Ansys CERTIFIED ELITE CHANNEL PARTNER

Antenna Array Simulation within HFSS

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

Ansys 2021 R1
Engineering What's Ahead.

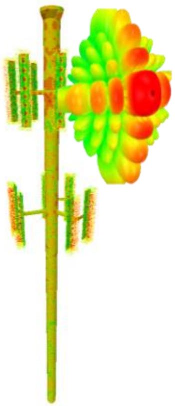
Solve it, with CYBERNET

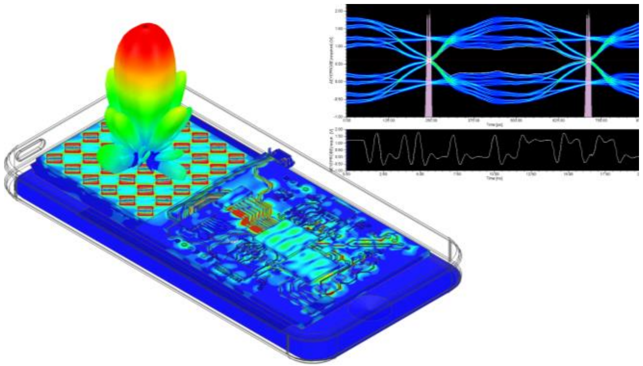
Today File

- https://github.com/ChangNicholas/20210326_HFSS

Outline

- Introduction
- 3D Component Array
- Lab 1. Estimate Regular Array Performance with Unit Cell
- Lab 2. Build Finite Regular Array
- Lab 3. Build 3D Component Array

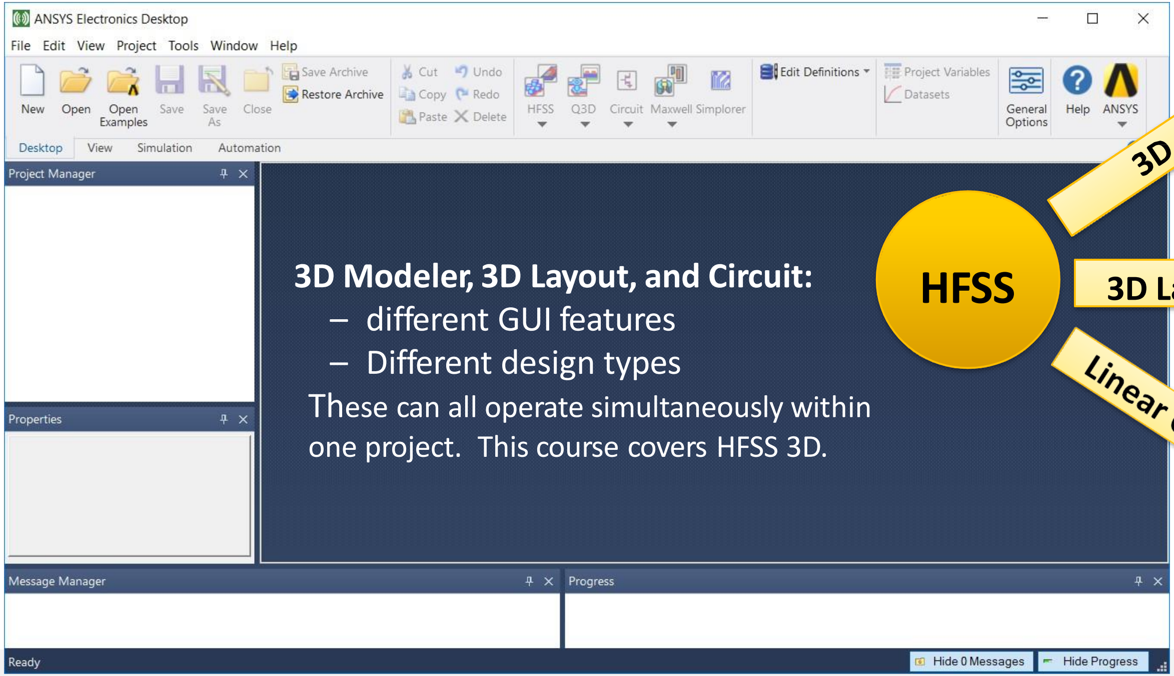




Introduction

HFSS in ANSYS Electronics Desktop (AEDT)

3 Basic Interfaces - 1 Desktop



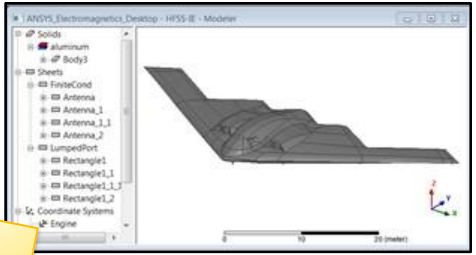
3D Modeler, 3D Layout, and Circuit:

- different GUI features
- Different design types

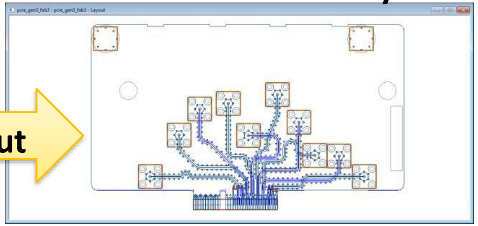
These can all operate simultaneously within one project. This course covers HFSS 3D.

HFSS → 3D · 3D Layout · Linear Circuit

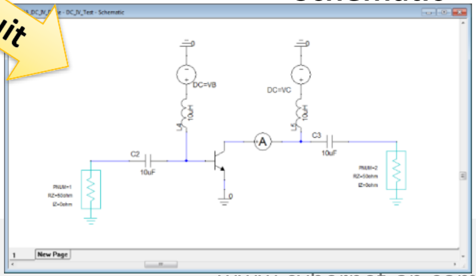
3D Modeler



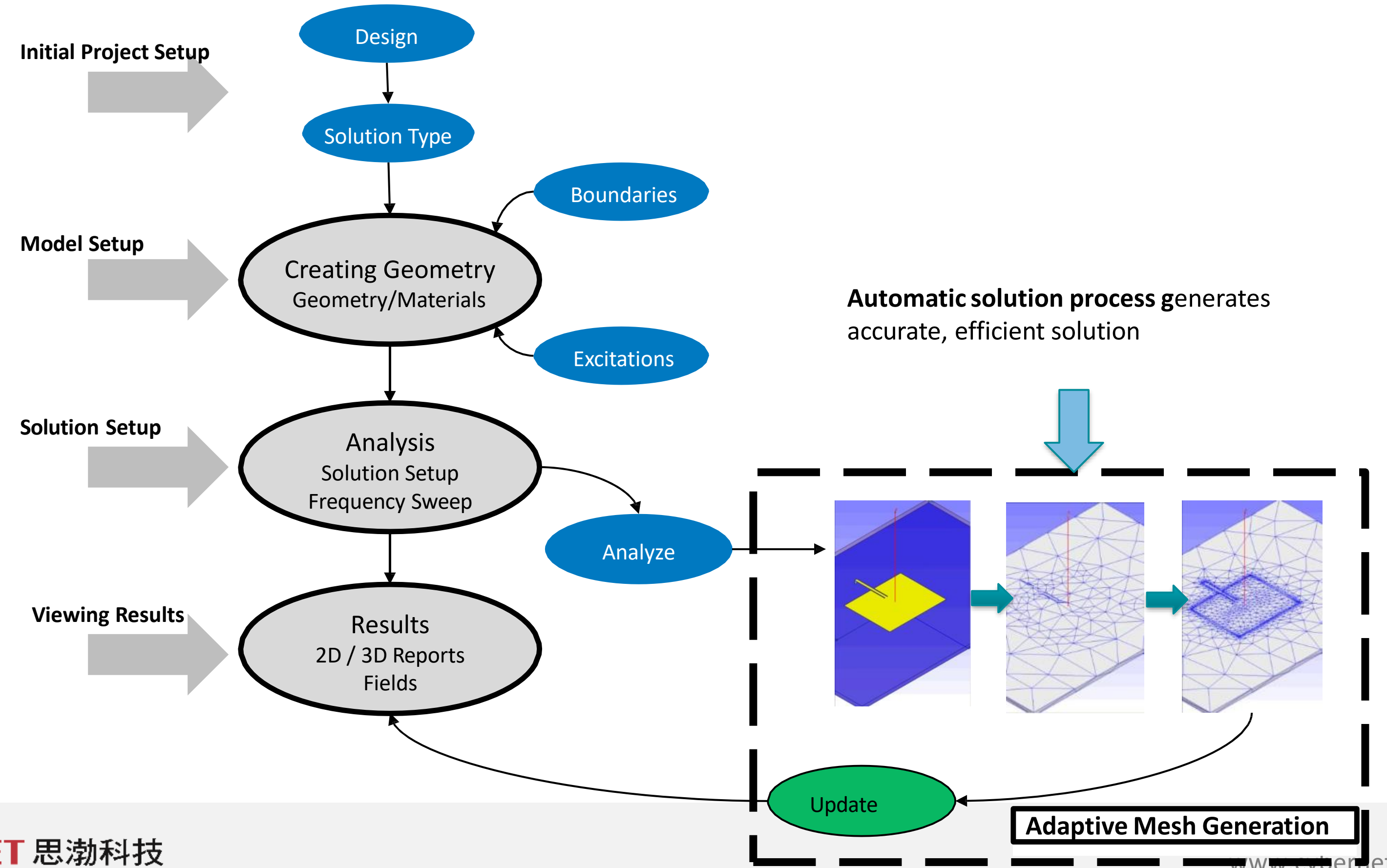
3D Layout



Schematic

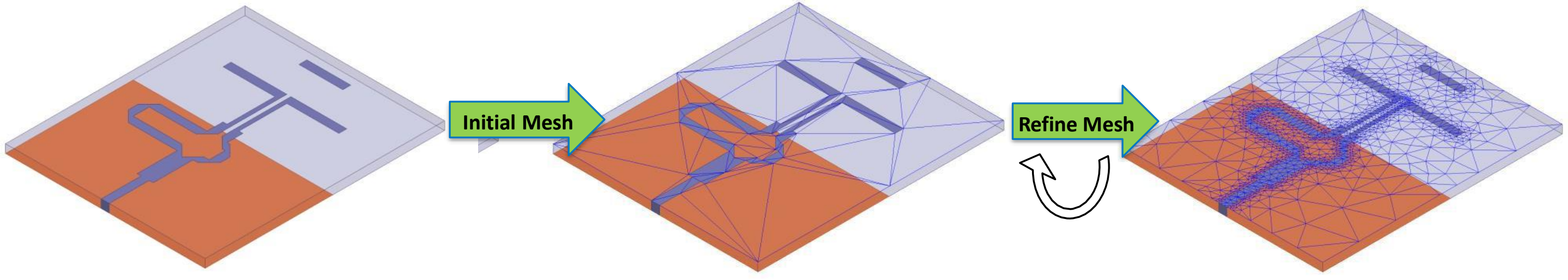


HFSS – Overview of Solution Process



Automatic solution process generates accurate, efficient solution

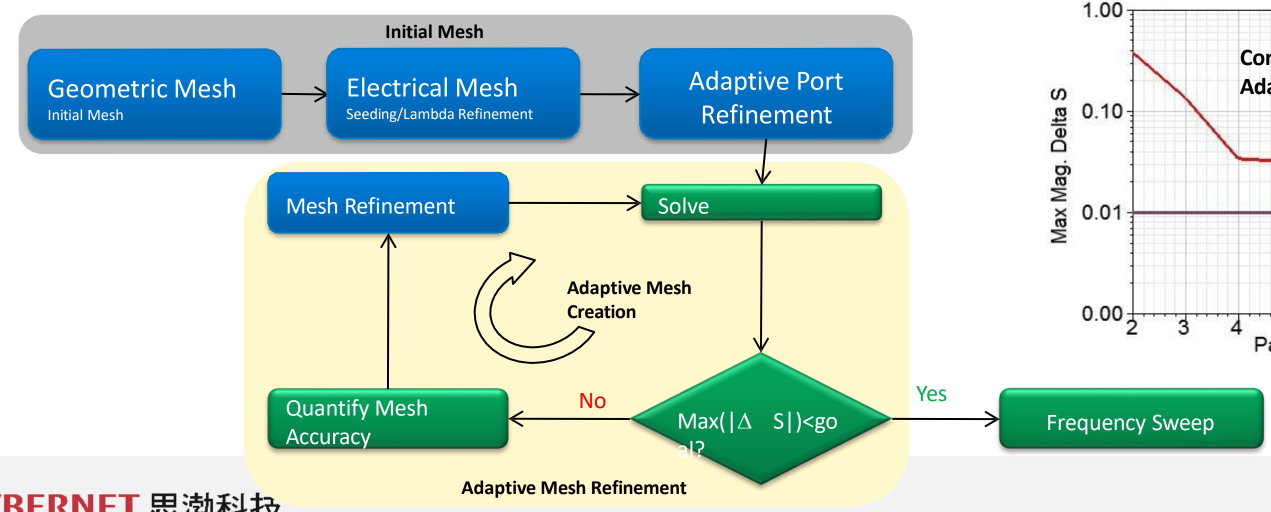
HFSS – Automated solution process

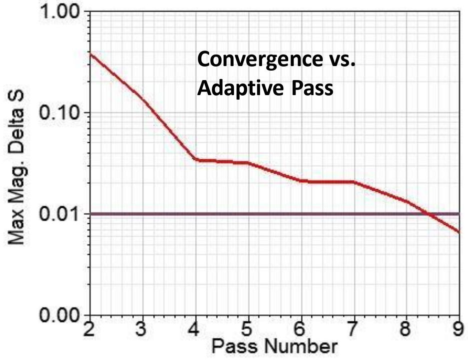


Geometry

Initial Mesh

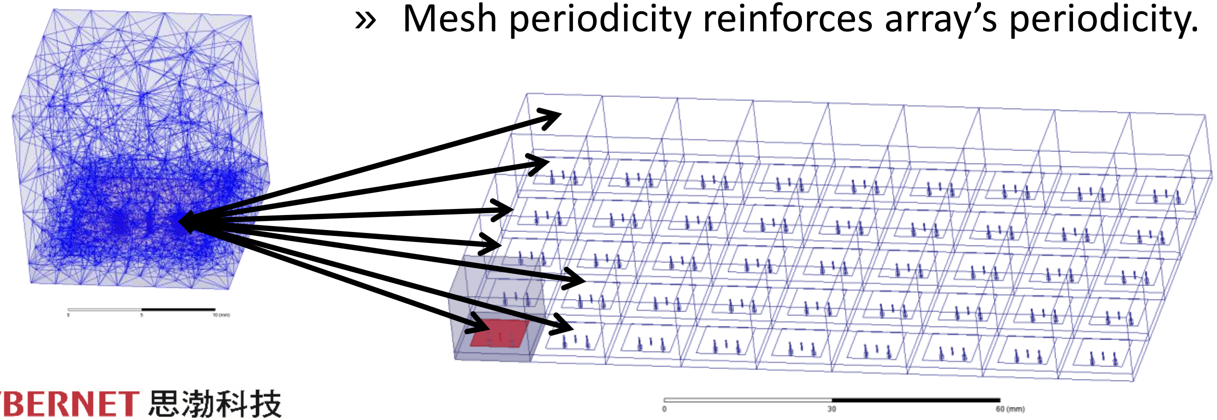
Converged Mesh





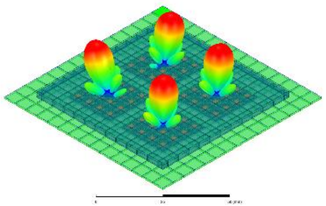
Finite Array Domain Decomposition

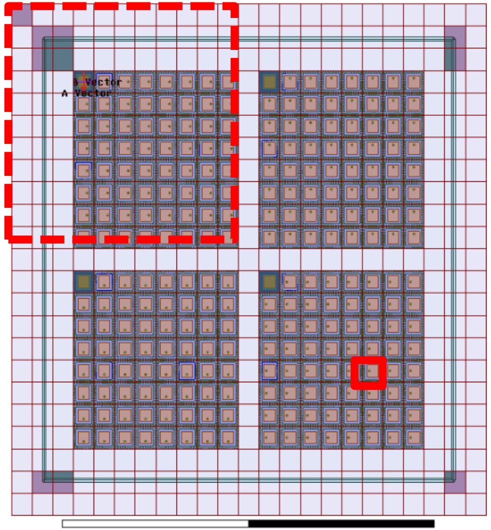
- Utilizes Replicated DDM Unit Cell to Address Array Concerns
- Geometry and Mesh copied directly from Unit Cell Model
  - Unit Cell geometry expanded to finite array through a simple GUI
  - Adaptive Meshing Process imported from Unit Cell Simulation
    - Dramatically reduces the meshing time associated with finite array analyses.
    - Mesh periodicity reinforces array's periodicity.

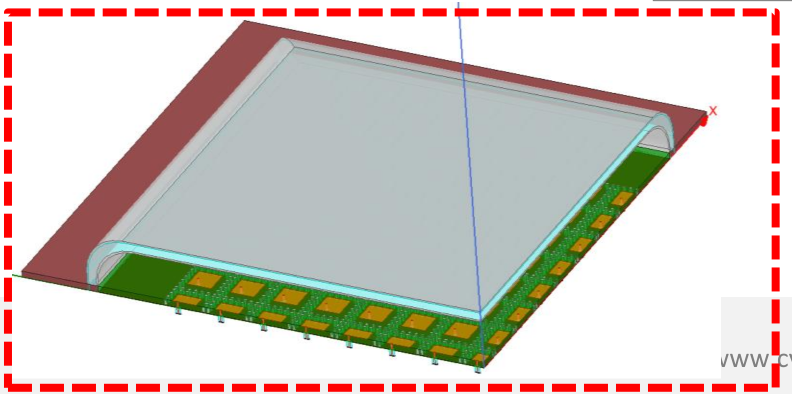


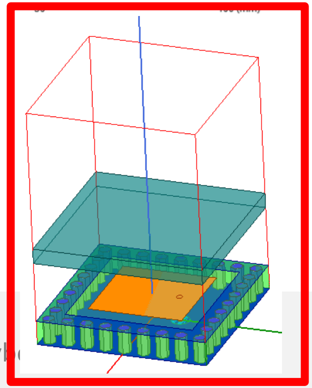
3D Component Array

- **256 Element Example**
  - Four times 8x8 array for 5G (28 GHz)
  - Radome covering all panels
  - Radome is bended to touch the ground plane
  - Each Array panel is separated with an antenna element size ground plane
  - Includes finite ground size (size of ground plane extension from antenna edge)
- **Full wave (FEM) Simulation time**
  - **Adaptive Mesh included**
  - **32 Cores: 3 hours**
  - **128 Cores: Approximately 1 hour**

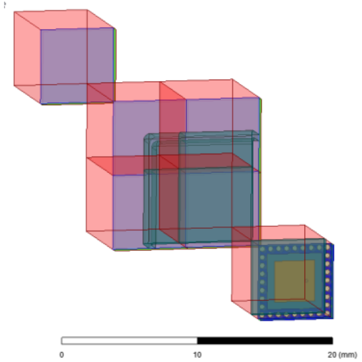








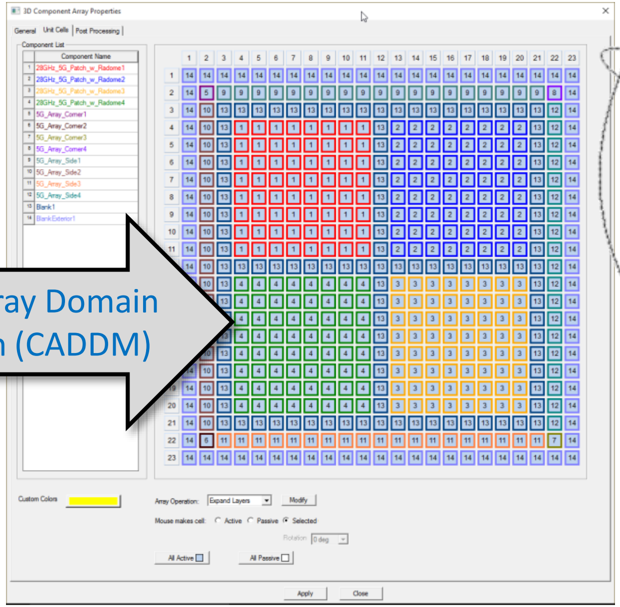
Array Antenna Analysis in HFSS

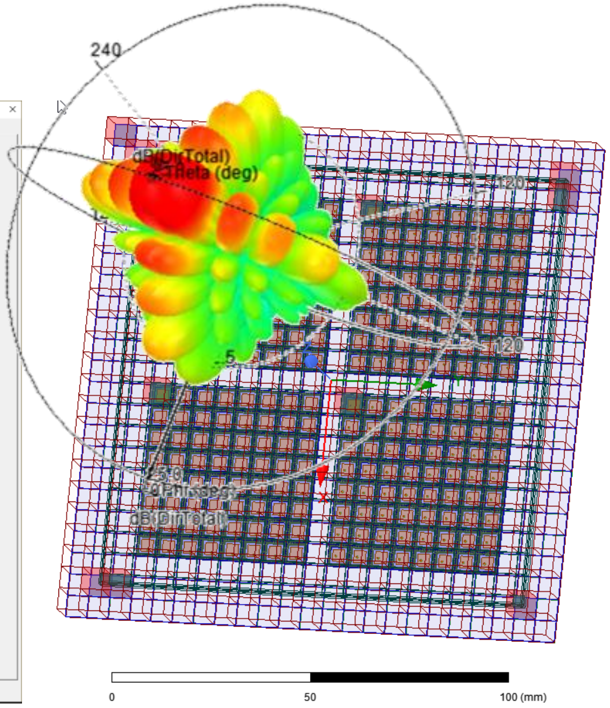


Unit Cells

- Models a single, repeated buildi[ng block] for a large array
- Low computational resources
- Can model finite edge effects, multiple element types on lattice, dummy elements and element failure effects

Component Array Domain Decomposition (CADDM)

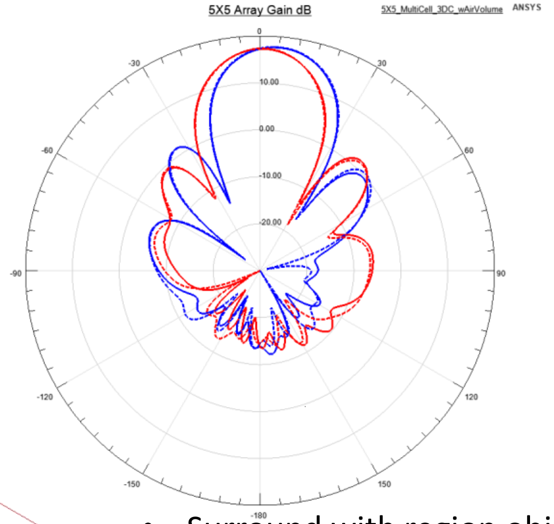




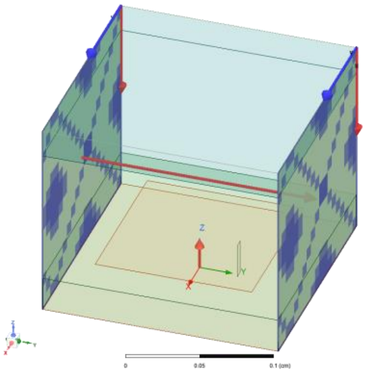
Compute scanned beam/directivity

3D Component Array: Global Region Volume

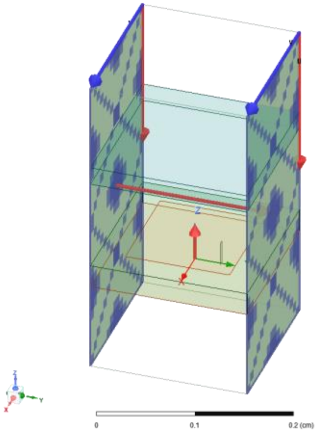
- Previously, 3D Component Array Unit cells required the surrounding radiation volume and ABC
- Now only requires unit cell volume and geometry
  - e.g. antenna element plus radome
  - Unit cell volume for defining lattice pairs and vectors



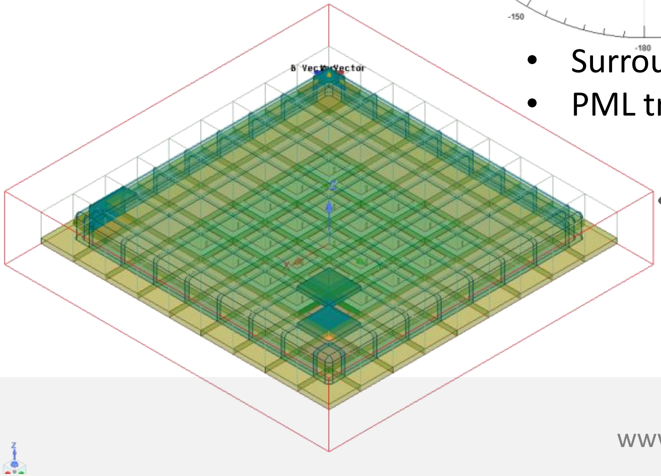
Unit Cell Only New!



Unit Cell and Rad. Vol.



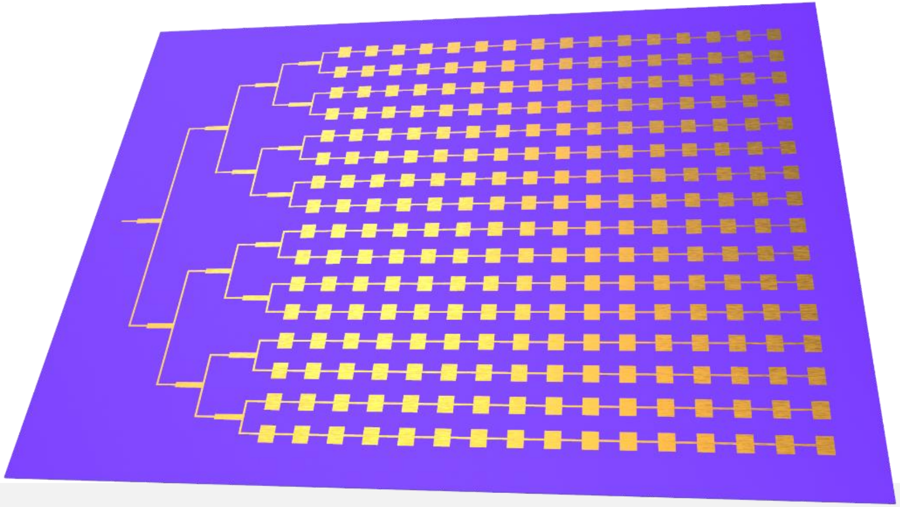
New in 2021 R1!

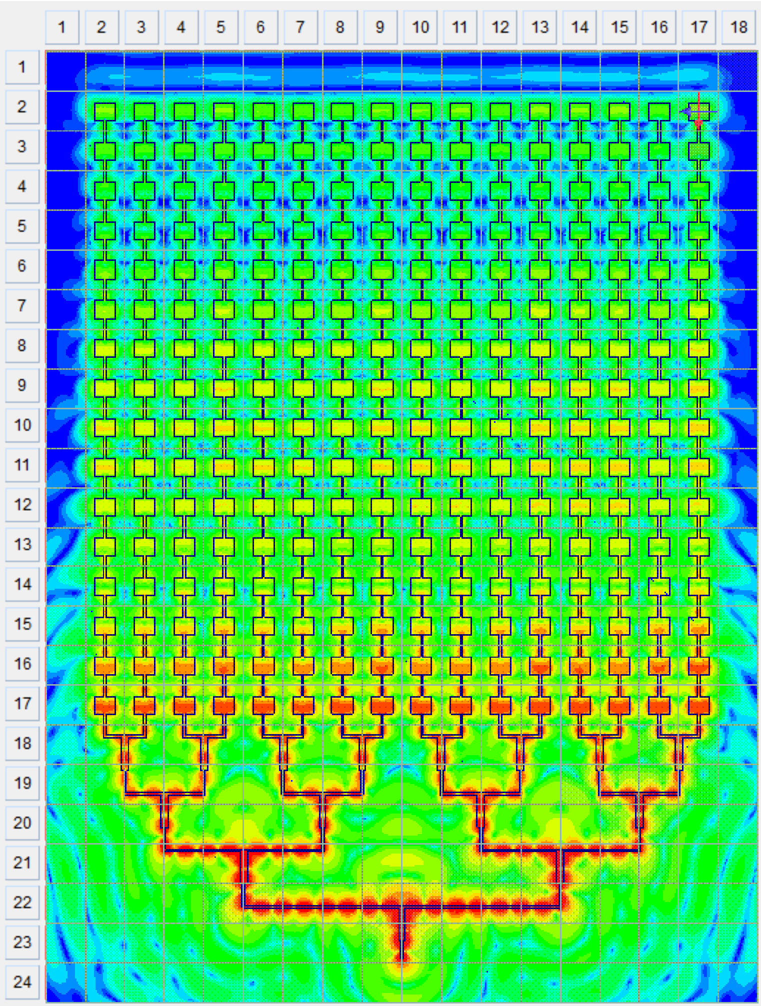


- Surround with region object
- PML truncation available

16 x 16 Array with Feed network

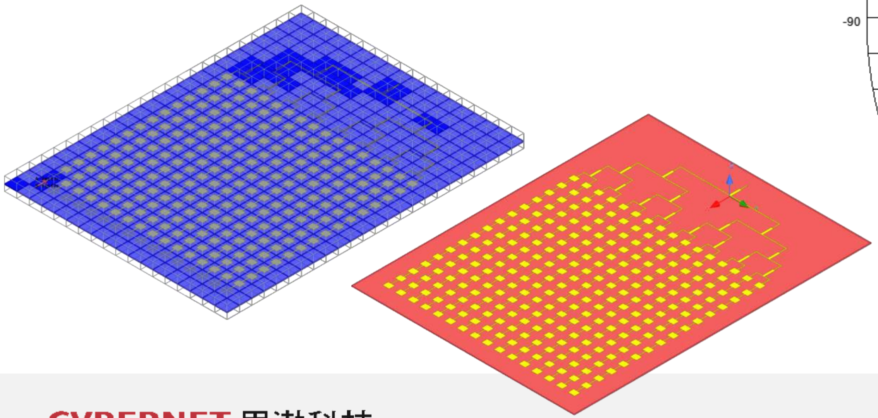
- Array Mask with 432 Elements
- 26 Individual Components
- Full Array
- Fields propagating through the domains

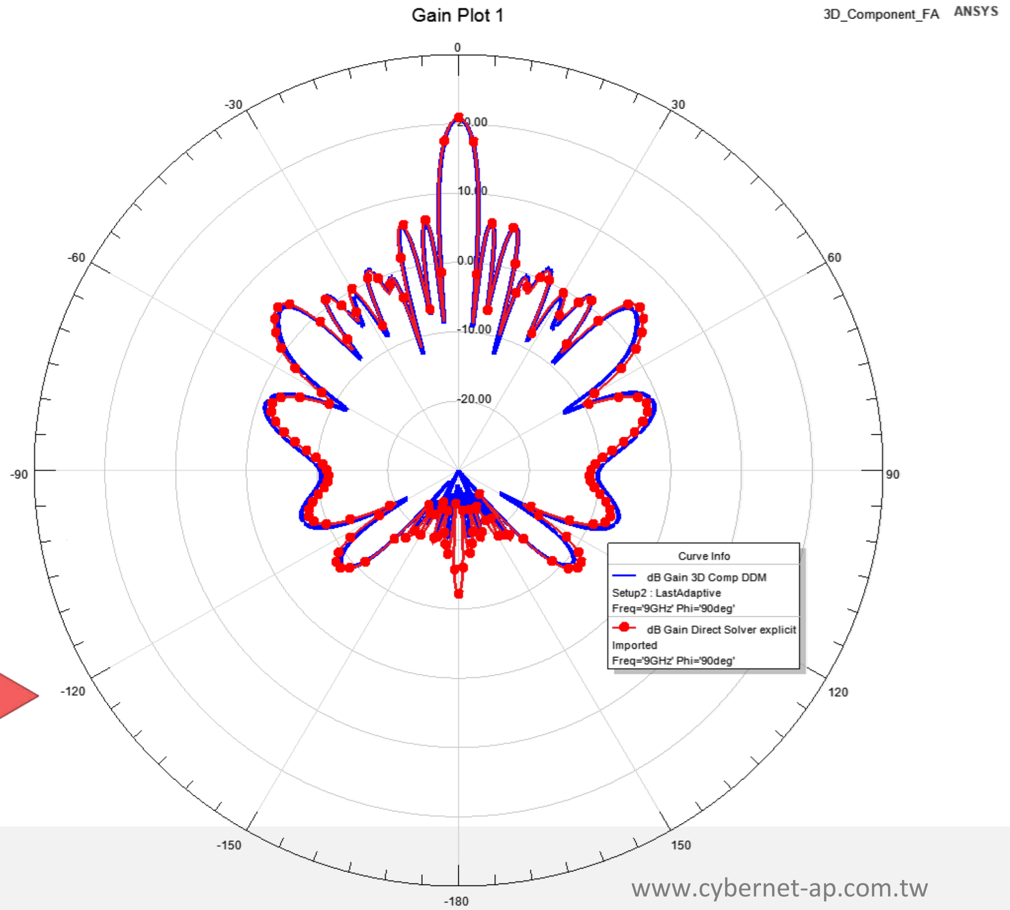




Results Comparison for 16 x 16 Array

- Compare explicit, direct matrix solve with 3D Comp DDM
- Excellent agreement in far field
- 3D Comp DDM
  - 67% savings in memory
  - 37% savings in simulation time

