Ansys CERTIFIED ELITE CHANNEL PARTNER

# HFSS Foundation Course

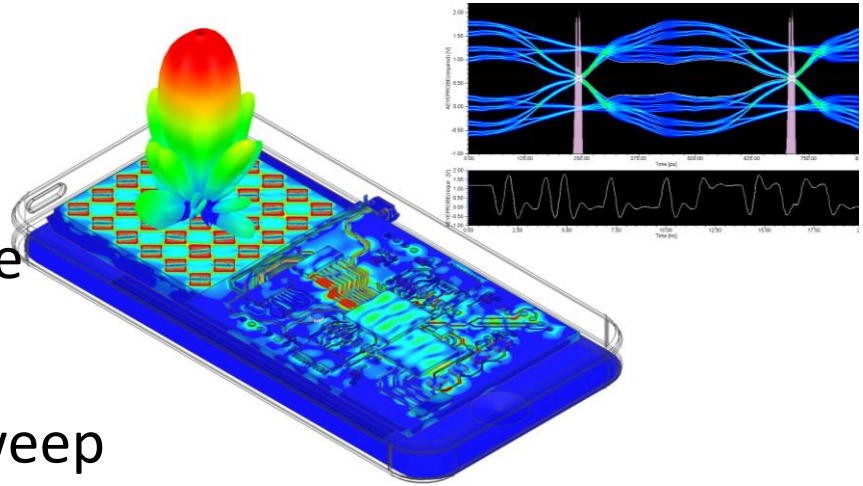
Nicholas Chang, Ph.D
High Frequency Application Engineer
**CYBERNET SYSTEMS TAIWAN Co. Ltd**

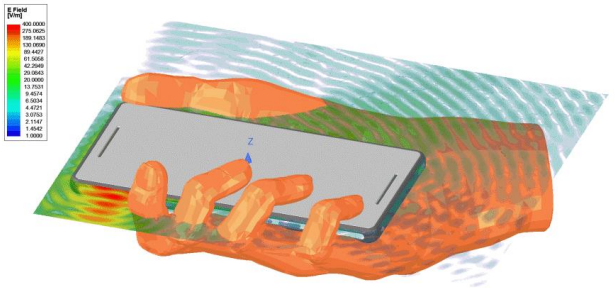
Ansys 2021 R1
Engineering What's Ahead.

Solve it, with CYBERNET

# Outline

- Introduction
- Geometry Construction
- Boundaries and Simulation Space
- HFSS Lumped and Wave Port
- FEM Solution Setup - Mesh & Sweep
- Post-Processing - S-Parameters and Fields
- Project Production





# Cybernet Systems Worldwide


## 關於 CYBERNET 集團

日本 CYBERNET SYSTEMS 公司自1985年成立至今，為日本一級的成功上市股票公司，旗下有數百位專業的碩、博士專業工程師，提供各種工程分析、顧問服務以及開發相關軟硬體介面、模式以及分析模塊開發與 IT 相關軟體工具。 CYBERNET SYSTEMS 擁有超過30年頂尖工程專業經驗，各行業之領導廠商皆是 CYBERNET 之忠誠客戶。

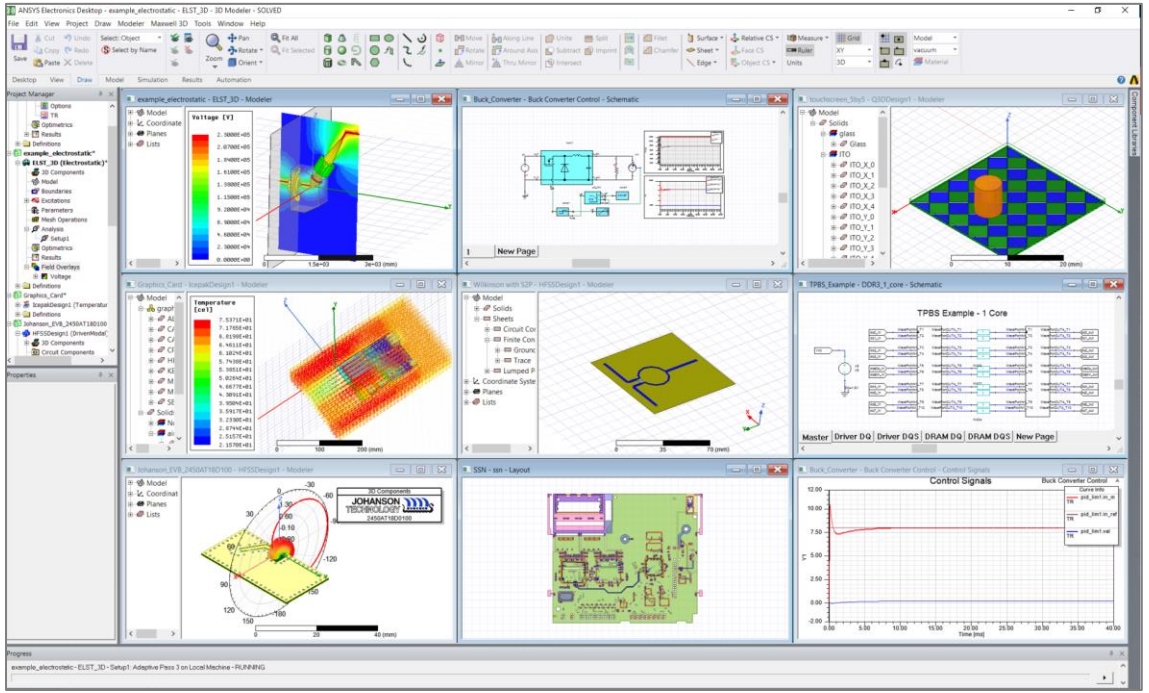
CYBERNET 集團於2006年始積極拓展版圖，積極於中國、台灣等處成立服務據點，並陸續併購公差分析軟體開發大廠 Sigmetrix、符號運算引擎開發大廠 Maplesoft，以及程序整合暨設計最佳化開發大廠 Noesis Solutions，期能提供更完整、更即時之工程服務予全球客戶。

# 關於思渤科技

CYBERNET SYSTEMS 集團於台灣經營據點思渤科技股份有限公司，秉以永續經營為理念，代理各國先進之研究開發工程工具。從設計到製造面，提供**軟硬體銷售**、**技術諮詢**、**工程專案**以及**教育訓練課程**等服務，解決您從設計到製造面的跨領域困難課題。

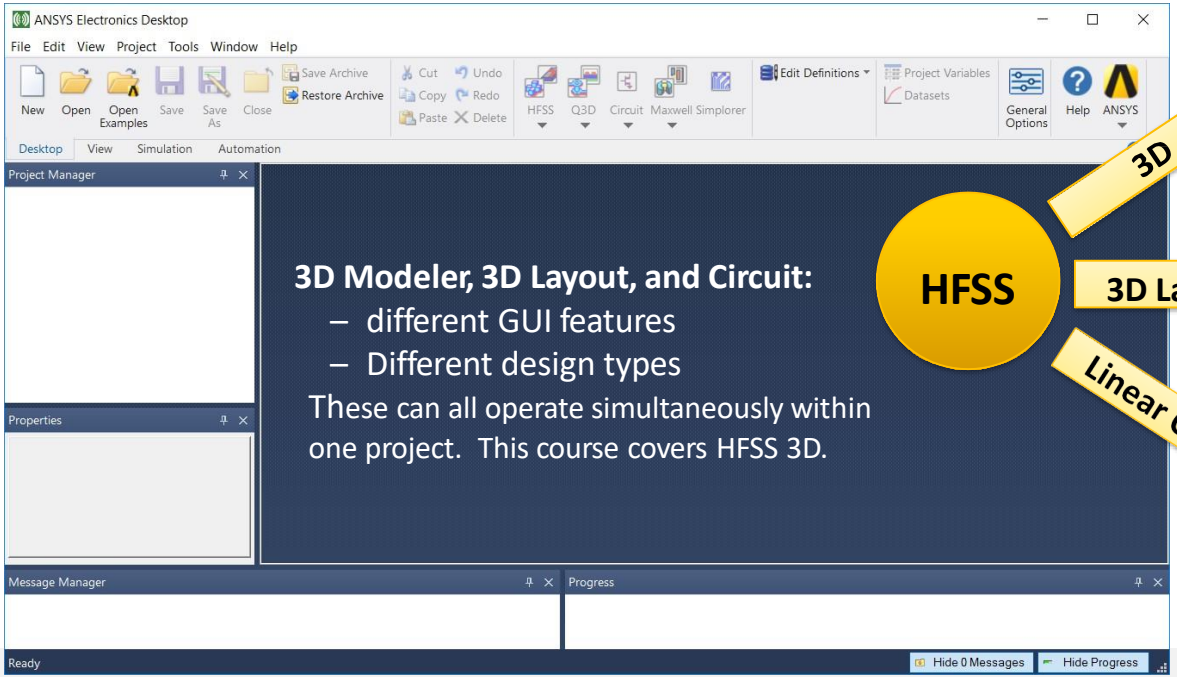
# Ansys Electronics Desktop (AEDT)

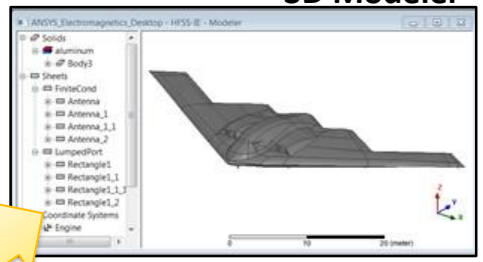
- Ansys Electronics Desktop – Electromagnetics Suite
- Single Unified Desktop for Electronics Tools
  - HFSS Design Types
  - Q3D Extractor Design Types
  - Circuit Design Types
  - ICEpak Designs
  - Maxwell Design Types
  - Simplorer
  - SBR+

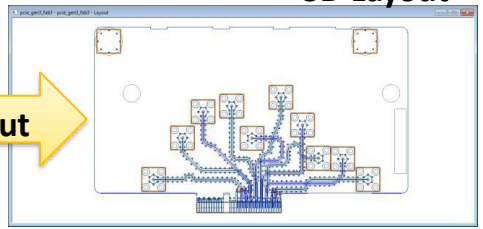


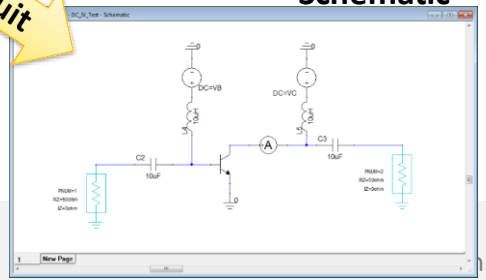
# HFSS in ANSYS Electronics Desktop (AEDT)

3 Basic Interfaces - 1 Desktop



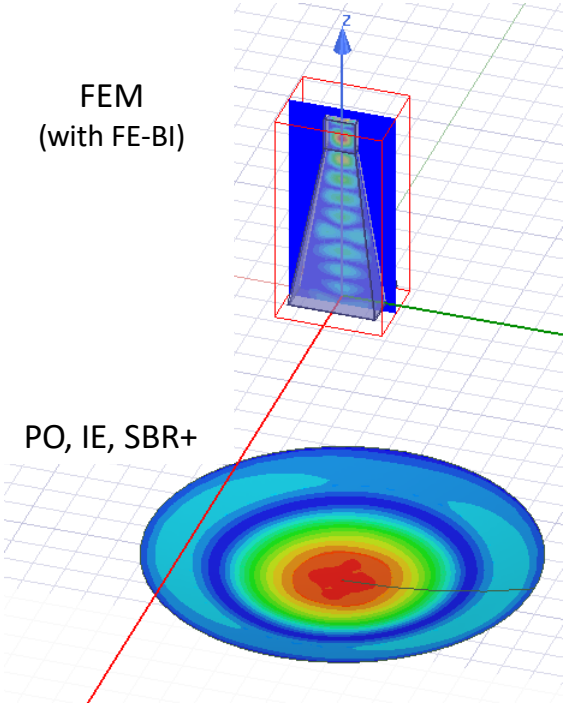



# HFSS Includes Multiple EM Solvers

- **HFSS FEM (Finite Element Method)**
  - Fully arbitrary 3D - the whole simulation space gets meshed
  - Used for microwave, antenna, and PCB signal integrity applications
  - HFSS is also a “design type” within the HFSS product.
- **HFSS IE (Integral Equation) Solver**
  - 3D surface meshing – but only meshes surfaces
  - Commonly used for antenna applications
  - Available within the HFSS design type
- **HFSS PO (Physical Optics) and SBR+ (Shooting Bouncing Ray) Solvers**
  - Approaches wave propagation in terms of rays
  - Commonly used for antenna applications
  - Available within the HFSS design type
- **HFSS Transient Solver**
  - Time domain formulation that can employ pulsed excitations
  - Commonly used for applications such as EMI (electromagnetic interference)
- **HFSS Eigenmode Solver**
  - Used to obtain fields in cavities and periodic structures along with the associated dispersion curves
  - No excitation needed - not a driven solution

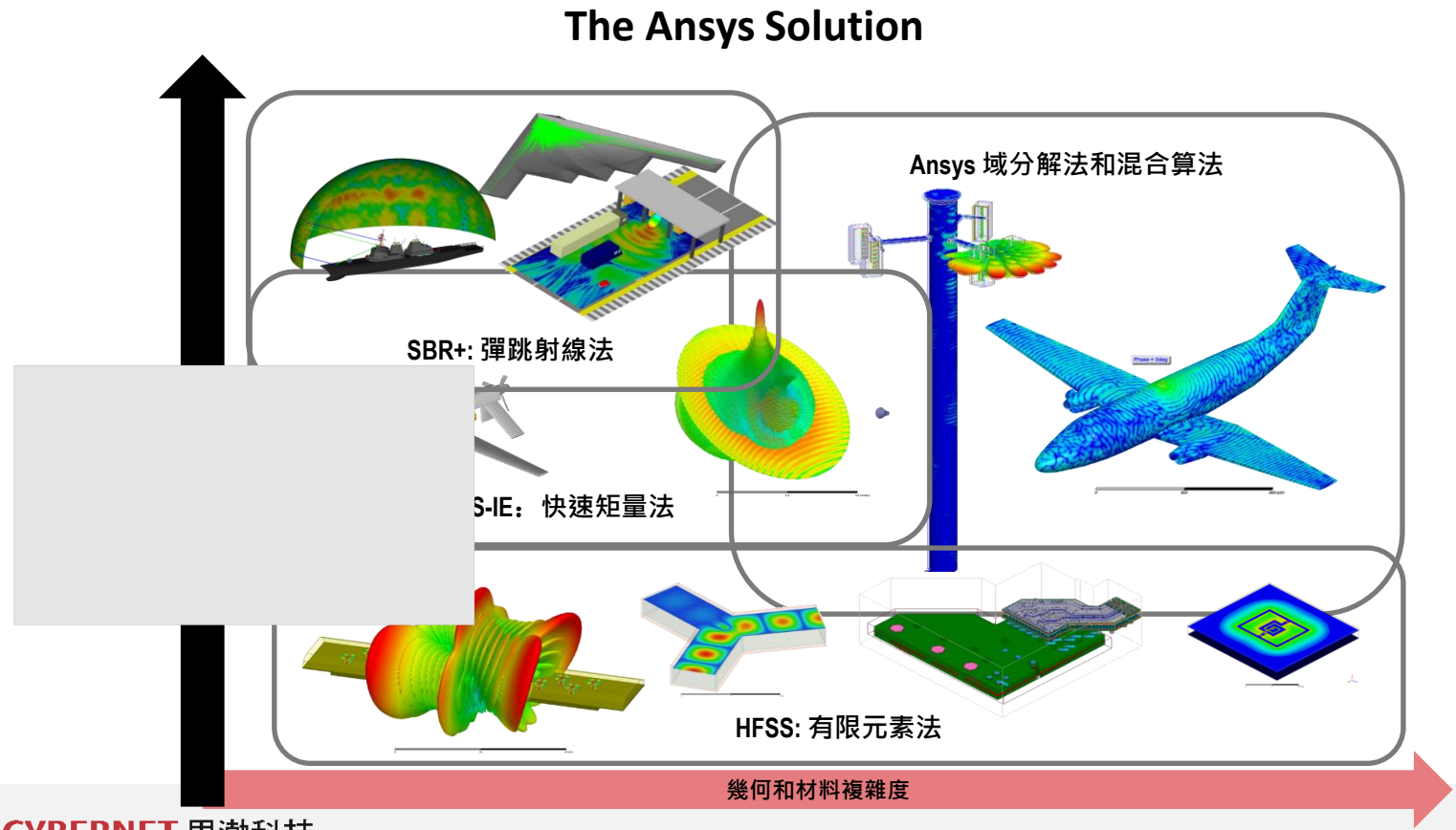
FEM
(with FE-BI)




PO, IE, SBR+

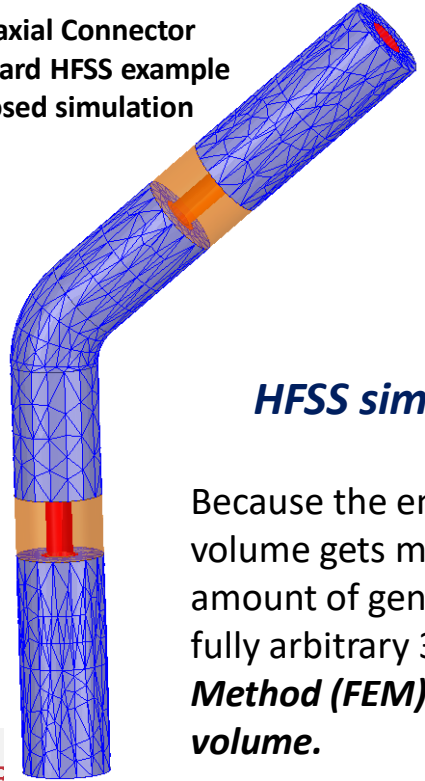
***HFSS FEM*** is the subject of this course.

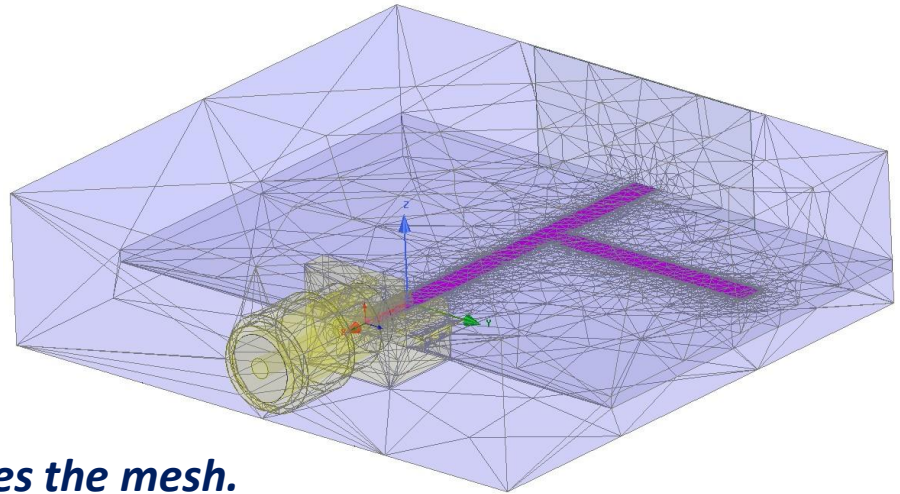
# HFSS 求解數值法


The Ansys Solution
Ansys 域分解法和混合算法
SBR+: 彈跳射線法
S-IE：快速矩量法
HFSS: 有限元素法
幾何和材料複雜度

# HFSS Fully Arbitrary 3D FEM Meshes

**Coaxial Connector**
**Standard HFSS example**
**Closed simulation**





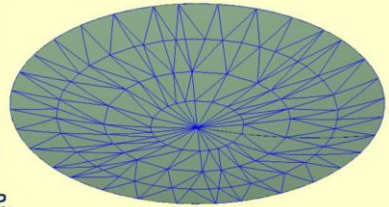
**SMA coaxial to microstrip transition**
**Open simulation**

***HFSS simulates the mesh.***

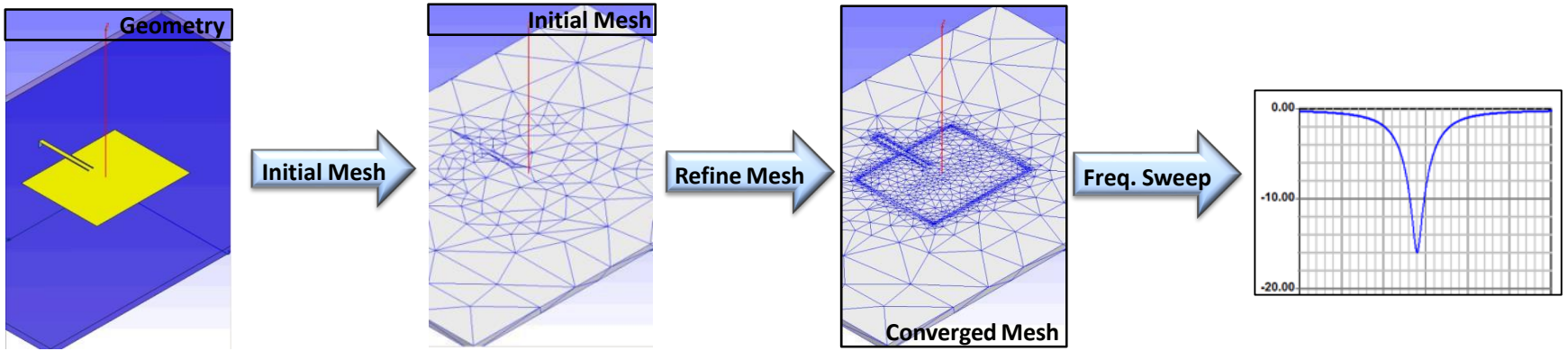
Because the entire computational volume gets meshed, the greatest amount of generality is available – fully arbitrary 3D. ***Finite Element Method (FEM) solves for fields in a volume.***

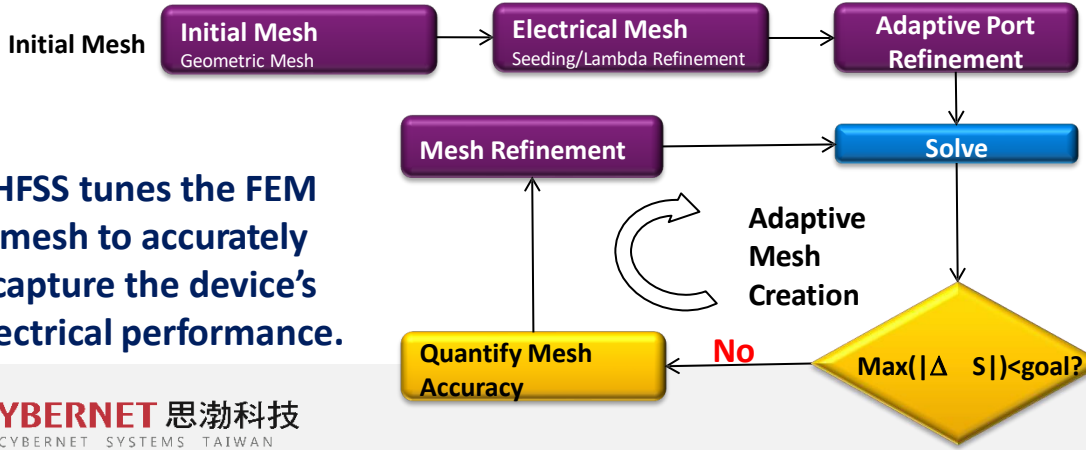
***What else is there…besides fully arbitrary 3D?***

*For comparison, the HFSS IE solver meshes surfaces and simulates fields at a distance away from that structure. IE reflector antenna shown here.*



# HFSS FEM Automated Meshing Process

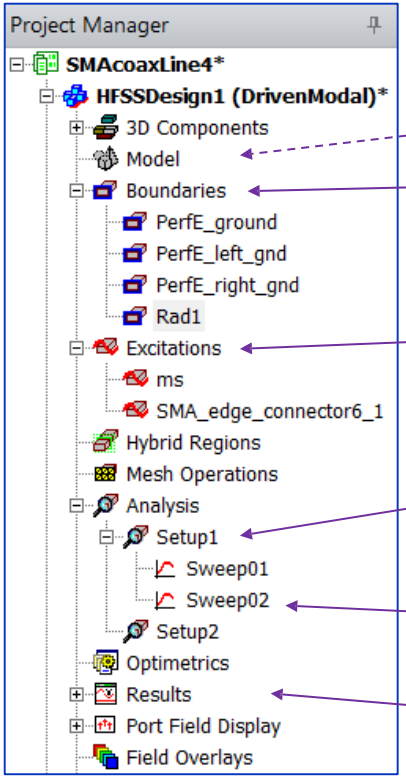



HFSS tunes the FEM mesh to accurately capture the device's electrical performance.

The HFSS setting used in the decision diamond is called *Delta S.*

# The HFSS Project Manager




Geometric Structure
Materials

Primarily addressed
in the Model Tree

Boundaries
Simulation Space

Excitations/Ports

Simulation Setup
Adaptive Process

Simulation Setup -
Frequency Sweep

Simulation Results

***HFSS FEM*** is the subject of this course. While HFSS represents and includes a collection of solvers, we're focusing on the finite element method solver only.

The document "***An Introduction to HFSS***", Chapter 5 ***"HFSS Modeling GUI Basics"*** section ***"Modeling Practice in HFSS"*** with an HFSS workflow.

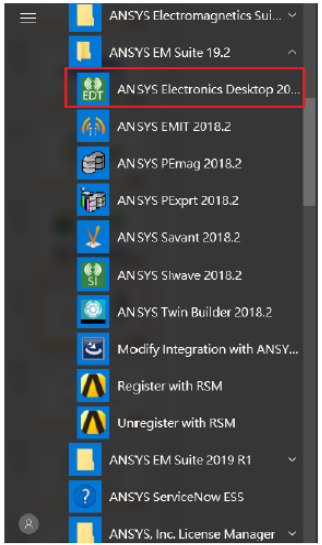
# Geometry Construction

- HFSS Antenna Toolkit, Creating Shapes in HFSS , Geometry Construction

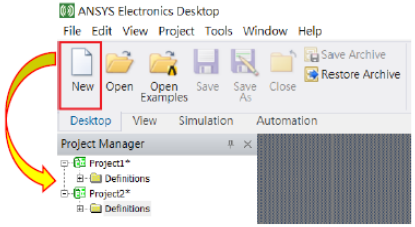
# How to Open HFSS?

## First of All

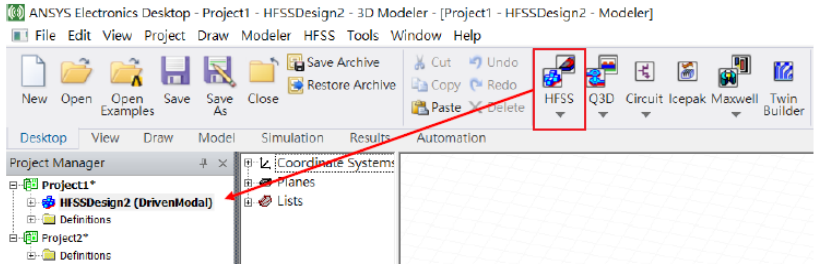
- Open HFSS


- New Project




- New Design
  - ✓ What's the difference between "project" and "design"?
  - ✓ How to select the active design?

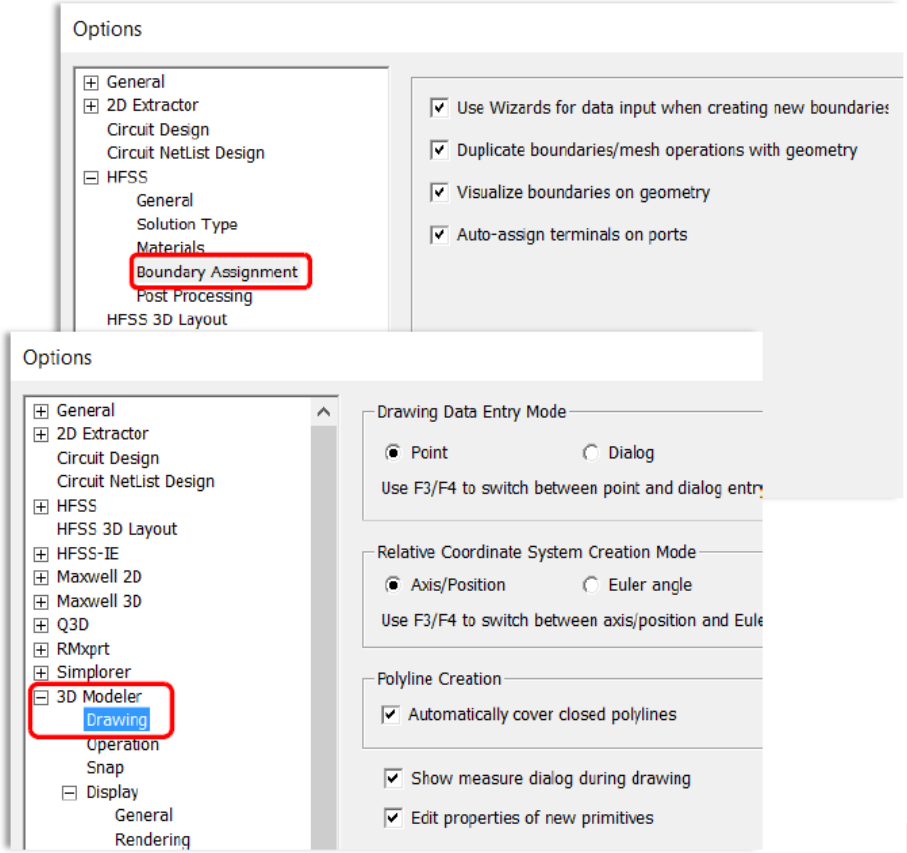


# Setting of Option

- Selected: ***Tools > Options > General Options***

- Expand ***HFSS*** (by clicking on the + sign) and Select ***Boundary Assignment***
  - ***Check all entries***

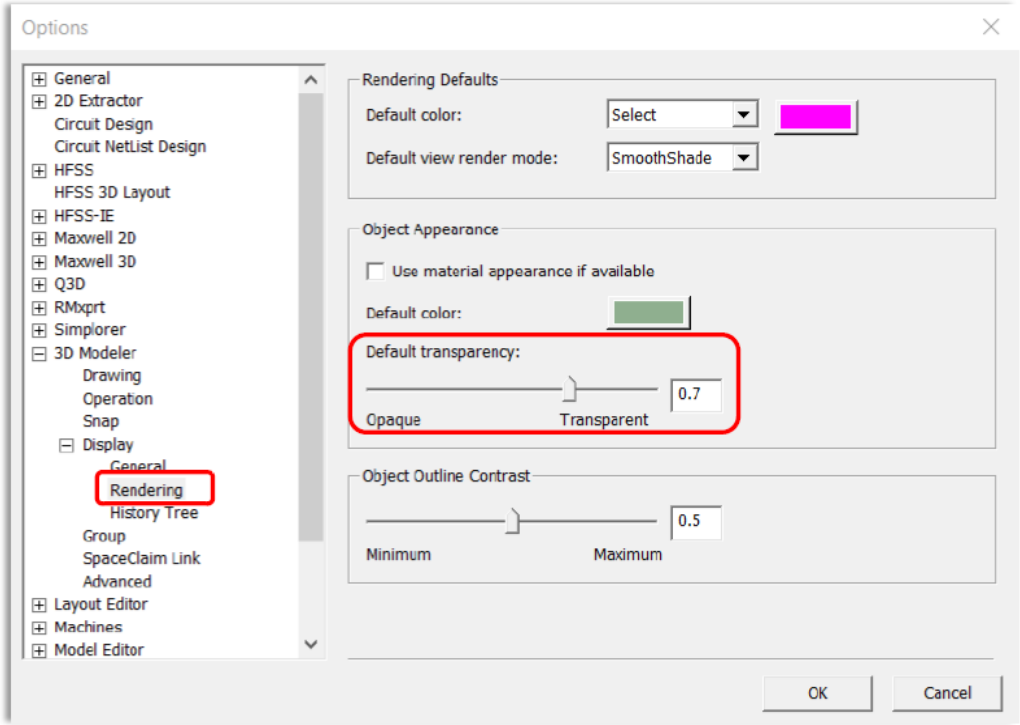
- Expand ***3D Modeler*** and click ***Drawing***
  - **Automatically cover closed polylines: ☑ Checked**
  - **Edit properties of new primitives: ☑ Checked**



# Display History and Transparency

- Expand ***Display***
  - Click ***Rendering*** and set ***Default Transparency*** to ***0.7***
  - Click ***History Tree*** and ***check all entries*** (not shown here)

- Click the ***OK*** button to close the Options dialog box



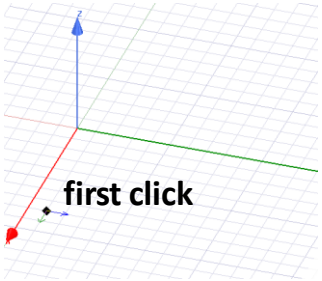
# HFSS Geometry Construction

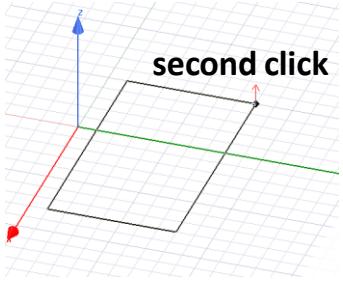
- **Section 1: Creating Shapes in HFSS with *Draw***
  - Box — Draw > Box
  - Line — Draw > Line
  - Cylinder — Draw > Cylinder
  - Coordinate Entry Bar — (lower right)
- **Section 2: HFSS 3D Parameterization**
  - Parameters on the Fly
  - Equations in Parameters
  - Centering Geometry with Parameters
  - Intrinsic Parameterization
- **Section 3: Viewing Features**
  - Visibility
  - Transparency
- **Section 4: Shape Modifications with *Modeler***
  - Thicken — Modeler > Surface > Thicken Sheet
  - Split — Modeler > Boolean > Split
  - Sweep along a path
  - Visibility and transparency settings
  - Section and Split lanes
- **Section 5: Geometry Construction with *Edit***
  - Duplicate Along Line
  - Duplicate Around Axis

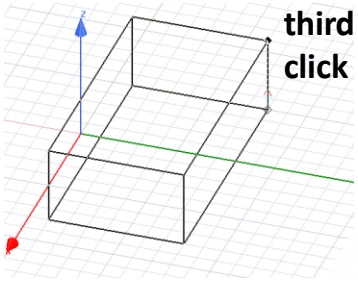
Additional detailed information on geometry construction in the HFSS modeler is available in the Help document ***HFSS.pdf*** which can be found in the HFSS installation directories in the ***Help/HFSS*** directory.

# Objects in HFSS Modeler

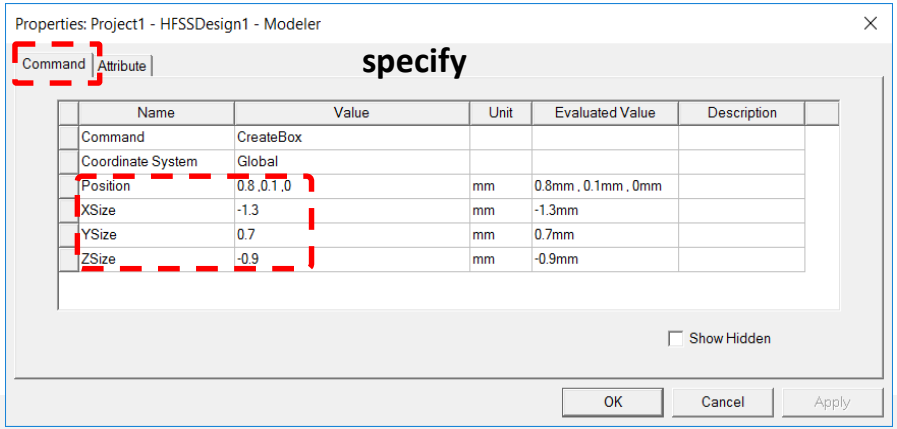
The easiest and most common way to draw a 3D object in HFSS is to click in the ***Model*** window three times to put a randomly sized shape in place. Then specify the exact coordinates in the dialog box that comes up after the third click.

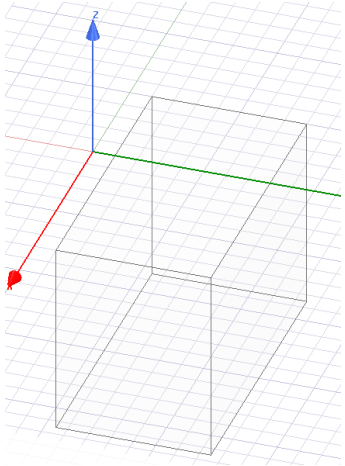



Upon the third click, which defines the 3D object, a dialog box appears. In the ***Command*** tab, specify the exact coordinates desired. *(These here were left unchanged from the draw step.)*

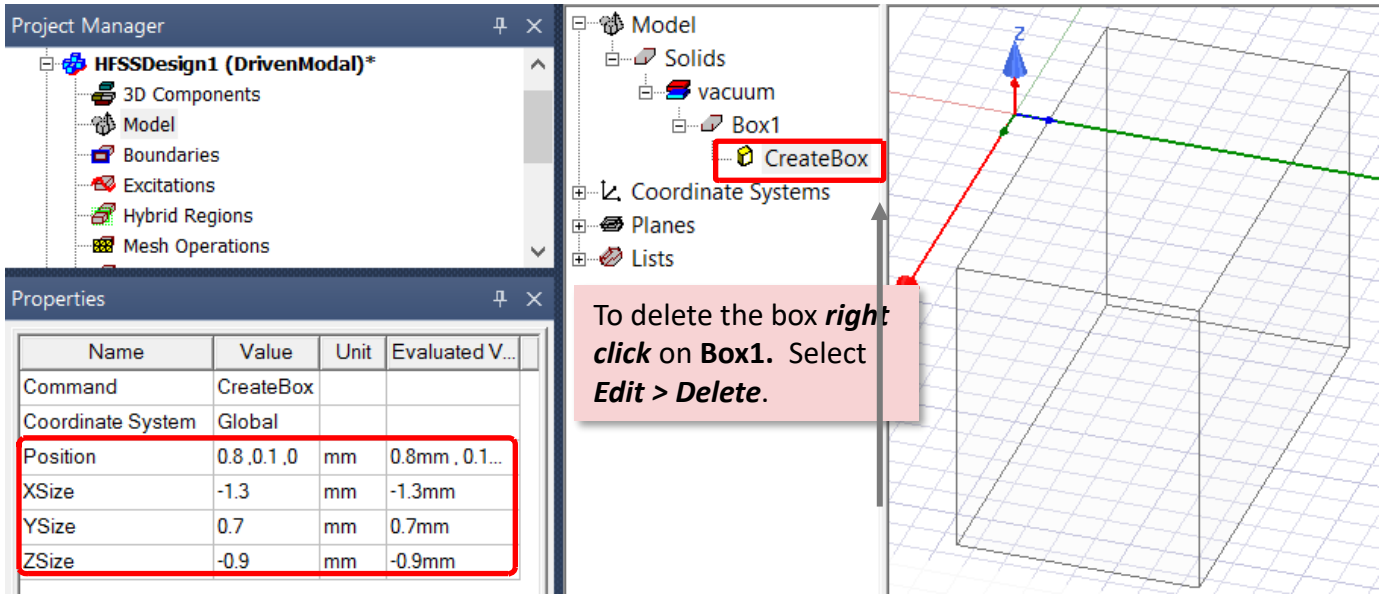


**final box 3D object in HFSS *Modeler***

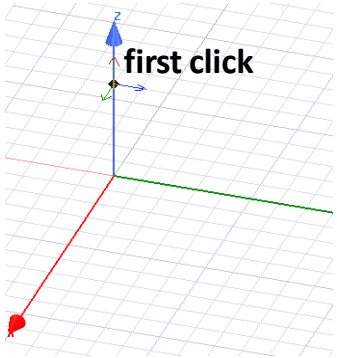
# Box in 3D Modeler and Properties

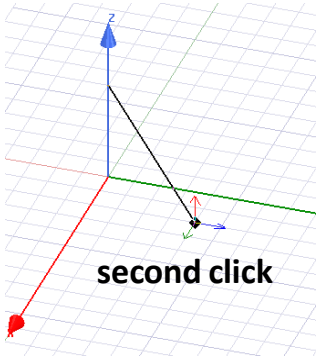
The box dimensions appear in the ***Properties*** window (lower left) when the object (Box1 here) is selected in the 3D modeler.



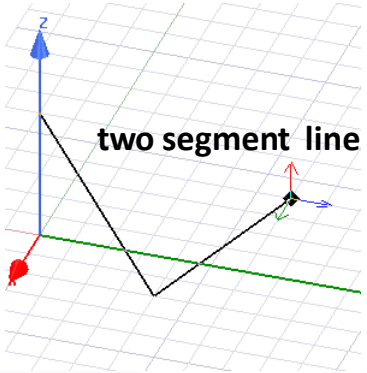
# Lines in HFSS Modeler

A line can be drawn in HFSS by clicking two places anywhere in the ***Model*** window to create a randomly placed line. Then specify the exact coordinates in the ***Command*** tab of the ***Properties*** dialog box. Click twice on the last point.

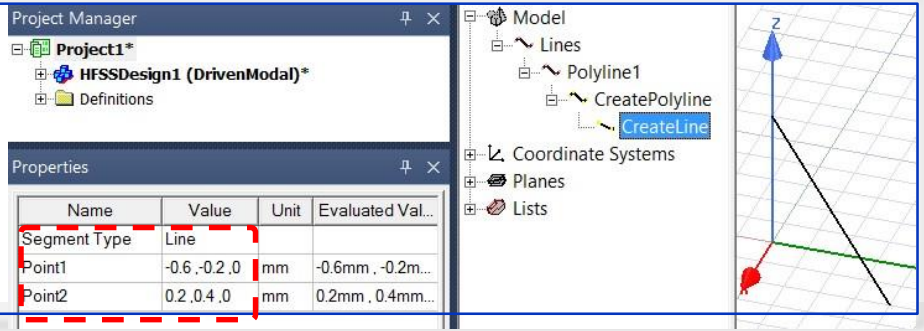



Slightly different from 3D objects, lines in HFSS can have multiple segments. A double-click is required to end the line drawing mode and bring up the ***Properties*** dialog box.

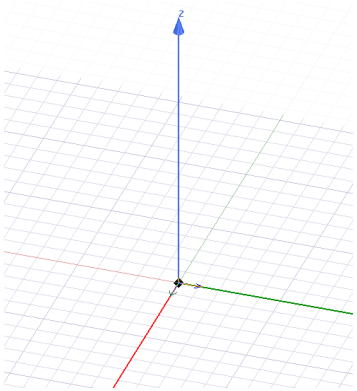



The coordinates for HFSS 3D modeler objects can also be accessed and changed in the ***Properties***. Notice that ***CreateLine*** is highlighted in the ***Modeler*** tree in order to show coordinates in ***Properties***.

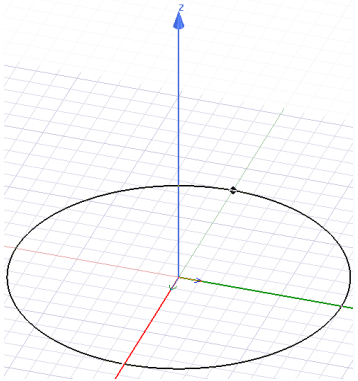


# Drawing Cylinders –Center, Radius, Height

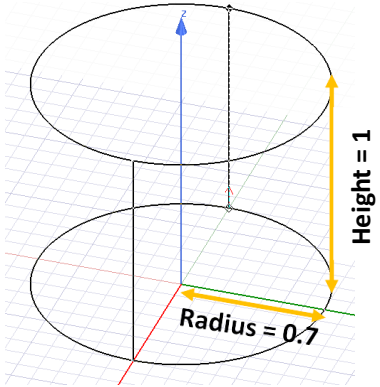
Cylinders can be drawn in HFSS with clicks. The first click is the center, the second is the radius, and the third is the height.



**First click is the center.**



**Second click is the radius.**



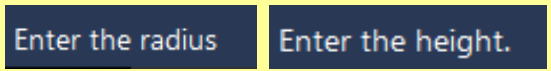

**Third click is the height.**

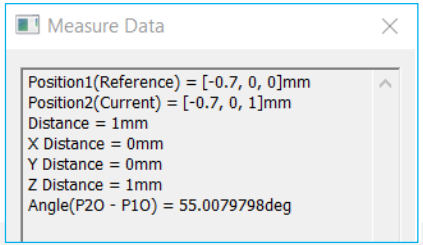
Command | Attribute

| | Name | Value | Unit |
|---|---|---|---|
| | Command | CreateCylinder | |
| | Coordinate System | Global | |
| | Center Position | 0,0,0 | mm |
| | Axis | Z | |
| | Radius | 0.7 | mm |
| | Height | 1 | mm |
| | Number of Segments | 0 | |

This cylinder is centered at the origin with a radius of 0.7 and a height of 1. This shows the attributes in the Properties dialog box.

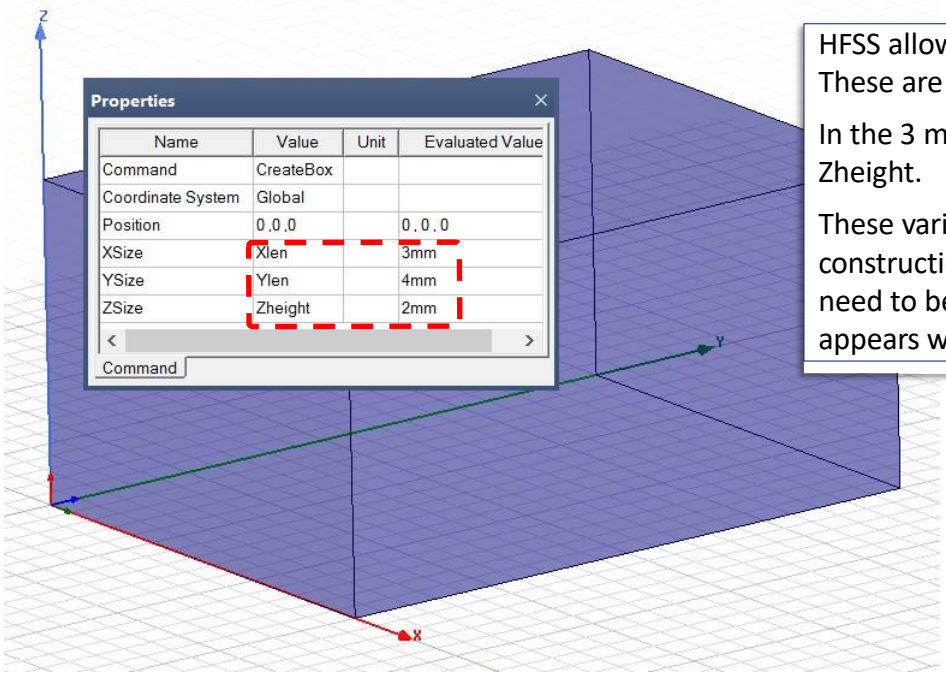
Look for Tool Tips in the lower left corner during geometry construction.




As an alternative to the “click, click, click, specify” approach, this measure function can be used to pick exact locations as you draw. This graphic on the left shows the placement of the height.

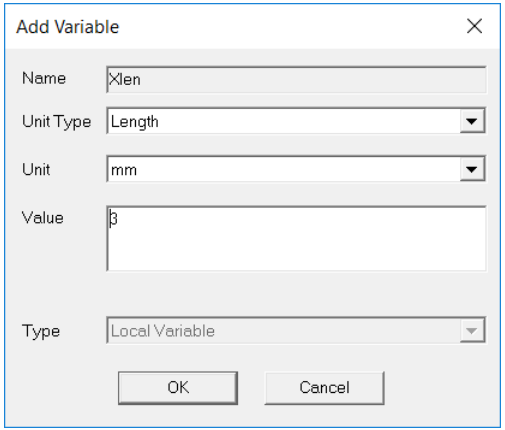
# Parameterized Geometry




HFSS allows the parameterization of geometry using local variables. These are also called ***design variables*** in documentation.

In the 3 mm by 4 mm by 2mm box, the variables are Xlen, Ylen, and Zheight.

These variables can be created and defined during the geometry construction process ("on the fly"). Geometry variables do not need to be defined ahead of their use. This dialog box below appears when a new variable name is introduced.

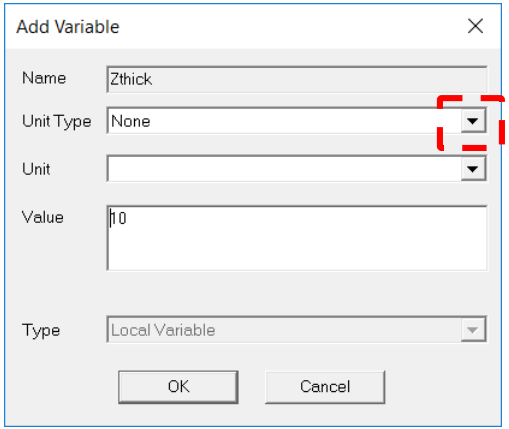


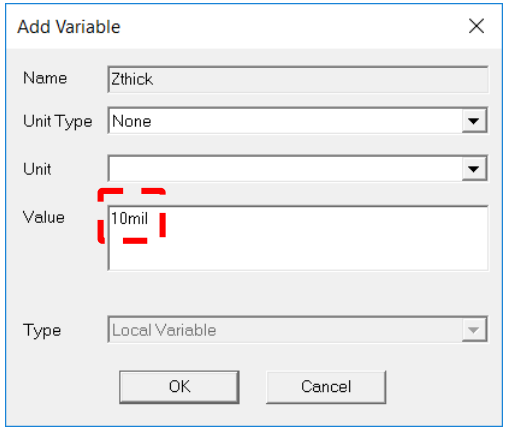

**Note1**: The origin of a Box is a corner.

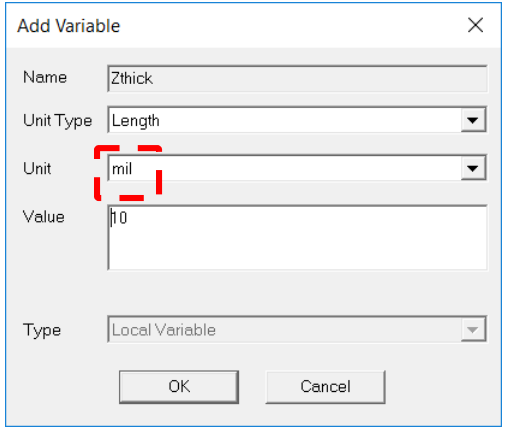
**Note2**: The coordinate bar entry (in the lower right) does not accept variables.

# Parameter Units in HFSS

In HFSS, you can type the unit that you want right next to the value.  HFSS dialog boxes pick this up and populate the “Unit” accordingly.  This method can be used in many HFSS dialog boxes.



