RV College of Engineering®
Mysore Road, RV Vidyaniketan Post,
Bengaluru - 560059, Karnataka, India

NBA Accredited (UG - 6Years)

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

Academic Year: 2023-24

Microwave and Radiating Systems
21EC72

UG Laboratory Companion Book
(Autonomous scheme 2021)

EXPERIMENT 6

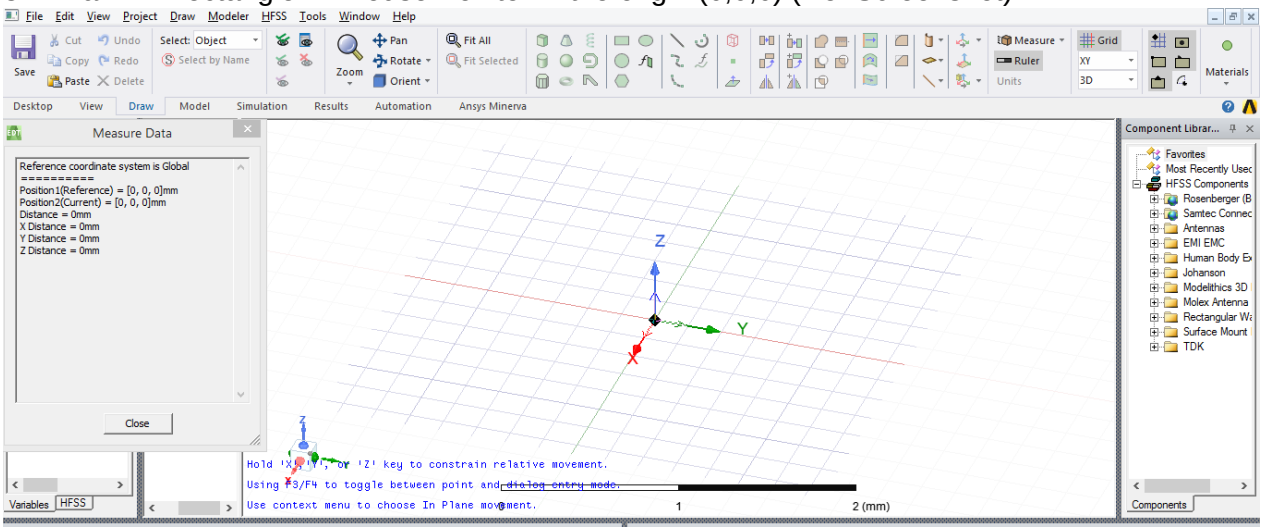
Design Simulation of Micro Strip Line and Hybrid Ring Using HFSS

a) Design of 50 Ω Microstrip line using HFSS

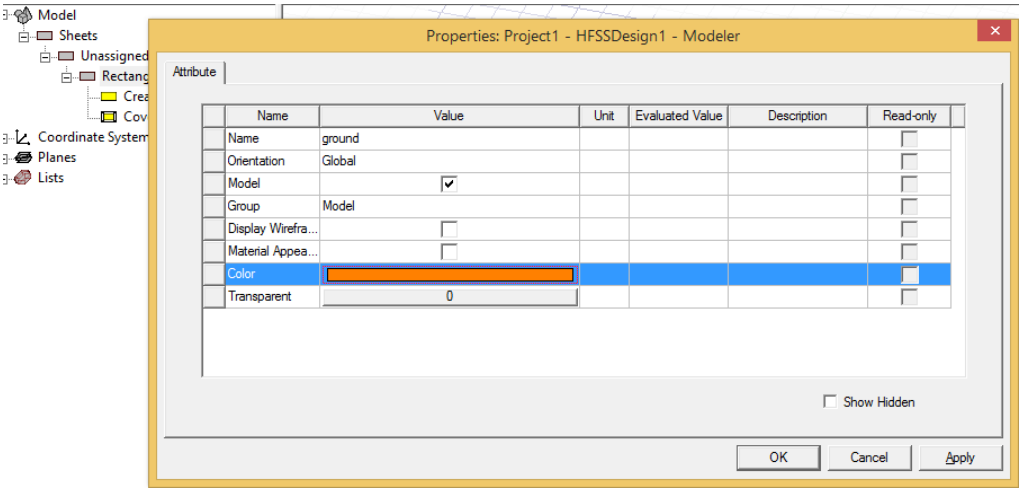
Detailed HFSS Steps

In ANSYS Electronics Desktop

1. Insert HFSS Design
    a. Project--> Insert HFSS Design
2. Creating ground plane for microstrip line
3. Draw--> Rectangle--> Mouse Pointer in the origin (0,0,0) (Ref Screenshot)

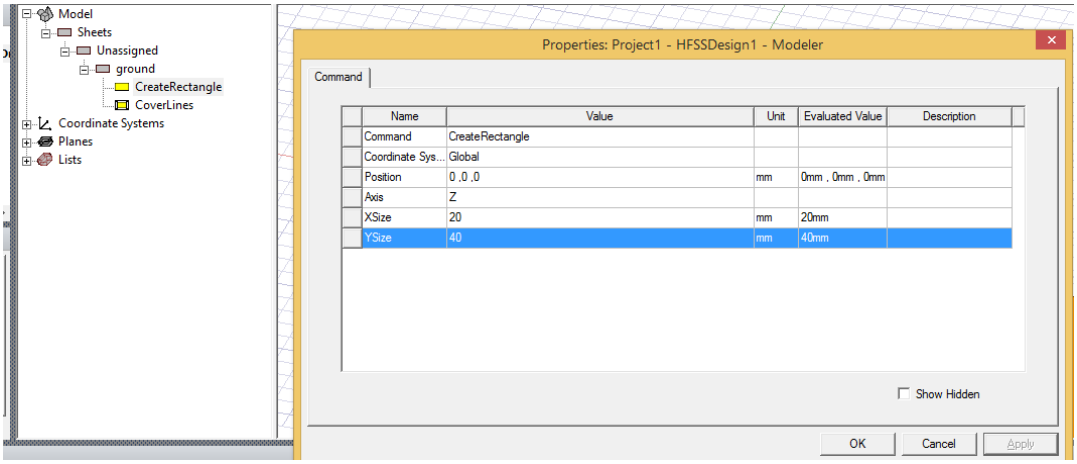


Draw rectangle with arbitrary X & Y Distances,
Rename the ground plane with name “ground”
Go to Model windows--> Rectangle1-->Attribute-->Name--> “ground”
Color--> Orange --> Enter OK



Select the ground plane dimension

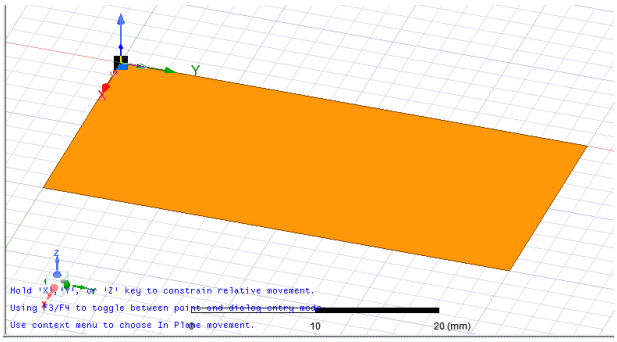
Model--> ground--> CreateRectangle-->XSize--> 20mm, YSize-->40mm --> Enter OK

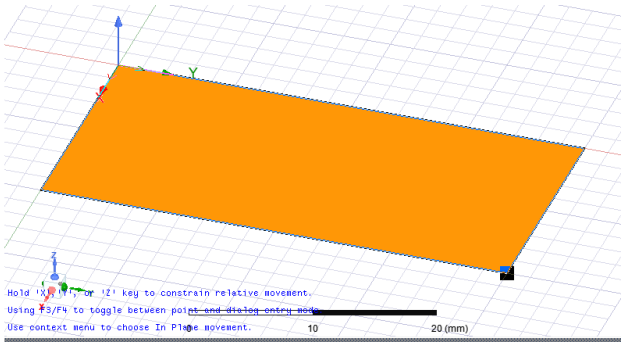


Note: To Fit all models in window, View--> Fit all (or Ctrl + D)

Design substrate box

Draw--> Box-->Position--> Using mouse-->Origin(0,0,0)-->XSize-->20mm, YSize-->40mm --> Enter OK



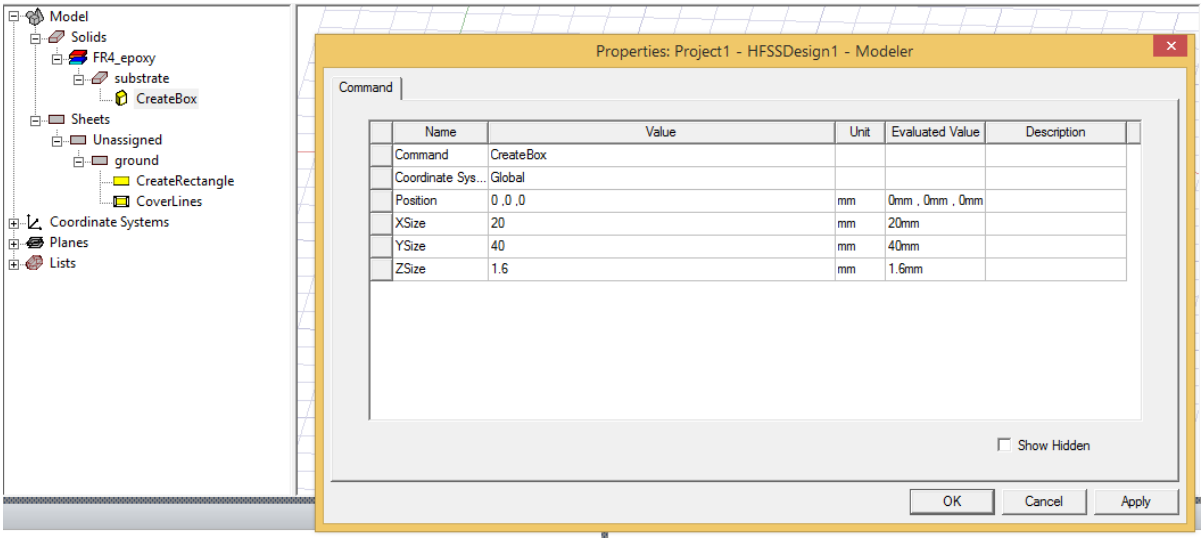


Rename the box with substrate & choose the substrate material

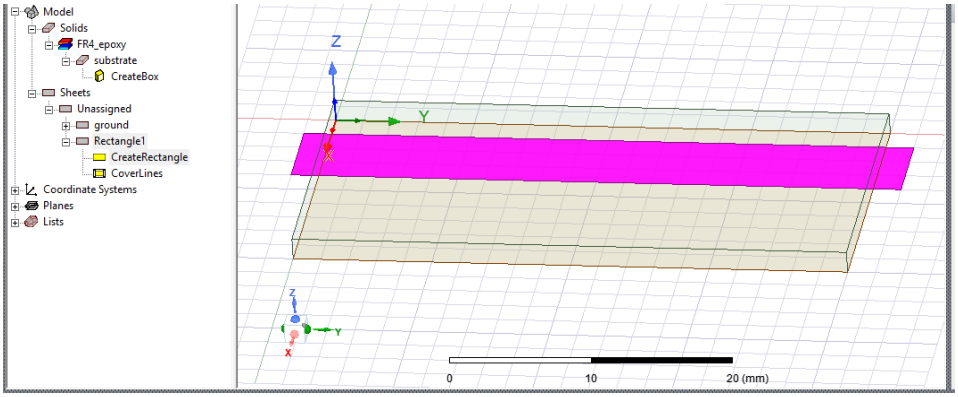
Solids-->vaccum-->Name-->" substrate" --> Material--> Edit..-->Search by Name--> "fr4" --
>Select--> "FR4_epoxy" --> Enter OK

Change the thickness of the substrate

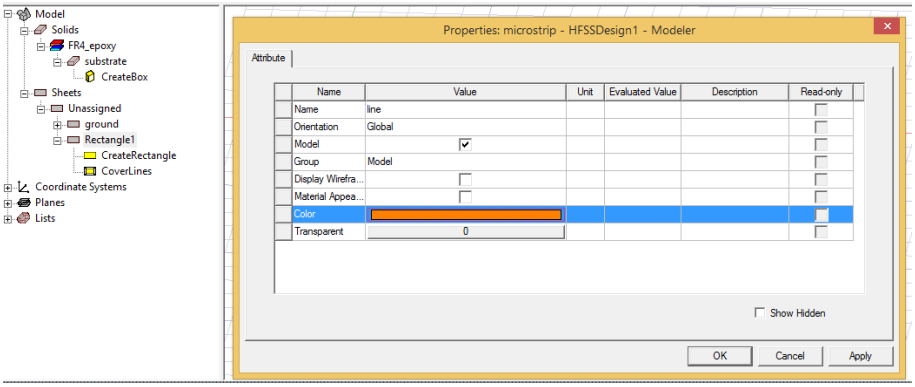
Model--> Solids--> FR4_epoxy-->substrate-->CreateBox-->ZSize-->1.6-->Enter OK



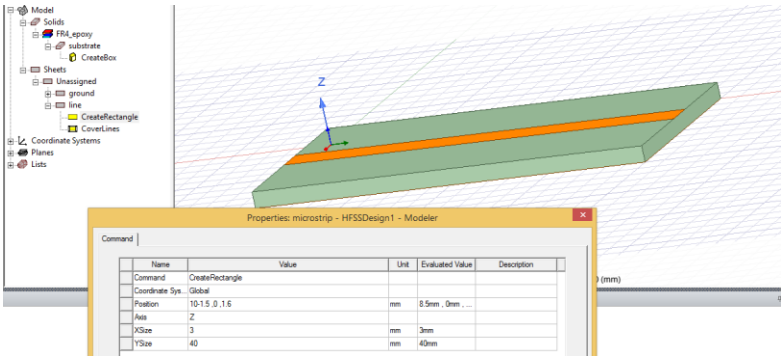
Design microstrip line at the top of the substrate
Draw Rectangle-->Using mouse pointer above substrate & change its attributes,



Sheets-->Unassigned-->Rectangle1-->Double click
Name--> "line", Color--> Orange --> Enter OK

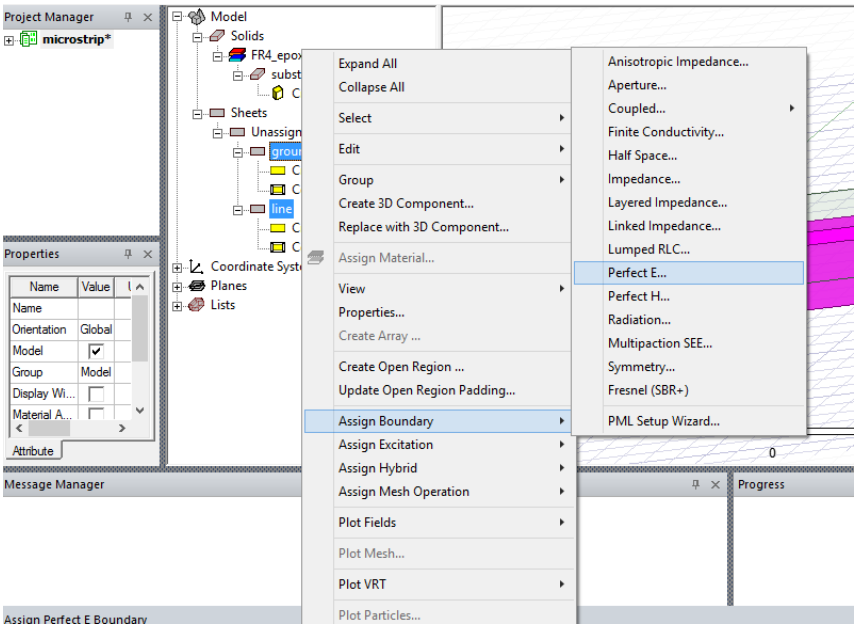


Change the dimension & position of micro strip line
Sheets-->line-->CreateRectangle
The microstrip line is positioned at top & center of the susbstrate.
Position--> 10-1.5, 0, 1.6
XSize-->3 (width of strip line)
YSize--> 40 (length of strip line)