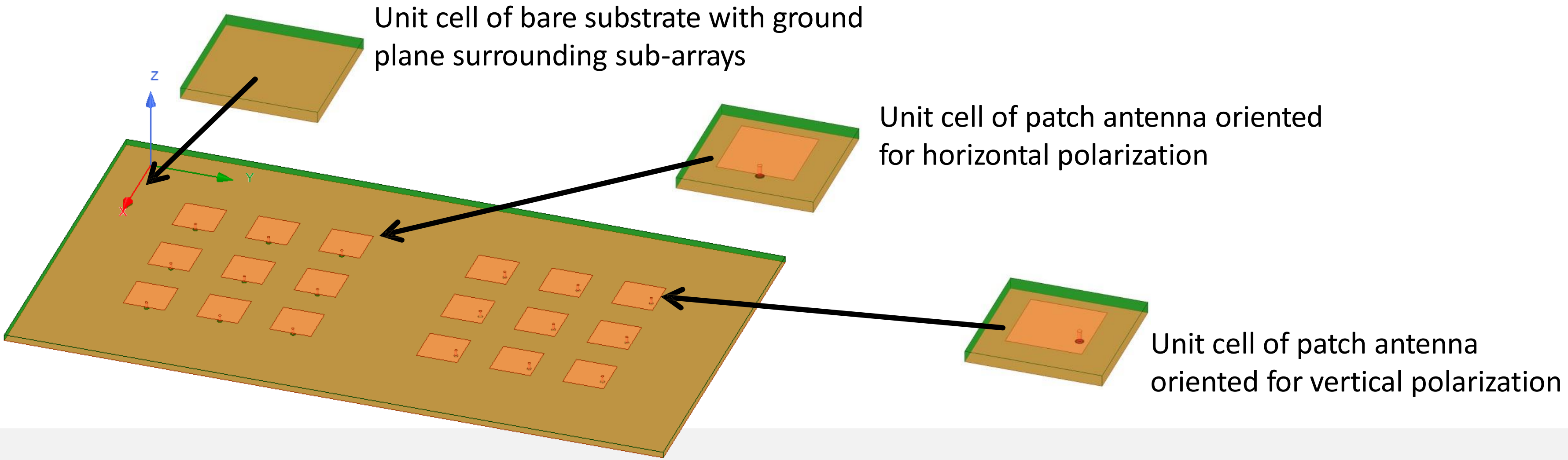




3D Component Array

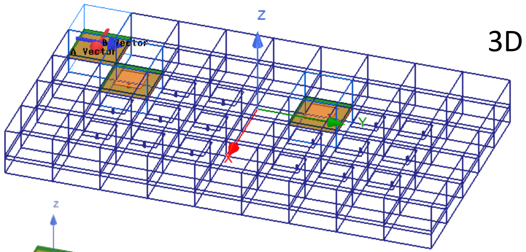
Objective

- The 3D Component Finite Array workflow is a new and efficient iterative domain-decomposition-based finite element technique introduced in ANSYS HFSS 2019 R3 for modeling finite periodic structures with non-identical unit cells. This new simulation technique enables faster simulation times with memory usage reduction and it can leverage distributed computing resources with shared memory.
- This tutorial will demonstrate the new 3D Component Finite Array workflow by setting up and analyzing the array shown below consisting of three non-identical unit cells

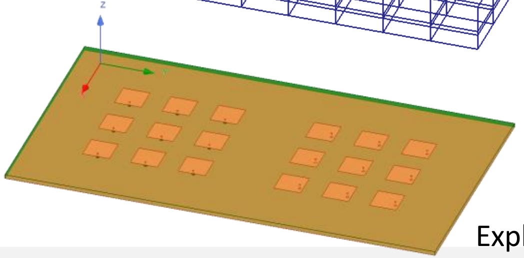


Background

- To utilize the new 3D Component Finite Array workflow with non-identical unit cells, the unit cells have to meet the following requirements:
  - Unit cells are defined as 3D Components
  - Dimensions of unit cells' bounding boxes are identical
  - Master and slave boundaries are defined on surfaces of unit cells
- During the solution process, HFSS creates non-conformal mesh interfaces between the unit cells, which reduces the memory footprint and improves the simulation performance

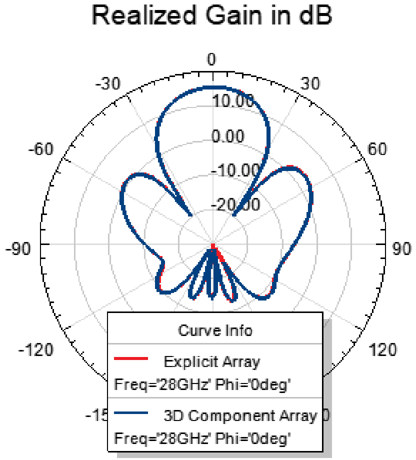


3D Component Finite Array



Explicit array

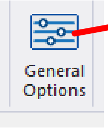
Adaptive meshing comparison on 32 cores:
3D Component Array: 2.7 minutes, 3.1 GB RAM
Explicit array: 20.9 minutes, 10.5 GB RAM

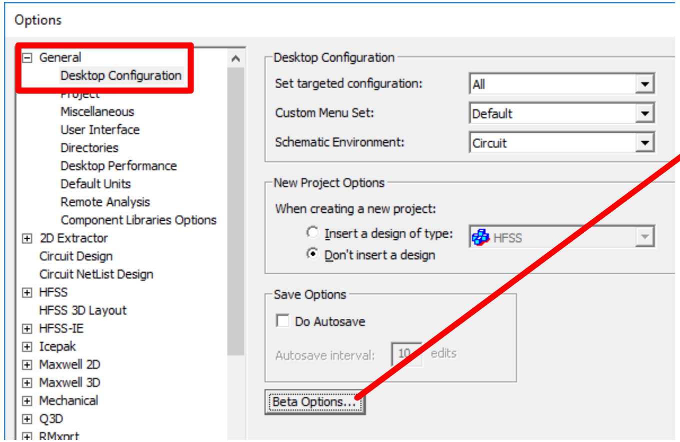


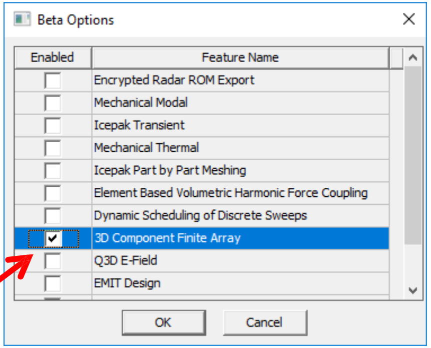
Getting Started

- Open HFSS
  - To access HFSS, click the Microsoft Start button and select: *Programs > ANSYS EM Suite 2019 R3 > ANSYS Electronics Desktop 2019 R3*

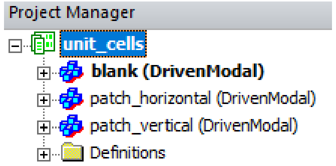
- Enable 3D Component Finite Array (if not already enabled)
  - In the Desktop ribbon, click on *General Options*
  - Expand *General* and select *Desktop Configuration*
  - Click on *Beta Options…*
  - Enable *3D Component Finite Array*



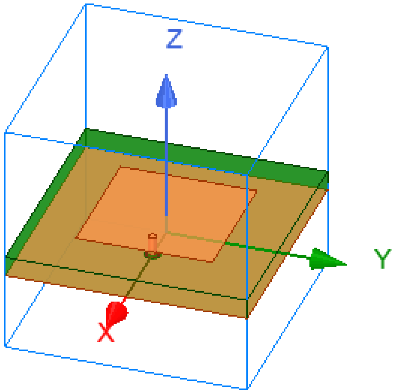




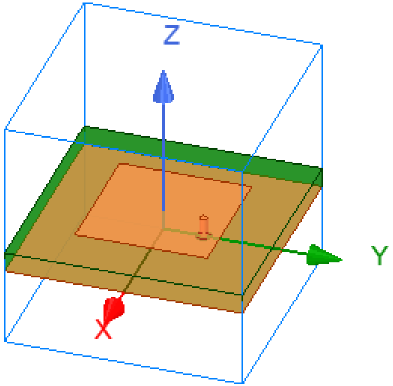
Unit Cells for Finite Array



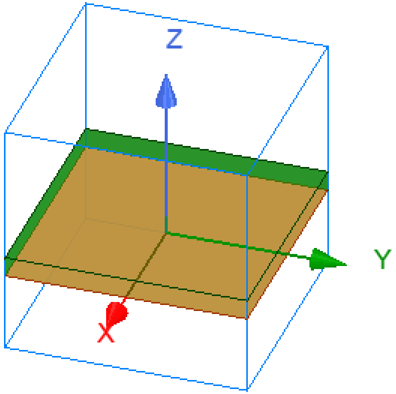
- Open the project *unit_cells.aedt*
- The project contains three designs:



patch_horizontal



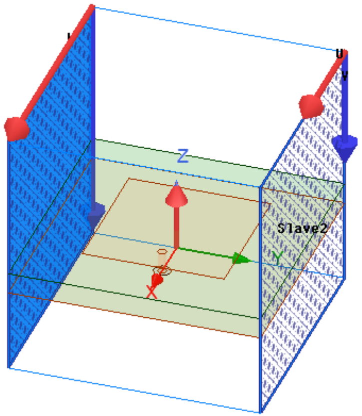
patch_vertical



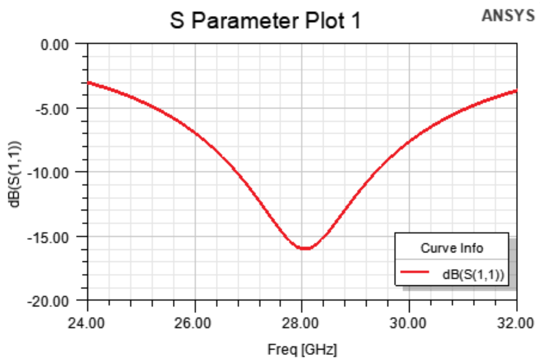
blank

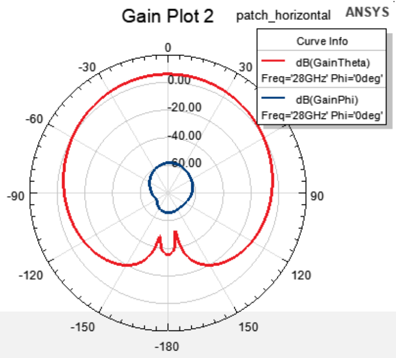
- On the next slides, we will explore how each of these designs is setup and we will create 3D Components for each unit cell to be used in the finite array.

Patch for Horizontal Polarizat



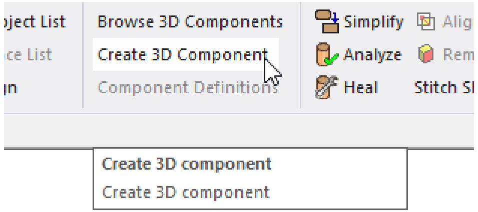
- Open the design *patch_horizontal*
- Explore the design
  - Unit cell design:
    - Probe fed patch with ground plane
    - Airbox contains master and slave boundaries on the sides and radiation boundaries on top and bottom
    - Note: Master1 and Slave1 pair is along the x-direction, Master2 and Slave2 pair is along the y-direction
    - The unit cell footprint is 6 mm x 6 mm
  - Run simulation
    - S-parameters show patch in infinite array (due to master and slave boundaries) resonates at 28 GHz
    - Radiation pattern shows horizontal polarized antenna: radiation of GainTheta is dominant compared to GainPhi

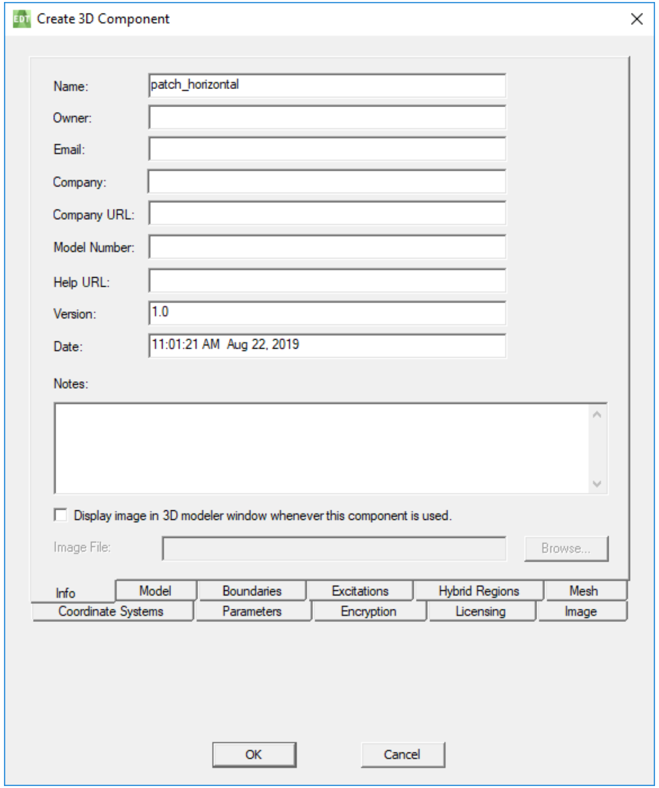




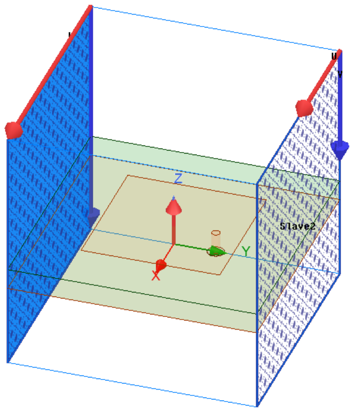
Create 3D Component for Horizontal Patch

- Select all objects in the model by clicking *Edit > Select All*
- In Model ribbon, click on *Create 3D Component*
  - Specify the name of the 3D Component (make sure it does not contain spaces) as *patch_horizontal*
  - Click *OK* to save the 3D Component as *patch_horizontal.a3dcomp*

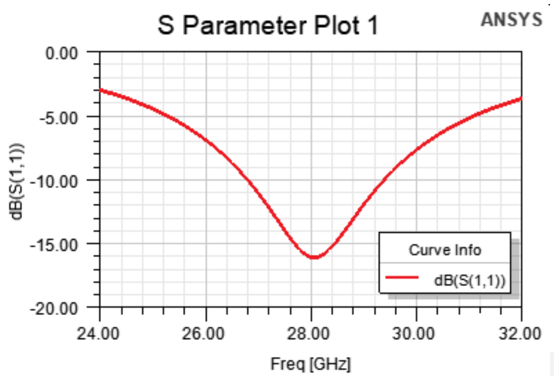


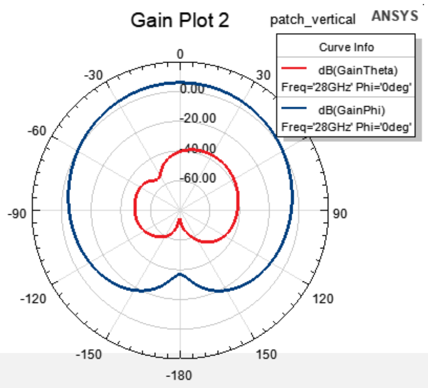


Patch for Vertical Polarizatic



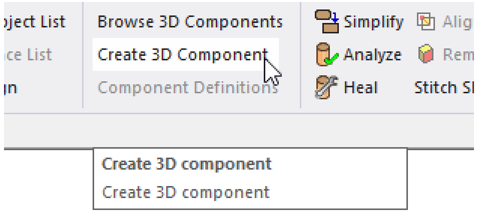
- Open the design *patch_vertical*
- Explore the design
  - Unit cell design:
    - Probe fed patch with ground plane
    - Airbox contains master and slave boundaries on the sides and radiation boundaries on top and bottom
    - Note: Master1 and Slave1 pair is along the x-direction, Master2 and Slave2 pair is along the y-direction
    - The unit cell footprint is 6 mm x 6 mm
  - Run simulation
    - S-parameters show patch in infinite array (due to master and slave boundaries) resonates at 28 GHz
    - Radiation pattern shows vertical polarized antenna: radiation of GainPhi is dominant compared to GainTheta

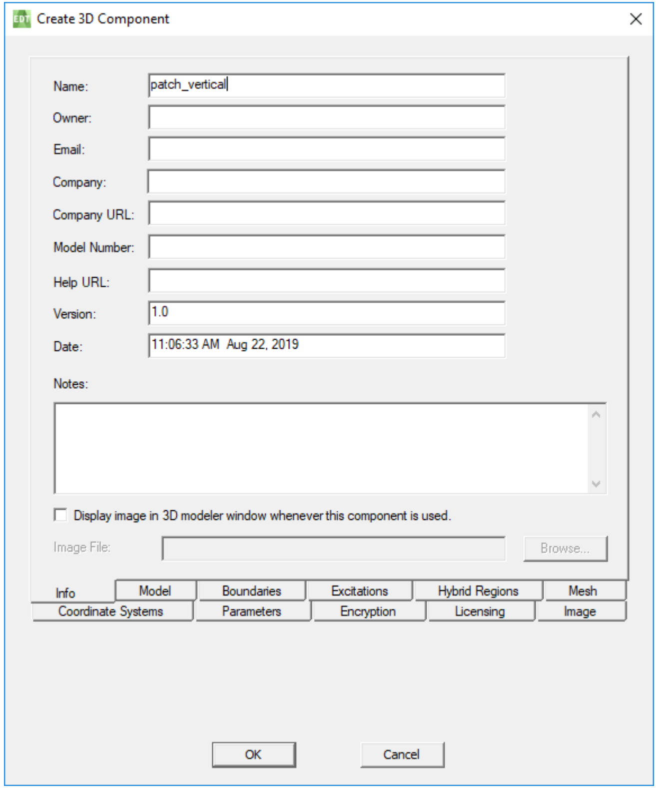




Create 3D Component for Vertical Patch

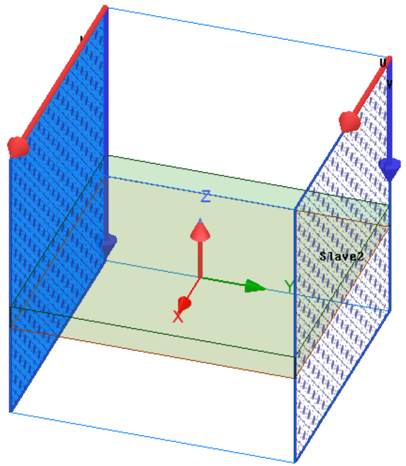
- Select all objects in the model by clicking *Edit > Select All*
- In Model ribbon, click on *Create 3D Component*
  - Specify the name of the 3D Component (make sure it does not contain spaces) as *patch_vertical*
  - Click *OK* to save the 3D Component as *patch_vertical.a3dcomp*





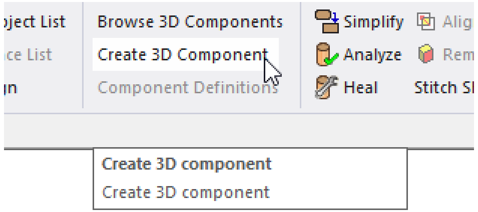
Unit Cell for Edges of Array

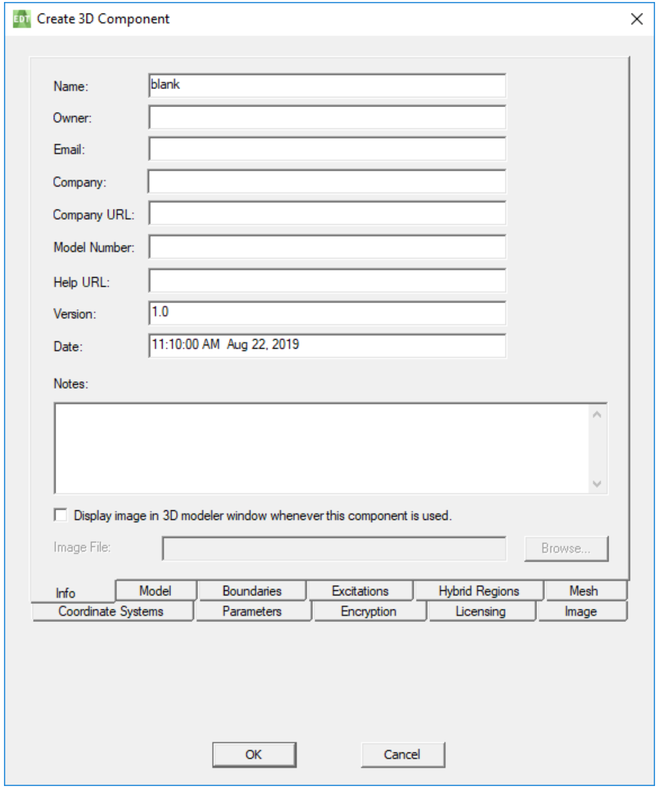
- Open the design *blank*
- Explore the design
  - Unit cell design:
    - Design is similar to the unit cells for the patches, but it only contains the substrate and ground plane
    - Airbox contains master and slave boundaries on the sides and radiation boundaries on top and bottom
    - Note: Master1 and Slave1 pair is along the x-direction, Master2 and Slave2 pair is along the y-direction
    - The unit cell footprint is 6 mm x 6 mm



Create 3D Component for Blank Element

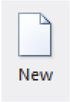
- Select all objects in the model by clicking *Edit > Select All*
- In Model ribbon, click on *Create 3D Component*
  - Specify the name of the 3D Component (make sure it does not contain spaces) as *blank*
  - Click *OK* to save the 3D Component as *blank.a3dcomp*



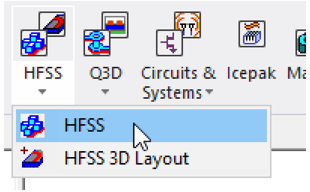


Create a Project for Array Simulation

- Create a new Project by clicking on *New* in the *Desktop* ribbon



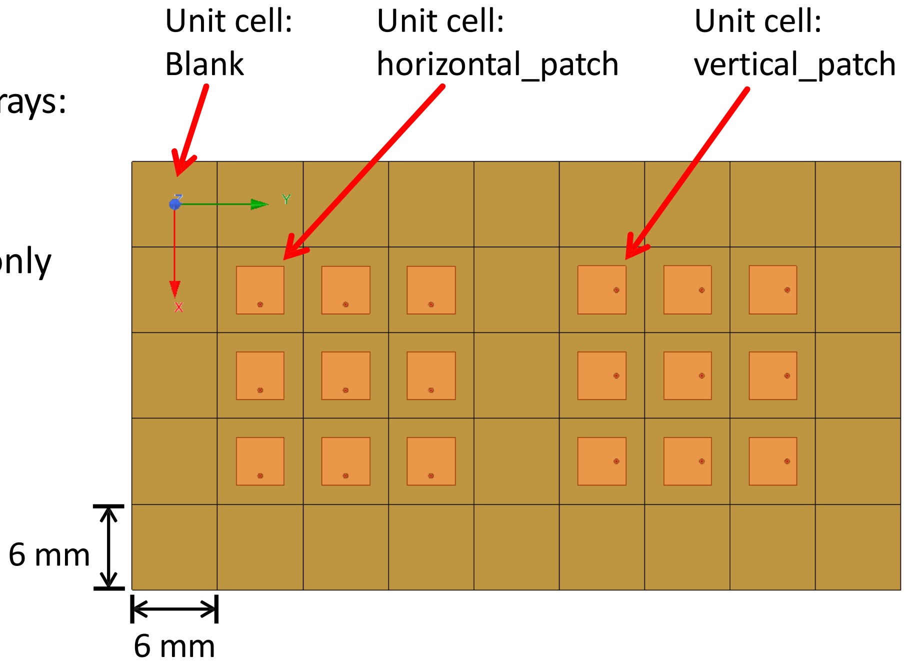
- Insert a HFSS design by clicking on *HFSS > HFSS* in the *Desktop* ribbon



- Save the project as *array.aedt*

Array Layout

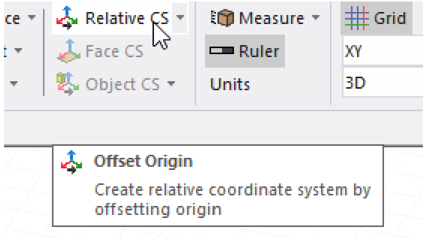
- We want to create an array consisting of two subarrays:
  - 3x3 array of horizontal polarized patches,
  - 3x3 array of vertical polarized patches,
  - Each subarray is surrounded by cells containing only the substrate and ground plane



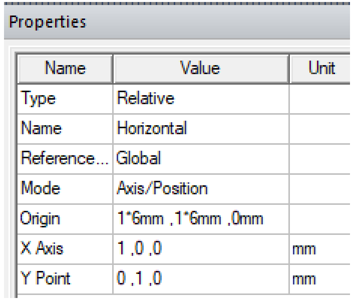
- 3D Components are placed on a target coordinate system
- On the next slides, we will create relative coordinate systems for each of the three unit cells and insert them into the model

Relative Coordinate Systems: Horizontal Patch

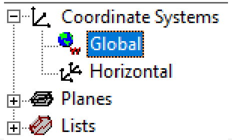
- For the horizontally polarized patch antenna, create a relative coordinate system by clicking on *Relative CS* in the *Draw* ribbon



- Place the new coordinate system anywhere in the modeler window, then edit the properties of *Relative CS1*:
  - Change its name to *Horizontal*
  - Place its origin at *1*6 mm, 1*6 mm, 0 mm*

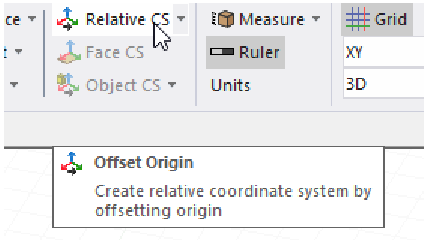


- Set the Global coordinate system as the working coordinate system by clicking on *Global* in the Coordinate Systems branch in the modeler tree

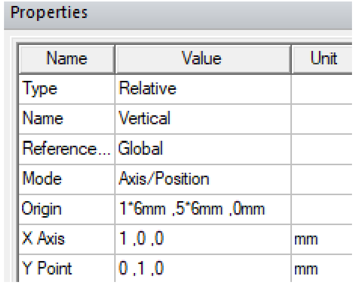


Relative Coordinate Systems: Vertical Patch

- For the horizontally polarized patch antenna, create a relative coordinate system by clicking on *Relative CS* in the *Draw* ribbon



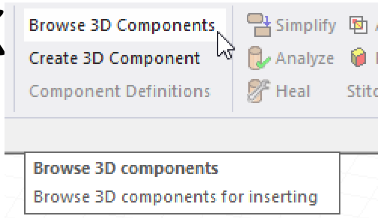
- Place the new coordinate system anywhere in the modeler window, then edit the properties of *Relative CS1*:
  - Change its name to *Vertical*
  - Place its origin at *1*6 mm, 5*6 mm, 0 mm*


Properties

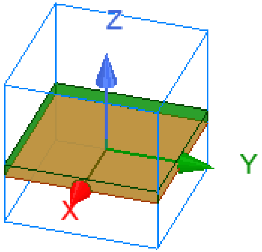
Name	Value	Unit
Type	Relative	
Name	Vertical	
Reference...	Global	
Mode	Axis/Position	
Origin	1*6mm ,5*6mm ,0mm	
X Axis	1 ,0 ,0	mm
Y Point	0 ,1 ,0	mm

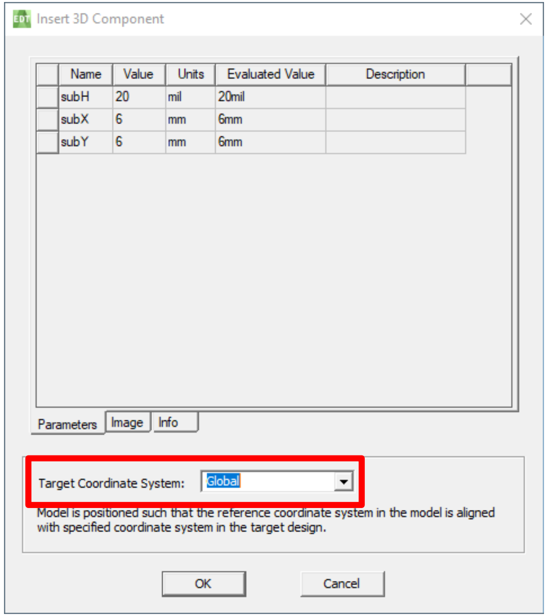
- Set the Global coordinate system as the working coordinate system by clicking on *Global* in the Coordinate Systems branch in the modeler tree

Insert 3D Component: Blank

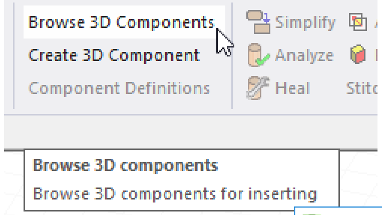


- In the *Model* ribbon, click on *Browse 3D Components*
- Browse to and open the previously created 3D Component *blank.a3dcomp*
- Ensure that *Target Coordinate System* is set to *Global*

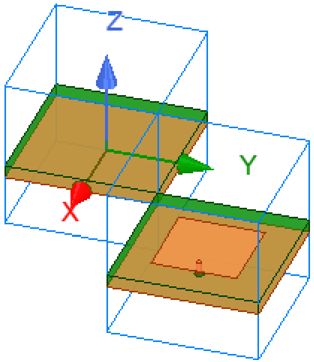


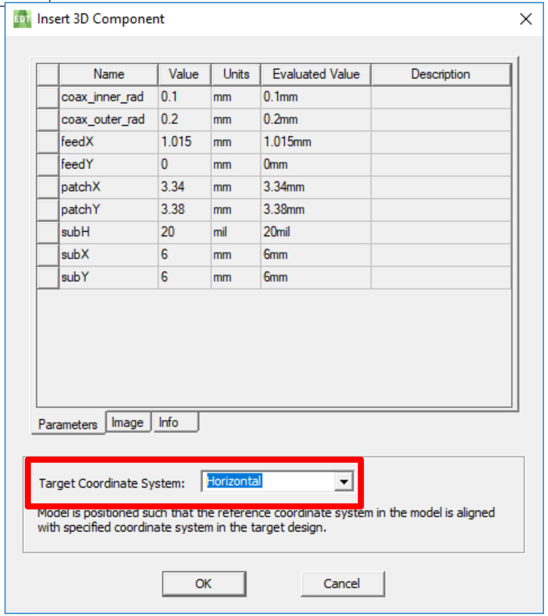


Insert 3D Component: Horizontal Patch



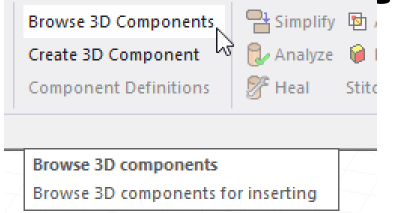
- In the *Model* ribbon, click on *Browse 3D Components*
- Browse to and open the previously created 3D Component *patch_horizontal.a3dcomp*
- Ensure that *Target Coordinate System* is set to *Horizontal*