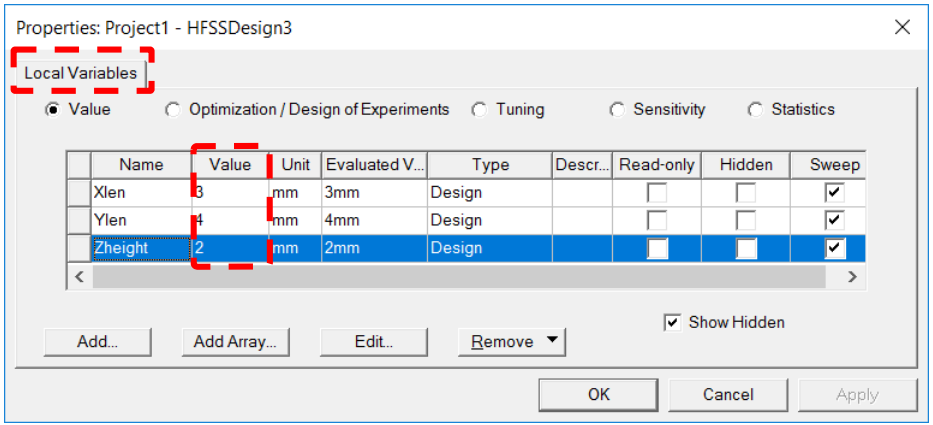




It is important and useful to diligently make sure correct units always get defined for new variables.

# Changing Local (Design) Variable

Local variables, also called design variables, can be changed under HFSS **Design Properties**, **Local Variables** tab, in the **Values** column.

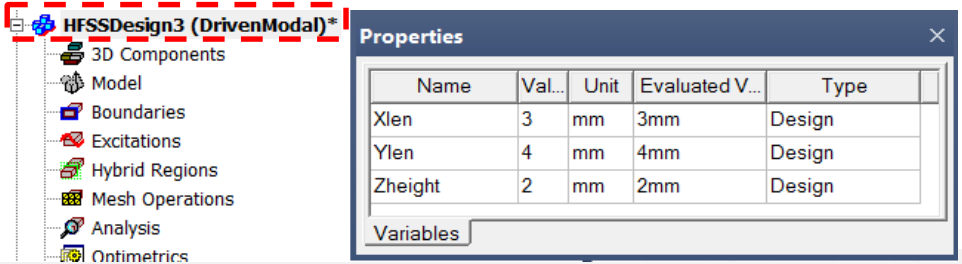



These Local Variables can be accessed two different ways:

1. ***HFSS > Design Properties***
2. Double clicking on the variable name in the ***Properties*** box that appears when the design is highlighted in the ***Project Manager***.

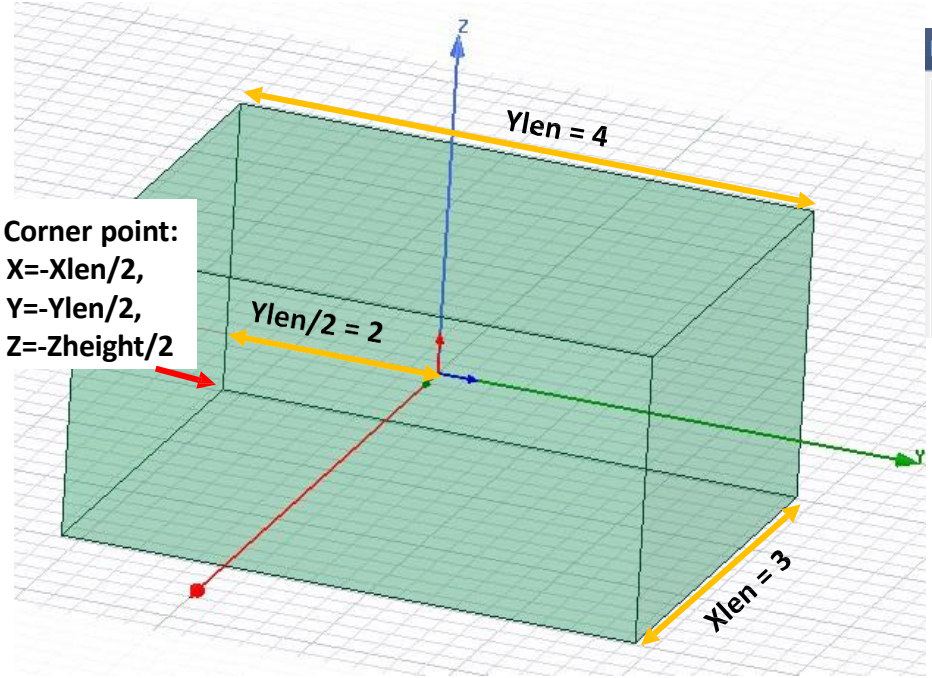
**Note1**: There are also Project Variables available, different from these local variables defined at the design level. See HFSS Help for further information.

This ***Properties*** dialog box is for Design Properties, and it appears when the HFSS design name is highlighted (selected). These variables are defined at the design level in HFSS and can be changed directly in this dialog box.




**Note2**: This is a different Properties box than the one related to the box command and shown on a previous slide.

# Centering a Box Around the Origin




Properties

| Name | Value | Unit | Evaluated Value |
|---|---|---|---|
| Command | CreateBox | | |
| Coordinate System | Global | | |
| Position | -Xlen/2 ,-Ylen/2 ,-Zheight/2 | | -1.5mm , -2mm , -1mm |
| XSize | Xlen | | 3mm |
| YSize | Ylen | | 4mm |
| ZSize | Zheight | | 2mm |

In order to center a box around the origin, use half the desired dimensions of the box for the center point (in the negative direction in this case).

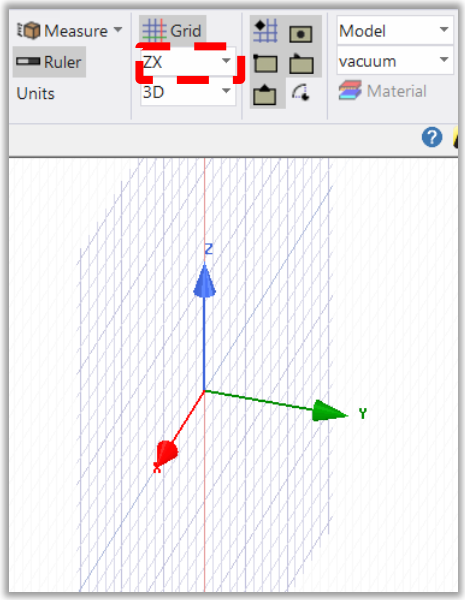
Use the full length for each dimension size.

**Note 1**: Equations can be used in a parameter specification.

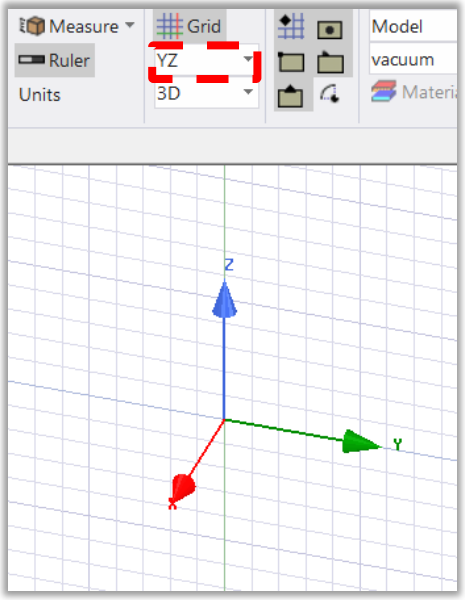
**Note 2**: There is additional information in the HFSS Help chapter “Drawing Objects”.

# Drawing Planes

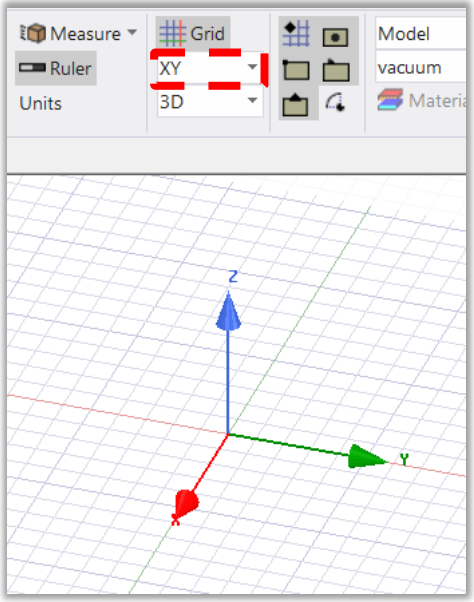
In the top ribbon, in the Draw tab, are drawing plane settings on the far right. These determine which 2D plane is used first in freehand drawing.




ZX




YZ

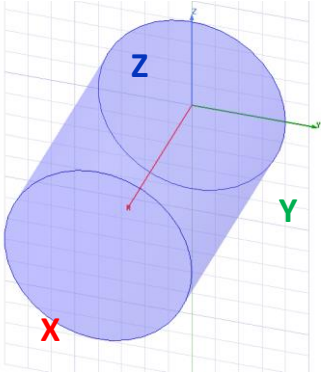



XY

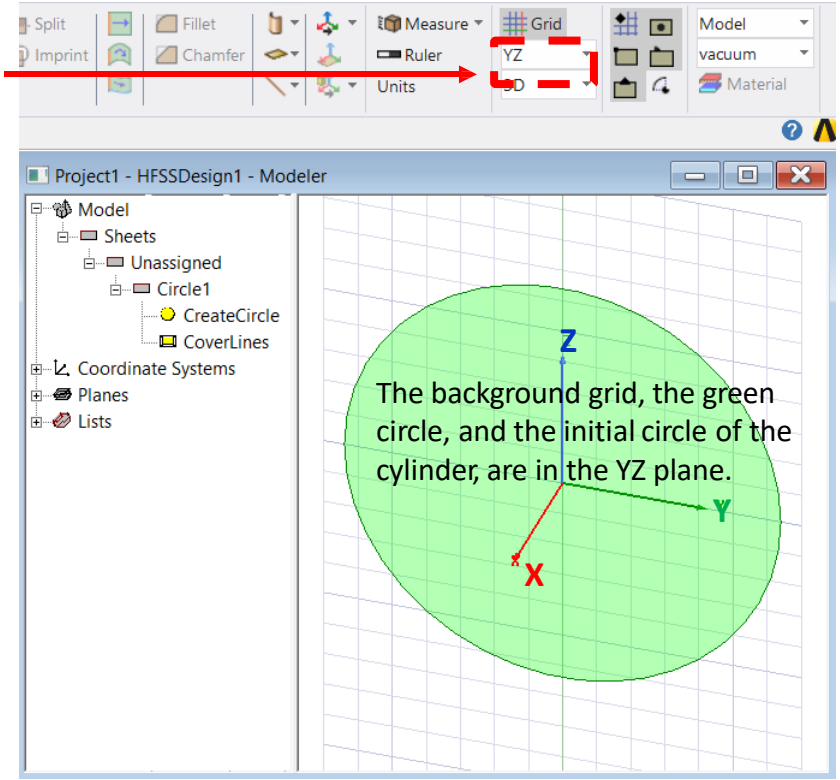
# Drawing in Planes – Modeler > Grid Plane

To select a plane in which 2D objects get drawn, click on this tumble box under Grid, in the Draw tab.

For 3D objects, this plane defines the starting plane or the orientation. A box or cylinder, drawn by clicking in the modeler, would draw the initial 2D shape in the YZ plane here.

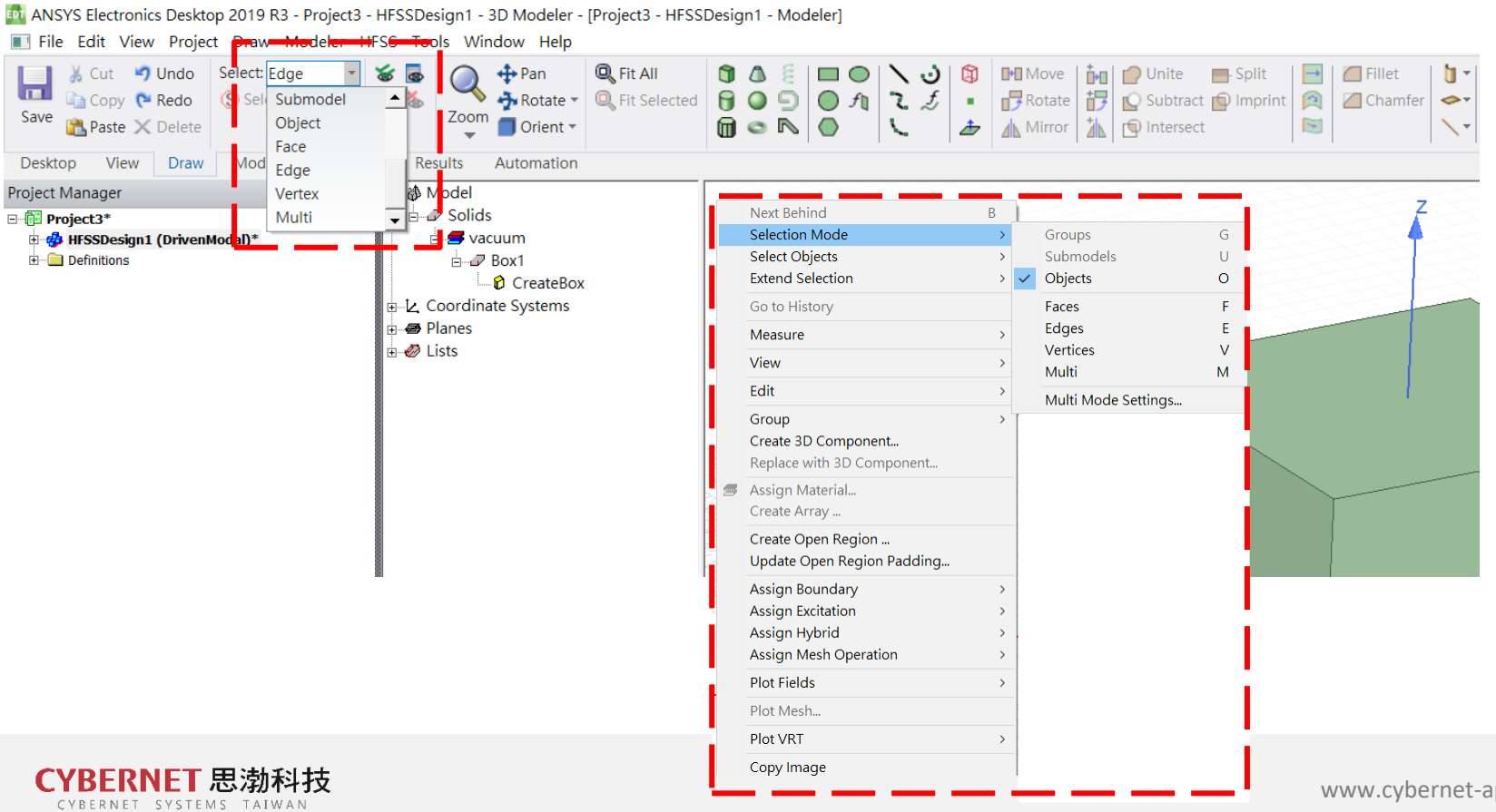



An existing 3D object can be modified by changing the **Axis** in the object's Properties dialog box.

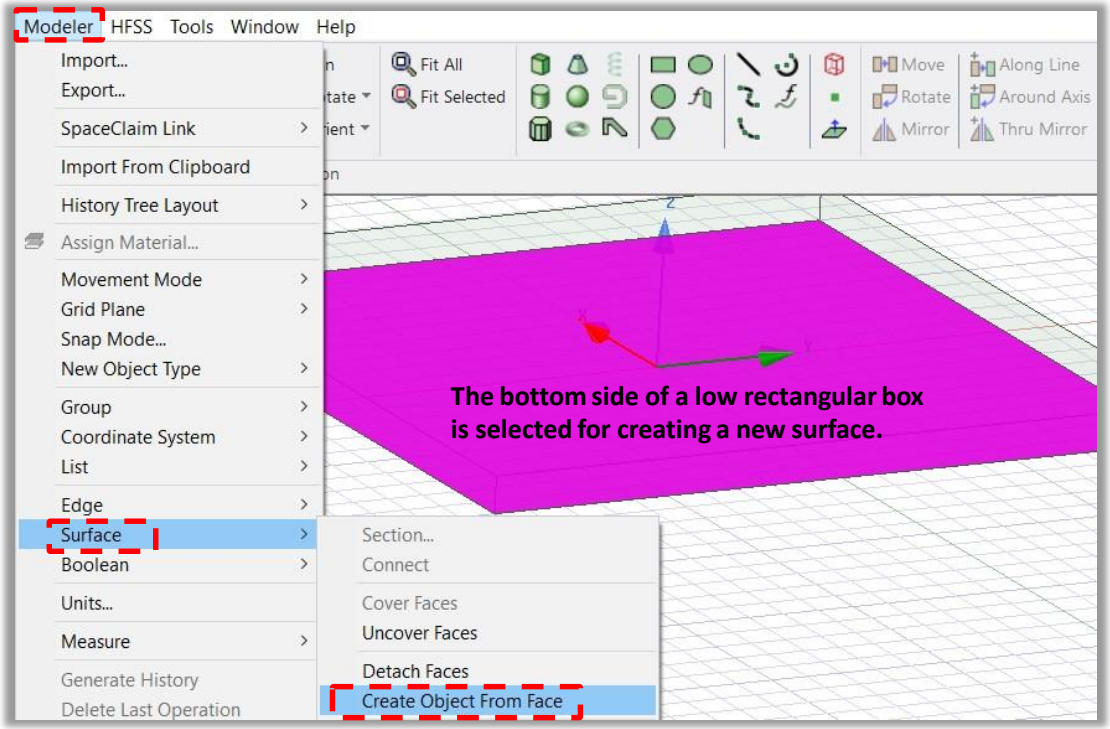



**Note**: The drawing plane can also be specified from ***Modeler > Grid Plane***.

# Selection mode



# Surface from Face – One from Another




***Edit > Select > Faces***

There are several ways to ***select*** in the HFSS **Modeler**, including **Faces**, **Edges**, and **Vertices**. Specific selections are often needed for these operations. These **Select** modes can be accessed from ***Edit > Select*** or by ***right clicking*** in the ***Modeler*** and choosing one of the **Select** modes.

***Modeler > Surface > Create Object From Face***

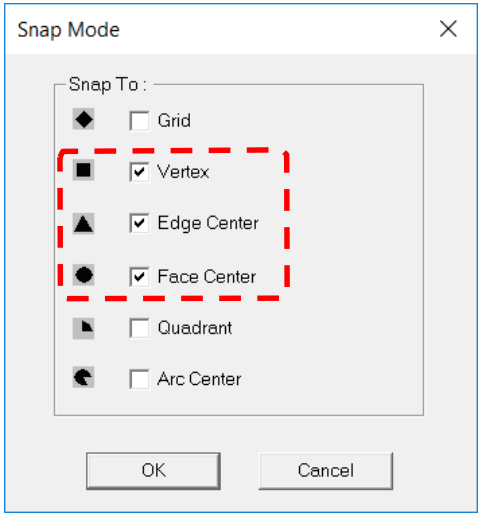
A new geometric surface object can be created from the face of an existing object, such as a box.

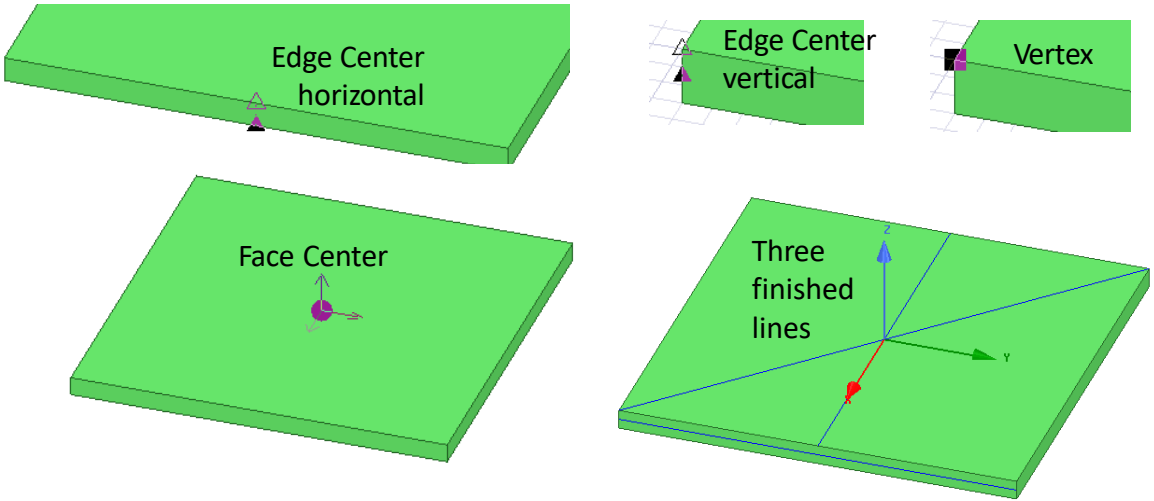
The appropriate face of the box must be selected/highlighted for executing this command.

**Note**: There are many operations in the **Modeler** where an object can be created from another object, hence the label "one from another" for these operations available in the **Modeler** pull down menu.

# Snap Mode for Drawing Lines on Surfaces

Objects can be drawn on existing objects using vertices and midpoints. This process starts with setting the HFSS ***Modeler > Snap Mode*** to include **Vertex, Edge Center,** and **Face Center**.



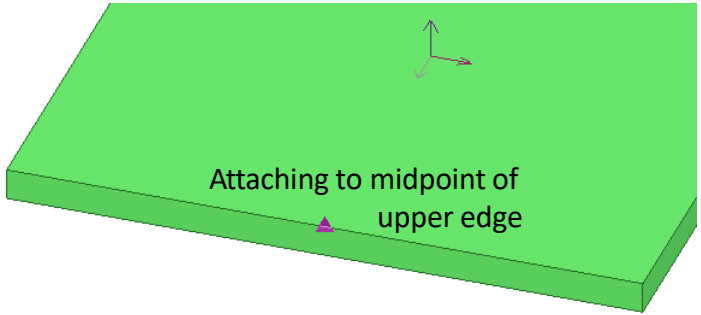



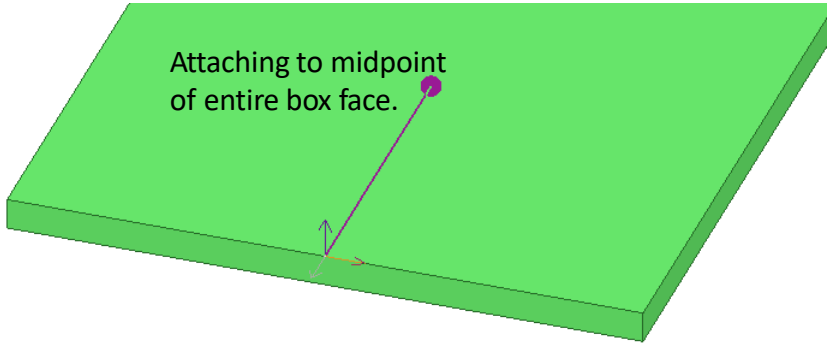

**Note1**: If Grid is checked, it may override snapping to other selections, such as **Vertex, Edge Center,** *etc.*

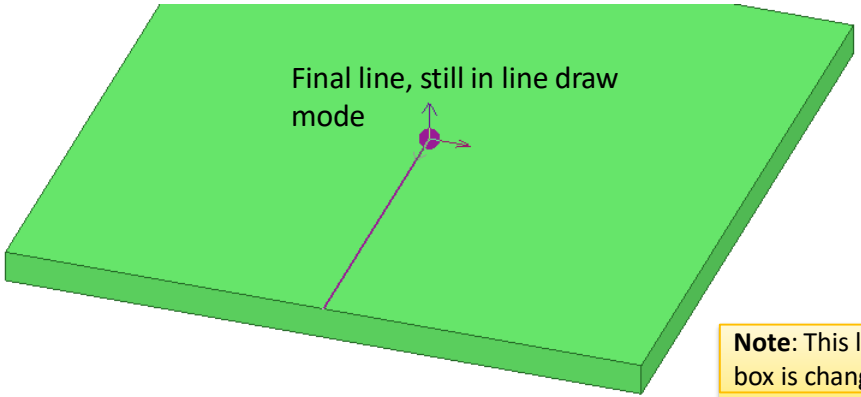
**Note2**: The coordinate system (axes) are hidden in these views. To change the coordinate systems view, select ***View > Coordinate Systems > Hide***.

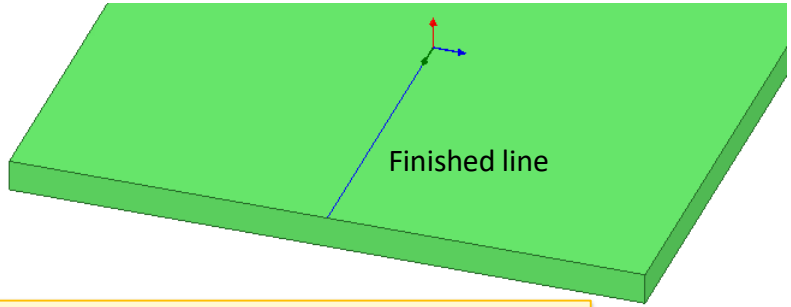
# Draw Lines on Surfaces using Snap Mode

Set HFSS ***Modeler > Snap Mode*** to include ***Face Center*** and ***Edge Center***. Select ***Draw > Line***, and click on the edge center and middle of the top face to draw a line from the middle of the edge to the center.



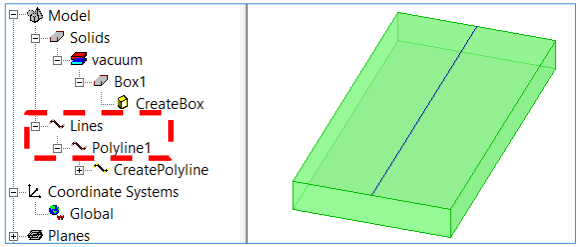



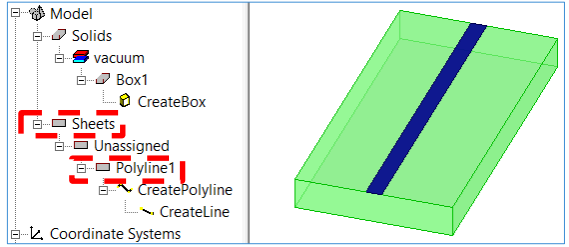



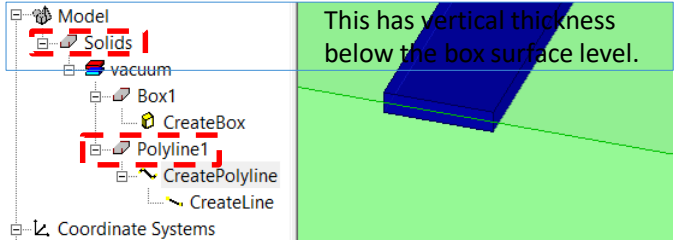

**Note**: This line will not change length if the length of the underlying box is changed; this line is not intrinsically parameterized to the box.

# Modify Lines into 2D and 3D Shapes

The ***Command*** dialog box allows changes of ***Type*** and dimensions. Here thickness, and then height, gets added to the original line. The object changes categories in the Modeler tree, from line to sheet to solid.




Command

| | Name | Value | Unit |
|---|---|---|---|
| | Command | CreatePolyline | |
| | Coordinate System | Global | |
| | Number of points | 2 | |
| | Number of curves | 1 | |
| | --Cross Section | | |
| | Type | Line | |
| | Orientation | Auto | |
| | Width/Diameter | 0 | mm |
| | Top Width | 0 | mm |
| | Height | 0 | mm |
| | Number of Segments | 0 | |
| | Bend Type | Corner | |

Command

| | Name | Value | Unit |
|---|---|---|---|
| | Command | CreatePolyline | |
| | Coordinate System | Global | |
| | Number of points | 2 | |
| | Number of curves | 1 | |
| | --Cross Section | | |
| | Type | Rectangle | |
| | Orientation | Auto | |
| | Width/Diameter | 1 | mm |
| | Top Width | 0 | mm |
| | Height | 0 | mm |
| | Number of Segments | 0 | |
| | Bend Type | Corner | |

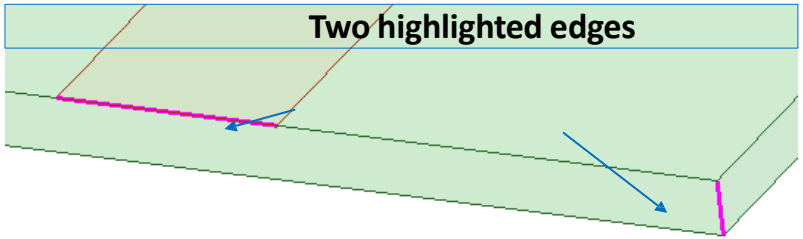
Command

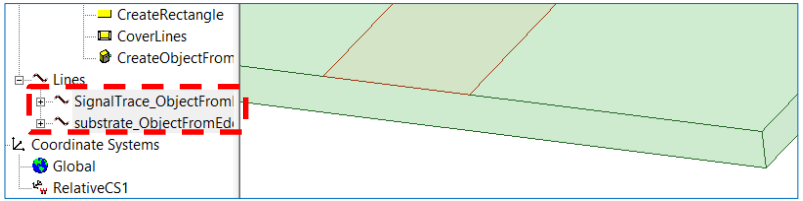
| | Name | Value | Unit |
|---|---|---|---|
| | Command | CreatePolyline | |
| | Coordinate System | Global | |
| | Number of points | 2 | |
| | Number of curves | 1 | |
| | --Cross Section | | |
| | Type | Rectangle | |
| | Orientation | Auto | |
| | Width/Diameter | 1 | mm |
| | Top Width | 0 | mm |
| | Height | 0.2 | mm |
| | Number of Segments | 0 | |
| | Bend Type | Corner | |

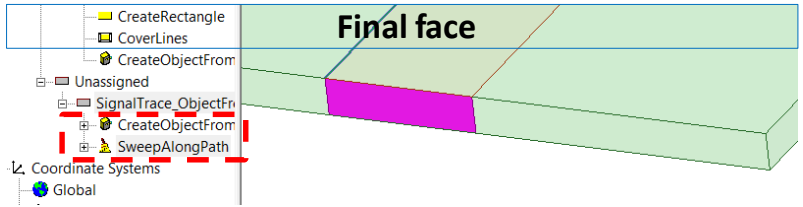
# Sweep Along Path to Create a Face for Port

This two step process first uses a ***Modeler > Edge > Create Object from Edge*** command, to create two new objects. Then the two new objects are used for a ***Draw > Sweep Along Path***.

1. Set ***Modeler*** to ***Select Edges***
2. Select the edge of the surface conductor in the middle of the block.
3. Holding down the control (Ctrl) key, also select a vertical edge of the block.
4. Select ***Modeler > Edge > Create Object from Edge*** Two lines appear in the Model tree.
5. With the two new objects selected….(Order of selection makes a difference)
6. Select ***Draw > Sweep > Along Path…***

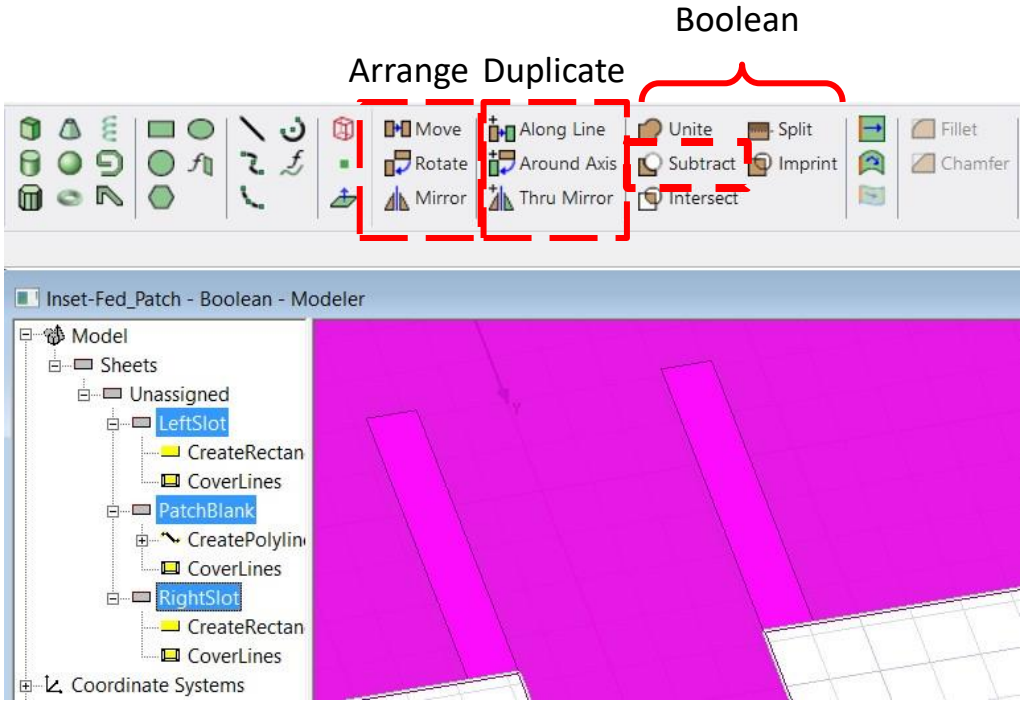


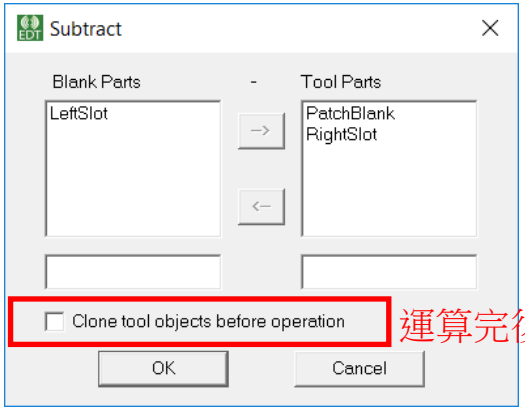




**Note**: Of course this approach can be used to create rectangles for many applications in addition to microstrip ports.

# Boolean Subtraction Icon




運算完後留下減物

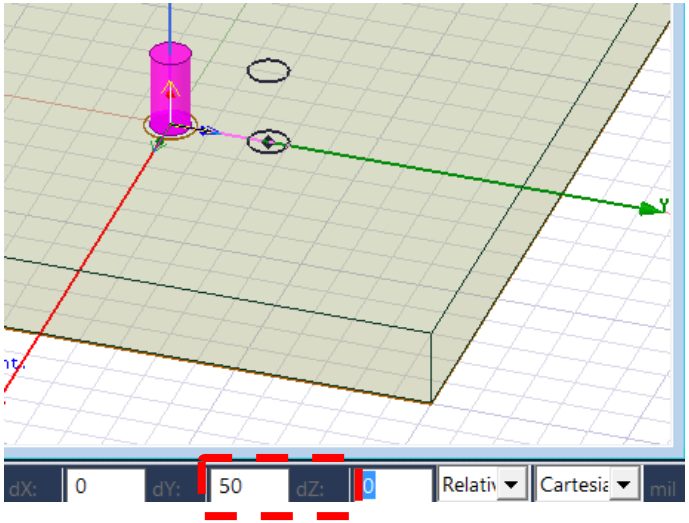
***Blank Parts*** is what gets modified and stays. ***Tool Parts*** is what gets removed (subtracted) and goes away…unless one clones first. Cloning keeps the original ***Tool Parts*** after the Boolean operation.

This Boolean subtraction is also available from ***Modeler > Boolean > Subtract***.

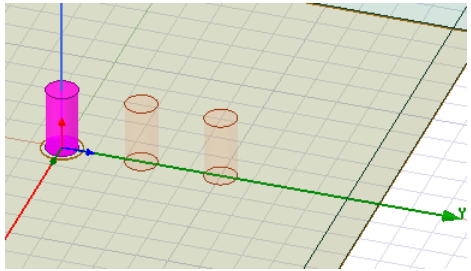
# Edit > Duplicate > Along line

Duplicates selected/highlighted objects along a line specified in the Modeler

- Found under the ***Edit*** pull down, not **Modeler** or **Draw**
- Select ***Edit > Duplicate > Along Line***
- Specify duplication vector by clicking in the Modeler or by using the coordinates in the lower right.

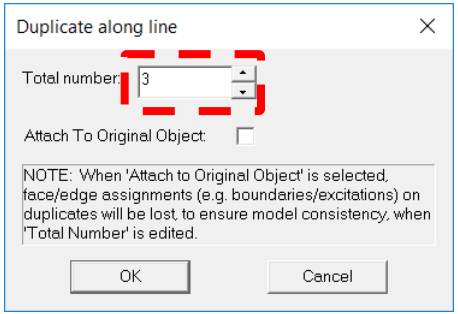



**50 mm Y offset chosen**



**Total number of 3 chosen from pop up dialog box**

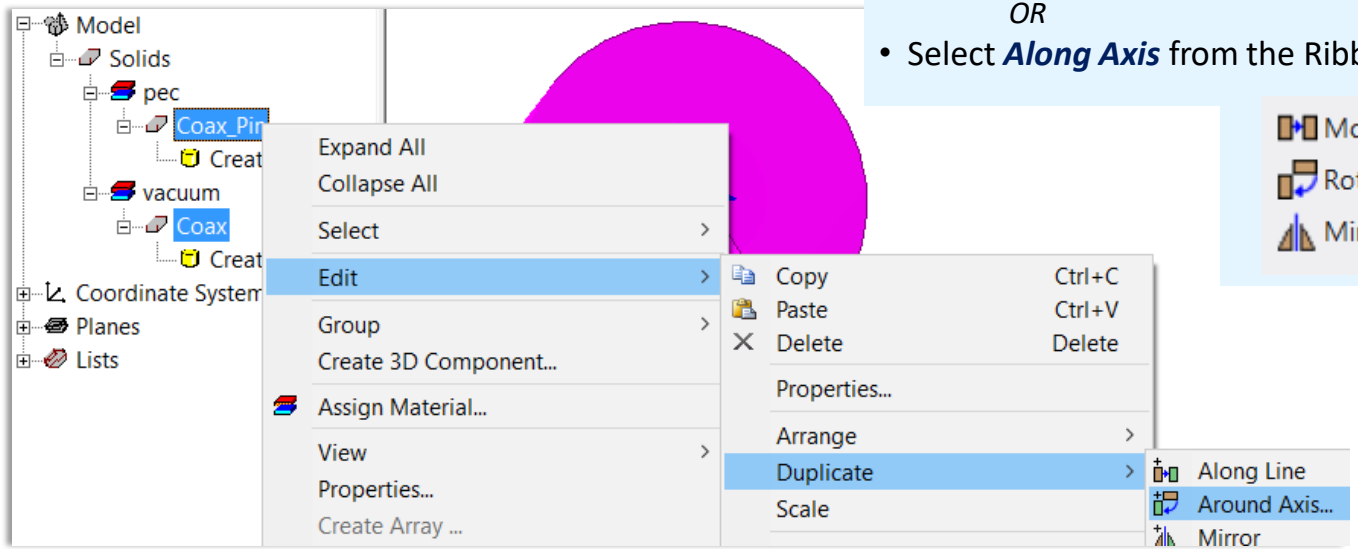

***Duplicate*** options in the Draw tab of the Ribbon



# Edit > Duplicate > Around Axis 1/3

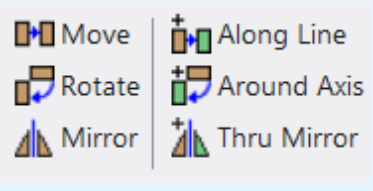
Steps for the ***Duplicate Around Axis*** command:

- Highlight both sections of the coax, Coax and Coax_Pin, using ***Ctrl-D*** to make multiple selections.
- Select ***Edit > Duplicate > Around Axis*** to get the ***Duplicate Around Axis*** dialog box.




*OR*

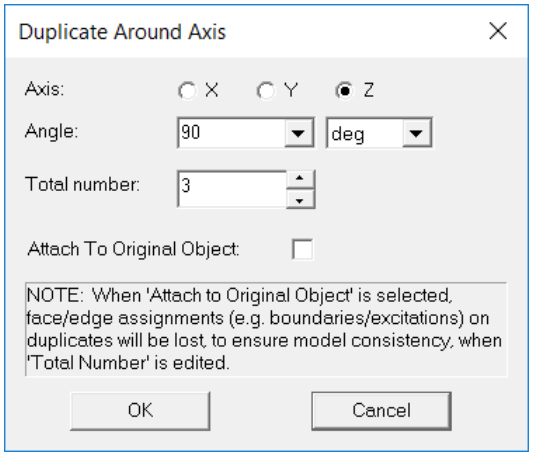
- Select ***Along Axis*** from the Ribbon in the Draw tab.

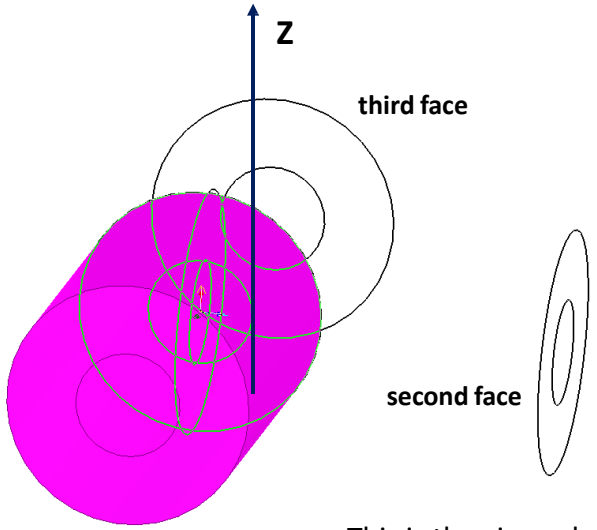


# Edit > Duplicate > Around Axis 2/3

When the ***Duplicate Around Axis*** dialog box appears:

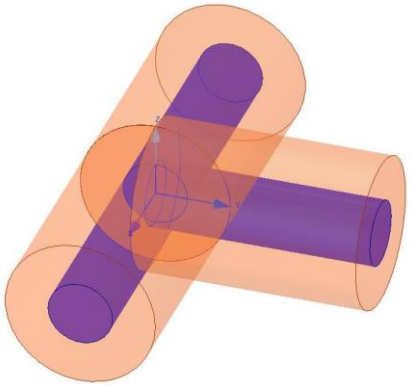
- Select or verify that ***Axis:*** is set to ***Z***
- Select or verify that ***Angle:*** is set to ***90 deg***
- Set ***Total Number***: to ***3***
- Click ***OK***



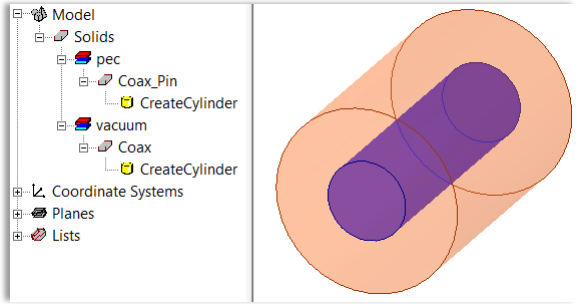

This is the view when the command is in progress.

# Edit > Duplicate > Around Axis 3/3

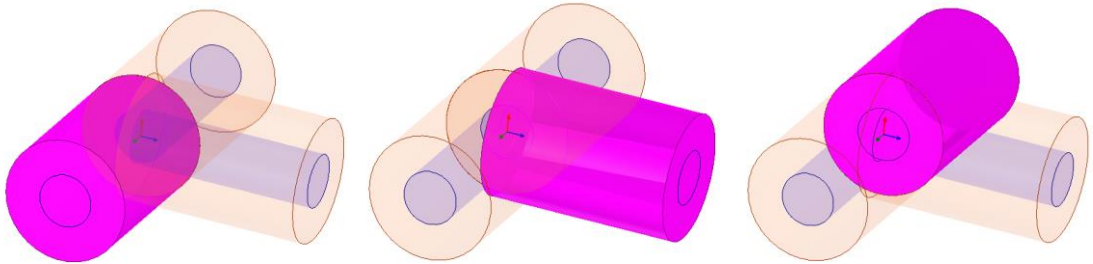


The ***Duplicate Around Axis*** command can be used to construct this 3-port coaxial tee by starting with just one section.

This comes from the Getting Started example in Help/HFSS/GSG file/HFSS Coax Tee.pdf.




The starting point is one section consisting of a coax outer shield and inner conductor pin.



The three sections, resulting from the ***Duplicate Around Axis*** operation, can be pictured by highlighting each section in the resulting finished structure.

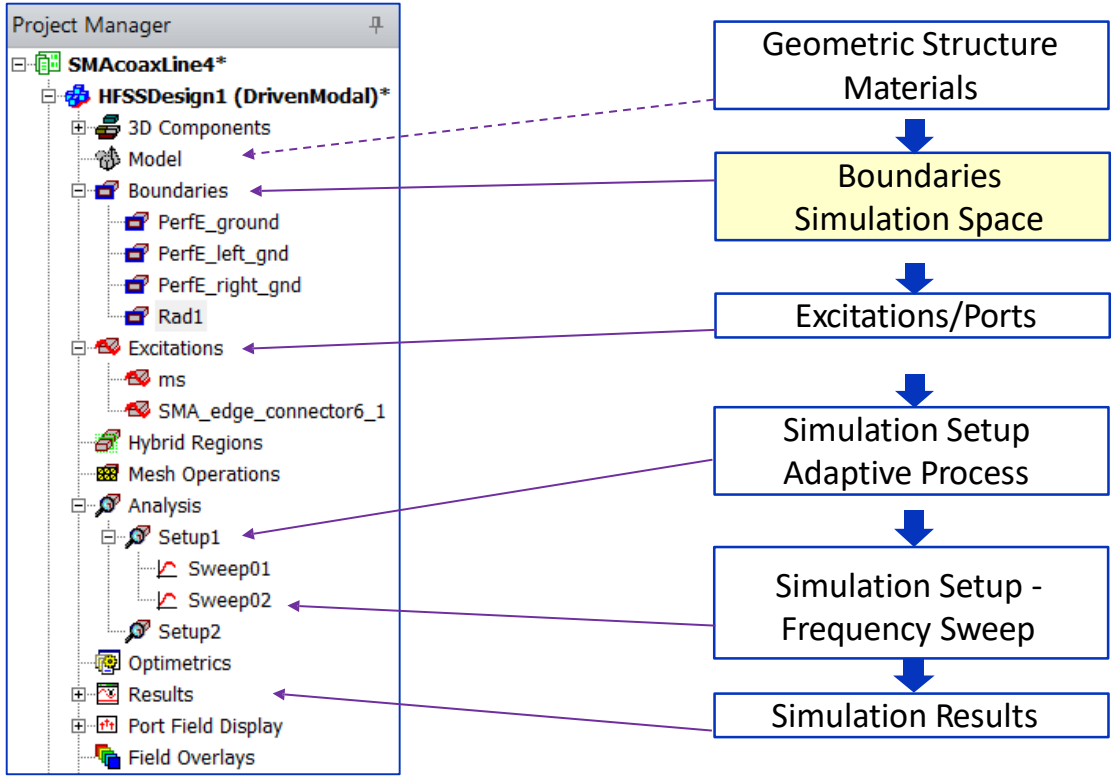
# Boundaries and Simulation Space

- Boundaries Define, Simulation Space and Types of Boundaries

# Boundaries and Simulation Space

- **Boundaries Define Computational Volume**
  - Boundaries define the simulation space (also called computational volume or solution space).
  - Closed models with PEC can serve as the outer surface of the geometry.
  - Air around a structure in the computational volume (simulation space).
  - Radiation boundaries allow energy out of the solution space.
- **Creating Boundaries and Simulation Space in HFSS FEM**
  - *Create Open Region* makes a geometric region and assigns a boundary condition.
  - *Draw > Region* creates a Region in the model with padding.
  - PML has a setup wizard.
  - Assigning Boundary Conditions to a radiating surface bounding box
- **Types of Boundaries in HFSS**
  - List of boundary types
  - PerfE - PEC - perfect electrical conductor
- **Boundaries versus Materials**
  - perfE can be used in place of ideal conductor material
  - Microstrip and stripline applications

# Boundaries in HFSS Simulation Workflow




This presentation focuses on Boundaries in HFSS.

The HFSS Boundaries define what gets meshed and included in the FEM simulation.

Boundaries relate to the HFSS model geometry and structure. Sometimes boundaries extend beyond the geometry and sometimes boundaries are defined on the outside of the model geometry.

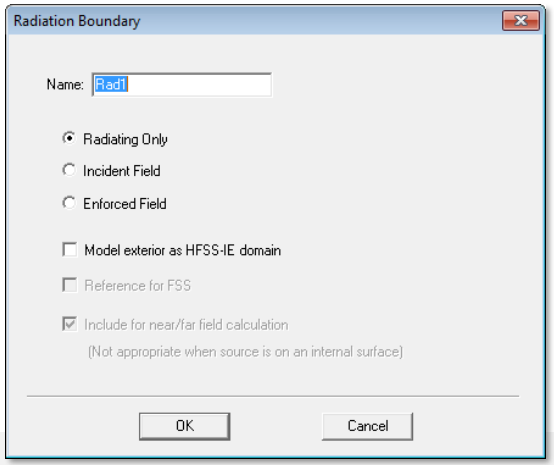
The HFSS Online Help document "***HFSS.pdf***", Chapter 17 ***"Assigning Boundaries in HFSS and HFSS-IE"*** is a good resource for Boundaries.

# Radiation Boundary

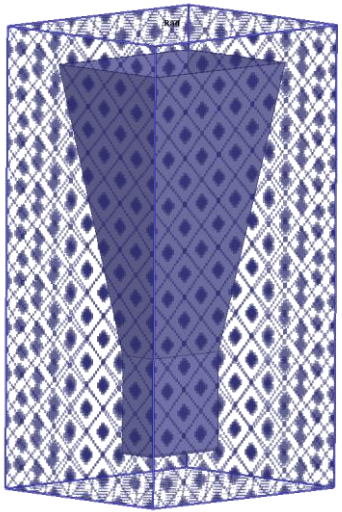
A Radiation Boundary mimics continued propagation beyond boundary plane.