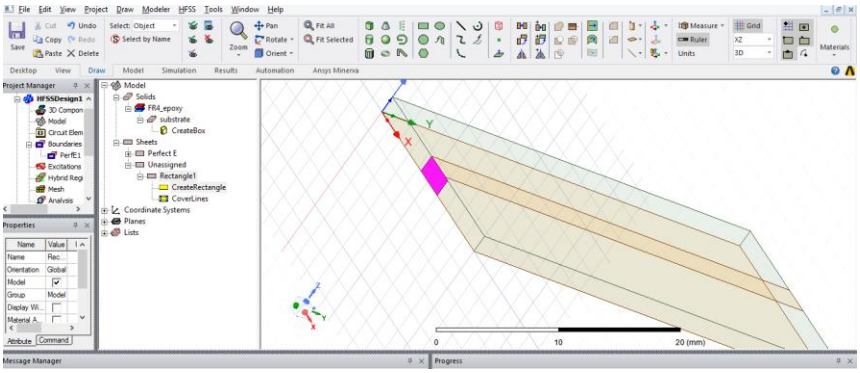


Assign field boundary to ground plane & micro strip line

Using Ctrl Command select both ground & line--> Right mouse click--> Assign Boundary--> Perfect E, Name-->PerfE1-->Enter OK

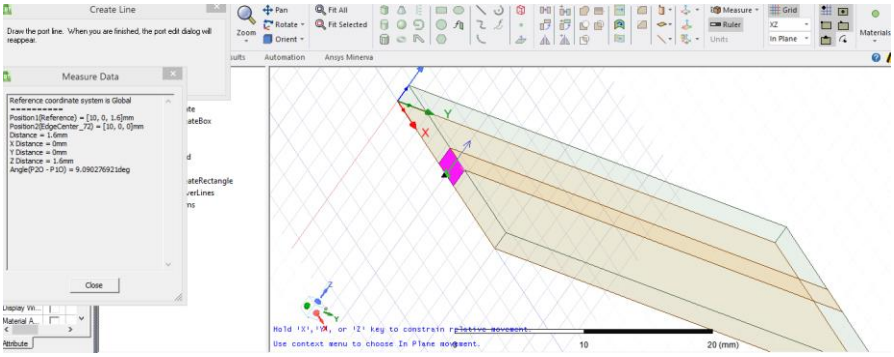


Assign excitation for port 1(input) & port 2(output)
In Draw panel, select X-Z plane



Draw--> Rectangle-->With mouse pointer select face of microstrip line
Change the attributes Model-->Sheets-->Unassigned--> Rectangle1-->Name--> “port1”-->Enter OK
Adjust the port1 dimension to accurate
Model-->Sheets-->Unassigned--> port1-->CreateRectangle
Position-->8.5,0,1.6
XSize-->3 (width of the strip)
Enter OK

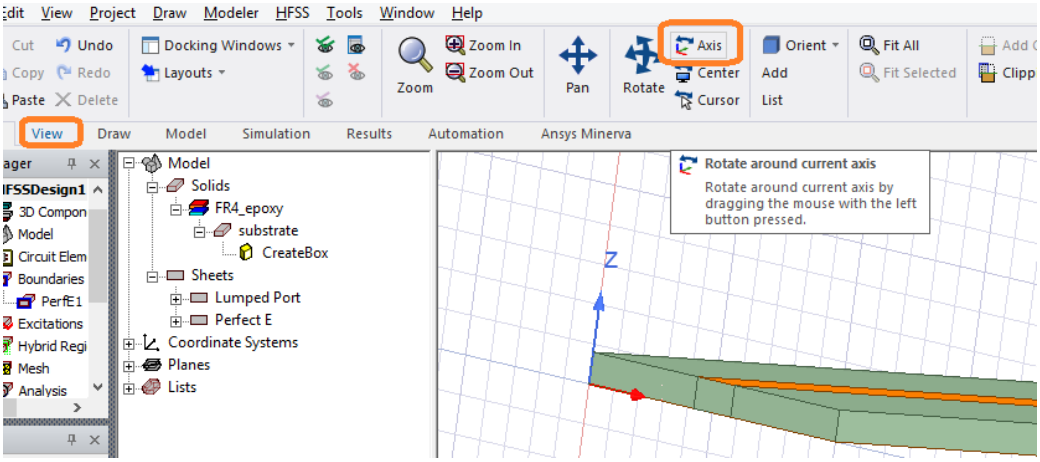
Assign Excitation to port 1
Model-->Sheets-->port1--> Right Click on Mouse, Assign Excitation--> Lumped port-->Name-->1, Next-->Integration Line-->New Line--> Select on face of port 1 from up to down as shown below,

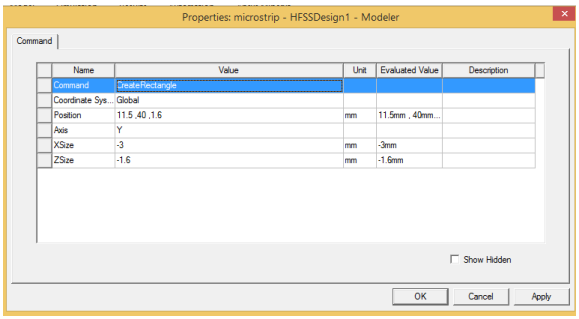


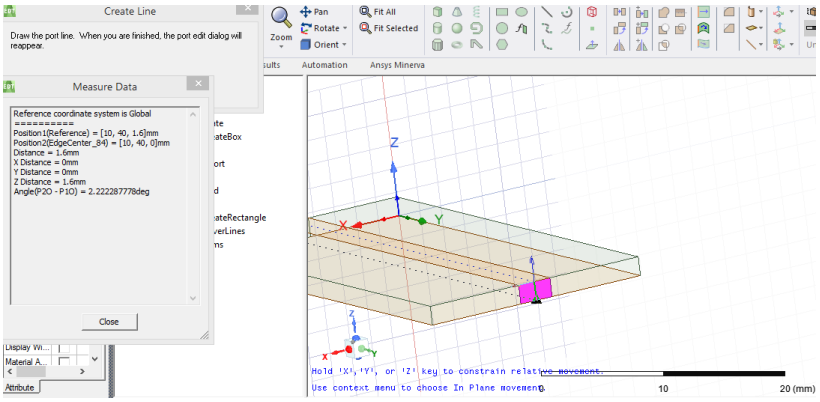
Next--> Enter Finish

Similarly assign port2

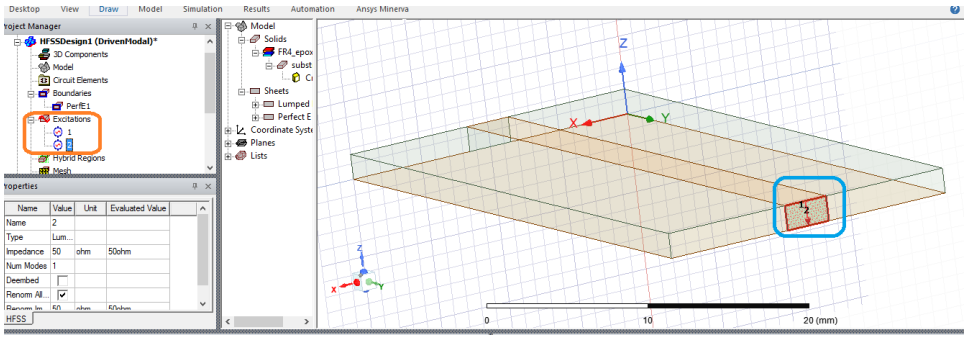
In View panel-->Rotate around current axis



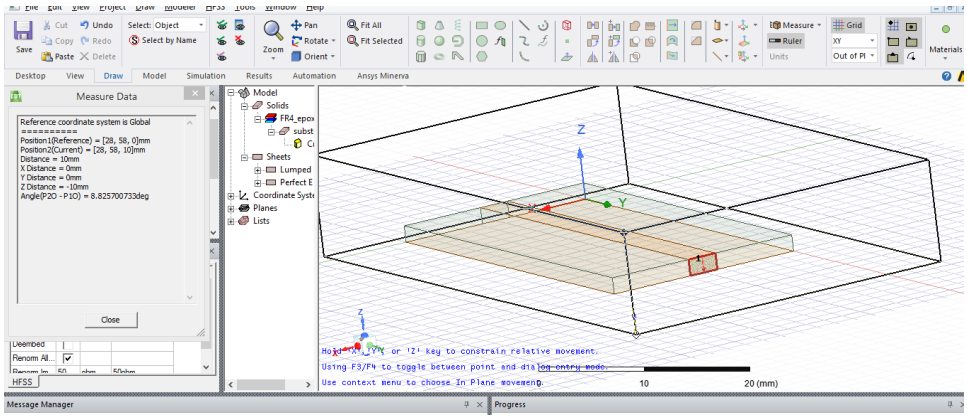




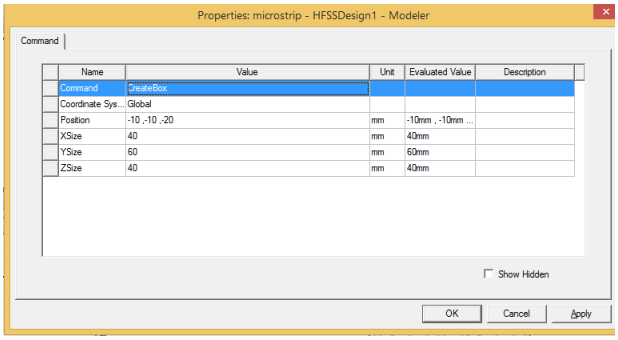
Verify the excitation in the Project manager explorer



Create Radiation Box bigger than design structure
Draw-->Box

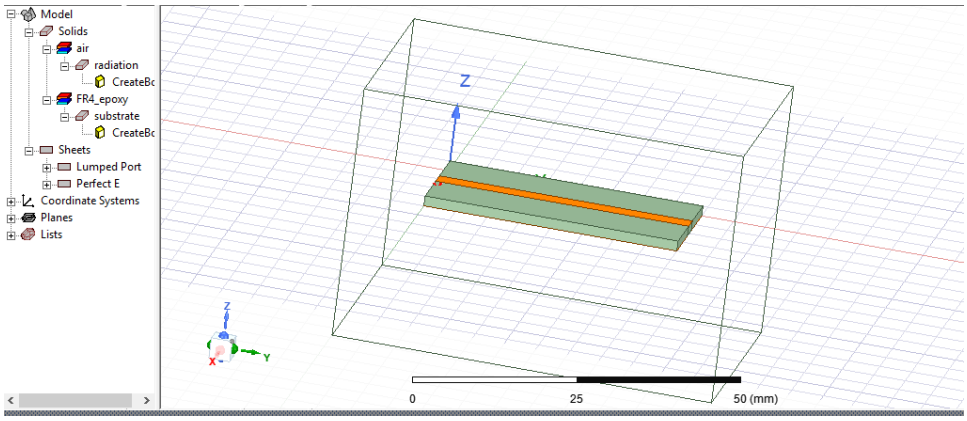


Change the attribute of the Box
Model-->Solid-->vacuum-->Box1-->Name--> “radiation”, material--> Search by name-->air-->Enter OK
Transparency--> 1 (To make design structure inside box visible)-->Enter OK
Fine tune the Radiation box position & dimensions
Model--> Solids-->air-->radiation-->CreateBox
Position: -10, -10, -20
XSize: 40
YSize: 60
ZSize: 40



To fit all structure

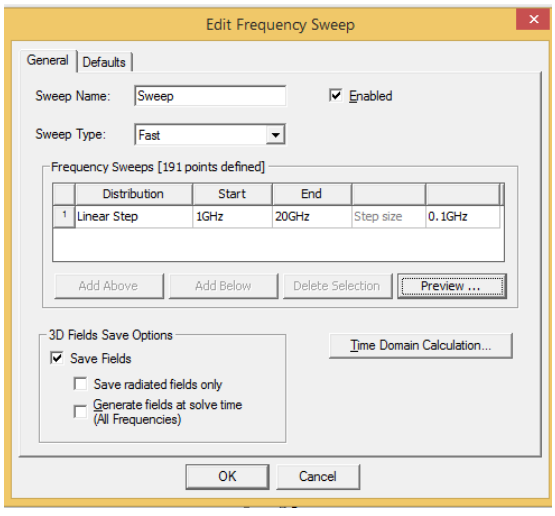
Ctrl +D or View-->Fit all



Assign radiation boundary to box
Model--> Solids-->air-->radiation, Right click on Mouse--> Assign Boundary--> Radiation--> Enter OK
Now micro strip line structure is designed and boundary & excitation are defined.

To analyze the results,

In project manager explorer
HFSS Design-->Analysis--> Right Click--> Add Solution Setup--> Advanced-->Solution Frequency 5 GHz--> Enter OK
Under Edit Sweep frequency
Distribution--> Linear Step--> 1 GHz (Start) 20 GHz (End), Step Size--> 0.1 GHz, Sweep Type--> FastClick on “Preview” to check the number of iterations execution



(Note: User can also use “interpolation” sweep type)
Enter OK

Note: Continue the remaining steps in the lab manual for the results

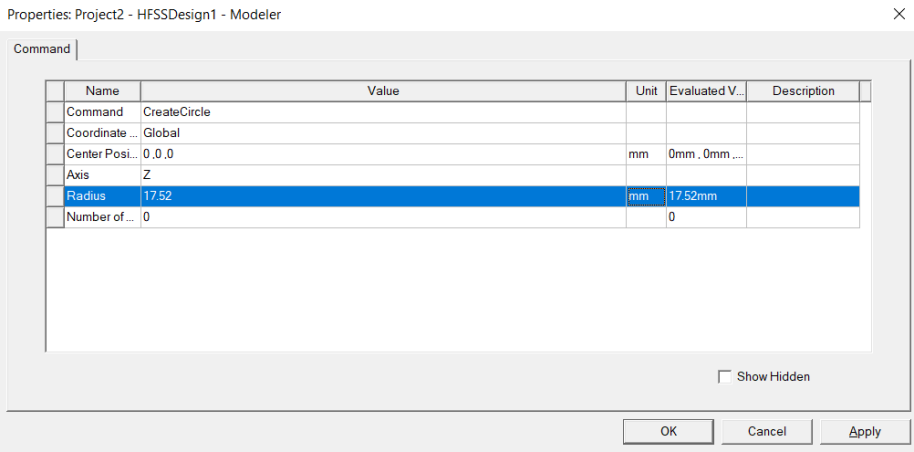
b) The hybrid ring design using microstrip line in HFSS

Draw Circle,

Change the attributes & position of circle as below,

Centre position: 0, 0, 0 mm

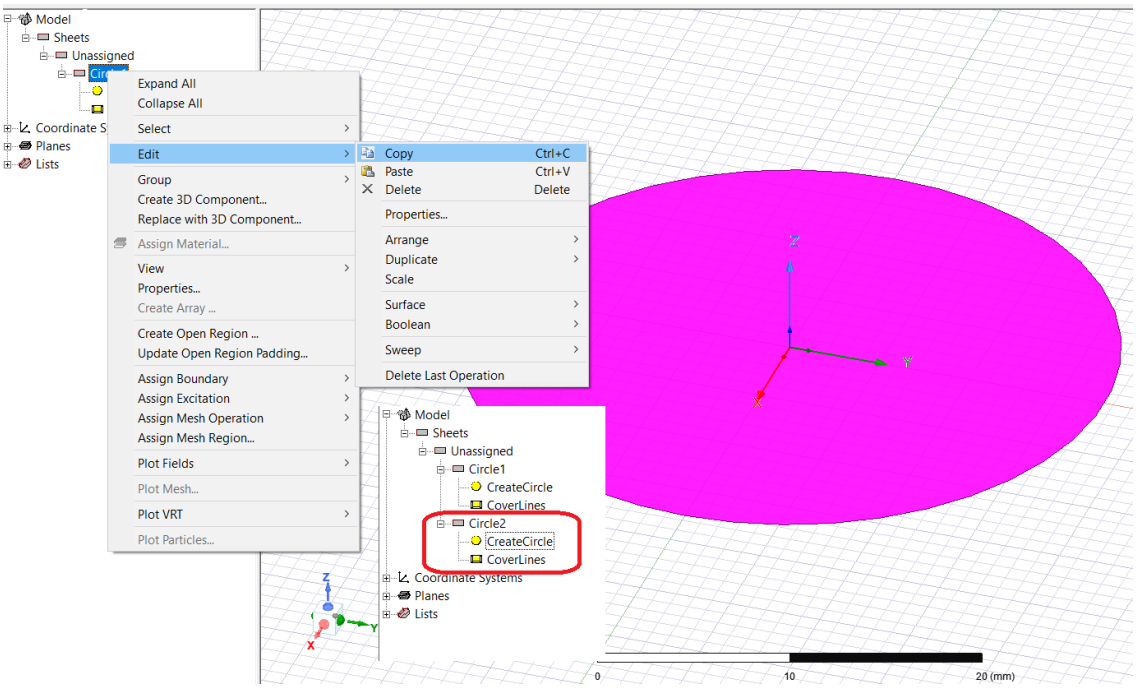
Radius= 17.52 mm (Outer Circle) → Enter OK