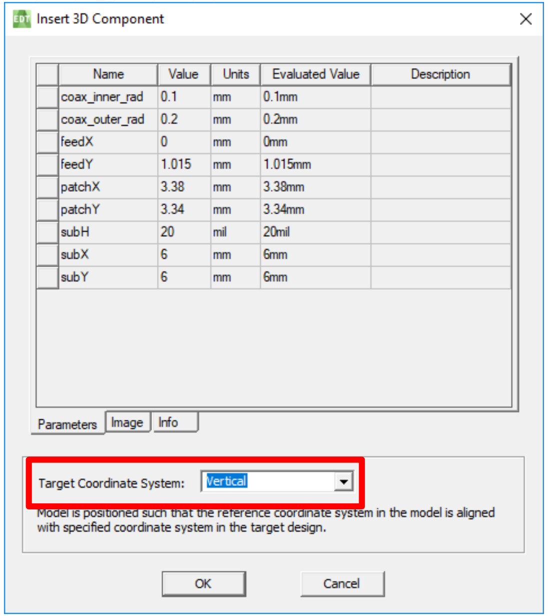


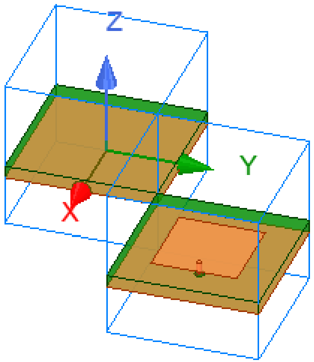
Name	Value	Units	Evaluated Value	Description
coax_inner_rad	0.1	mm	0.1mm	
coax_outer_rad	0.2	mm	0.2mm	
feedX	1.015	mm	1.015mm	
feedY	0	mm	0mm	
patchX	3.34	mm	3.34mm	
patchY	3.38	mm	3.38mm	
subH	20	mil	20mil	
subX	6	mm	6mm	
subY	6	mm	6mm	

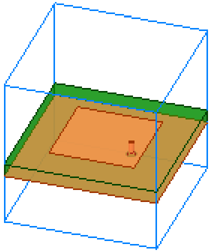
Insert 3D Component: Horizontal Patch



- In the *Model* ribbon, click on *Browse 3D Components*
- Browse to and open the previously created 3D Component *patch_vertical.a3dcomp*
- Ensure that *Target Coordinate System* is set to *Vertical*

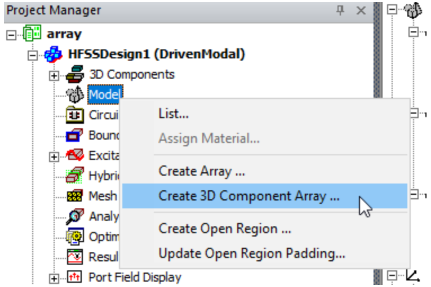




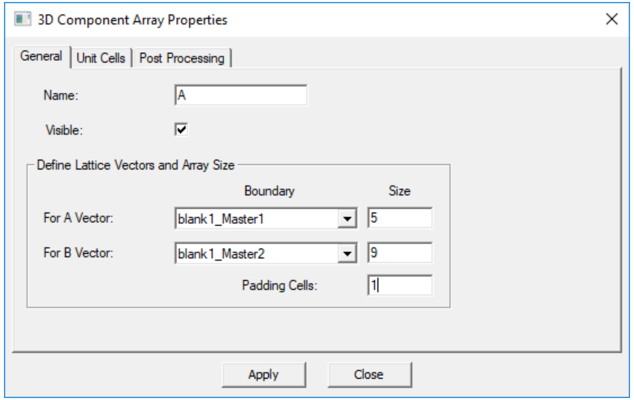


Create the 3D Component Array

- In the project manager, right-click on *Model* and select *Create 3D Component Array…*

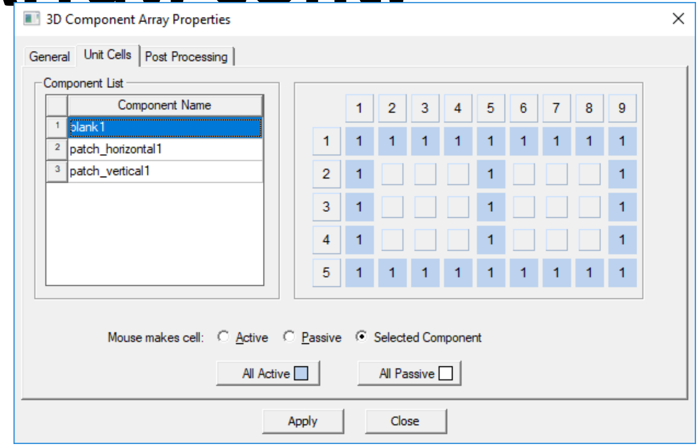


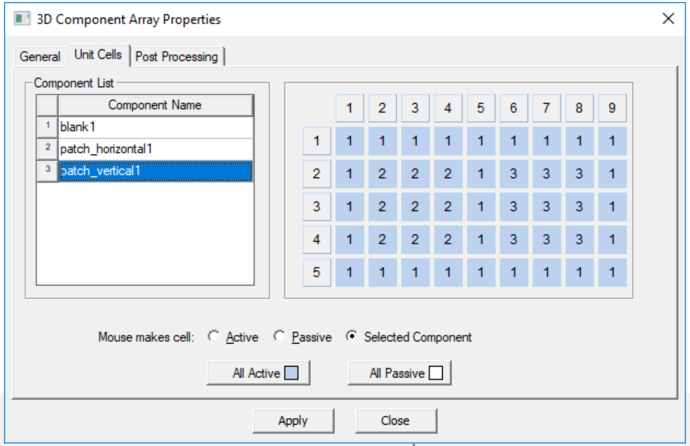
- On the *General* tab, specify the layout of the array
  - Enable *visible*
  - The array should consist of 5 elements in the x-direction and 9 elements in the y-direction
    - Set A vector to *blank1_Master1* (along x-direction, defined by unit cell boundary setup) with *5* cells
    - Set B vector to *blank1_Master2* (along y-direction, defined by unit cell boundary setup) with *9* cells
  - Set *Padding Cells* to *1*
    - This setting defines how many empty cells surround the array, i.e. how much airspace surrounds the array



Create the 3D Component Array, cont.

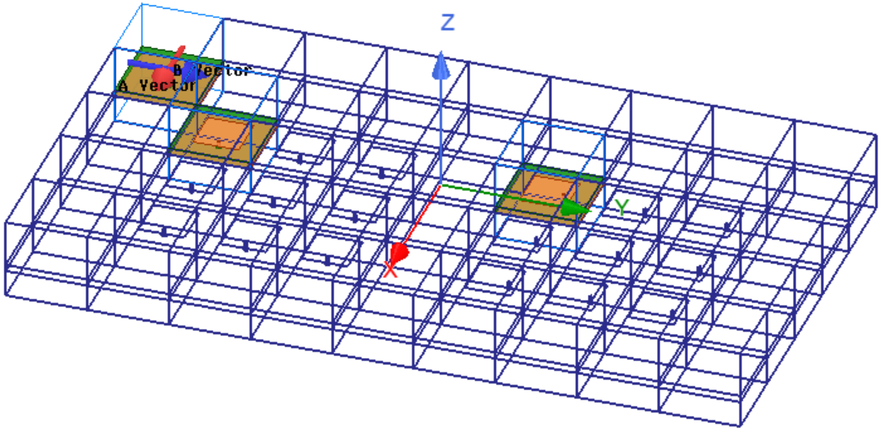
- On *Unit Cells* tab, specify the content of each array element
  - Select component *blank1* and click on the cells surrounding the sub-arrays:
    - Note, clicking on the number indicating the row or column will fill the entire row or column with the selected component
  - Fill in the areas of the sub-arrays with the component *patch_horizontal1* and *patch_vertical1*
  - Click *Apply* to visualize the array
- Click *Close*



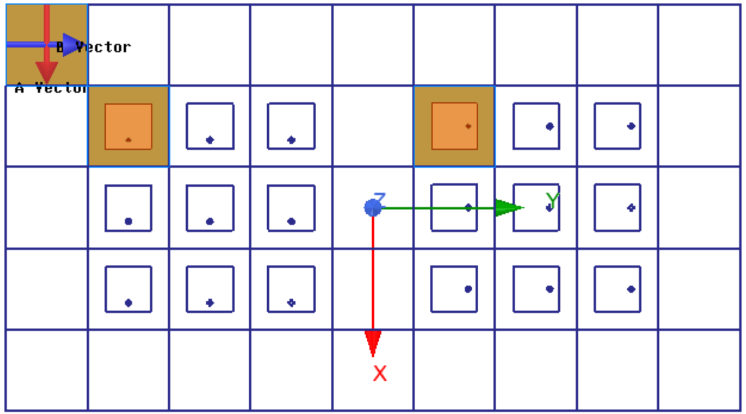


Create the 3D Component Array, cont.

- The arrayed unit cells are visualized by showing the edges of the contained objects
- These objects cannot be selected, only objects of the inserted 3D Components

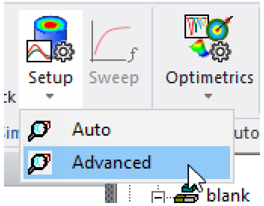


Trimetric view

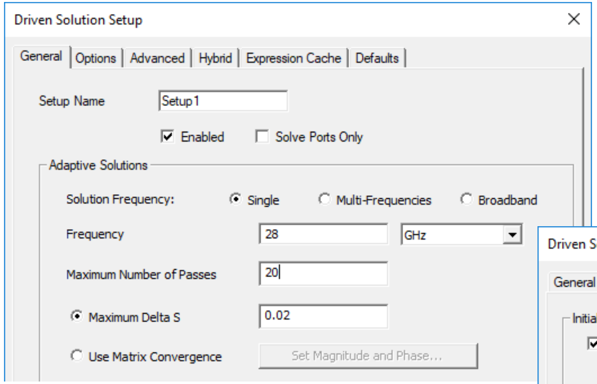


Top view

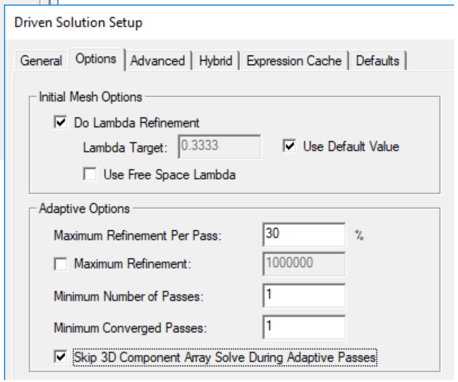
Solution Setup



- In the *Simulation* ribbon, click on *Setup > Advanced*
  - On the *General* tab, set
    - Solution frequency to 28 GHz,
    - Maximum number of passes to 20
    - Maximum Delta S to 0.02



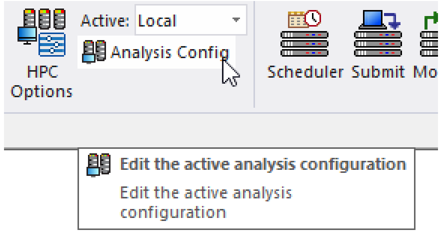
  - On *Options* tab, ensure that *Skip 3D Component Array Solve During Adaptive Passes* is enabled
    - This option will adapt the mesh for each 3D Component without spending the time to solve the entire array
  - Click *OK* to accept settings

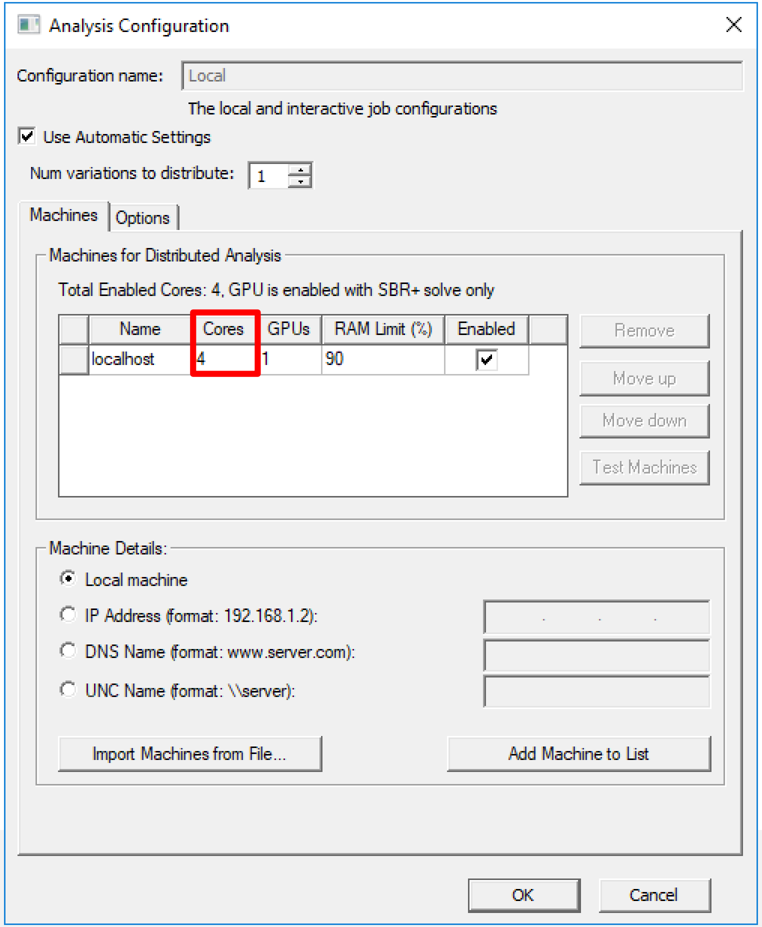


- Click *Cancel* for the frequency sweep dialog that pops up
  - For this tutorial, we are only interested in the array performance at 28 GHz

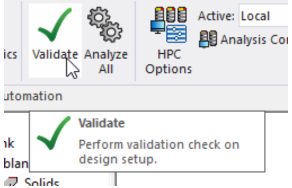
Verify HPC Configuration

- Before running the simulation, verify the HPC configuration
- On the *Simulation* ribbon, click on *Analysis Configuration*
  - Ensure that you are using as many cores as available on your computer and are enabled by your HPC license configuration
  - Ensure that *Use Automatic Settings* is enabled

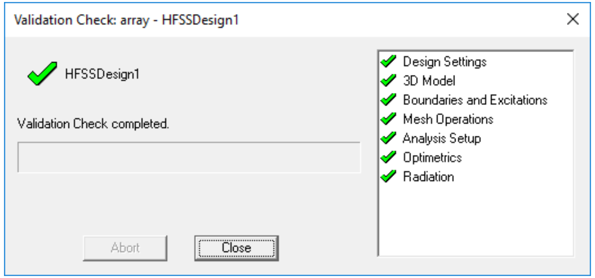




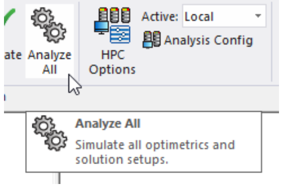
Analyze the 3D Component Finite Array



- Validate and run the simulation
  - On the *Simulation* ribbon, validate the simulation by clicking *Validate*



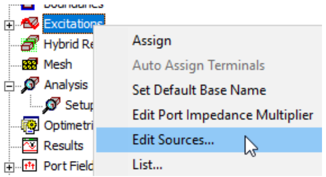
  - If no errors are indicated, run the simulation by clicking *Analyze All*



- Once the simulation is completed, save the project

Edit Sources

- The radiation pattern is dependent on how each of the array elements is excited
- Open the edit sources dialog by right-clicking on *Excitations* in the Project Manager and selecting *Edit Sources*



- For broadside excitation of the horizontal elements, set all 9 sources that contain *horizontal* in their name to
  - Magnitude = 1 W,
  - Phase = 0 deg
- All other sources should be set to 0 W
- This can be done manually, or by clicking on *Load From File…* and browsing to and selecting *horizontal.csv*
- Click *OK*

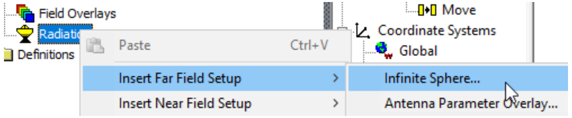
Edit post process sources

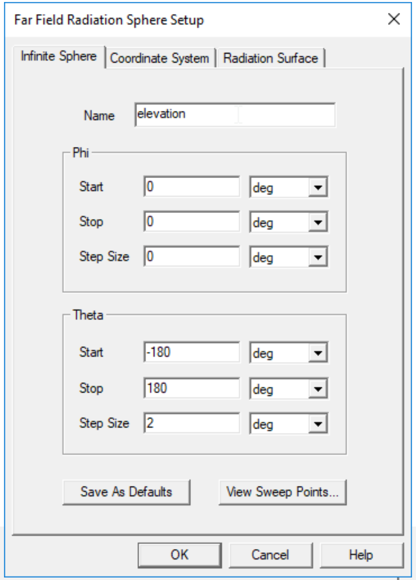
Spectral Fields

	Source	Type	Magnitude	Unit	Phase	Unit
1	A[2,2]patch_horizontal1_1:1	Port	1	W	0	deg
2	A[2,3]patch_horizontal1_1:1	Port	1	W	0	deg
3	A[2,4]patch_horizontal1_1:1	Port	1	W	0	deg
4	A[2,6]patch_vertical1_1:1	Port	0	W	0	deg
5	A[2,7]patch_vertical1_1:1	Port	0	W	0	deg
6	A[2,8]patch_vertical1_1:1	Port	0	W	0	deg
7	A[3,2]patch_horizontal1_1:1	Port	1	W	0	deg
8	A[3,3]patch_horizontal1_1:1	Port	1	W	0	deg
9	A[3,4]patch_horizontal1_1:1	Port	1	W	0	deg
10	A[3,6]patch_vertical1_1:1	Port	0	W	0	deg
11	A[3,7]patch_vertical1_1:1	Port	0	W	0	deg
12	A[3,8]patch_vertical1_1:1	Port	0	W	0	deg
13	A[4,2]patch_horizontal1_1:1	Port	1	W	0	deg
14	A[4,3]patch_horizontal1_1:1	Port	1	W	0	deg
15	A[4,4]patch_horizontal1_1:1	Port	1	W	0	deg
16	A[4,6]patch_vertical1_1:1	Port	0	W	0	deg
17	A[4,7]patch_vertical1_1:1	Port	0	W	0	deg
18	A[4,8]patch_vertical1_1:1	Port	0	W	0	deg

Radiation Pattern

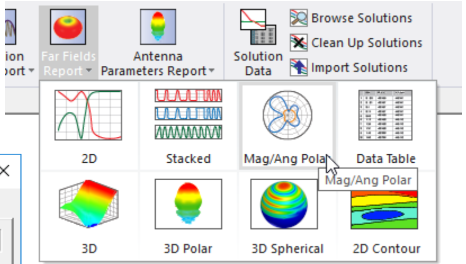
- Before creating a radiation pattern, we have to define an infinite sphere setup to specify at what angles we want to observe the far-field quantities
- In the Project Manager, right-click on *Radiation* and select *Insert Far Field Setup > Infinite Sphere*
  - Name: elevation
  - Phi start: 0 deg
  - Phi stop: 0 deg
  - Phi step size: 0 deg
  - Theta start: -180 deg
  - Theta stop: 180 deg
  - Theta step size: 2 deg

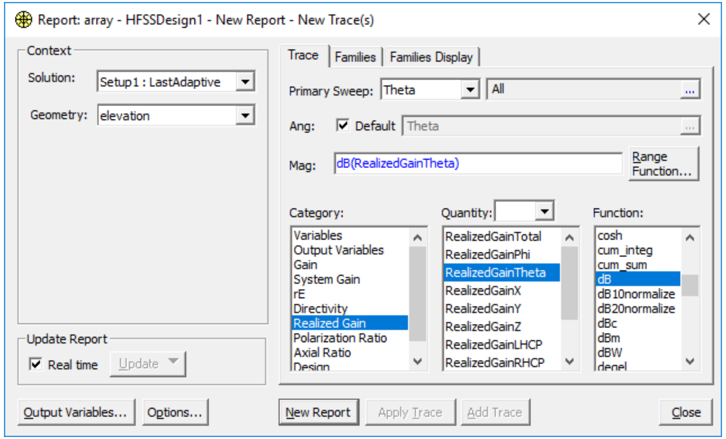


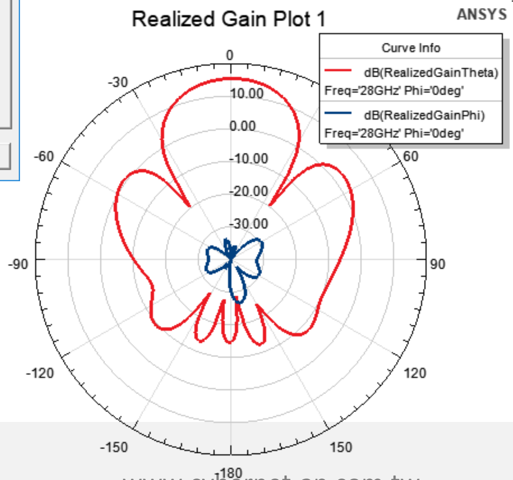


Radiation Pattern, cont.

- Create a radiation pattern by clicking on *Far Fields Report > Mag/Ang Polar* in the *Results* ribbon
- In the report creation dialog, select:
  - Category: *Realized Gain*
  - Quantity: *RealaizedGainTheta*
  - Function: *dB*
  - Click *New Report*
  - Category: *Realized Gain*
  - Quantity: *RealaizedGainPhi*
  - Function: *dB*
  - Click *Add Trace*



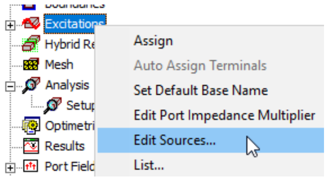




- Note: The radiation pattern shows that the Theta polarization is dominant

Edit Sources

- The radiation pattern of the vertically polarized sub-array can be visualized by turning on only the sources of those patches
- Open the edit sources dialog by right-clicking on *Excitations* in the Project Manager and selecting *Edit Sources*



- For broadside excitation of the vertical patch elements, set all 9 sources that contain *vertical* in their name to
  - Magnitude = 1 W,
  - Phase = 0 deg
- All other sources should be set to 0 W
- This can be done manually, or by clicking on *Load From File…* and browsing to and selecting *vertical.csv*
- Click *OK*

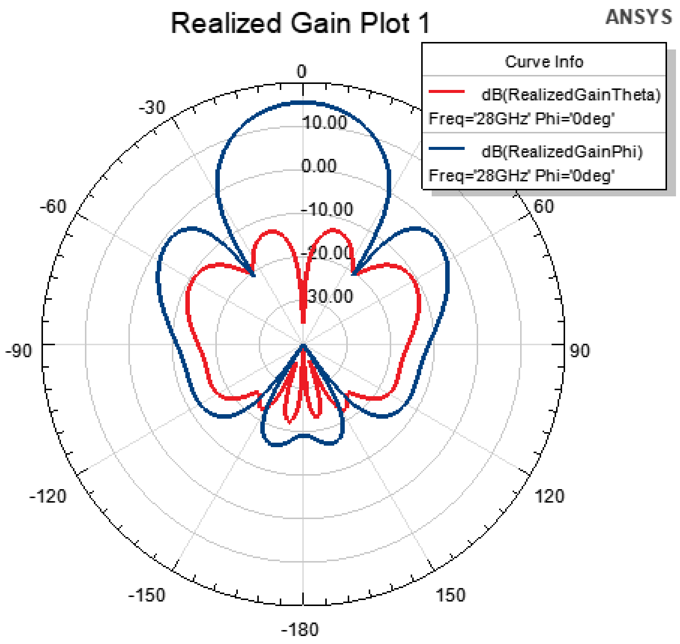
Edit post process sources

Spectral Fields

	Source	Type	Magnitude	Unit	Phase	Unit
1	A[2,2]patch_horizontal1_1:1	Port	0	W	0	deg
2	A[2,3]patch_horizontal1_1:1	Port	0	W	0	deg
3	A[2,4]patch_horizontal1_1:1	Port	0	W	0	deg
4	A[2,6]patch_vertical1_1:1	Port	1	W	0	deg
5	A[2,7]patch_vertical1_1:1	Port	1	W	0	deg
6	A[2,8]patch_vertical1_1:1	Port	1	W	0	deg
7	A[3,2]patch_horizontal1_1:1	Port	0	W	0	deg
8	A[3,3]patch_horizontal1_1:1	Port	0	W	0	deg
9	A[3,4]patch_horizontal1_1:1	Port	0	W	0	deg
10	A[3,6]patch_vertical1_1:1	Port	1	W	0	deg
11	A[3,7]patch_vertical1_1:1	Port	1	W	0	deg
12	A[3,8]patch_vertical1_1:1	Port	1	W	0	deg
13	A[4,2]patch_horizontal1_1:1	Port	0	W	0	deg
14	A[4,3]patch_horizontal1_1:1	Port	0	W	0	deg
15	A[4,4]patch_horizontal1_1:1	Port	0	W	0	deg
16	A[4,6]patch_vertical1_1:1	Port	1	W	0	deg
17	A[4,7]patch_vertical1_1:1	Port	1	W	0	deg
18	A[4,8]patch_vertical1_1:1	Port	1	W	0	deg

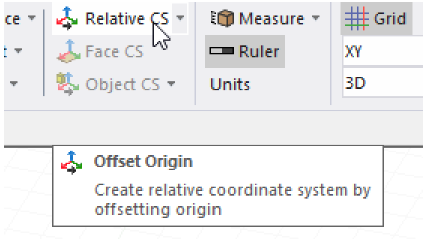
Radiation Pattern of Vertical Patch Sub-Array

- By changing the sources in the Edit Sources dialog, the radiation pattern will update
- The radiation patterns shows that the sub-array is polarized in Phi



Field Overlay

- A field overlay plot can quickly show what elements in the array are excited and which ones are not
- Fields can be plotted and animated on non-model cut planes, as well as on selected model object faces
- To create a non-model cut plane, define a new relative coordinate system
  - In the *Draw* ribbon click on *Relative CS*

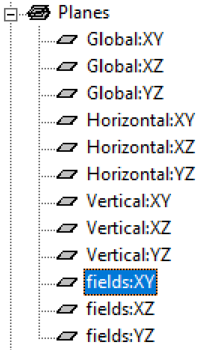


  - Place it anywhere in the modeler window and edit its properties
    - Name: *fields*
    - Origin: *0 mm, 0 mm, 3.5 mm*

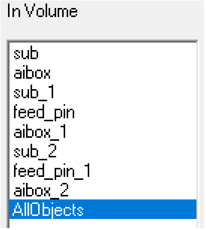
Properties

Name	Value	Unit
Type	Relative	
Name	ields	
Reference...	Global	
Mode	Axis/Position	
Origin	0 ,0 ,3.5	mm
X Axis	1 ,0 ,0	mm
Y Point	0 ,1 ,0	mm

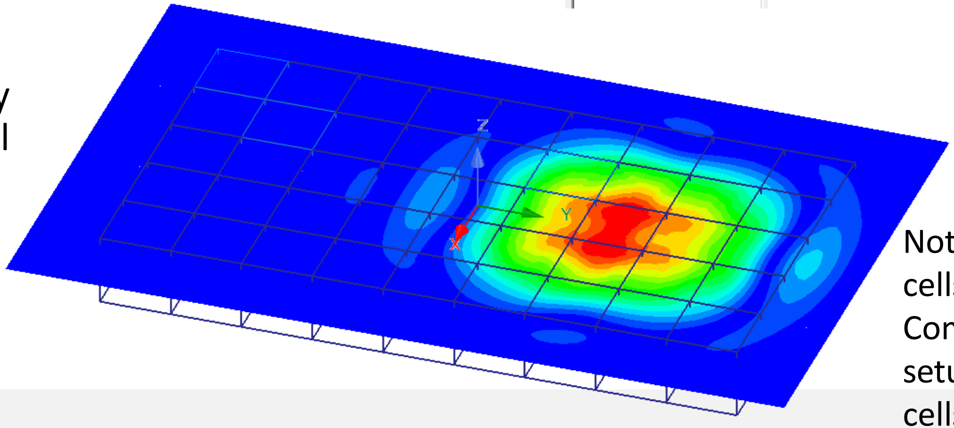
Field Overlay, cont.



- From the modeler tree, select the XY-plane of the fields coordinate system: *fields:XY*
- Right-click in the modeler window and select *Plot Fields > E > Mag_E*
- In the *Create Field Plot* dialog you can limit in what subset of volumes the fields will be plotted
  - Select *AllObjects*
- Accept settings by clicking *Done*



- The field overlay plot clearly shows that only the vertical subarray is active



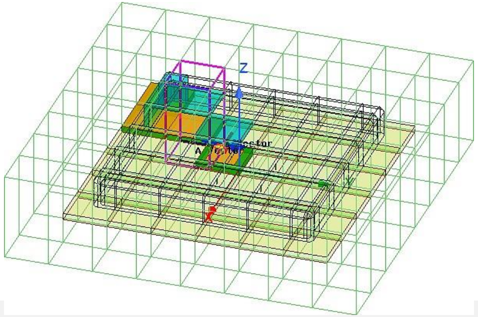
Note: Fields are shown in all cells defined by the 3D Component Finite Array setup including the padding cells surrounding the array

Overview

- This workshop shows the workflow of a 3D component finite array simulation also known as Component Array Domain Decomposition Method (CADDM) in HFSS. This workshop presents a 77 GHz 2x2 phased array antenna with Radome integration by using the 3D component finite array workflow.
- This feature utilizes efficient domain decomposition-based finite element technique for modeling finite semi-periodic structures which contains non-identical unit cells.
- This simulation technique enables faster simulation and less memory usage compared to simulating the Finite Array Domain Decomposition Method (FADDM), and it can leverage distributed computing resources.
- The overall workflow starts with importing the 3D Components into HFSS representing different unit cells in the model. Then creating the array like creating Finite Array from a unit cell. The dialog box however lists all the unit cell components and allow any arrangement of those unit cells within the array dimension that user defines.