- Absorption achieved via 2nd order radiation boundary
- Absorbs best when incident energy flow is normal to surface
- Distance from radiating structure
  - **Place at least λ /4 from strongly radiating structure**
  - Place at least λ /10 from weakly radiating structure
- Must be concave to all incident fields from within modeled space

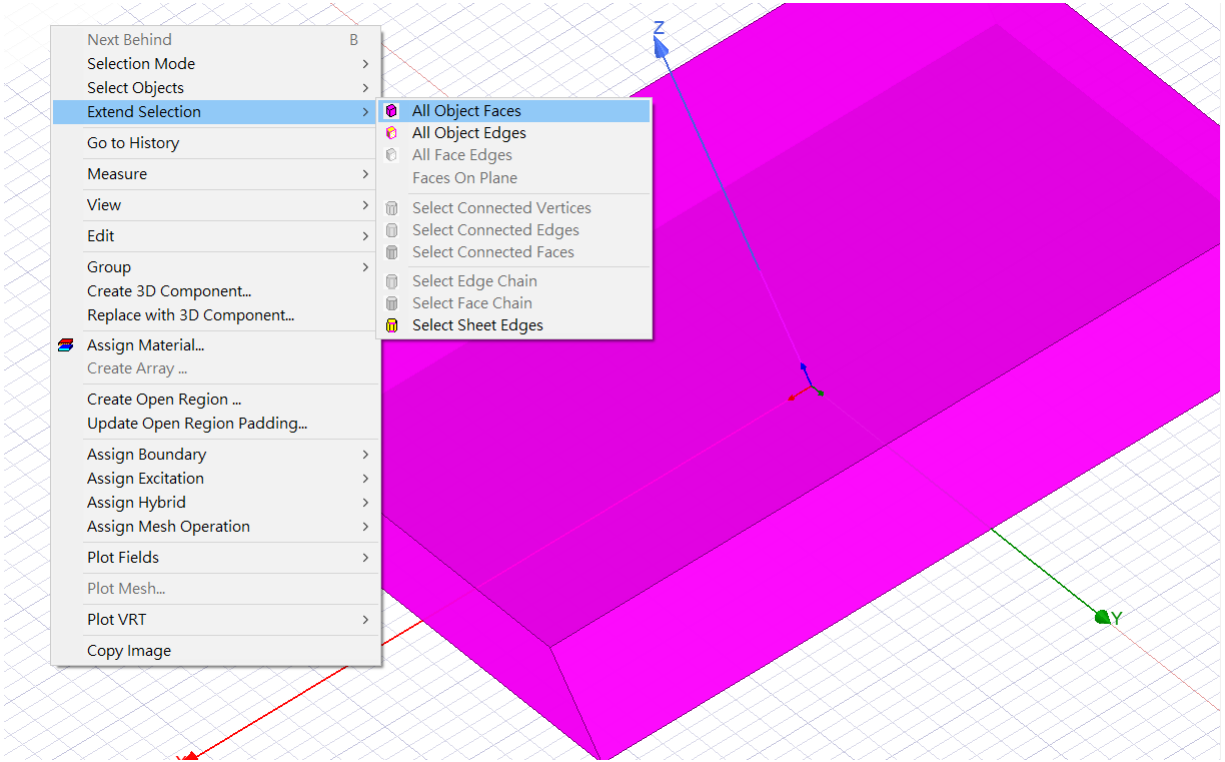
$$(\nabla\times\boldsymbol{E})_{tan} = jk_0\boldsymbol{E}_{tan} - \frac{j}{k_0}\nabla_{tan}\times(\nabla_{tan}\times\boldsymbol{E}_{tan}) + \frac{j}{k_0}\nabla_{tan}(\nabla_{tan}\bullet\boldsymbol{E}_{tan})$$



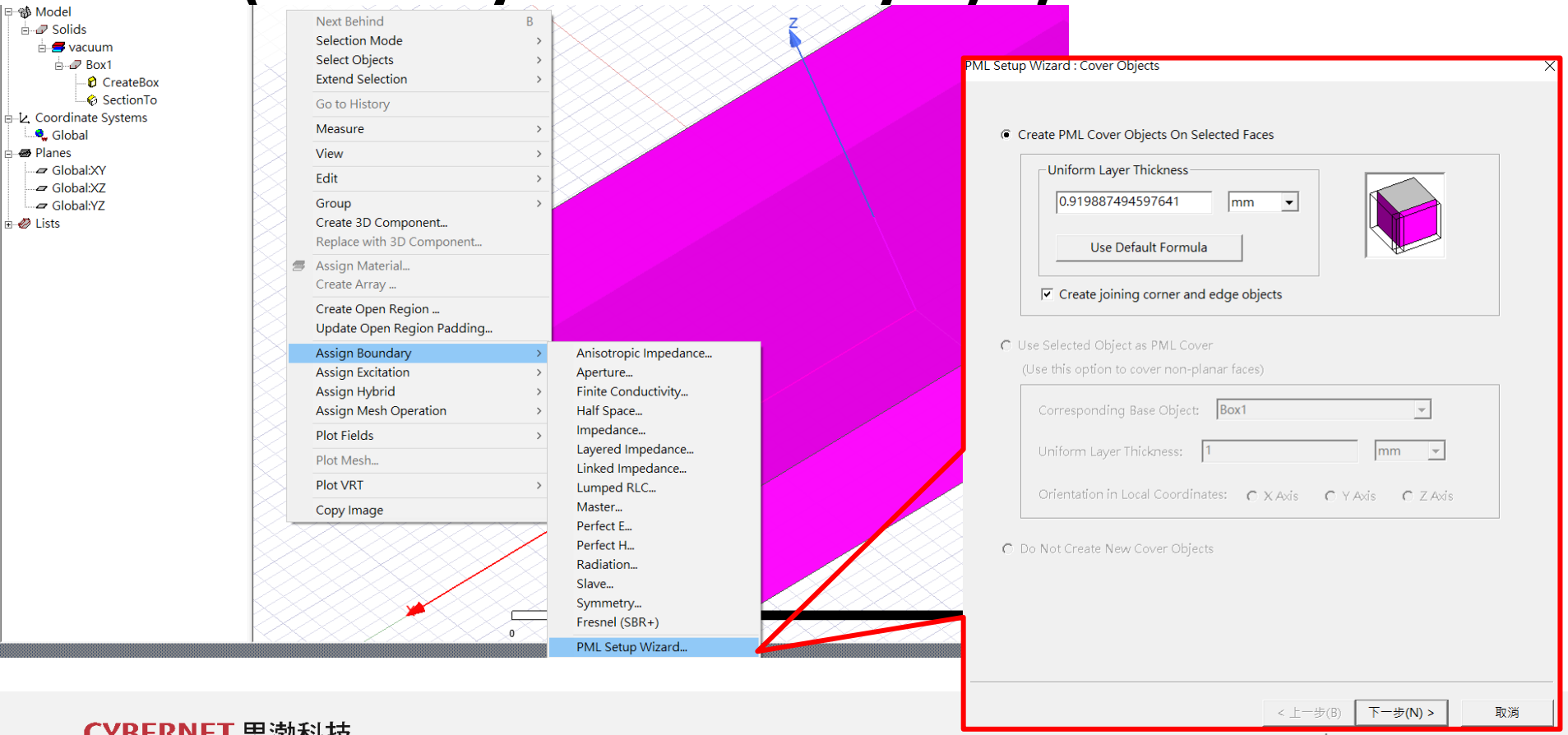

**Boundary is λ /4 away from horn aperture in all directions**



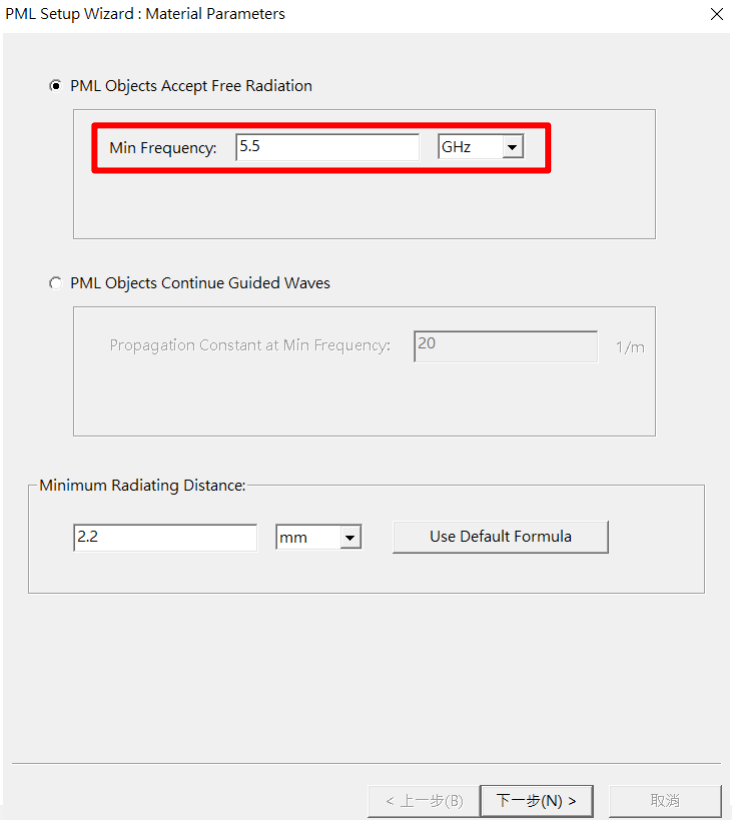
# PML (Perfectly Matched Layer) 1/3

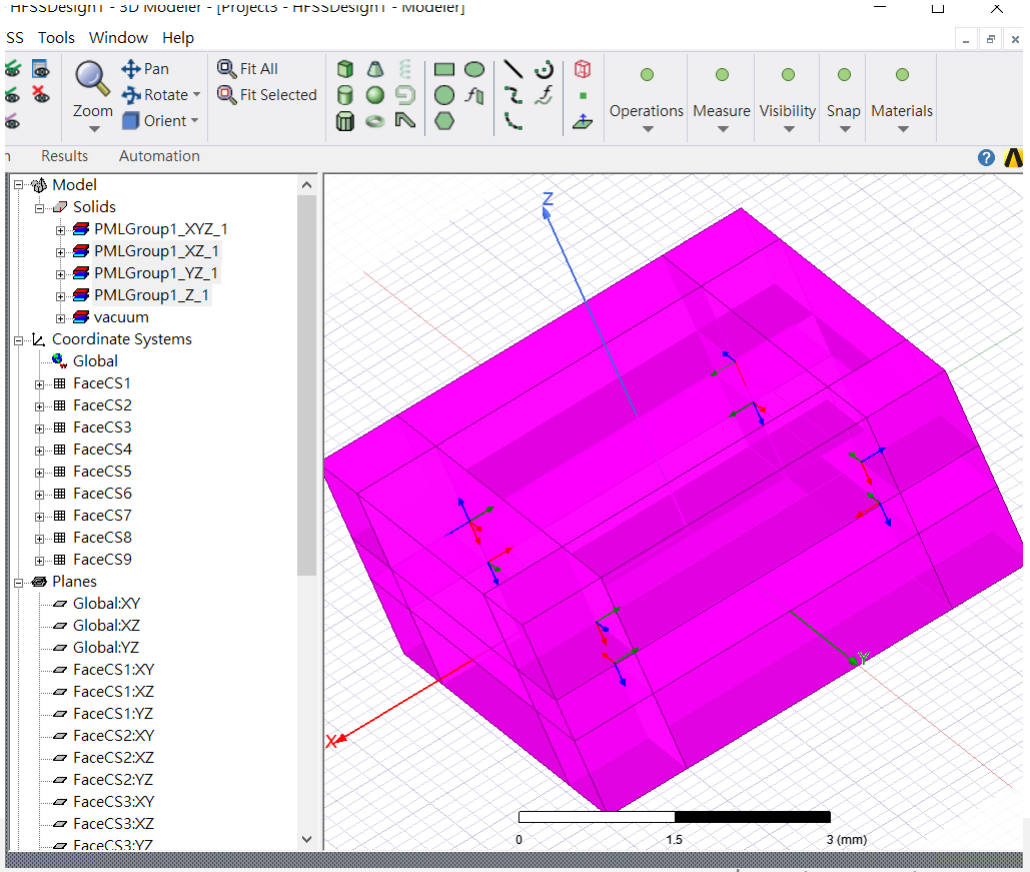


# PML (Perfectly Matched Layer) 2/3



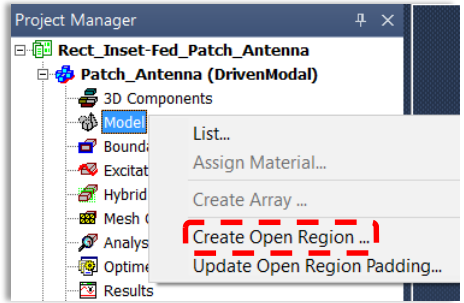
# PML (Perfectly Matched Layer) 3/3



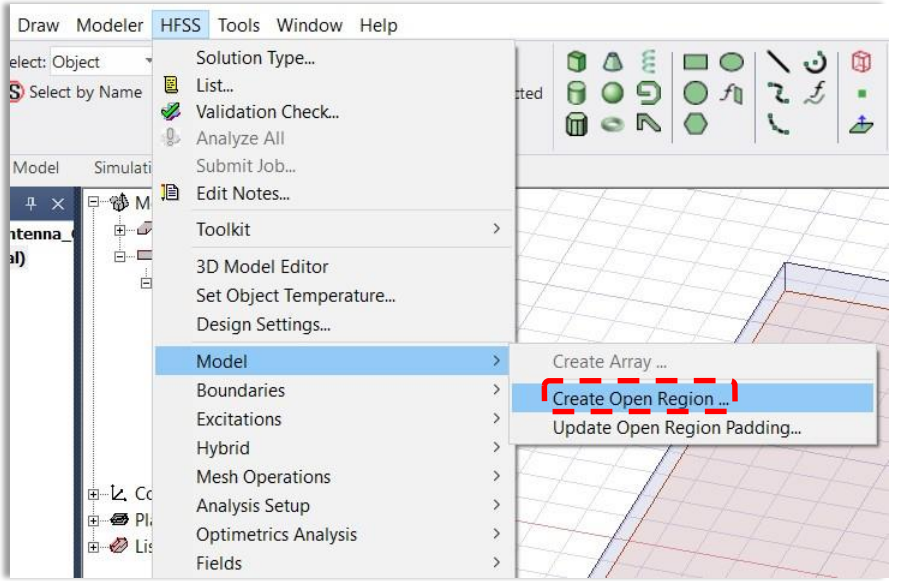
# Automated Open Boundary Setup

In order to create a radiating boundary quickly in HFSS, using ***Create Open Region…***




In the ***Project Manager***, Right-click on ***Model*** and select ***Create Open Region…***

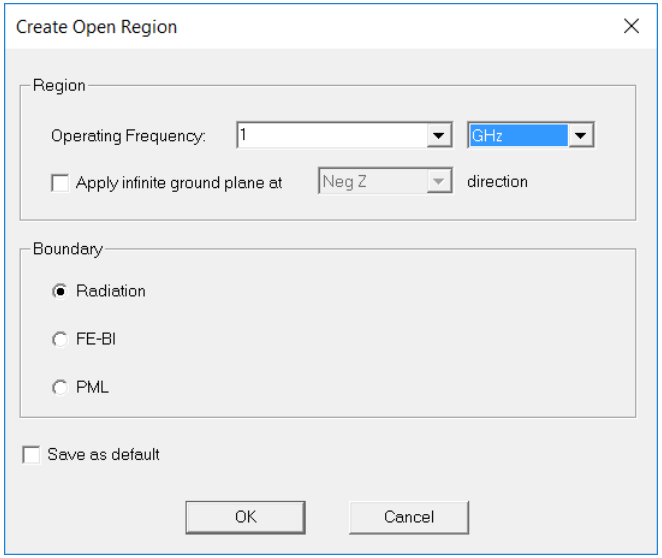
**…OR…**



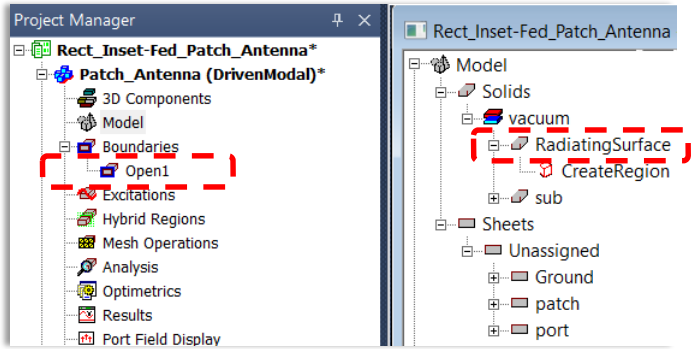

Select on Model and select ***HFSS > Model > Create Open Region…***

…to bring up the ***Create Open Region…*** dialog box

# Specify Frequency in Create Open Region




The main specification for the ***Create Open Region…*** dialog box is the ***Operating Frequency***. This determines the size of the geometric box, the surface where the boundary is applied.

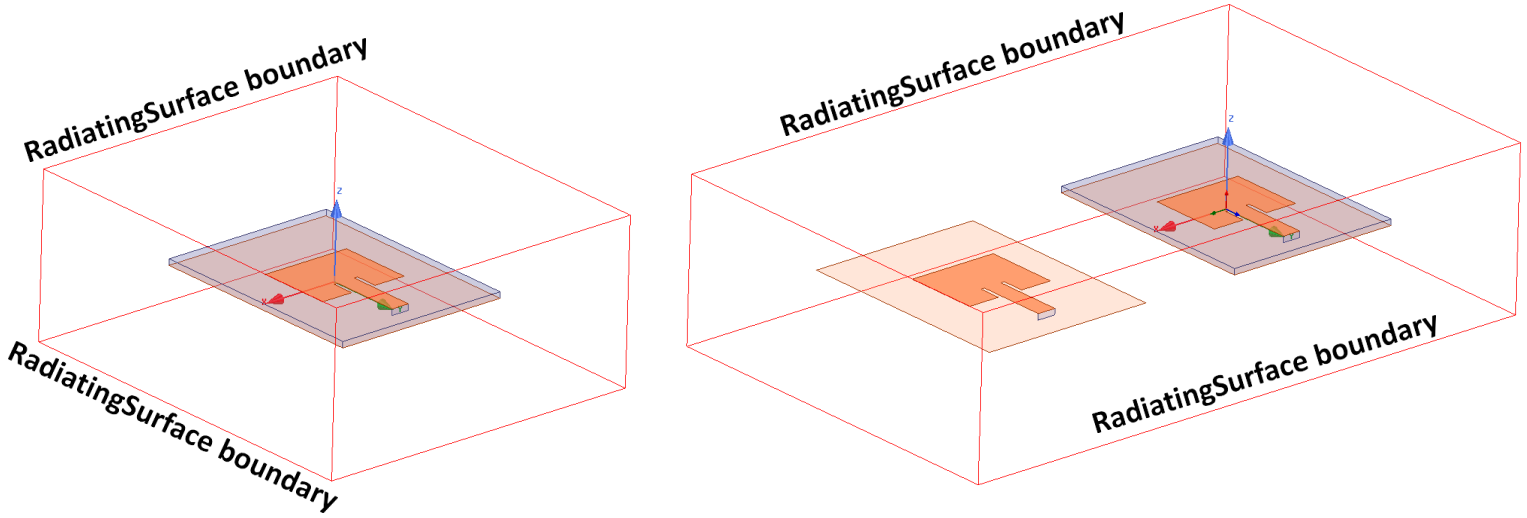



The ***Create Open Region…*** dialog box offers three choices for open boundaries, Radiation (ABC), PML (perfectly matched layer), and FE-BI (finite element - boundary integral) which is not discussed in this module.

***Create Open Region…*** places an entry ***Open***1 in the ***Project Manager*** under ***Boundaries***, and ***Create Open Region*** puts a ***RadiatingSurface*** in the ***Model*** Tree.

# Open Region Automatically Adjusts Size

The size of the bounding box RadiatingSurface is automatically adjusted based on the size of the model structure. If the geometry gets bigger, or more objects get added, the boundary and the simulation space get larger.



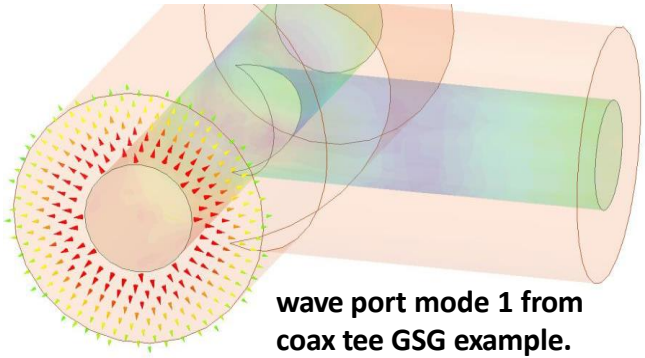

The ***Operating Frequency*** specification determines the size of the geometric box (the Region…the RadiatingSurface) in the ***Create Open Region…*** operation. It automatically adjusts the size of the box.

# HFSS Lumped and Wave Port

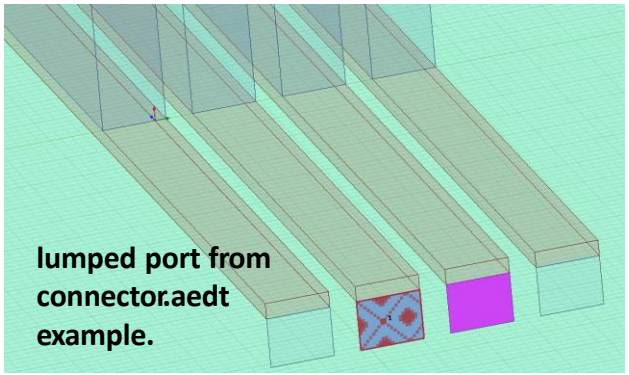
- HFSS Lumped and Wave Ports, Ports and Solution Type

# Lumped and Wave Ports

- **HFSS Lumped and Wave Ports Introduction**
  - Excitations (Ports) in the HFSS workflow
  - Ports are excitations that provide S-Parameters
- **Lumped Ports**
  - Need impedance specified by the user
  - Internal placement
  - Single mode (TEM & quasi-TEM)
- **Wave Ports**
  - Can calculate the impedance of the port
  - External placement
  - Multiple modes
- **Ports and Solution Type**
  - Terminals come with Terminal Solution Type
  - Modes and integration lines come with Modal Solution Type
- **Lumped Ports vs. Wave Ports**

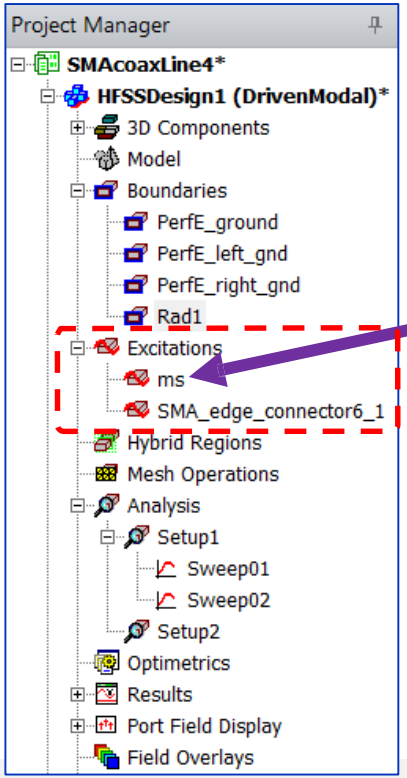


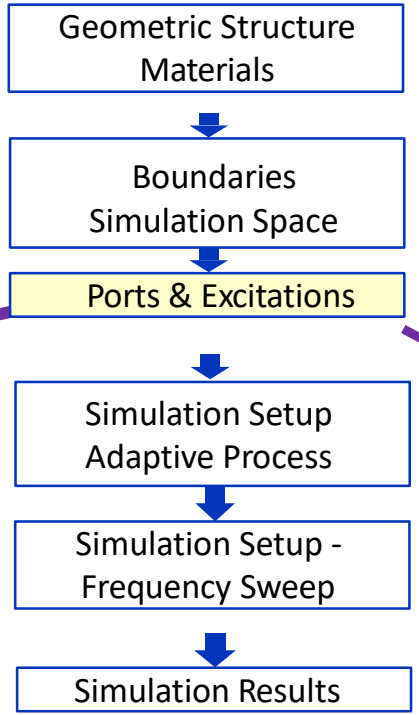
**wave port mode 1 from coax tee GSG example.**

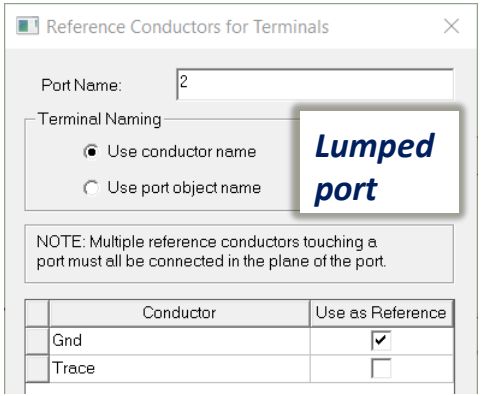


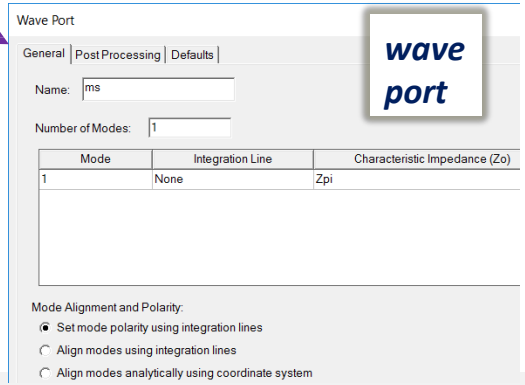
**lumped port from connector.aedt example.**

# HFSS Ports and Excitations in the Workflow



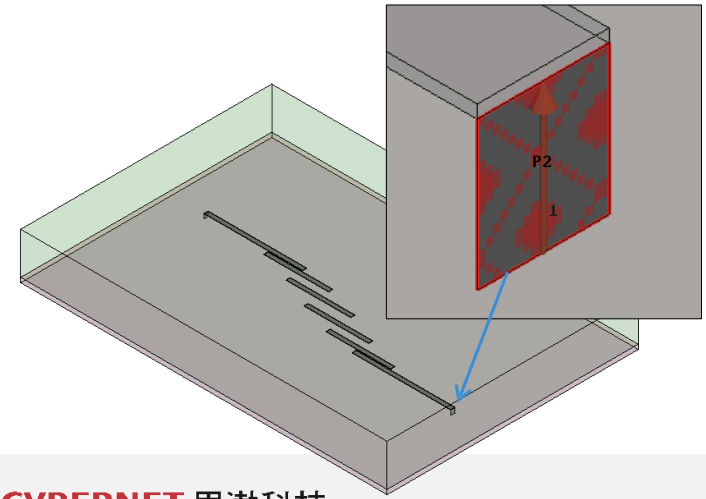



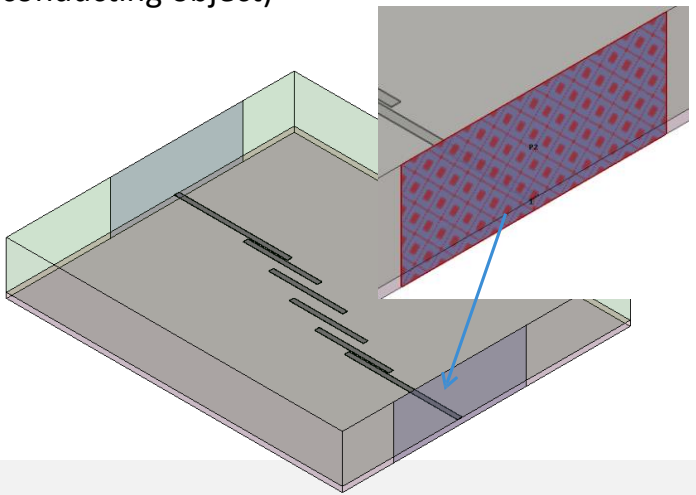



# Lumped versus Wave Ports

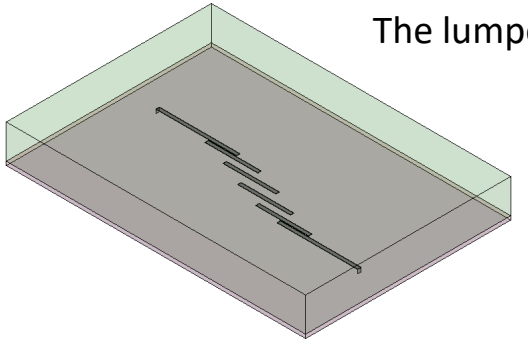
– **Lumped ports can be used to feed printed transmission lines.**
  - S-parameters normalized to ***user-specified*** characteristic impedance
  - Single mode propagation
  - No de-embedding operations available
  - Port must be located inside the model



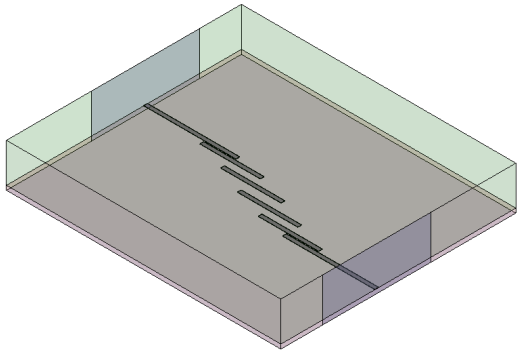

– **Wave ports can be used to feed printed transmission lines.**
  - S-parameters normalized to ***computed*** characteristic impedance (Generalized S-Parameters)
  - Multiple propagating modes possible
  - De-embedding available as post-processing operation
  - Port must touch background object (or be backed by conducting object)



# Simulation Comparison

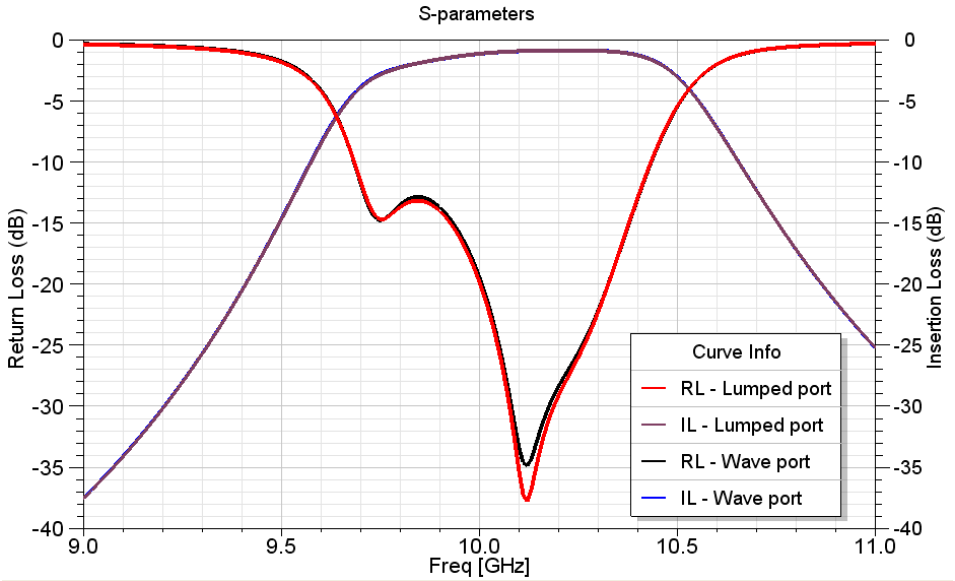


Lumped Ports



Wave Ports

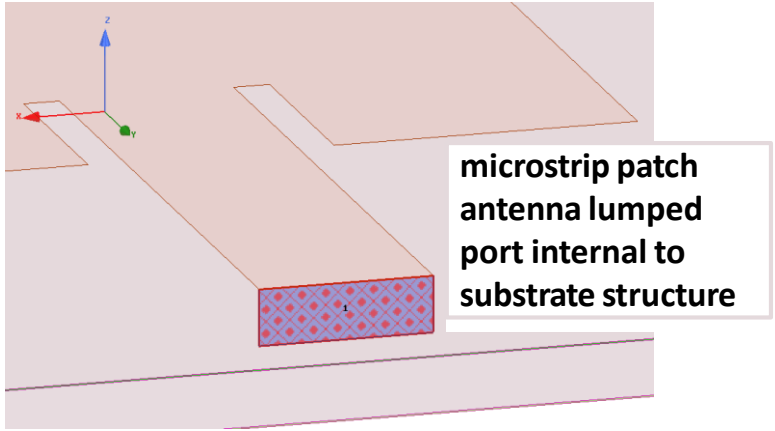
The lumped port and wave port simulation results compare very closely.

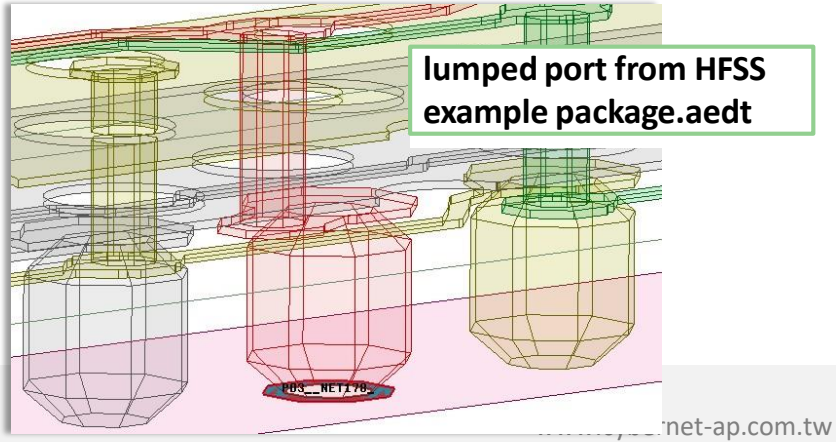


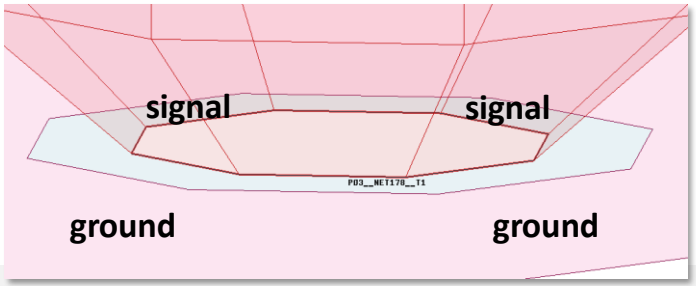
# Lumped Ports

HFSS Lumped ports are used for:

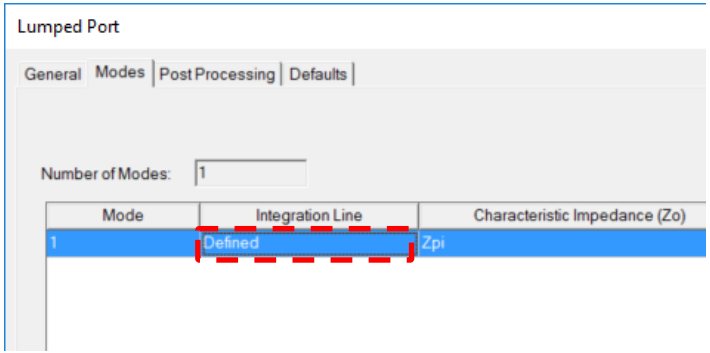
- Signal integrity applications
- TEM (transverse electric magnetic) single-mode propagation
- Circuits that are not well-defined wave guides
- Structures with non-uniform shapes, like connectors and BGAs (ball grid arrays)
- Open structures like antennas



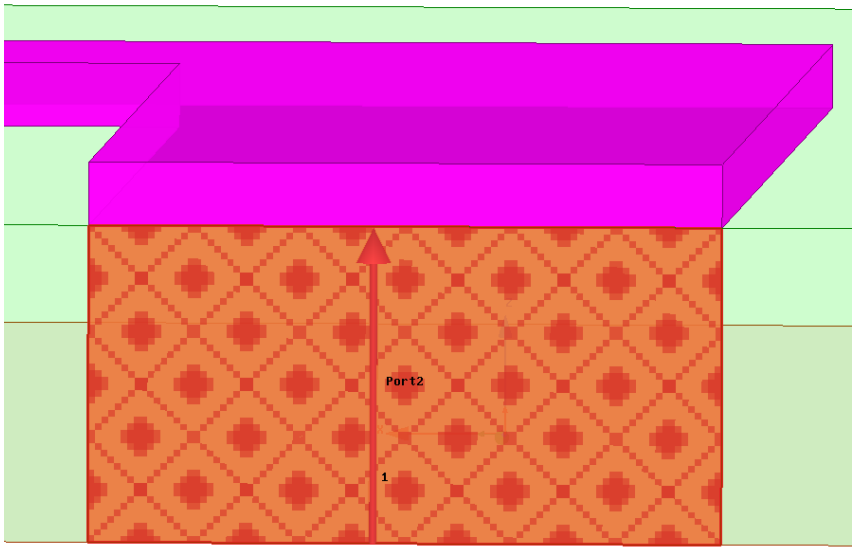



# Lumped Port Mode




Integration lines can be used to define and specify the modes and their polarity. Clicking in the ***Integration Line*** box can bring up a menu for drawing a new line. Integration lines correspond to ***Driven Modal Solution Type***.

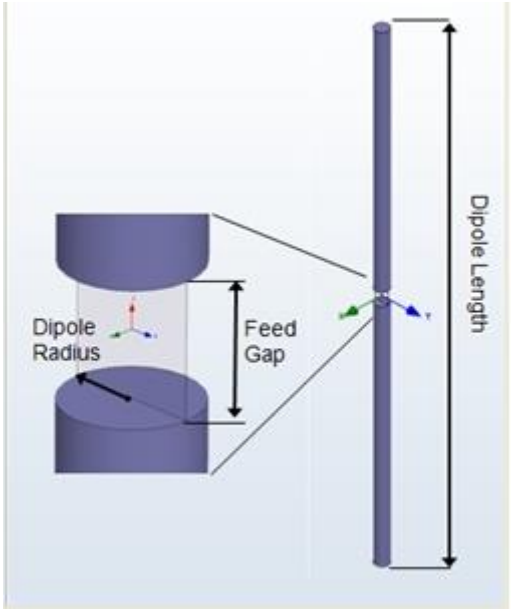



**Integration line in Lumped Port - Driven Modal Solution Type**

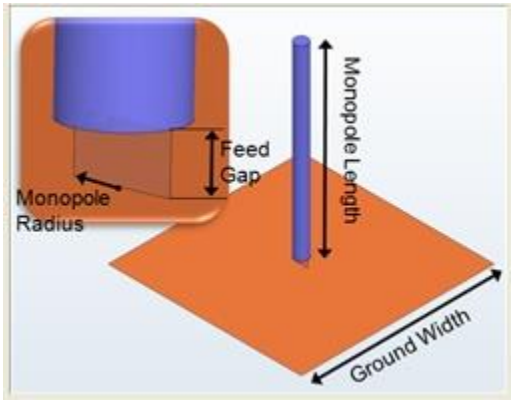
See ***HFSS.pdf***, Chapter 19 ***Assigning Excitations for HFSS… > Wave Port Dialog for Modal Solutions > Set Mode Polarity Using Integration Line.***

# Lumped Ports in Antenna

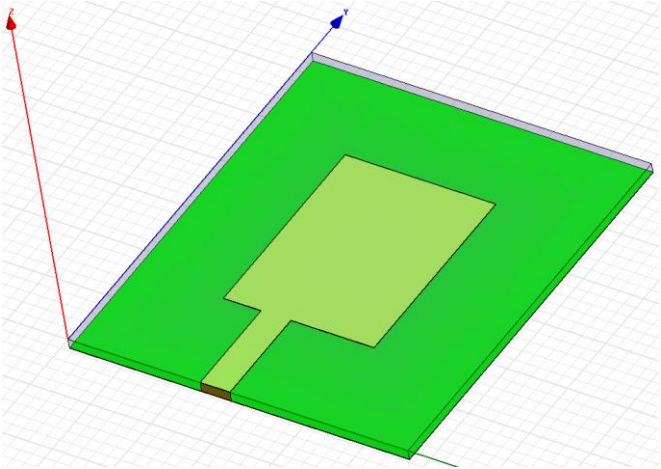
Dipole Antenna




Monopole Antenna



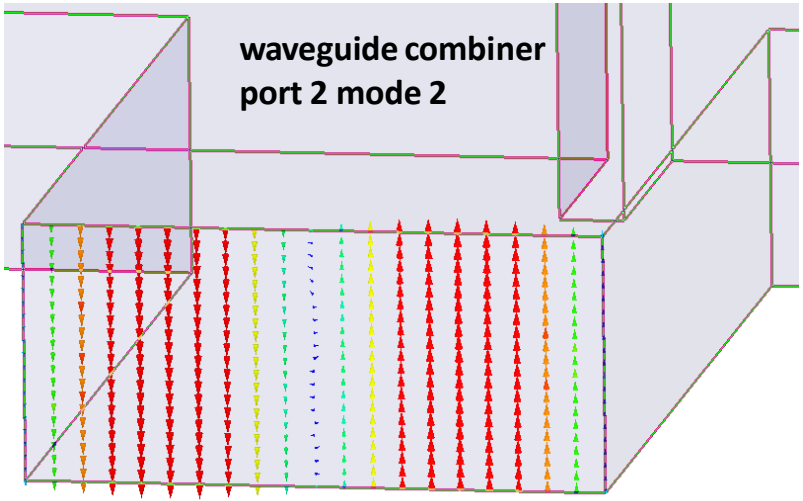

Microstrip Antenna



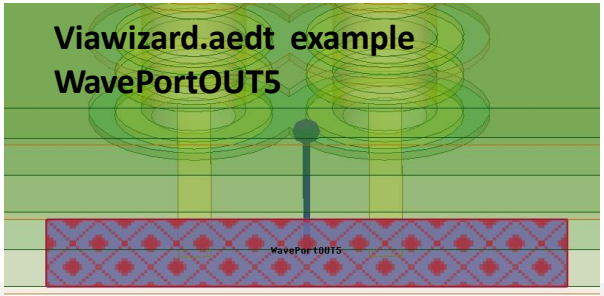
# Wave Ports

HFSS Wave ports are used for:

- Closed structures like waveguides and coaxial cables
- multiple propagation modes like waveguides and odd/even modes on differential pairs
- Surfaces exposed to background object
- Structures with uniform waveguide or transmission line where the port attaches.




The waveguide combiner is a closed waveguide structure that can support multiple propagation modes.



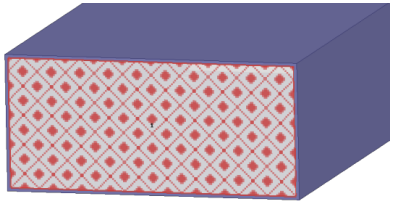

The document "***An Introduction to HFSS***", Chapter 3 ***"HFSS Excitations"*** is available in the installation directories ***/Help/HFSS/GSG***.

See also ***HFSS.pdf***, Chapter 19 ***Assigning Excitations for HFSS…*** available in the HFSS installation directories ***Help/HFSS.***

# Wave Port Sizing

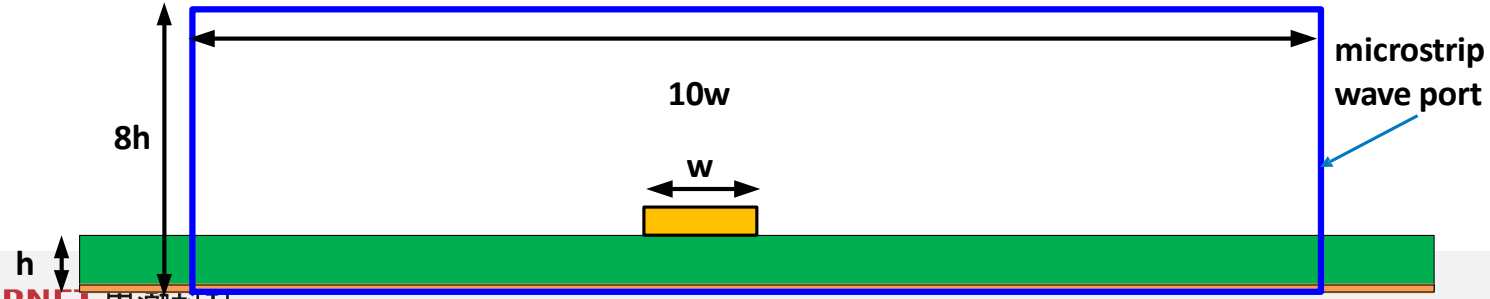
- **Closed Transmission Line Structures**
  - The boundary enforced on the port's edge implies the transmission line modeled by the Wave Port always sits inside a waveguide structure. The enclosing material forms the port's edge boundary.

Coax

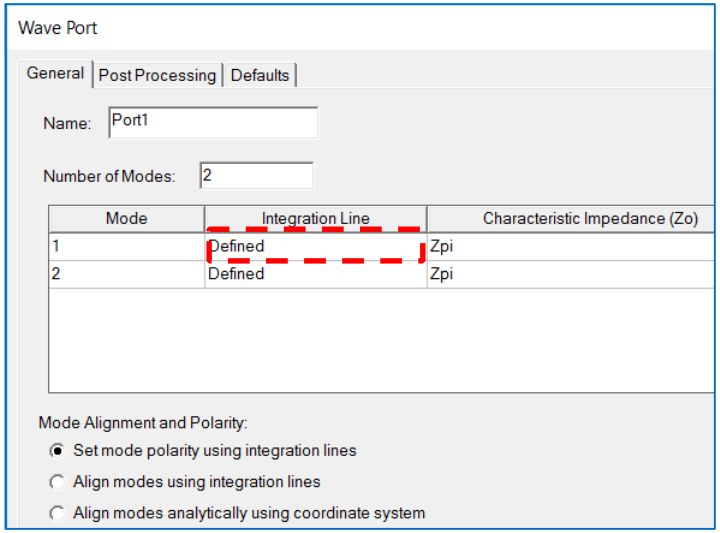


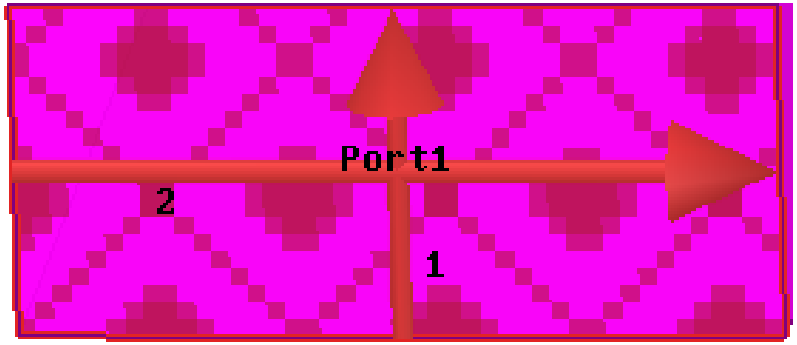
Waveguide

- **Open transmission line structures require additional consideration**
  - Microstrip, Co-Planar Waveguide, Slotline
  - Wave Ports must be large enough to capture the entire transmission line's field structure
    - For open transmission line structures the Wave Port must surround the entire structure.
    - Make sure the transmission line fields are not interacting with the port's boundary condition.
    - Wave Ports too small can lead to incorrect characteristic impedances, and add additional reflection to the results.



# Wave Port Mode



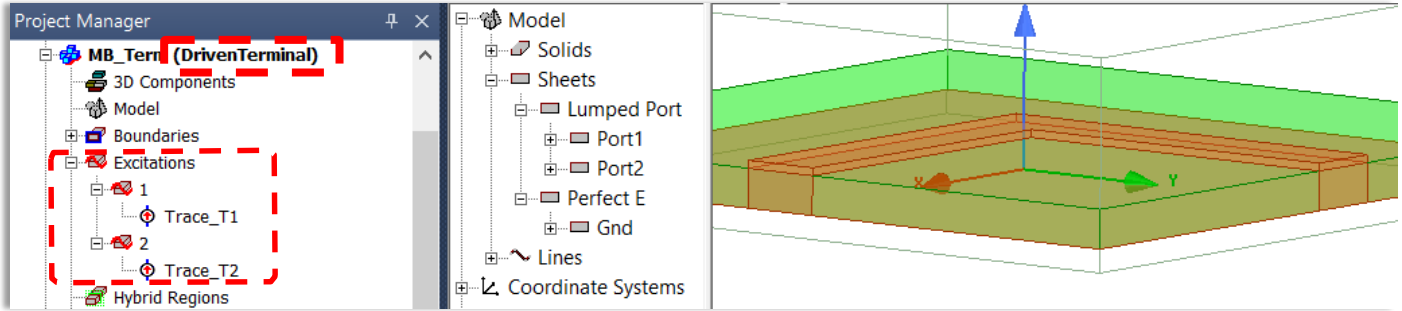




Integration lines can be used to define and specify the modes and their polarity. Clicking in the ***Integration Line*** box can bring up a menu for drawing a new line.

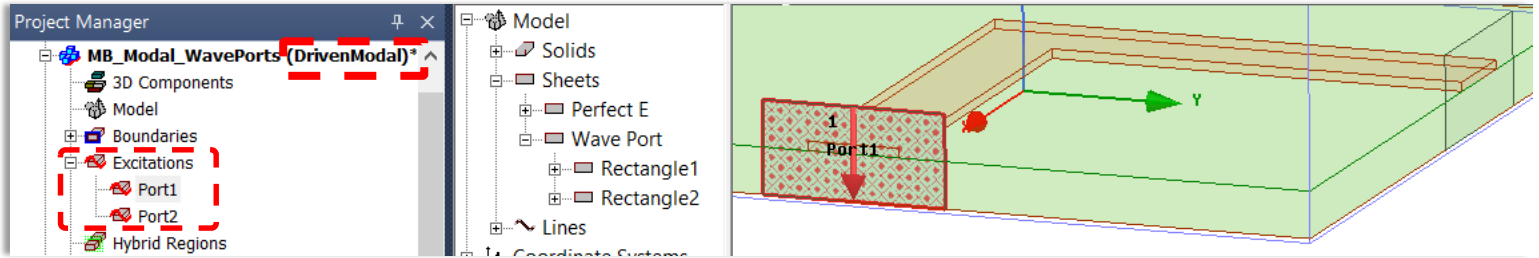
See ***HFSS.pdf***, Chapter 19 ***Assigning Excitations for HFSS… > Wave Port Dialog for Modal Solutions > Set Mode Polarity Using Integration Line.***

# Ports and Solution Types

## Lumped Ports with Driven Terminal Solution Type

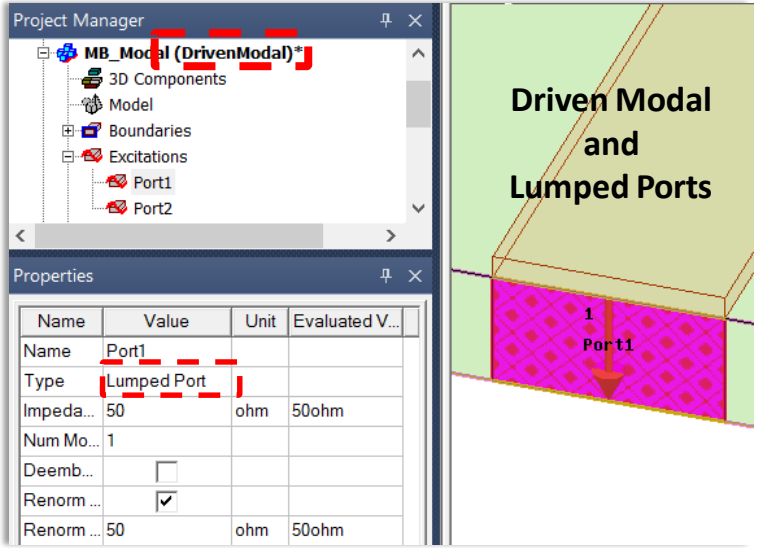



## Wave Ports with Driven Modal Solution Type

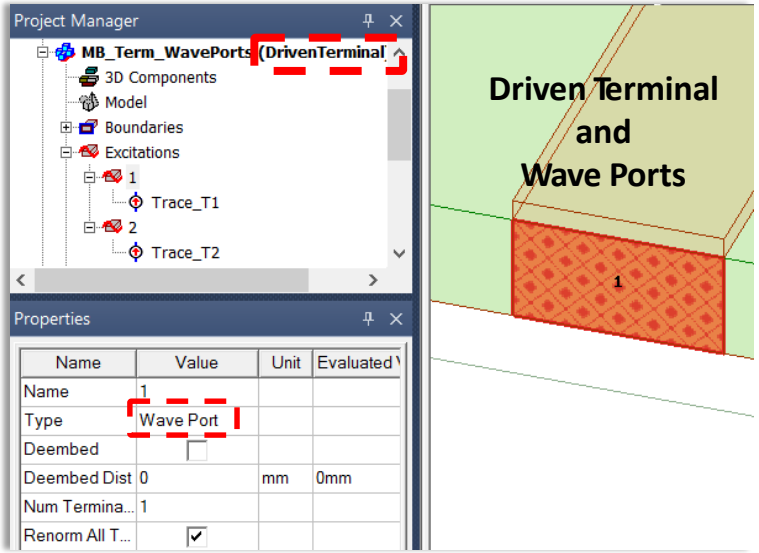



These are common combinations. Notice that the type of port is listed in Properties when the port is highlighted.