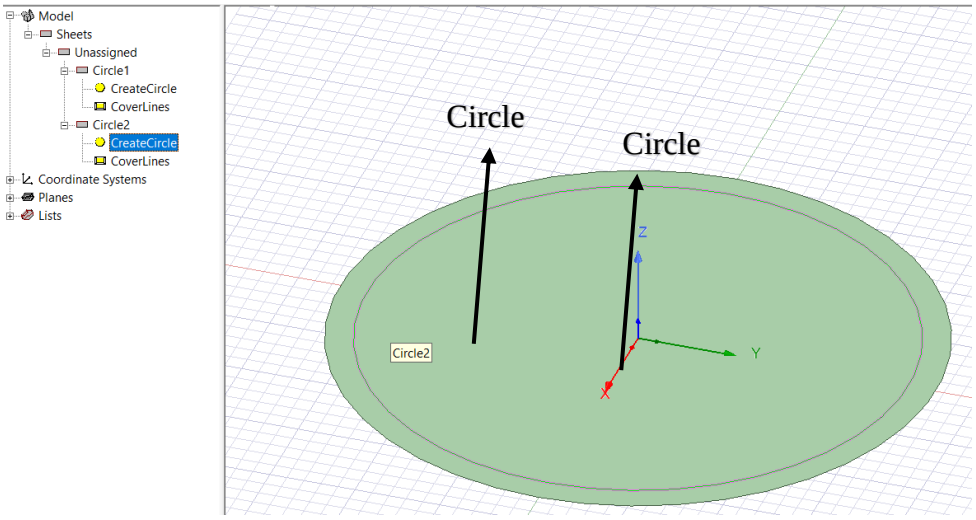
Fit View→ Ctrl+D or View→ Fit all

Duplicate the same circle

Under Model→ Sheets→Unassigned, Right Click on Mouse Circle1, Edit→ Copy and Paste in the Same directory



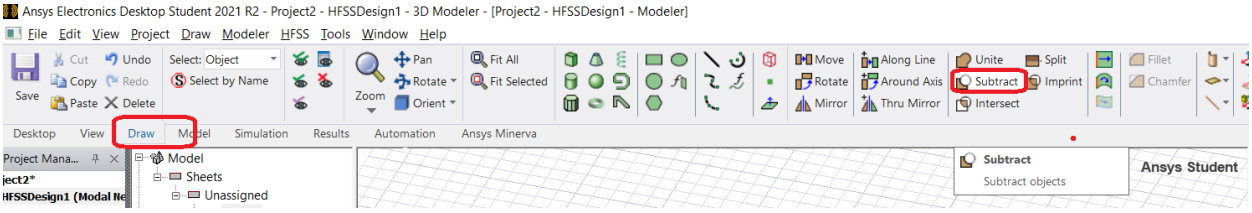
Change the radius of Circle2 as 15.9 mm as mentioned step above.



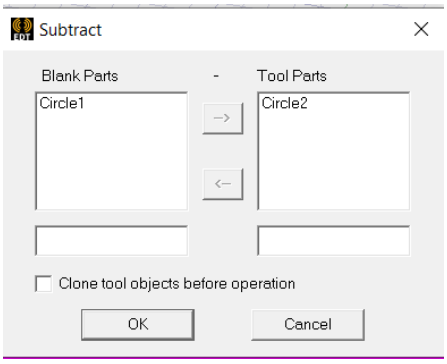
Subtract Circle1-Circle2

Using Mouse pointer, Select Circle1 and Circle 2

Modeler→ Boolean→ Substrate or in Draw panel→ Subtract



In Subtract pane→ Enter OK



Drawing the first port

Change the TOP view orientation by

View→ Modify attribute→ Orientation→ Orientation List→ top→ Apply to View→ Close

Draw rectangle with the port line width

Draw→ Rectangle

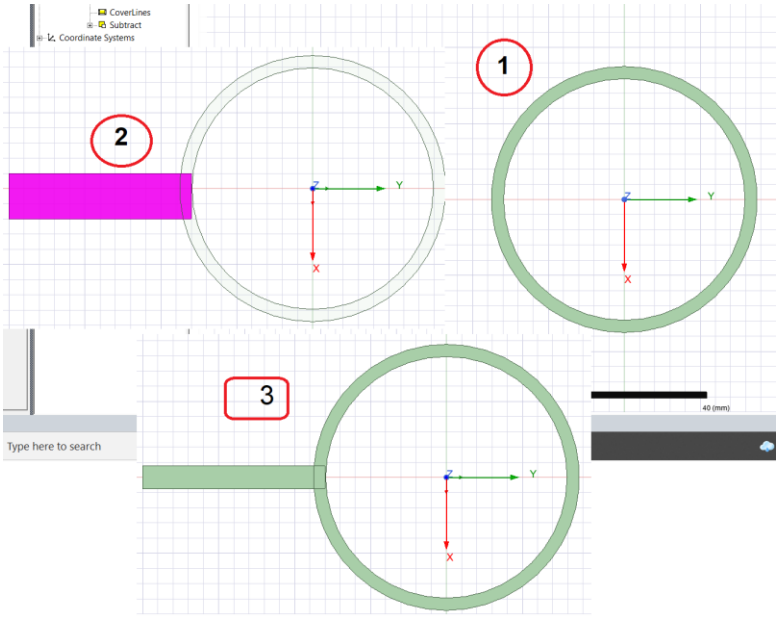
Model→ Sheet→ Rectangle1→CreateRectangle

Position: -1.5, -40, 0 (since width is measure along X-axis along origin, the X range is (-1.5 to 1.5 mm, total 3mm and Y-axis position -40 + YSize of 24 =-16 position which overlap on ring)

XSize→ 3 mm

YSize→24 mm

Enter OK



Note: The port line length can be shortened by changing the position & size of rectangle. For e.g,

In Model→ Sheet→ Rectangle1→CreateRectangle

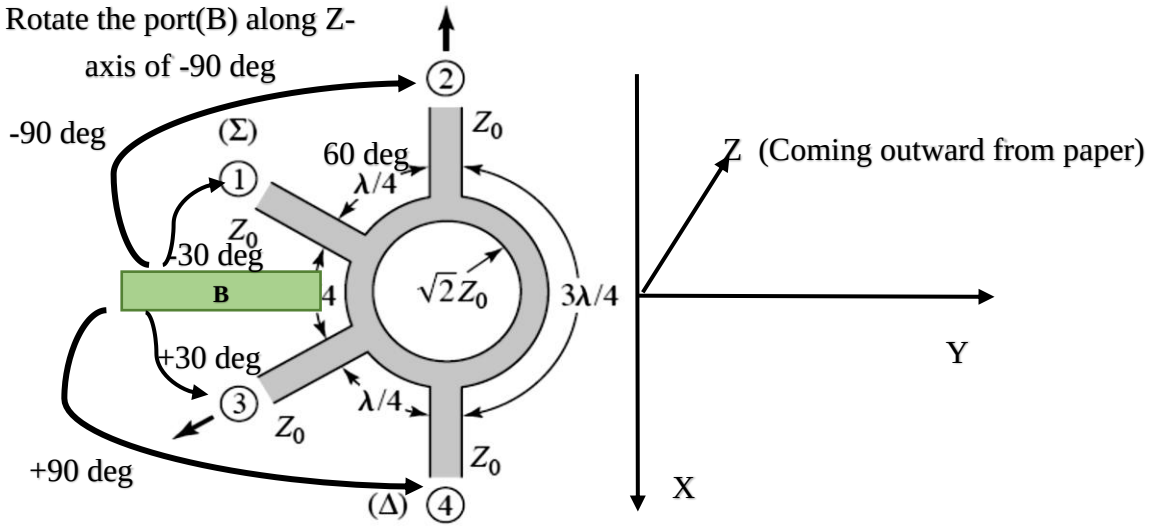
Position: -1.5, -30, 0

XSize: 3

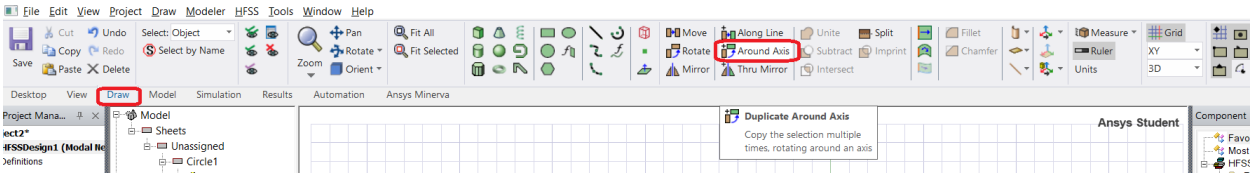
YSize: 14

Draw other ports

From the design, duplicate and create other ports



Select the port and in draw panel, Duplicate around Axis→ Axis→Z, Angle→-90 deg, Total number→2 (port 2)



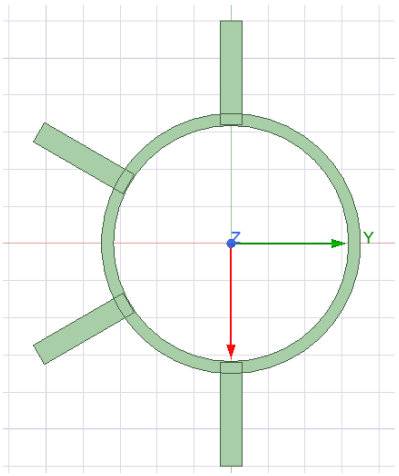
Similarly,

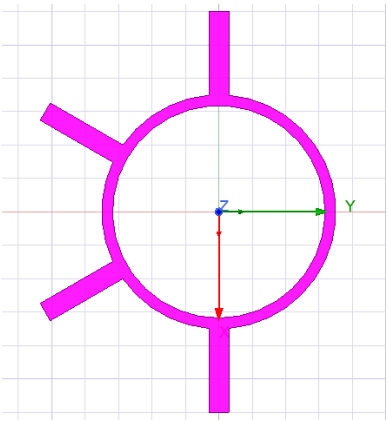
Select the port and in draw panel, Around Axis→ Axis→Z, Angle→90 deg, Total number→2 (port 4)

Select the port and in draw panel, Around Axis→ Axis→Z, Angle→-30 deg, Total number→2 (port 1)

Move the remaining port to +30^{o} degree

Select the port and in draw panel, Rotate→ Axis→Z, Angle→30 deg (port 2)





Select ALL objects and unite

In Draw panel→ Unite (or) Modeler→ Boolean→ Unite

Change the properties of ALL objects as “ring” by changing it attribute Name as “ring”

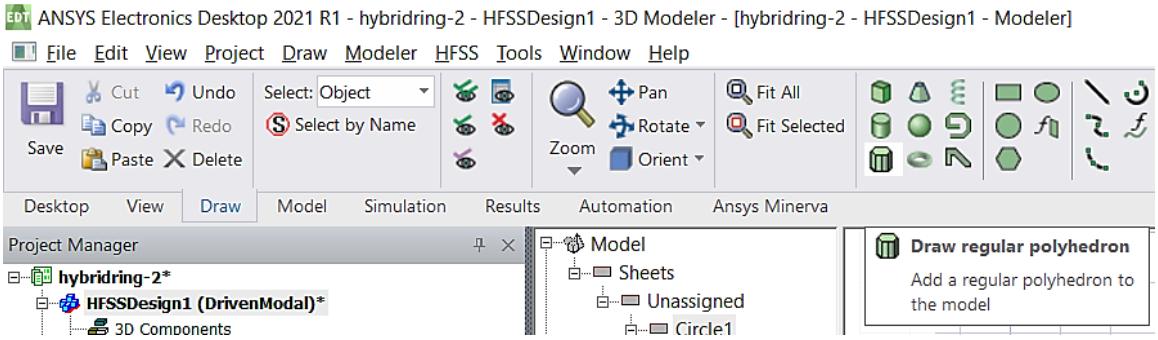
Draw the substrate & ground plane beneath hybrid ring

Select ALL Object→Edit→ Select All,

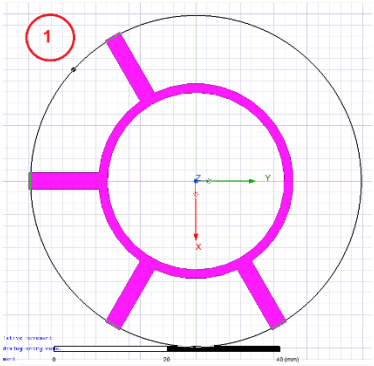
In Draw panel, Rotate→Axis→Z, Angle→30 deg→ Enter OK

Creating ground plane:

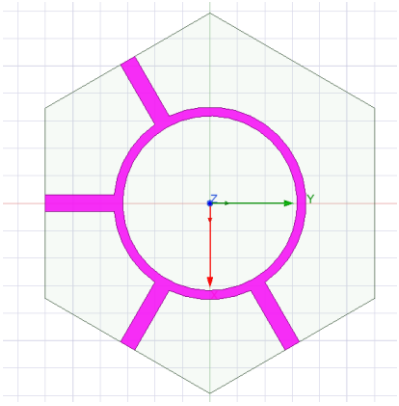
Draw regular polyhedron



Use mouse pointer, select origin and draw the circle and choose Number of segments: 6

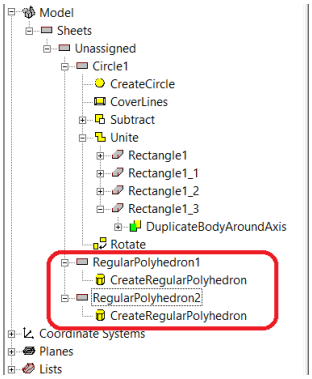


Choose the properties of polyhedron (ground plane),
Model→ Sheets→Unassigned→RegularPolyhedron1→ CreateRegularPolyhedron
Select properties as,
Centre position: 0, 0, 0
Start position: 30mm/cos(30*pi/180) ,0mm ,0mm (**Note:** radius of polygon is port line length/ cos(30*pi/180). Pythagoras theorem)
Axis: Z
Height: 0 mm



Duplicate polyhedron for substrate:

Select the ground plane, Model→ Sheets→Unassigned→RegularPolyhedron1, Right Mouse Click→ Edit→ Copy & Paste in same location (RegularPolyhedron2 will be created)



Choose the properties of polyhedron (substrate),

Model→ Sheets→Unassigned→RegularPolyhedron2→ CreateRegularPolyhedron

Select properties as,

Centre position: 0, 0, 0

Start position: 30mm/cos(30*pi/180) ,0mm ,0mm (**Note:** radius of polygon is port line length/ cos(30*pi/180). Pythagoras theorem)

