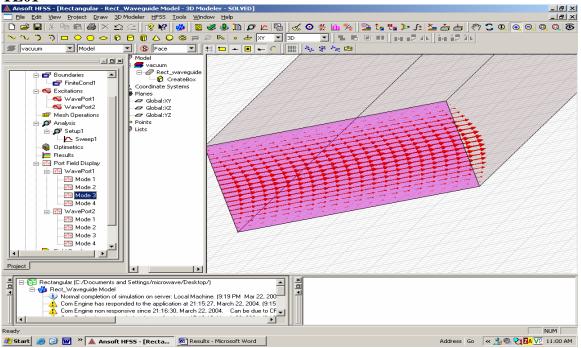
On the project tree click on **Port Field Display>wave port 1>mode1** The TE10 mode will be displayed in the model



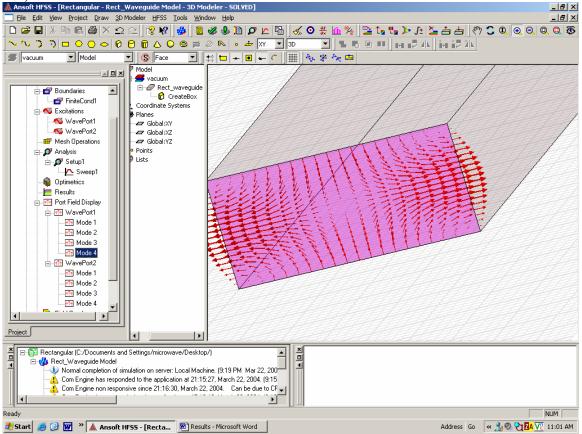
# Similarly click on mode2, mode3 and mode 4 TE20



#### **TE01**

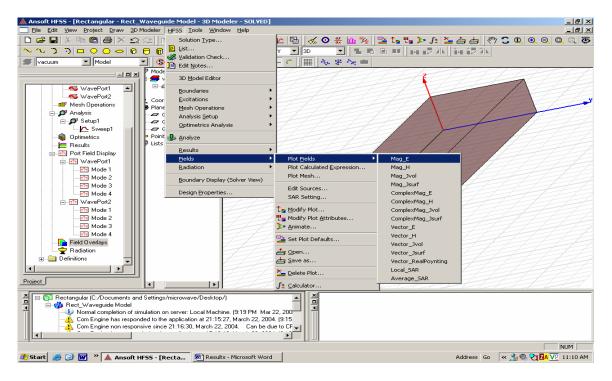


# TE,TM11

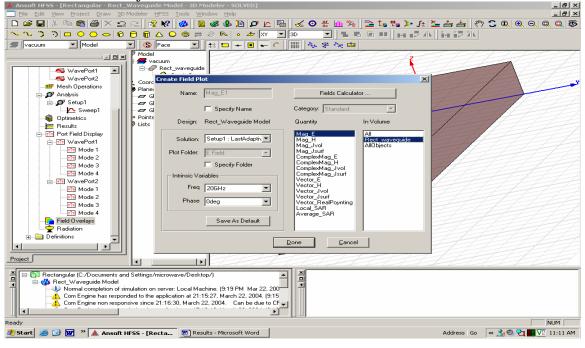


# To view E and H field patterns

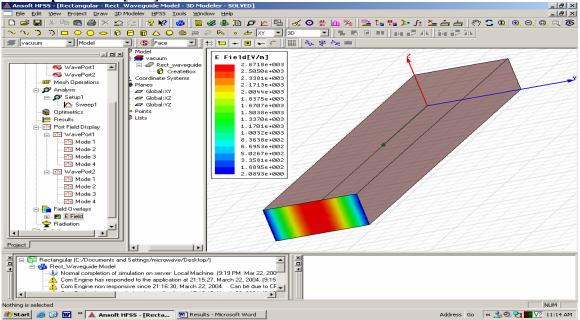
Select the face for which you want to view the field pattern On the **HFSS** menu click on **Fields>plot fields>mag E** For H field pattern click on **Fields>plot fields>mag H** 



**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]