# Modal and Terminal Solution Types



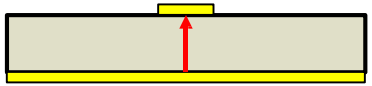

Modes and integration lines in ports come from ***Modal Solution Type***.



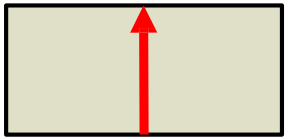

Terminals in ports come from ***Terminal Solution Type***.

# Wave Ports vs Lumped Ports

| | Wave port | Lumped port |
|---|---|---|
| Accessibility | External Faces | Internal to Model |
| Higher order modes | Yes | No |
| De-embedding | Yes | No |
| Re-normalization | Yes | Yes |
| Setup complexity | Moderate | Low |
| Gamma propagation | Yes | No |

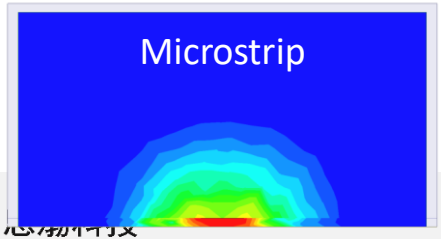


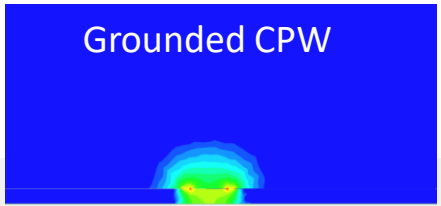
Microstrip line

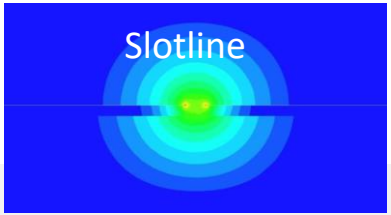


Waveguide

Slotline



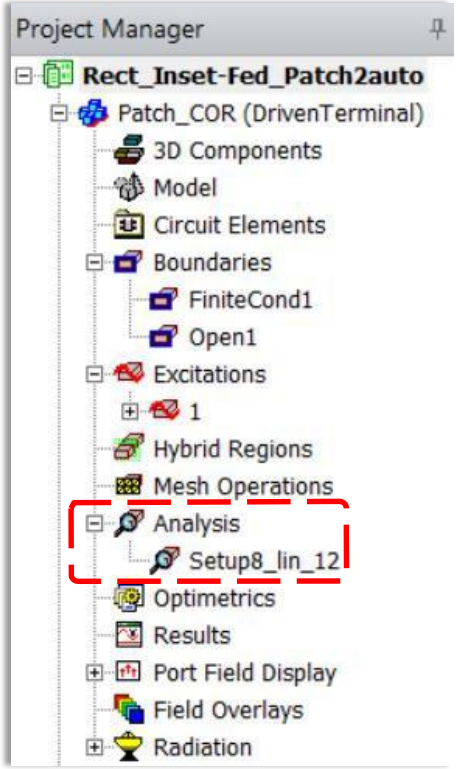



# FEM Solution Setup - Mesh & Sweep

- Simulation Solution Setup, HFSS Adaptive Meshing

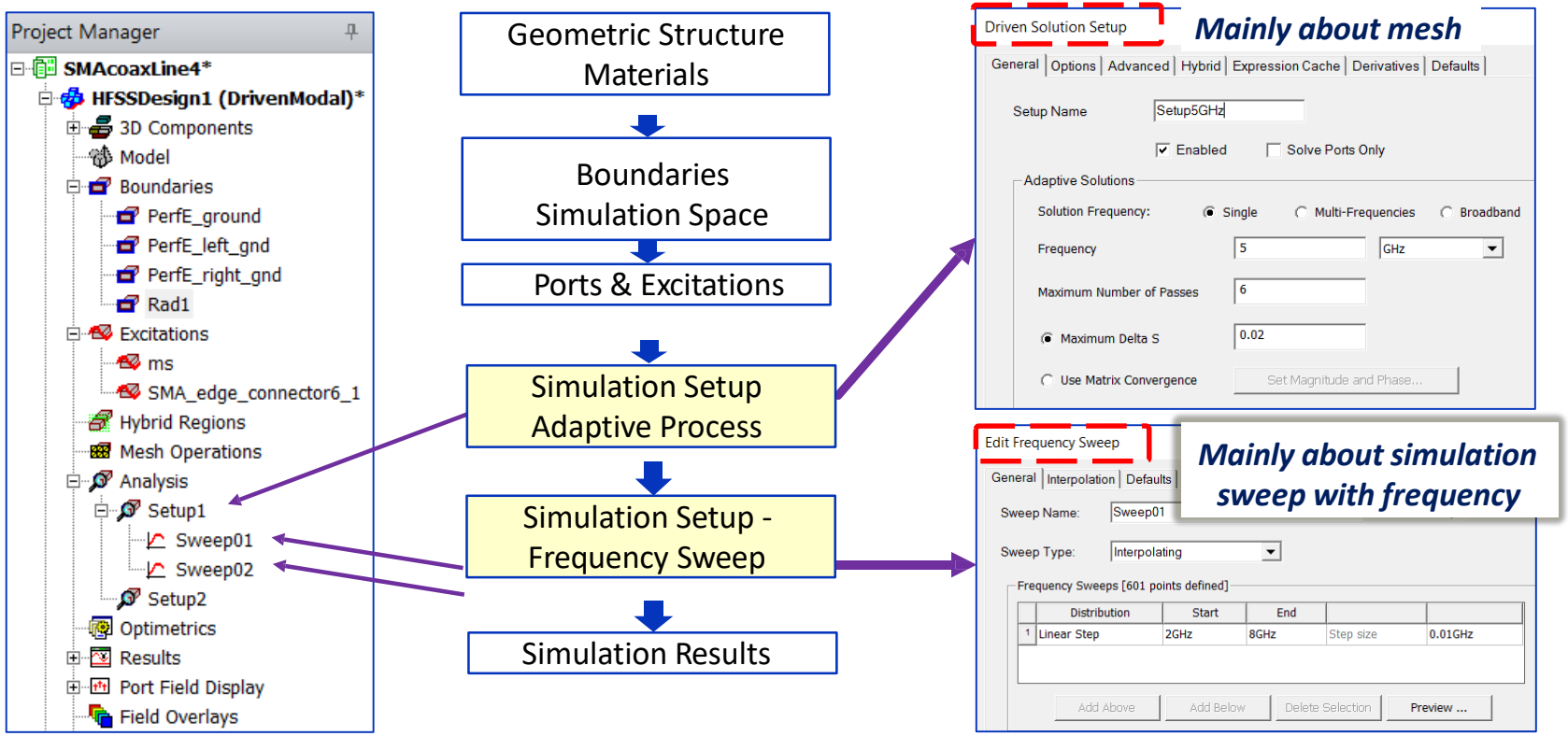
# Solution Setup

- **Simulation Solution Setup Auto**
  - Speed versus Accuracy
  - Frequency Sweep Settings
- **Simulation Solution Setup Advanced**
  - Driven Solution Setup
  - Frequency Sweep
- **HFSS Adaptive Meshing**
  - Adaptive Meshing algorithm flow
  - Frequency, Max # Passes,  and Delta S
  - Specifying Delta S
- **Accuracy and Resolution**
  - Delta S & Maximum Number of Adaptive Passes
  - Sweep - density of data with frequency
  - Solution Frequency




This shows the ***HFSS Solution Setup Auto*** option.

# HFSS Simulation Setup in the Workflow

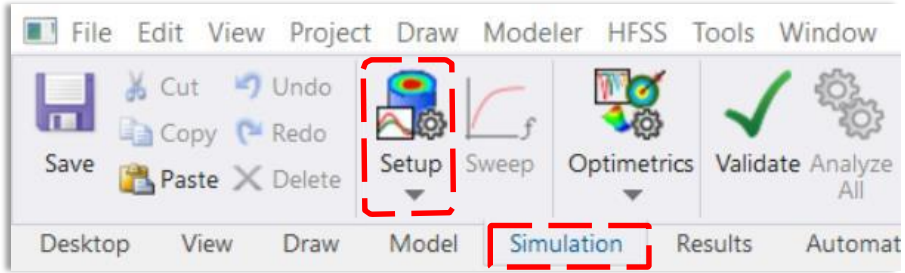


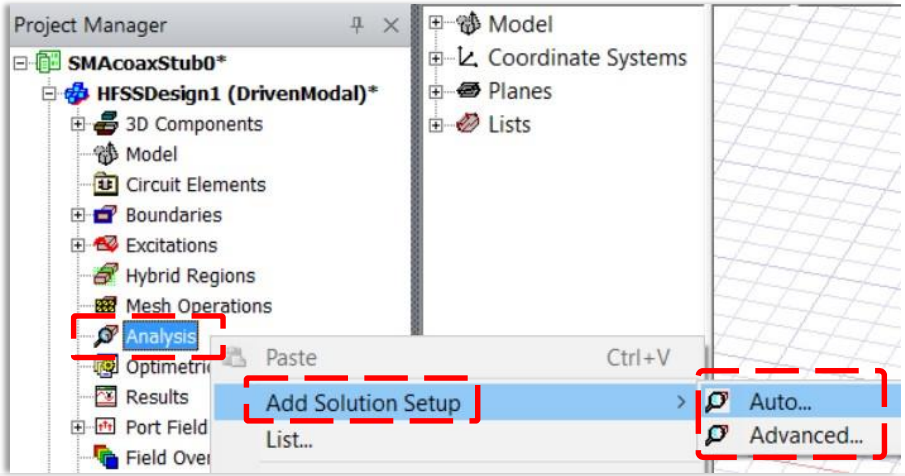

This shows the ***HFSS Solution Setup Advanced*** option.

# Accessing the HFSS Driven Solution Setup

From the ***Ribbon***, ***HFSS Solution Setup*** can be accessed from the Simulation tab.

From the ***Project Manager***, ***HFSS Solution Setup*** can be accessed from ***Analysis > Add Solution Setup***.




The two choices for ***HFSS Solution Setup*** are ***Auto*** and ***Advanced***.

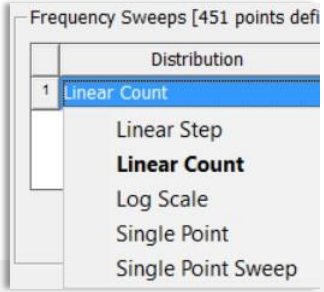
# HFSS Auto Solution Setup

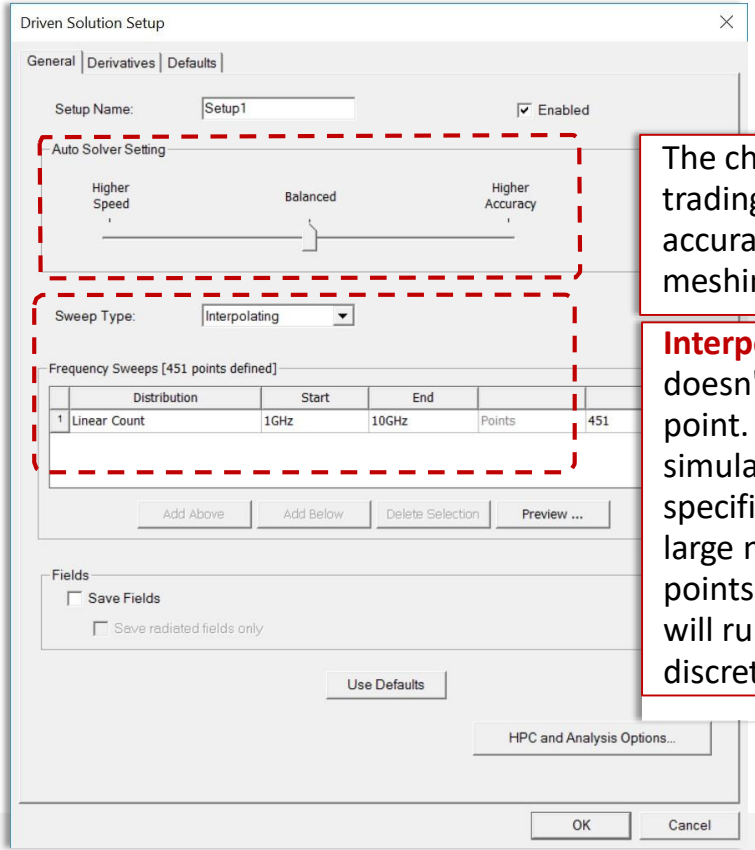
Two aspects of HFSS solution process are the

1. **meshing** … and the
2. **frequency sweep**.

In the ***HFSS Solution Setup Auto*** option, both the meshing and the frequency sweep are specified on the ***General*** tab of the ***Driven Solution Setup*** dialog box.

There are several ***Distributions.***



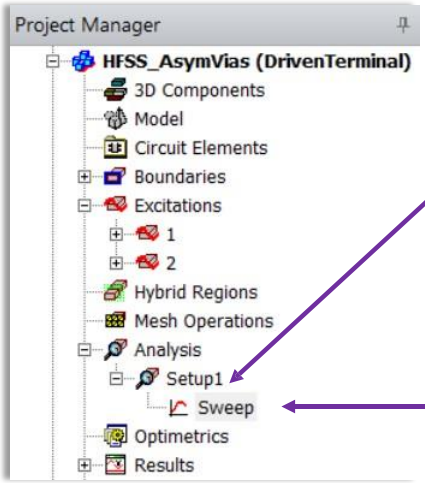



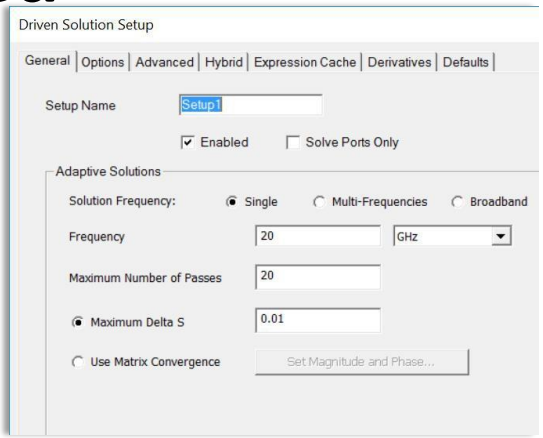

The choice of **slider** setting, trading off speed versus accuracy, specifies the HFSS meshing.

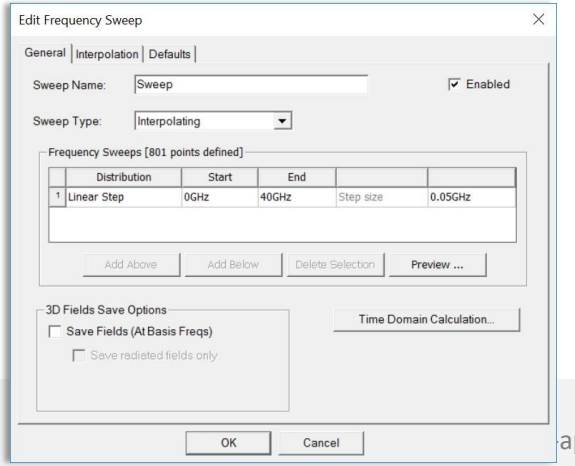
**Interpolating** sweep type doesn't simulate every single point. **Discrete** sweep does simulate every single point specified in the sweep. For large numbers of frequency points, interpolating sweep will run much faster than discrete sweep.

# HFSS Solution Setup Advanced

The ***HFSS Solution Setup Advanced*** option separates the meshing and the frequency sweep into distinct parts.

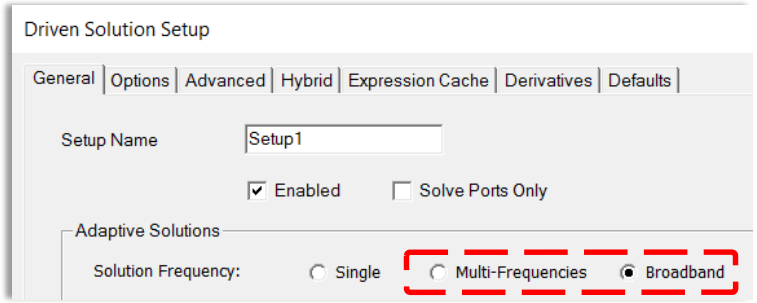


# Broadband and Multi-Frequency

In addition to ***Single*** frequency meshing, HFSS offers ***Broadband*** and ***Multi-Frequencies*** meshing.




## Broadband Frequency

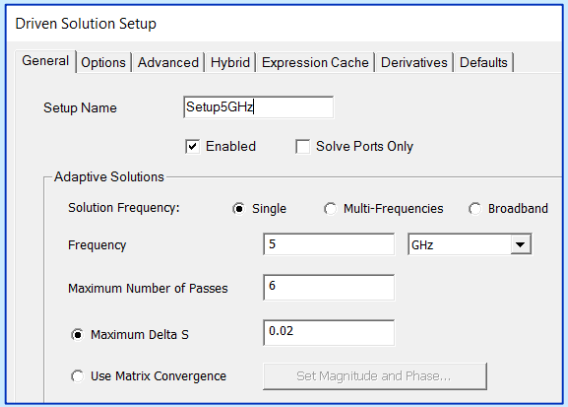
For most problems, specifying the Single solution frequency setup is adequate to obtain accurate results. If you desire increased reliability for broadband devices and more accurate solutions, you can specify the **Broadband** option for the Solution Frequency setup. The Broadband setup enables HFSS to intelligently determine the appropriate frequencies at which to adapt the mesh. The virtue of automatic broadband adaptive meshing lies in eliminating uncertainty in choosing the best frequency for adapting the mesh. You need only specify the highest and lowest frequencies of the range and HFSS determines the frequencies at which to adapt the mesh. The mesh is always adapted for a minimum of three frequencies within the specified frequency range. Adaptive meshing at additional frequencies require enabling the high performance computing (HPC) feature and availability of sufficient computational resources.

From the HFSS Online Help document ***HFSS.pdf***, Chapter 19 ***Specifying Solution Settings*** section titled ***Broadband Frequency***.

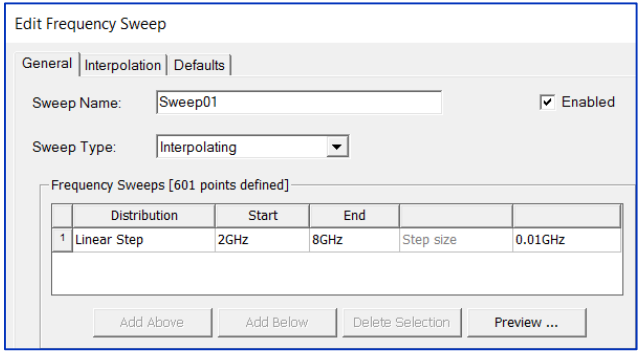
# HFSS Solution Setup

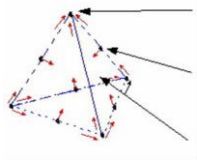
**HFSS adaptive mesh settings are:**

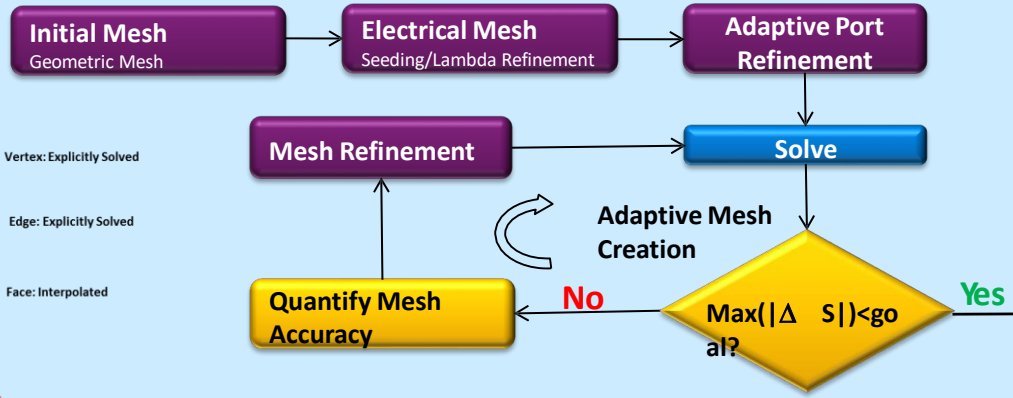
1. **Frequency**
2. **Maximum Number of Passes**
3. **Maximum Delta S.**

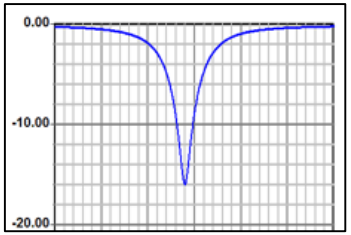



**S-parameter resolution comes from frequency sweep settings.**

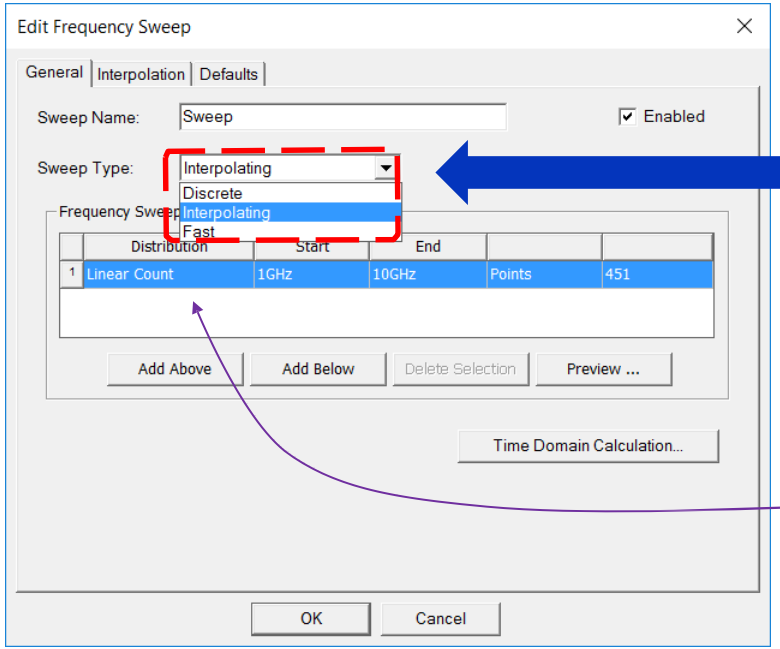


# Edit Frequency Sweep




Three Sweep Types in HFSS

1. ***Interpolating***
   Good for wideband - no fields
2. ***Discrete***
   Generates fields at every frequency. Can also save fields.
3. ***Fast -*** (ALPS) - Use with caution
   Recommended only for the very narrow frequency sweeps when field information is necessary for many frequency points.

**Linear Count**

Total number of frequencies, spread over the whole frequency range

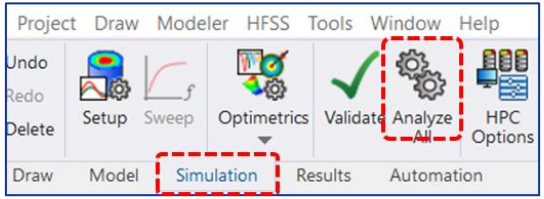
**Linear Step**

Frequency between steps

The document ***An Introduction to HFSS***, Chapter 4 ***HFSS Solution Setup*** includes a section ***Frequency Sweeps*** with more information.

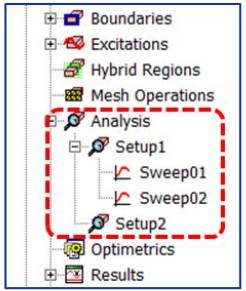
# HFSS FEM Simulation - Analyze

1.




2. ***HFSS > Analyze All***

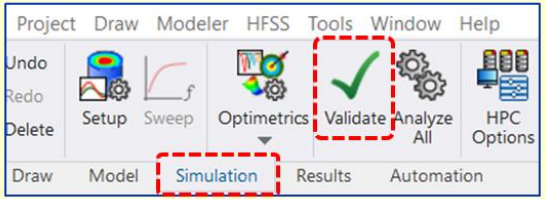
3.

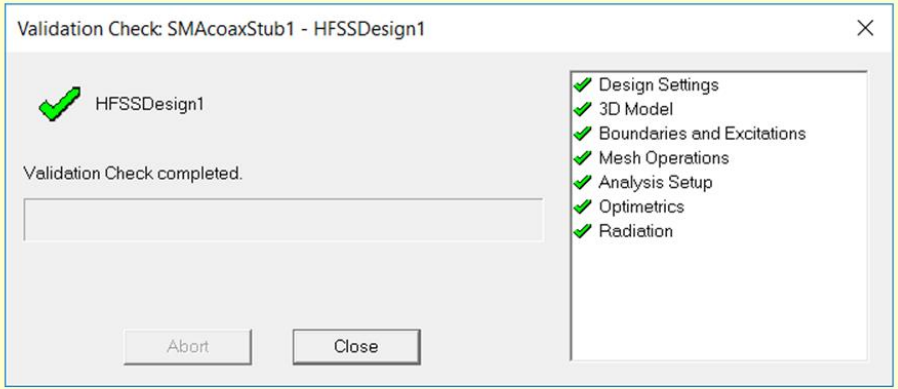



***Right-click on Analysis***

***Three ways to start an HFSS simulation***

**Validate before simulation**



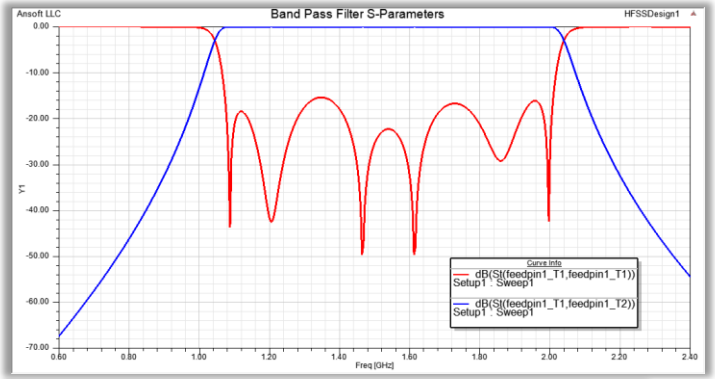

**Validation is an important and useful check before starting an HFSS simulation.**

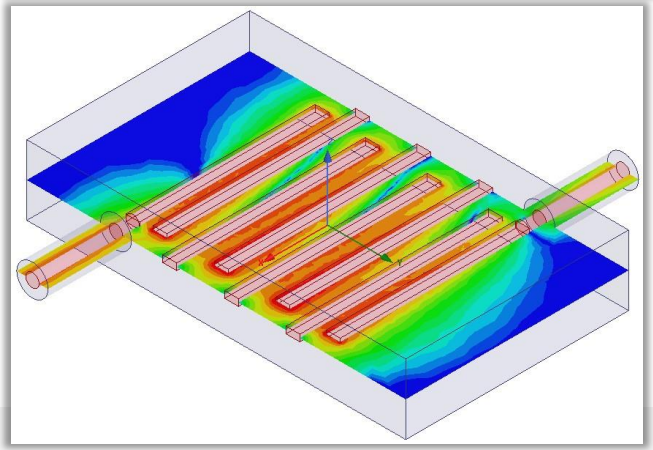
# Post-Processing - S-Parameters and Fields

- Plotting S-Parameters, Post Processing

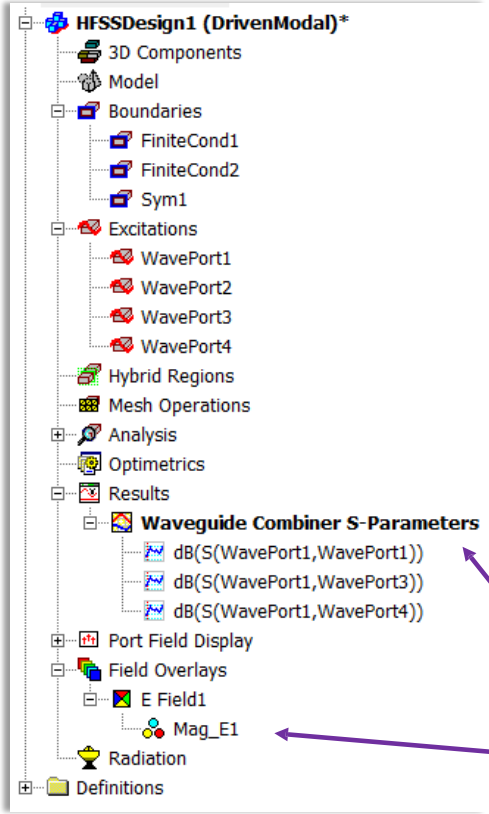
# Post-Processing

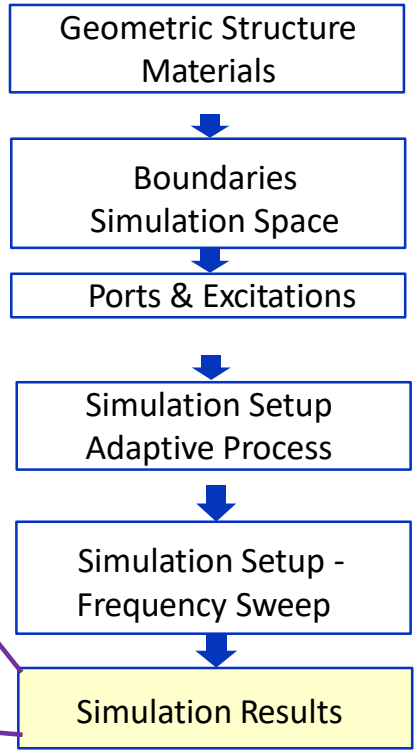
- **Plotting S-Parameters with Rectangular Plots**
  - Setting up and modifying S-Parameter plots
  - Modifying visual appearance of plots
  - Copying rectangular plot Data and Definitions
- **Overview of Available Post Processing**
  - Quantities
  - Formats
  - Report Editor
- **Plotting Fields and Overlays**
  - What simulation is needed for a field
  - Setting up Field Plots
  - Field Plot Overlays on Geometries

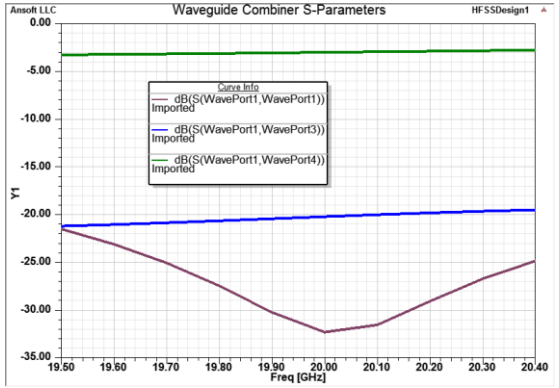


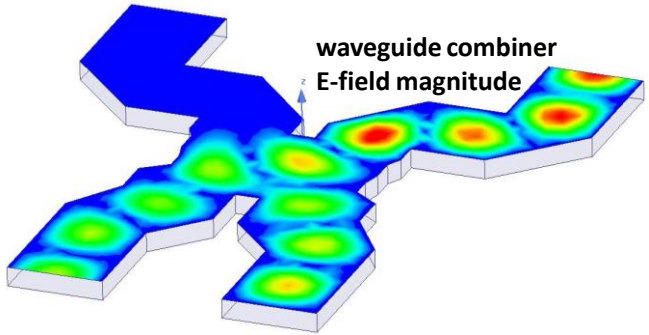



# Simulation Results and Field in Workflow

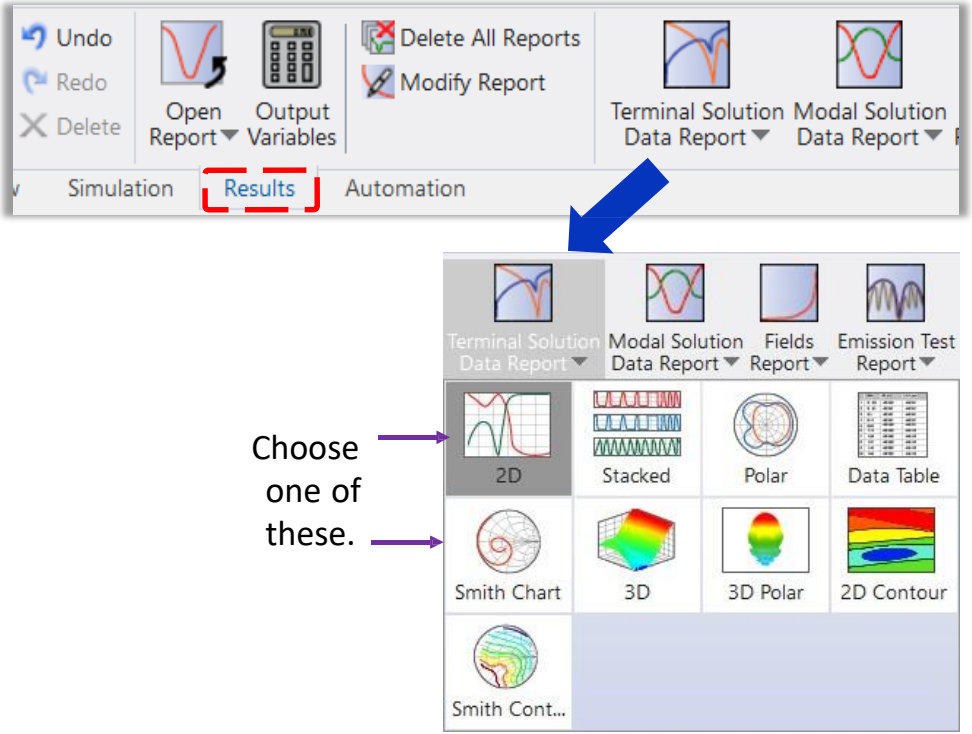


# S-Parameter Plot Setup




**…from the Ribbon**

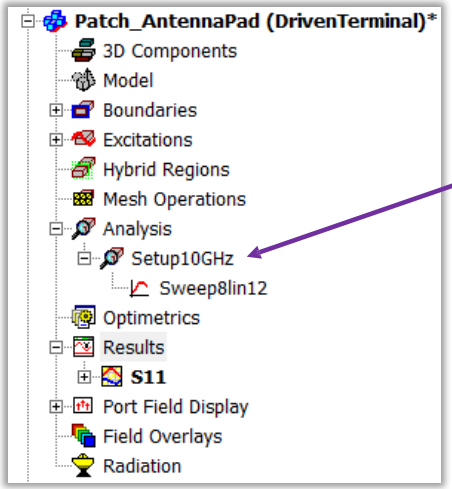
- In the Ribbon, highlight Results.
- Click on ____***Solution Data Report***.
- Choose ***2D*** or ***Smith Chart***.

The blank ____gets filled in with a ***Solution Type***, such as ***modal***, ***terminal***, or ***eigenmode***.

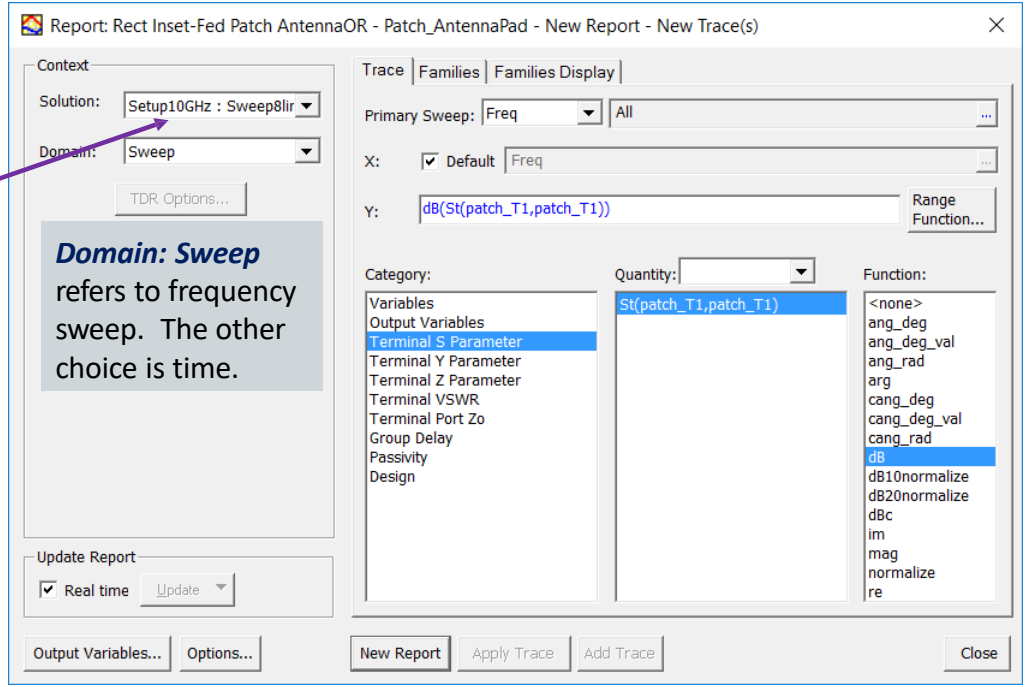
Both the ***2D*** and the ***Smith Chart*** are more general than just S-parameters. For example, impedance can be graphed in either.

Setting up S-parameter plots is also available from: ***HFSS > Results > Create ____Solution Data Report.***

# S-Parameter Rectangular Report




This was accessed via:
***Terminal Solution Data Report > Rectangular Plot.***

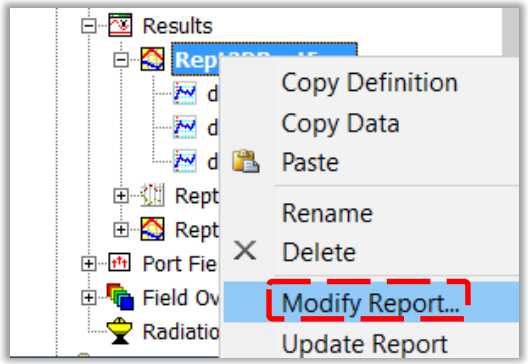



For information on the ***Families*** tab, see ***HFSS.pdf***, Chapter 22, section 56 ***Using Families Tab for Reports***.

# Adding Traces to S-Parameter Report (Plot)

To modify an existing Report:

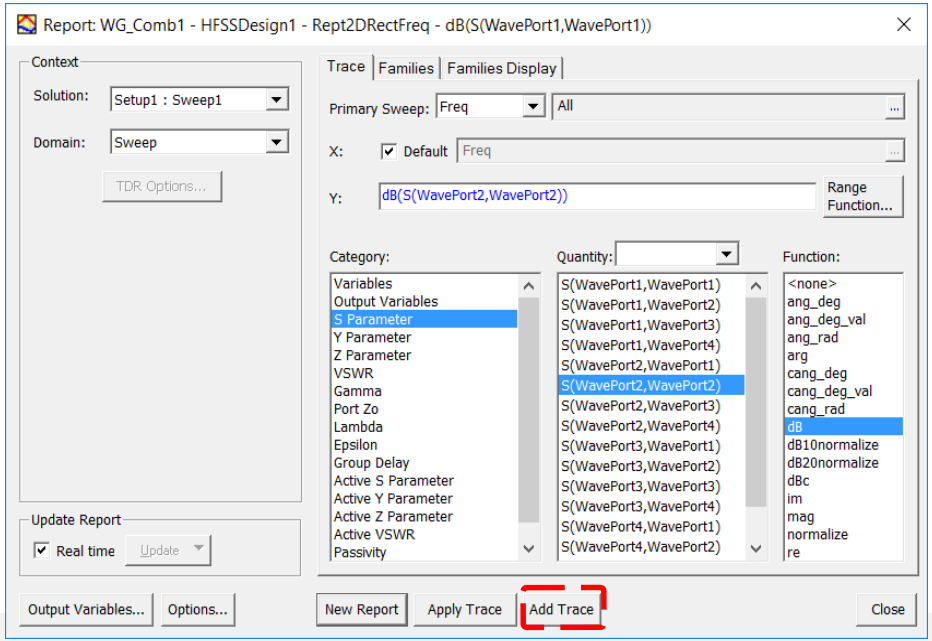
1. In the ***Project Man***ager, select the report
2. Right-click and select ***Modify Report*** …which brings up the dialog box.



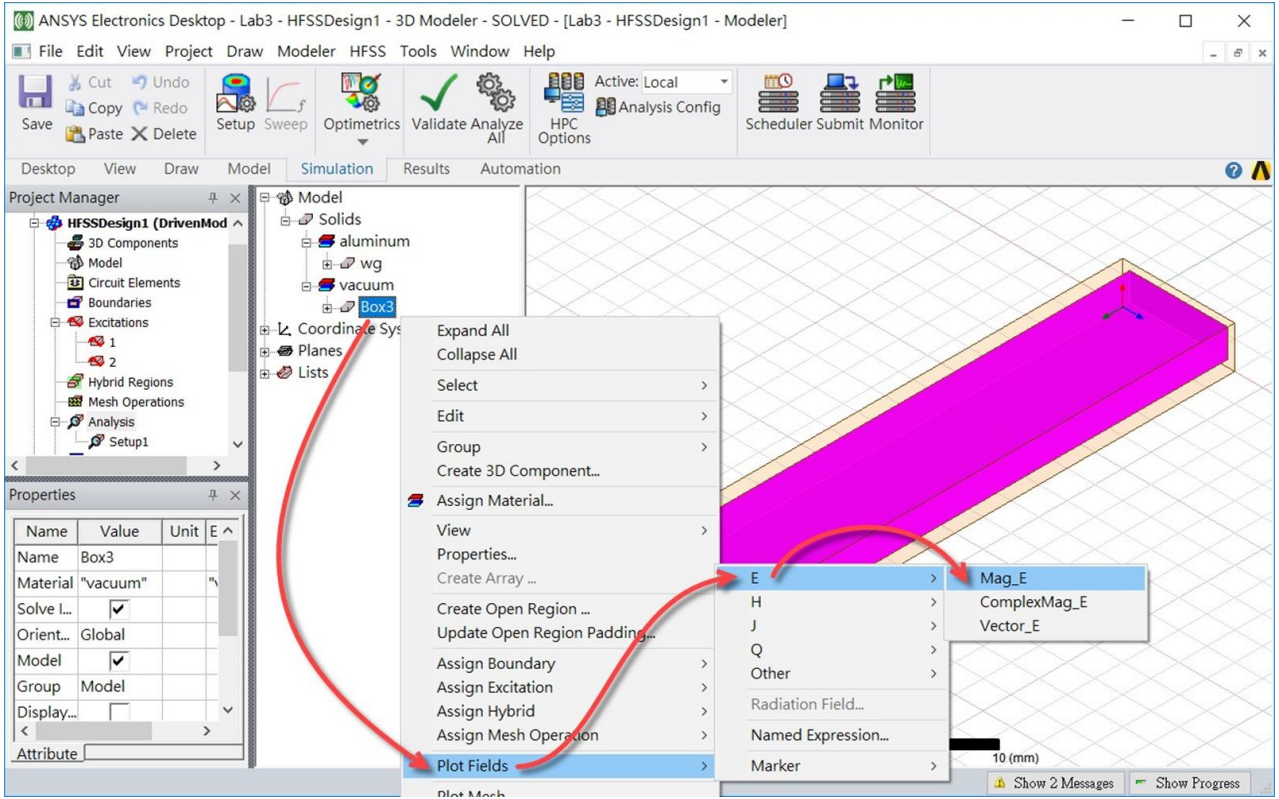

For more information see ***HFSS.pdf***, Chapter 22 ***Modifying Reports.***

To add a trace:

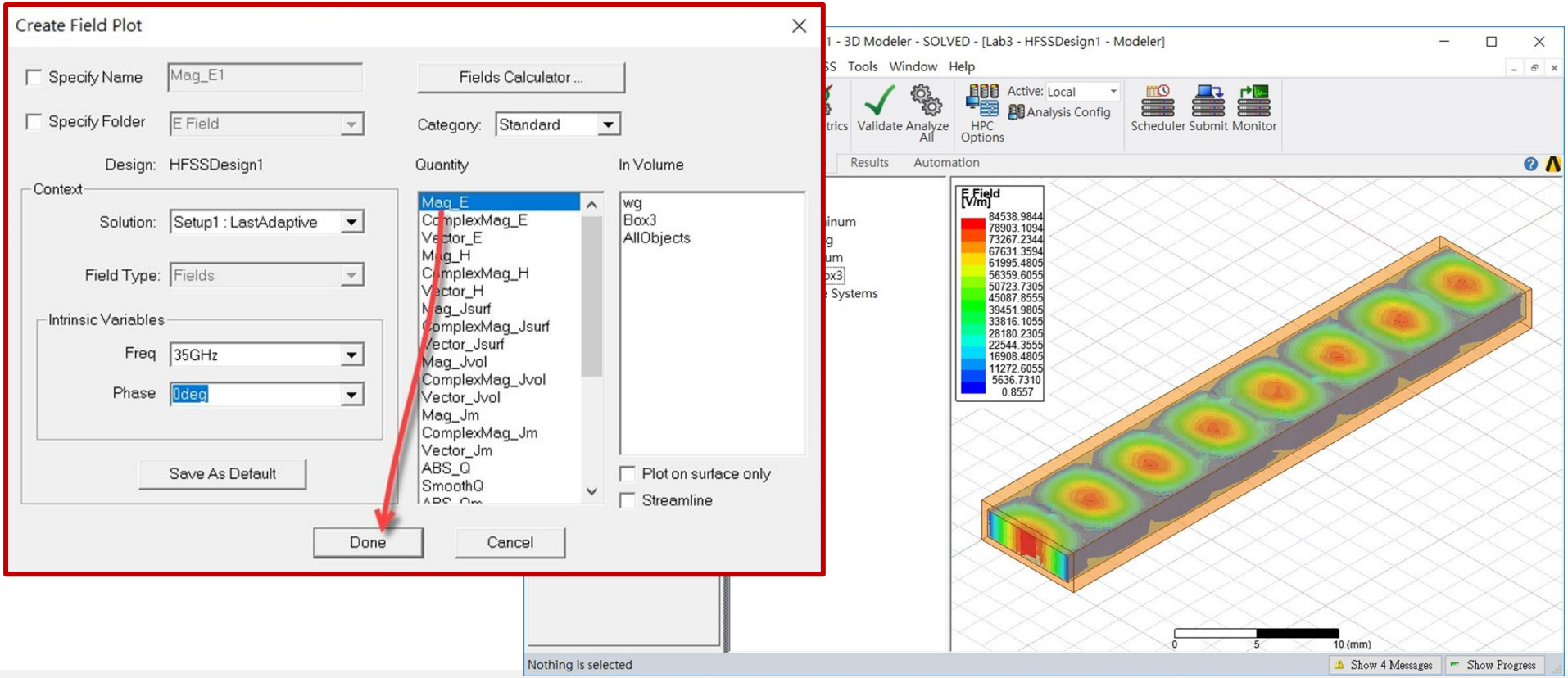
1. Select the desired trace (e.g. in blue )
2. Click on ***Add Trace*** (bottom middle)
3. Click on Close



# Plot E-Field


ANSYS Electronics Desktop - Lab3 - HFSSDesign1 - 3D Modeler - SOLVED - [Lab3 - HFSSDesign1 - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save Cut Copy Paste Undo Redo Delete Setup Sweep Optimetrics Validate Analyze All HPC Options Active: Local Analysis Config Scheduler Submit Monitor
Desktop View Draw Model Simulation Results Automation
Project Manager
HFSSDesign1 (DrivenMod
3D Components
Model
Circuit Elements
Boundaries
Excitations
1
2
Hybrid Regions
Mesh Operations
Analysis
Setup1
Properties
Name Value Unit
Name Box3
Material "vacuum"
Solve I...
Orient... Global
Model
Group Model
Display...
Attribute
Model
Solids
aluminum
wg
vacuum
Box3
Coordinate Sys
Planes
Lists
Expand All
Collapse All
Select
Edit
Group
Create 3D Component...
Assign Material...
View
Properties...
Create Array ...
Create Open Region ...
Update Open Region Padding...
Assign Boundary
Assign Excitation
Assign Hybrid
Assign Mesh Operation
Plot Fields
E
H
J
Q
Other
Radiation Field...
Named Expression...
Marker
Mag_E
ComplexMag_E
Vector_E
10 (mm)
Show 2 Messages
Show Progress