Axis: Z

Height: -1.6 mm

Assign substrate material,

Solid→ Vacuum→ RegularPolyhedron2→ Material→ Edit→Search by Name→FR4_epoxy

Move the ground-plane polyhedron beneath substrate

Draw panel → Orient→ Trimetric

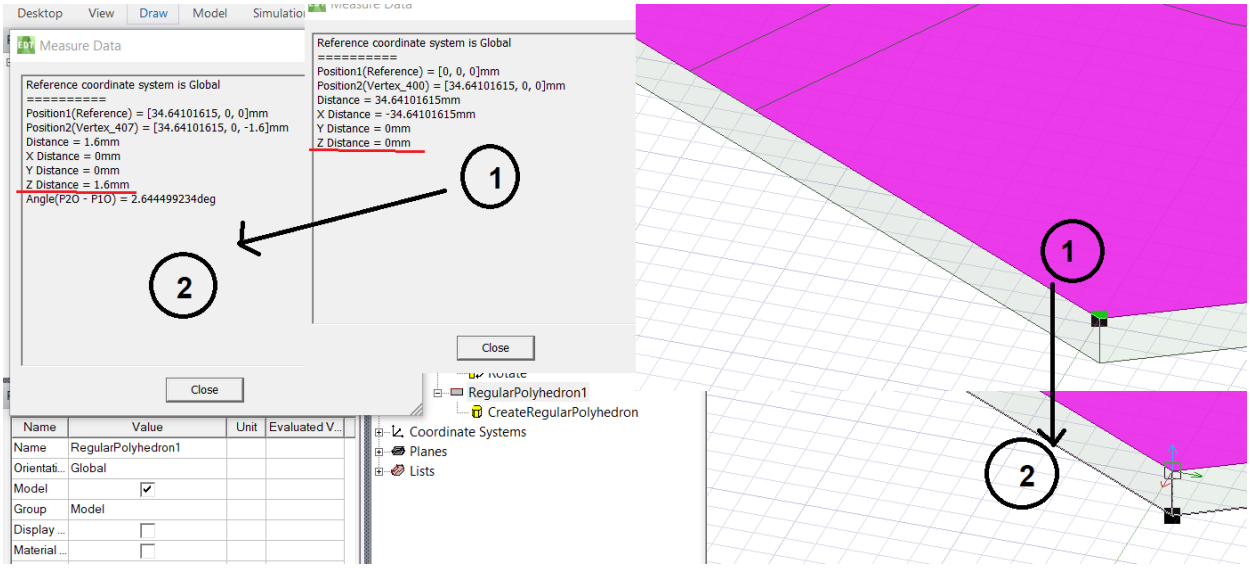
Select Face of RegularPolyhedron1,

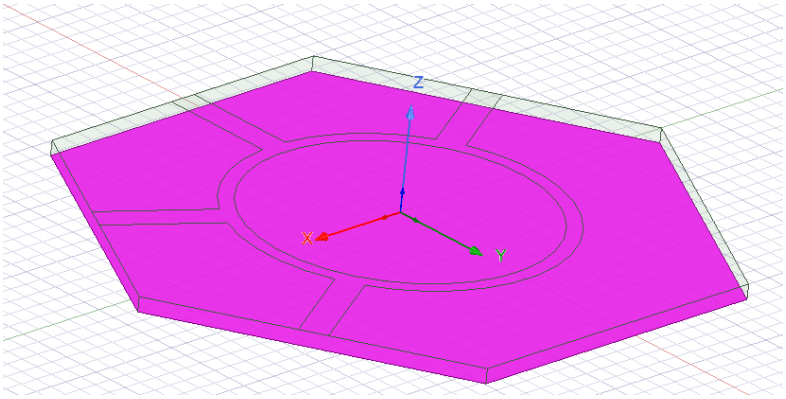
In Draw panel, Select→Select by Name→ RegularPolyhedron1

Move in Z-Axis

Draw panel→ Move

Select the corner of RegularPolyhedron1 and move down in Z-axis by -1.6mm





Place the rectangle sheet in all port as a lumped port

Note: Every time rotate 30/60^{o} degree and draw rectangular sheet for each port

Select ALL Object, Edit→ Select All

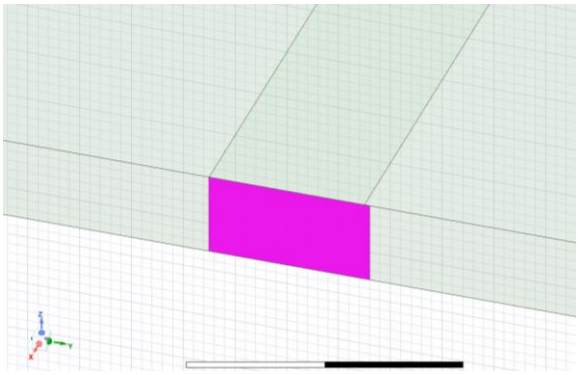
Draw panel→ Rotate→ Axis: Z, Angle→ 30 deg

Drawing rectangle on each port:

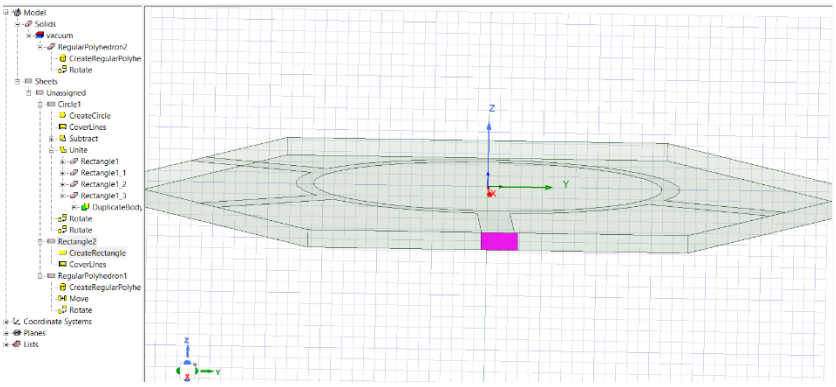
In Draw panel→

Change the drawing plane choose the plane accordingly (YZ)

Draw→ Rectangle on every port

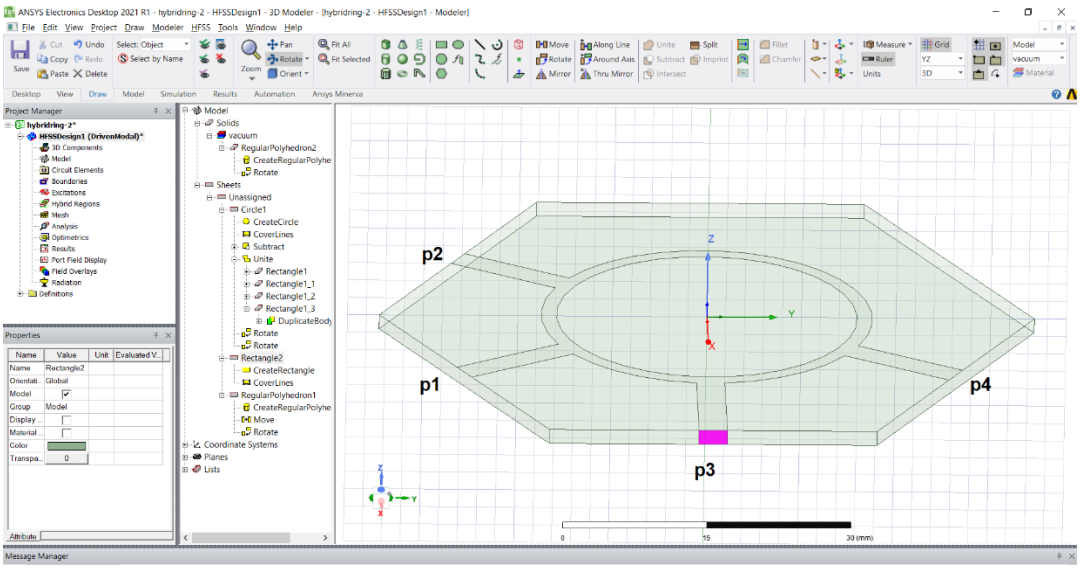


For e.g port 3 (Note: In view panel, Use Pan & Zoom in to draw)



Adjust the dimension of rectangle to match with width of strip line.

(Rectangle2→ CreateRectangle→ YSize→3 mm



Before rotating the object,

Change the orientation: Draw→Orient->Top(-Z)

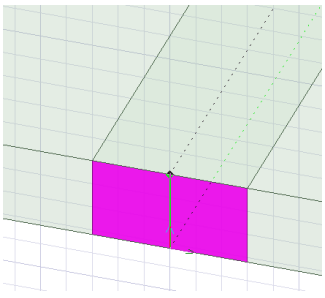
Select ALL Object, Edit→ Select All

Draw panel→ Rotate→ Axis: Z, Angle→ 60 deg (for port 1 & 2) 180 deg (for port 4)

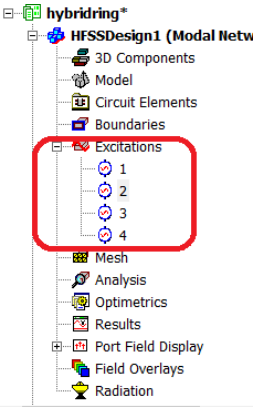
Draw→ Rectangle on every other port (Hint: Use view→ Fit all, Rotate around axis to choose other ports for drawing).

Assign Port excitation

1. Select the port face object individually→ Right mouse click→ Assign excitation→Port→ Lumped port→Name→1 or 2 or 3 or 4 (Provide the name according to hybrid ring design)→ Integration line→ New line→Select on face of port from bottom to up at middle(Ref microstrip line)



2. Assign excitation for remaining ports. (Note: While selecting integrating line on face of port object, keep relevant plane of reference either XY or YZ or XZ)
3. Excitation port can be verified in project manager under excitation numbered 1 to 4.



4. Assign Boundary to ring and ground plane
    a. Select the ring, Model→ Sheet→ Unassigned→ring than
    b. Right mouse click→ Assign the Boundary→ Perfect E→ Enter OK
    c. Select the ground plane, Sheets→ Unassigned→ RegularPolyhedron1 than
    d. Right mouse click→ Assign the Boundary→ Perfect E→ Enter OK
    e. Create Radiation bigger enough to accommodate hybrid ring
5. Draw Box, In Attribute→Position→ (-50, -50, -50)
    a. XSize:100
    b. YSize:100
    c. ZSize:100
6. Assign material as “Air” to Box, Solids→ vacuum→Box2→Right Mouse click→Assign material→ Search by Name→ air.
7. Increase the transparency of Box to view the device structure inside the box
    a. Solids→ air→Box2→Right Mouse click→properties→transparency→ 0.7 to 1
8. Assign Boundary as Radiation to Air Box
    a. Solids→ air→Box2→Right Mouse click→Assign the Boundary→ Radiation→ Enter OK

Note: Continue the remaining steps in the lab manual for the results

EXPERIMENT 7

Design and Simulation of Rectangular Waveguide and Magic-Tee using HFSS

a) Simulation of Rectangular Waveguide

HFSS: Getting Started

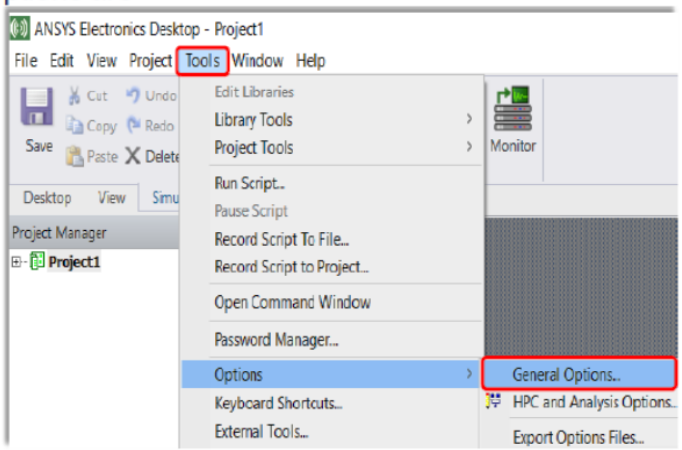
CADFEM

Launching ANSYS Electronics Desktop

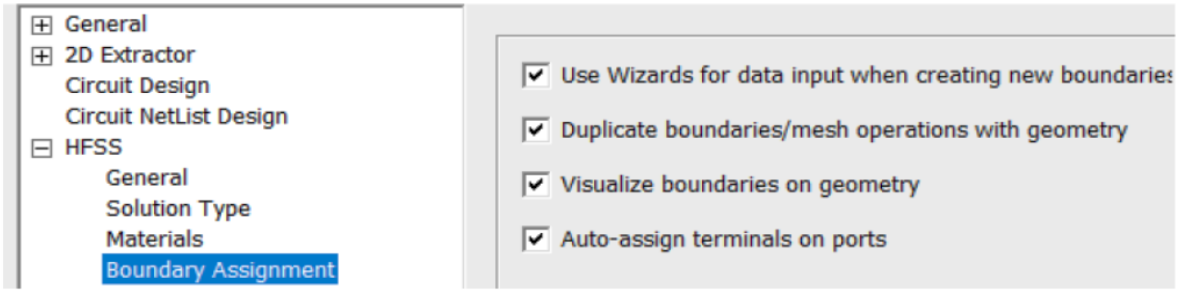
- To access ANSYS Electronics Desktop, click the Microsoft Start button,
  - Select Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite. Select ANSYS Electronics Desktop

Setting Tool Options

- Note: In order to follow the steps outlined in this example, verify that the following tool options are set :
- Select the menu item Tools > Options > General Options…
  - Expand the HFSS Branch
    - Left Mouse Click Boundary Assignment
      - Use Wizards for data input when creating new boundaries: ☑ Checked
      - Duplicate boundaries/mesh operations with geometry: ☑ Checked



Options



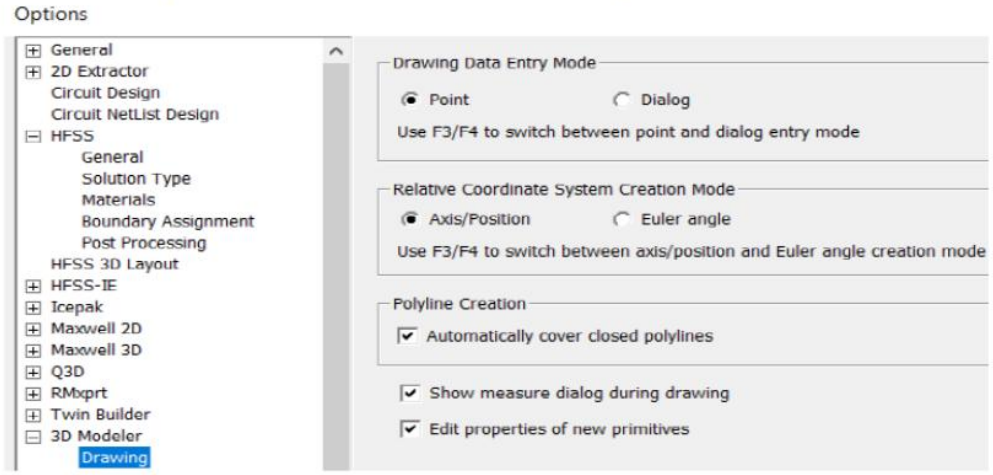
© CADFEM 2020

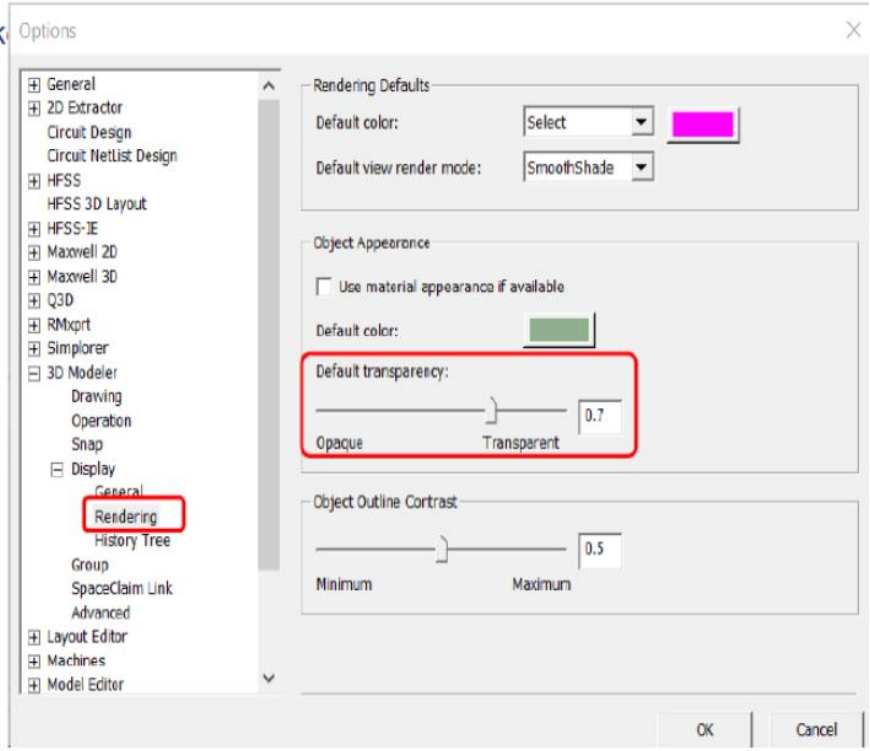
HFSS: Getting Started

CADFEM

Setting Tool Options (Continued)