Learning Outcome

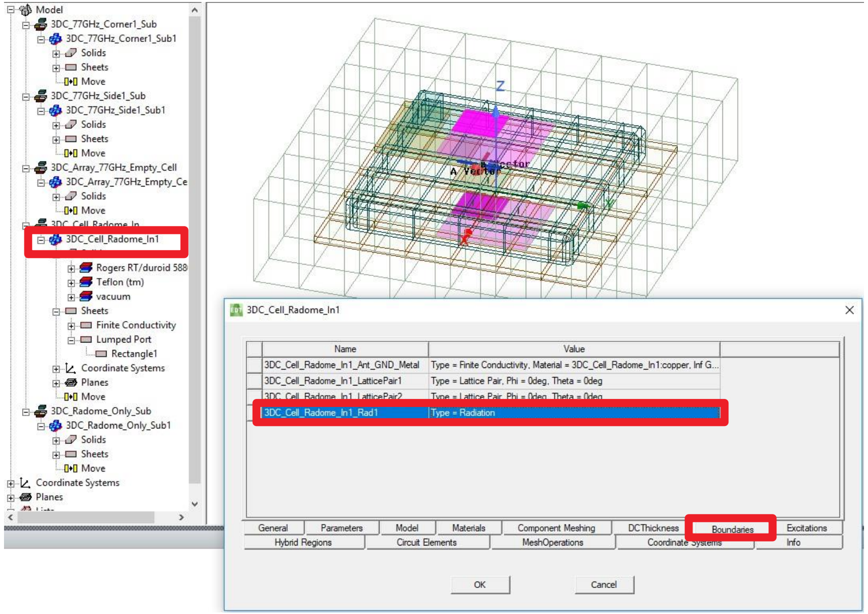
- Know how to use and arrange non-identical unit cells in a finite array
- Know how to mesh CADDM in a way that reduces the memory footprint and improves the simulation performance
- Know how to define Source Groups and apply them for Far-Field patterns



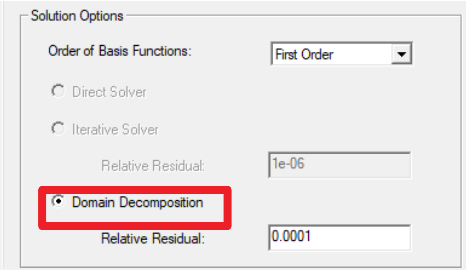
Overview

- To utilize the 3D Component Finite Array workflow with non-identical unit cells, the unit cells must meet the following requirements: **1**
    - Unit cells must be defined as 3D Components
    - Dimensions of unit cells' bounding boxes must be identical
    - Appropriate Lattice Pairs and boundary conditions must be defined on surfaces of unit cells
- Details about each Unit Cell's geometry, boundaries, excitations, parameters can be found in ***Modeler Tree*** by double clicking on HFSS model of each ***3D Component -> Design Properties*** **2**
- Once the array is defined, any cell in the array can be designated as active/passive/padding. The padding cell designation can be utilized to define arbitrarily irregular arrays. Padding cells are treated as background material for fields calculations **3**
- And finally select the ***Domain Decomposition*** in ***Solution Setup*** and set up the ***distributed processor pool***. Note that designs with arrays require ***HPC licenses*** **4**

2



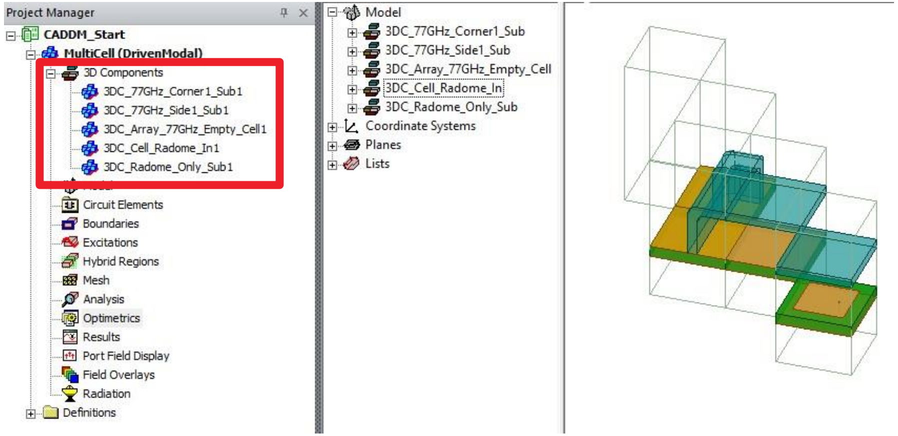
4

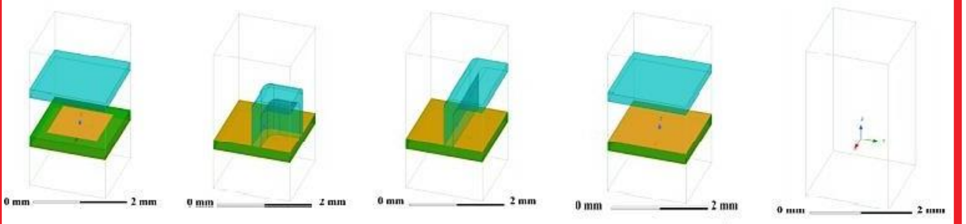


Note: Native geometries and 3D components cannot be mixed in one array

Project Setup

- The project file **CADDM_Start.aedt** is composed of five different 3D components to represent the 77 GHz phased array antenna with Radome
- The active element unit cell is an optimized square patch antenna to operate at 77 GHz with a finite size ground plane, two Lattice Pairs, and a radiation boundary on top and bottom faces. It is simulated over a 254-um thick Rogers RT/duroid® 5880 substrate with a 200-um thick Teflon Radome structure with 1-mm separation from the substrate
- Note, for importing 3D Components in the ***Project Manager*** expand the project (here ***MultiCell***) and right click on ***3D Components -> Browse 3D Components…***
- The finite array beam angle toolkit (***HFSS -> Toolkit -> Finite Array Beam Angle***) will be used to explore beam steering capabilities on this phased array antenna by using parameterized values for the magnitude and phases in the array's excitations

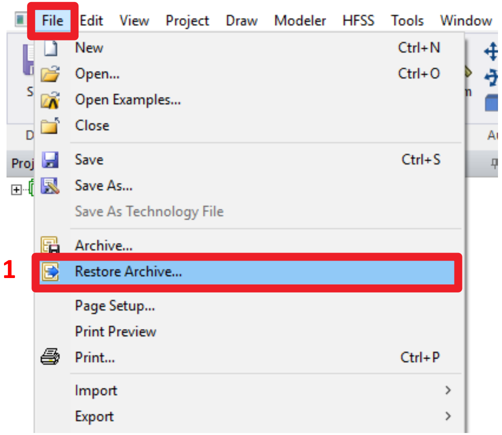




3D components unit cells used in the finite array: Patch antenna, Radome corner and substrate, Radome side and substrate, Top Radome and substrate, and Empty unit cell

Open Project

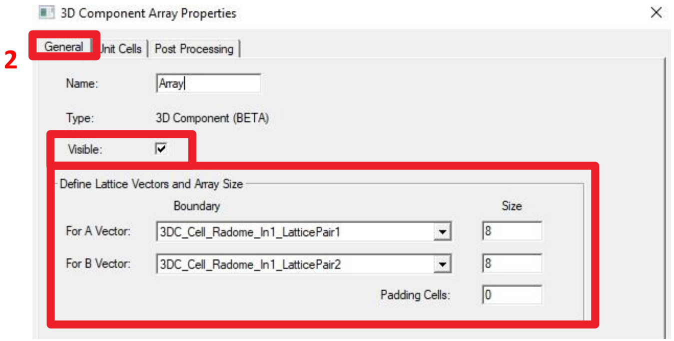
- **Starting ANSYS Electronic Desktop**
  - Windows 10
    - To Launch AEDT, click the Microsoft Start button and type in **Electronic Desktop**. Select **ANSYS Electronic Desktop 2020 R2**
  - Windows 7
    - To Launch AEDT, click the Microsoft Start button and select: **All Programs -> ANSYS EM Suite 2020 R2 -> Ansys Electronic Desktop 2020R2**
- **Load Design File**
  - Click on **File -> Restore Archive** 1
    - Navigate to the training files and choose **CADDM_Start.aedtz**
    - Click **Open**
    - A prompt will ask to save the file. Choose an appropriate directory to save the file. Click **Save** as **CADDM_Start.aedt**

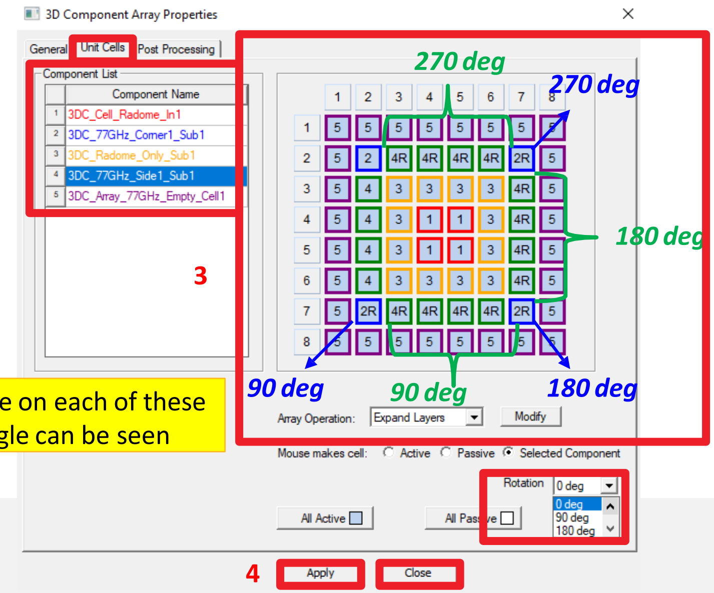


The *z* in .aedt*z* indicates an archive. Think zip.

Create 3D Component Array

- Since all the 3D Components are already imported in the project file **CADDM_Start.aedt** , now the array can be created
- From HFSS menu bar select ***HFSS -> Model -> Create Array…***, then ***3D Component Array Properties*** window appears
- In ***3D Component Array Properties*** window under ***General*** tab set: 1
  - Check the box for ***Visible*** 2
  - In ***Define Lattice Vectors and Array Size:***
    - ***For A Vector: > Boundary:* 3DC_Cell_Radome_In1_LatticePair1*, Size:* 8**
    - ***For B Vector: > Boundary:* 3DC_Cell_Radome_In1_LatticePair2*, Size:* 8**
    - ***Padding Cells:* 0** (because there are empty unit cells part of the configuration)
- In ***3D Component Array Properties*** window under ***Unit Cells*** tab set:
  - Select the desired 3D component from ***Component List*** and click on associated cell on the 8x8 array configuration on the right side 3
  - Apply the ***Rotation*** for corner and side 3D components (90 deg, 180 deg, 270 deg), letter ***R*** next to the unit cell numbers shows the cell is rotated
- Select ***Apply***, and then select ***Close*** 4
- The created array model can be found in the ***Project Manager -> Model -> Array*** 5

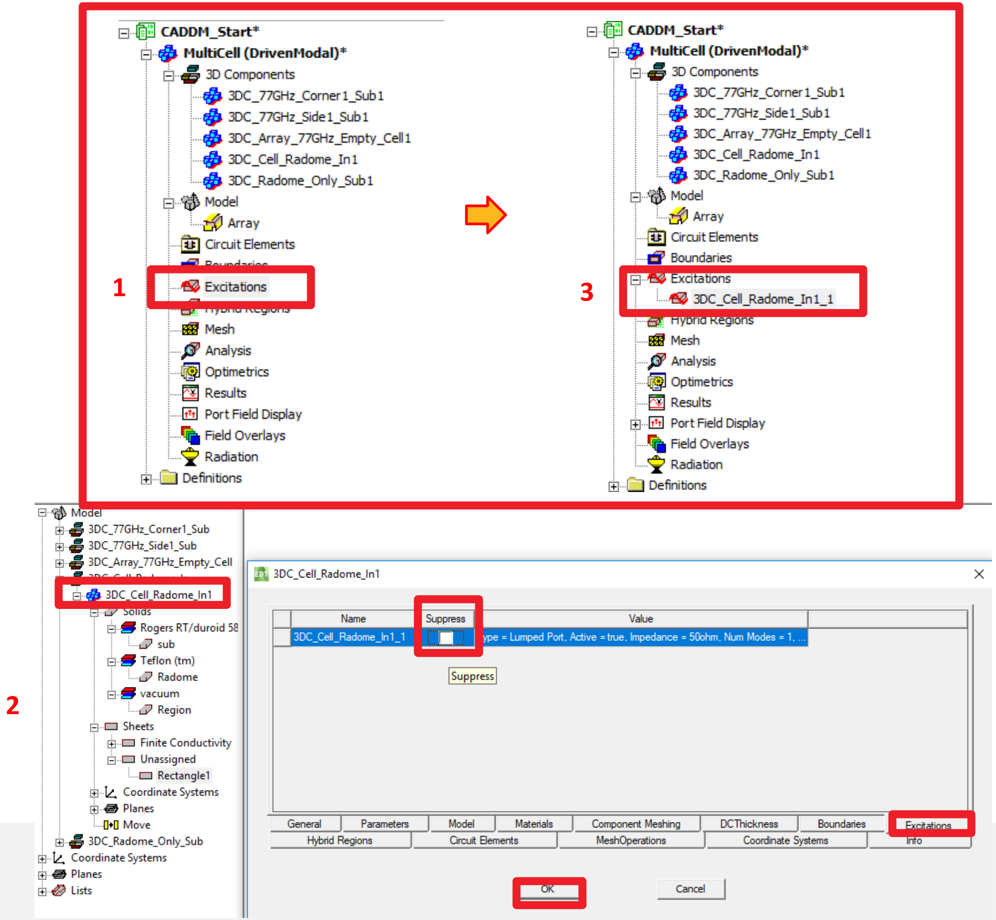




Note: By hovering the mouse on each of these rotated cells the rotation angle can be seen

Define Excitation in Finite Array

- During Importing the 3D Components, the excitation port of the active cell may be suppressed. That is the reason that in **CADDM_Start.aedt** in the ***Project Manager -> Excitation*** there is no defined port 1
- To define the port, in ***Modeler Tree*** double click on **3DC_Cell_Radome_In1** HFSS model and in the ***Properties*** window select ***Excitations*** tab and uncheck ***Suppress***, and click ***OK*** 2
- The activated port can be found in the ***Project Manager -> Excitation*** with the name of **3DC_Cell_Radome_In1_1** 3

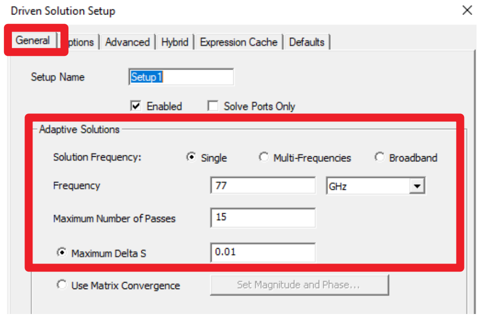


Define Solution Setup

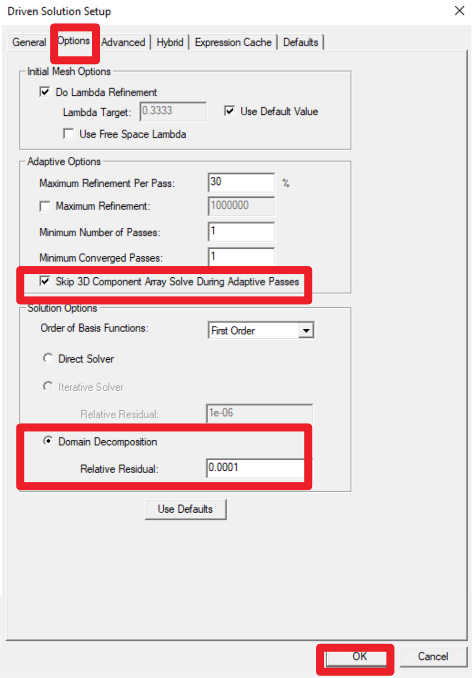
- In the ***Project Manager*** right click on ***Analysis -> Add Solution Setup -> Advanced…*** to open ***Driven Solution Setup*** window **1**
- In ***Driven Solution Setup*** dialog box under ***General*** tab set: **2**
  - ***Frequency: 77***
  - ***Maximum Number of Passes: 15***
  - ***Maximum Delta S: 0.01***
- In ***Driven Solution Setup*** dialog box under ***Options*** tab set: **3**
  - Check ***Skip 3D Component Array Solve During Adaptive Passes***
  - Select ***Domain Decomposition,*** here ***Direct Solver*** can also be used since ***distributed processor pool*** is not defined for this work
- Click ***OK*** **4**

> ***Note:*** by default, ***Skip 3D Component Array Solve During Adaptive Passes*** is selected. This option allows HFSS to perform individual adaptive passes for each 3D components, then with a converged mesh for each 3D component, a final adaptive pass is performed on the whole array. Keep this option selected in order to take advantage of speed and RAM efficiency of 3D Component Finite Array

2



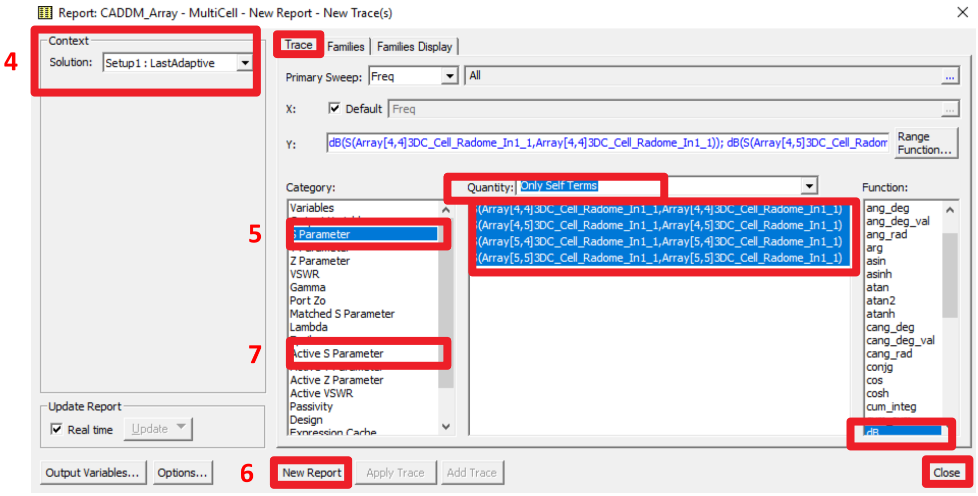
3



4

Analyze and View Results

- In the ***Ribbon*** with the ***Simulation*** tab selected click on the ***Validate*** icon **1**
- In the ***Ribbon*** with the ***Simulation*** tab selected click on the ***Analyze All*** icon **2**
- When the simulation is finished, in the ***Project Manager***, right-click on ***Results -> Create Modal Solution Data Report -> Data Table*** **3**
- In the ***Report*** window in ***Context*** set: **4**
  - ***Solution***: LastAdaptive
- In the ***Report*** window in ***Trace*** set: **5**
  - ***Category***: S Parameter
  - ***Quantity***: select ***Only Self Terms*** and select all
  - ***Function***: dB
- Click ***New Report***, and click ***Close*** **6**
- Repeat it for similar ***Data Table*** to show Active S Parameter **7**
- Save the project as **CADDM_Array.aedt** **8**



S Parameter Table 1

	Freq [GHz]	dB(S(Array[4,4]3DC_Cell_Radom... Setup1 : LastAdaptive	dB(S(Array[4,5]3DC_Cell_Radome_I... Setup1 : LastAdaptive	dB(S(Array[5,4]3DC_Cell_Radome_In... Setup1 : LastAdaptive	dB(S(Array[5,5]3DC_Cell_Radome_In... Setup1 : LastAdaptive
1	77.000000	-9.387170	-7.473782	-9.383816	-7.465124

Active S Parameter Table 1

	Freq [GHz]	dB(ActiveS(Array[4,4]3DC_Cell... Setup1 : LastAdaptive	dB(ActiveS(Array[4,5]3DC_Cell_Ra... Setup1 : LastAdaptive	dB(ActiveS(Array[5,4]3DC_Cell_Rad... Setup1 : LastAdaptive	dB(ActiveS(Array[5,5]3DC_Cell_Rado... Setup1 : LastAdaptive
1	77.000000	-6.766033	-5.428204	-6.780372	-5.416730

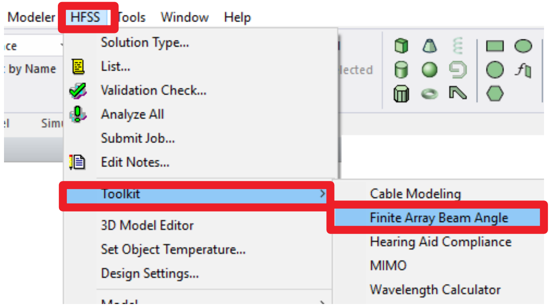
S-Parameters and Active S-Parameters values at 77 GHz for the four center elements of the array

Steer the Beam of Finite Array

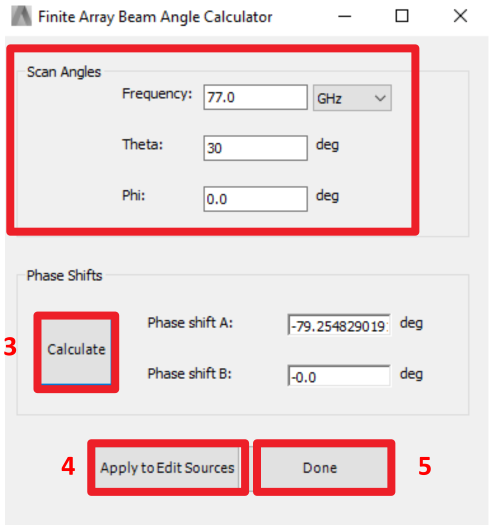
Finite Array Beam Angle Calculator Toolkit

- In order to steer the beam of finite array HFSS provides an automated beam angle calculation of phase delay between the unit cells. This toolkit adjusts the ***Edit Sources*** values accordingly
- To open the calculator, select ***HFSS -> Toolkit -> Finite Array Beam Angle*** 1
- In ***Finite Array Beam Angle Calculator*** choose the desired ***Theta*** and ***Phi***, here ***Theta***=30 and ***Phi***=0 2
- Click ***Calculate*** 3
- Click ***Apply to Edit Sources*** 4
- Click ***Done*** 5
- Check the ***Edit Sources*** values In ***Project Manager*** right-click on ***Excitations -> Edit Sources*** 6

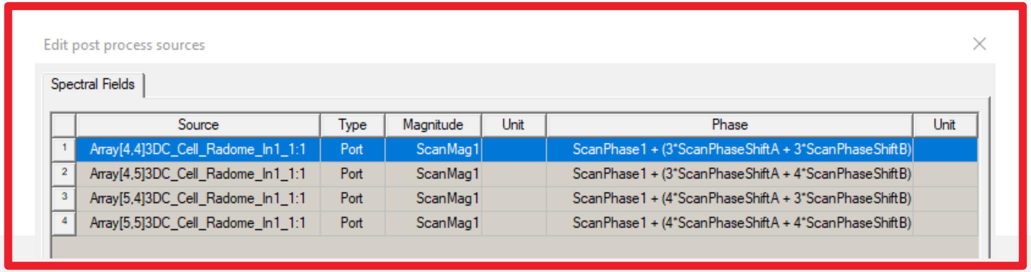
1



2 3 4 5



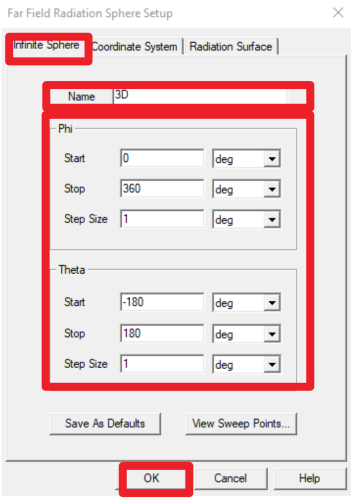
6



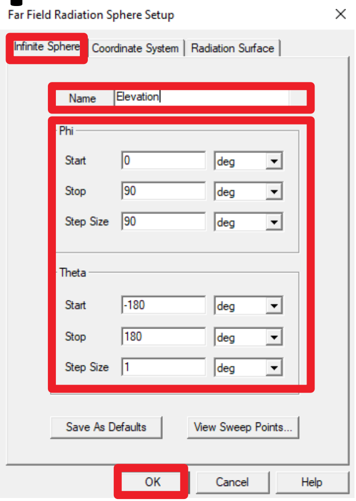
Create Far-Field Radiation Infinite Spheres

- In the ***Project Manager***, right-click on ***Radiation*** and select ***Insert Far Field Setup -> Infinite Sphere*** 1
- In ***Far Field Radiation Sphere Setup*** window under ***Infinite Sphere*** tab set: 2
  - ***Name***: 3D
  - ***Phi***: Start: 0, Stop: 360, Step Size: 1
  - ***Theta***: Start: -180, Stop: 180, Step Size: 1
  - Click ***OK***
- Repeat the same procedure for Elevation ***Radiation Sphere*** with this differences 3
  - ***Name***: Elevation
  - ***Phi***: Start: 0, Stop: 90, Step Size: 90
  - ***Theta***: Start: -180, Stop: 180, Step Size: 1
- The defined ***Radiation Spheres*** can be found In ***Project Manager -> Radiation*** 4

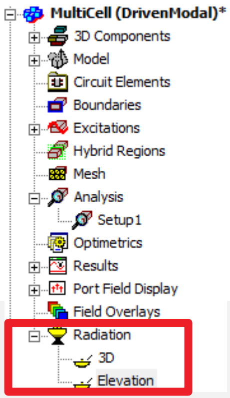
2



3

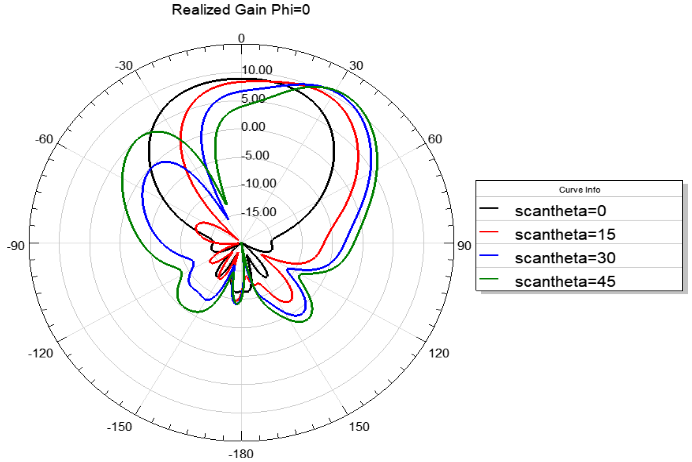


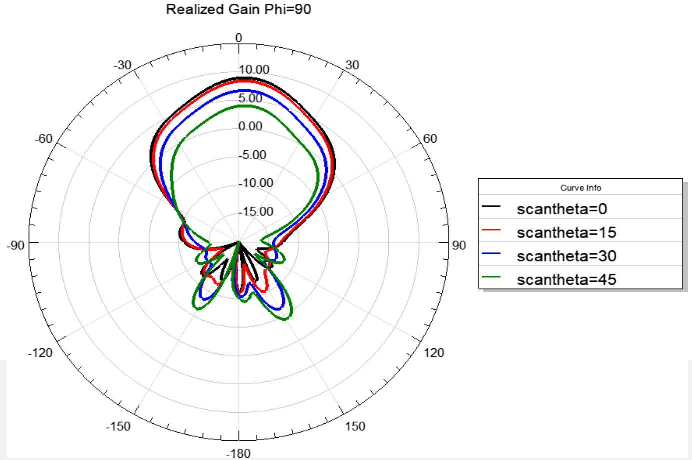
4



View the Steered Beam (Elevation)

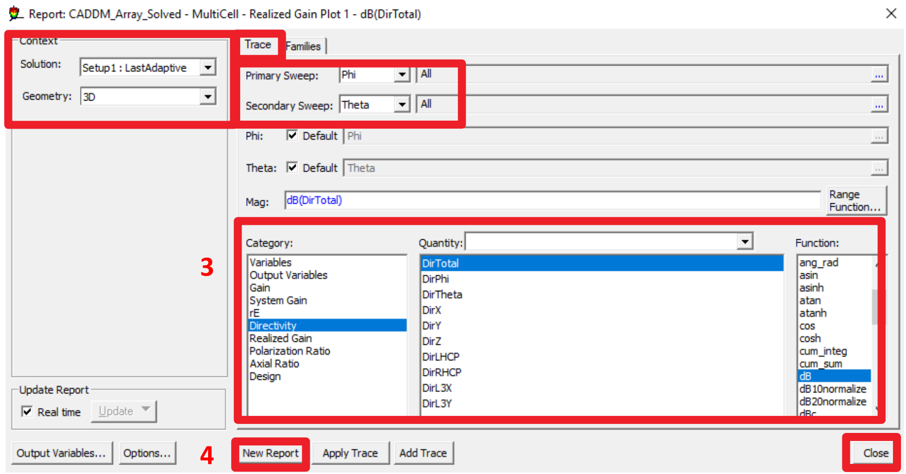
- Due to beam steering, new variables are generated automatically which can be found in design ***Properties*** 1
- In ***Finite Array Beam Angle Calculator*** choose the ***Theta***= 0, 15, 30, 45 and ***Phi***=0, and click ***Calculate*** and ***Apply to Edit Sources*** for each of them to be able to see the generated patterns separately 2
- In the ***Project Manager*** right-click on ***Results*** and select ***Create Far Fields Report -> Radiation Pattern*** 3
- In the ***Report*** window in ***Context*** set: 4
  - ***Solution***: Setup1: LastAdaptive, ***Geometry***: Elevation
- In the ***Report*** window in ***Trace*** set: 5
  - ***Primary Sweep***: Theta, ***Category***: Realized Gain, ***Quantity***: RealizedGainTotal, ***Function***: dB
- In the ***Report*** window in ***Families*** set: 6
  - ***Phi***: 0deg (first plot), 90deg (second plot)
- Click ***New Report***, and click ***Close*** 7

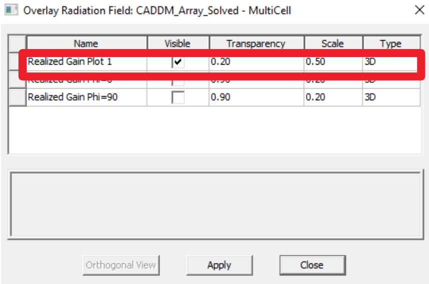


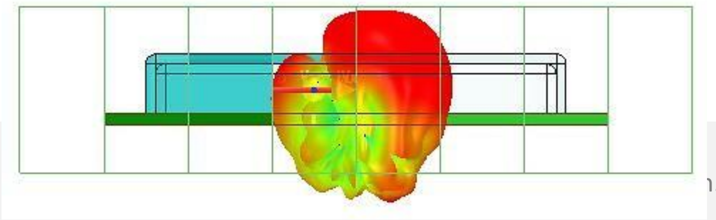


View the Beam (3D)

- In the ***Project Manager*** right-click on ***Results*** and select ***Create Far Fields Report -> 3D Polar Plot*** 1
- In the ***Report*** window in ***Context*** set: 2
  - ***Solution***: Setup1: LastAdaptive, ***Geometry***: 3D
- In the ***Report*** window in ***Trace*** set: 3
  - ***Primary/Secondary Sweep***: Phi/Theta, ***Category***: Directivity, ***Quantity***: DirTotal, ***Function***: dB
- Click ***New Report***, and click ***Close*** 4
- To overlay the radiation pattern on the model with scan angle theta set to 30 in the ***Project Manager***, right-click on ***Field Overlays -> Plot Fields -> Radiation Field…***. Size, transparency and visibility can be controlled in the ***Overlay Radiation Field*** dialog box 5
- Save the project as **CADDM_Array_Solved.aedt** 6



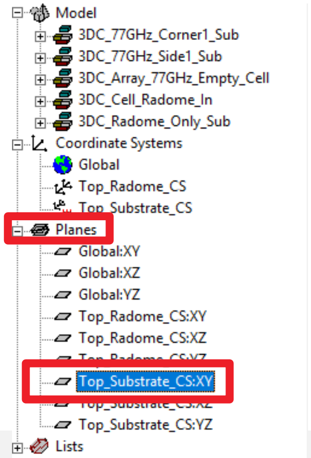




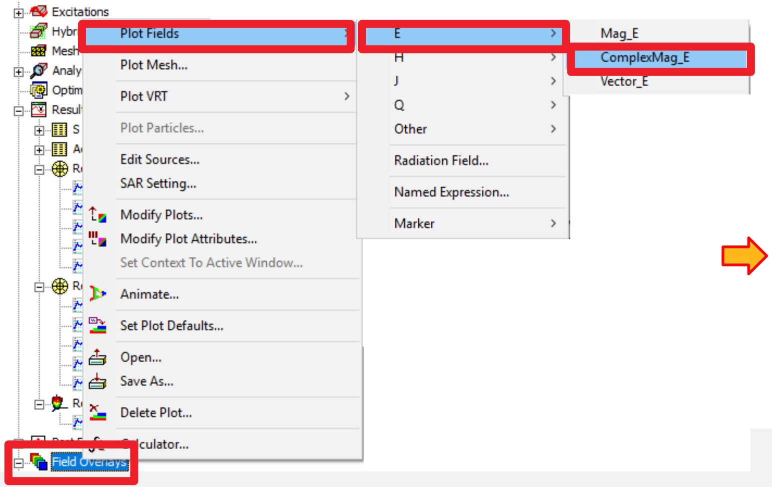
View the Electric Field

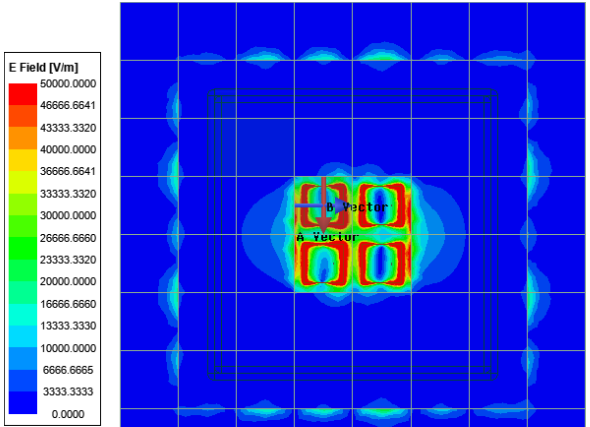
- To display the complex magnitude of the electric field ***(Complex Mag E)*** on substrate/top surface/XY plane, in ***3D Modeler Tree*** select ***Planes -> Top_Substrate_CS:XY*** 1
- In ***Project Manager -> Field Overlays*** uncheck the visibility of the 3D radiation pattern 2
- To plot the fields in ***Project Manager*** right-click on ***Field Overlays -> Plot Fields -> E -> Complex_Mag_E*** 3
- In ***Create Field Plot*** window select ***Done*** 4
- The field can be displayed for different scan angles by changing the ***Theta*** and ***Phi*** in ***Finite Array Beam Angle Calculator*** 5

