# **HFSS Microstrip Tutorial**

Kisa Avrutina

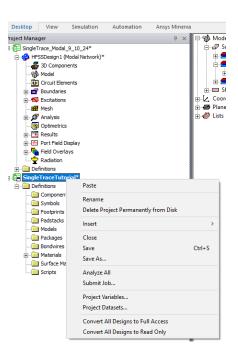
October 2024

## Contents

Setting Things Up	2
Building the Model	4
Ground plane:	4
Substrate:	
Microstrip:	6
Create an air boundary:	7
Create ports:	8
Running Simulations	0
Simulation Setup:1	
Running the Simulation:1	3
Viewing Solutions:1	4
Miscellaneous1	7

## **Setting Things Up**

- 1. Open Ansys Electronics Display
- 2. Create a new model file
  - a. Hover over file in the top banner, and select New
  - b. Right click on file name in Project Manager, select Insert, select Insert HFSS design



- 3. Check solution type
  - a. On top banner, click on HFSS

- b. Select Solution Type
- c. For wave port method, select Modal
- d. Check "Save as default" to ensure new models will be of the modal solution type
- e. Click Ok

Solution Type: SingleTraceTutorial - HFSSDesign2	×
Solution Types	
C SBR+ C Eigenmode C Characteristic Mode	
Options  Network Analysis  Composite Excitation	
<ul> <li>Modal</li> <li>C Terminal</li> </ul>	
Auto-Open Region	
Save as default	

## **Building the Model**

#### Ground plane:

- 1. Select the Draw tab on the top banner
- 2. Select the rectangular prism



- 3. Construct the shape anywhere on the work plane
- 4. The properties window should pop up. Adjust Position, XSize, YSize, and ZSize.
  - a. To access the properties window, double click
     "CreateShape" (where Shape is refers to whatever shape your object is—such as Box, Cylinder, etc). This can be found by expanding the object's tree.



- 5. Create variables for all important properties
  - a. Type "\$" followed by your variable name into any of the size or position boxes. Click the enter key. Ansys will recognize any non-numerical value as a variable, and prompt you to define it.
  - b. In the Add Variable window, set the variable's value. Make sure to set the units appropriately.
    - i. \$ ensures the variable is saved as a "Project Variable", making it easy to access later.

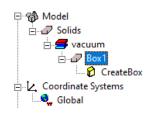
		Add Varia	ble	×		-
Name Command Coordinate Sys Position	0.0.0	Name Unit Type	\$StripT Length	•	ed Value	Description
XSize YSize	\$SubW \$SubL	Unit	mm	•		
ZSize	\$Strip T	Value	0.005	~ ~		
		Туре	Project Variable	Ţ		

- ii. To access variables later:
  - Right click on the file name in Project Manager, and select Project Variables.
  - The Project Variables window allows you to view and edit all project variables.

	8	
SingleTraceTutorial	Paste	
🖃 🍻 HFSSDesign2 (Mc		
🚭 3D Components	Rename	
	Delete Project Permanently from Disk	
🔃 Circuit Elements	,,,,,,	
- 🗗 Boundaries	Insert	>
- 🚭 Excitations	Close	
🔊 Analysis	Save	Ctrl+S
	Save As	
Port Field Displa	Analyze All	
	Submit Job	
Radiation	Project Variables	
Definitions		
Components	Project Datasets	

Value	Optimization /	Design of I	Experiments	C Tuning	C Sensitivit	y C	Statistics
Name	Value	Unit	Evaluated Value	Description	Read-only	Hidden	Sweep
\$SubW	5	mm	5mm				~
\$SubL	7	mm	7mm				<b>v</b>
\$Strip T	0.005	mm	0.005mm				~

- 6. Assign materials
  - a. Double click on the object's name in the model tree.



b. The properties window will pop up. You can assign a name, appearance, and material. For ease of visibility later on, you may want to raise the transparency.