- Expand the 3D Modeler Branch
  - Drawing: Edit properties of new primitives: ☑ Checked
- Expand the Display Branch
  - Rendering Set Default Transparency to 0.7
  - History Tree: Select last command on object/submodel select: ☑ Check





- Click the **OK** button

© CADFEM 2020

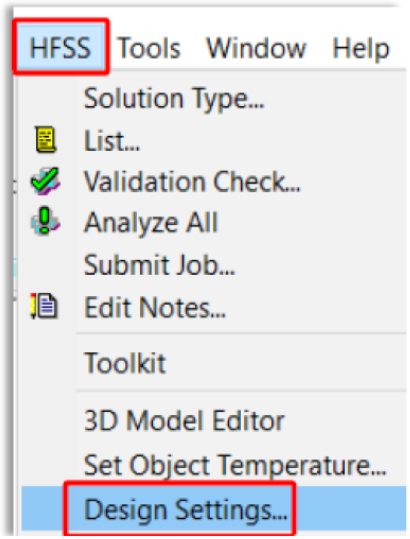
HFSS: Getting Started: Enable Material Override

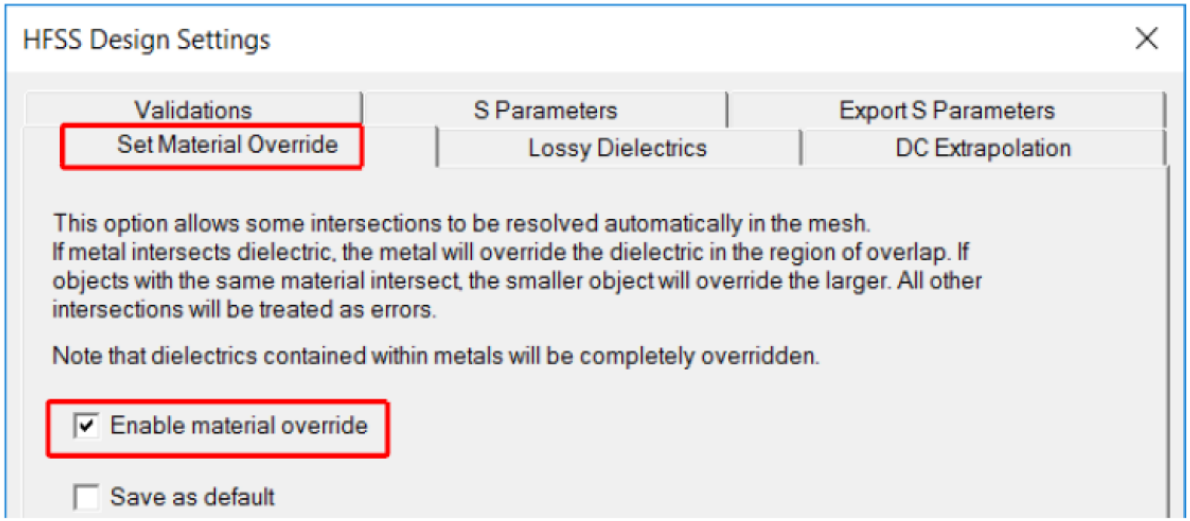
CADFEM

Sometimes it is inconvenient to manually subtract partially overlapping solids (ex: via in PCB).
Enable Material Override adds additional rules to help the 3D Model Editor resolves partial overlaps

1. If the overlap exists between a good conductor and a dielectric the good conductor takes precedence
2. Overlaps between solids of the same material are reduced to warnings messages

Enable Material Override eliminates the need to subtract the vias from the substrates. HFSS automatically performs the subtraction because the metal vias override the dielectric substrate in the region of overlap.
From HFSS's menu bar HFSS > Design Settings: Enable material override: ☑ Checked





- **Click the OK button**

© CADFEM 2020

Simulation of a WR-90 Rectangular waveguide

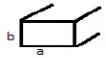
CADFEM

1. Start HFSS and define geometry
   - Define Inner wall of waveguide
   - Define outer wall of waveguide
   - Boolean subtraction to obtain hollow waveguide
   - Inside of waveguide as vaccum
   - Radiation boundry

2. Define Excitation (Wave port)
3. Add solution setup (Freq Sweep)
4. Valiodate and simulate
5. View results

Design parameters

WR-90 Operating Frequency is X band (8.4 GHz to 12.4 GHz)



Standard dimentions (a X b) = 22.86 mm X 10.16 mm
(0.2 mm thickness of waveguide wall, length of 60 mm)
TE10 cutoff Freq = 6.56 GHz
TE20 cutoff Freq = 13.12 GHz
TE01 cutoff Freq = 14.76 GHz
TE11 cutoff Freq = 16.16 GHz

$$f_{c_{mn}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}}\sqrt{\left(\frac{m\pi}{a}\right)^2 + \left(\frac{n\pi}{b}\right)^2}_{m,n=0,1,\ldots}$$

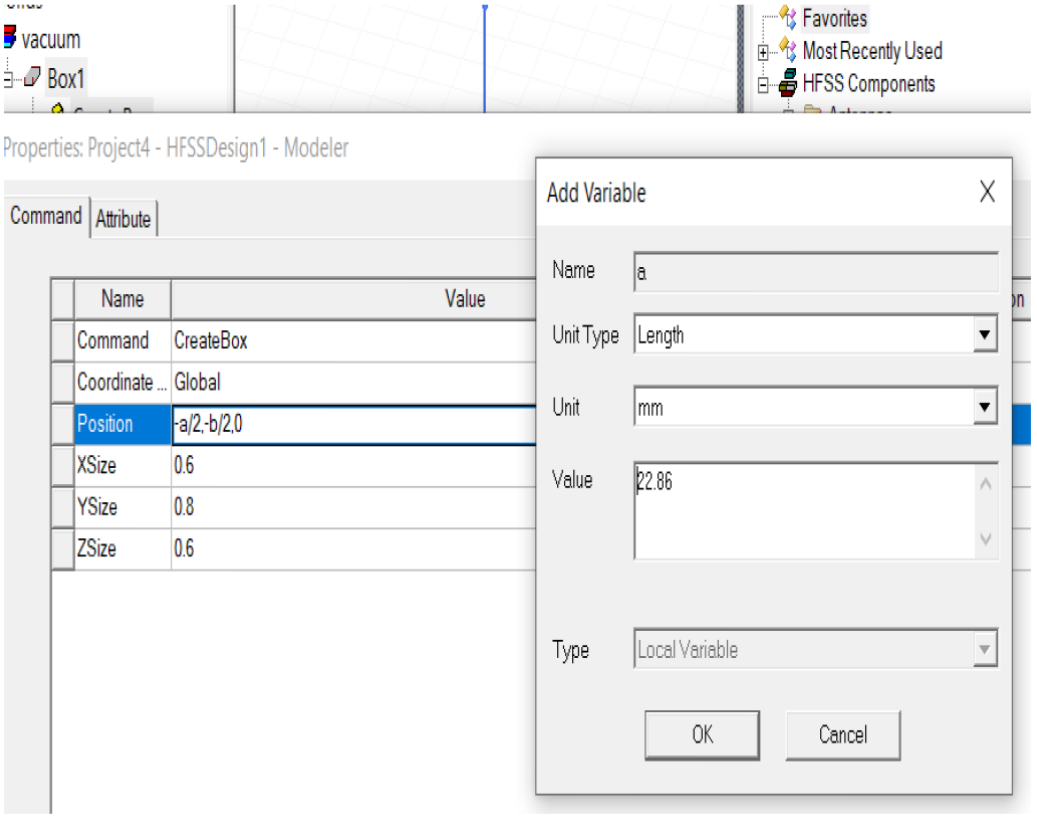
For TE_{10} dominant mode,

$$f_{c_{10}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}}\sqrt{\left(\frac{m\pi}{a}\right)^2} = \frac{1}{2\sqrt{\mu\varepsilon}}\frac{1}{a} = \frac{c}{2a} = \frac{3\times10^8}{2\times22.86\times10^{-3}} = 6.56\,\text{GHz}$$

© CADFEM 2020

Define Inner wall of waveguide

Select> HFSS > Solution Type > Model
Select> Modeler > Units > mm
Define Geometry: Draw Box > click three times in the main area
In Box properties tab > define position as
Position: -a/2,-b/2,0
In automatically opened ass variable windows,
we can define the parameters we have used:
Unit Type = Length
Unit: mm
Value for a = 22.86
Value for b = 10.16
Now we can use those variables throughout the rest of the exercise
Xsize: a
Ysize: b
Zsize: L
Click ok



Define Inner wall of waveguide

Again we define the parameter ‘L’, representing the length of the waveguide along the Z-direction
Unit Type = Length
Unit: mm
Value for L = 60, ok
Press ‘Ctrl’ and ‘D’ to fit view

Properties: Project4 - HFSSDesign1 - Modeler

Command | Attribute

Name	Value	Unit	Evaluated Value	Description
Command	CreateBox			
Coordinate ...	Global			
Position	-a/2 ,-b/2 ,0mm		-11.43mm , -5.08mm , 0m...	
XSize	a		22.86mm	
YSize	b		10.16mm	
ZSize	L		60mm	

Define Outer Wall of Waveguide

CADFEM

We will copy the inner wall definition and then modify it to give the waveguide a thickness.

In the model tree, under solids, vacuum, click on the box object Box 1 and copy (ctrl+c) and paste (ctrl+v) it in the same place, notice Box 2 is now created

Double click on box2 to open the attribute tab:

Name: Metal

Material: Copper

Note that this box is now defined in the modeler tree as a solid copper object.

Open up the command tab for this airbox by double create box under the copper solid now called metal and change the properties as follows:

Positon: -a/2-th,-b/2-th,0

Define Outer Wall of Waveguide

In the automatically opened Add variable window, we define the parameter ‘th’ we have used to denote the thickness of the waveguide wall:

Unit Type = Length

Unit: mm

Value for th = 0.2

Xsize: a+2*th

Ysize: b+2*th

Click ok

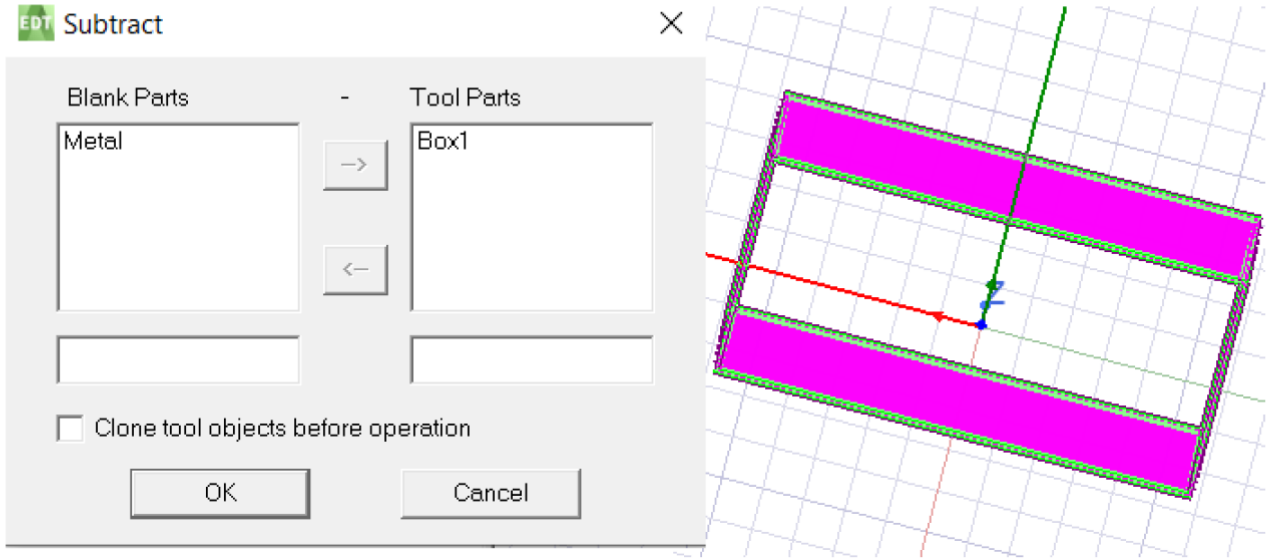
Properties: Project4 - HFSSDesign1 - Modeler

Command

	Name	Value	Unit	Evaluated V...	Desc
	Command	CreateBox			
	Coordinate ...	Global			
	Position	-a/2-th ,-b/2-th ,0mm		-11.63mm , -...	
	XSize	a+2*th		23.26mm	
	YSize	b+2*th		10.56mm	
	ZSize	L		60mm	

Boolean Subtraction to obtain Hollow waveguide

In the modeler tree, click on Box 1 and hold down ctrl and select Metal, then use the menu modeler>Boolean>subtract
Ensure blank parts has metal and tool parts has air and click ok (Can give different colour)



Define inside of waveguide as air

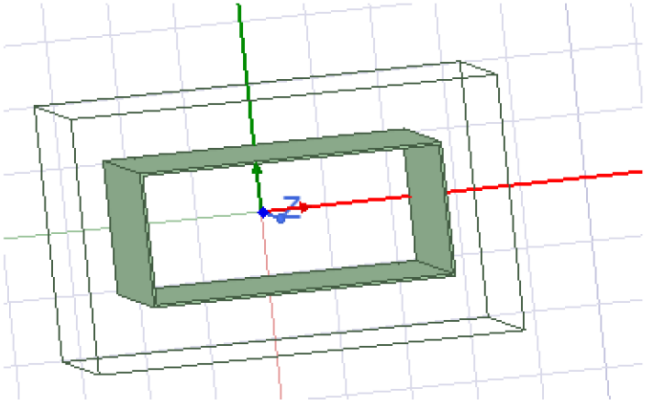
We will create another box to define the inside of the hollow waveguide as vacuum
In the modeler tree, select Box 1 and copy and paste
With the Not assigned solid, we will apply the following properties:
Name: Air
Material: air
Transparent:1

Define outside of waveguide + Rad Boundary

In the modeler tree, select air and copy and paste it
With the new air1 solid, we will apply the following properties
Name: Radiation
Material: air
Transparent: 1

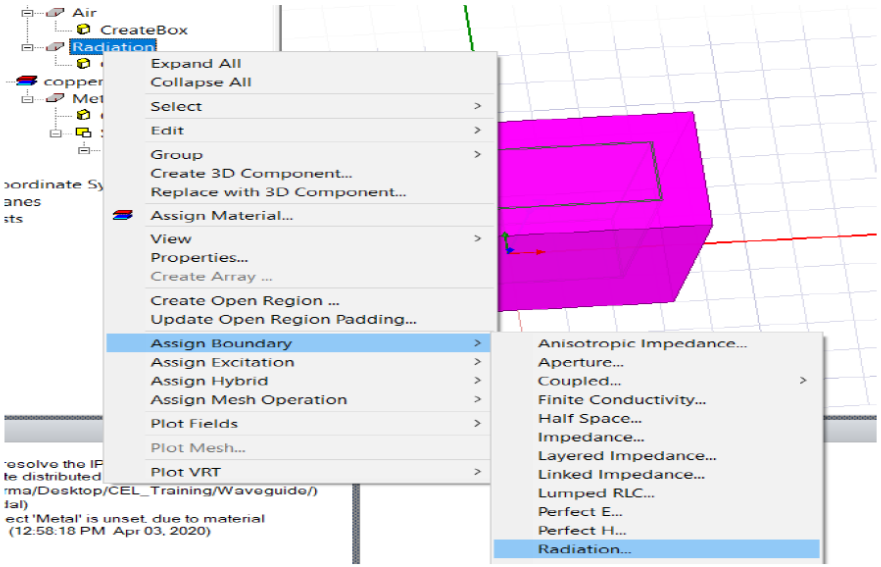
Position: -a/2-5mm, -b/2-5mm,0
Xsize: a+10mm
Ysize: b +10mm
Zsize: L