1



3



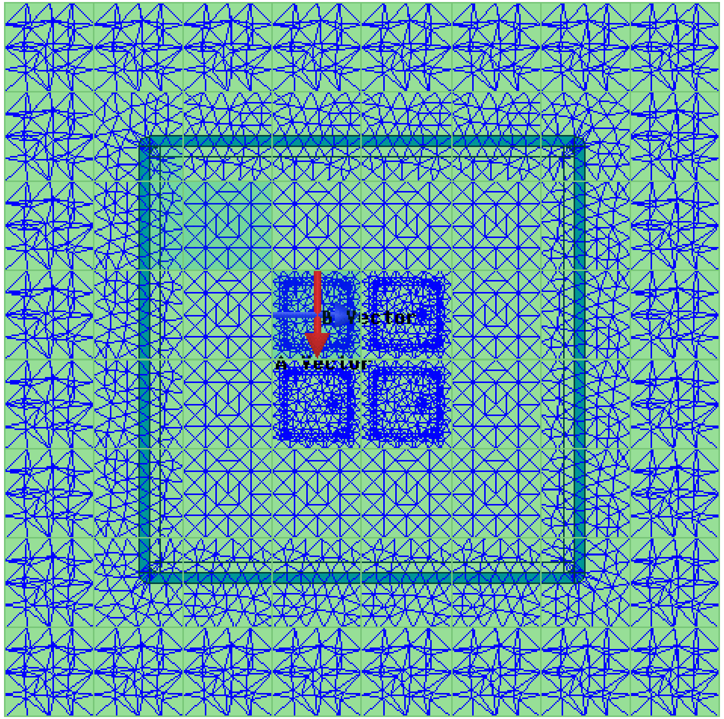


Complex Mag E at Theta=0 and Phi=0

View the Mesh

- To display the mesh on substrate/top surface/XY plane, in ***3D Modeler Tree*** select ***Planes -> Top_Substrate_CS:XY*** **1**
- In ***Project Manager -> Field Overlays*** uncheck the visibility of the ComplexMag_E **2**
- To plot the mesh in ***Project Manager*** right-click on ***Field Overlays -> Plot Mesh…*** **3**
- In ***Create Mesh Plot*** window select ***Done*** **4**

Note: The displayed mesh shows how HFSS creates non-conformal mesh interfaces between each 3D component unit cells, which reduces the memory footprint and improves the simulation performance

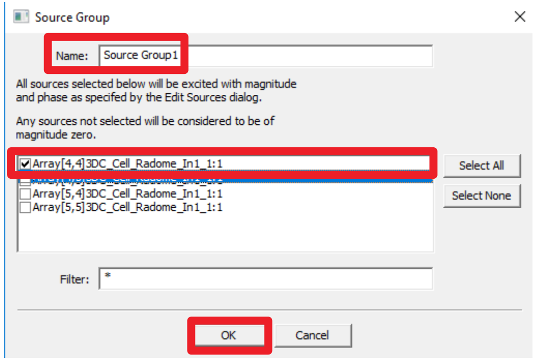


Set Up Source Groups for Finite Array

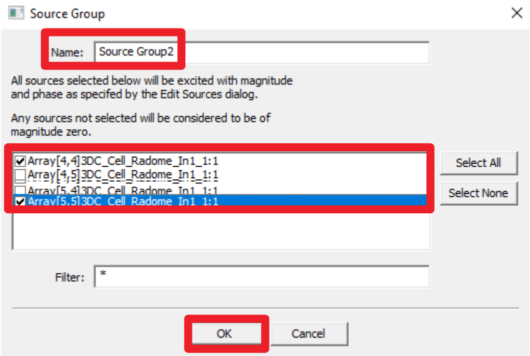
- In ***Project Manager*** right-click on ***Radiation -> Insert Far Field Setup -> Source Group…*** 1
- In ***Source Group*** window use check boxes and/or ***Select***
- ***All/None*** buttons to specify which sources to excite with the magnitude and phase as specified in the ***Edit Sources*** dialog. 2 Any sources not selected will be assigned magnitude zero 3
- Click ***OK*** to close the dialog box 4
- The ***Source Groups*** appear in the ***Project Tree*** under ***Radiation***

Note: HFSS Edit Sources has had the concept of 'Source Contexts' for far field plots which is adequate for simple antenna simulation, but it fails to meet users' needs with respect to finite arrays. So, when multiple 'Source Contexts' are needed, and each context maintain its own set of active excitation in a finite array, the ***Source Group*** is useful. This feature allows users to activate a subset of the virtual excitations instead of the physical excitations associated with the unit cell(s).
In Radiated Fields report, the created Source Groups will be listed with other sources in the Context window.

2,3

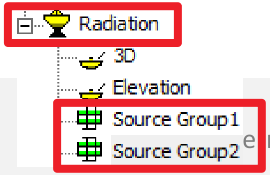


Source Group1



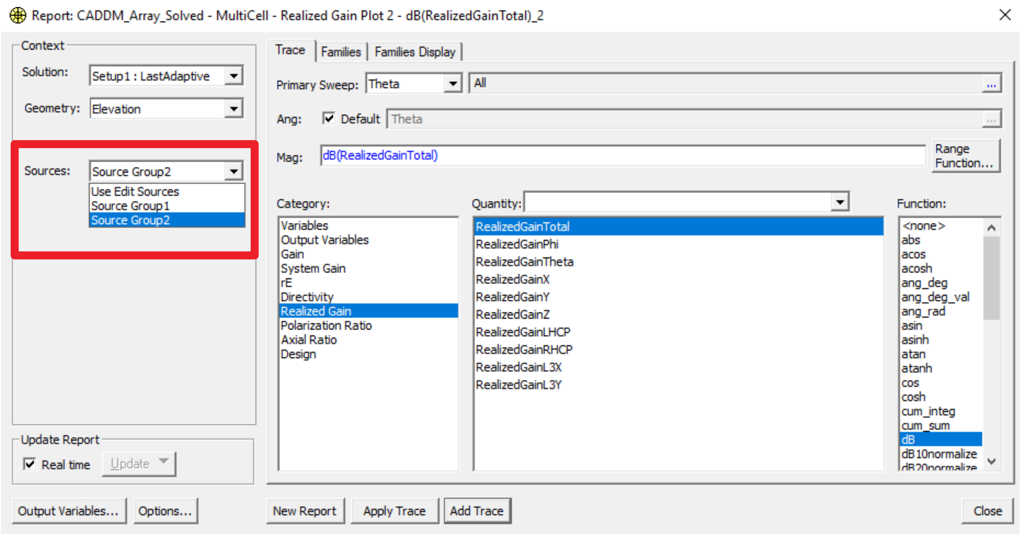
Source Group2

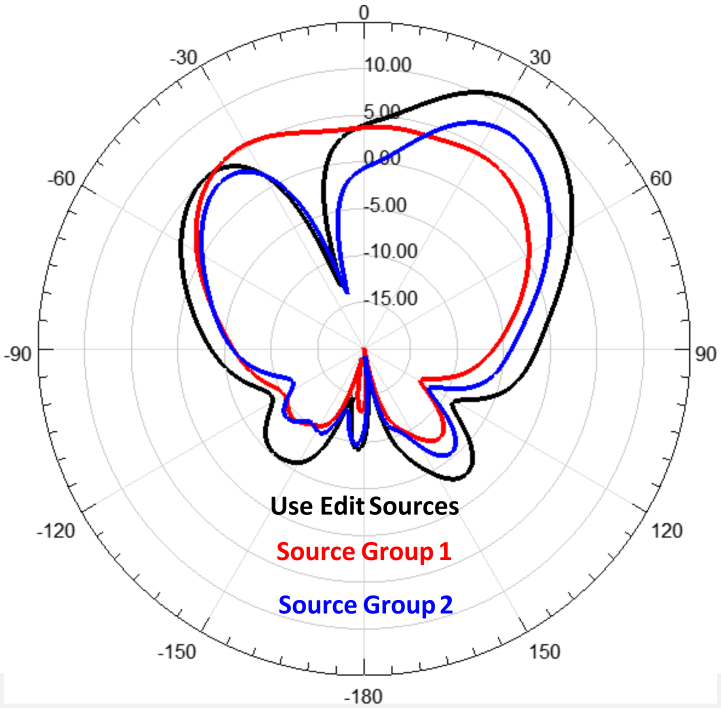
4



View the Radiation Patterns Associated to Source Groups

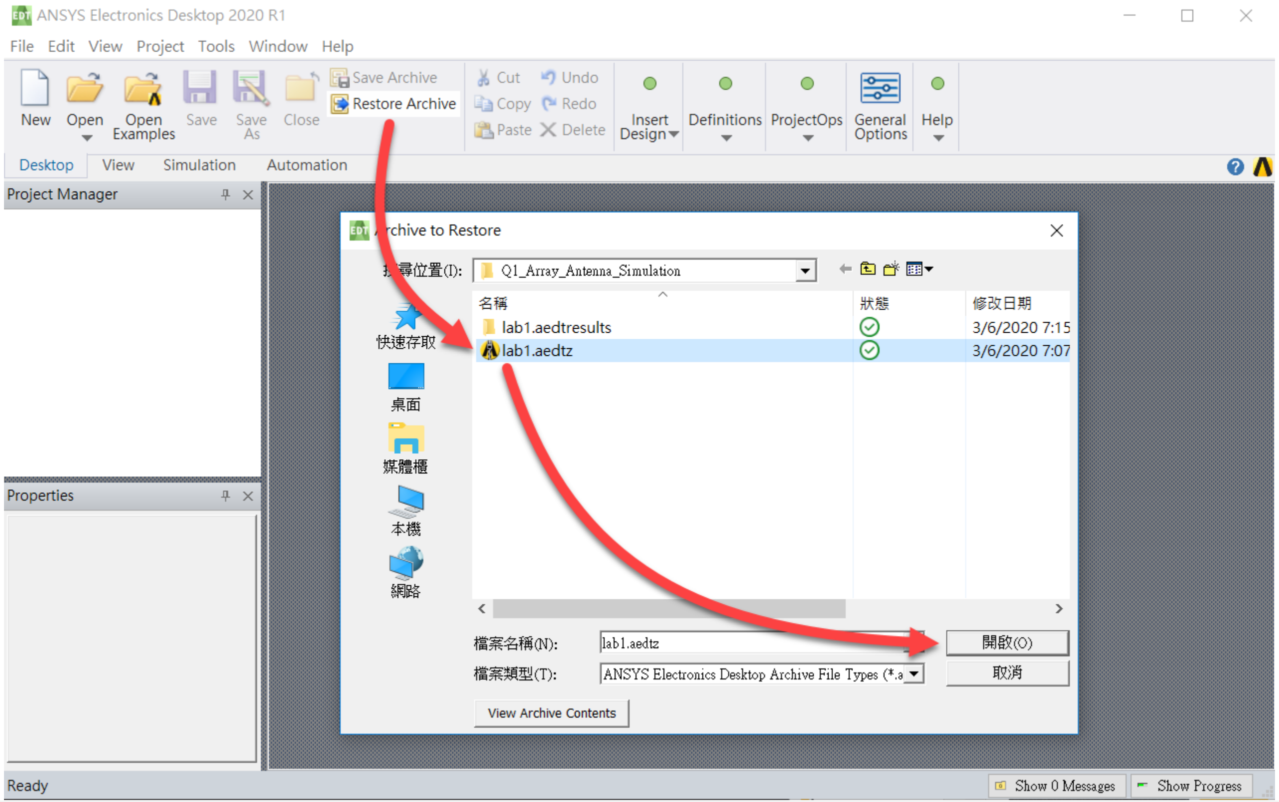
- Plot the ***Radiation Pattern*** at ***Theta Scan***=45 in ***Elevation Cut*** and ***Phi***=0, with activating different ***Sources*** 1
- In the ***Report*** window in ***Context*** set the ***Sources*** 2
- Save the project as **CADDM_Array_Solved.aedt** 3





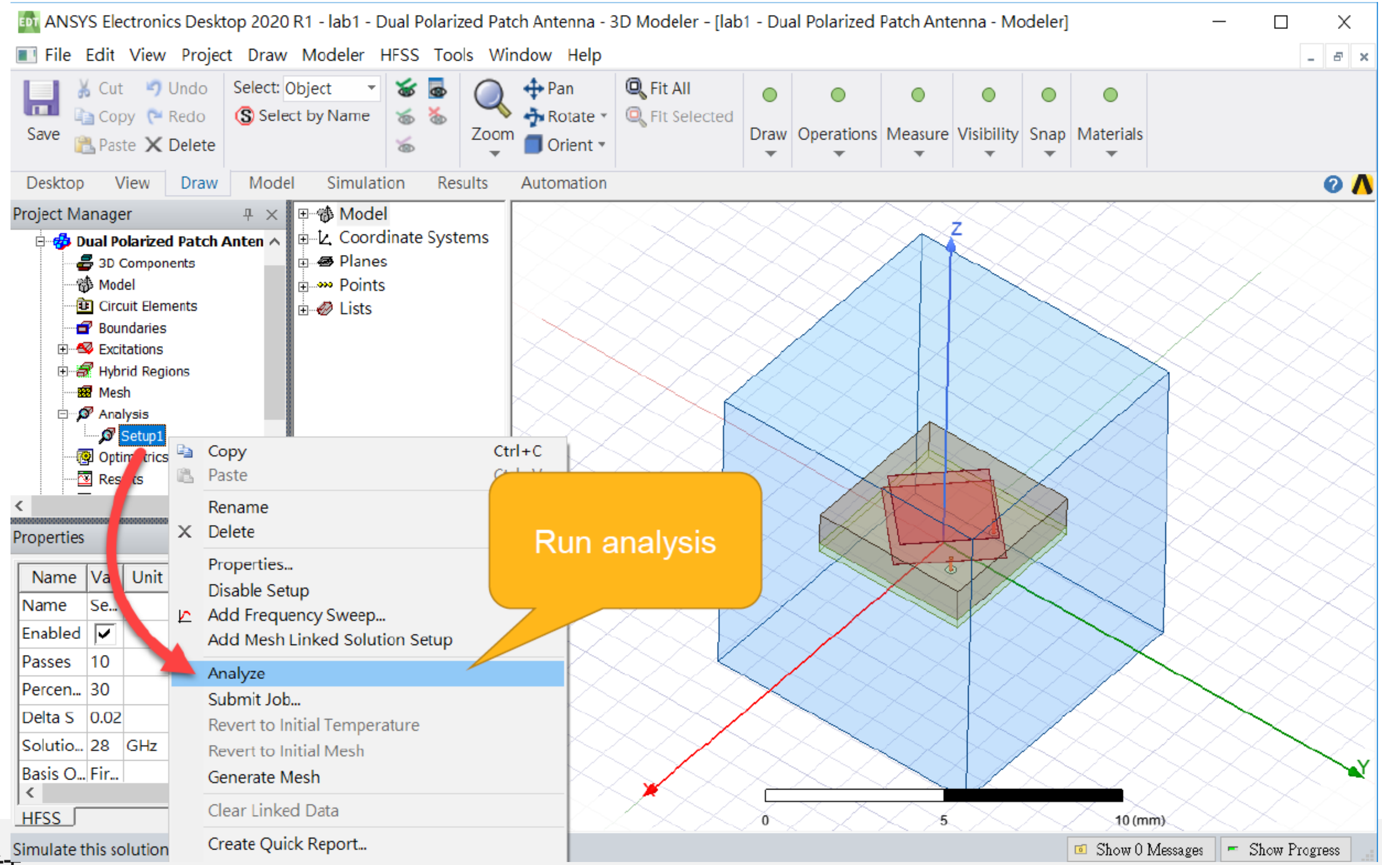
Lab 1. Unit Cell

Open Lab1

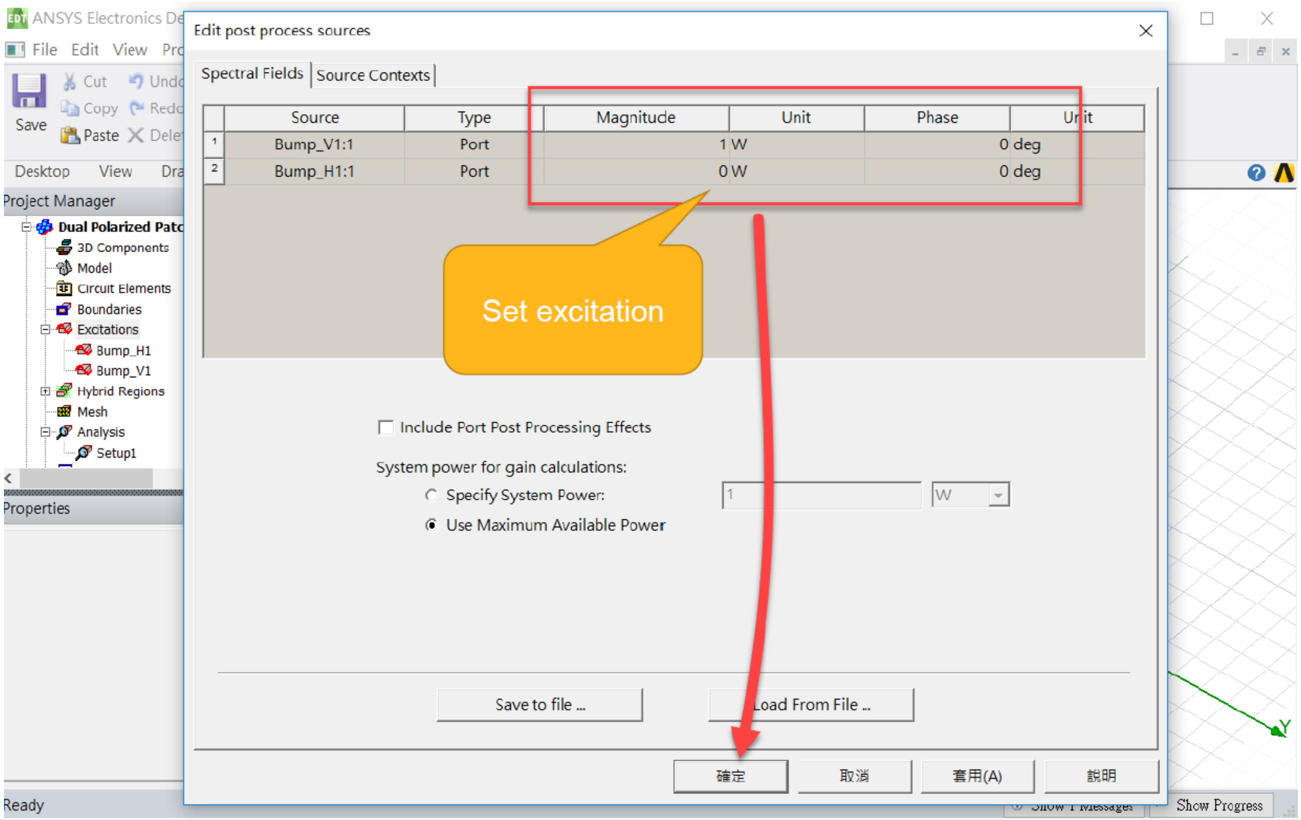


ANSYS Electronics Desktop 2020 R1
File Edit View Project Tools Window Help
New
Open
Open Examples
Save
Save As
Close
Save Archive
Restore Archive
Cut
Copy
Paste
Undo
Redo
Delete
Insert Design
Definitions
ProjectOps
General Options
Help
Desktop
View
Simulation
Automation
Project Manager
Properties
Ready
Show 0 Messages
Show Progress
rchive to Restore
尋位置(I):
Q1_Array_Antenna_Simulation
名稱
狀態
修改日期
lab1.aedtresults
3/6/2020 7:15
lab1.aedtz
3/6/2020 7:07
快速存取
桌面
媒體櫃
本機
網路
檔案名稱(N):
lab1.aedtz
開啟(O)
檔案類型(T):
ANSYS Electronics Desktop Archive File Types (*.a
取消
View Archive Contents