Attribute	1						
	Name	Value	Unit	Evaluated Value	Description	Read-only	
	Name	Box1					
	Material	"vacuum"		"vacuum"			
	Solve Inside	✓					
	Orientation	Global					
	Model	~					
	Group	Model					
	Display Wireframe						
	Material Appearance						
	Color						
	Transparent	0					
						<b>— — — — — — — — — —</b>	
						Show Hidden	

- c. Expand the material drop-down. The material should be set to "vacuum" by default.
- d. Select "Edit ... "
- e. You can select a material from the library, or add your own material and manually assign values to material properties. The ground plane's material is typically copper.

arch Parameters arch by Name opper Search	Search Criteria by Name Relative Permittivity	C by Property	Libraries 🔽 [sys] Materials	Show Project definitions	Select all libraries		< >	-
/Name	Location	Origin	Relative Permittivity	Relative Permeability	Bulk Conductivity	Dielectric Loss Tangent		^
cast_iron	SysLibrary	Materials	1	60	1500000siemens/m	0	0	1
chromium	SysLibrary	Materials	1	1	7600000siemens/m	0	0	
cobalt	SysLibrary	Materials	1	250	10000000siemens/m	0	0	
conformal_coat	SysLibrary	Materials	3.1	1	Osiemens/m	0.035	0	
copper	SysLibrary	Materials	1	0.999991	58000000siemens/m	0	0	
coming_glass	SysLibrary	Materials	5.75	1	0	0	0	
cyanate_ester	SysLibrary	Materials	3.8	1	0	0	0	
diamond	SysLibrary	Materials	16.5	1	0	0	0	
diamond_hi_pres	SysLibrary	Materials	5.7	1	0	0	0	
diamond_pl_cvd	SysLibrary	Materials	3.5	1	0	0	0	
Dupont Type 100 HN Film (tm)	SysLibrary	Materials	3.5	1	0	0.0026	0	
Duroid (tm)	SysLibrary	Materials	2.2	1	0	0.0009	0	
epoxy Kevlar xv	SvsLibrarv	Materials	3.6	1	0	0	0	~

f. Click Ok

#### Substrate:

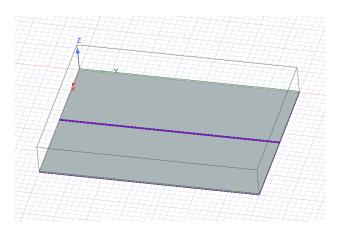
- 1. Construct the shape the same way you did for the ground plane. Use variables in the properties window to assign size and position.
- 2. Assign a material. This is often aluminum nitride, called "Al\_N" in Ansys.

#### Microstrip:

- 1. Construct the shape. Use variables to assign size and position. The trace's thickness is typically the same as the ground plane.
- 2. Assign a material. This is typically copper.

#### Create an air boundary:

1. Create a box of the same x and y-dimensions of the ground-plane, but significantly bigger in the z-dimension (about 1-1.5 mm is usually good). This box should be placed on the same work plane as the ground plane, and overlap the ground plane and substrate exactly.



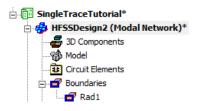
- 2. Assign the material of the box to be air.
- 3. Make sure Object is chosen in the Select menu found under the Draw tab on the top banner. You may want to change this later to select specific faces or edges.

	2022 R1 - SingleTraceTutorial - HFSSD tt Draw Modeler HFSS Tools
→     Cut     ♥     Undo       □     □     Copy     ♥     Redo       Save     □     Paste     X     Delete	Select: Object Vame Select: Dipect Select by Name Select by Name
Desktop View Draw	Model Simulation Res

- 4. Select your air box by clicking on it in the work plane.
- 5. Right click on the box, select Assign Boundary, and select "Radiation..."
- 6. The Radiation Boundary pop-up will prompt you to name your boundary. Choose an appropriate name and click OK.