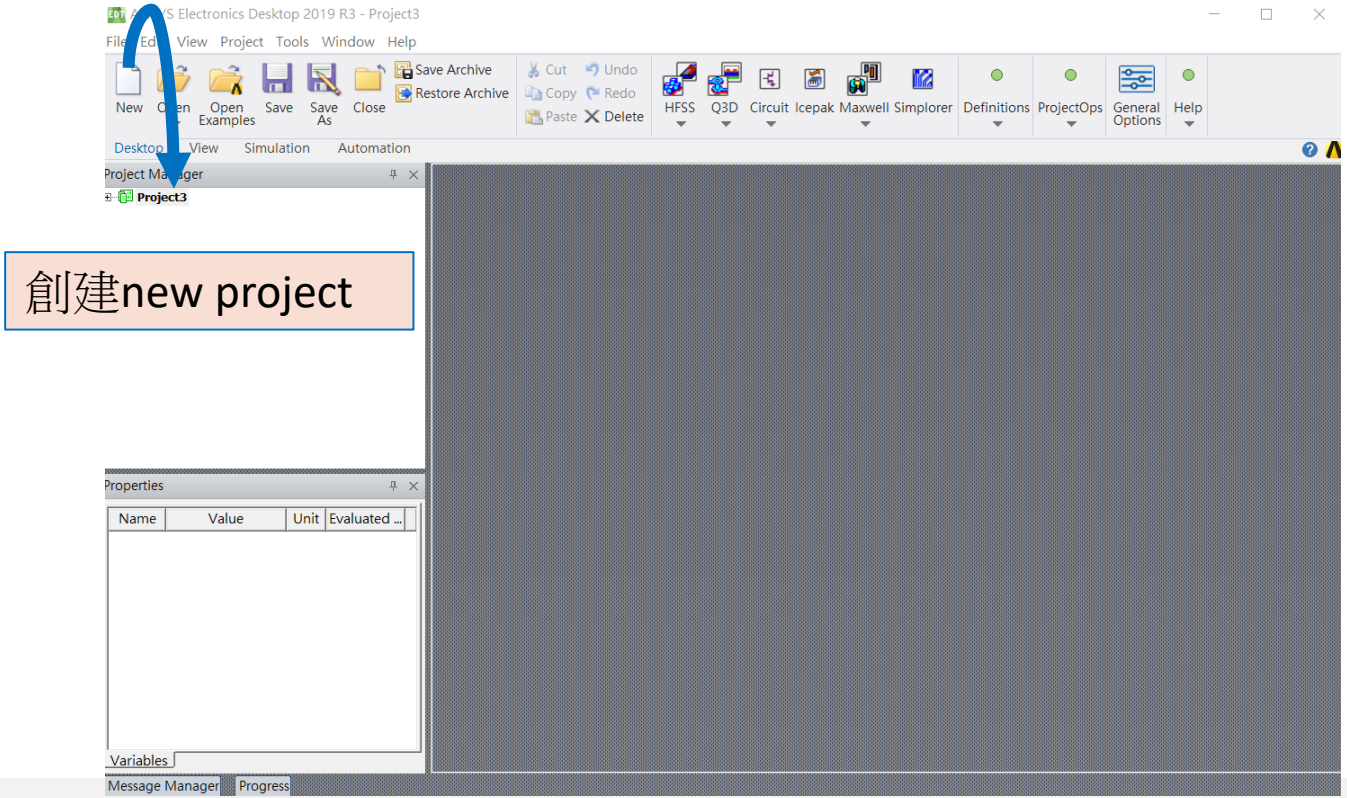
# EM Propagation in Waveguide



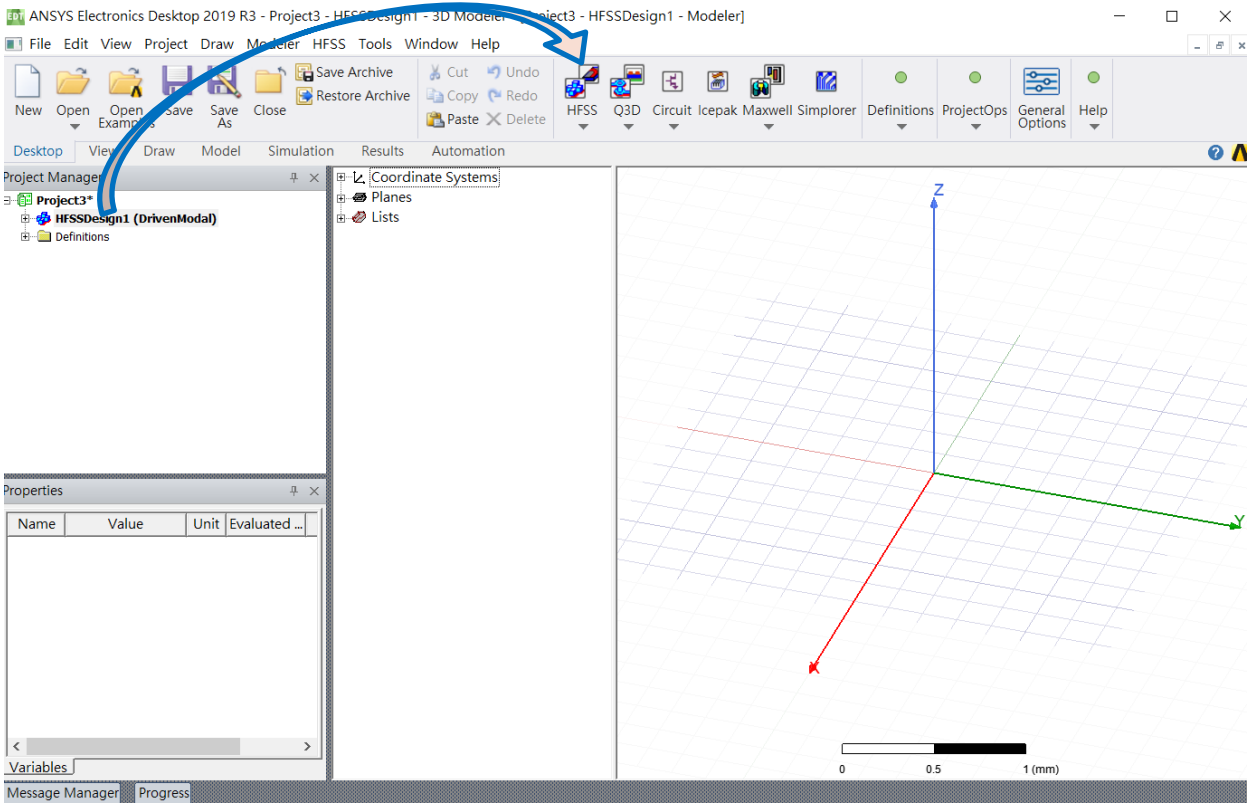
# Project Production 1

- A patch antenna simulation by HFSS from create new project to Post-Processing

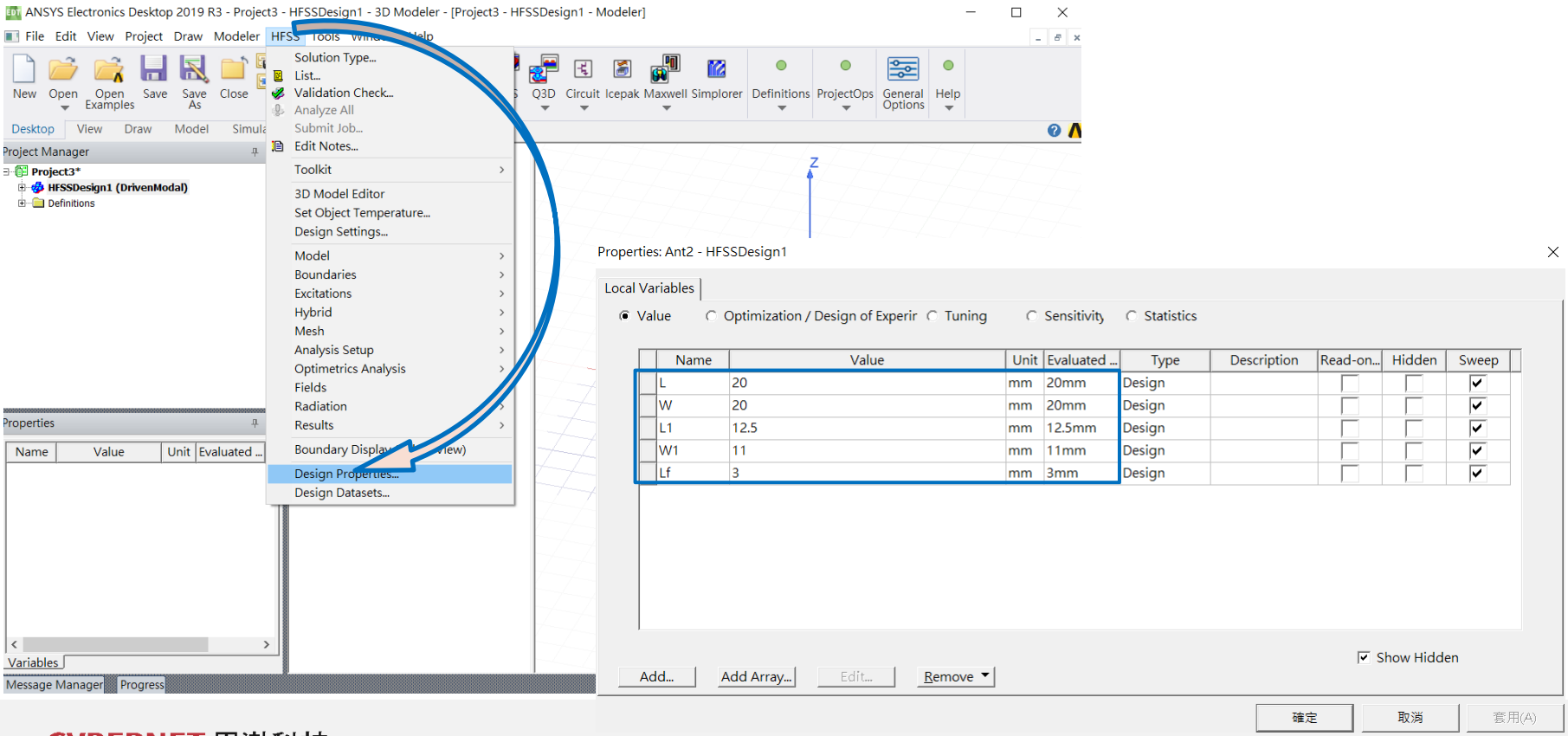
# New Project


Electronics Desktop 2019 R3 - Project3
File Edit View Project Tools Window Help
New
Open
Open Examples
Save
Save As
Close
Save Archive
Restore Archive
Cut
Copy
Paste
Undo
Redo
Delete
HFSS
Q3D
Circuit
Icepak
Maxwell
Simplorer
Definitions
ProjectOps
General Options
Help
Desktop
View
Simulation
Automation
Project Manager
Project3
創建new project
Properties
Name
Value
Unit
Evaluated ...
Variables
Message Manager
Progress

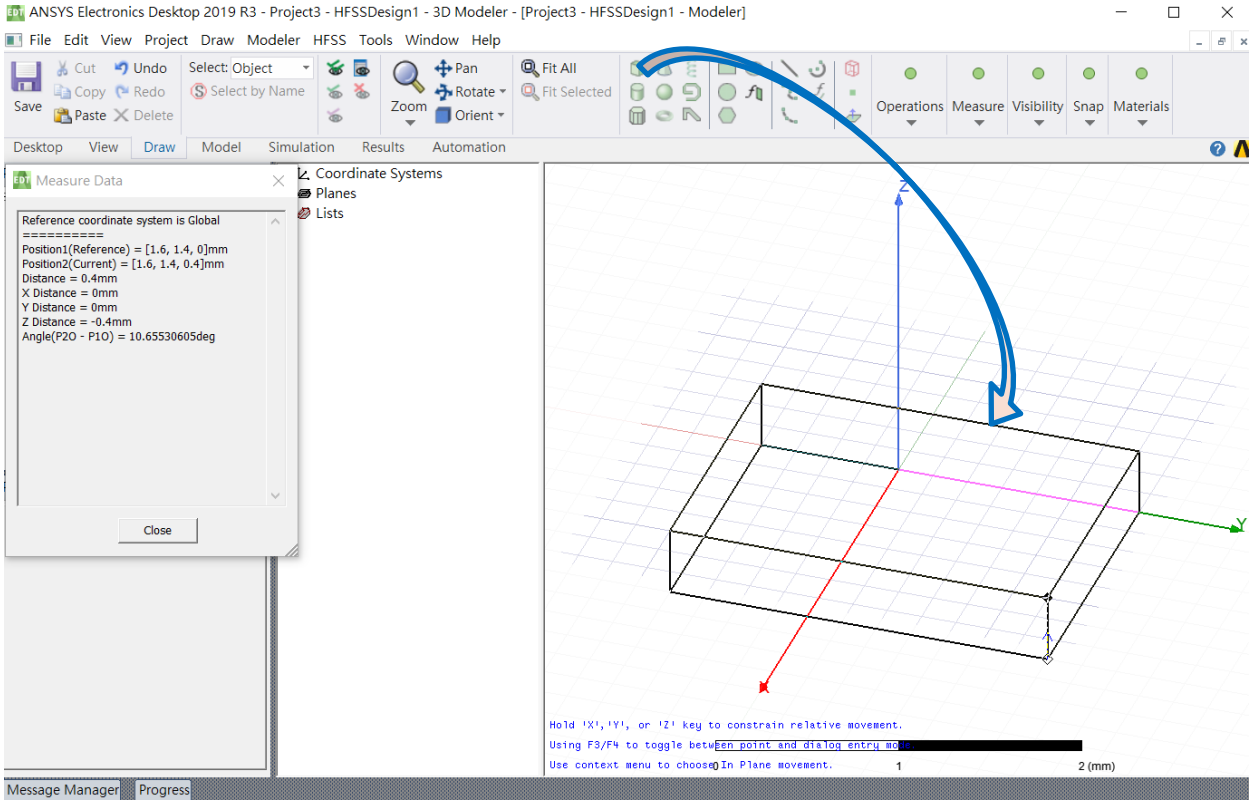
# New HFSS Design



# Design Properties

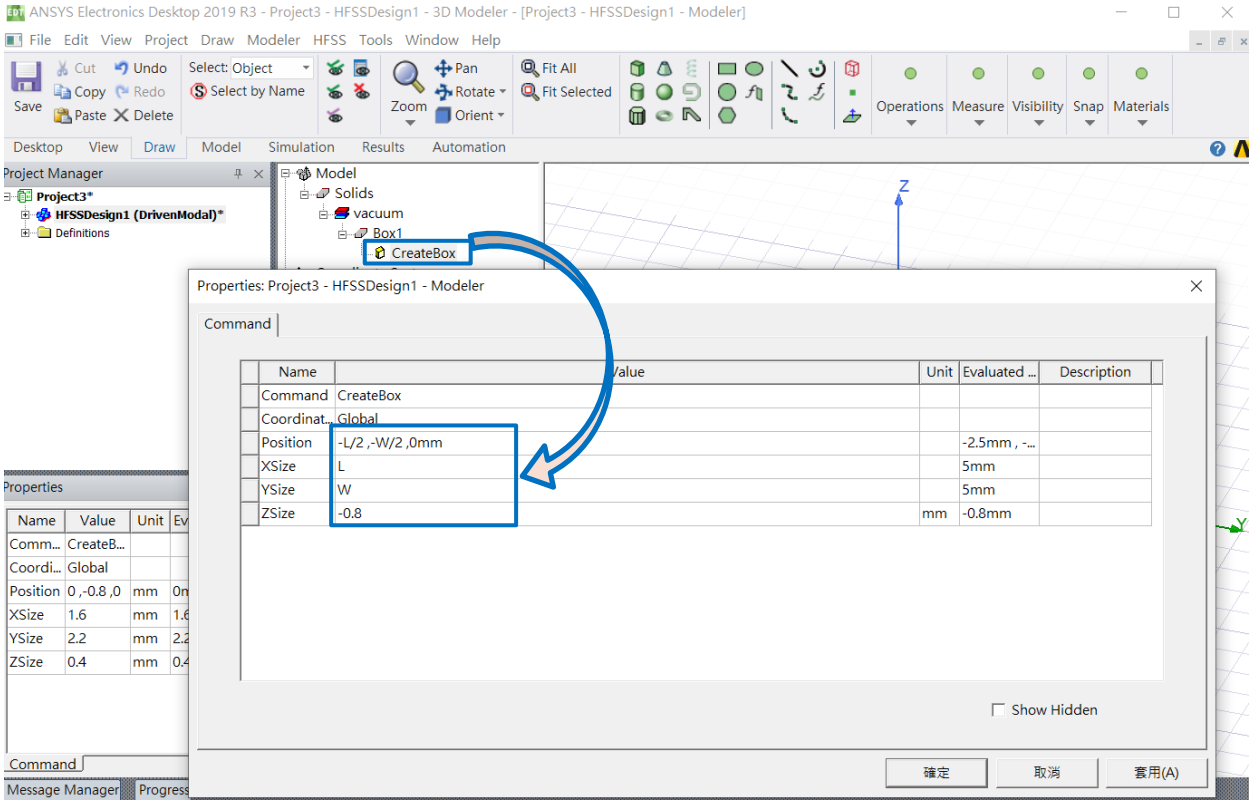


# Draw Box

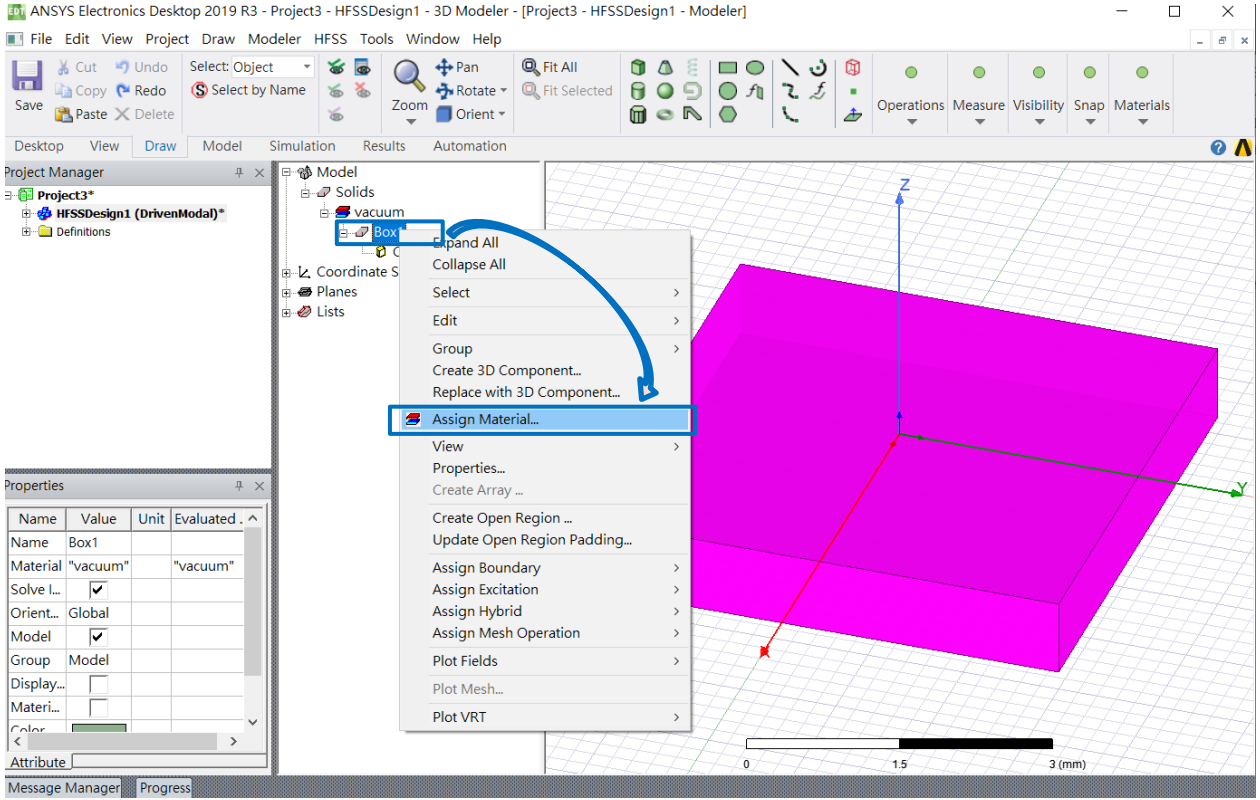



任意拖拉即可，事後會進行參數化

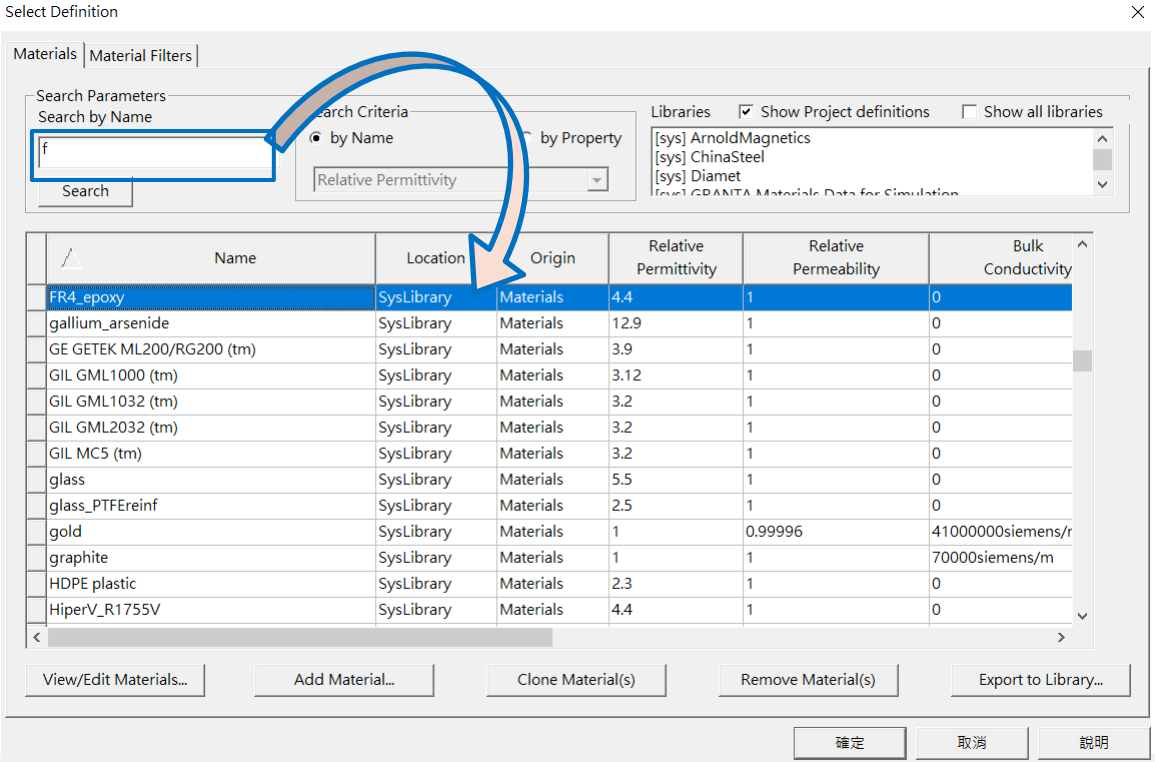
# Edit Properties



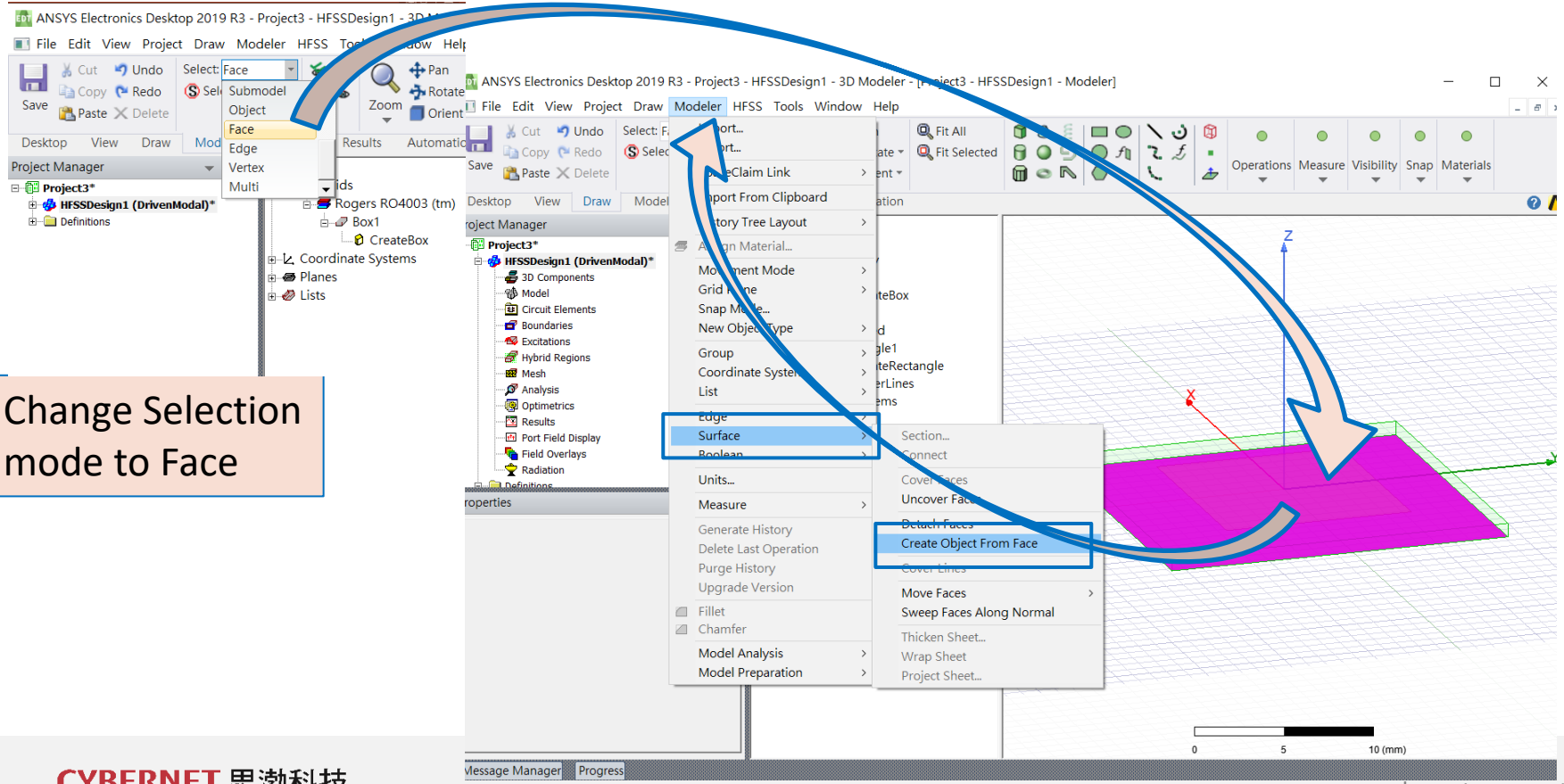
# Assign Material



# Select Material

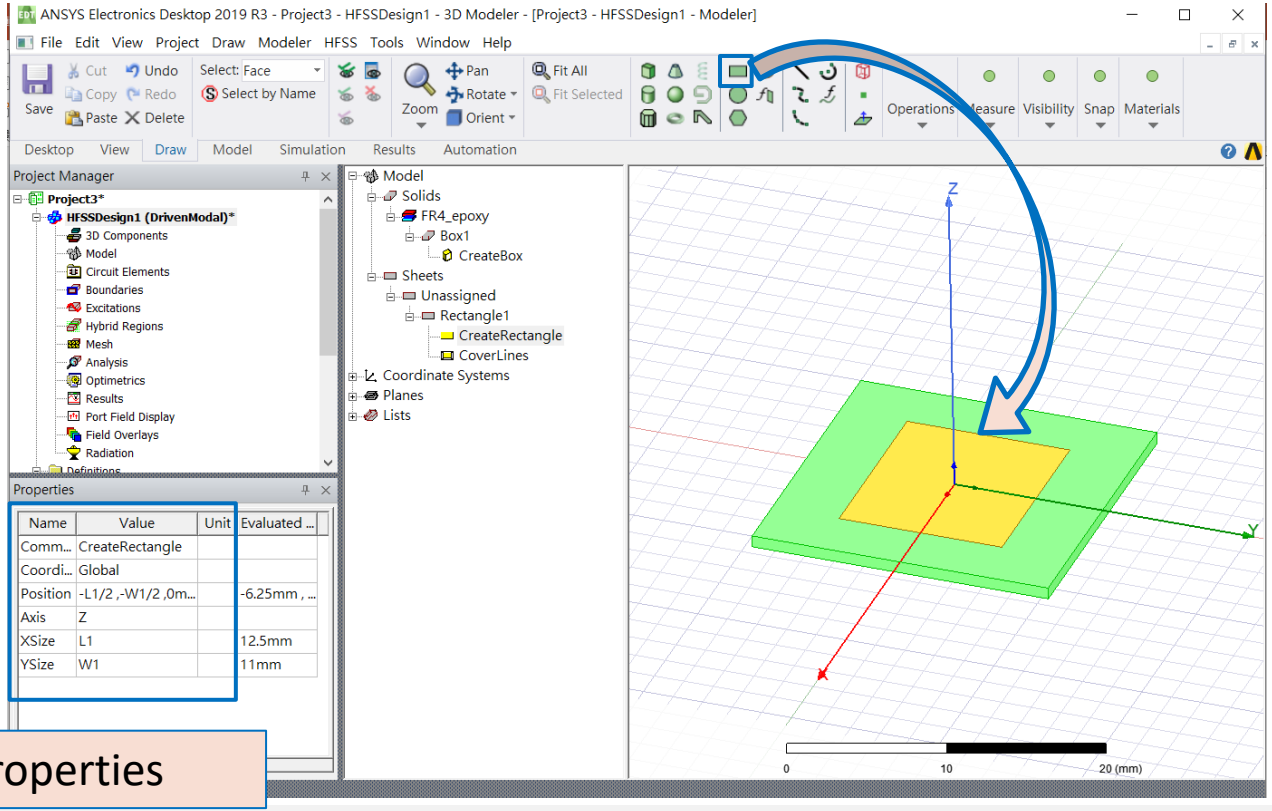


# Draw Ground Plane



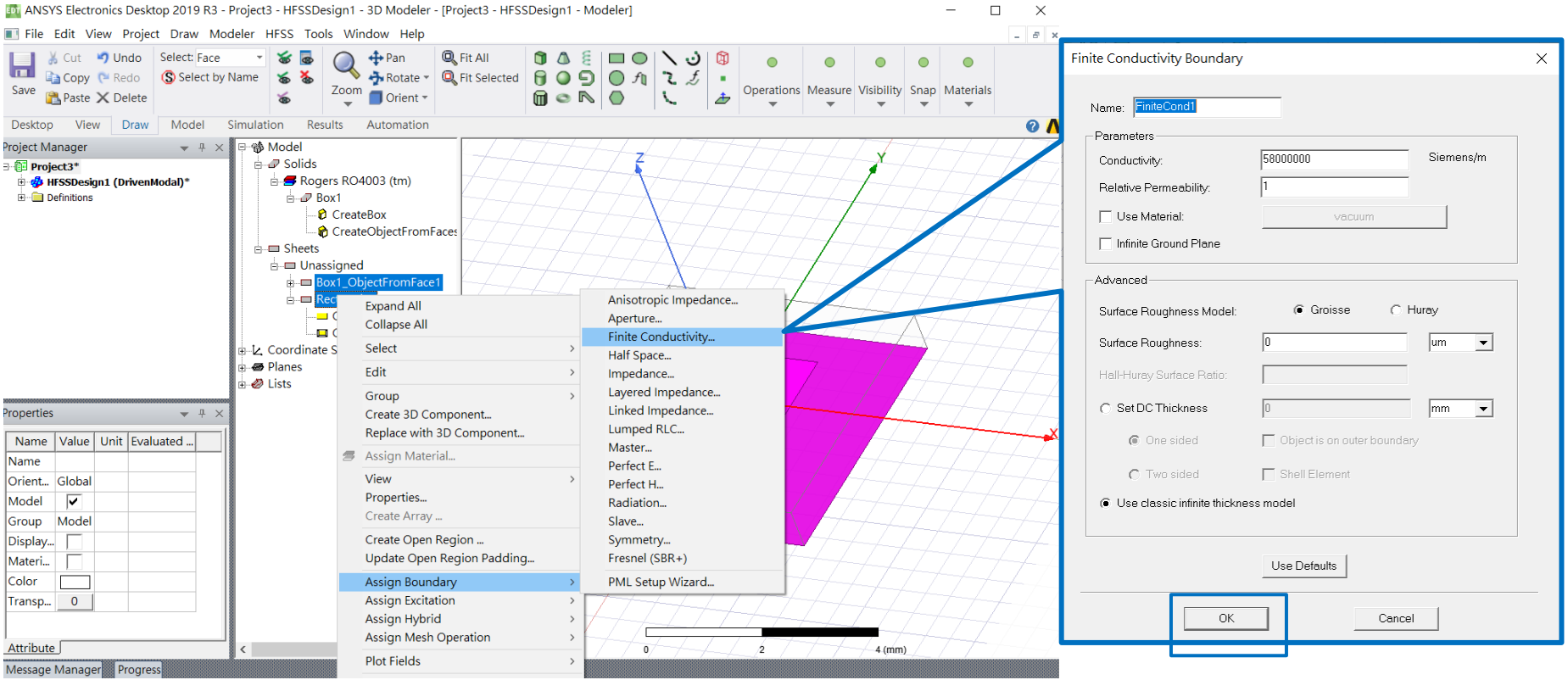

Change Selection mode to Face

# Draw Patch

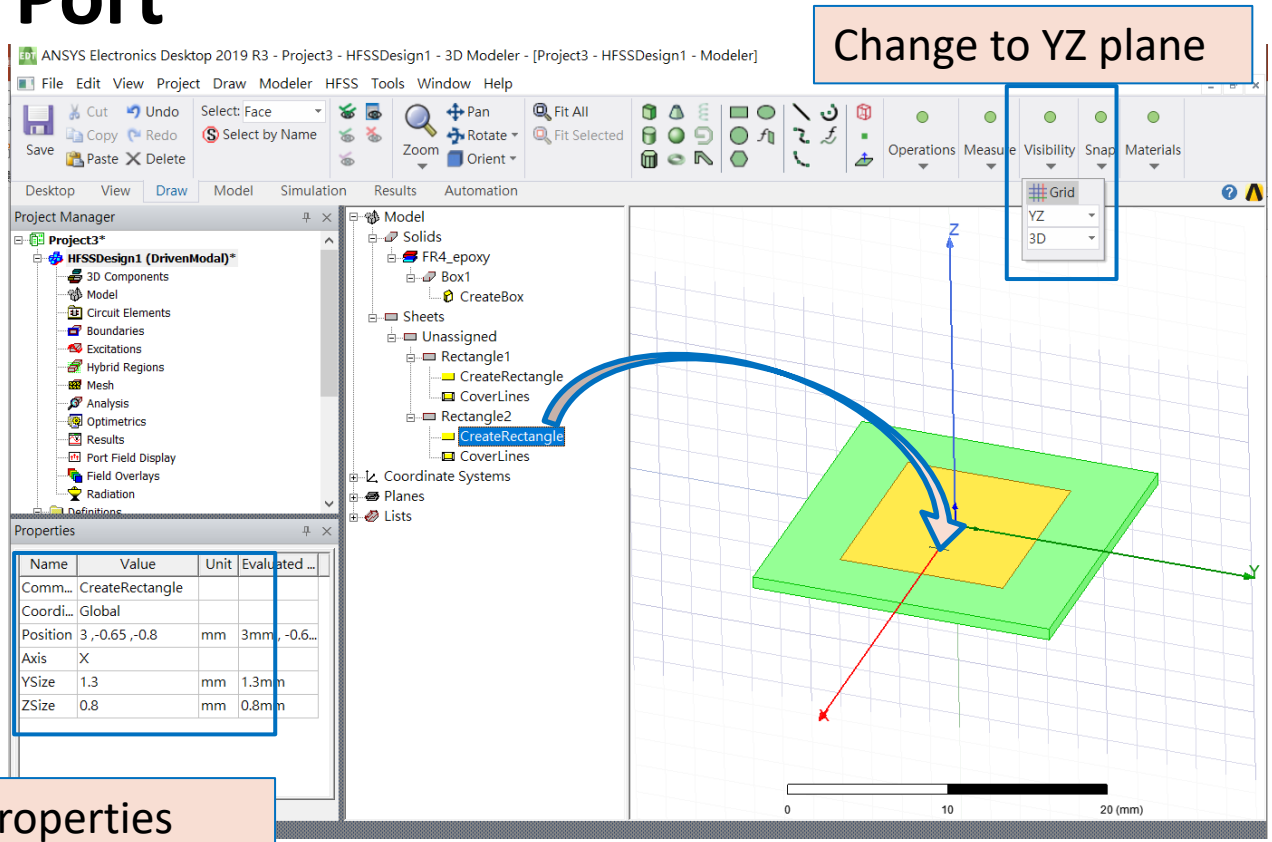



Edit Properties

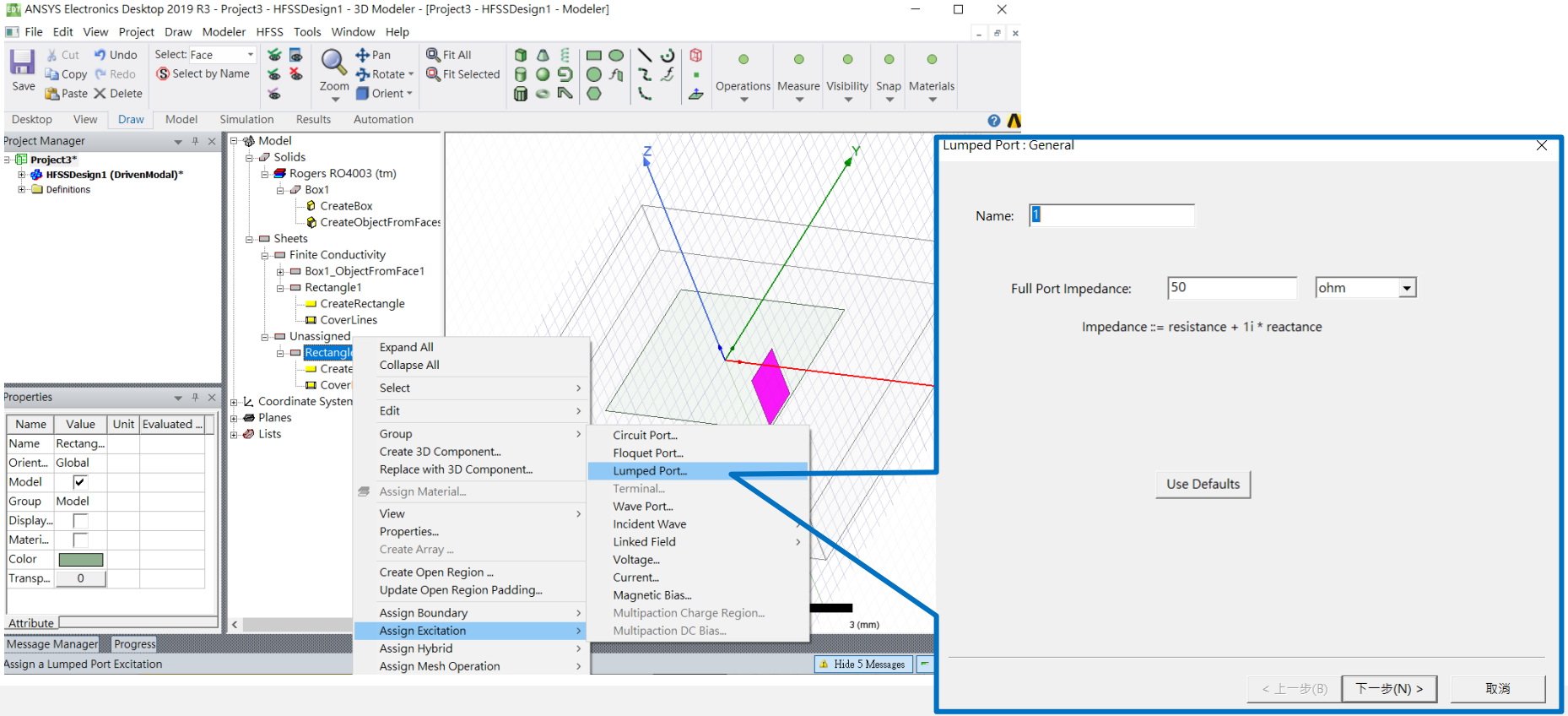
# Assign Boundary



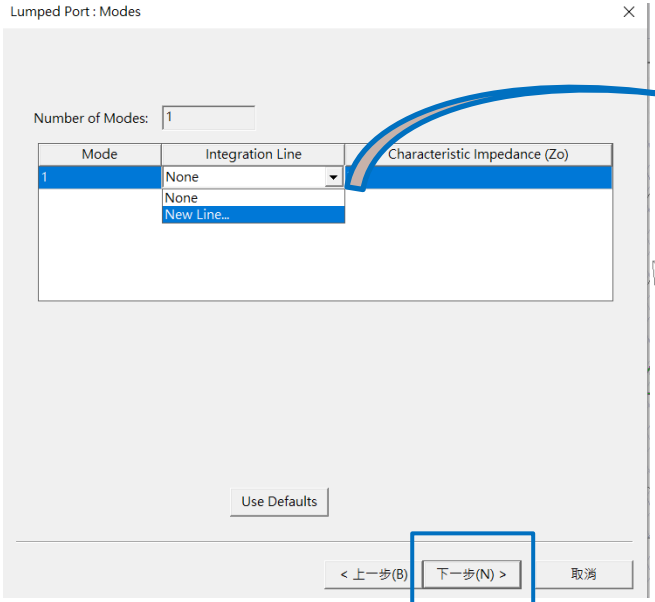
# Draw Port

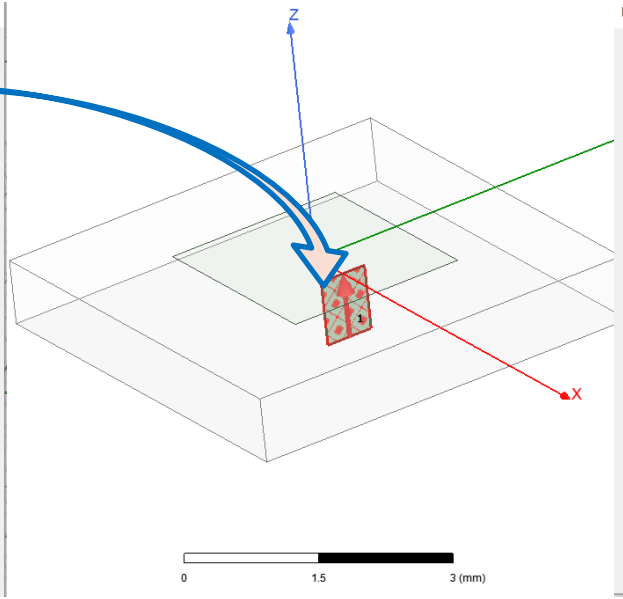

Change to YZ plane
ANSYS Electronics Desktop 2019 R3 - Project3 - HFSSDesign1 - 3D Modeler - [Project3 - HFSSDesign1 - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save Cut Copy Paste Undo Redo Delete
Select: Face
Select by Name
Zoom Pan Rotate Orient Fit All Fit Selected
Operations Measure Visibility Snap Materials
Desktop View Draw Model Simulation Results Automation
Grid YZ 3D
Project Manager
Project3*
HFSSDesign1 (DrivenModal)*
3D Components
Model
Circuit Elements
Boundaries
Excitations
Hybrid Regions
Mesh
Analysis
Optimetrics
Results
Port Field Display
Field Overlays
Radiation
Model
Solids
FR4_epoxy
Box1
CreateBox
Sheets
Unassigned
Rectangle1
CreateRectangle
CoverLines
Rectangle2
CreateRectangle
CoverLines
Coordinate Systems
Planes
Lists
Properties
Name Value Unit Evaluated ...
Comm... CreateRectangle
Coordi... Global
Position 3 ,-0.65 ,-0.8 mm 3mm , -0.6...
Axis X
YSize 1.3 mm 1.3mm
ZSize 0.8 mm 0.8mm
Z
Y
0 10 20 (mm)
Edit Properties