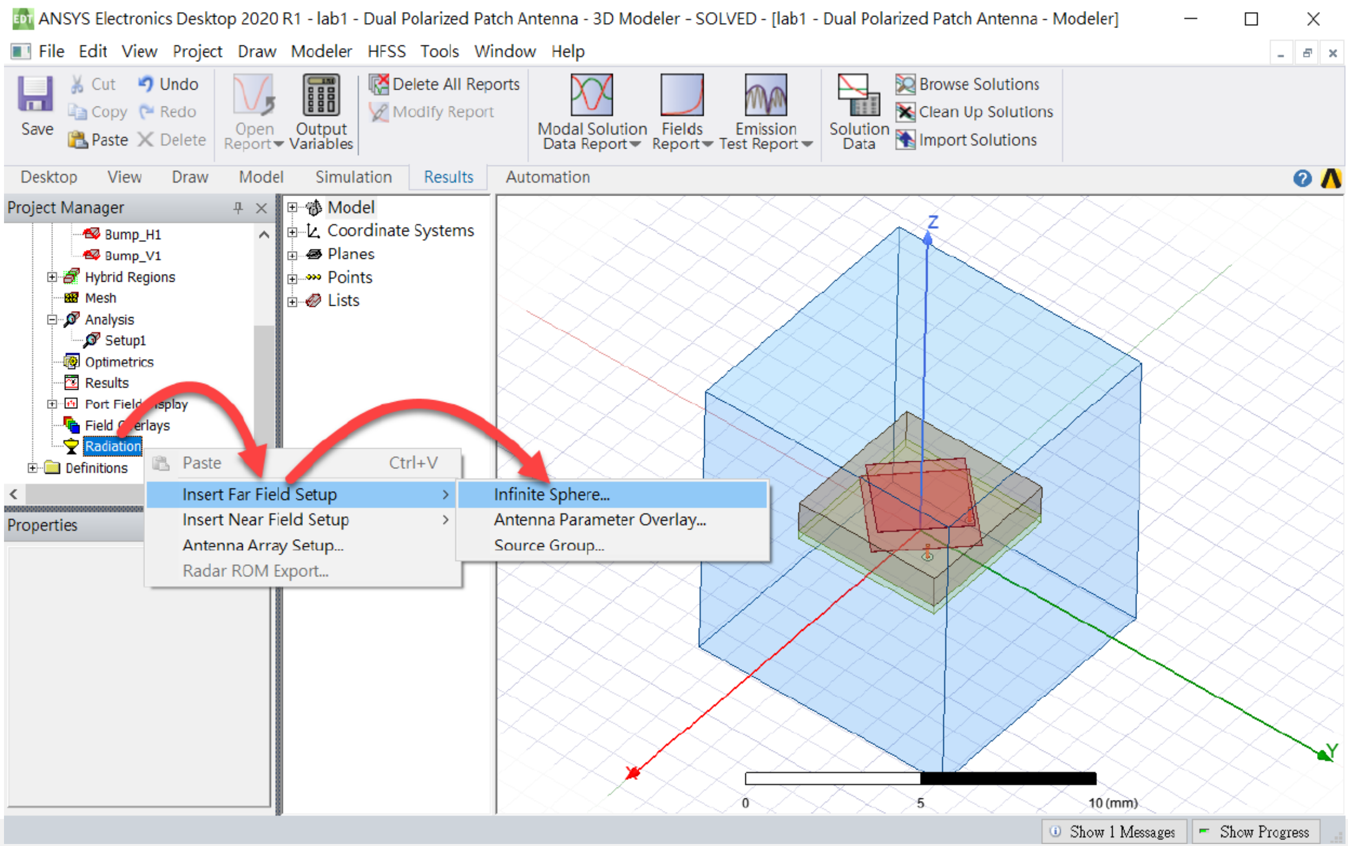
Run Analysis



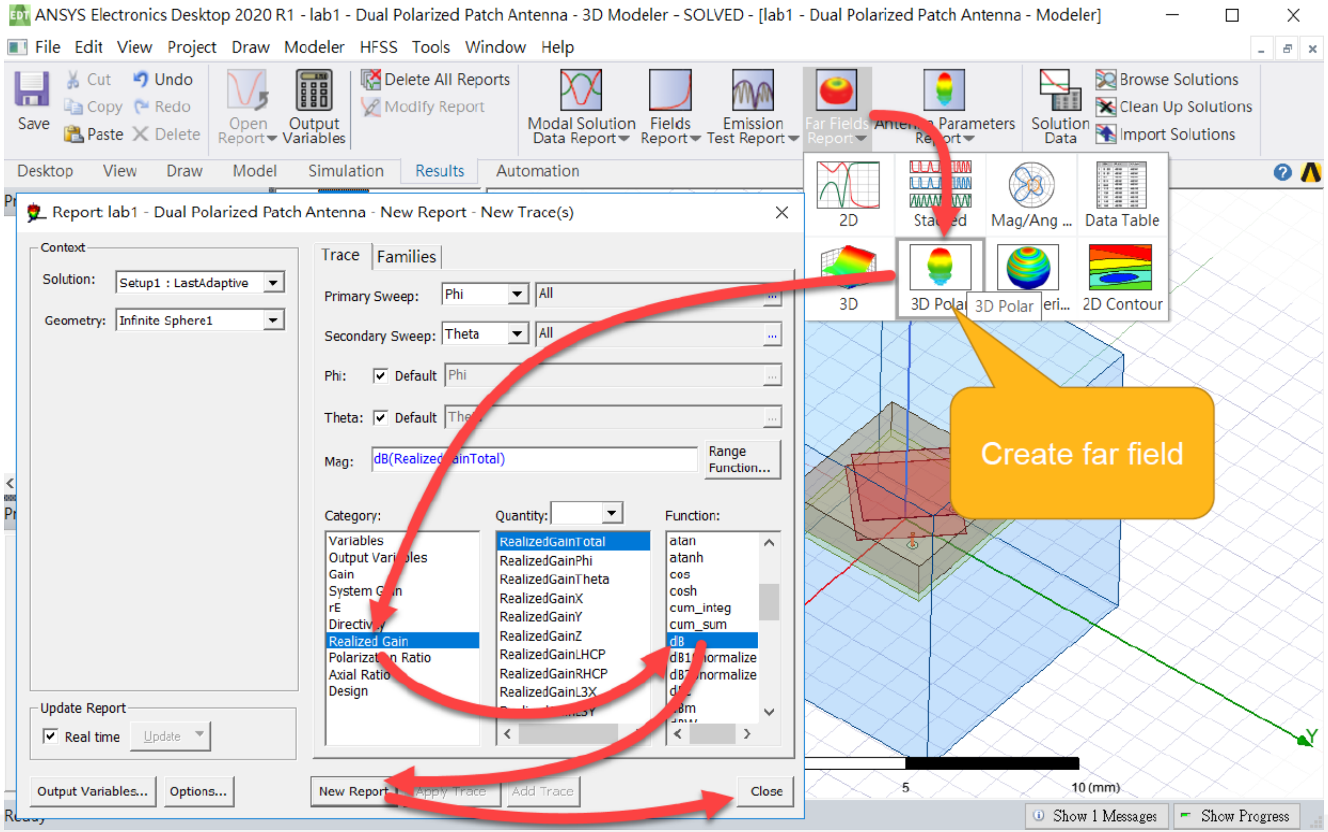
Set Excitation



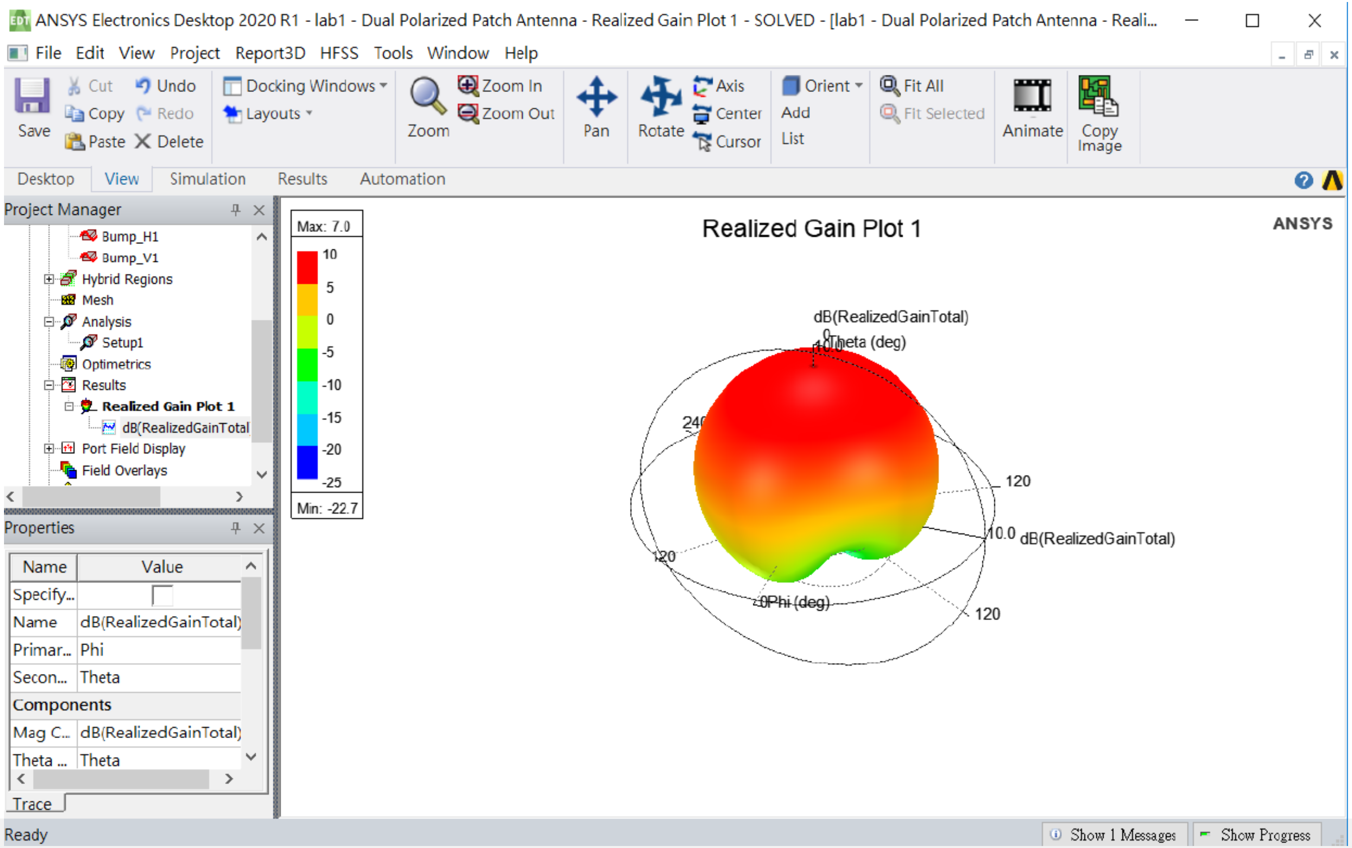
Insert Far Field Setup



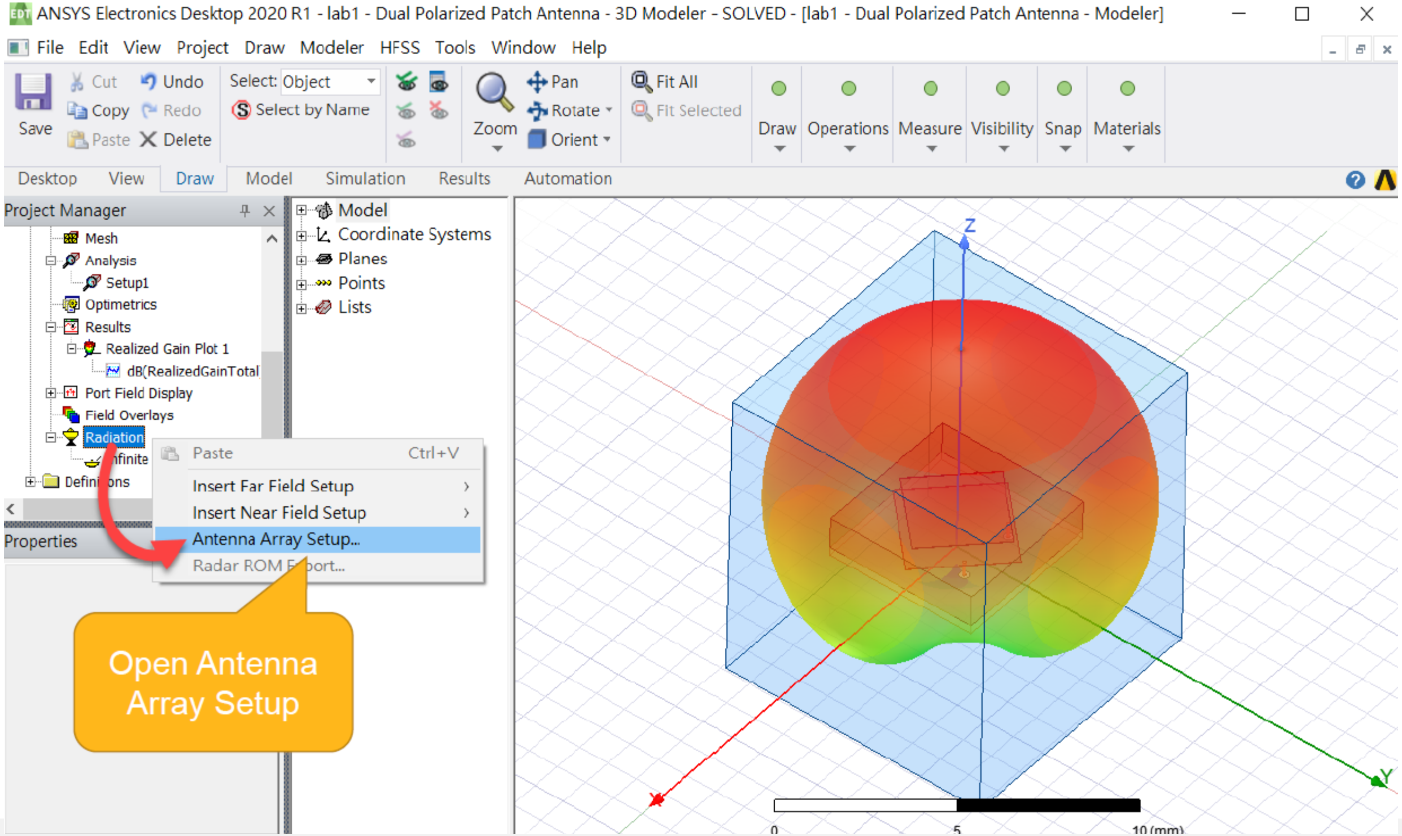
Plot Far Field



Far Field Result

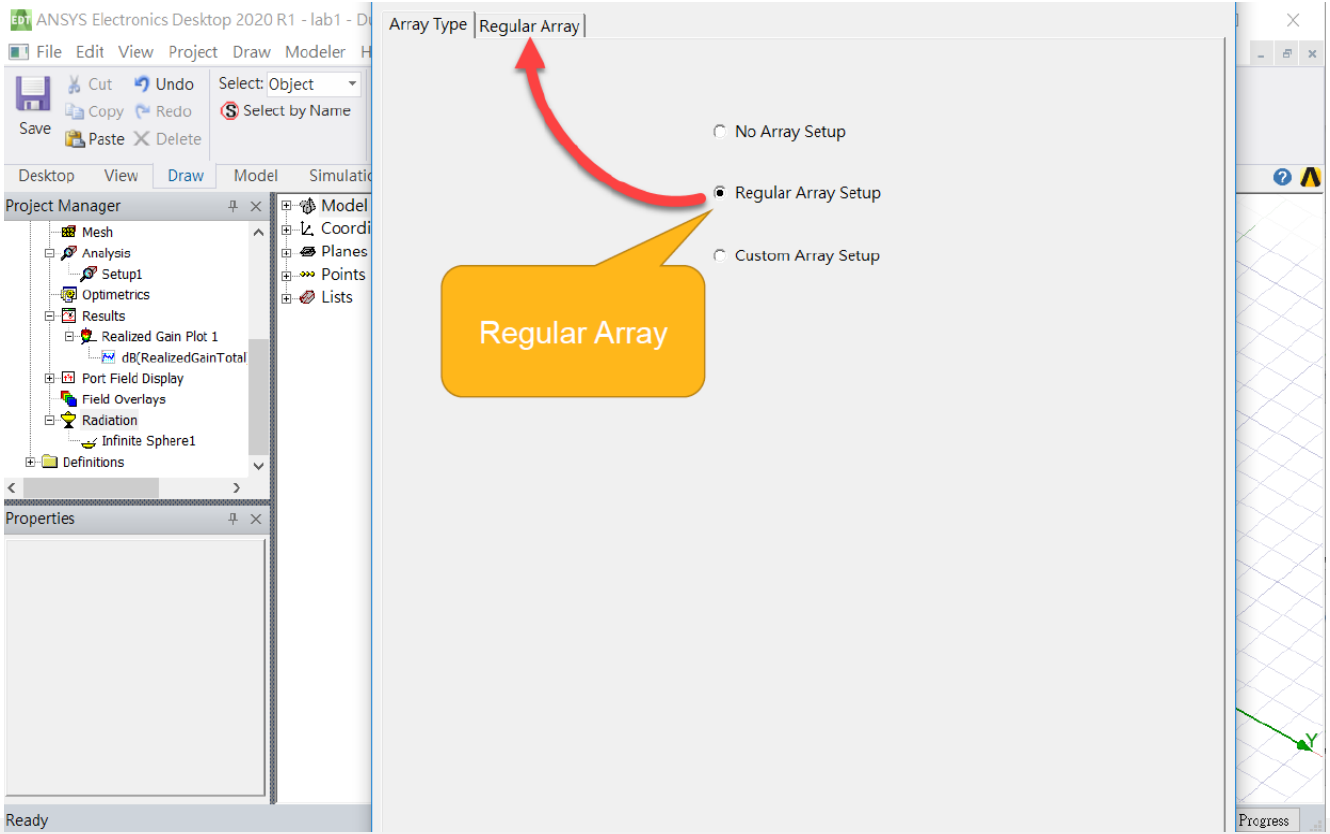


Array Setup

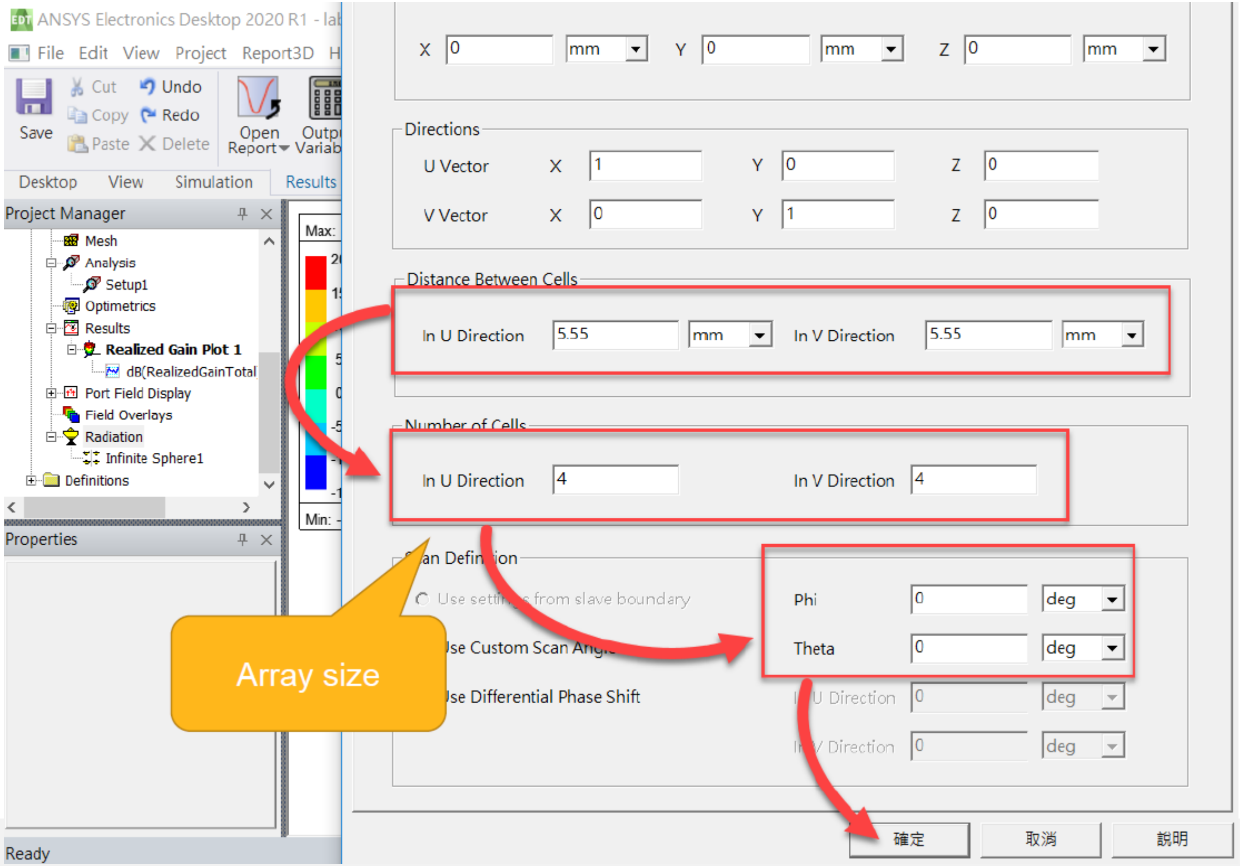


ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [lab1 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut
Copy
Paste
Undo
Redo
Delete
Select: Object
Select by Name
Zoom
Pan
Rotate
Orient
Fit All
Fit Selected
Draw
Operations
Measure
Visibility
Snap
Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
Mesh
Analysis
Setup1
Optimetrics
Results
Realized Gain Plot 1
dB(RealizedGainTotal
Port Field Display
Field Overlays
Radiation
Properties
Model
Coordinate Systems
Planes
Points
Lists
Paste Ctrl+V
Insert Far Field Setup
Insert Near Field Setup
Antenna Array Setup...
Open Antenna Array Setup
Z
Y
0
5
10 (mm)

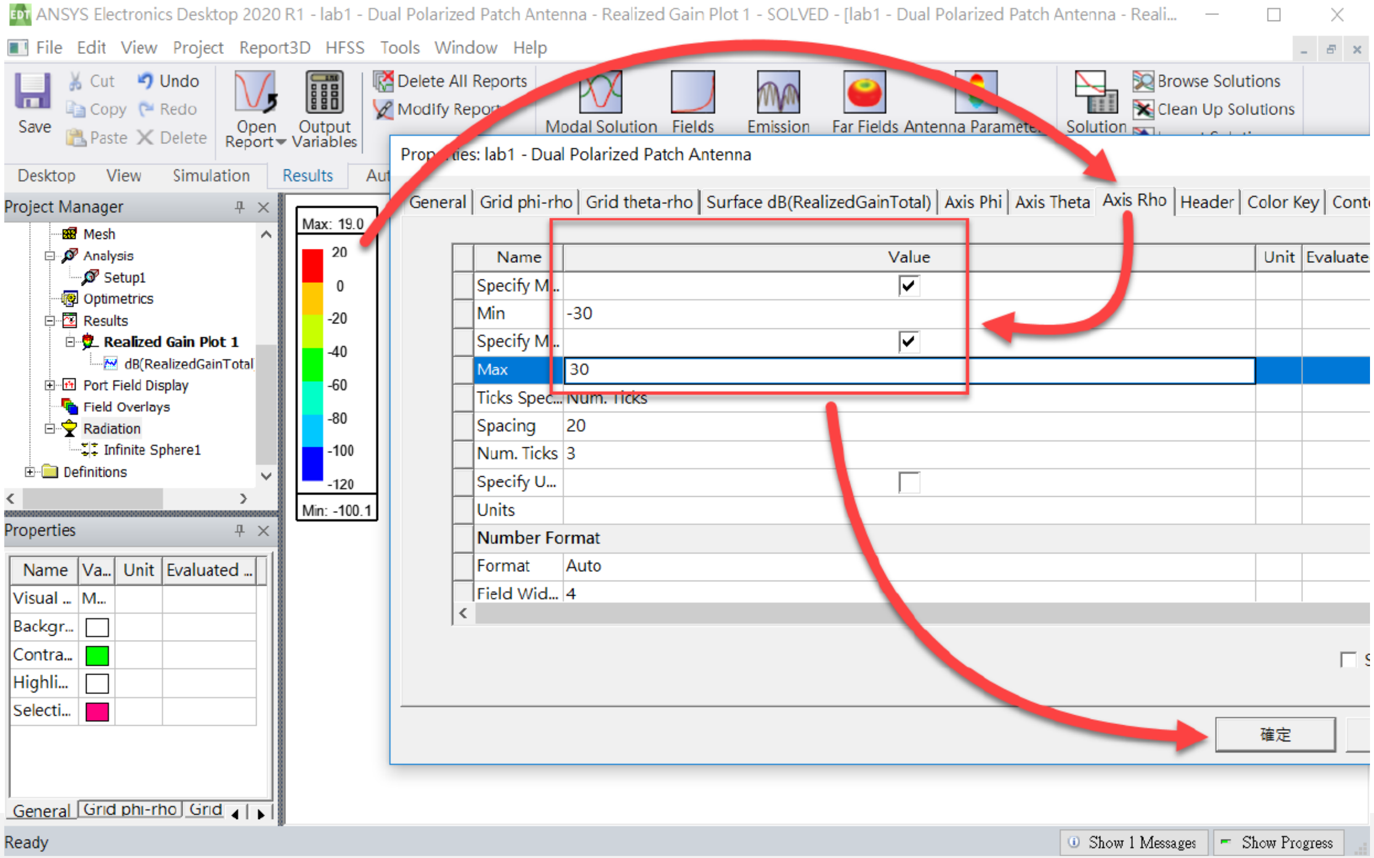
Regular Array



Set Array Size

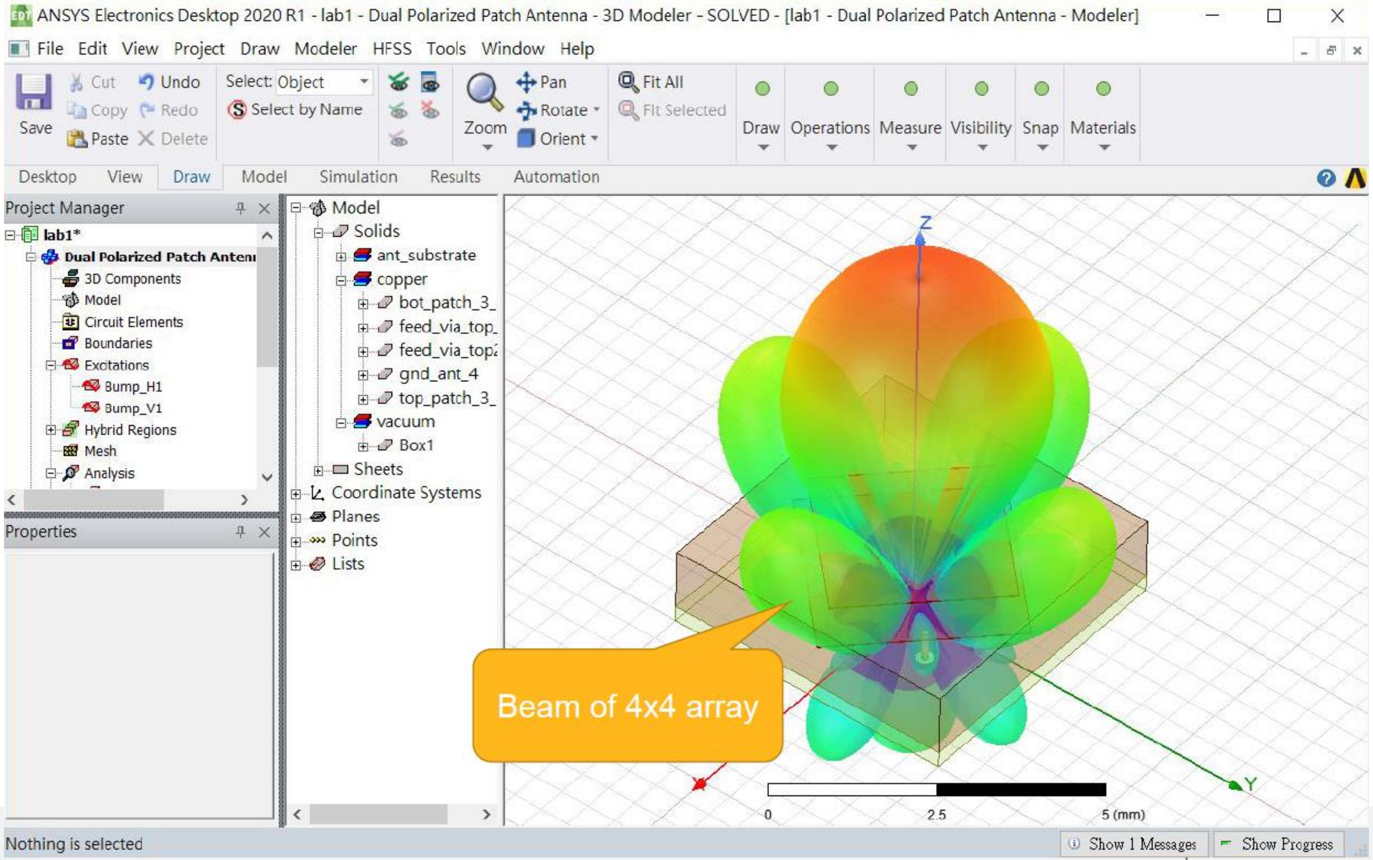


Axis Range



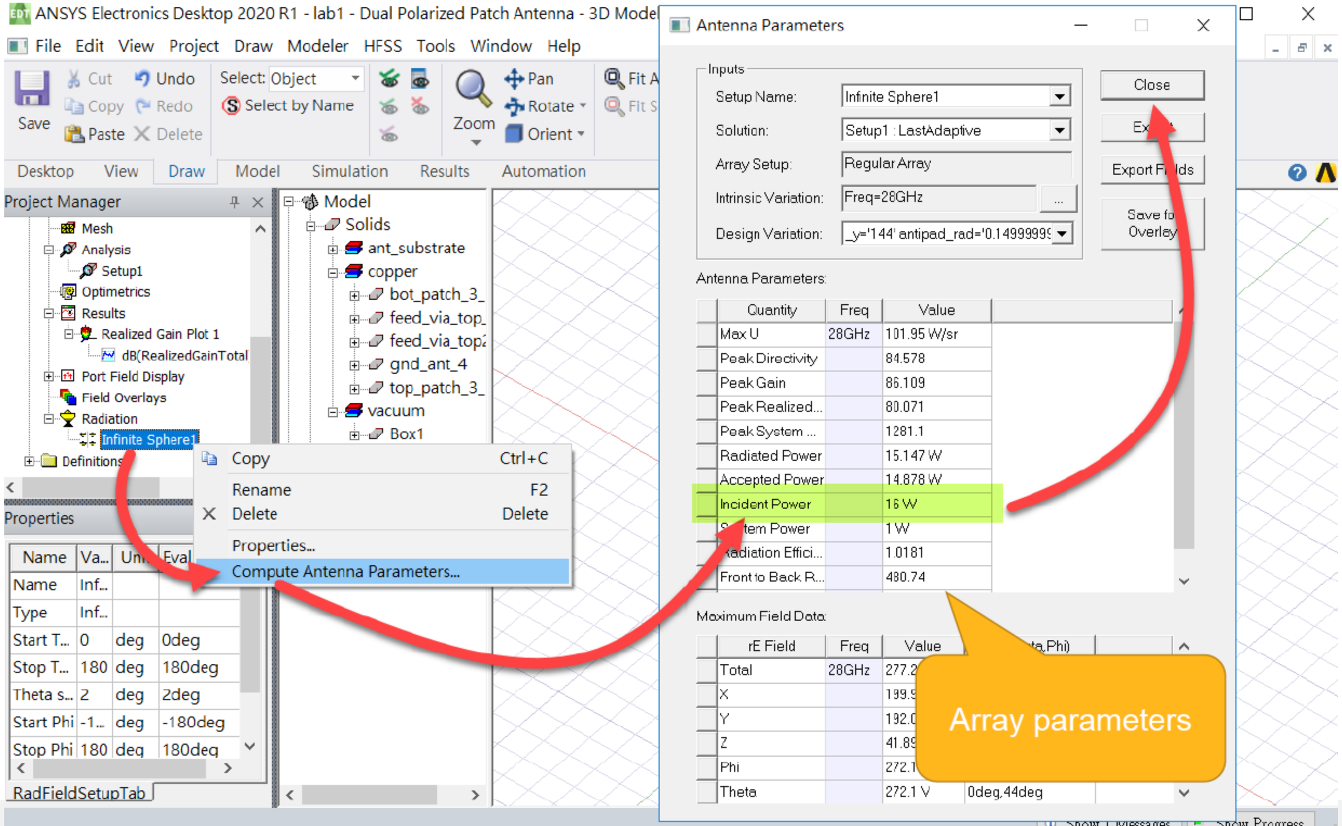
ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - Realized Gain Plot 1 - SOLVED - [lab1 - Dual Polarized Patch Antenna - Reali...
Properties: lab1 - Dual Polarized Patch Antenna
General
Grid phi-rho
Grid theta-rho
Surface dB(RealizedGainTotal)
Axis Phi
Axis Theta
Axis Rho
Header
Color Key
Name
Value
Unit
Evaluate
Specify M..
Min
-30
Specify M..
Max
30
Ticks Spec...
Num. Ticks
Spacing
20
Num. Ticks
3
Specify U...
Units
Number Format
Format
Auto
Field Wid...
4
確定
Project Manager
Mesh
Analysis
Setup1
Optimetrics
Results
Realized Gain Plot 1
dB(RealizedGainTotal
Port Field Display
Field Overlays
Radiation
Infinite Sphere1
Definitions
Max: 19.0
Min: -100.1
Ready

Array Beam

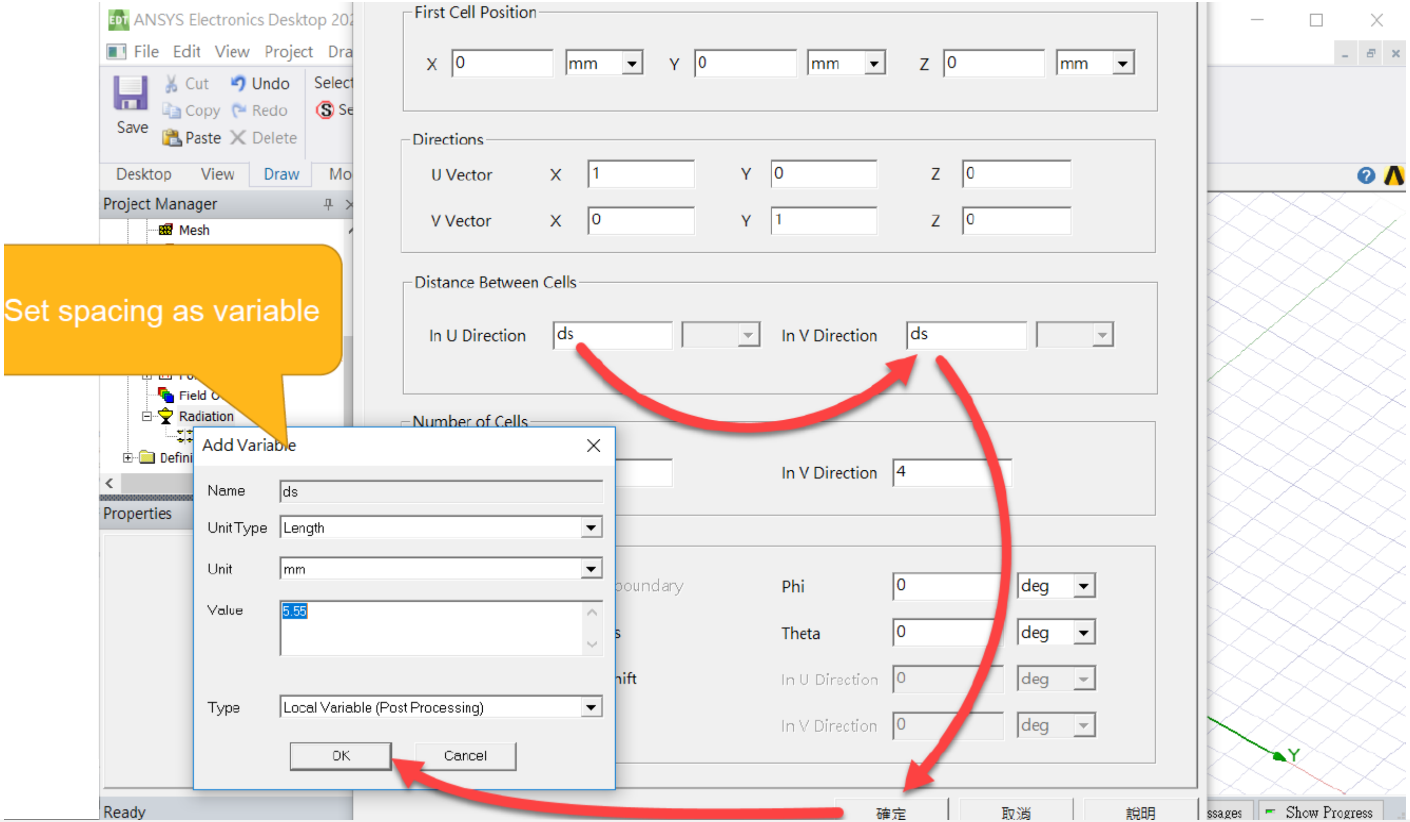


ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [lab1 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save Cut Copy Paste Undo Redo Delete
Select: Object
Select by Name
Zoom Pan Rotate Orient
Fit All Fit Selected
Draw Operations Measure Visibility Snap Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
lab1*
Dual Polarized Patch Anten
3D Components
Model
Circuit Elements
Boundaries
Excitations
Bump_H1
Bump_V1
Hybrid Regions
Mesh
Analysis
Properties
Model
Solids
ant_substrate
copper
bot_patch_3_
feed_via_top_
feed_via_top
gnd_ant_4
top_patch_3_
vacuum
Box1
Sheets
Coordinate Systems
Planes
Points
Lists
Z
Y
0
2.5
5 (mm)
Beam of 4x4 array
Nothing is selected
Show 1 Messages
Show Progress