Radiation	Boundary	×
Name:	Rad1	
	OK	

7. Check that your boundary appears under Boundaries in the design tree in Project Manager.

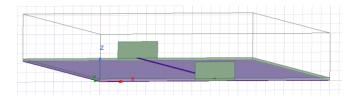


#### *Create ports:*

- 1. Creating the wave ports involves drawing rectangles at either end of the microstrip. 2D shapes are drawn along the work plane grid, which is oriented on the XY-plane by default. To draw a shape along a different plane, you will need to change the grid. This can be done under the Draw tab.
  - a. You will most likely want the XZ grid.

∰ Grid		₩.	Model *
XZ	-		vacuum *
3D	Ŧ	📩 🦂	/ Material
			0 🖊

- 2. Draw a rectangle at one end of your microstrip, then assign appropriate position and dimensions.
  - a. The wave port should be centered at the center of your microstrip. Its bottom end should be aligned with the top edge of the ground plane.
  - b. <u>Ideal Dimensions:</u>
    - i. The height should be >4h, where h is the thickness of your substrate.
    - ii. The width should be  $\sim$ 5w, where w is the width of your microstrip.
    - iii. You can go bigger than this! If your simulation results look strange, it may be worth increasing the size of your wave port.



- 3. Assign excitations
  - a. Select the created rectangle. Right click on the selection, click Assign Excitation, then Port, then Wave Port.
  - b. Ansys allows you to set a number of modes and set an integration line for each mode. The default setting are fine for many basic applications.
    - i. You can also change the characteristic impedance. The options are Zpi, Zpv, Zvi, and Zwave. These will generally give different results. Zpv and Zvi will only appear if an integration line is drawn such that a unique voltage can be computed for that mode. Zwave is for homogenous waveguides.
  - c. For this model, I used 1 mode with Zpi characteristic impedance.

Wave Port : Genera	I		×
Name:	: 1		
Mode	Integration Line	Characteristic Impedance (Zo)	1
1	None	Zpi	1
C Align modes	arity using integration lines using integration lines		
U Axis Line:	analytically using coordinate	Reverse V Direction	
Filter modes fo	Undefined r reporter Use Defaults		
		< Back Next > Cancel	

d. Click Next. Renormalize all modes to port impedance of 50 Ohm. Click Finish.

Full Port Impedance:	50	ce + 1i * reactance	ohm	•
C Renormalize Specific M		Edit Mode Impedar	nces	
eembed Settings				
Deembed Distance:	0		mm	Ŧ
Positive distance will deemb	ed the por	t into the model.		
Get I	Distance (	iraphically		
	Use De	faulto		
	036 De	ruuro -		

- e. You will be able to find the created rectangle under Sheets in the model tree. You will find the wave ports under Excitations in the design tree in Project Manager.
- f. Repeat this process for the second port.

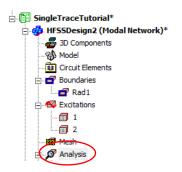
## **Running Simulations**

#### Simulation Setup:

1. Go to the Simulation tab on the top banner. Click Setup and select Advanced.



2. Alternatively, right click on Analysis under the design tree in Project Manager. Hover over Add Solution Setup and select Advanced.



SingleTraceTutorial

🗄 姆 HFSSDesign2 (Modal Network)

- 🚑 3D Components -- 🛞 Model

Circuit Elements

Boundaries Boundaries Rad1 Control Rad1 

2

- 3. You can solve for either a single frequency or multiple frequencies. Ansys will sweep through a range of frequencies either way, and use this to generate data for S-parameter plots. At this stage, you are selecting frequencies at which you will be able to plot the H-field on the model itself.
  - a. Multiple frequencies can be useful when trying to diagnose problems in the model.
- 4. In the Driven Solution Setup window, you can adjust the frequencies you solve for, maximum delta S, and maximum number of passes.