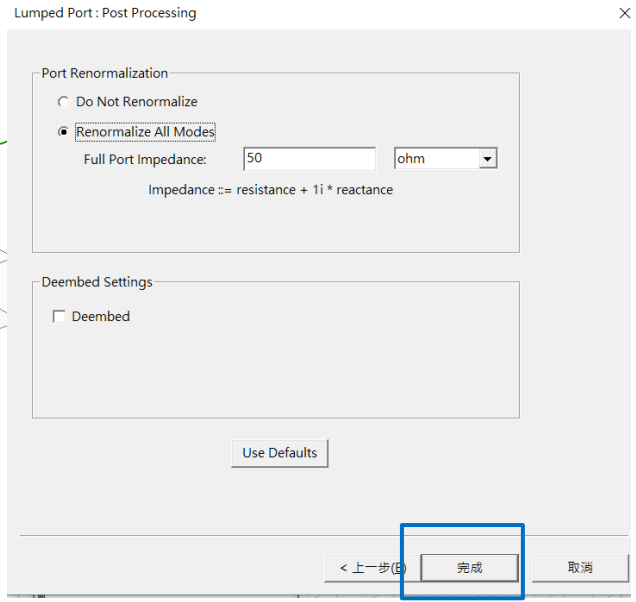
# Assign Port 1/2



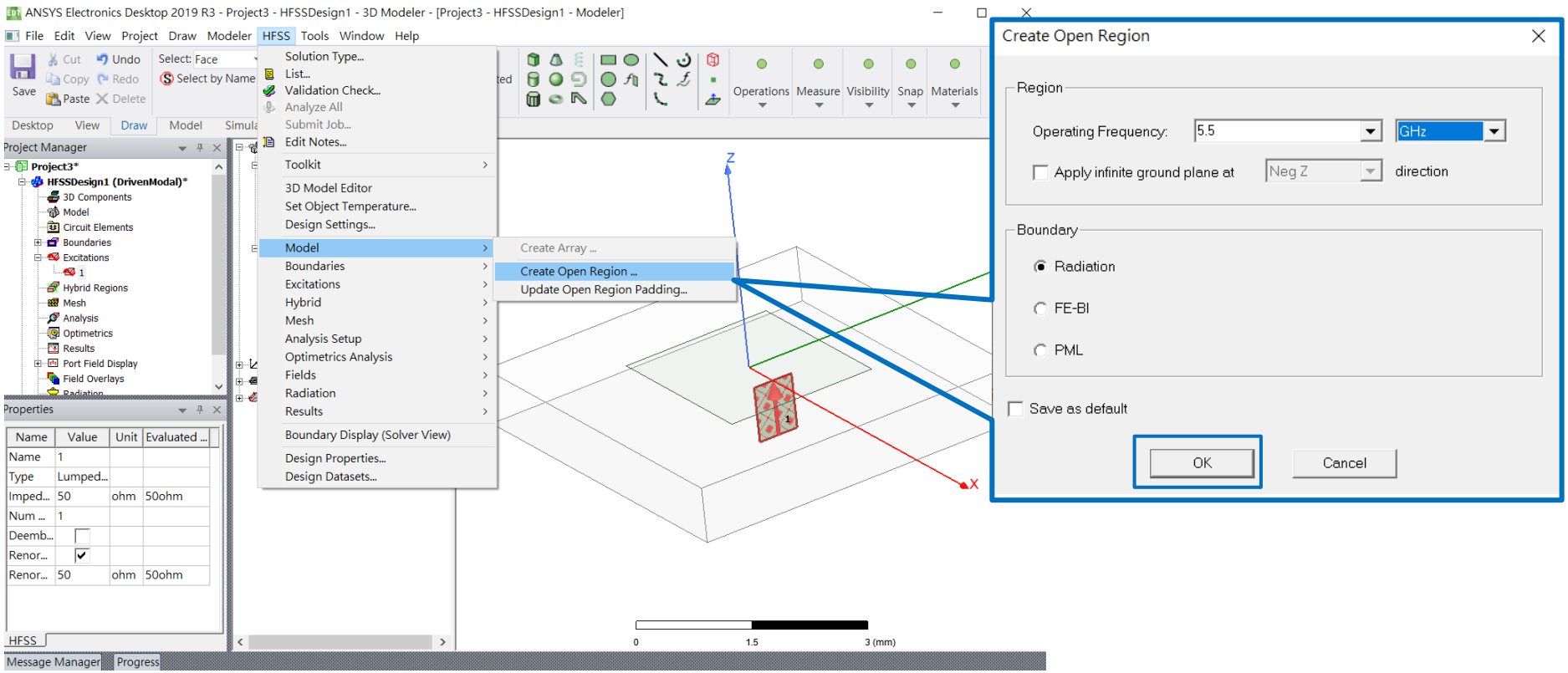
# Assign Port 2/2



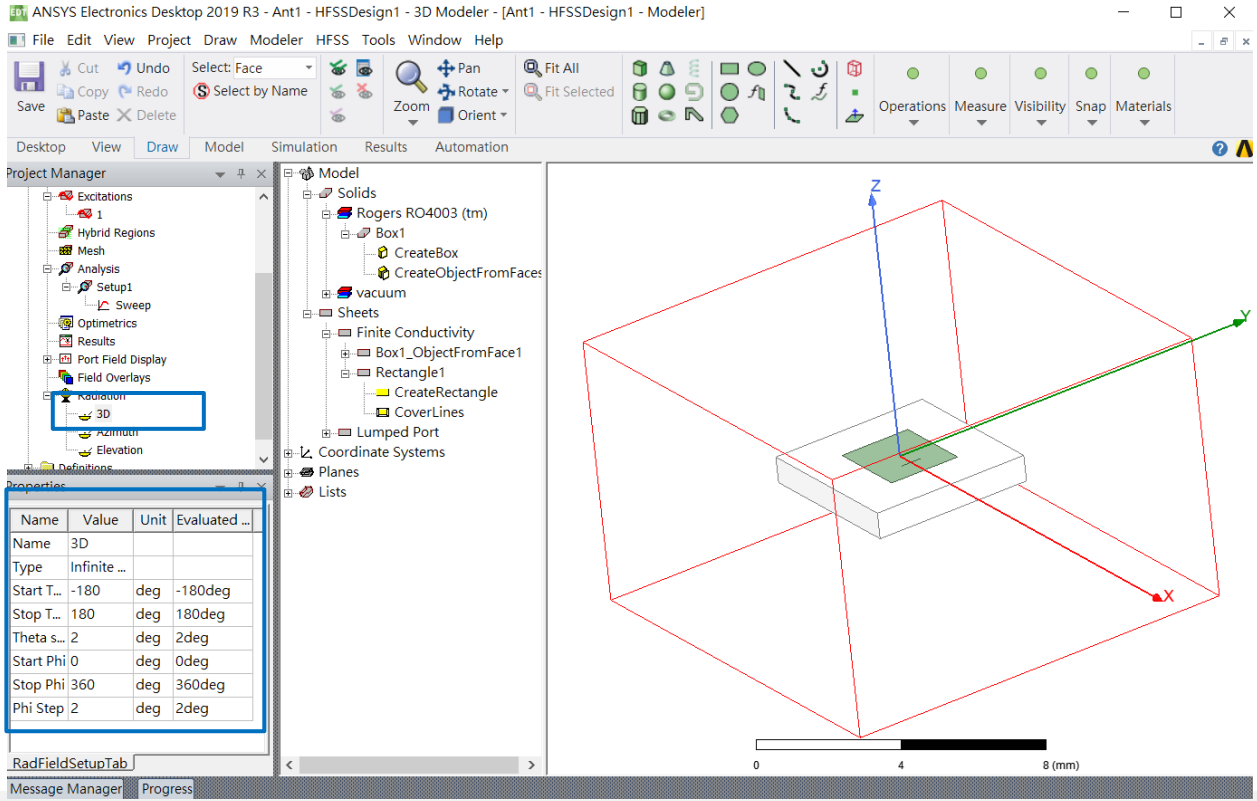



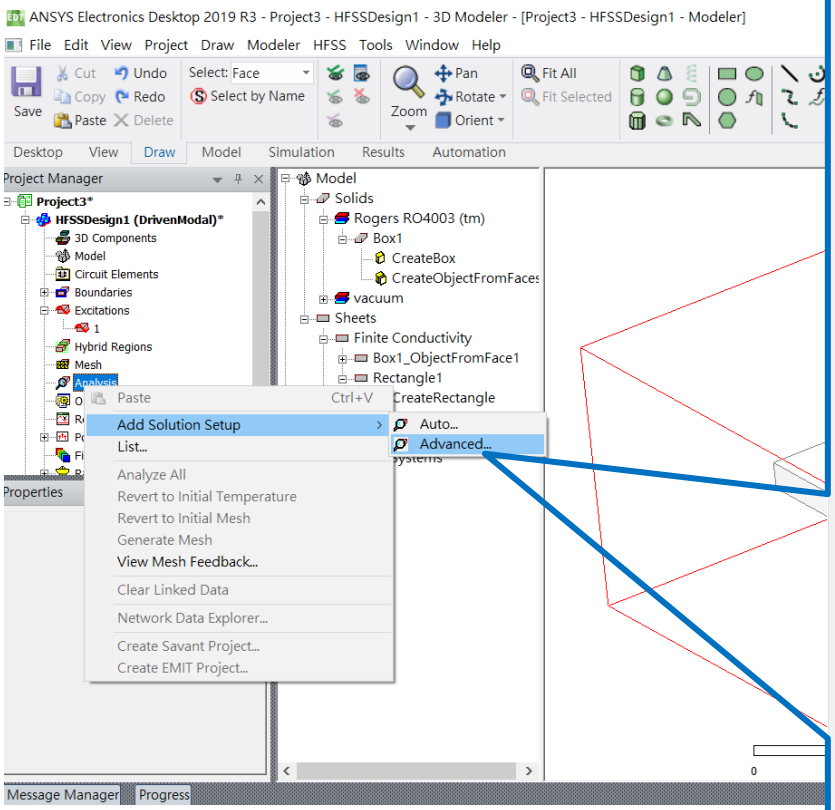
# Assign Radiation Boundary

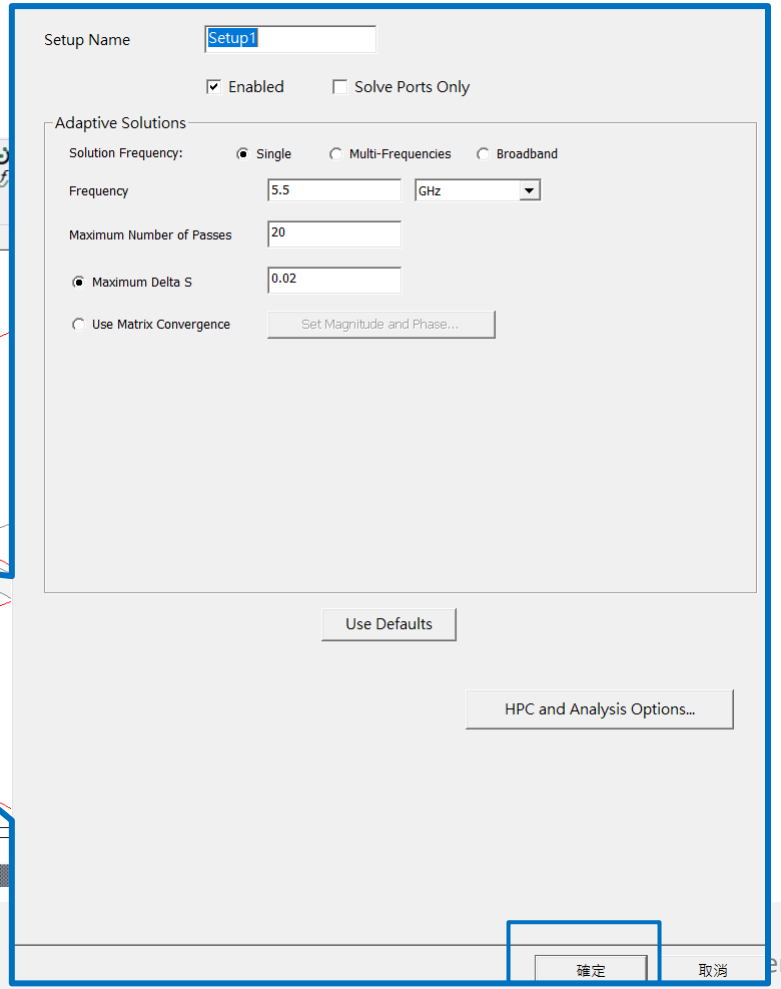


# Edit Radiation Sphere

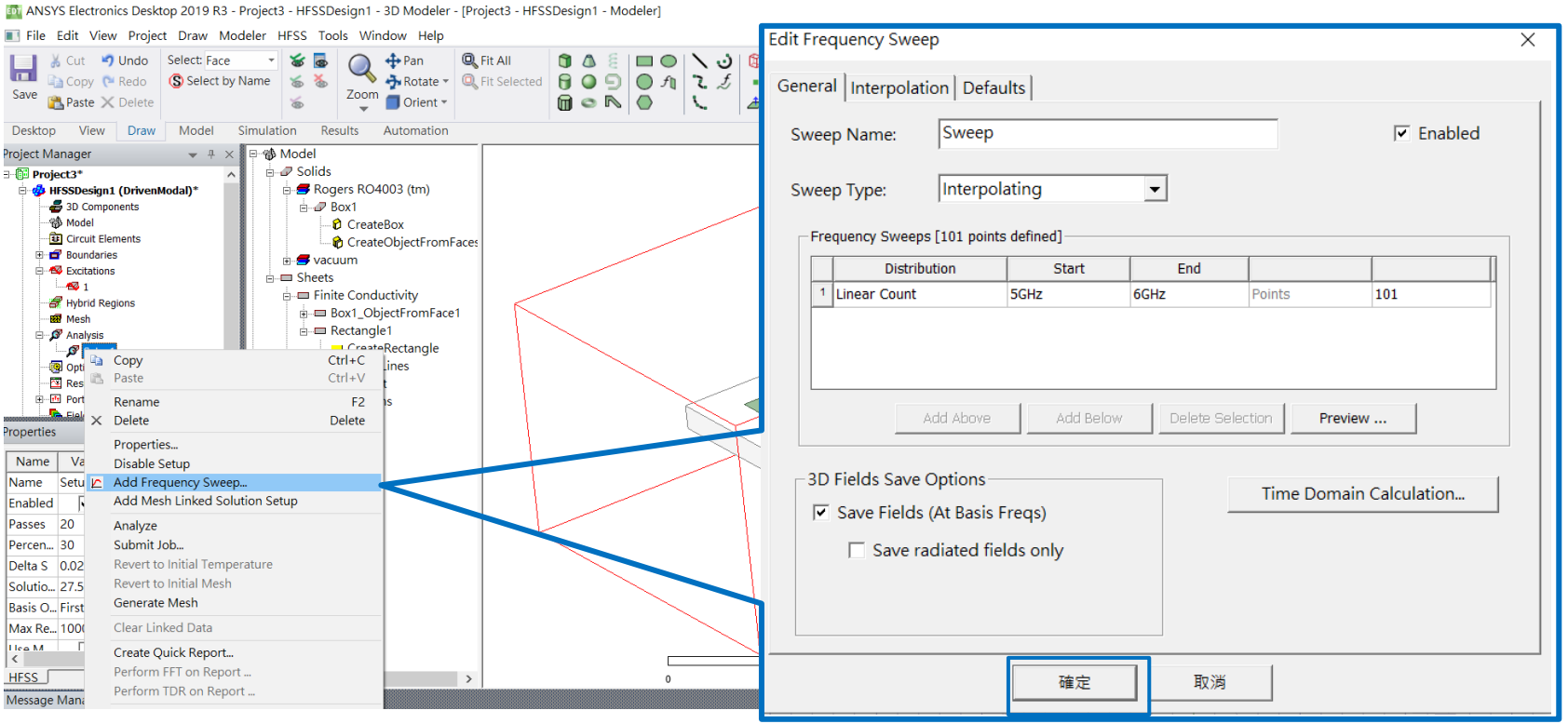


# Add Solution Setup

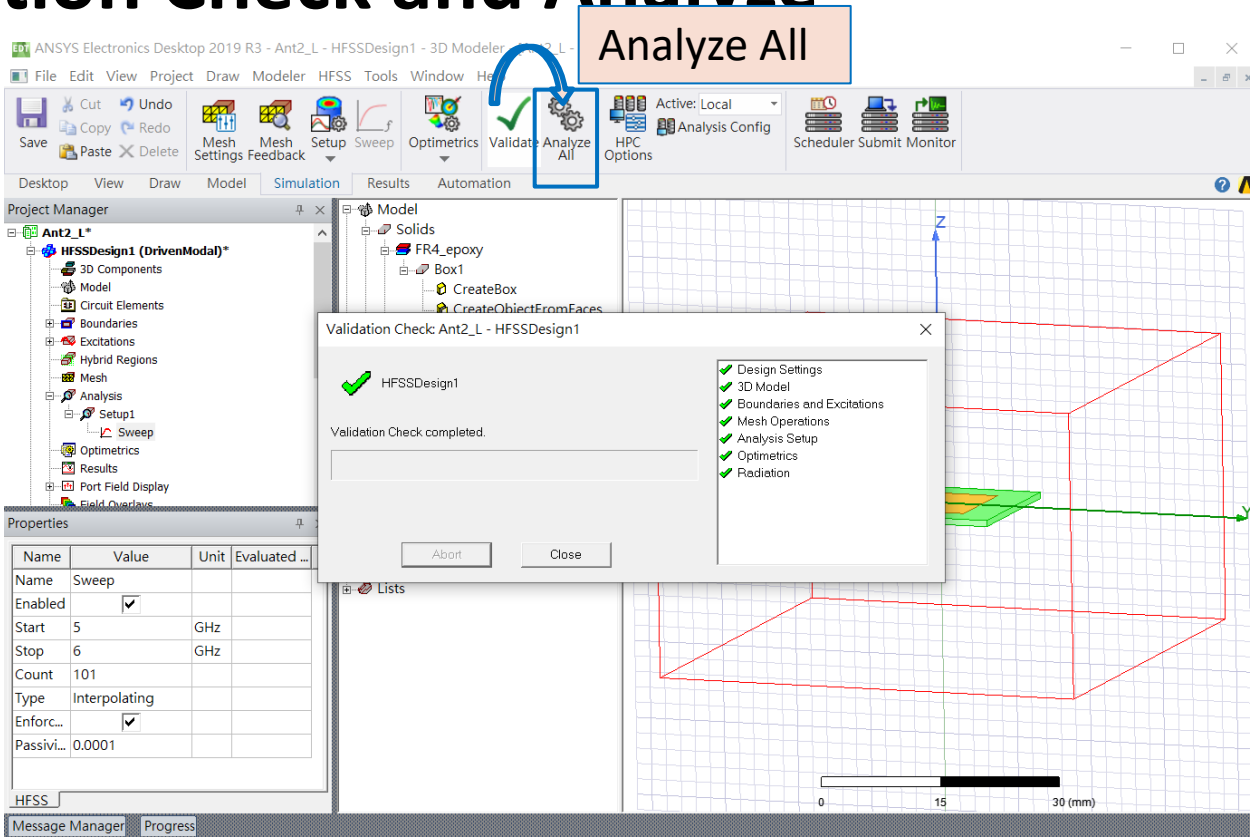


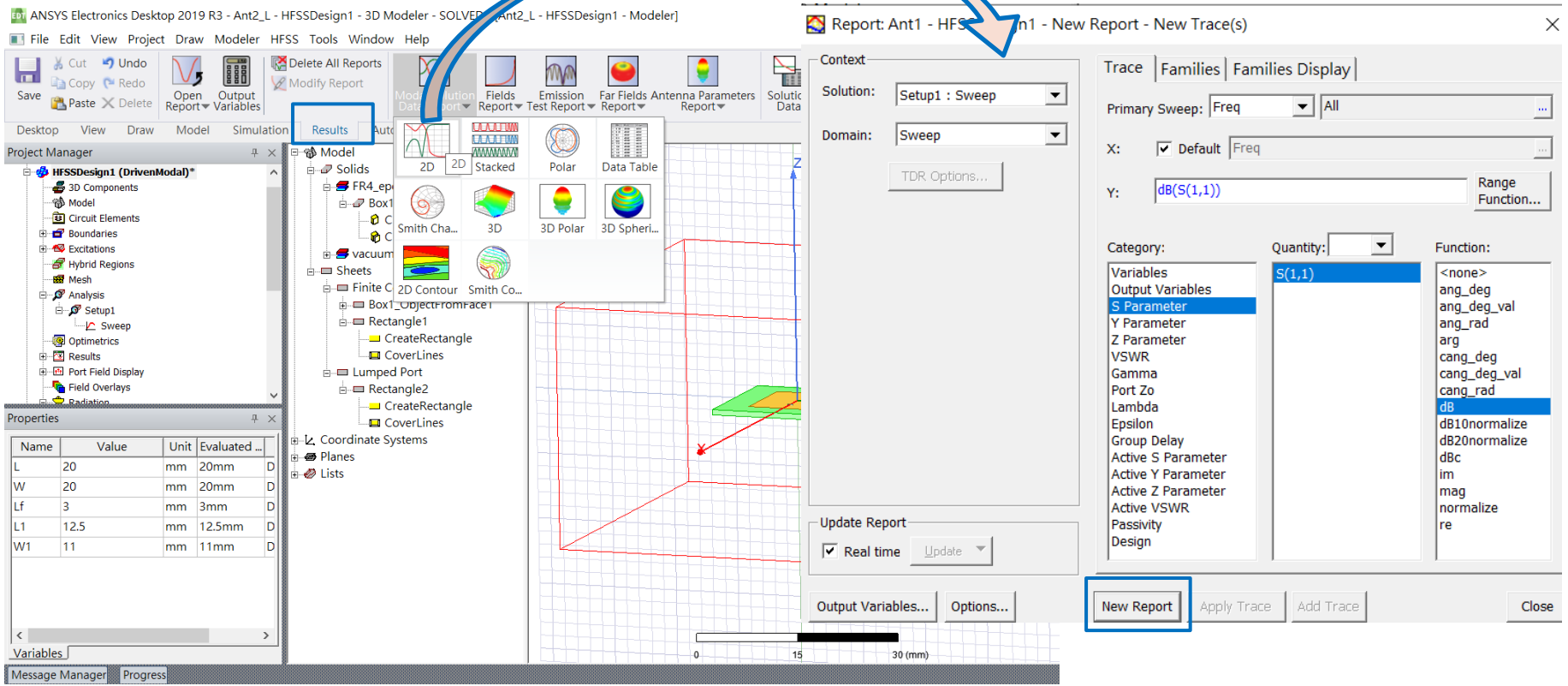
# Add Frequency Sweep



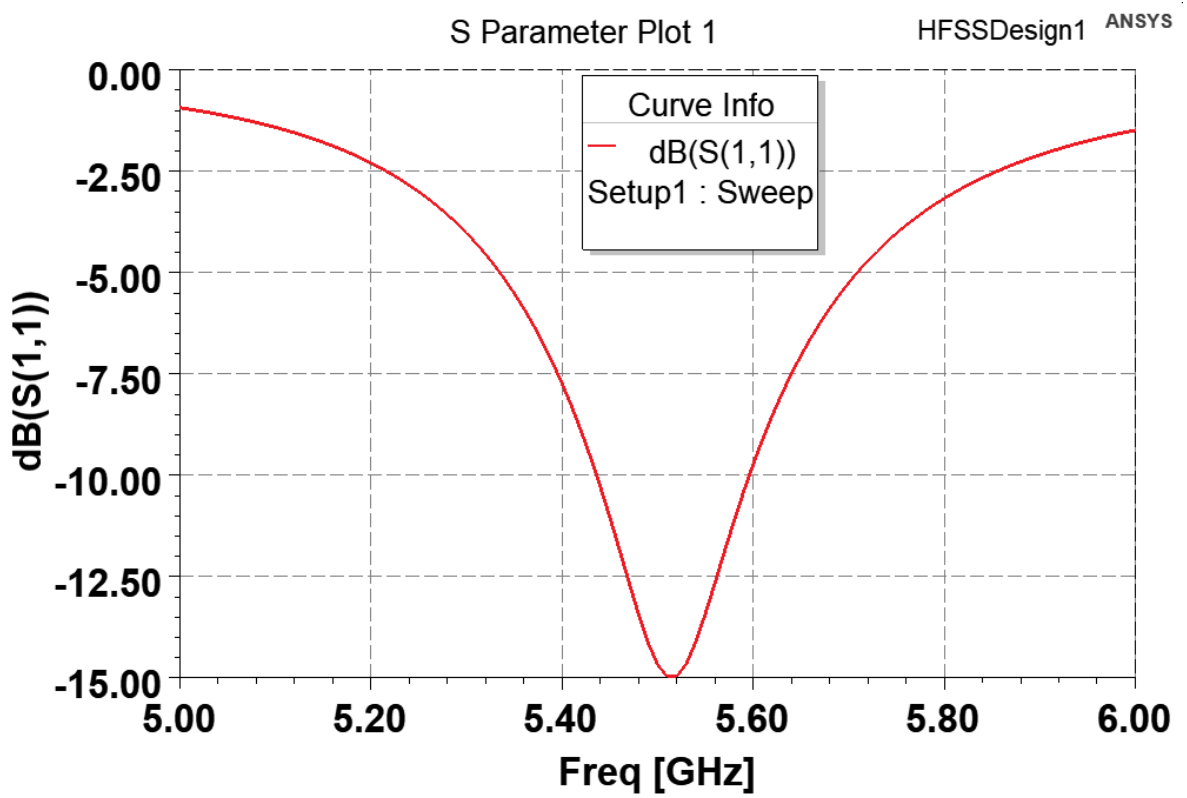
# Validation Check and Analyze



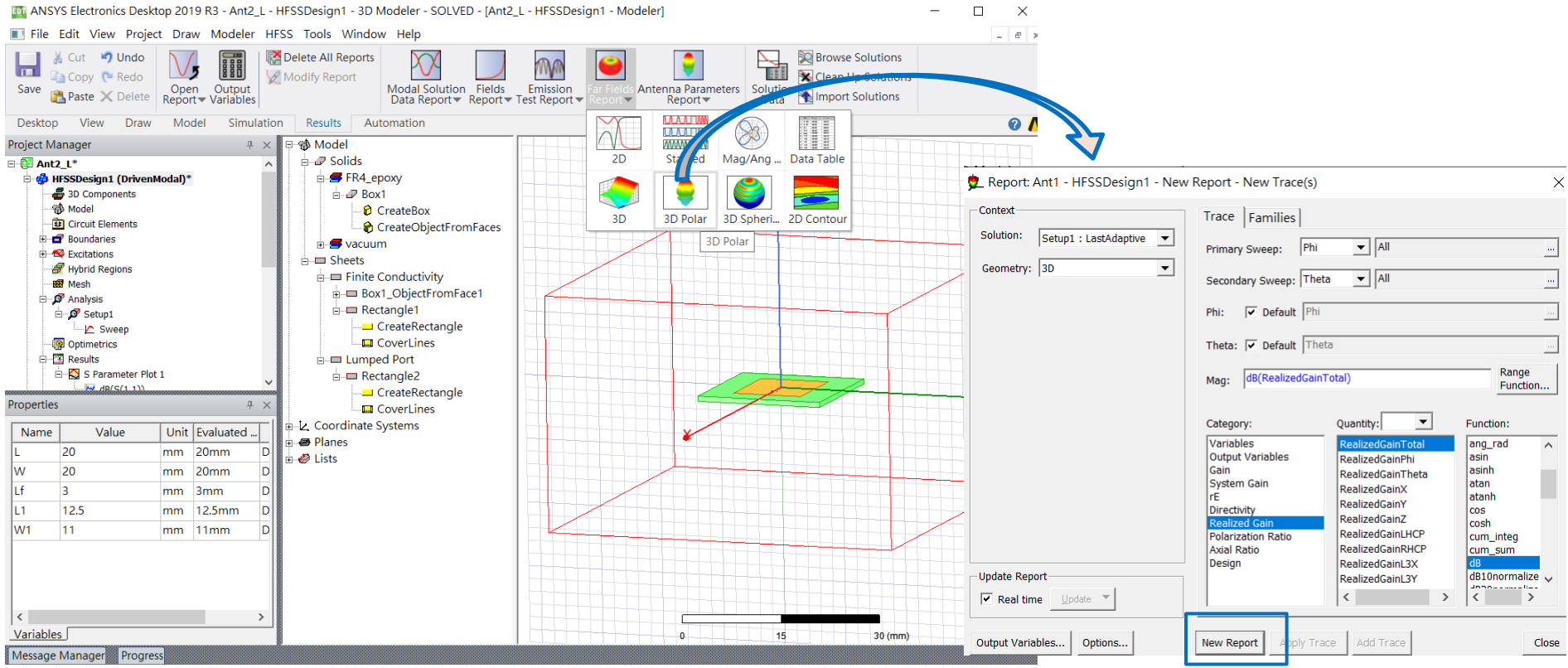
# Plot |S11| Results



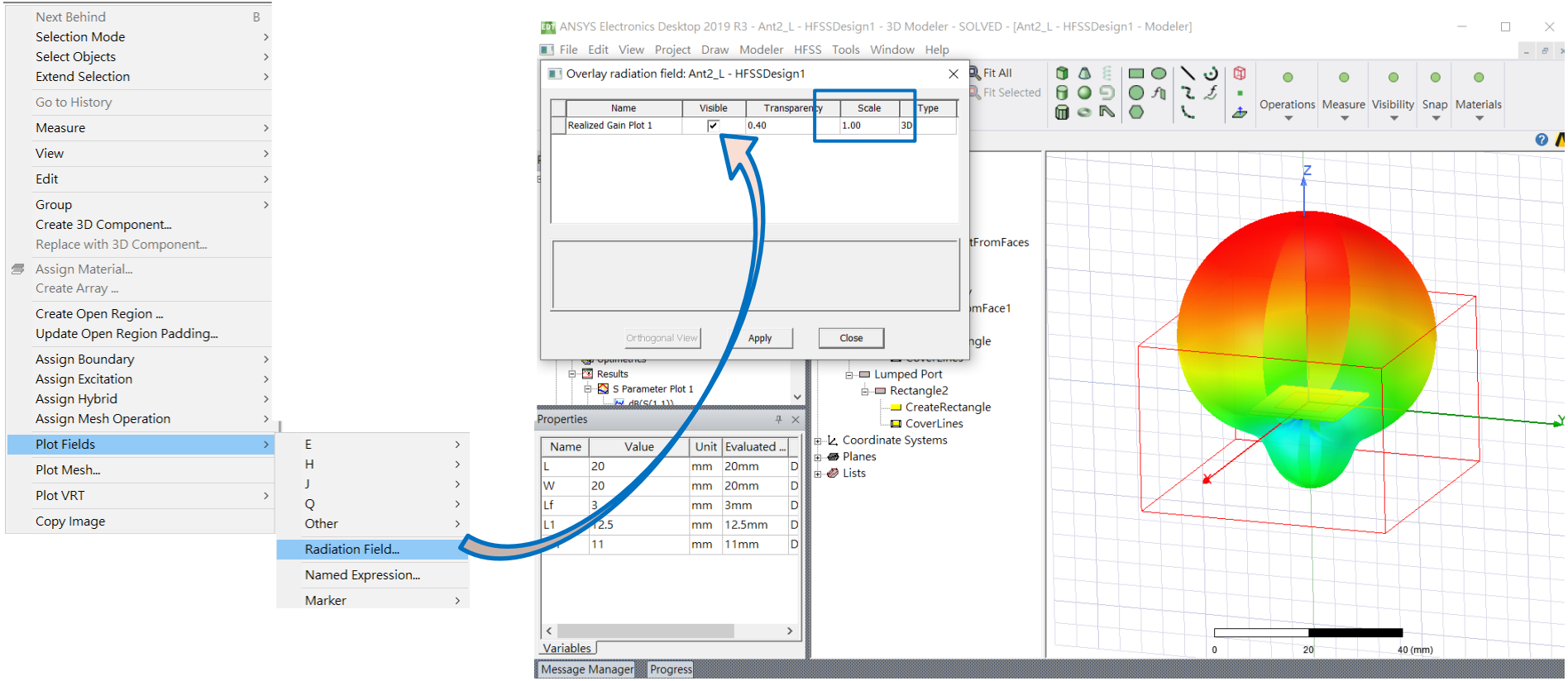
# |S11|


S Parameter Plot 1
HFSSDesign1
ANSYS
Curve Info
dB(S(1,1))
Setup1 : Sweep
0.00
-2.50
-5.00
-7.50
-10.00
-12.50
-15.00
dB(S(1,1))
5.00
5.20
5.40
5.60
5.80
6.00
Freq [GHz]

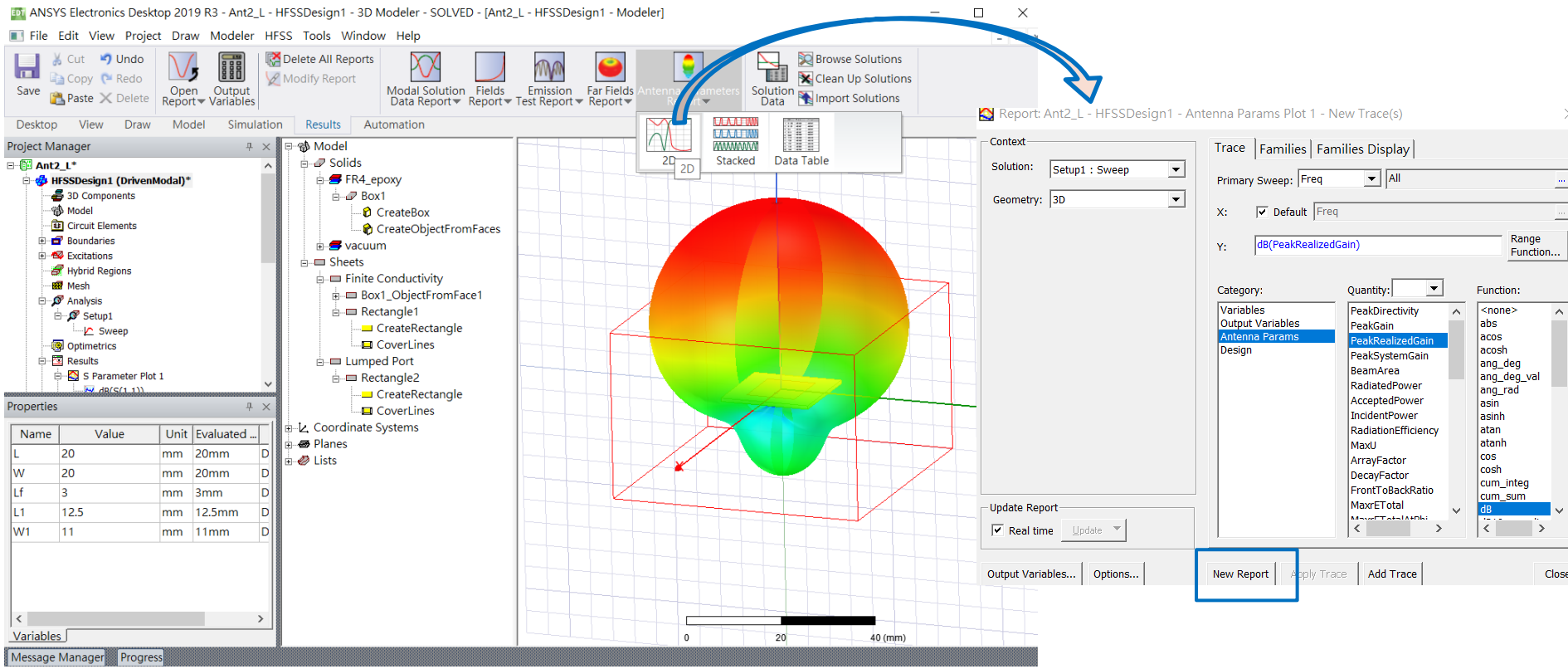
# Plot 3D Pattern



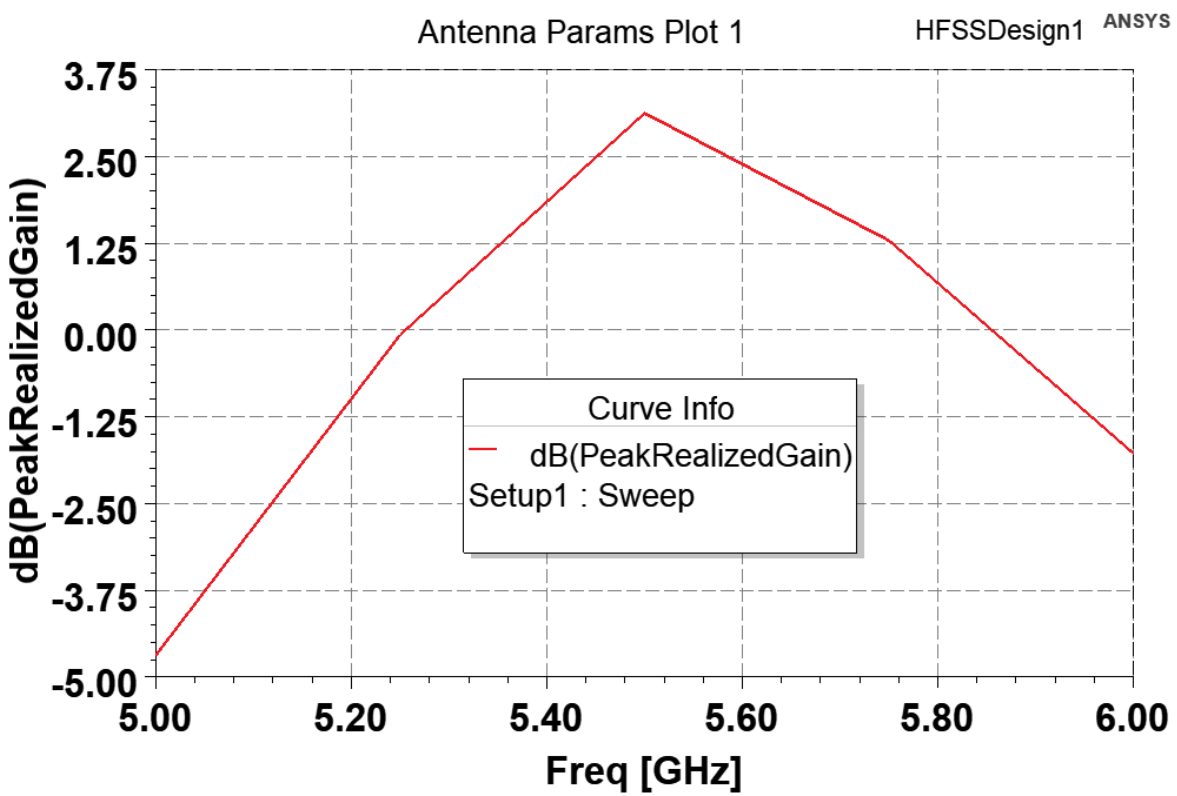
# 3D Pattern


Next Behind B
Selection Mode
Select Objects
Extend Selection
Go to History
Measure
View
Edit
Group
Create 3D Component...
Replace with 3D Component...
Assign Material...
Create Array ...
Create Open Region ...
Update Open Region Padding...
Assign Boundary
Assign Excitation
Assign Hybrid
Assign Mesh Operation
Plot Fields
Plot Mesh...
Plot VRT
Copy Image
E
H
J
Q
Other
Radiation Field...
Named Expression...
Marker
ANSYS Electronics Desktop 2019 R3 - Ant2_L - HFSSDesign1 - 3D Modeler - SOLVED - [Ant2_L - HFSSDesign1 - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Overlay radiation field: Ant2_L - HFSSDesign1
Name Visible Transparency Scale Type
Realized Gain Plot 1 0.40 1.00 3D
Orthogonal View
Apply
Close
Fit All
Fit Selected
Operations
Measure
Visibility
Snap
Materials
Results
S Parameter Plot 1
Lumped Port
Rectangle2
CreateRectangle
CoverLines
Coordinate Systems
Planes
Lists
Properties
Name Value Unit Evaluated ...
L 20 mm 20mm D
W 20 mm 20mm D
Lf 3 mm 3mm D
L1 12.5 mm 12.5mm D
11 mm 11mm D
Variables
Message Manager
Progress
Z
Y
0
20
40 (mm)

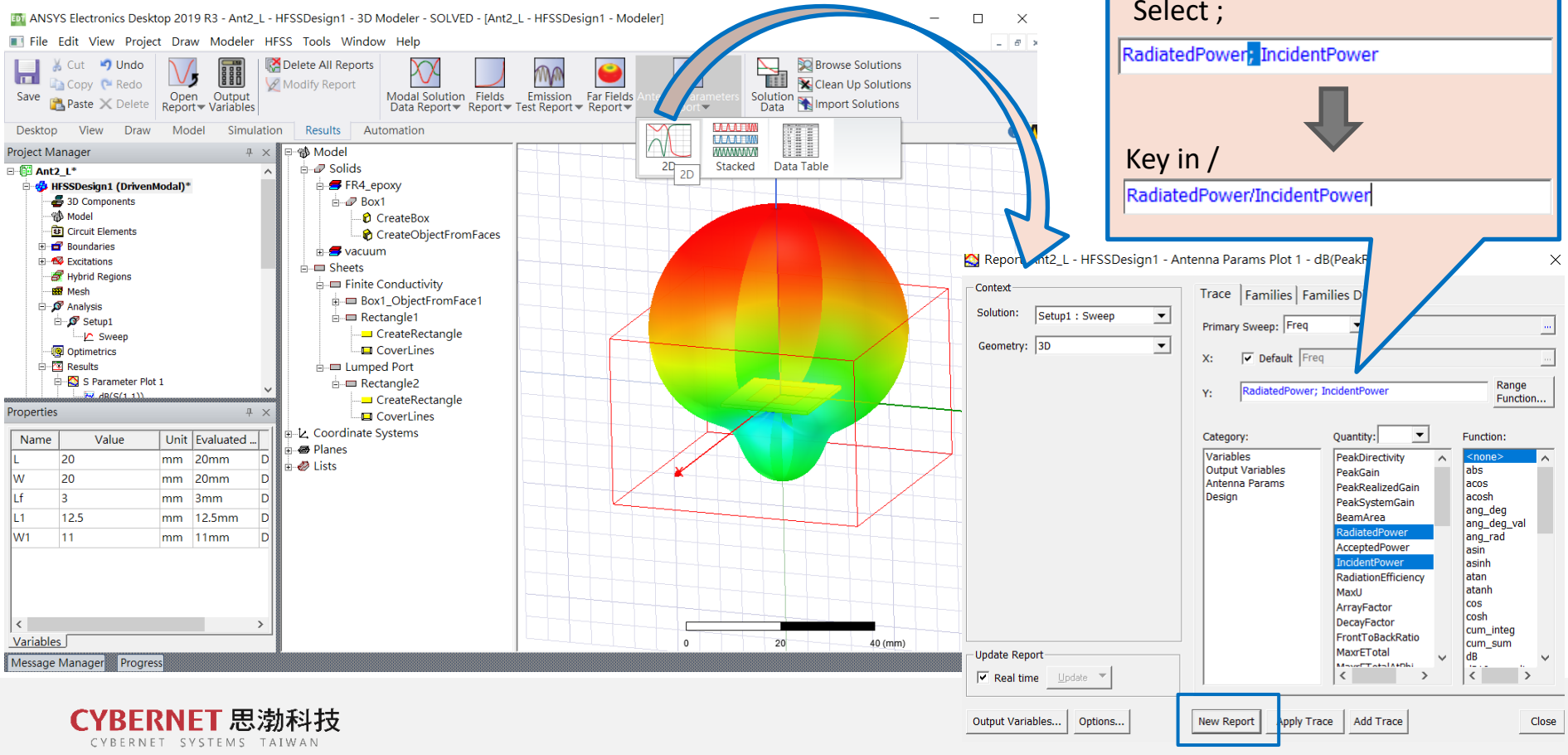
# Plot Peak Gain



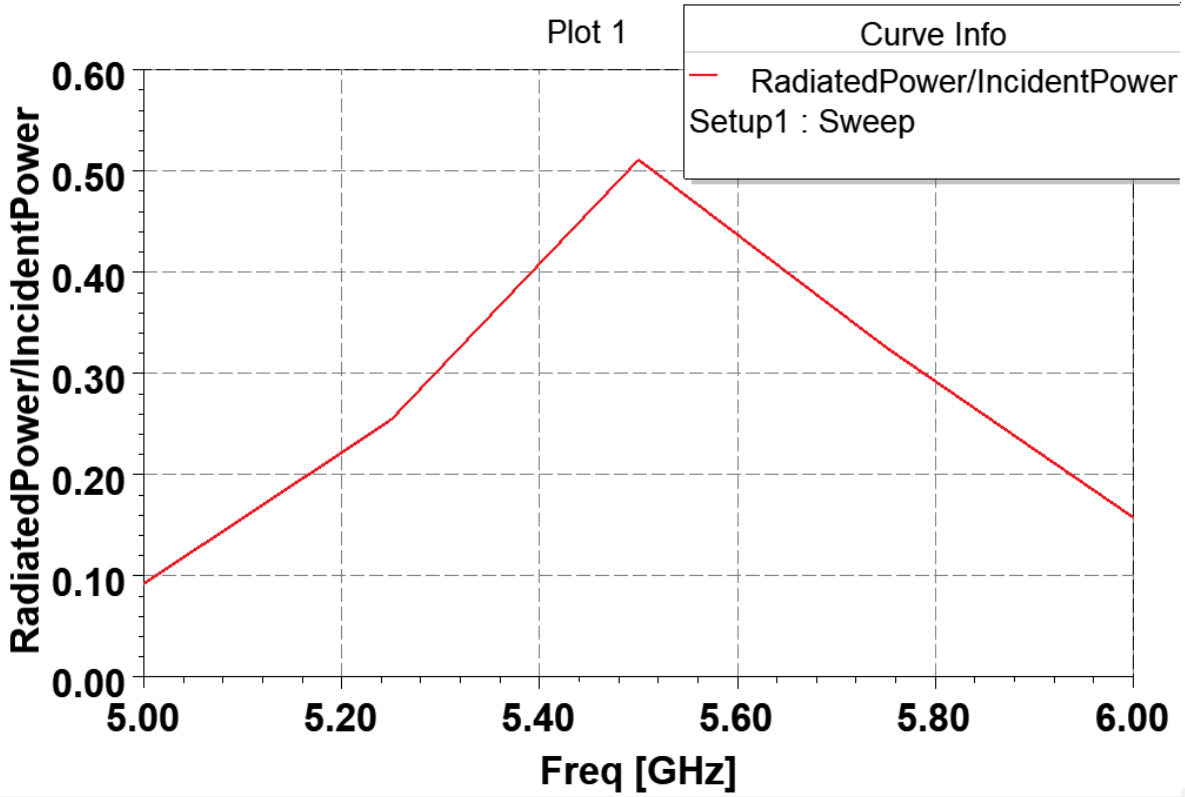
# Peak Gain


Antenna Params Plot 1
HFSSDesign1
ANSYS
dB(PeakRealizedGain)
3.75
2.50
1.25
0.00
-1.25
-2.50
-3.75
-5.00
5.00
5.20
5.40
5.60
5.80
6.00
Freq [GHz]
Curve Info
dB(PeakRealizedGain)
Setup1 : Sweep

# Plot Efficiency



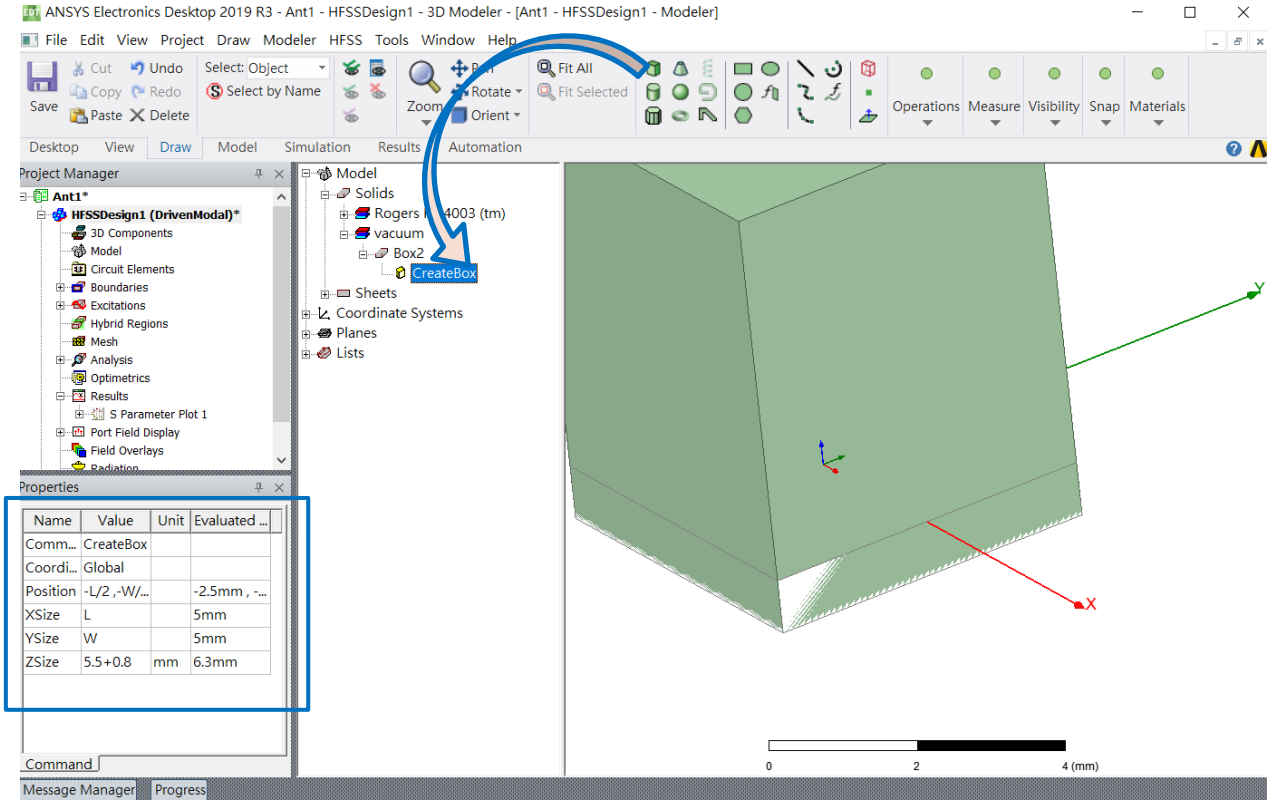
# Efficiency


Plot 1
Curve Info
RadiatedPower/IncidentPower
Setup1 : Sweep
RadiatedPower/IncidentPower
0.60
0.50
0.40
0.30
0.20
0.10
0.00
5.00
5.20
5.40
5.60
5.80
6.00
Freq [GHz]

# Project Production 2

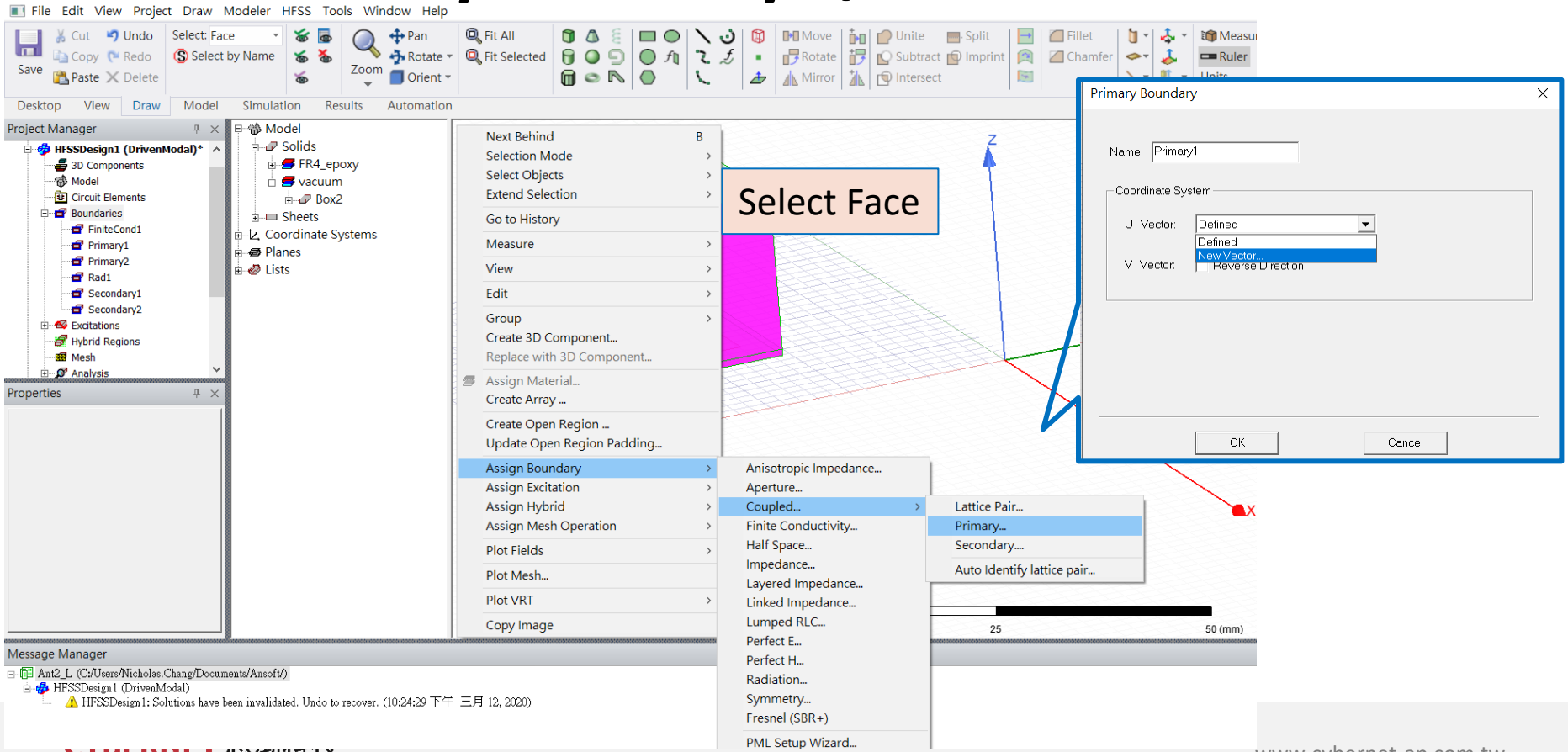
- A 1×4 array with patch antenna simulation by HFSS in finite array distribution