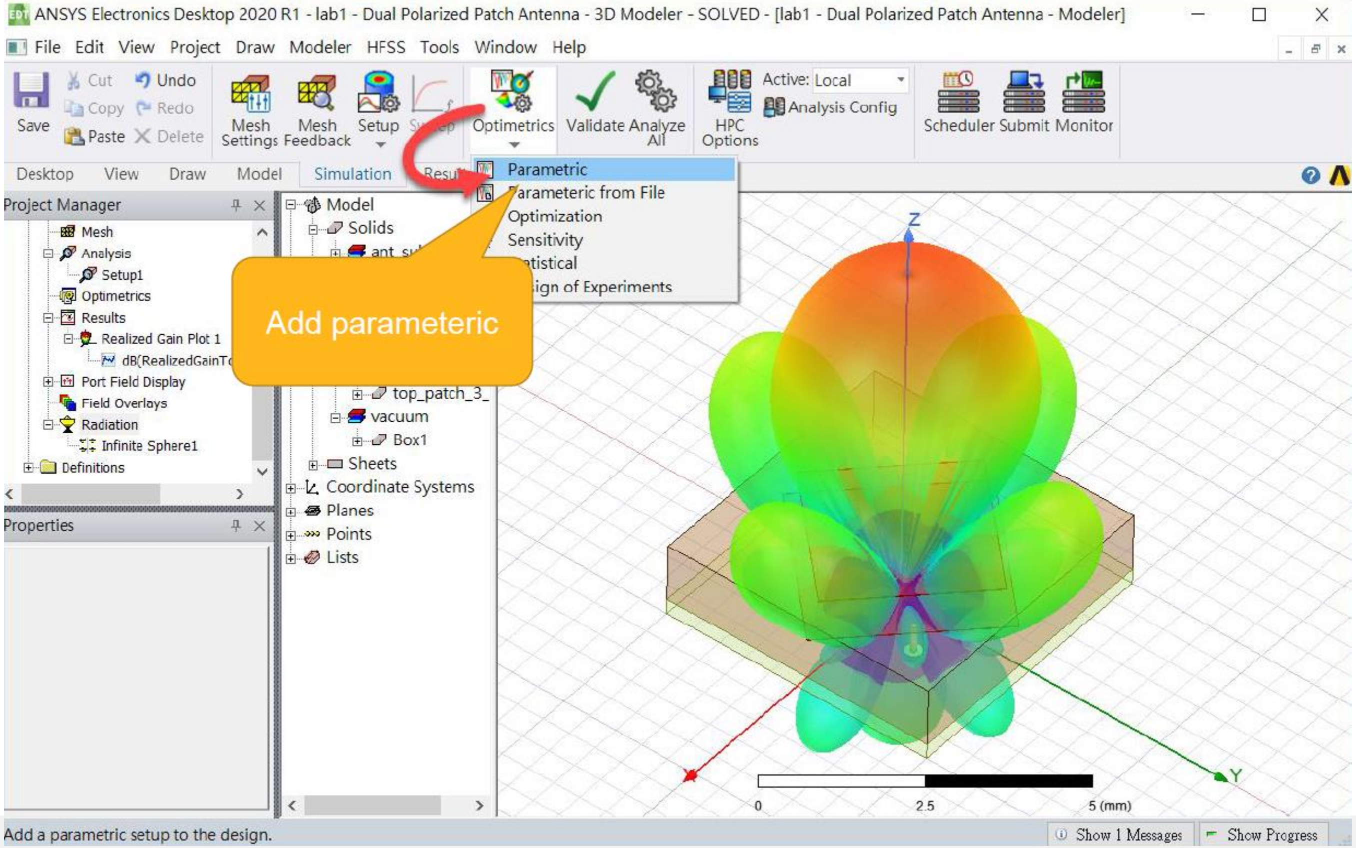
Array Parameters



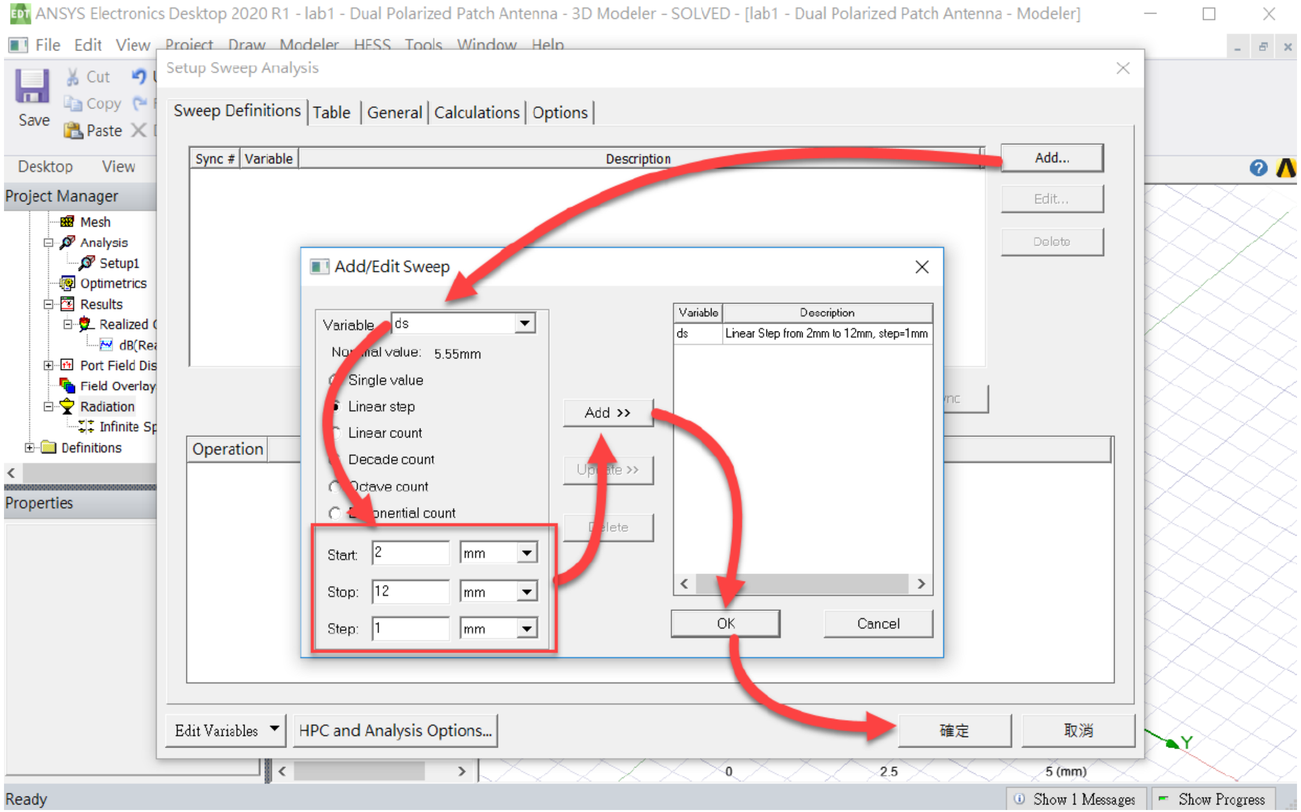
Parametric with Distance



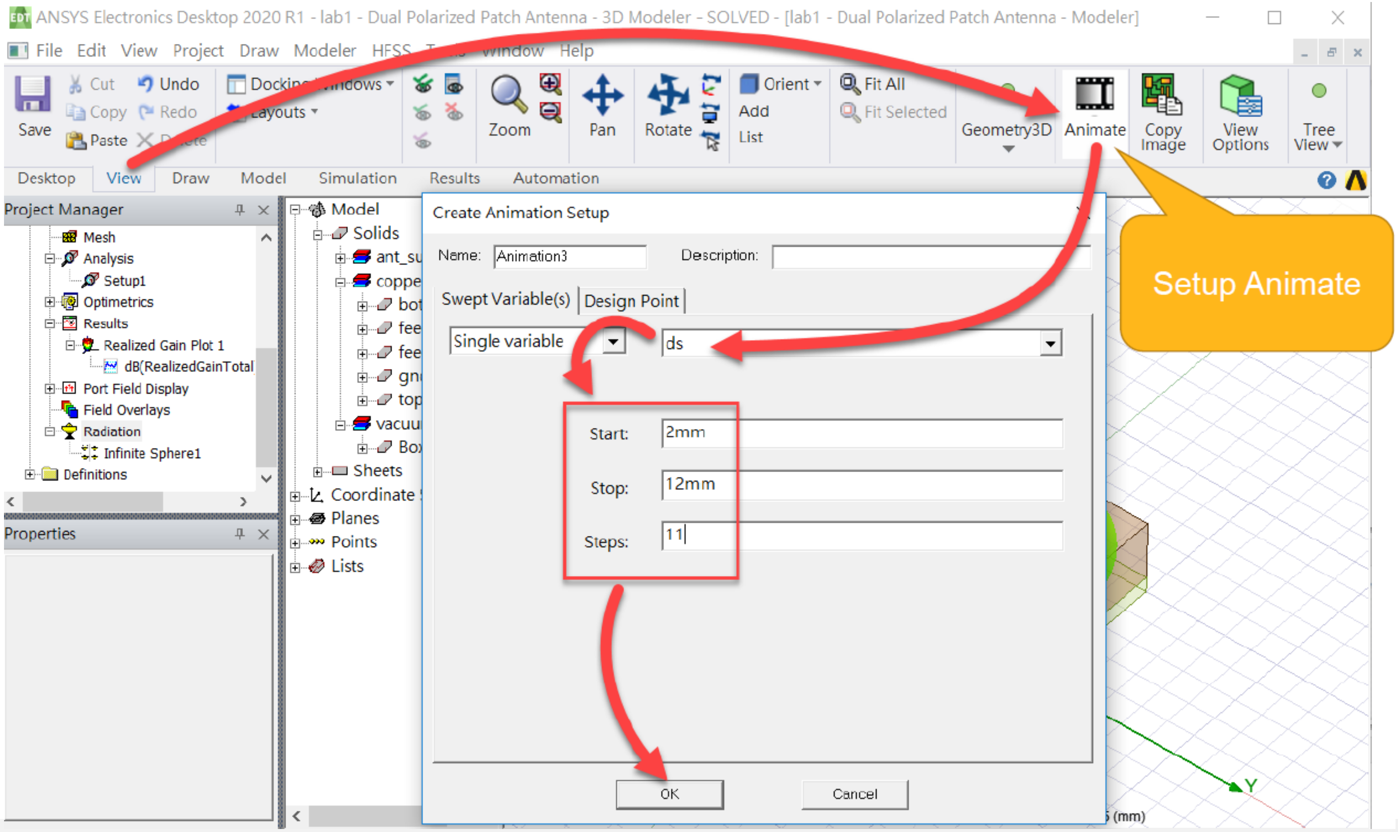
Add Parameteric



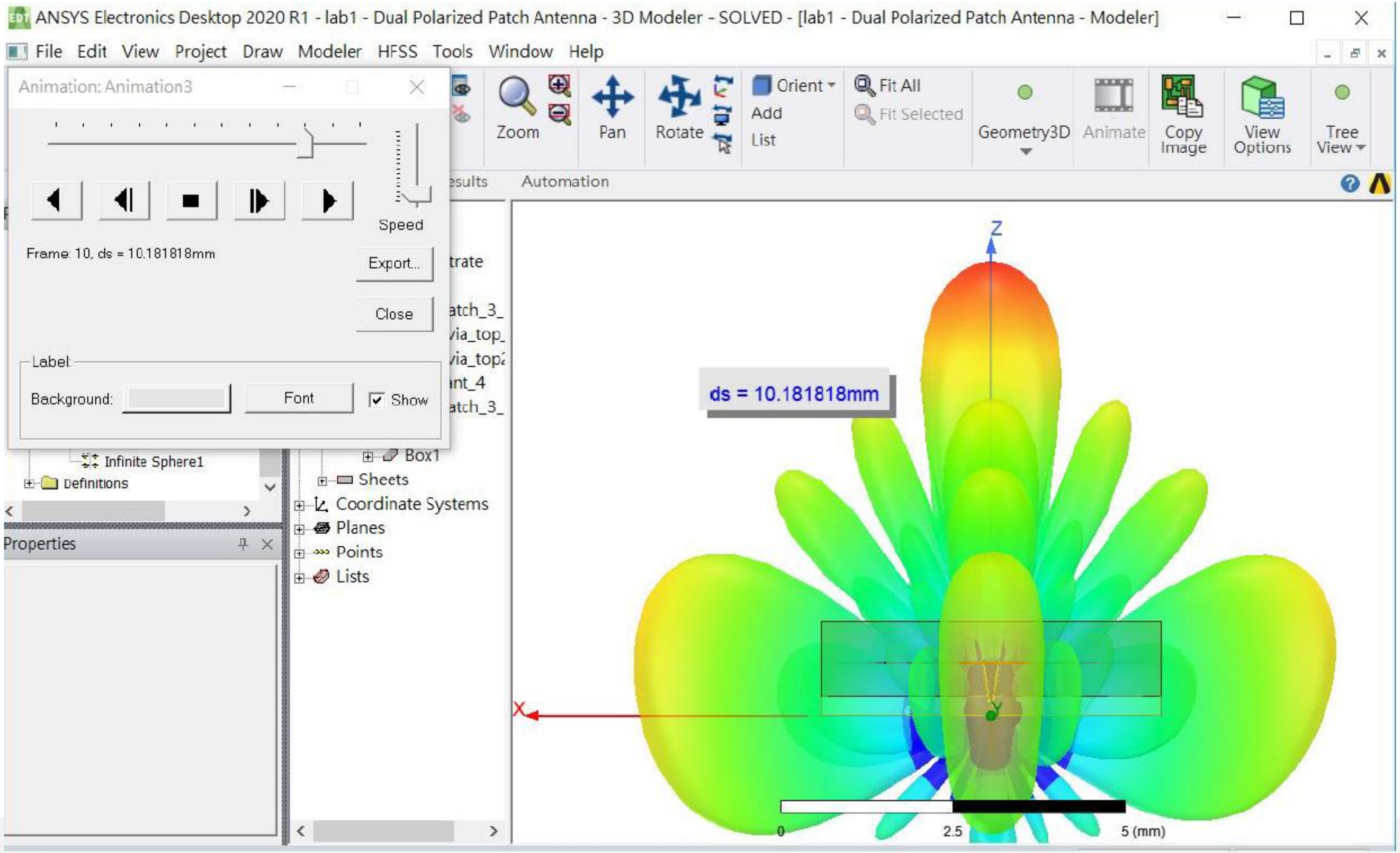
Set Range


ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [lab1 - Dual Polarized Patch Antenna - Modeler]
Setup Sweep Analysis
Sweep Definitions
Table
General
Calculations
Options
Sync #
Variable
Description
Add...
Edit...
Delete
Add/Edit Sweep
Variable
ds
Single value
Linear step
Linear count
Decade count
Start:
2
mm
Stop:
12
mm
Step:
1
mm
Add >>
Variable
Description
ds
Linear Step from 2mm to 12mm, step=1mm
OK
Cancel
Operation
Edit Variables
HPC and Analysis Options...
確定
取消
Project Manager
Properties
Ready

Setup Animate

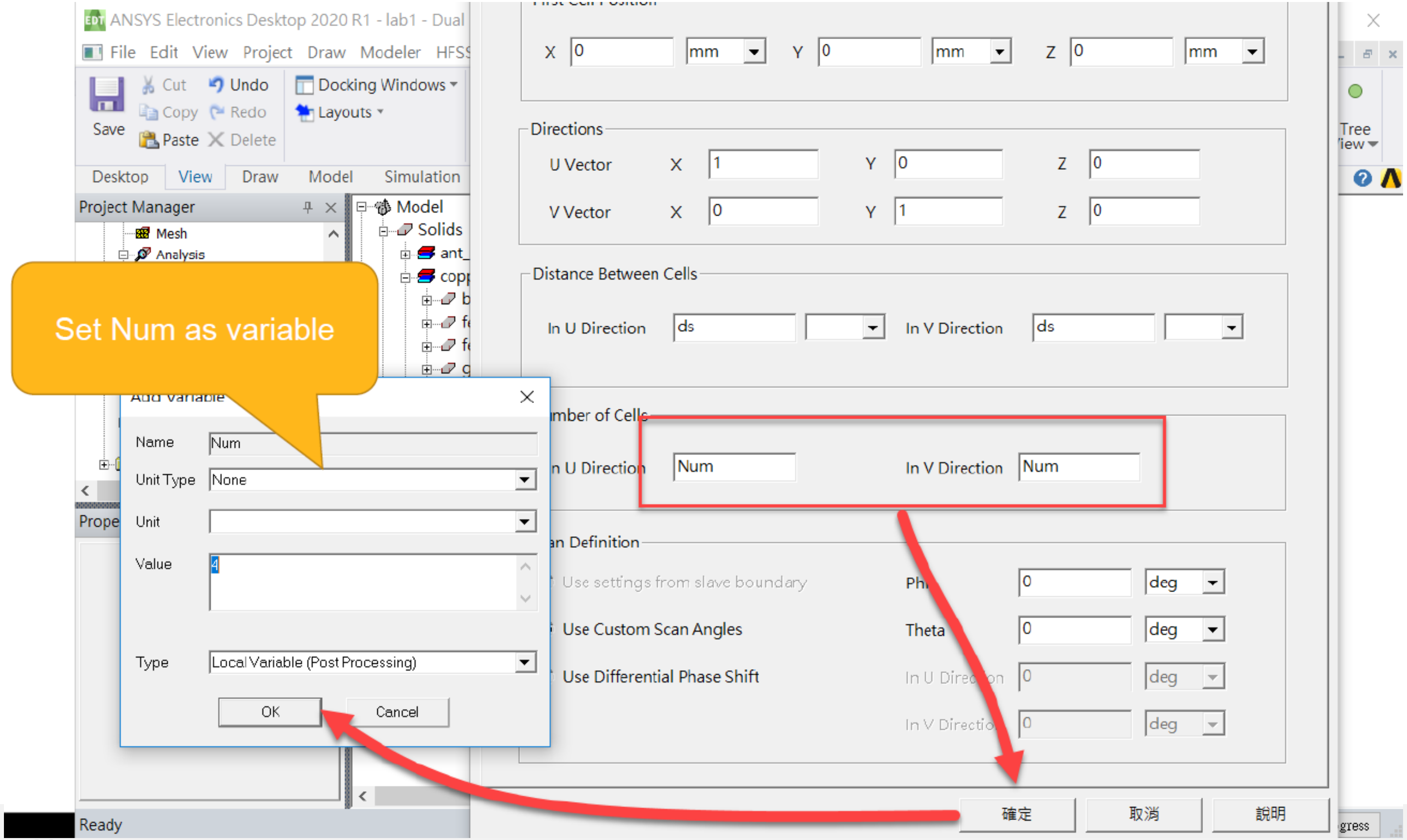


Result

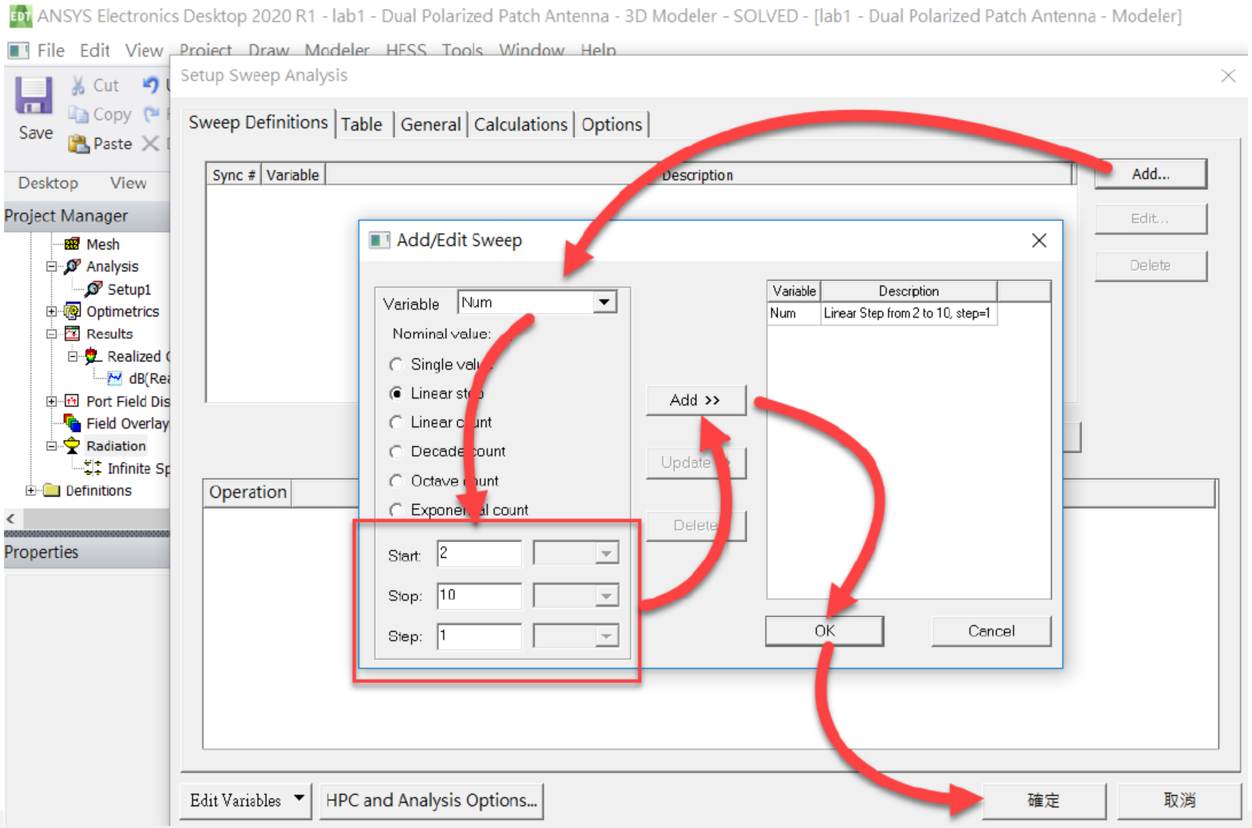


ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [lab1 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Animation: Animation3
Speed
Frame: 10, ds = 10.181818mm
Export...
Close
Label
Background:
Font
Show
Zoom
Pan
Rotate
Orient
Add
List
Fit All
Fit Selected
Geometry3D
Animate
Copy Image
View Options
Tree View
Automation
ds = 10.181818mm
Infinite Sphere1
Definitions
Box1
Sheets
Coordinate Systems
Planes
Points
Lists
Properties
Z
X
0
2.5
5 (mm)

Parametric with Array Number

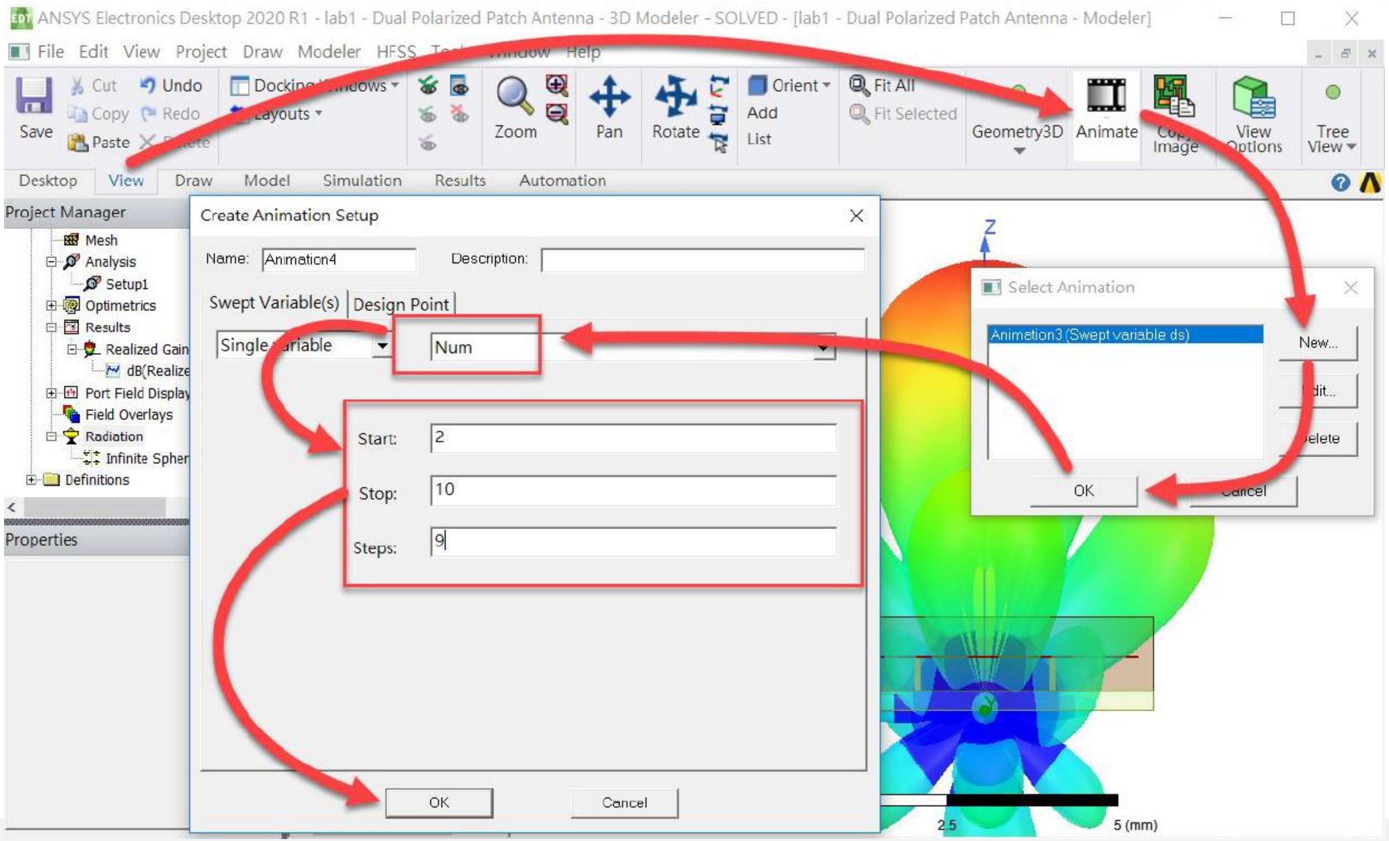


Set Range

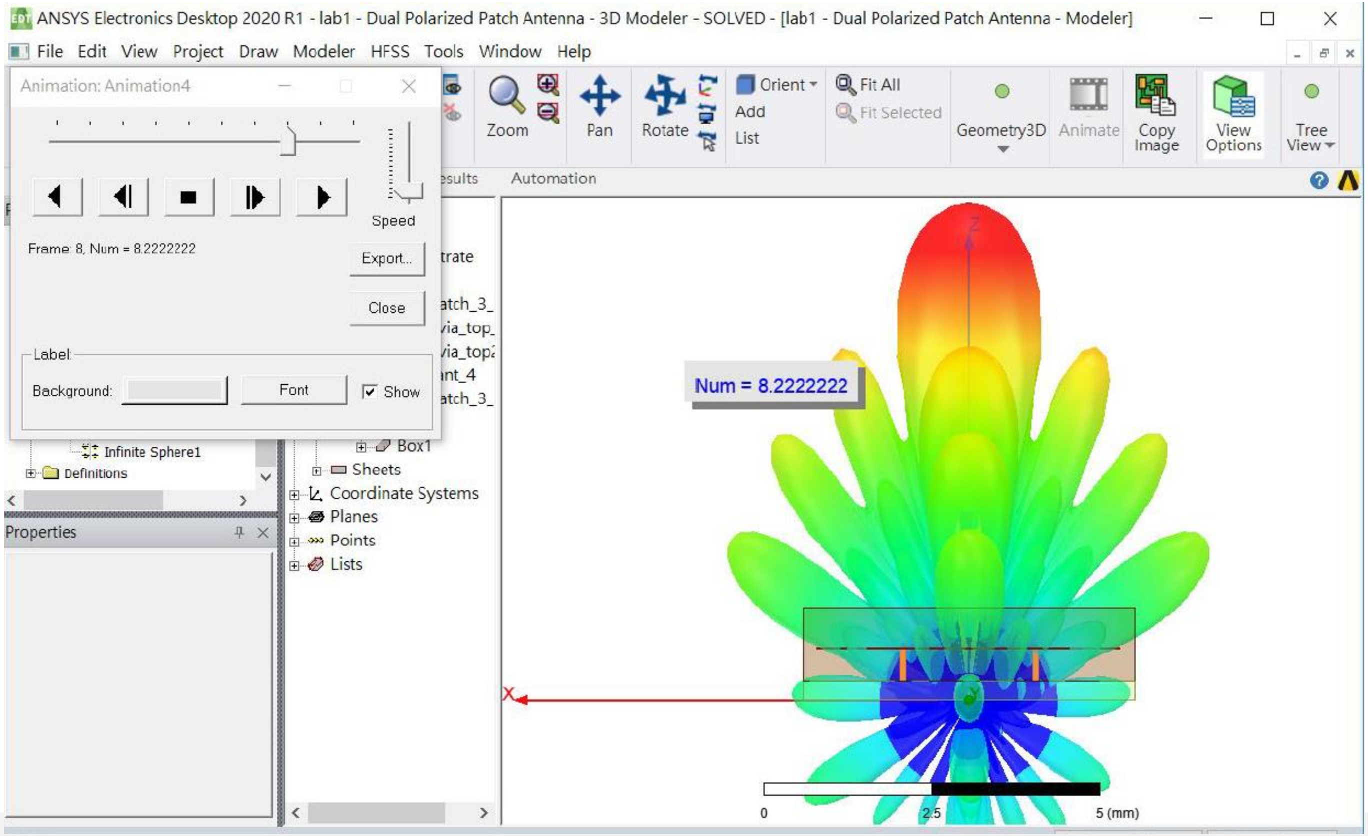


ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [lab1 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut
Copy
Paste
Desktop View
Project Manager
Mesh
Analysis
Setup1
Optimetrics
Results
Port Field Dis
Field Overlay
Radiation
Definitions
Properties
Setup Sweep Analysis
Sweep Definitions
Table
General
Calculations
Options
Sync #
Variable
Description
Add...
Edit...
Delete
Operation
Add/Edit Sweep
Variable
Num
Nominal value:
Linear count
Start:
2
Stop:
10
Step:
1
Add >>
Update
Delete
Variable
Description
Num
Linear Step from 2 to 10, step=1
OK
Cancel
Edit Variables
HPC and Analysis Options...
確定
取消

Setup Animate



Result


ANSYS Electronics Desktop 2020 R1 - lab1 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [lab1 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Animation: Animation4
Speed
Frame: 8, Num = 8.2222222
Export...
Close
Label
Background:
Font
Show
Zoom
Pan
Rotate
Orient
Add
List
Fit All
Fit Selected
Geometry3D
Animate
Copy Image
View Options
Tree View
Automation
Num = 8.2222222
Infinite Sphere1
Definitions
Properties
Box1
Sheets
Coordinate Systems
Planes
Points
Lists
0
2.5
5 (mm)

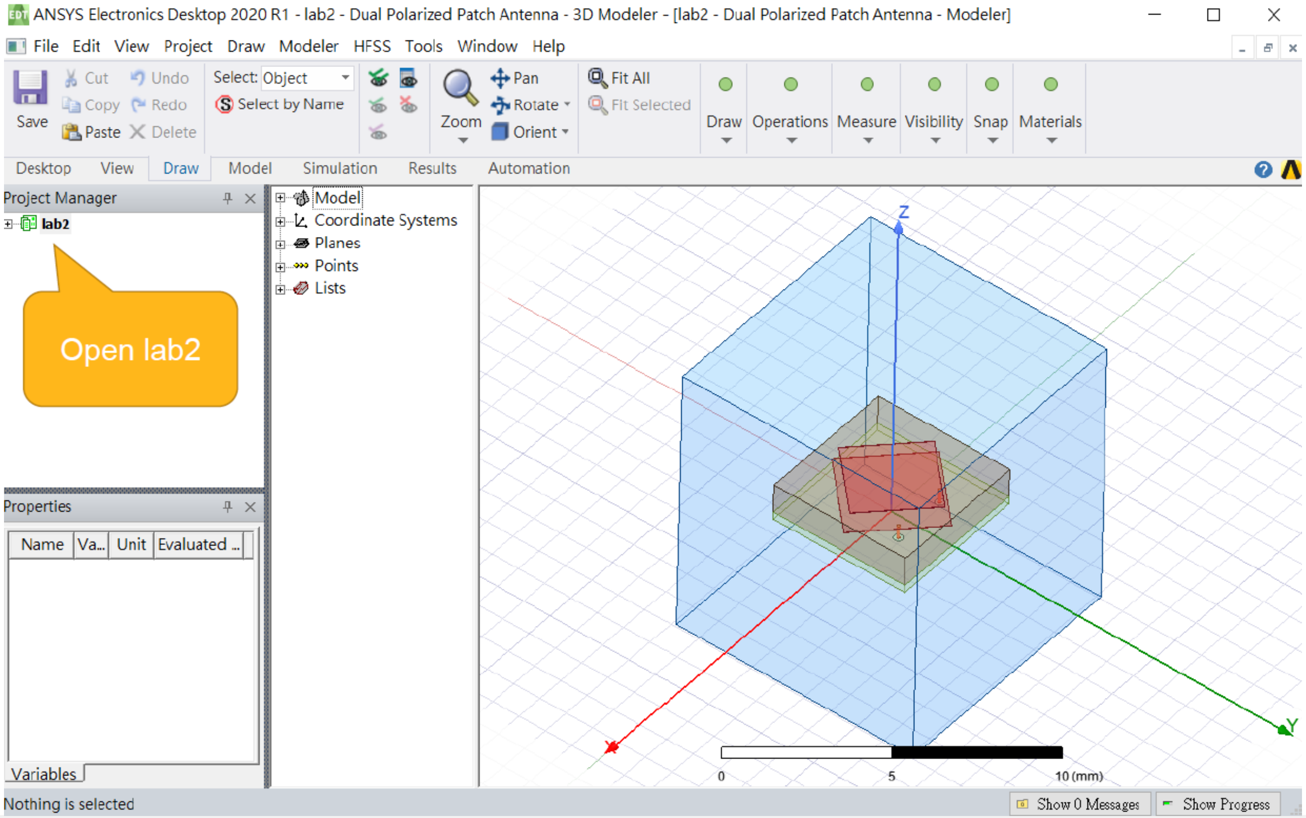
Ansys / CERTIFIED ELITE CHANNEL PARTNER /

Lab 2. Build Finite Regular Array

Ansys 2021 R1
Engineering What's Ahead.