- a. Delta S characterizes the mesh sensitivity for the S-matrix, which controls the convergence of the adaptive solution. A smaller delta S results in a more accurate solution.
- b. For most cases, a delta S of 0.02 is best.
- c. The maximum number of passes is the maximum amount of times HFSS will refine the mesh as it attempts to converge. If the allowed amount of adaptive passes is low, the solution will not fully converge, and HFSS will return a warning.
- d. 20 passes is generally sufficient. You may go lower if the solution is taking especially long. If your solution does not fully converge, increase the number of allowed passes and re-run the simulation.

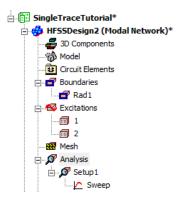
Driven Sc	olution Setup				$\times$
General	Options Advanced	Expression Cad	che Derivatives Defaults	3	
Setup	Name Setup1				
		bled 🗔 :	Solve Ports Only		
Ada	aptive Solutions				
	Solution Frequency:	C Single	Multi-Frequencie	es C Broadband	
	Frequency	Unit	Max. Delta S	Add	
	5	GHz	0.02		
	10	GHz	0.02		
				Remove	
		es 20			
	Maximum Number of Pass	ses 20			
		Us	e Defaults		
					S
			HPC and	d Analysis Options	
				OK Cance	el

- 5. The Edit Frequency Sweep window should appear once you confirm your solution setup.
  - a. If this window does not pop up on its own, right-click on your solution setup (which should have appeared under Analysis) and select Add Frequency Sweep.
  - b. Set the sweep type to "Interpolating".

- c. Set your sweep start and end to reasonable values. Make sure the frequencies you chose in your solution setup fall within this range.
- d. ~400 points for 10 GHz typically works well for smooth plotting.
- e. Check the box for Save Fields (At Basis Freqs)
  - i. This is what allows you to plot the magnetic field onto your model! This is very useful for assessing whether the simulation has run correctly.

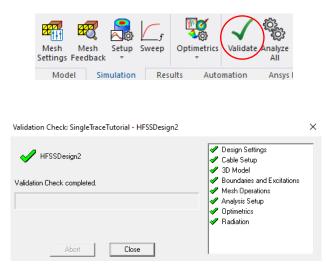
Edit Frequency Sweep					×
General Interpolation Defaul	ts				
Sweep Name: Sweep		Enable	đ		
Sweep Type: Interpolatin	g 💌	]			
Frequency Sweeps [401 pc	oints defined] —				_
Distribution	Start	End			
<sup>1</sup> Linear Count	2.5GHz	12.5GHz	Points	401	
Add Above       Add Below       Delete Selection       Preview         3D Fields Save Options       Time Domain Calculation       Save Fields (At Basis Freqs)         Save Fields (At Basis Freqs)       S Matrix Only Solve       S Matrix Only Solve         Save radiated fields only       C Manual - Allow for frequencies above					ve
	ОК	Cancel		lz <u>v</u>	

f. You will be able to find your Setup and Sweep under Analysis under the design tree in Project Manager. Double click on either of them to edit.



## **Running the Simulation:**

1. Under the Simulation tab in the top banner, click Validate. This runs a validation check. Make sure there are no errors or warnings.



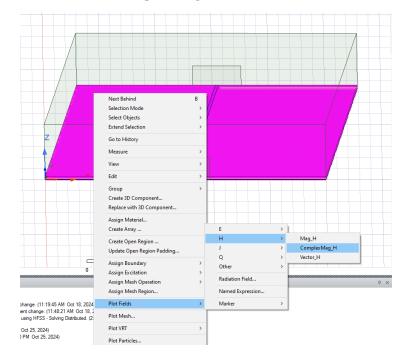
2. Under the Simulation tab, click Analyze All.



3. Wait for the simulation to finish running!

## **Viewing Solutions:**

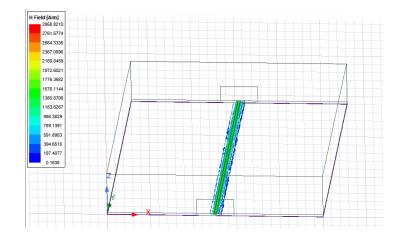
- 1. At this point, it is best to make sure all model objects are transparent.
- 2. Plotting magnetic field
  - a. Select any geometry on your model. Right click, hover over Plot Fields, hover over H, and select ComplexMag\_H.