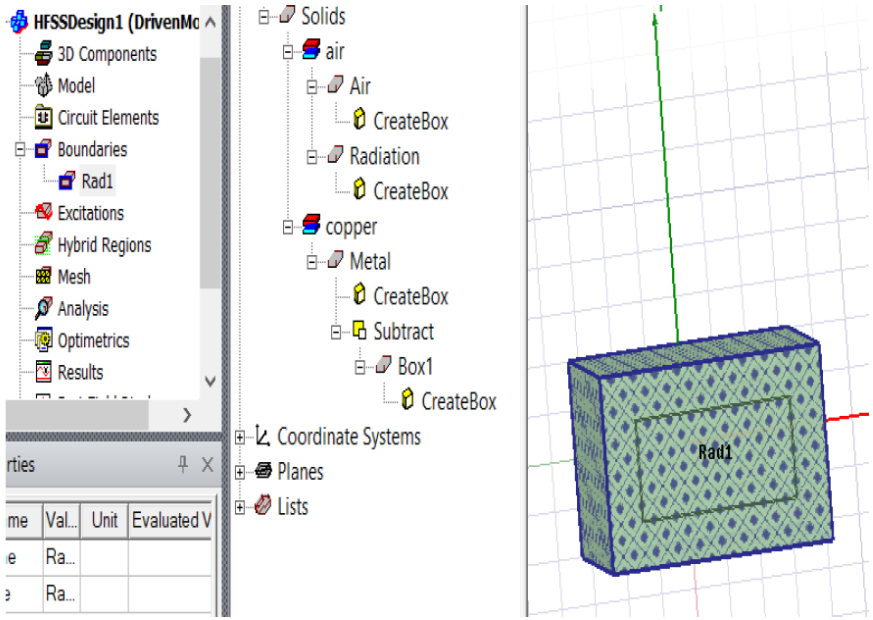
	Name	Value	Unit	Evaluated V...
	Command	CreateBox		
	Coordinate ...	Global		
	Position	-a/2-5mm ,-b/2-5mm ,0mm		-16.43mm , -...
	XSize	a+10mm		32.86mm
	YSize	b+10mm		20.16mm
	ZSize	L		60mm



Define outside of waveguide + Rad Boundary

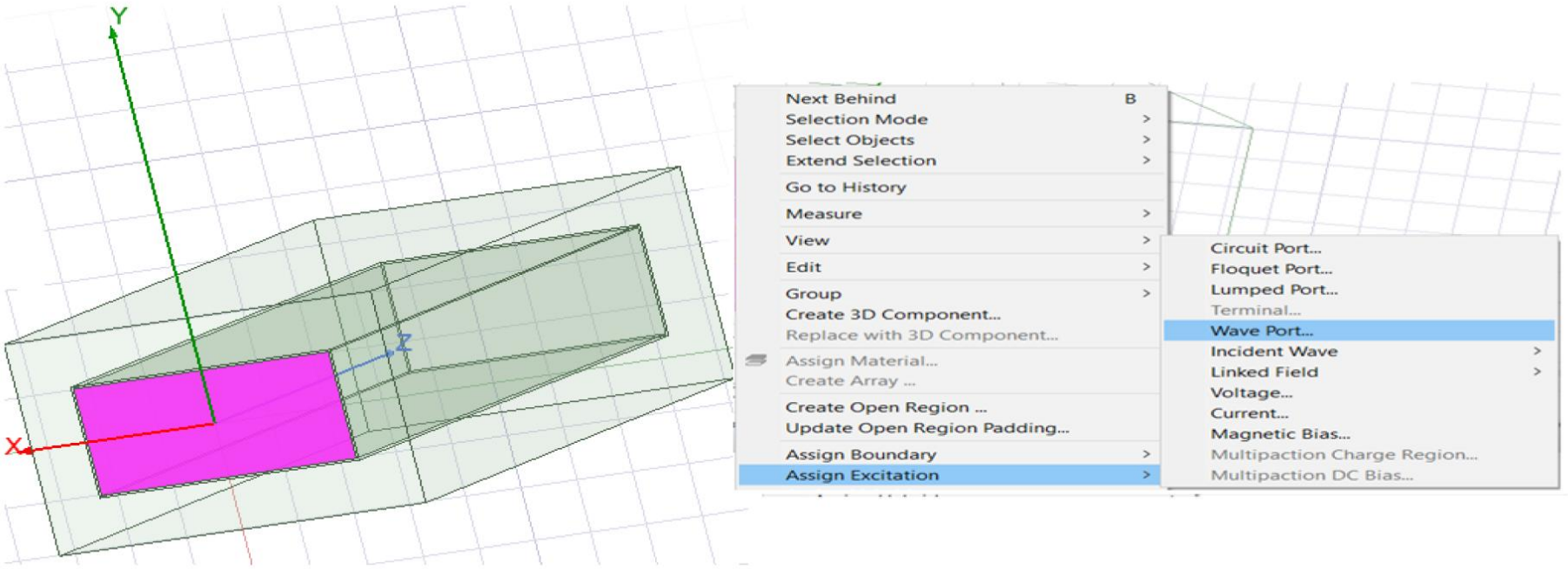
New assign this box as a radiation boundary
In the Modeler tree, RC on radiation solid>Assign Boundary>radiation>ok





Define Wave Ports

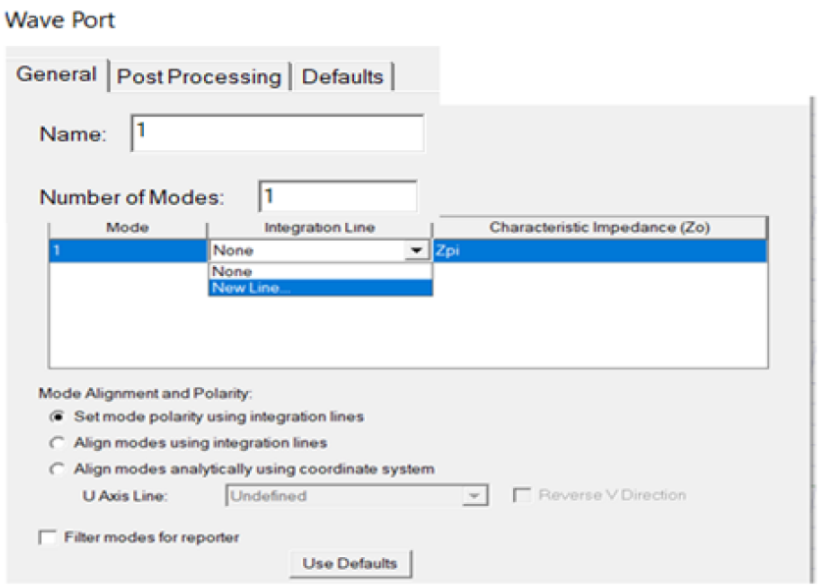
In the main window, click outside the waveguide, On your keyboard, press F to enter face selection mode
Select face on the x-y plane at the entrance of the waveguide
(By left clicking on it as shown below) and R click on face>Assign Excitation>Wave port (as shown below)

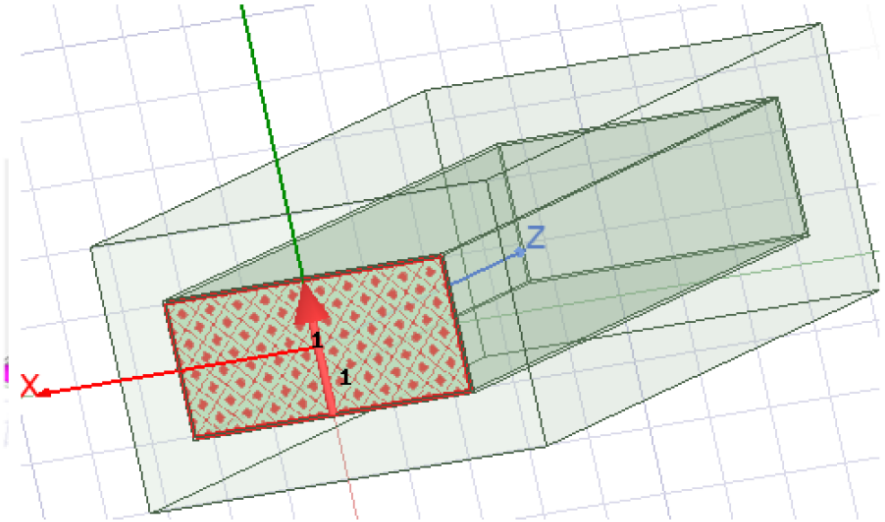


(Note: Wave port is used to solve the fields for the 2D port scenario and excite the 2D structure with it)

Define Wave Ports

In the Wave port window, click on Integration line > new line > in the 3d model editor, click on the Centre bottom and top of the selected face to define the line as shown below:



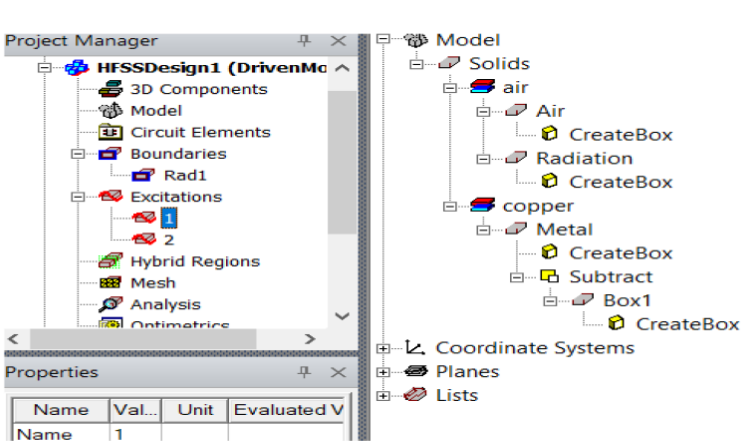


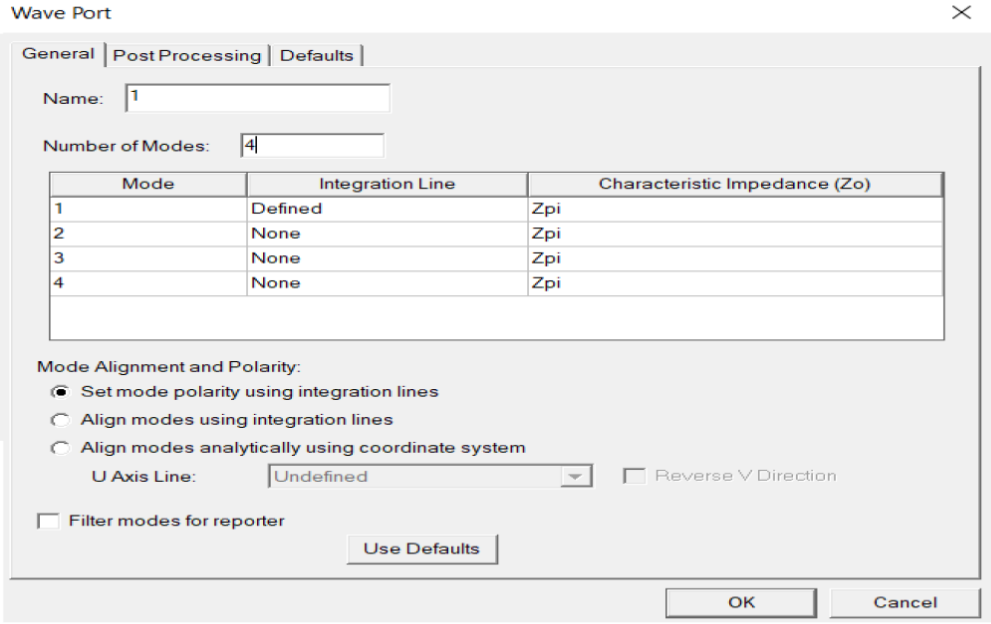
Click Next and then Finish (We do not want to renormalize to 50 OHM)
Repeat the same process for opposite face

Define Wave Ports

Set number of modes 4 to simulate 4 modes.

In project manager, open Excitation and click on 1 and set number of modes to 4 as shown below (We do not need to redefine integration line)

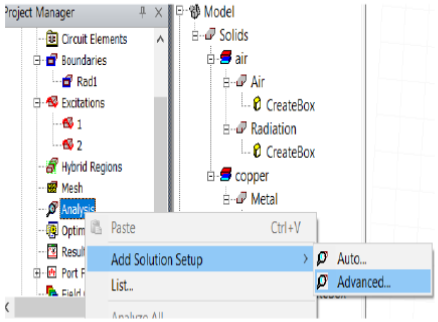


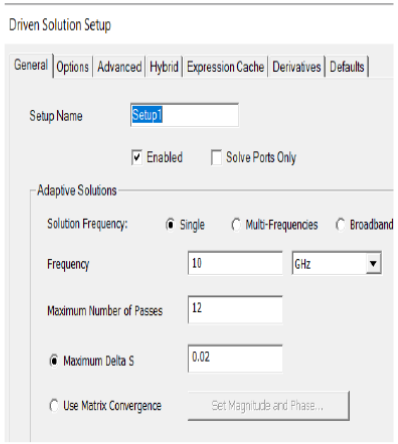


Repeat the same process for Wave port 2 and set Modes to 4.

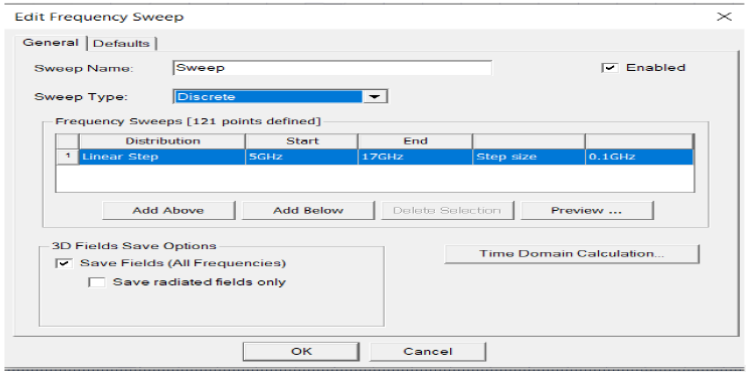
Add Solution Setup

In project manager window click on analysis> Add solution Setup> Advance and use below settings:
Freq: 10 GHz
Num of Pass: 12 > ok





In project manager window, open up analysis, RClick on setup1> Add freq Sweep and use below settings:
Sweep Type: Discrete
Distribution: Linear step
Start: 5 GHz
End: 17 GHz
Step: 0.1 GHz
Save Field: All frequencies Checked



© CADFEM 2020

Note: Continue the remaining steps in the lab manual for the results

b) **Design and Simulation of Magic-Tee using HFSS**

Magic Tee HFSS Procedure

Setting Tool Options

•Note: In order to follow the steps outlined in this example, verify that the following tool options are set :

•Select the menu item Tools > Options > General Options…

•Expand the HFSS Branch

Left Mouse Click Boundary Assignment

•Use Wizards for data input when creating new boundaries:

☑ Checked

•Duplicate boundaries/mesh operations with geometry:

☑ Checked

•Expand the 3D Modeler Branch

Drawing: Edit properties of new primitives:

☑ Checked

Expand the Display Branch

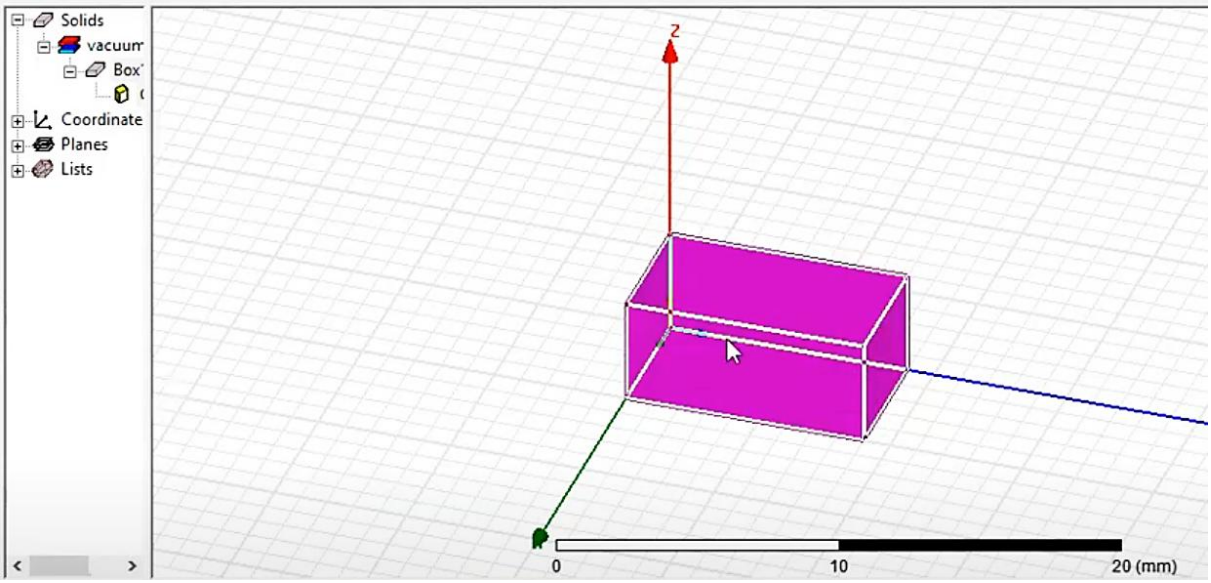
Rendering Set Default Transparency to 0.7