# Draw ABC Box

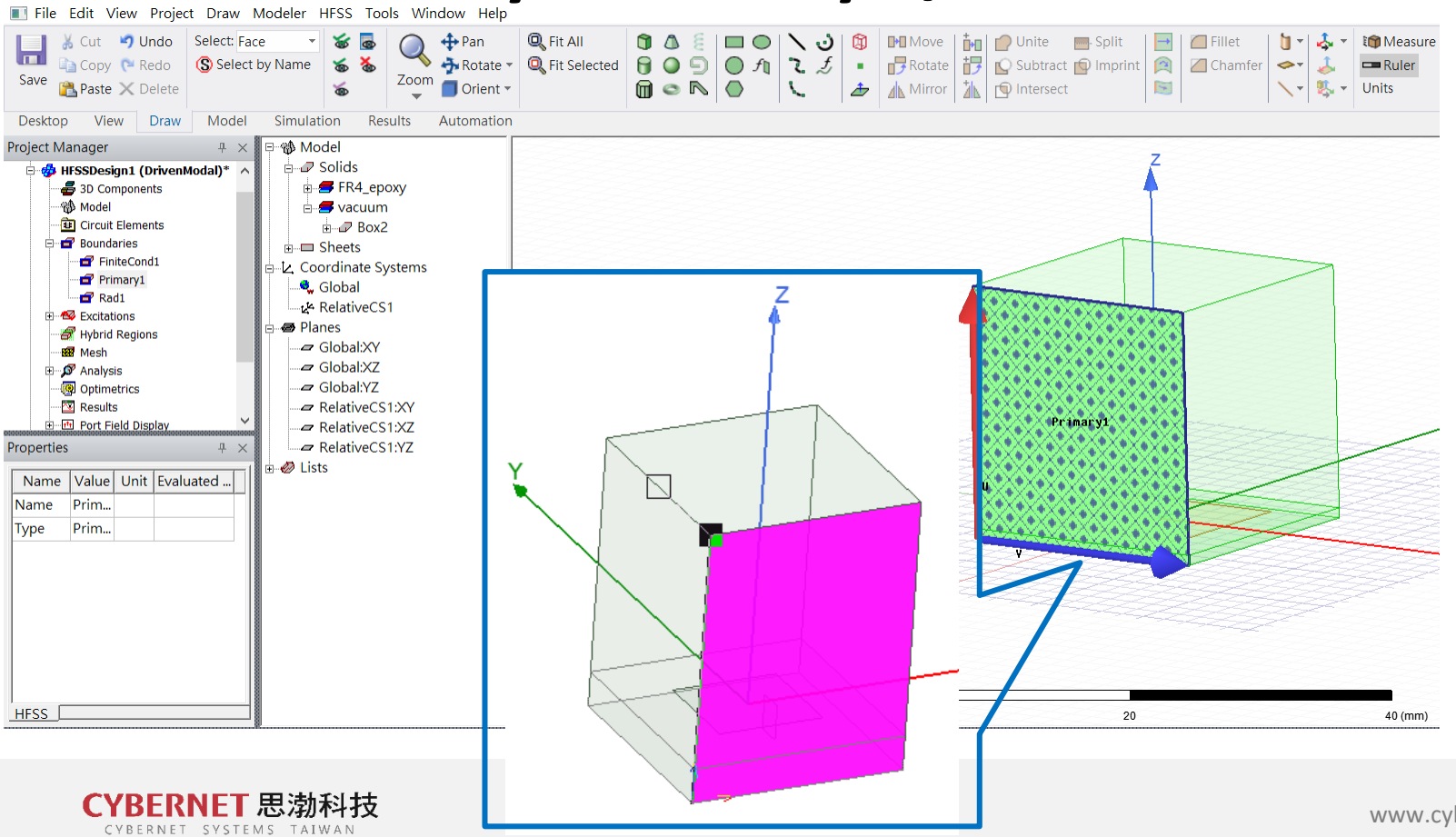



將原本的air box 刪除，繪製新的Box

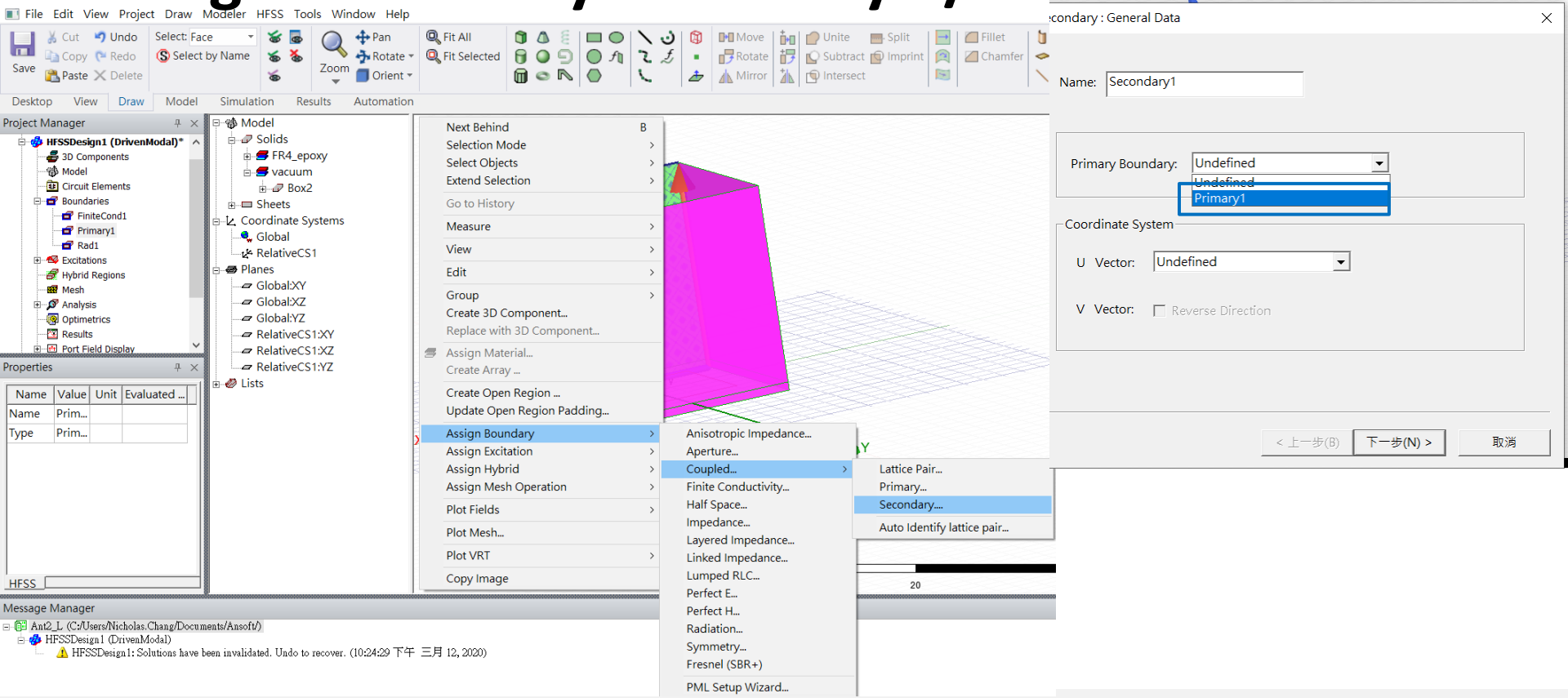
# Draw Primary Boundary 1/2



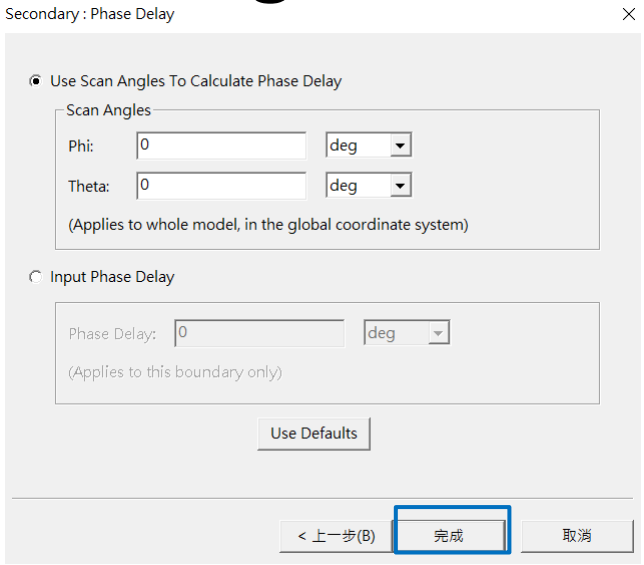
# Draw Primary Boundary 2/2

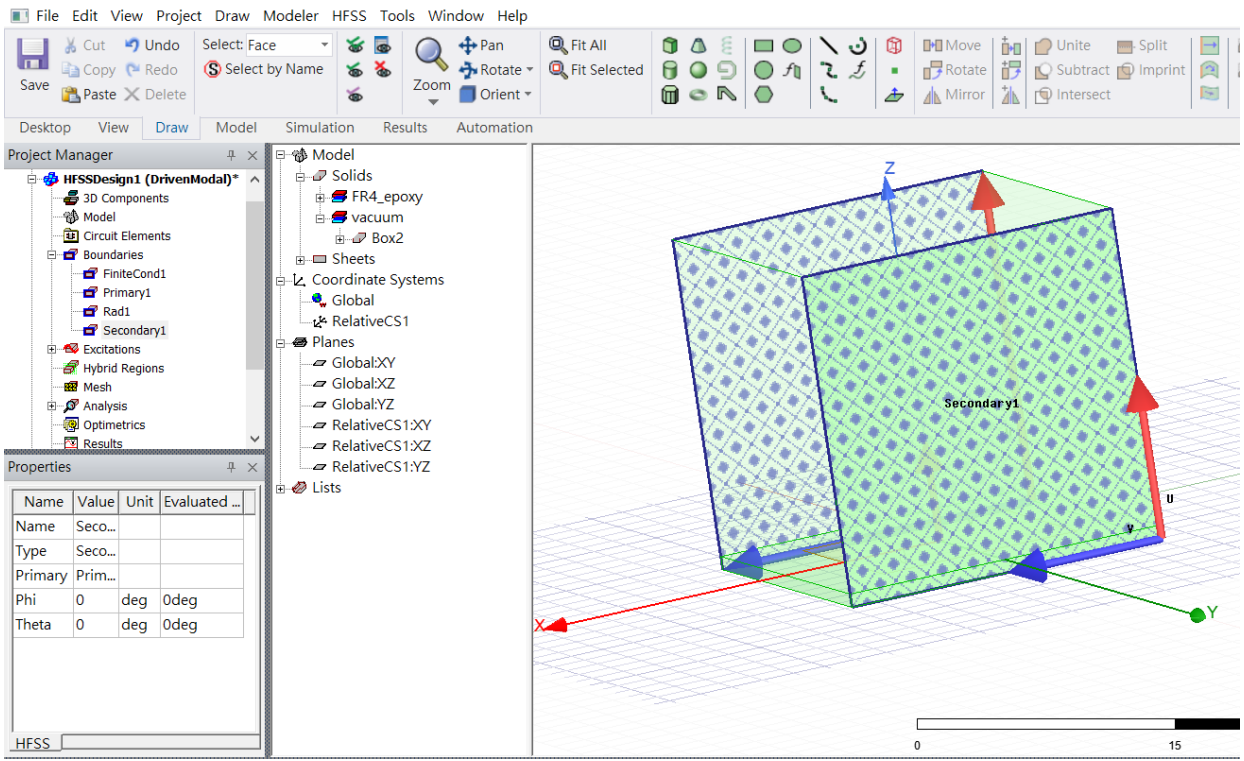


# Assign Secondary Boundary 1/2

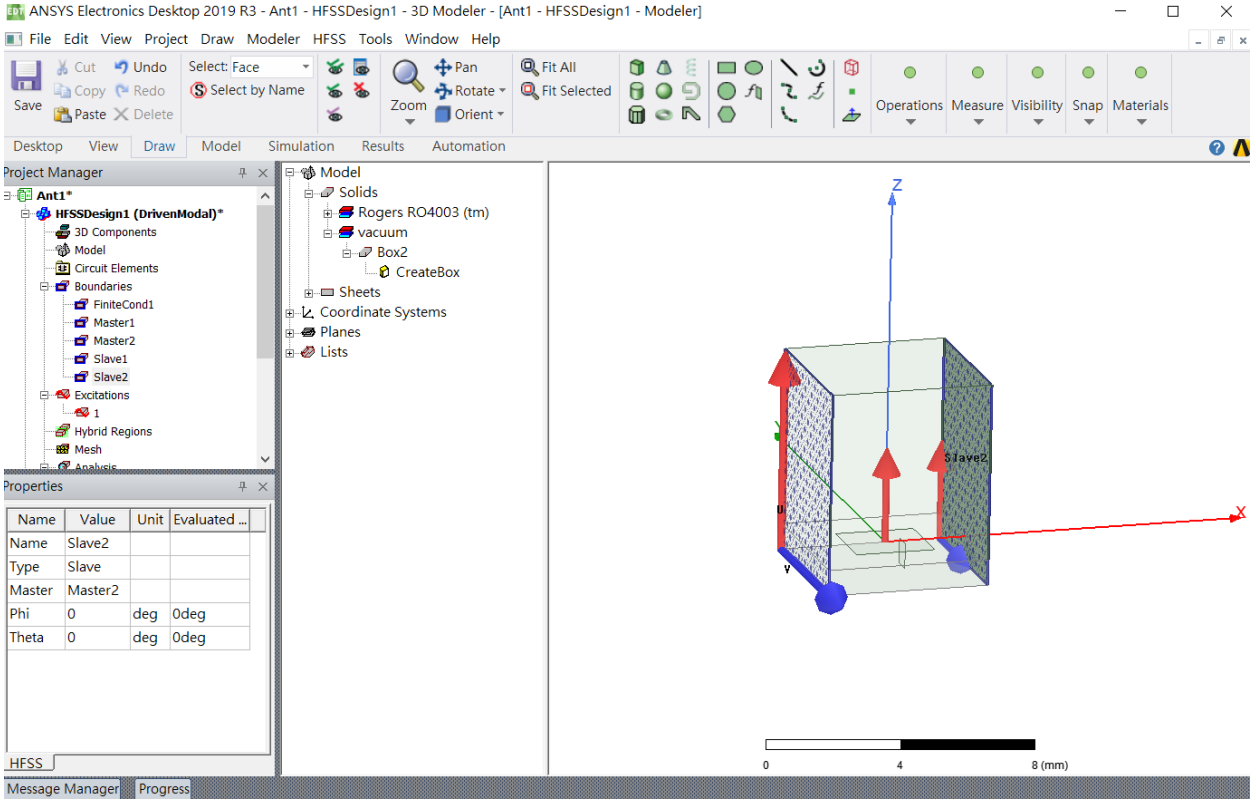


# Assign Secondary Boundary 2/2

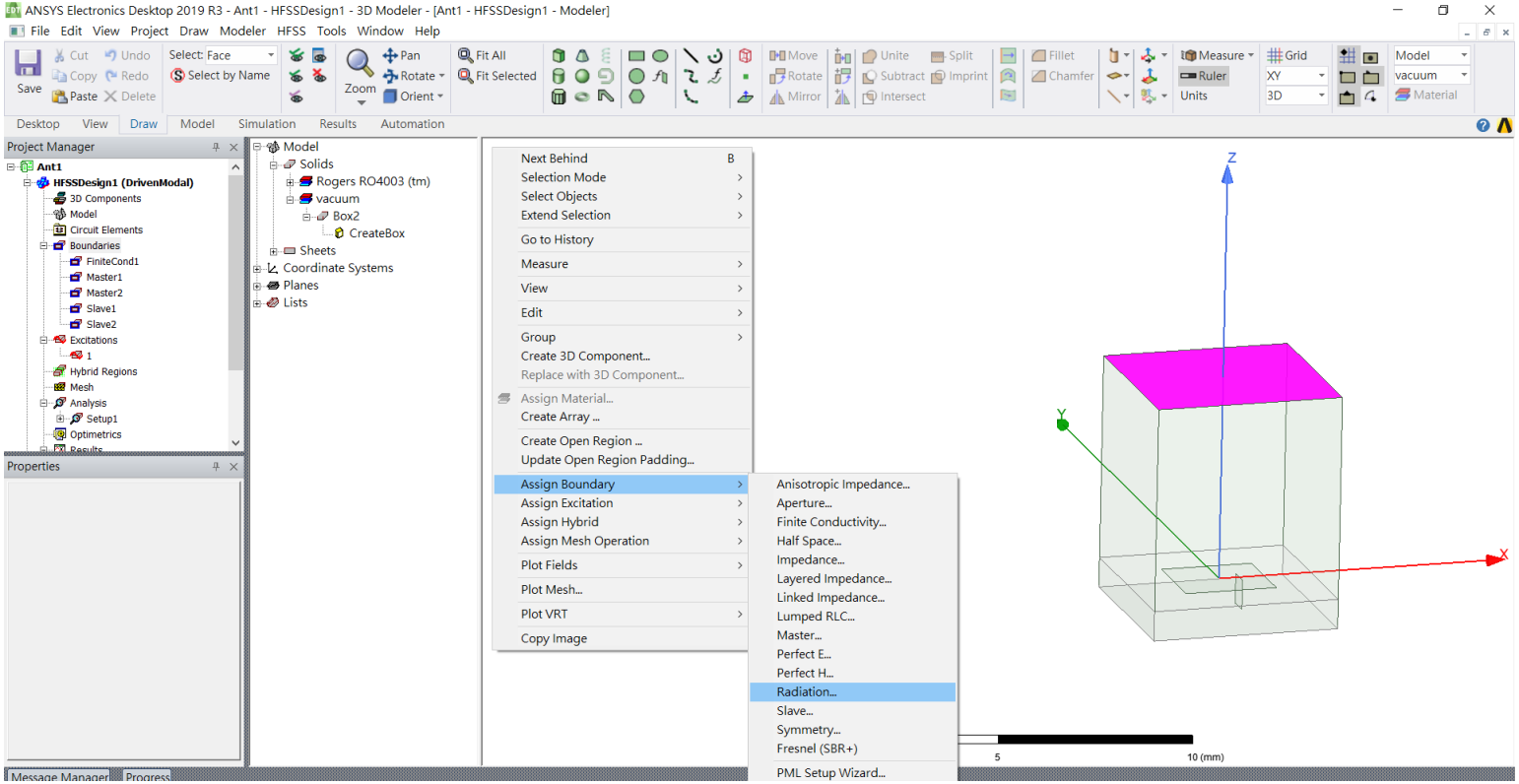


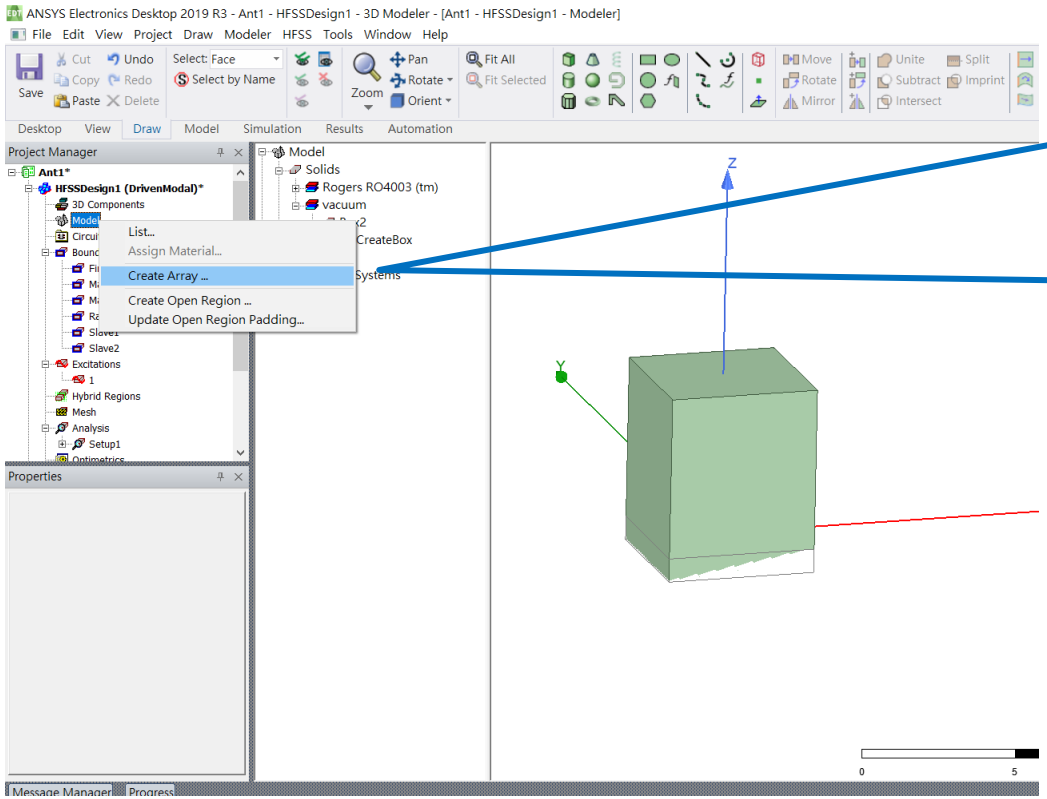
# Assign ABC in The Other Side

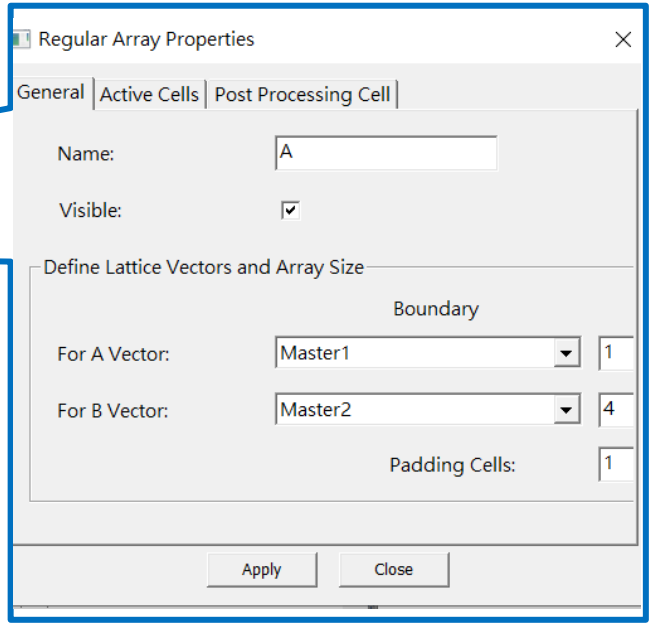


# Assign Radiation Boundary

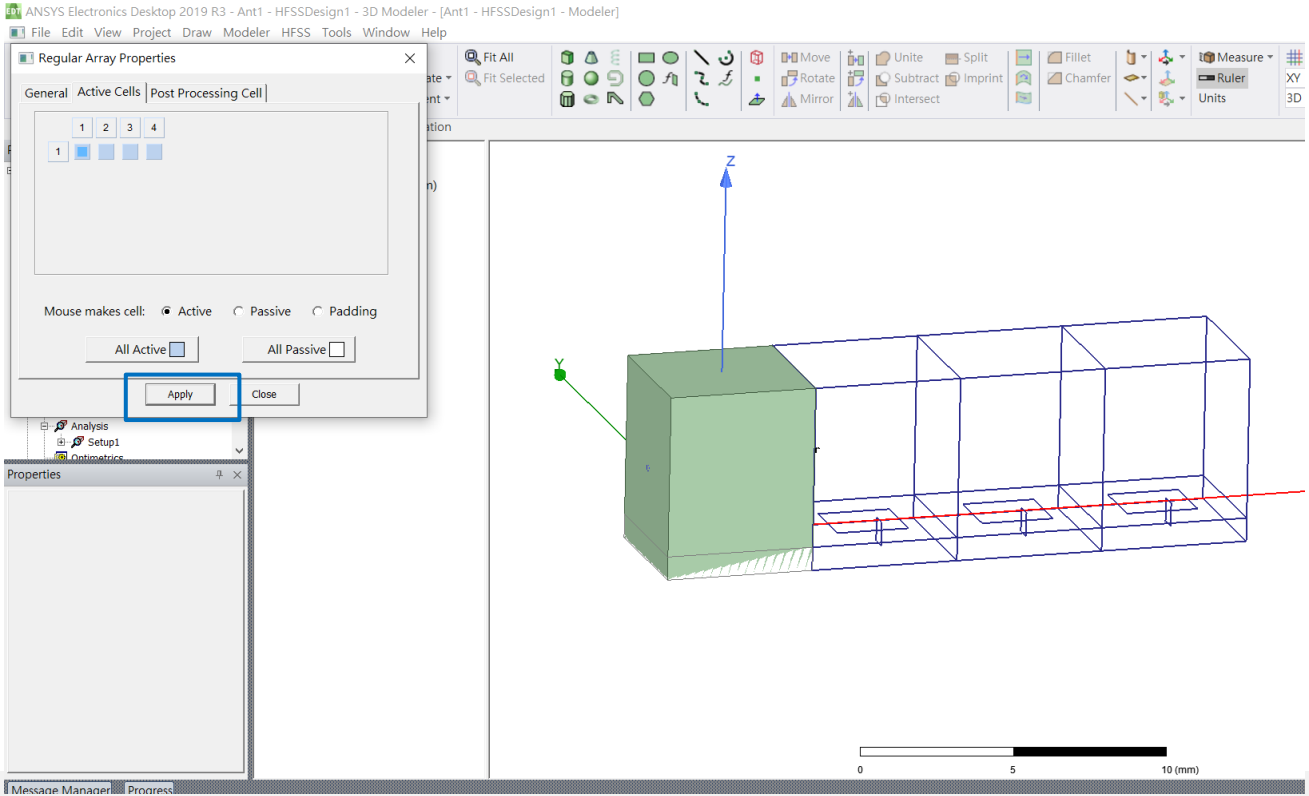


# Create Array 1/2

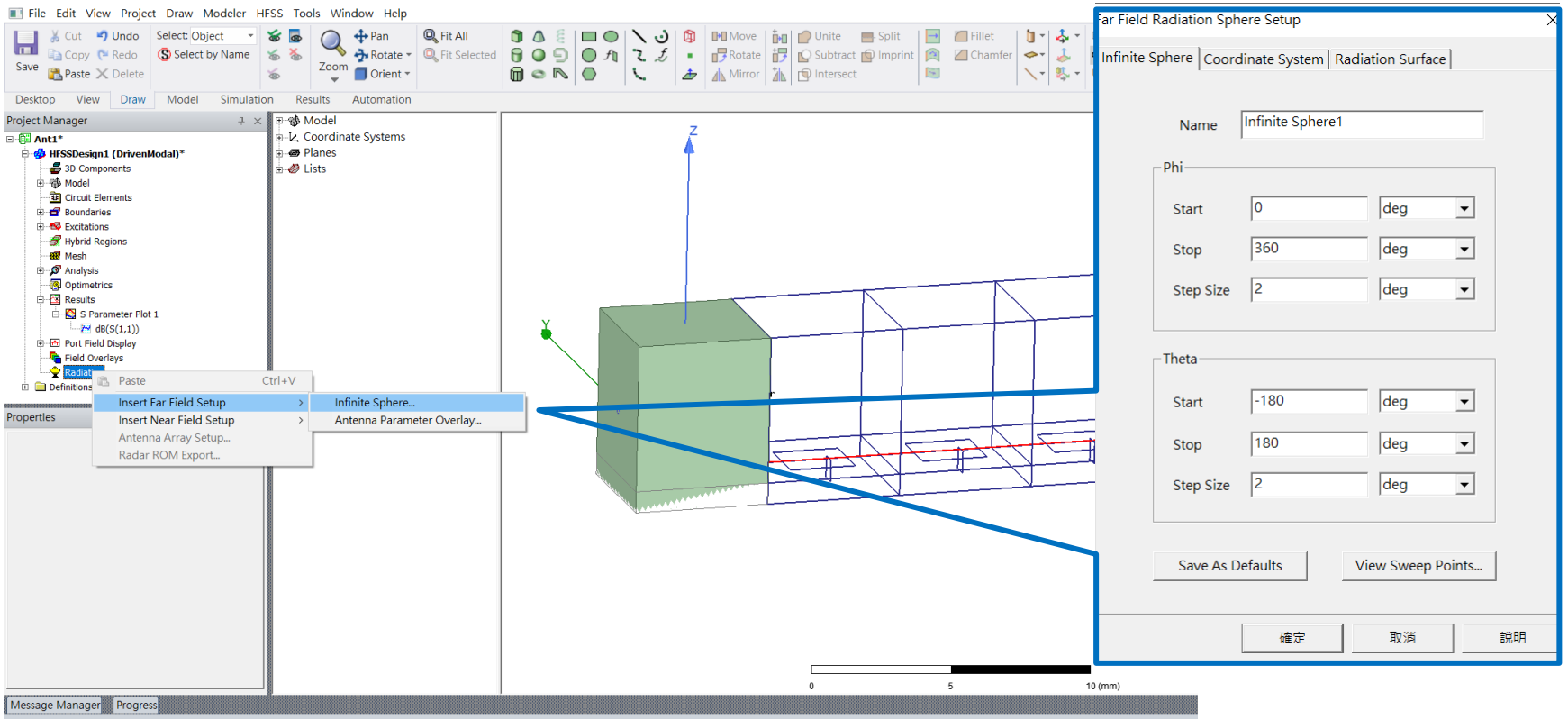


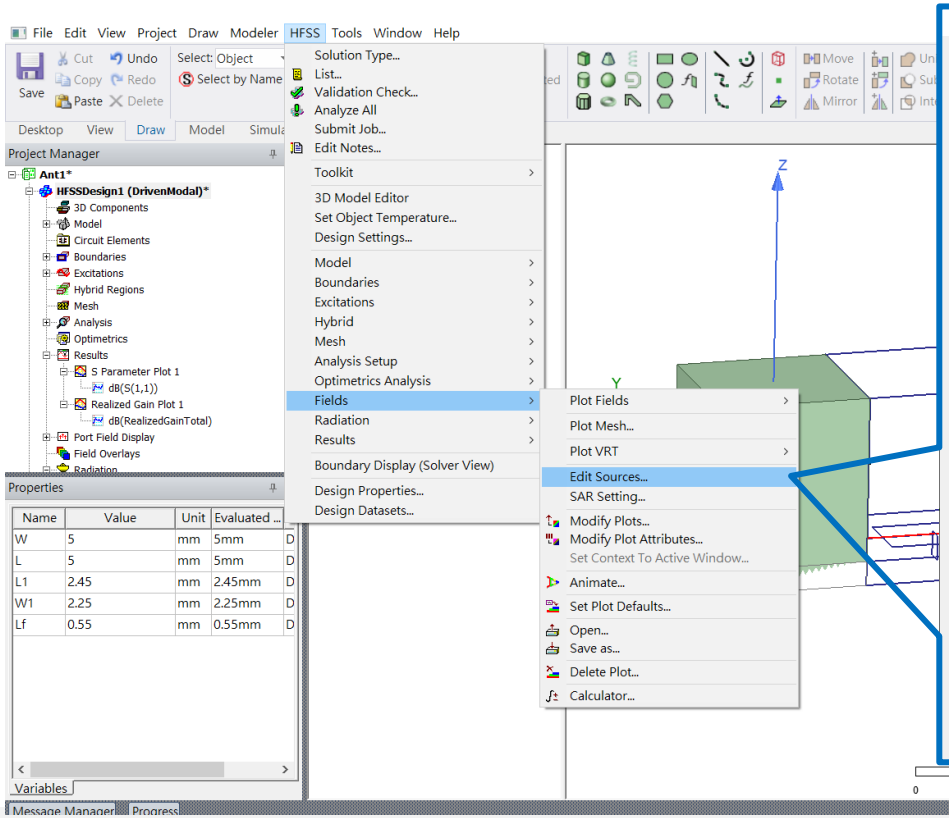
# Create Array 2/2

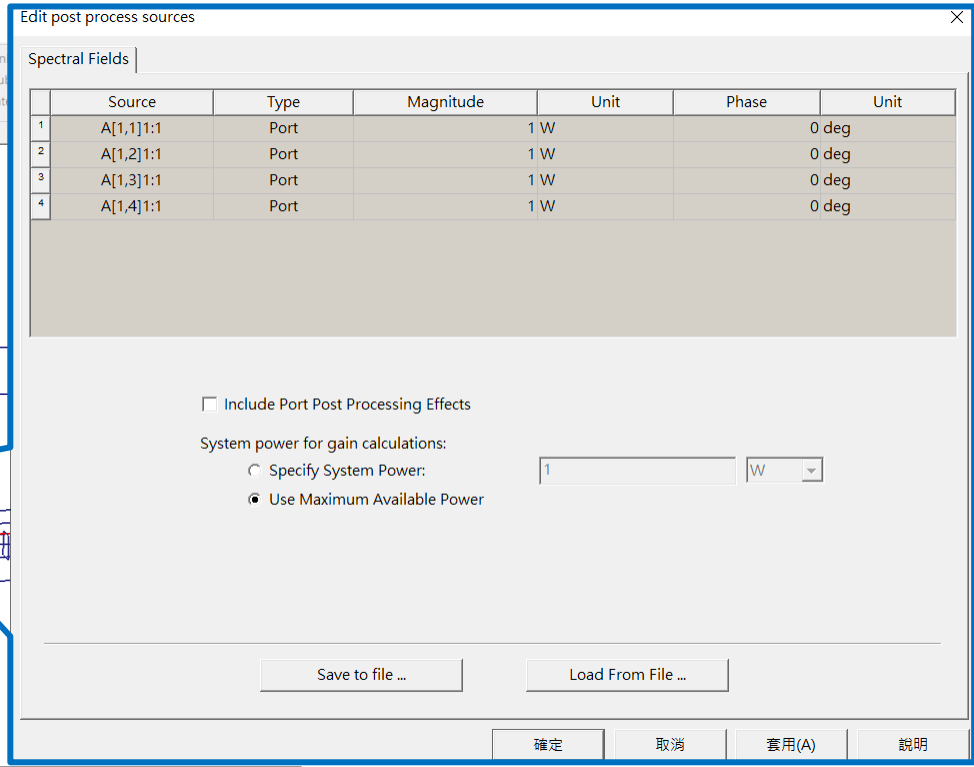


# Insert Far Field

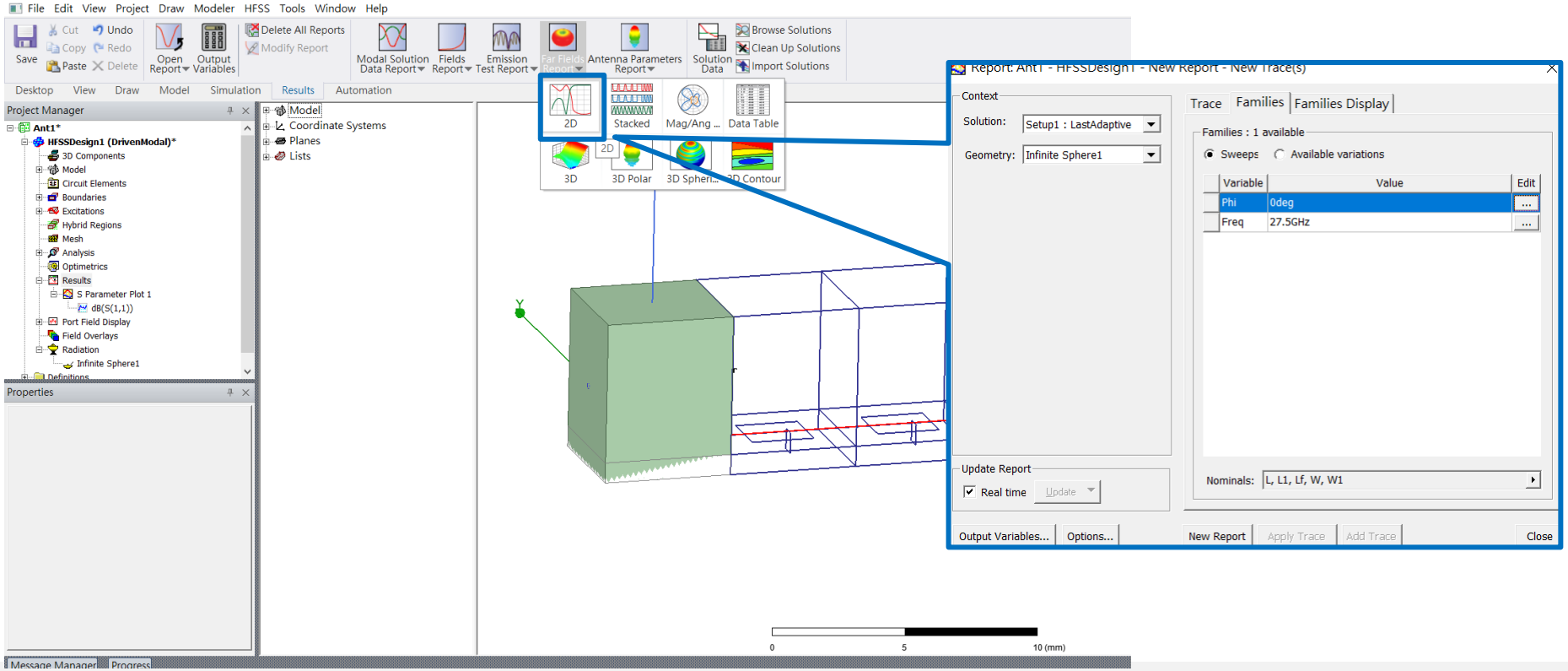


# Edit Sources

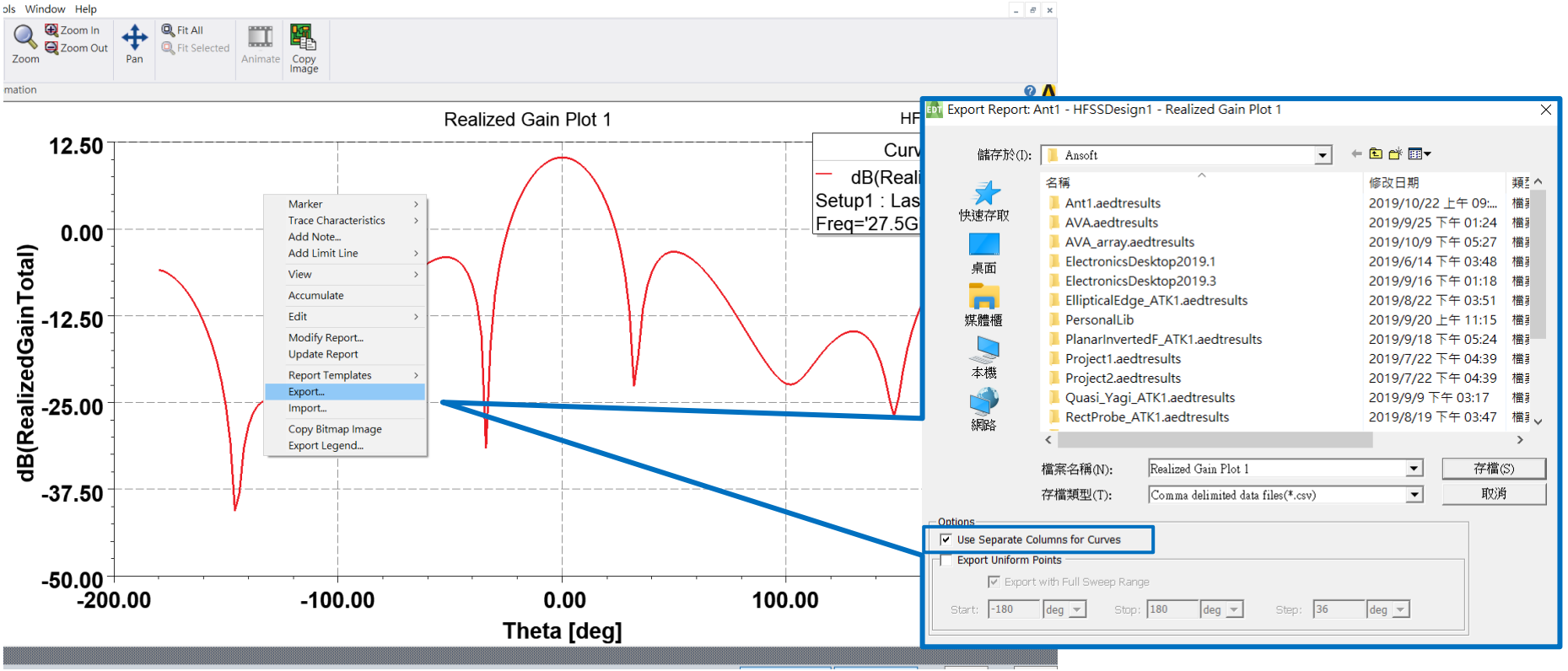


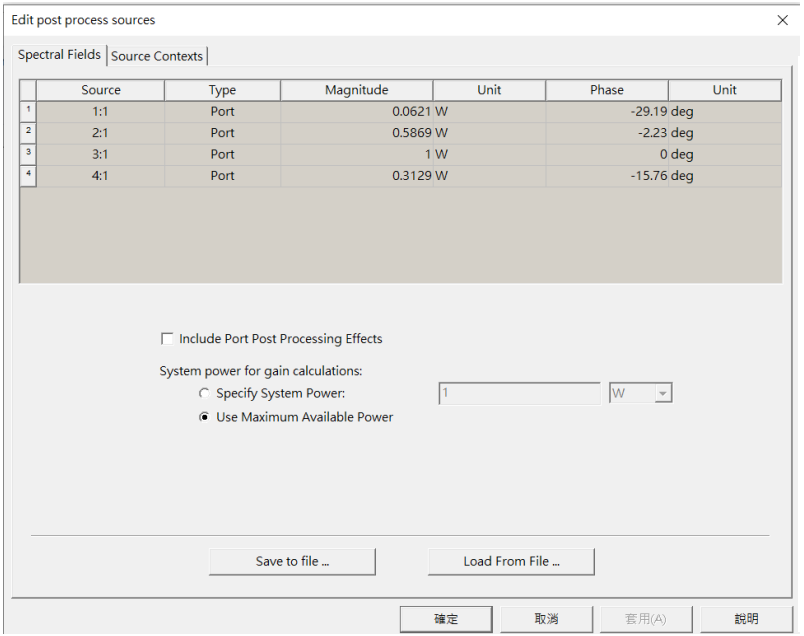
# Plot Radiation Pattern in 2D Report

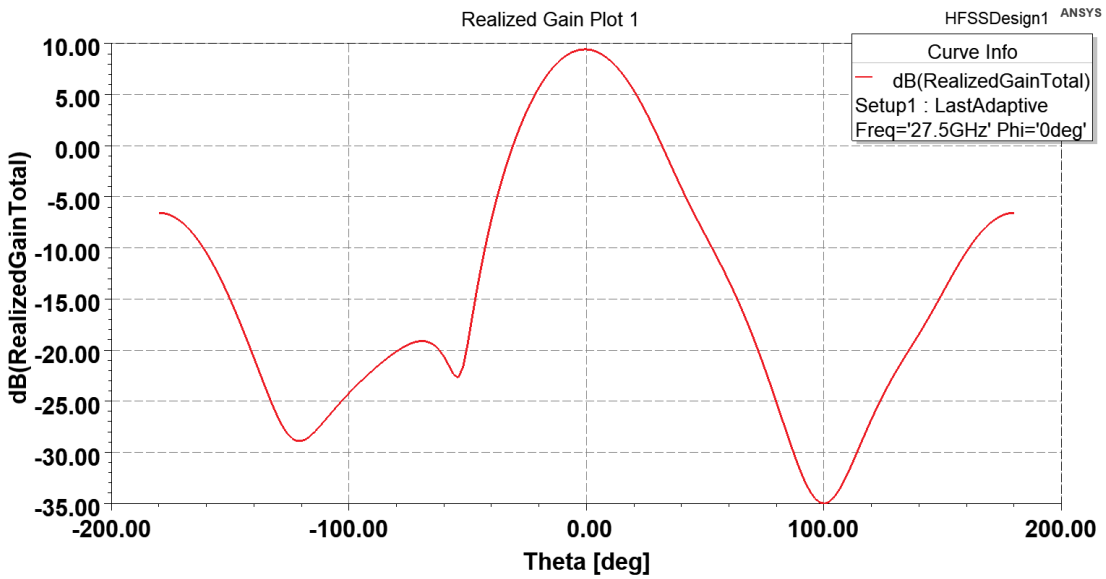


# Export 2D Results



# Edit Source


Edit post process sources
Spectral Fields
Source Contexts
Source
Type
Magnitude
Unit
Phase
Unit
1:1 Port 0.0621 W -29.19 deg
2:1 Port 0.5869 W -2.23 deg
3:1 Port 1 W 0 deg
4:1 Port 0.3129 W -15.76 deg
Include Port Post Processing Effects
System power for gain calculations:
Specify System Power:
1
W
Use Maximum Available Power
Save to file ...
Load From File ...
確定
取消
套用(A)
說明


Realized Gain Plot 1
HFSSDesign1
ANSYS
Curve Info
dB(RealizedGainTotal)
Setup1 : LastAdaptive
Freq='27.5GHz' Phi='0deg'
dB(RealizedGainTotal)
10.00
5.00
0.00
-5.00
-10.00
-15.00
-20.00
-25.00
-30.00
-35.00
-200.00
-100.00
0.00
100.00
200.00
Theta [deg]

# HFSS Antenna Toolkit

- HFSS ACT Extensions


CYBERNET 思渤科技
CYBERNET SYSTEMS TAIWAN

# Starting HFSS and ACT Extensions

Launching ANSYS Electronics Desktop

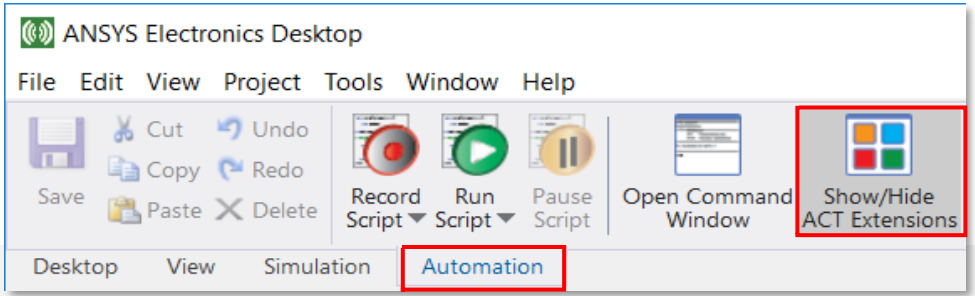
- To access *ANSYS Electronics Desktop*, click the Microsoft *Start* button
- Select *Programs* > *ANSYS Electromagnetics* > *ANSYS Electromagnetics Suite*.
- Select *ANSYS Electronics Desktop*

Viewing ACT Extensions

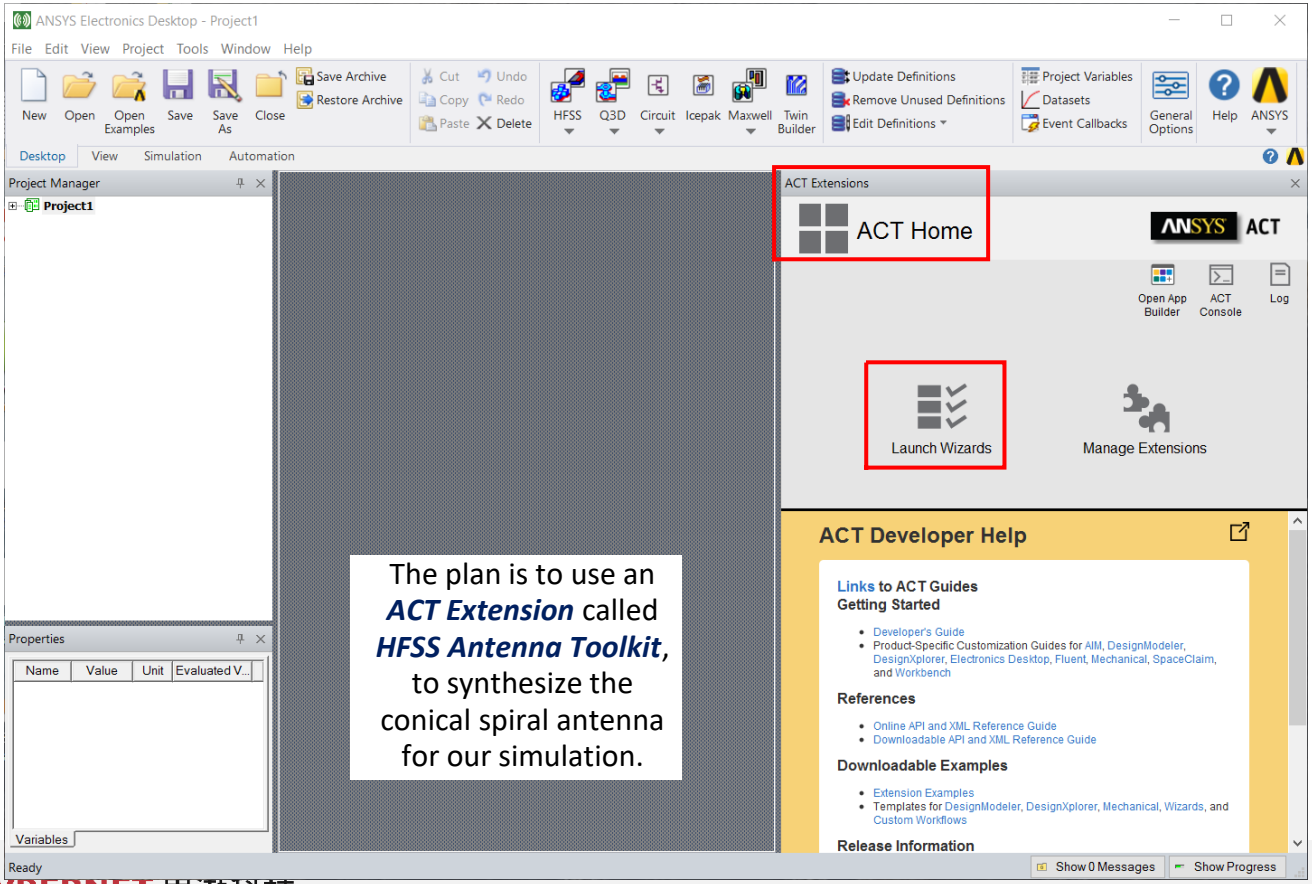
- In ANSYS Electronics Desktop, with the Automation tab highlighted, click Show/Hide ACT Extensions. This brings up the ACT Extensions window. It may show ACT Home.

If ACT Extensions are not available…

- If the ACT Extensions are not available on your installation, click on File > Open, browse to the training file location, and open ConicalLog_ATK1.aedt.

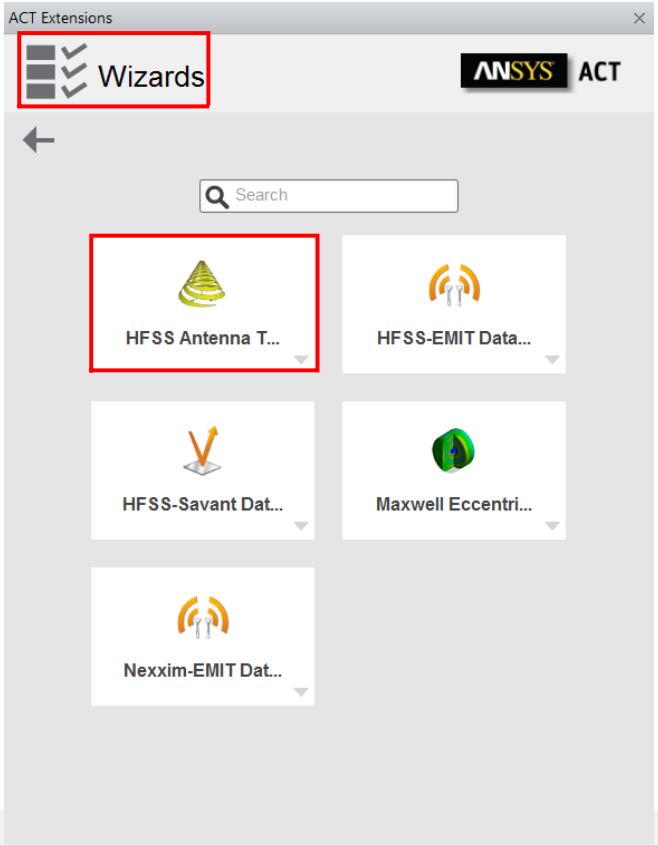


# ACT Extensions Home

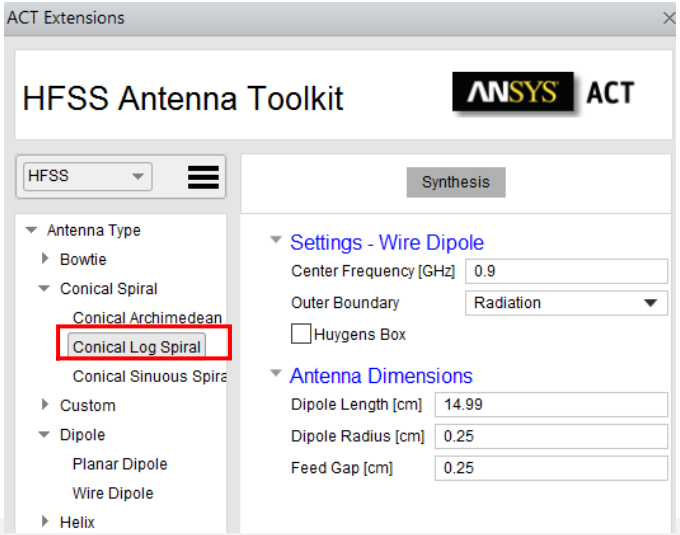



- In *ACT Home*, click on *Launch Wizards*.