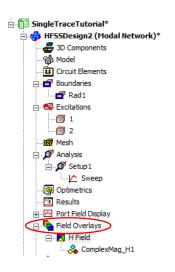
b. The Create Field Plot window will pop up. You can plot the magnetic field at any of your basis frequencies and choose which model object to plot on.

Create Field Plot				×
🔲 Specify Name	ComplexMag_H1	Fields Calculator		Surface Smoothing
🔲 Specify Folder	H Field	Category: Standard	•	
Design:	HFSSDesign2	Quantity	I	n Volume
Solution:	Setup1 : LastAdaptive 💌	ComplexMag_E Vector_E		Ground_Plane Substrate Microstrip Box1
Field Type:	Fields 💌	Mag_H ComplexMag_H Vector H		AllObjects
- Intrinsic Variable	s	Mag_Jsurf ComplexMag_Jsurf		
Freq	5GHz 💌	Vector_Jsurf Mag_Jvol		
Phase	Odeg 🗸	ComplexMag_Jvol Vector_Jvol Mag Jm		
		ComplexMag_Jm Vector Jm		
	Save As Default	ABS_Q SmoothQ ABS_Qm	Г Г Ч	Plot on surface only Adjacent side Streamline
	Done	Cancel		

c. You should be able to view the magnetic field overlay on your model.

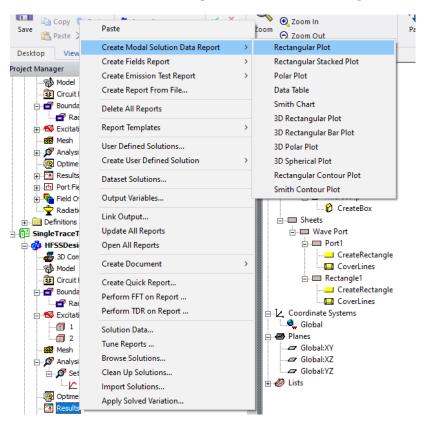


d. You will find plotted fields under Field Overlays under the design tree in project manager. Right click on a plot to modify or delete it.



#### 3. Plotting S-parameters.

a. Right click on Results under the design tree in project manager. Hover over Create Modal Solution Data report, and select Rectangular Plot.



b. You will be prompted to choose a variable and units. Choose these appropriately and click New Report.

Context	Trace Families Families	s Display		
Solution: Setup1:Sweep	Primary Sweep: Freq	All		
Iomain: Sweep	X: Default Free	1		
TDR Options	Y: dB(S(1,1))			Range Function
	Category:	Quantity:	-	Function:
sdate Report	Variables Output Variables S Parameter 2 Parameter VSWR Gamma Port 20 Lambda Epsilon Catobe S Parameter Active S Parameter Active S Parameter Active VSWR Passivity Design	S(1,1) S(2,1) S(1,2) S(2,2)		crone> , ang_deg ang_deg_val ang_rad arg cang_deg cang_deg_val cang_rad db 10normalize db 20normalize db db dbu m m mag normalize

c. The appropriate plot will be generated.

## Miscellaneous

- 1. To delete an object: Right click on the object name, hover over Edit, then select Delete.
- 2. Create global variables for all important properties!
  - a. Type "\$" followed by your variable name into any size or position box in the properties window and click the enter key. Ansys will recognize any non-numerical value as a variable, and prompt you to define it.
    - i. \$ ensure the variable is saved as a "Project Variable" and can be accessed from Project Manager (right click on the file name, and click Select Project Variables).
- 3. To easily orient to a specific view of your model:
  - a. Right click on your model, hover over View, and select Apply Orientation.