History Tree: Select last command on object/submodel select:

☑ Checked

a) Draw the Box and its dimension

Draw→ Box→Use Mouse Pointer→ Select the Origin (0,0,0) as position and draw(Ref Fig below)



b) Changing the dimensions

Model→vacuum→ Box→CreateBox

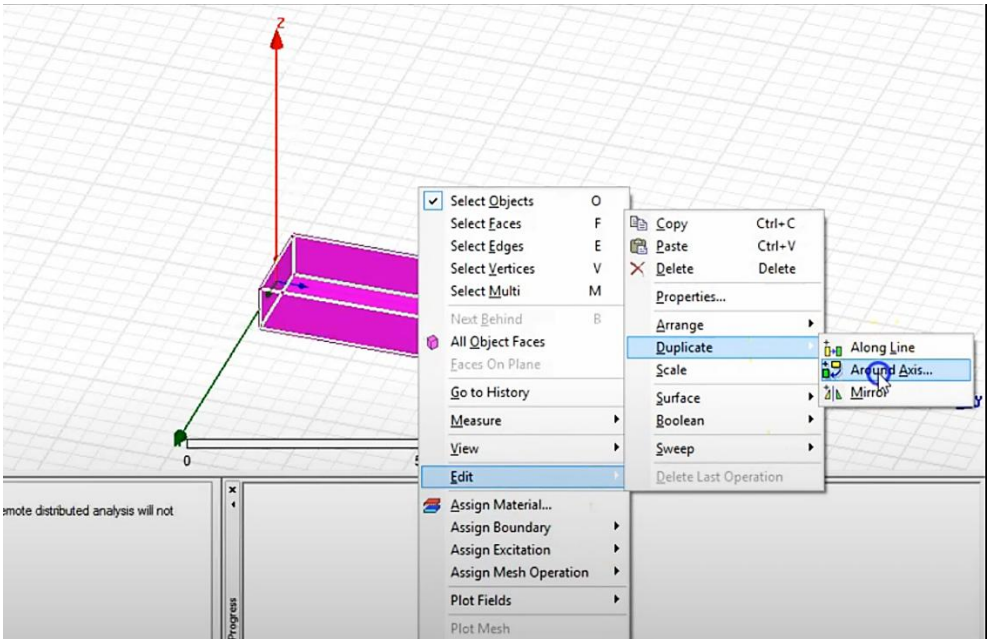
Position→(-11.43, 0, -5.08) (Note: To keep the waveguide positioning at centre of XZ plane axis)

XSize→ 22.86

YSize→100

ZSize→10.16

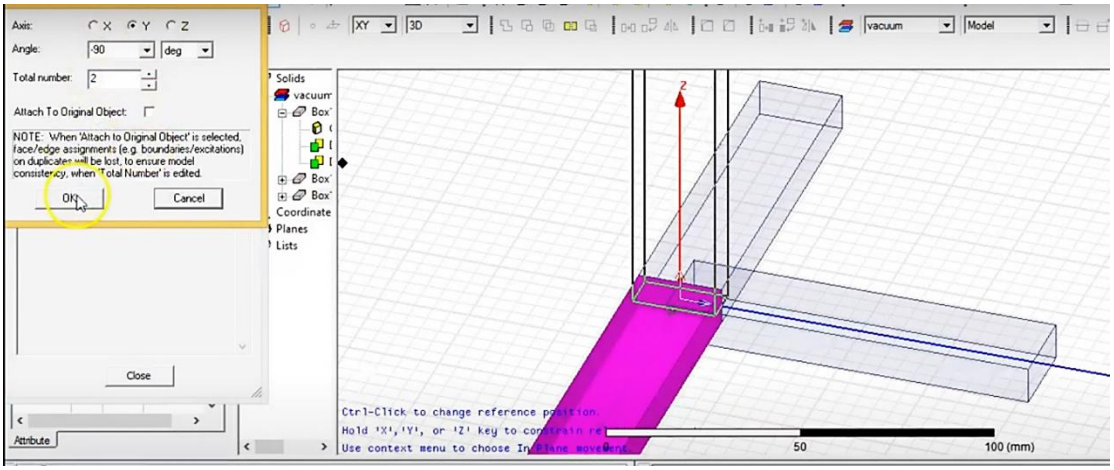
c) Fit View Ctrl +D or View→ Fit all & duplicate the box for all 4 arms
Select Box→ Edit→ Duplicate→ Around Axis (Ref Fig below)



Coplanar-arm (Port 2)
Axis: Z
Angle: 90 deg
Total number: 2

Similarly, Coplanar-arm (Port 1)
Axis: Z
Angle: -90 deg
Total number: 2

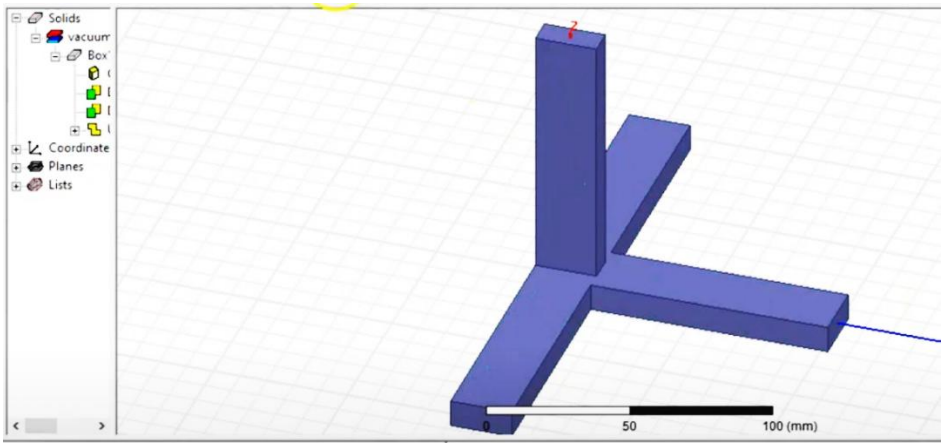
For E-plane arm (Port 3) (Ref Fig below)
Select the coplanar port1 arm,
Axis: Y
Angle: -90 deg
Total number: 2



d) Unite all the Box

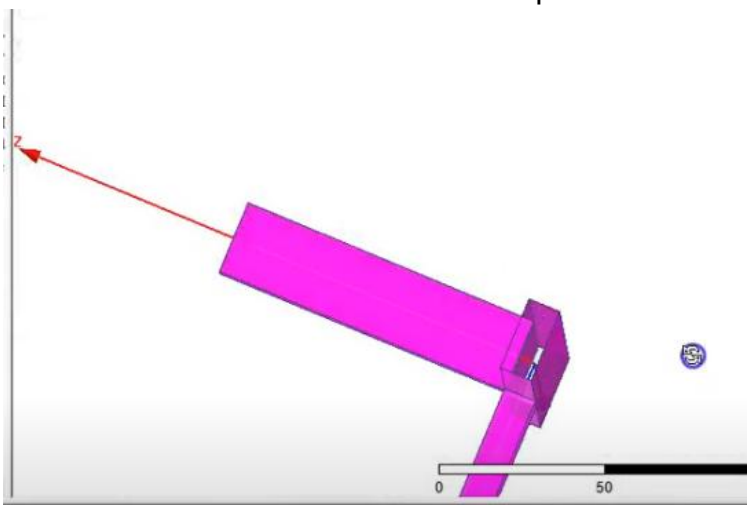
Edit→ Select All Visible or Ctrl + A

Modeler→ Boolean→Unite



e) Assign Boundary and excitation to the port arms

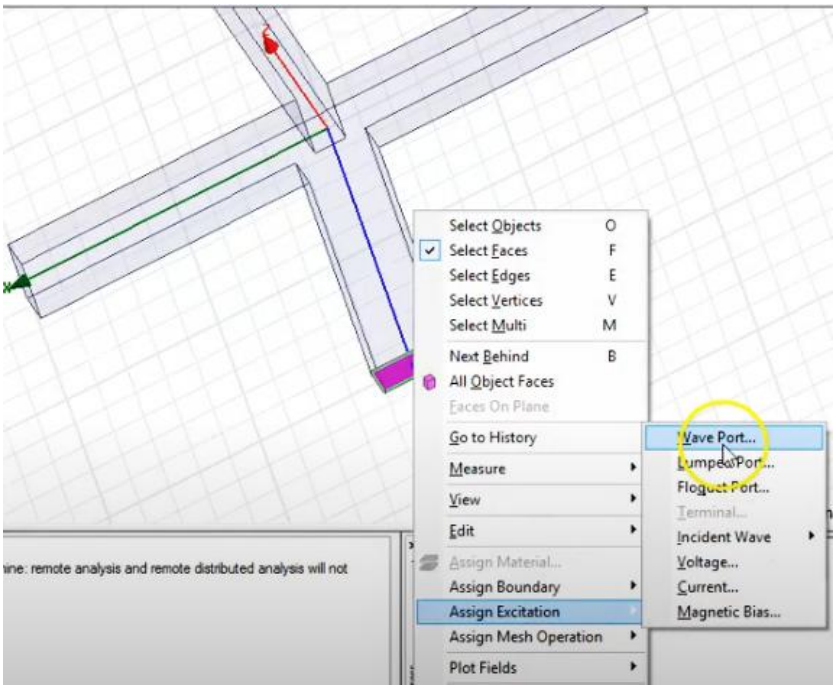
Select Ctrl + Select all the arms except faces



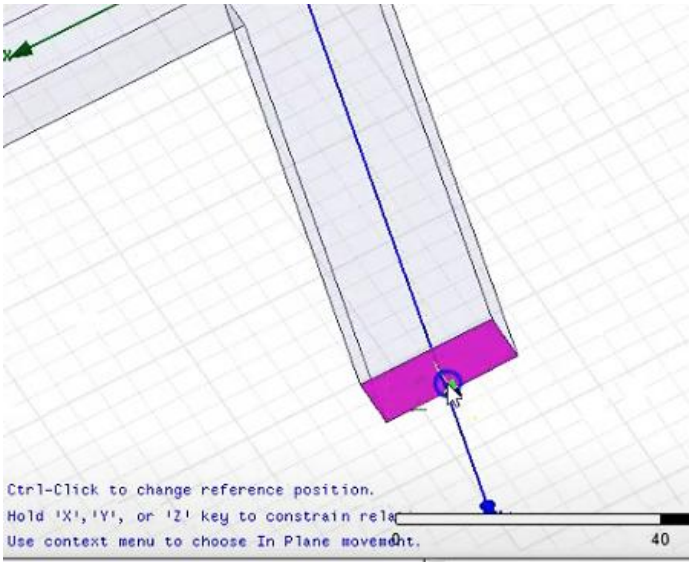
Right click on mouse→ Assign Boundary→ Perfect E→Name→PerfE1

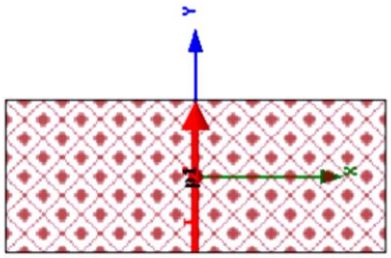
f) Assign excitation to wave ports

Use mouse pointer click on face of port and Assign Excitation→Waveport→ Name→3, Integration line→ New Line→



Select the integration line from bottom to top on the face as shown below





Wave Port Face

Similarly do for all the arms port1, 2 & 4

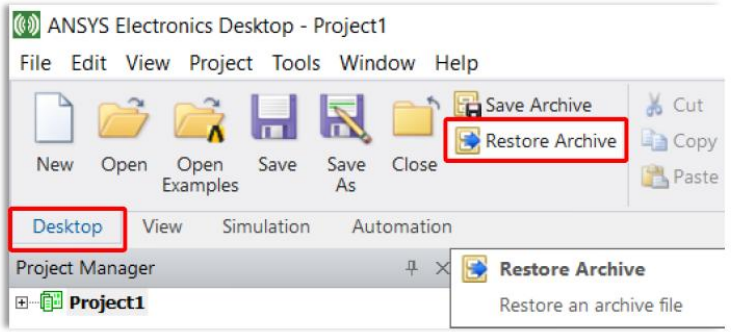
Note: Continue the remaining steps in the lab manual for the results

EXPERIMENT 8

Design and Simulation of Patch Antenna and Optimetrics using HFSS

Restore Archive and Save Optimetrics_Patch1.aedt

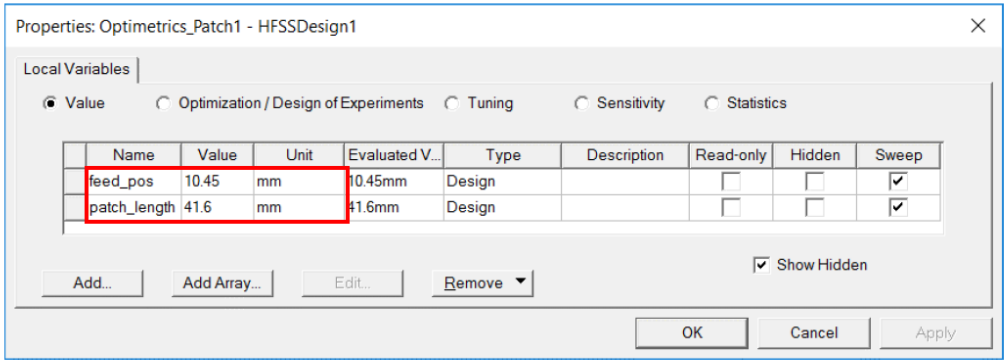
- In the ribbon, in the **Desktop** tab, selected: ***Restore Archive*** to bring up the ***Archive*** *to* ***Restore*** search browser.
- Browse to the training files location and select: **Optimetrics_Patch.aedtz**.
- Click ***Open***
- Save the file as **Optimetrics_Patch1.aedt**.

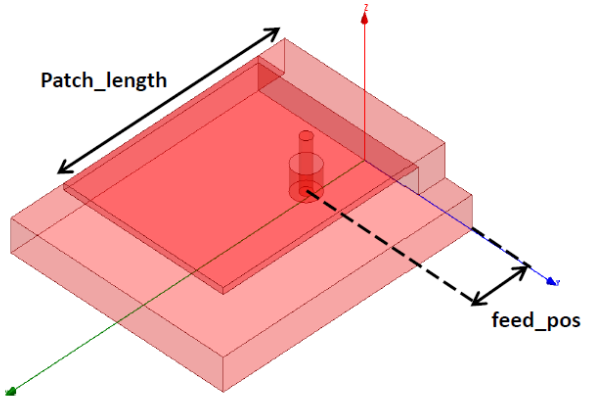


FileName.aedtz indicates an archive, where ***FileName.aedt*** indicates the usual Ansys Electronics Desktop file extension.