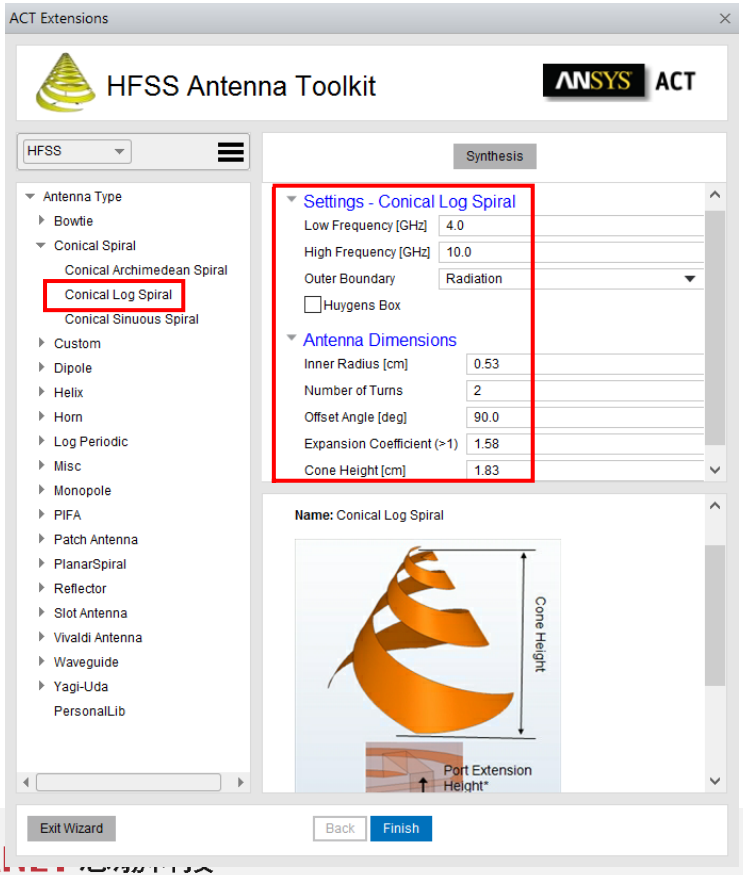
# HFSS Antenna Toolkit




- From *Wizards,* click *HFSS Antenna Toolkit.*
- Select *ANSYS Electronics Desktop*
- When the *Toolkit* comes up, under *Antenna Type* select *Conical Spiral > Conical Log Spiral.*

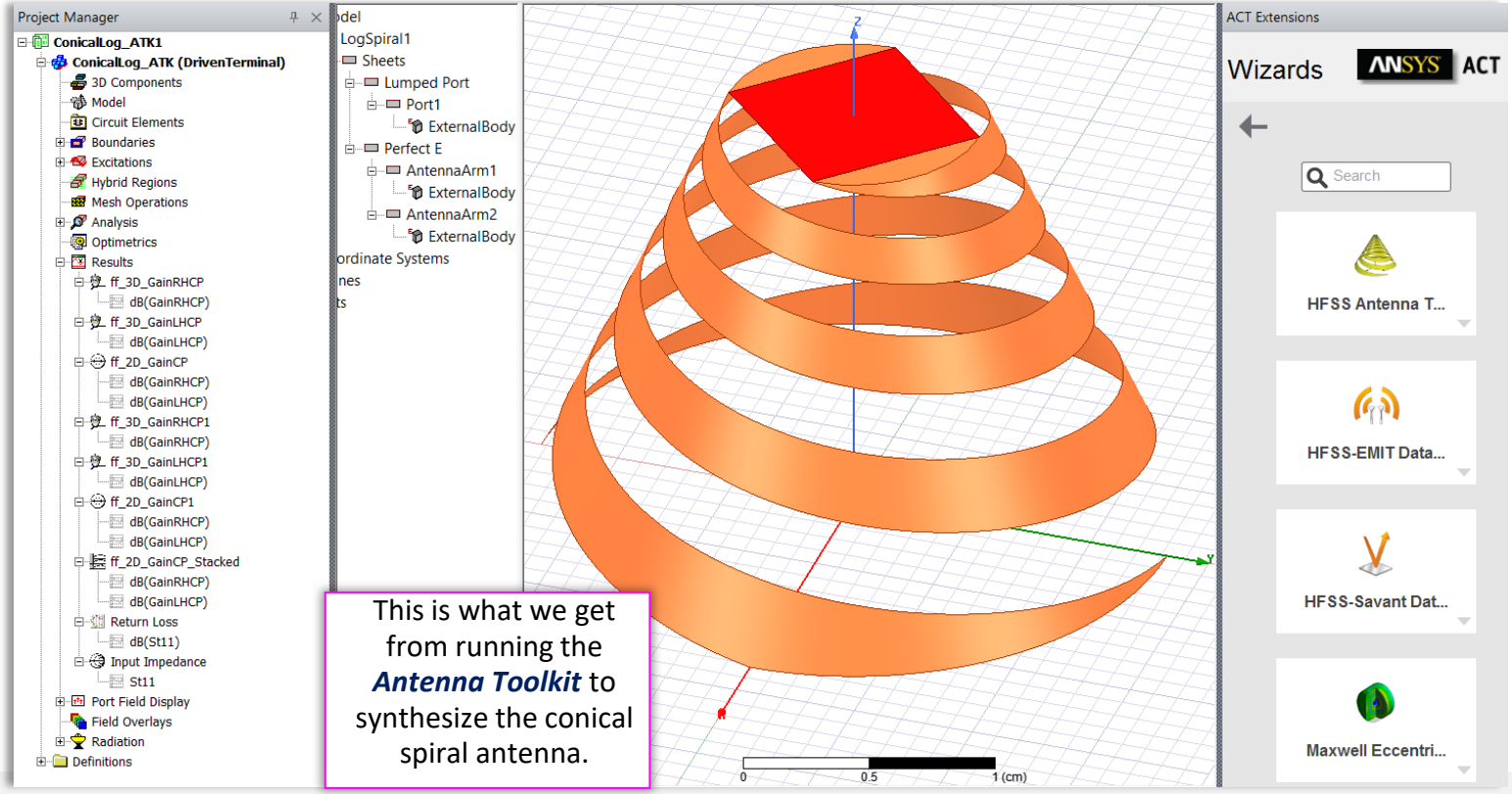


# Antenna Toolkit Conical Log Spiral Settings




- In the ***HFSS Antenna Toolkit***, under ***Antenna Type Conical Spiral,*** set (or verify):
  - ***Low Frequency: 4 GHz***
  - ***High Frequency: 10 GHz***
  - ***Outer Boundary: Radiation***
  - ***Inner Radius*: 0.53 cm**
  - ***Number of Turns: 2***
  - ***Offset Angle*: 90 deg**
  - **Expansion Coefficient: 1.58**
  - **Cone Height: 1.83 cm**
- Click on ***Finish*** to execute the toolkit operation. This may take a few seconds.
- Save File As **ConicalLog_ATK1.aedt**.

# Conical Spiral Project from Antenna Toolkit




This is what we get from running the ***Antenna Toolkit*** to synthesize the conical spiral antenna.

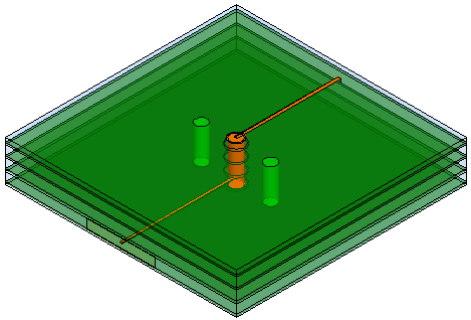
# Project Production 3

- Create geometry for single ended via, then assign appropriate boundary conditions and excitations to analysis and look at the results

# Single Ended Via

- This example is intended to demonstrate how to draw and parameterize geometry in HFSS 3D Modeler
  We will
  - Create geometry for single ended via
  - Assign appropriate boundary conditions and excitations
  - Run the analysis and look at the results

This geometry is of a Layout type. It is recommended to use Layout Design interface for drawing layout geometry and assigning boundary conditions and ports, and then exporting it into HFSS Design if necessary. The workflow that started in Layout Design UI is featured in the Workshop "Differential vias". However, this example provides ample practice in drawing techniques.



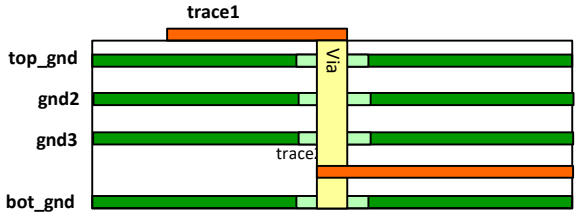
**Nominal Design:**

**Board:**
Thickness = 70 mils
$\varepsilon_r$ = 4.4 tan d = 0.02

**Ground:**
Thickness = 3.7 mils

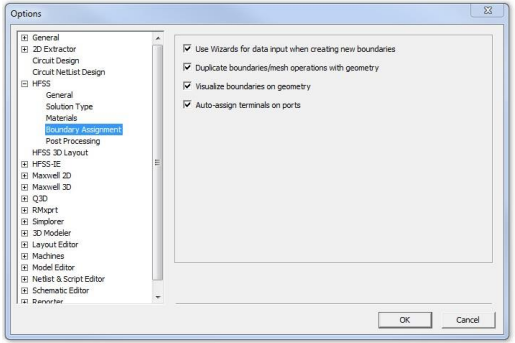
**Trace:**
Width = 4.75 mils
Thickness = 0.6 mils

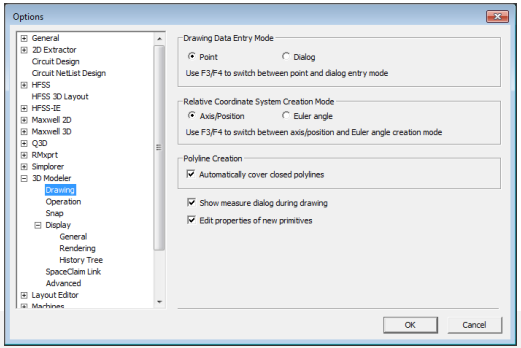
**Via:**
Diameter = 20 mils
Height = 70 mils



# HFSS: Getting Started

- **Launching ANSYS Electronics Desktop 2017.0**
  - To access HFSS, click the Microsoft Start button, Select *Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 18.0 > ANSYS Electronics Desktop 2017.0*
  - Setting Tool Options
    - Select the menu item *Tools > Options > General options*
      - Expand HFSS and select Boundary Assignment
        - Check all entries
    - Expand *3D Modeler*
      - Click Drawing
        - Automatically cover closed polylines: ☑ Checked
        - Edit properties of new primitives: ☑ Checked
      - Expand Display
        - Click Rendering and set Default Transparency to 0.7
        - Click History Tree and check all entries
      - Click the OK button

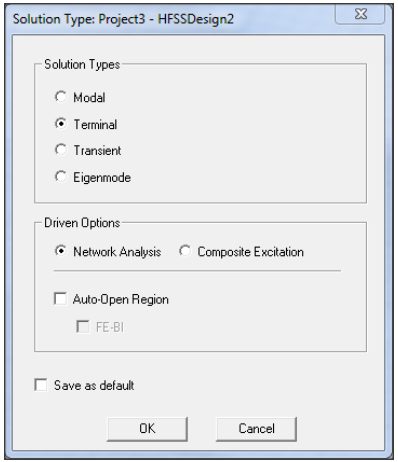


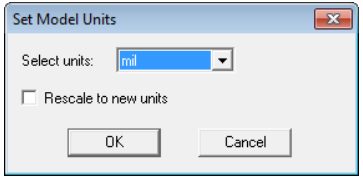




**Option settings suggested here ensure that the user can consistently follow the steps in the Workshop. A user can prefer his/her own option configuration**

# Creating the Design

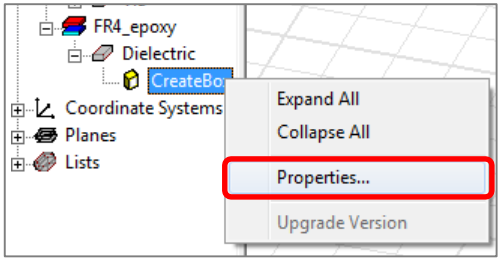
- **Opening the Project**
  - In Electronic Desktop, click the 🗋 icon on the Standard toolbar, or select the menu item *File > New*
  - From the Project menu, select Insert HFSS Design
- **Set Solution Type**
  - Select the menu item *HFSS > Solution Type*
    - Choose Driven Terminal / Network Analysis
    - Click the OK button
- **Set Model Units**
  - Select the menu item *Modeler > Units*
  - Note that the following is selected
    - Select Units: mil
    - Click the OK button
- **Set Default Material**
  - Using the 3D Modeler Materials toolbar, choose Select
    - Type FR4_epoxy in the Search by Name field
    - Click the OK button



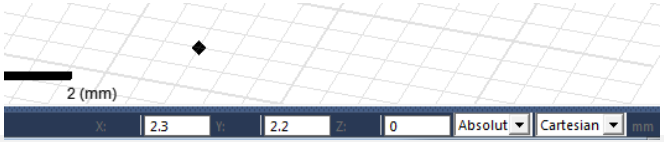

# Create Dielectric

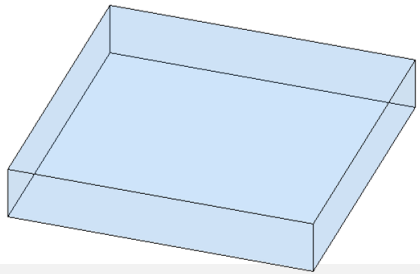
- Create Dielectric
  - Select menu item *Draw > Box*
  - Using the coordinate entry fields, enter the box position
    - X: -200, Y: -200, Z: 0.3, Press the Enter key
  - Using the coordinate entry fields, enter the opposite corner of the base rectangle
    - dX: 400, dY: 400, dz: -70, Press the Enter key
  - Select the Attribute tab from the Properties window
    - For the Value of Name type: Dielectric
    - Click the Ok button
  - To fit the view:
    - Select menu item *View > Fit All > Active View*. Or press CTRL+D




To access Attribute or Command Properties window of the object, right click either on the name of the object for Attribute or on the line below for Command and choose Properties

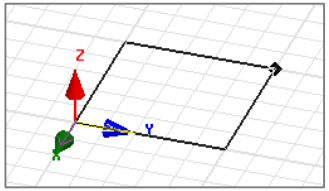
Tip: use Tab key to navigate between coordinate entry fields. Moving mouse cursor inside the Modeler window will change the values of coordinate entry fields.

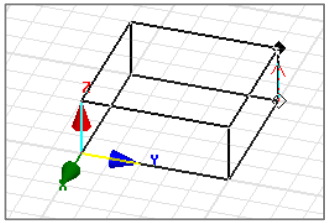


# Drawing box: alternative way

- Select menu item ***Draw > Box***
  - Left Mouse click anywhere in the Modeler window to draw the first corner of the box
  - Move mouse in XY plane and click again to draw the second corner of the box
  - Move mouse along Z axis and click again to draw the third corner of the box
  - Select the **Command** tab from the pop up Properties window
    - Edit position and sizes of the box
      - It is possible to use variables instead of values
    - Define Variables in the pop up windows as requested:




Properties: Project1 - HFSSDesign1 - Modeler

Command | Attribute

| | Name | Value | Unit | Evaluated Value | Description |
|---|---|---|---|---|---|
| | Command | CreateBox | | | |
| | Coordinate Sys... | Global | | | |
| | Position | Hom_length ,-a/2 ,-b/2 | | 5.2in , -0.45in , -0.2in | |
| | XSize | WG_lenght | | 1in | |
| | YSize | a | | 0.9in | |
| | ZSize | b | | 0.4in | |

Show Hidden

OK | Cancel | Apply

Add Variable

Name: a

Unit Type: Length

Unit: in

Value: 0.9

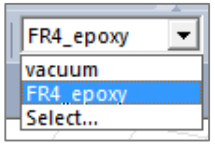
Type: Local Variable

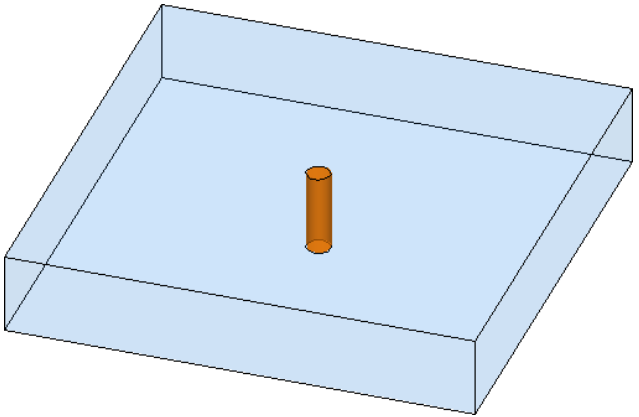
OK | Cancel

This example relies on method that uses coordinate entry fields. This is an alternative method you may use during exercise as well to draw any part of geometry.

# Create Via

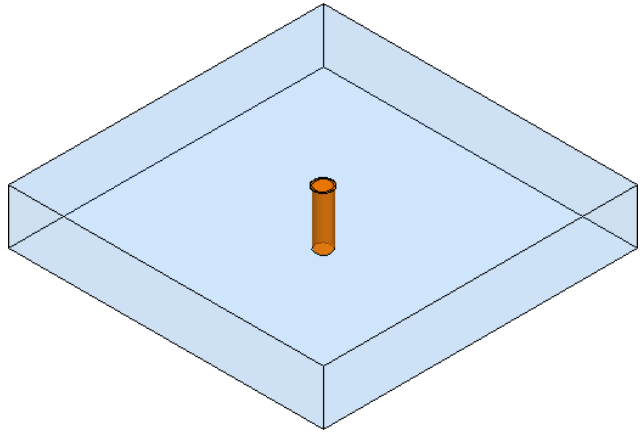
- Set Default Material
  - Using the 3D Modeler Materials toolbar, choose Select
    - Type copper in the Search by Name field
    - Click the OK button
- Create Via
  - Select menu item *Draw > Cylinder*
  - Using the coordinate entry fields, enter the center of the base
    - X: 0.0, Y: 0.0, Z: 0.0, Press the Enter key
  - Using the coordinate entry fields, enter the radius
    - dX: 0.0, dY: 10.0, dZ: 0.0 Press the Enter key
  - Using the coordinate entry fields, enter the height:
    - dX: 0.0, dY: 0.0, dZ: -70 Press the Enter key
    - Click the OK button
  - Select the Attribute tab from the Properties window.
    - For the Value of Name type: Via
  - Select the Command tab from the Properties window.
    - Click the OK button

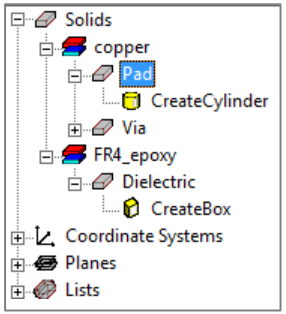


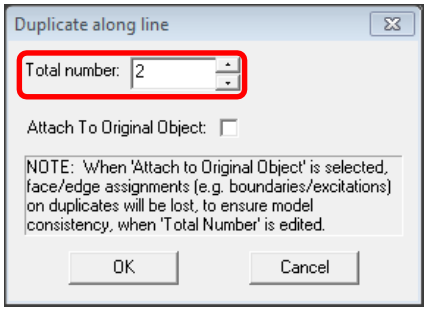



# Create Via Pad

- Create Via Pad
  - Select menu item *Draw > Cylinder*
  - Using the coordinate entry fields, enter the center of the base
    - X: 0.0, Y: 0.0, Z: -0.3, Press the Enter key
  - Using the coordinate entry fields, enter the pad radius
    - dX: 0.0, dY: 12.0, dZ: 0.0 Press the Enter key
  - Using the coordinate entry fields, enter the height:
    - dX: 0.0, dY: 0.0, dZ: 0.6 Press the Enter key
  - Select the Attribute tab from the Properties window.
    - For the Value of Name type: Pad
    - Click the OK button
- Duplicate the object
  - From the model tree, highlight Pad
  - Select menu item *Edit > Duplicate > Along Line*
  - Using the coordinate entry fields, enter the box position
    - X: 0.0, Y: 0.0, Z: 0.0, Press the Enter key
    - dX: 0.0, dY: 0.0, dZ: -58, Press the Enter key
    - Total Number: 2
    - Click the OK button
  - Click the OK button when Properties window appears

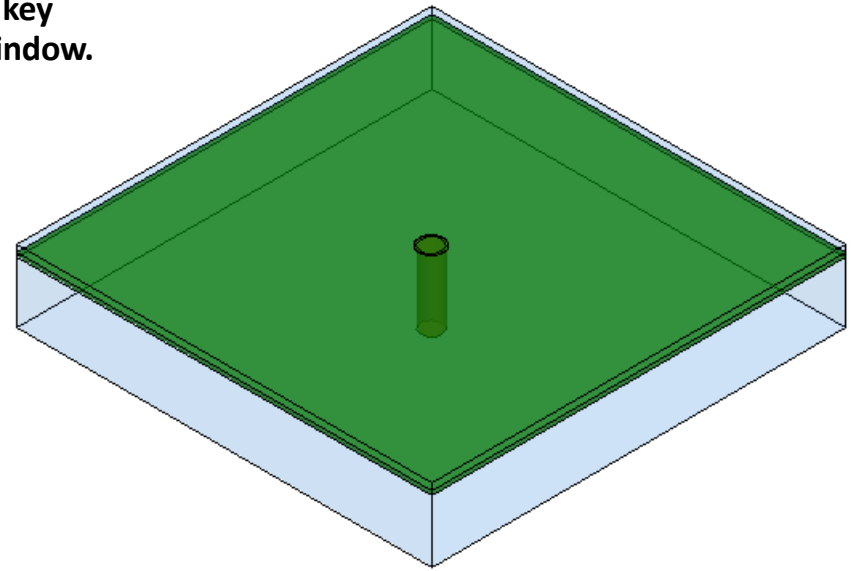




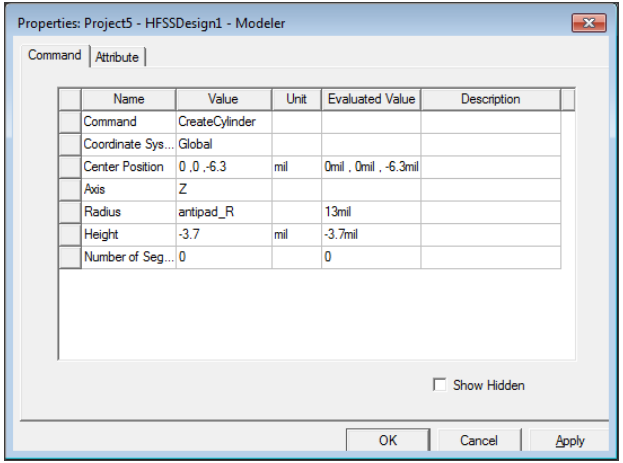
# Create Ground Plane

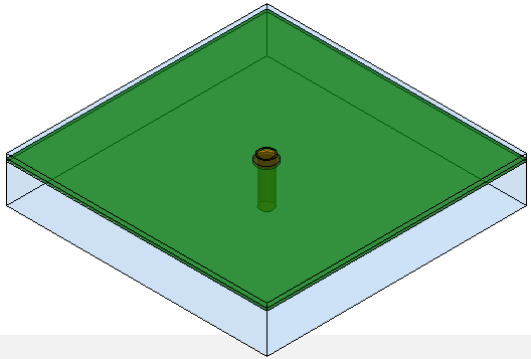
- Create Ground Plane
  - Select menu item *Draw > Box*
  - Using the coordinate entry fields, enter the center of the base
    - X: -200.0, Y: -200.0, Z: -6.3, Press the Enter key
  - Using the coordinate entry fields, enter the box dimensions
    - dX: 400.0, dY: 400.0, dZ: -3.7 Press the Enter key
  - Select the Attribute tab from the Properties window.
    - For the Value of Name type: Gnd_plane1
    - Click the OK button

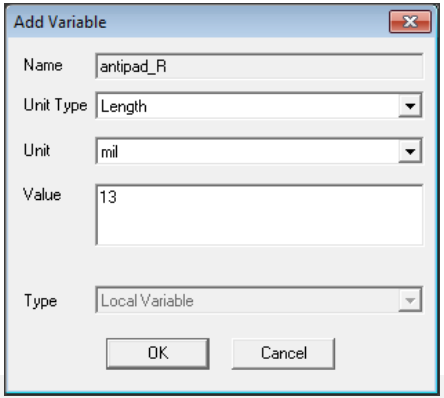


# Create Antipad

- Create Ground plane Antipad
  - Select menu item *Draw > Cylinder*
  - Using the coordinate entry fields, enter the center of the base
    - X: 0.0, Y: 0.0, Z: -6.3, press the Enter key
  - Using the coordinate entry fields, enter the pad radius
    - dX: 0.0, dY: 13.0, dZ: 0.0 press the Enter key
  - Using the coordinate entry fields, enter the height:
    - dX: 0.0, dY: 0.0, dZ: -3.7 press the Enter key
  - Select the Attribute tab from the Properties window.
    - For the Value of Name type: Antipad
  - Select the Command tab from the Properties window.
    - For Radius Value type: antipad_R
    - Add Variable Window
      - Unit Type: Length
      - Unit: mil
      - Value: 13 mil
      - Click the OK button
    - Click the OK button

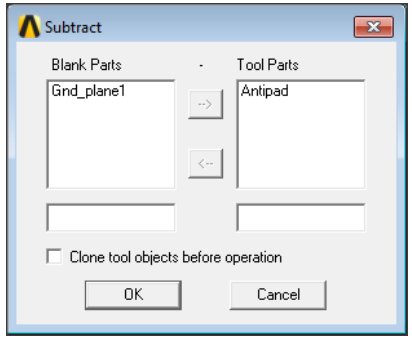


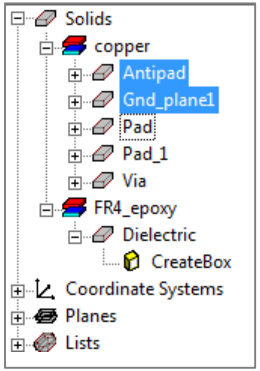





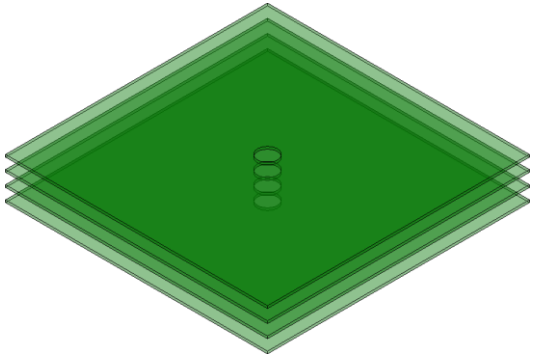
# Create Antipad

- **Subtract objects**
  - In the model tree, highlight: Gnd_plane1, Antipad
  - Select menu item *Modeler > Boolean > Subtract*
  - In the subtract window
    - Blank Parts: gnd_plane1
    - Tool Parts: Antipad
    - Clone tool parts before subtracting: ☐ Unchecked
    - Use the arrows to move the objects to the appropriate column
    - Click the OK button



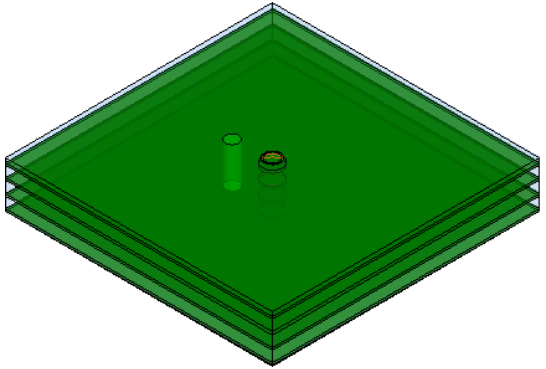



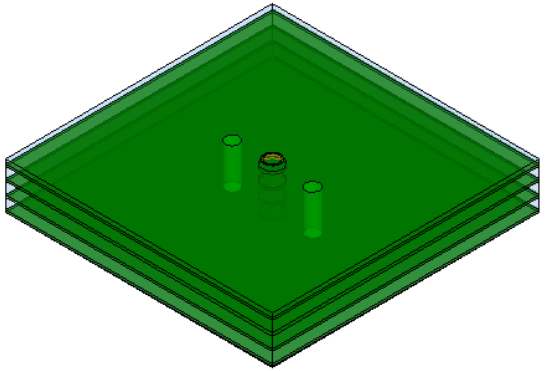

- **Duplicate the object**
  - From the model tree, highlight Gnd_plane1
  - Select menu item *Edit > Duplicate > Along Line*
  - Using the coordinate entry fields, enter the box position
    - X: 0.0, Y: 0.0, Z: 0.0, Press the Enter key
    - dX: 0.0, dY: 0.0, dZ: -20, Press the Enter key
    - Total Number: 4
    - Click the OK button
  - Click the OK button when Properties window appears



# Create Ground Via

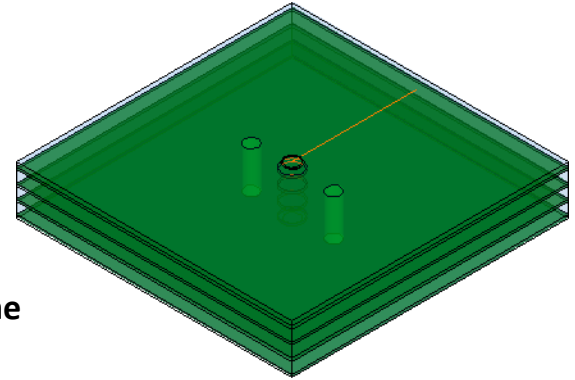
- Create Ground Via
  - Select menu item *Draw > Cylinder*
  - Using the coordinate entry fields, enter the center of the base
    - X: -60.0, Y: 0.0, Z: -6.3, Press the Enter key
  - Using the coordinate entry fields, enter the ground via radius
    - dX: 0.0, dY: 10.0, dZ: 0.0 Press the Enter key
  - Using the coordinate entry fields, enter the height:
    - dX: 0.0, dY: 0.0, dZ: -60 Press the Enter key
  - Select the Attribute tab from the Properties window
    - For the Value of Name type: gnd_via1
    - Click the OK button
- Duplicate the object
  - From the model tree, highlight gnd_via1
  - Select menu item *Edit > Duplicate > Along Line*
  - Using the coordinate entry fields, enter the box position
    - X: 0.0, Y: 0.0, Z: 0.0, Press the Enter key
    - dX: 120.0, dY: 0.0, dZ: 0.0, Press the Enter key
    - Total Number: 2
    - Click the OK button
    - Click the OK button when Properties window appears

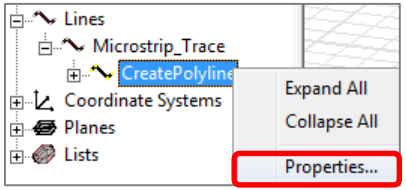


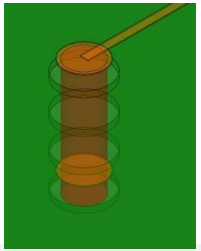


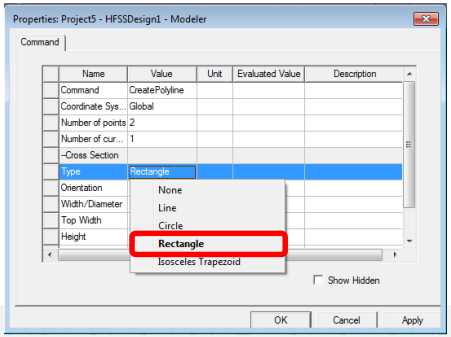
# Create Microstrip

- Create Microstrip Via Trace
  - To create the path, select menu item *Draw > Line*
    - Using the coordinate entry fields, enter the vertex point:
      - X: 0.0, Y: 0.0, Z: 0.0 Press the Enter key
    - Using the coordinate entry fields, enter the vertex point
      - X: 0.0, Y: 180, Z: 0.0 Press the Enter key
    - Using the mouse, right-click anywhere in the Modeler window and select Done
  - To set the name, select the Attribute tab from the Properties window
    - For the Value of Name type: Microstrip_Trace
    - Click the OK button
- To assign trace width and thickness
  - In Modeler Tree expand trace and right-click on Create Polyline, choose Properties
  - In Properties window change Type to Rectangle
  - Width/Diameter: 4.75 mil
  - Height: 0.6 mil
  - Click the OK button



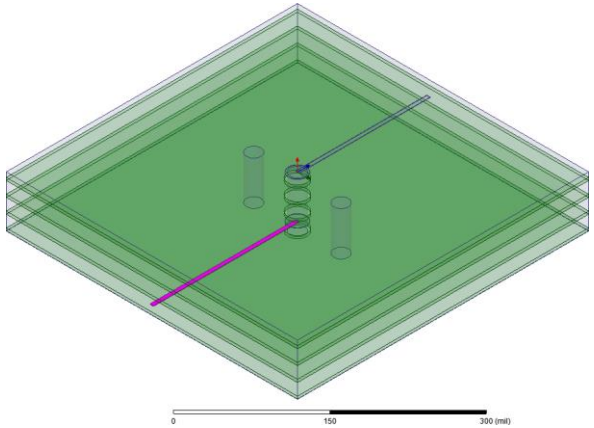




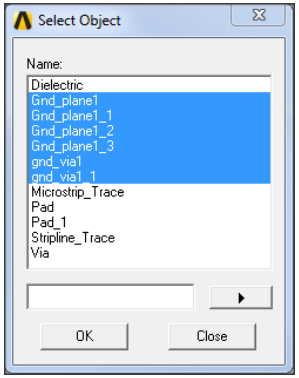
# Create Stripline

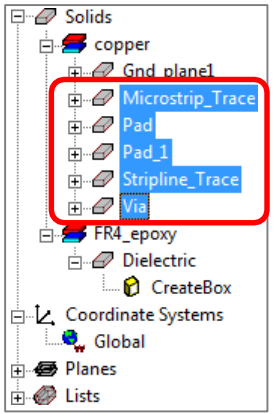
- Create Stripline Via Trace
  - To create the path, select menu item *Draw > Line*
    - Using the coordinate entry fields, enter the vertex point:
      - X: 0.0, Y: 0.0, Z: -58 Press the Enter key
    - Using the coordinate entry fields, enter the vertex point
      - X: 0.0, Y: -200, Z: -58 Press the Enter key
    - Using the mouse, right-click anywhere in the Modeler window and select Done
  - To set the name, select the Attribute tab from the Properties window
    - For the Value of Name type: Stripline_Trace
    - Click the OK button when the Properties dialog appears
- To assign trace width and thickness
  - In Modeler Tree expand trace and right-click on Create Polyline, choose Properties
  - In Properties window change Type to Rectangle
  - Width/Diameter: 4.75 mil
  - Height: 0.6 mil
  - Click the OK button
- To fit the view:
  - Select menu item *View > Fit All > Active View*. Or press CTRL+D



# Unite Objects

- Unite Objects
  - To select all ground planes and ground vias
    - Select menu item *Edit > Select > By Name*
    - Select Object Dialog,
      - Select the objects named: Gnd_plane1, Gnd_plane1_1, Gnd_plane1_2, Gnd_plane1_3, gnd_via1 and gnd_via1_1
      - Click the OK button
      - Select the menu item *Modeler > Boolean > Unite*
  - Signal Via
    - In the model tree, highlight: Via, and Pad_1, Pad_1, Microstrip_Trace and Stripline_Trace
    - Select menu item *Modeler > Boolean > Unite*
- To fit the view:
  - Select menu item *View > Fit All > Active View*. Or press CTRL+D

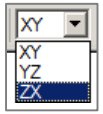


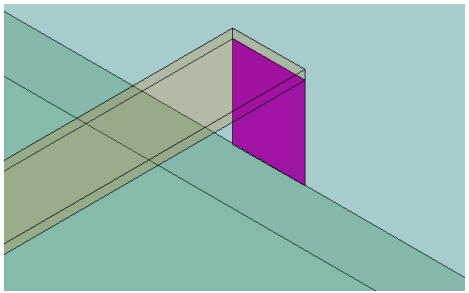

Note that objects created as the result of uniting several objects has the name of the object that was selected first.

# Excitation: Create Port

- Create Microstrip Trace Lumped Port Excitations
  - To create lumped port rectangle:
    - Select menu item *Modeler > Grid Plane > XZ*
    - Select menu item *Draw > Rectangle*
    - Using the coordinate entry fields, enter the vertex point:
      - X: -2.375 , Y: 180.0, Z: -0.3 Press the Enter key
    - Using the coordinate entry fields, enter the vertex point
      - X: 4.75, Y: 0.0, Z: -6.0 Press the Enter key
  - To set the name:
    - Select the Attribute tab from the Properties window
    - For the Value of Name type: Port1
    - Click the OK button when the Properties dialog appears



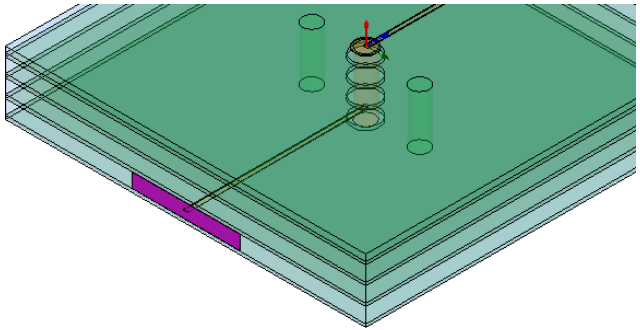

In this design, it is necessary to create a separate geometry for port. In Layout Design, this geometry is created automatically during port assignment



- To fit the view:
  - Select menu item *View > Fit All > Active View*. Or press CTRL+D

# Create second Port: Stripline Trace Wave Port Excitations

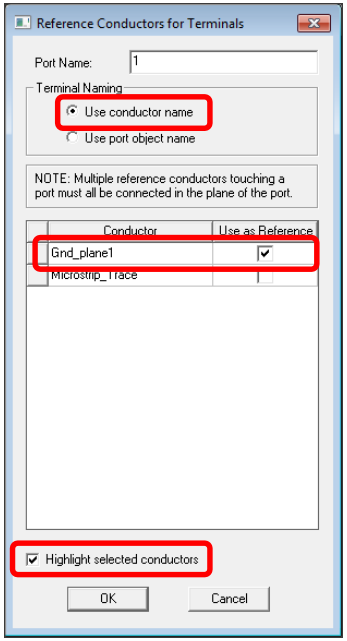
- Create Offset Coordinate System
  - Select menu item *Modeler > Coordinate System > Create > Relative CS > Offset*
  - Using the coordinate entry fields, enter the origin
    - X: -60.0, Y: -200.0, Z: -50.0, Press the Enter key
- To create wave port rectangle:
  - Select menu item *Draw > Rectangle*
  - Using the coordinate entry fields, enter the vertex point:
    - X: 0.0, Y: 0.0, Z: 0.0 Press the Enter key
  - Using the coordinate entry fields, enter the vertex point
    - X: 120.0, Y: 0.0, Z: -16.3 Press the Enter key

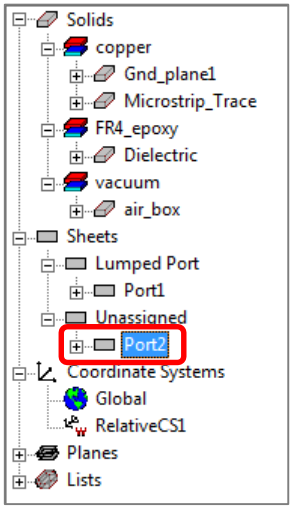


  - To set the name:
    - Select the Attribute tab from the Properties window.
    - For the Value of Name type: Port2
    - Click the OK button when the Properties dialog appears
- To fit the view:
  - Select menu item *View > Fit All > Active View*. Or press the CTRL+D key

# Assign Excitations

- Assign Lumped Port Excitation
  - In the Modeler Tree under Sheets highlight Port1
  - To assign lumped port excitation
    - Select menu item *HFSS > Excitations > Assign > Lumped Port*
    - Port Name: 1
    - Conductor: Gnd_plane1
      - Use as Reference: ☑ Checked
    - Highlight Selected conductors: ☑ Checked
      - Click the OK button
- Assign Wave Port Excitation
  - In the Modeler Tree under Sheets highlight Port2
  - Select menu item *HFSS> Excitations> Assign> Wave Port…*
    - Port name: 2
    - Terminal naming: Use Conductor Name
    - Conductor: Gnd_plane1
      - Use as reference: ☑ Checked
    - Highlight Selected Conductors: ☑ Checked
    - Click the OK button

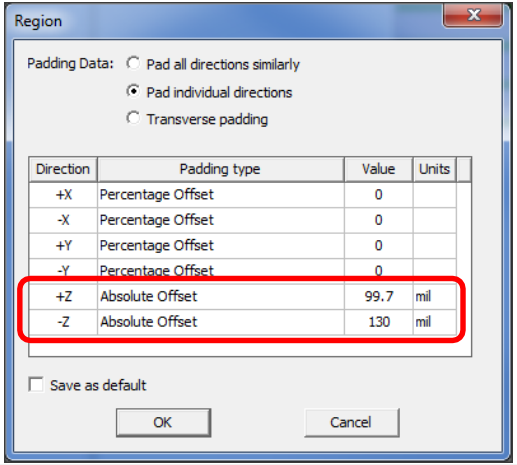



Wave port is typically on the external part of the geometry and the lumped port internal part of the geometry

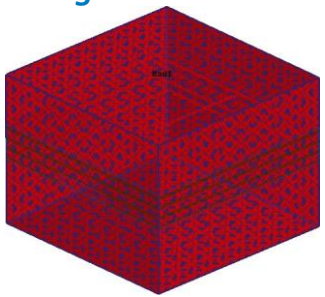
# Boundary Conditions: First - Create Airbox

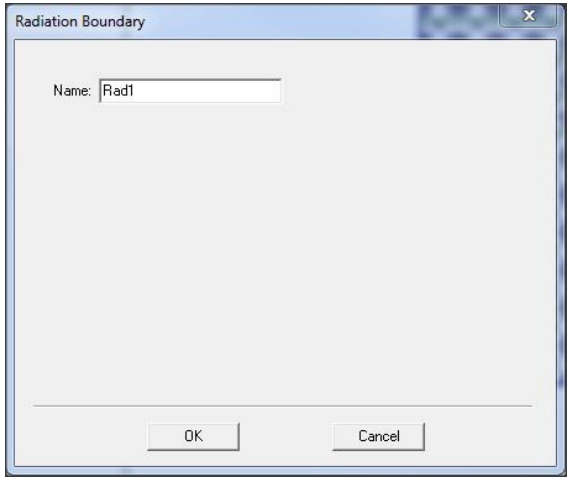
- Create Region
  - Select menu item *Draw > Region* or click the  icon on the tool bar
  - Padding is only needed in the z direction so, select Pad individual directions
    - For +Z, change Padding Type from the drop down menu to Absolute Offset, with value 99.7 and units in mil
    - For -Z, change Padding Type from the drop down menu to Absolute Offset, with value 130 and units in mil
    - Click the OK button
  - To fit the view:
    - Select menu item *View > Fit All > Active View*. Or press CTRL+D



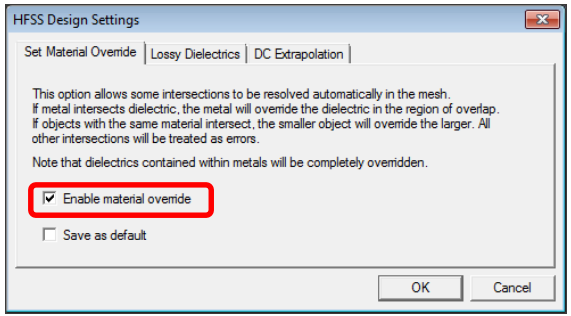
# Boundary Conditions: Assign Radiation Boundary

- **Assign Radiation Boundary**
  - Select menu item *Edit> Select> By Name*
    - Object: Region depending on which method you chose to create the Airbox
    - Click the OK button
  - Select menu item *HFSS> Boundaries> Assign> Radiation…*
    - Name: Rad1
    - Click the OK button



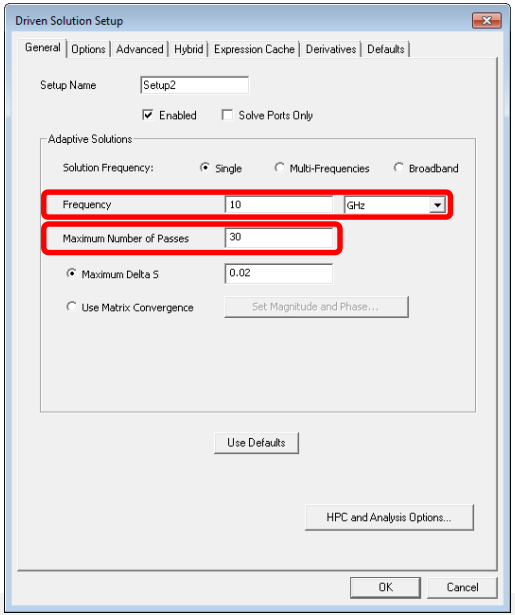



- **Set Material Override**
  - This option will allow metals to override dielectric objects and it will also allow overlaps between objects of same material properties. This eliminates the need to subtract metal objects from dielectric objects.
    - Select menu item *HFSS > Design Settings*
      - Enable Material Override: ☑ Checked



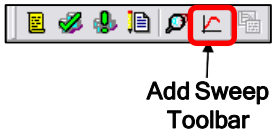
# Solution Setup

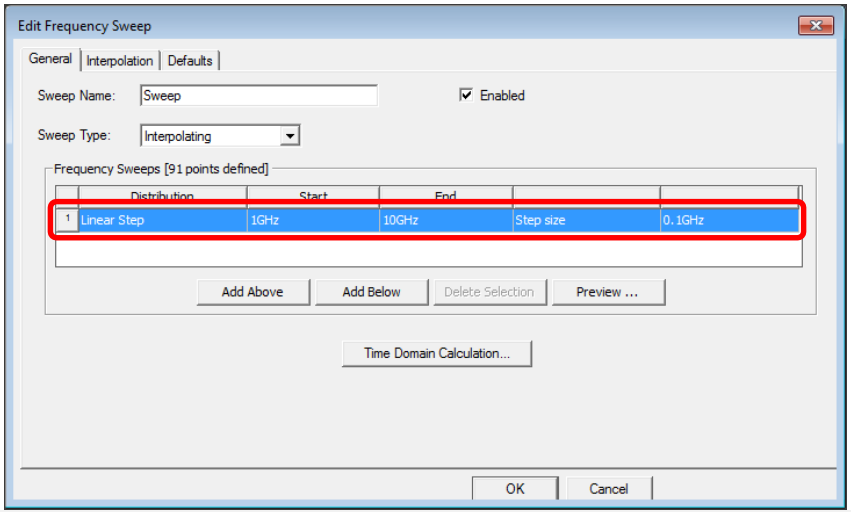
- Creating an Analysis Setup
  - Select menu item *HFSS > Analysis Setup > Add Solution Setup*
    - Click the General tab:
      - Solution Frequency: 10 GHz
      - Maximum Number of Passes: 30, to allow for enough iterations to reach convergence.
    - Click the OK button

# Add Frequency Sweep

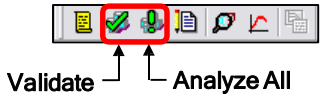
- Adding a Frequency Sweep
  - Select menu item *HFSS > Analysis Setup > Add Sweep*
    - Select: Solution Setup:Setup1
    - Click the OK button
  - Edit Sweep Window:
    - Sweep Type: Interpolating
    - Frequency Setup Type: Linear Step
      - Start: 1.0 GHz
      - Stop: 10.0 GHz
      - Step: 0.1 GHz
    - Click the OK button



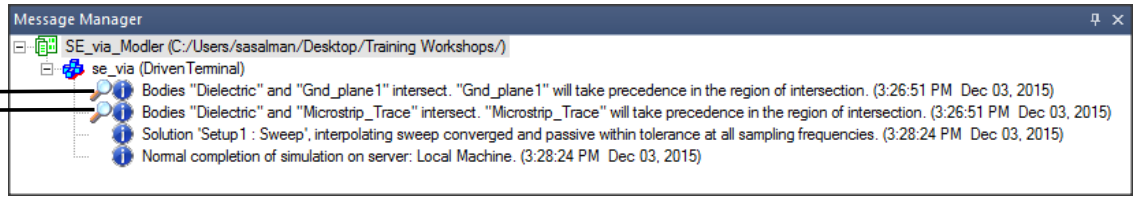
# Analyze

- Save Project
  - Select menu item *File > Save As*
    - Filename: se_via.aedt
    - Click the Save button




- Model Validation
  - Select menu item *HFSS > Validation Check*
    - Click the Close button
      - Note: To view any errors or warning messages, use the Message Manager

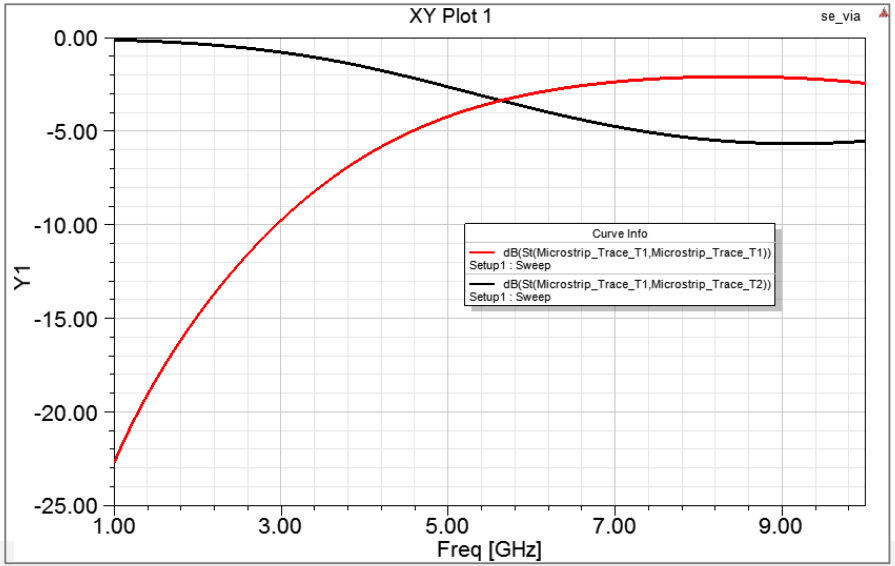
Material Override at work




- Analyze
  - Select menu item *HFSS > Analyze All*

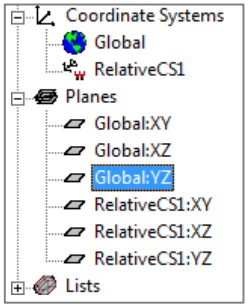
# Viewing Results

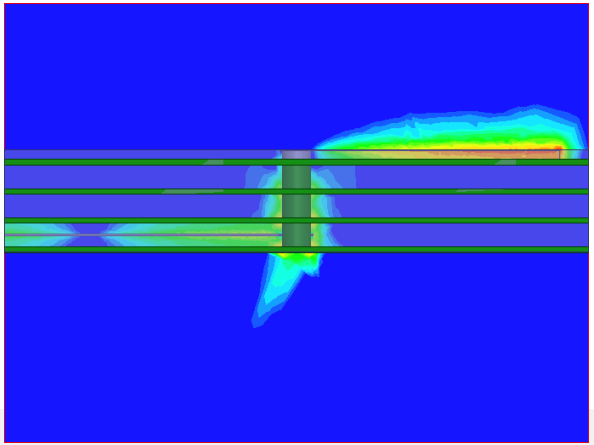
- Create Reports
  - Select menu item *HFSS > Results > Create Terminal Solution Data Report > Rectangular Plot*
    - Solution: Setup1:Sweep
    - Domain: Sweep
      - Category: Terminal S Parameter
      - Quantity: St(Microstrip_Trace_T1, Microstrip_Trace_T1), St(Microstrip_Trace_T1, Microstrip_Trace_T2))
      - Function: dB
      - Click New Report button
    - Click Close button



# Post Processing – Field Overlay

- Return to 3D modeler
  - To return to the 3D modeler window, select the menu item *Window > 1 se_via – HFSS Design1 - Modeler*
- Create Field Overlay
  - From the 3D Model tree, expand the Planes and select the Global:YZ
  - Select menu item *HFSS > Fields > Plot Fields > E > Mag_E*
    - Solution: Setup1:LastAdaptive
    - Quantity: Mag_E
    - Click the Done button
  - Select menu item *HFSS > Fields > Modify Plot Attributes*
    - Select Plot Folder Window, click the OK button
    - E-Field Window:
      - Click the Scale tab
        - Scale: Log
        - Limit Max/Min precision: ☑ Checked
        - digits: 2
      - If real time mode is not checked, click the Apply button
    - Click the Close button
  - To Animate the field plot:
    - Select menu item *HFSS > Fields> Animate*
      - Click the OK button





## 思渤科技股份有限公司

新竹市公道五路二段178號5樓

www.cybernet-ap.com.tw

+886-3-611-8668