Checking *Local Variables* feed_pos and patch_length

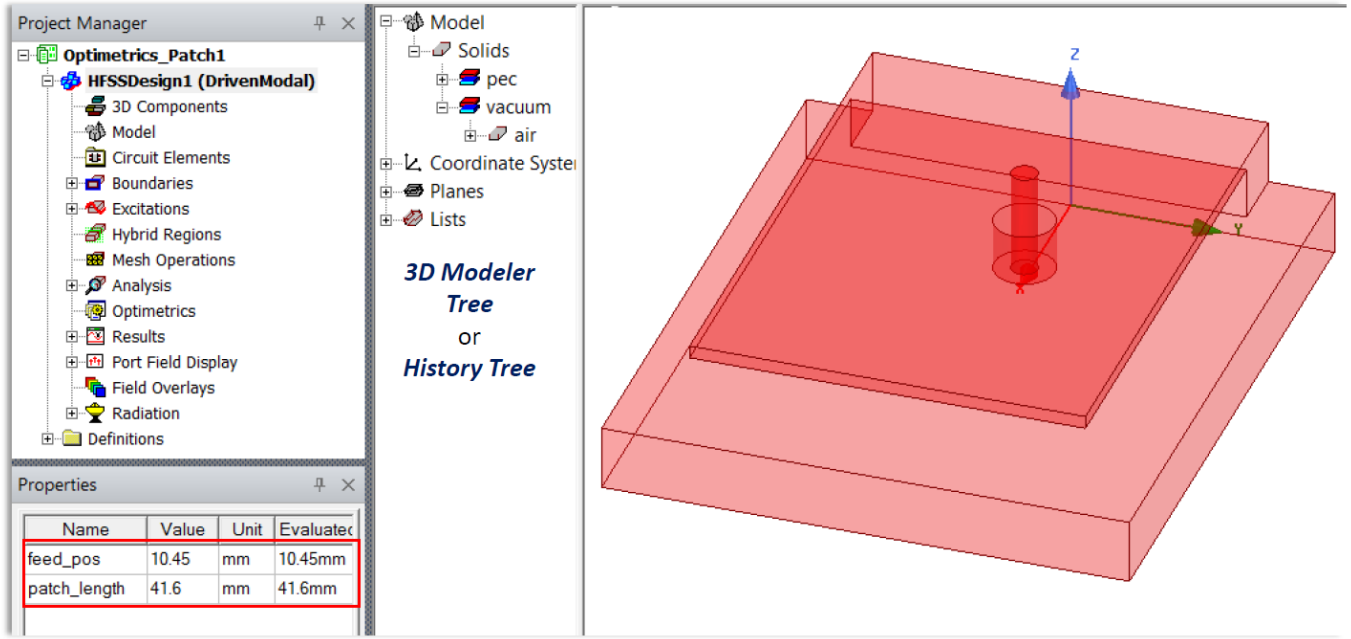
- There are two design variables that have been created. They control the location of the feed of the patch antenna, **feed_pos** in X, and the length of the patch, **patch_length** in X.
- To view a list of any design variables that have been created for this design:
  - Go to the menu item ***HFSS > Design Properties.***
  - Verify that the variable **feed_pos** is assigned the value **10.45 mm.**
  - Verify that the variable **patch_length** is assigned the value **41.6mm.**
  - Press the ***OK*** button.





Design Variables are *Local Variables*

- Alternatively, in the ***Project Manager,*** click on the design name **HFSSDesign1**. The design variables will be displayed in ***Properties***. Think ***local*** to this ***design*** to understand this equivalence in terms.

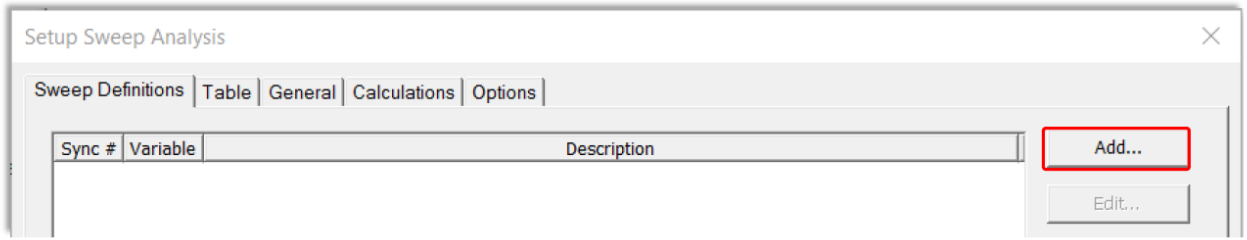


To get this view of the antenna, in the ***3D Modeler Tree,*** right-click on ***air*** and select ***Hide in Active View.***

Start Parametric Analysis Setup

- **Parametric Sweep of Feed Position feed_pos**
  - We will now complete the parametric project simulation setup using the defined variable **feed_pos** to vary the coaxial feed position in order to achieve optimal match between the patch antenna and its coaxial feed line. The ratio of the coaxial feed inner and outer diameters was chosen to achieve a 50 Ohm characteristic impedance. We will change the value of the feed offset location until we find a position which presents a 50 Ohm load impedance to the coaxial feed line.
- **Create Parametric Sweep**
  - In the ***Project Manager***, right-click on ***Optimetrics*** and select ***Add > Parametric...***
    - In in the ***Setup Sweep Analysis*** window, click the ***Add...*** button to open the ***Add/Edit Sweep*** window

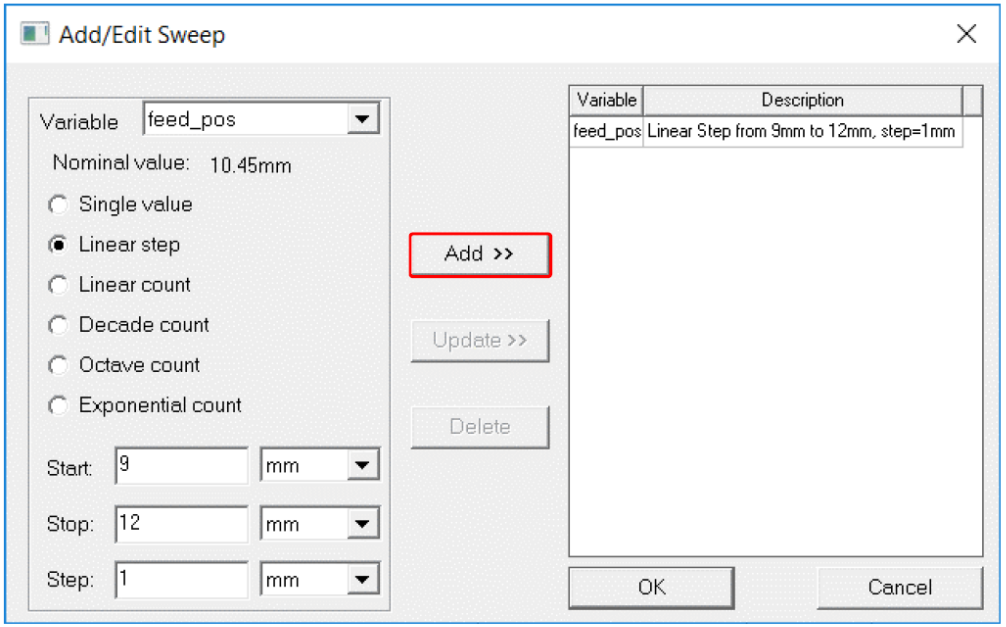
...continued...



Parametric Analysis Setup - *Add/Edit Sweep*

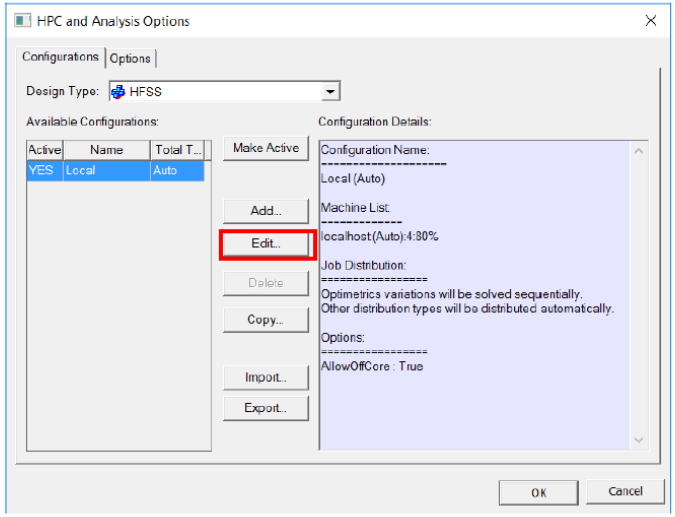
Creating the Parametric Sweep for *feed_pos*

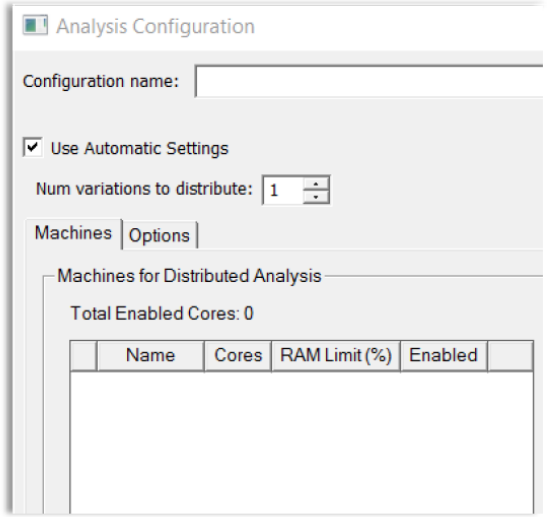
- In the ***Add/Edit Sweep*** window:
  - Select the ***Variable* feed_pos**
  - Select ***Linear Step***
    - ***Start: 9mm***
    - ***Stop: 12mm***
    - ***Step: 1mm***
    - Click the ***Add>>*** button
    - Click the ***OK*** button to close the ***Add/Edit Sweep*** dialog box.
  - Click the ***OK*** button to close the ***Setup Sweep Analysis*** window.



HPC and Analysis Options

- **High Performance Computing Configuration**
  - Parametric sweeps can be accelerated by solving multiple variations of a design in parallel. The HPC ***Analysis Configuration*** allows us to specify the number of cores and the number of tasks we would like to run. The number of tasks will correspond to the number of parametric variations or frequency points to run in parallel.
- **Configuring HPC Settings**
  - Select the menu item ***Tools > Options > HPC and Analysis Options...***
  - In the ***Configurations*** tab, click ***Edit*** button to bring up the ***Analysis Configuration*** dialog box.

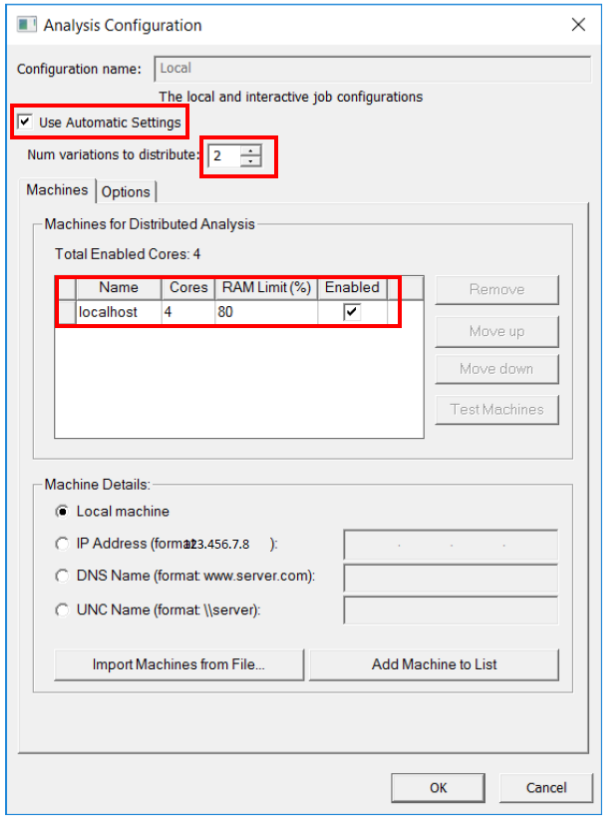




See Also:
High Performance Computing section in ***HFSS Help*** documentation.

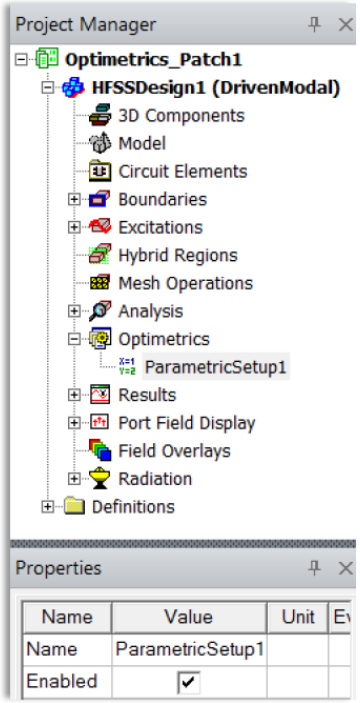
HPC: Analysis Configuration - Analyze Optimetrics_Patch1

- **In the *Analysis Configuration* window**
  - *Use **Automatic Settings***: ☑ ***Checked***
  - ***Num variations to distribute**: 2*
  - change or verify the following in the ***Machines*** tab:
    - ***Name: localhost***
    - ***Cores: 4***
    - ***RAM Limit(%): 80***
    - ***Enabled**: ☑ **Checked***
  - Click the ***OK*** button to finish and close ***Analysis Configuration*** window.
  - Click the ***OK*** button to close ***HPC** and **Analysis Options*** dialog box.

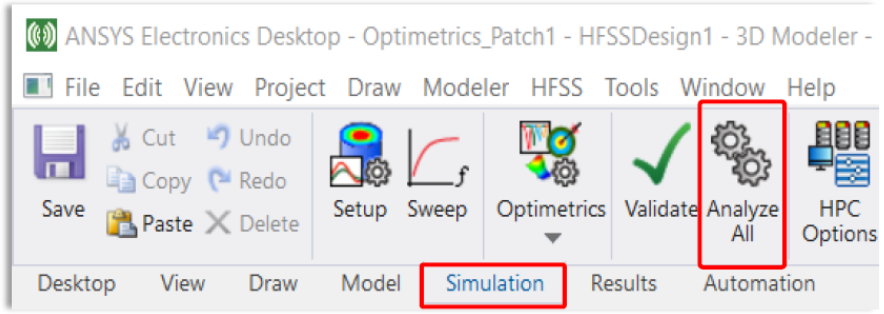


Note: ***Num variations to distribute*** refers to the number of parallel simulations to run for parametric and ***Optimetrics*** variations. With 4 total cores, this means 2 cores run each of those distributed simulations. The number of cores might vary with different computers.

Save, Validate, and Analyze HFSS Design Optimetrics_Patch1

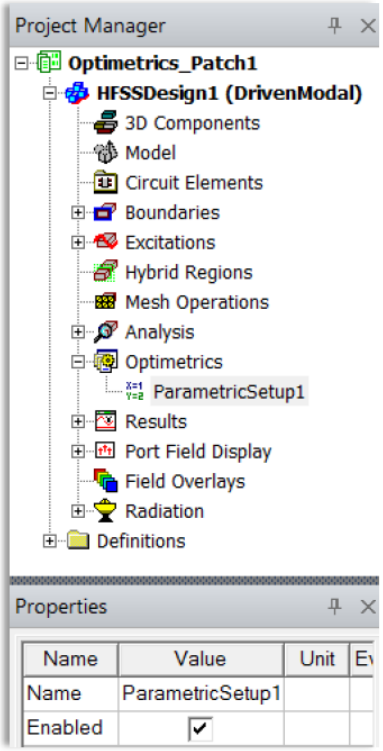


Note the* ParametricSetup1 *under Optimetrics.

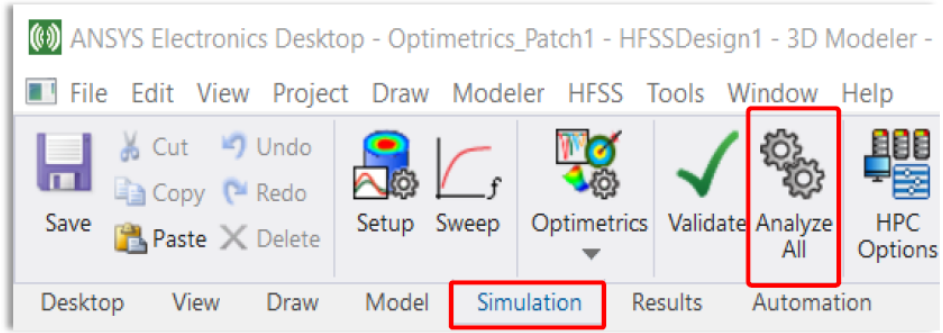


- In the ***Ribbon***, with the Simulation tab chosen, click the ***Validate*** greer check mark to validate the project.
- Select ***File > Save***
- In the ***Project Manager***, under ***Optimetrics,*** right click on ***ParametricSetup1*** and select ***Analyze***.
- Select ***File > Save*** when the simulation finishes.

Save, Validate, and Analyze HFSS Design Optimetrics_Patch1



Note the* ParametricSetup1 *under Optimetrics.



- In the ***Ribbon***, with the Simulation tab chosen, click the ***Validate*** greer check mark to validate the project.
- Select ***File > Save***
- In the ***Project Manager***, under ***Optimetrics***, right click on ***ParametricSetup1*** and select ***Analyze***.
- Select ***File > Save*** when the simulation finishes.

Note: Continue the remaining steps in the lab manual for the results