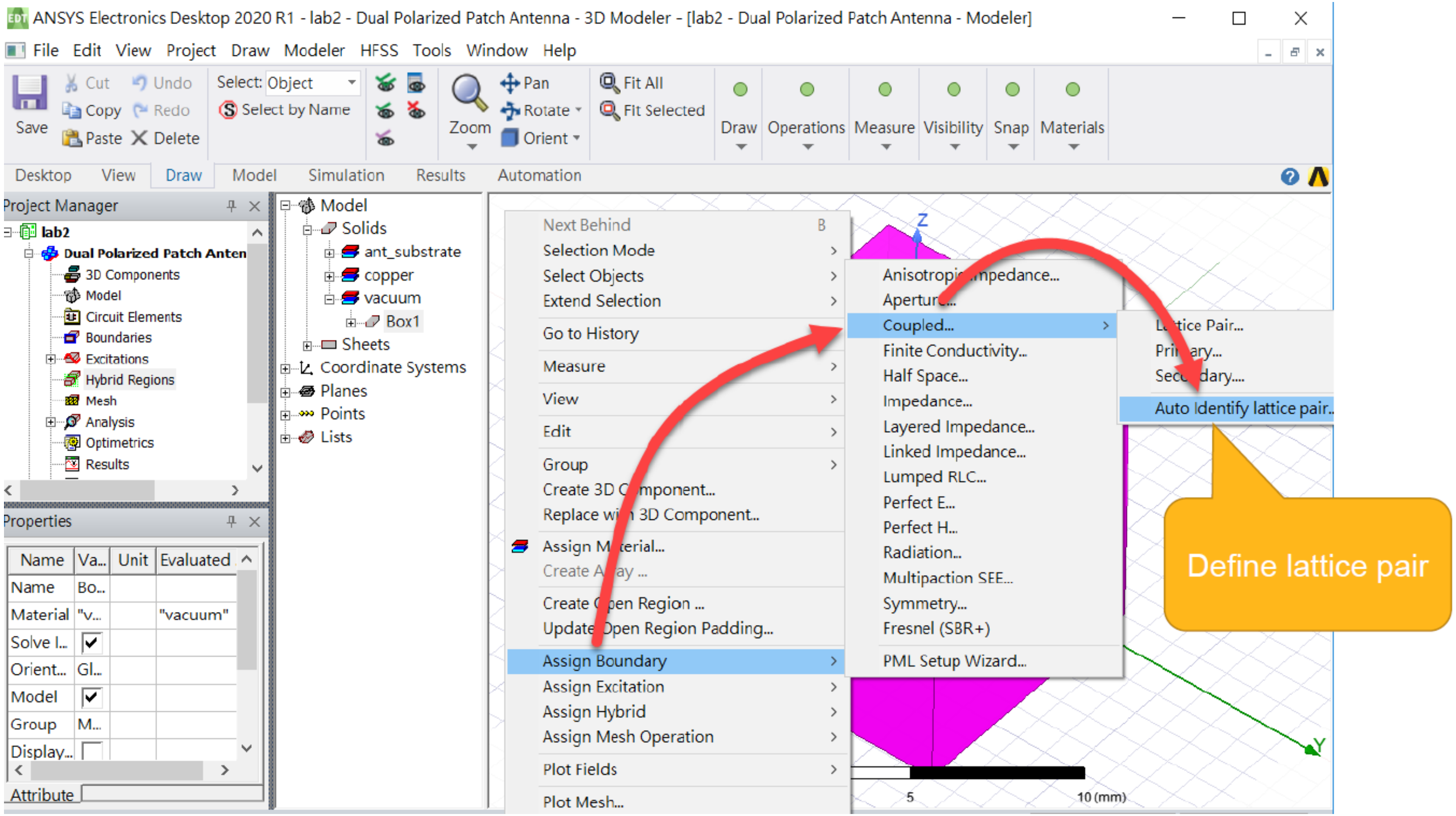
CYBERNET 思渤科技
CYBERNET SYSTEMS TAIWAN

Open Lab2

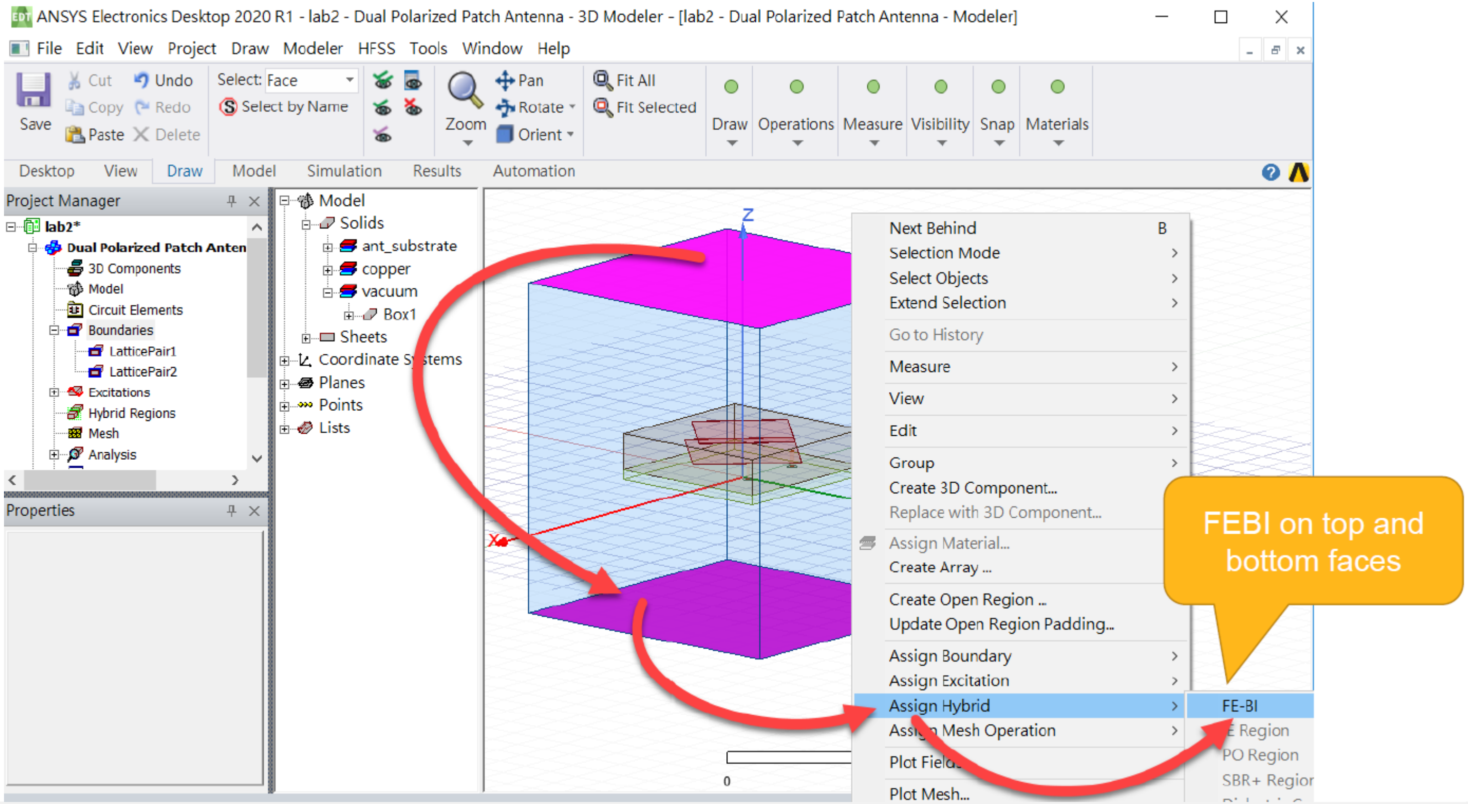


ANSYS Electronics Desktop 2020 R1 - lab2 - Dual Polarized Patch Antenna - 3D Modeler - [lab2 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut
Copy
Paste
Undo
Redo
Delete
Select: Object
Select by Name
Zoom
Pan
Rotate
Orient
Fit All
Fit Selected
Draw
Operations
Measure
Visibility
Snap
Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
lab2
Open lab2
Model
Coordinate Systems
Planes
Points
Lists
Properties
Name Va... Unit Evaluated ...
Variables
Nothing is selected
Show 0 Messages
Show Progress
Z
Y
0
5
10 (mm)

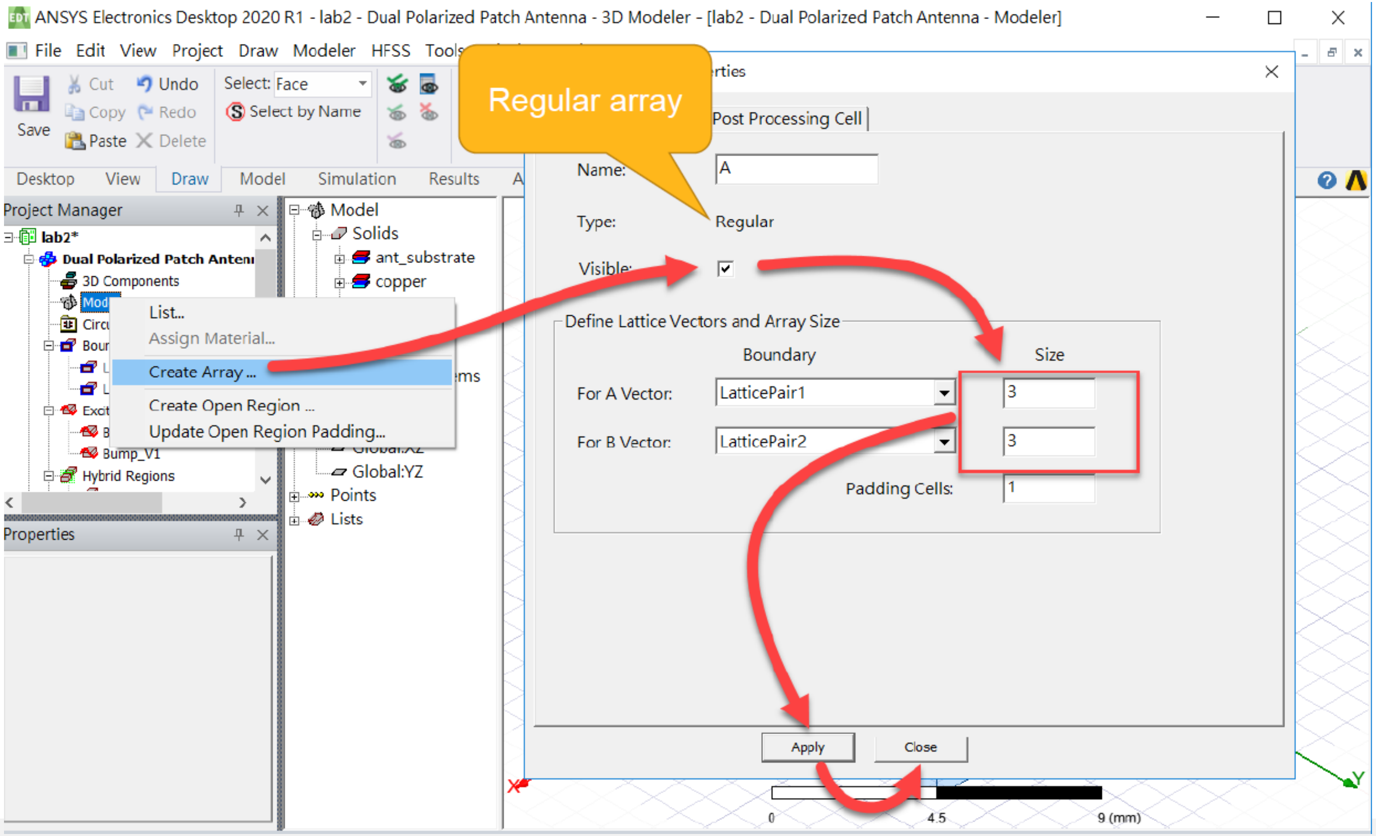
Define Lattice Pair



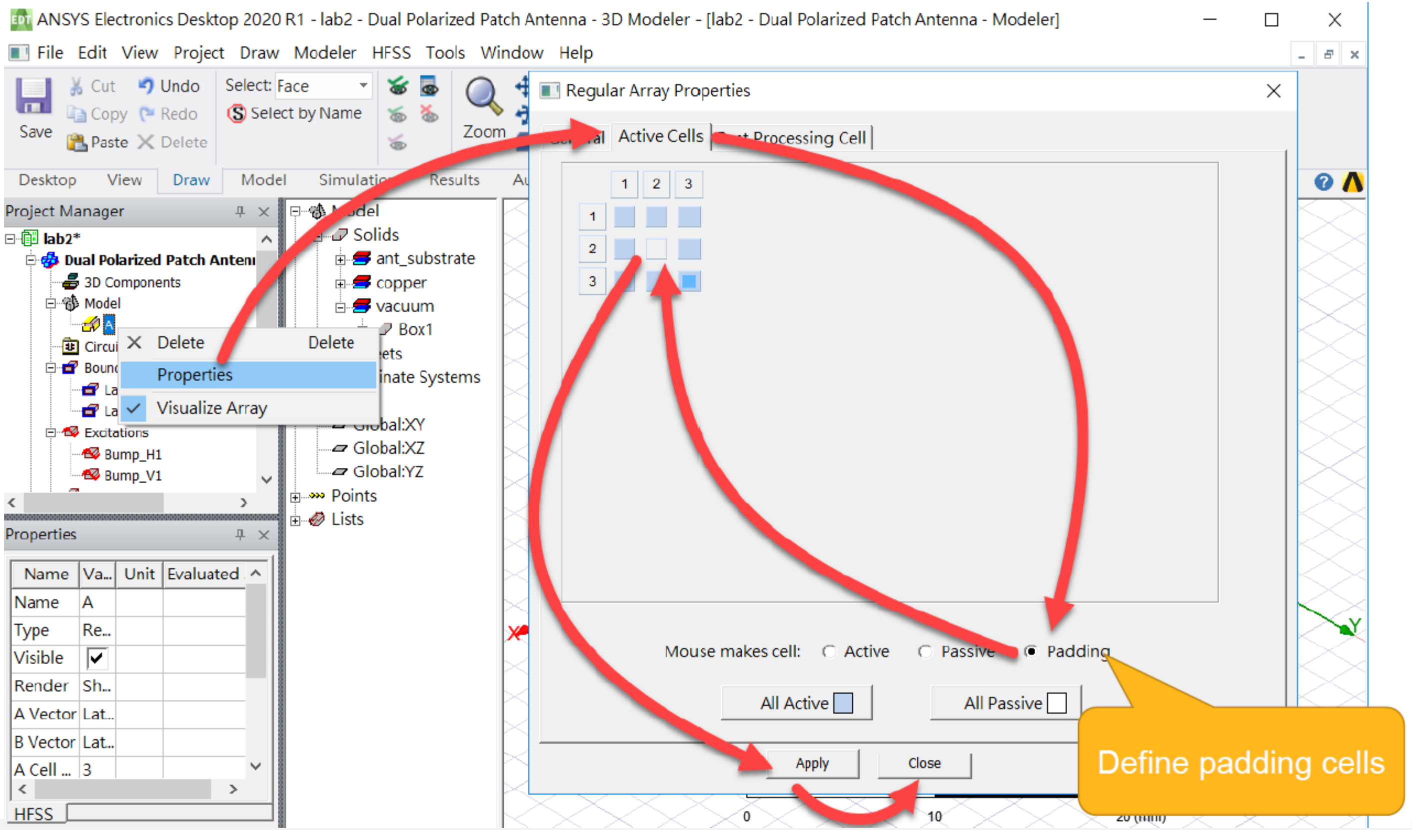
Set FEBI Boundary



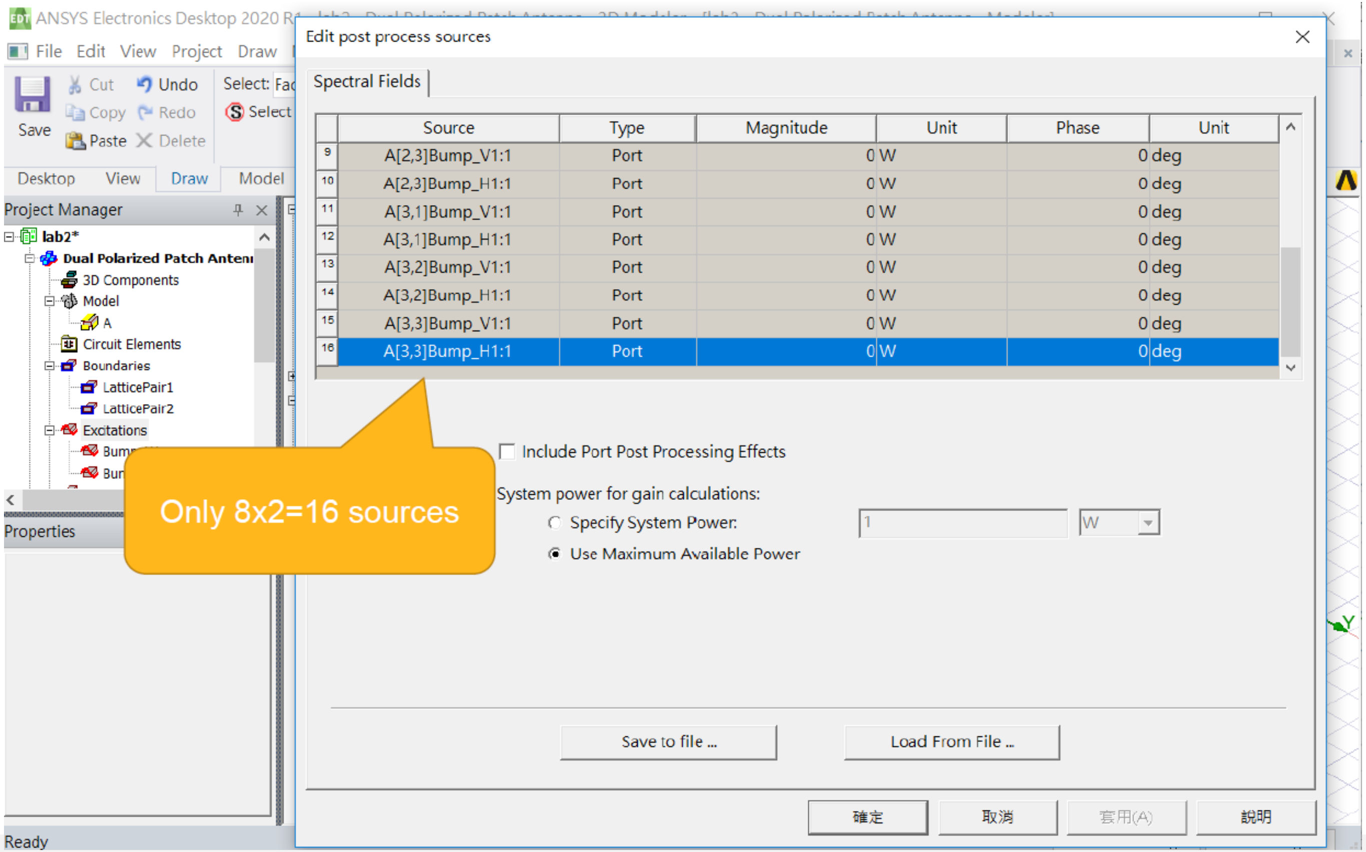
Create Array



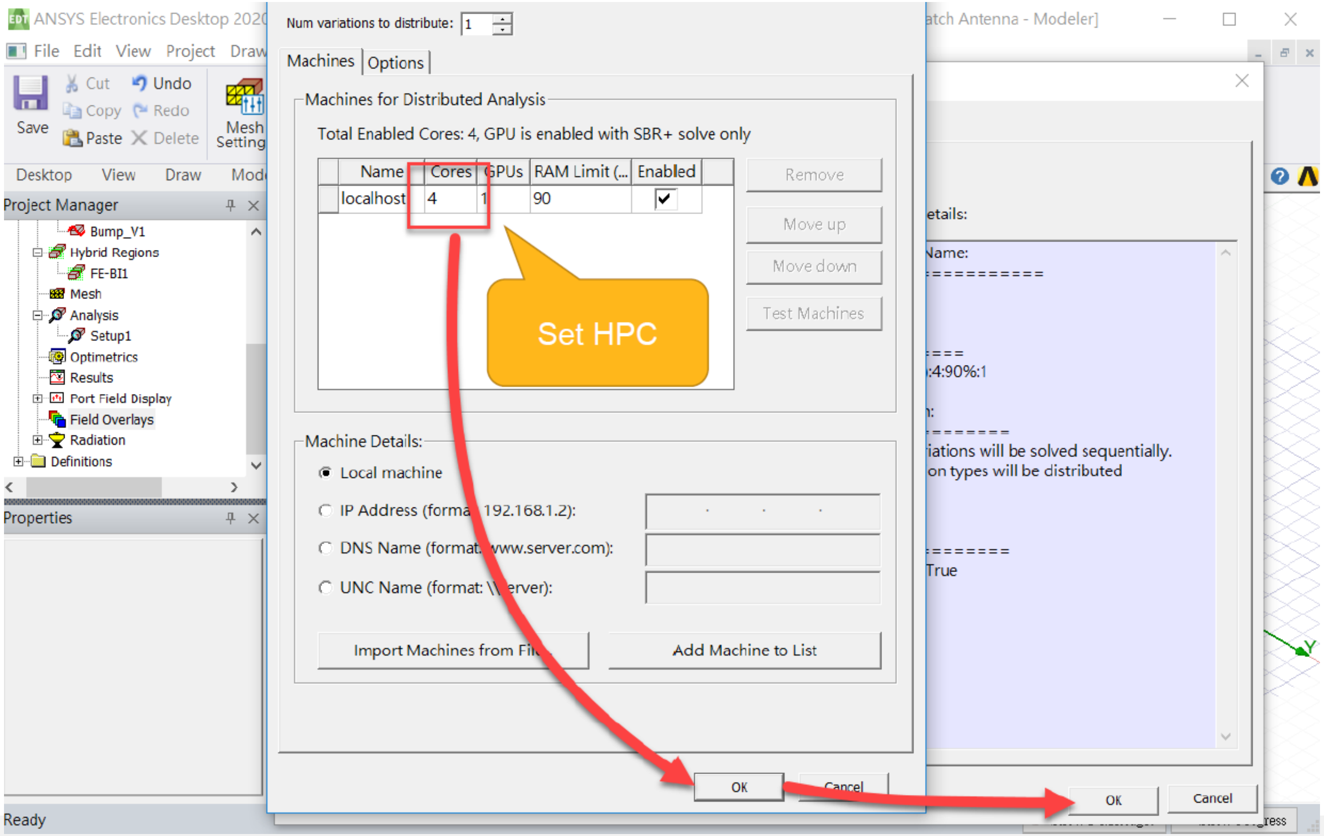
Define Padding Cells



Edit Sources

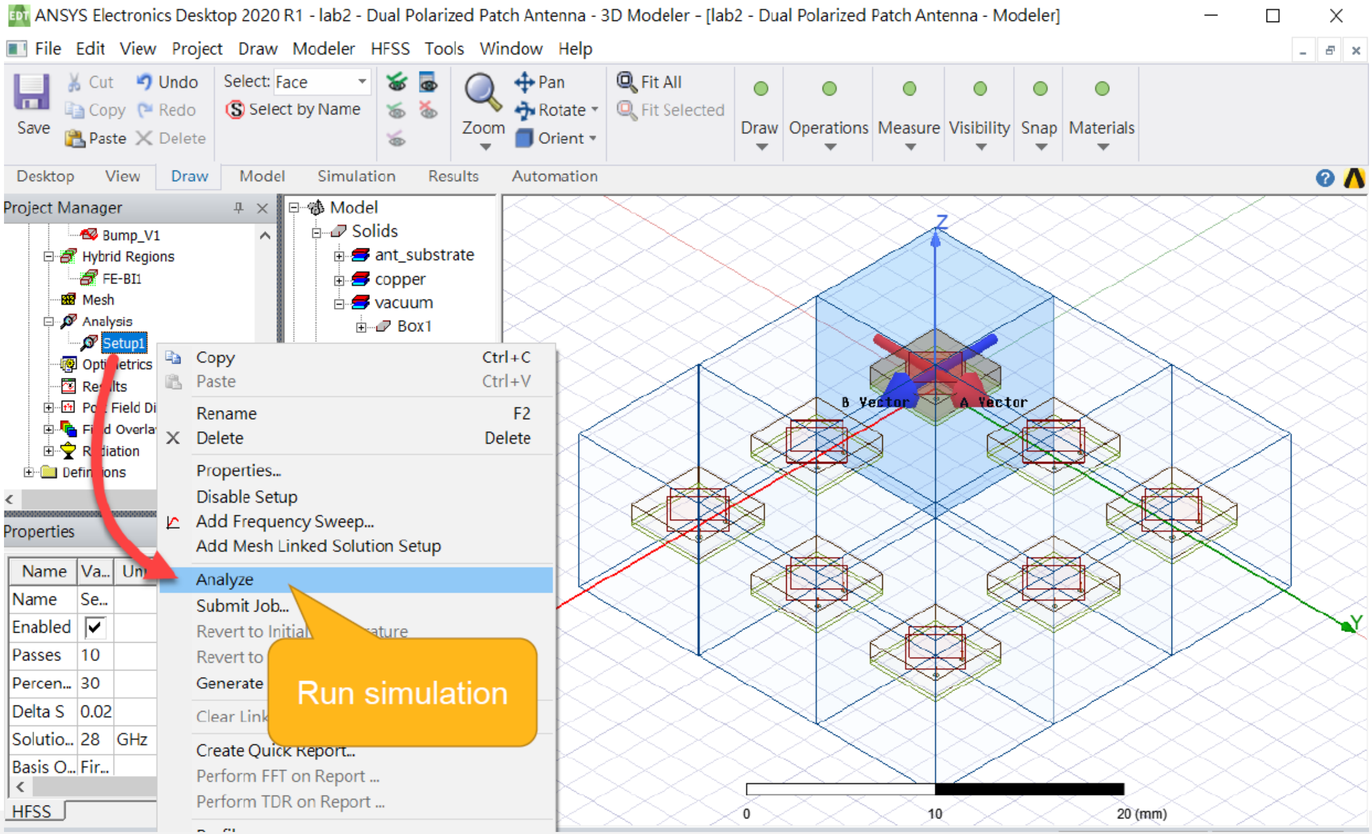


Set HPC



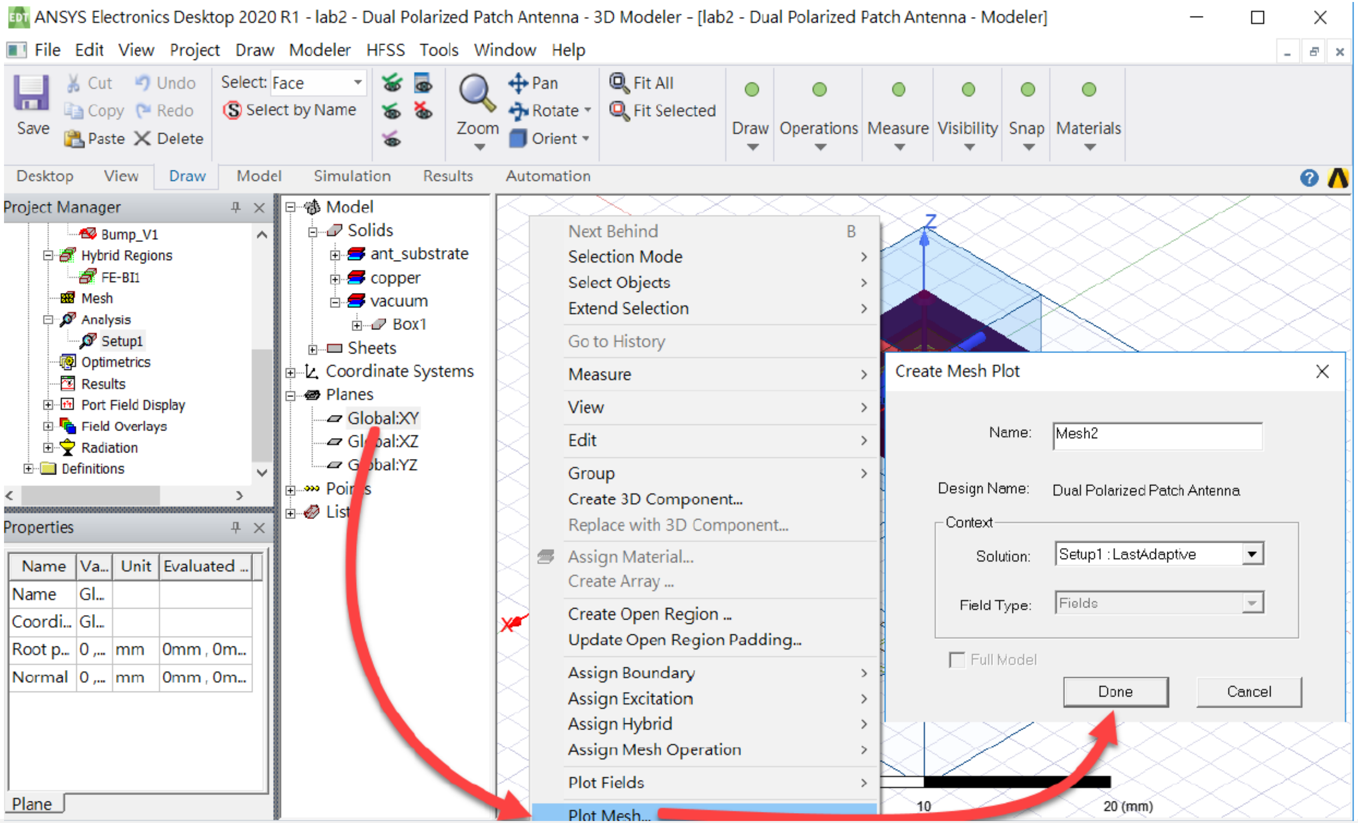
ANSYS Electronics Desktop 2020
atch Antenna - Modeler]
File Edit View Project Draw
Cut Undo
Copy Redo
Paste Delete
Save
Mesh Setting
Desktop View Draw Mod
Project Manager
Bump_V1
Hybrid Regions
FE-BI1
Mesh
Analysis
Setup1
Optimetrics
Results
Port Field Display
Field Overlays
Radiation
Definitions
Properties
Ready
Num variations to distribute: 1
Machines
Options
Machines for Distributed Analysis
Total Enabled Cores: 4, GPU is enabled with SBR+ solve only
Name Cores GPUs RAM Limit (... Enabled
localhost 4 1 90
Remove
Move up
Move down
Test Machines
Set HPC
Machine Details:
Local machine
IP Address (forma 192.168.1.2):
DNS Name (format www.server.com):
UNC Name (format: \\ erver):
Import Machines from Fil
Add Machine to List
OK
Cancel
etails:
Name:
:4:90%:1
iations will be solved sequentially.
on types will be distributed
True
OK
Cancel

Analyze



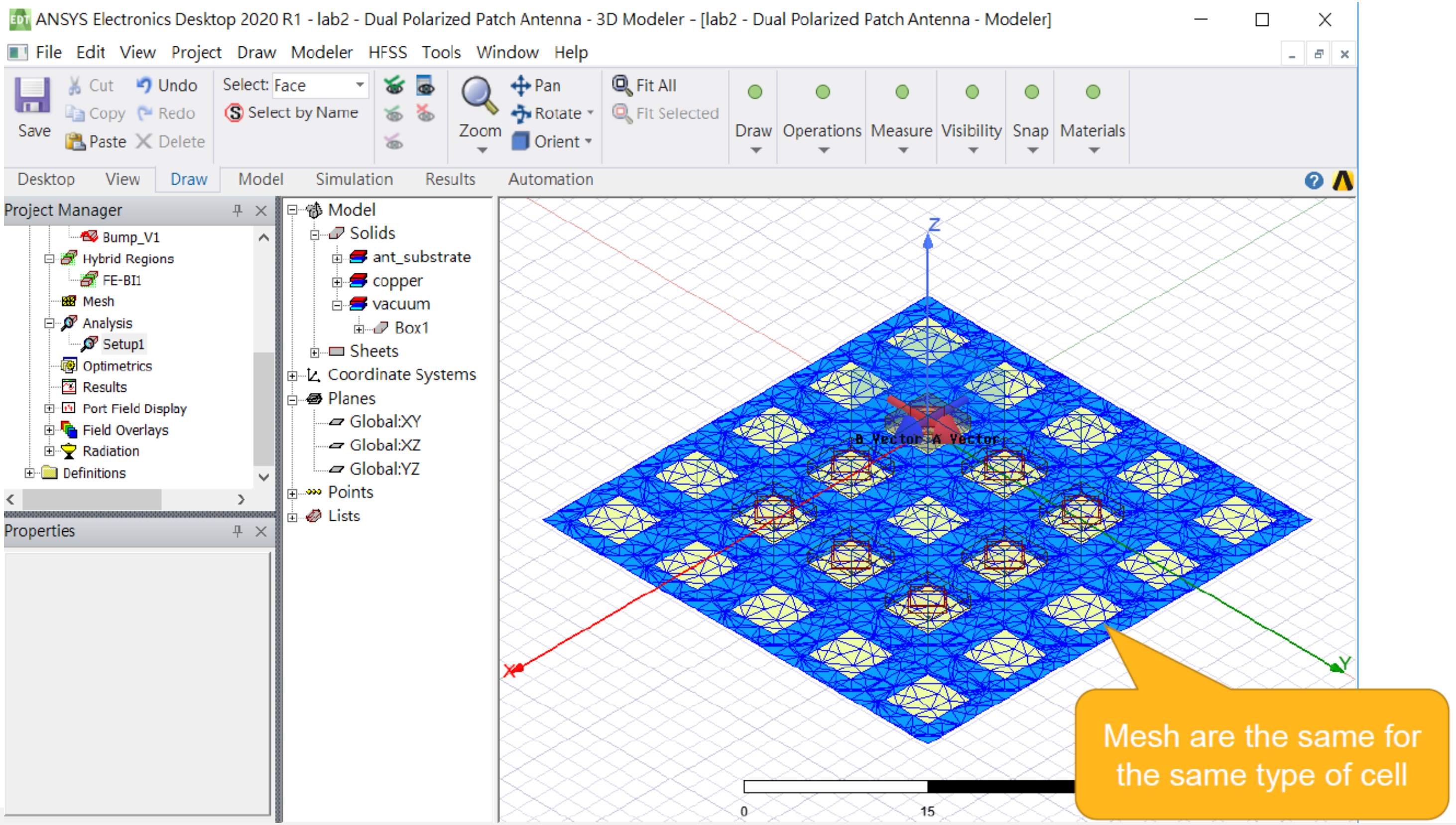
ANSYS Electronics Desktop 2020 R1 - lab2 - Dual Polarized Patch Antenna - 3D Modeler - [lab2 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Project Manager
Bump_V1
Hybrid Regions
FE-BI1
Mesh
Analysis
Setup1
Model
Solids
ant_substrate
copper
vacuum
Box1
Copy Ctrl+C
Paste Ctrl+V
Rename F2
Delete Delete
Properties...
Disable Setup
Add Frequency Sweep...
Add Mesh Linked Solution Setup
Analyze
Submit Job...
Create Quick Report...
Perform FFT on Report ...
Perform TDR on Report ...
Run simulation
Properties
Name
Enabled
Passes 10
Delta S 0.02
Solutio... 28 GHz
B Vector
A Vector
0
10
20 (mm)

Plot Mesh

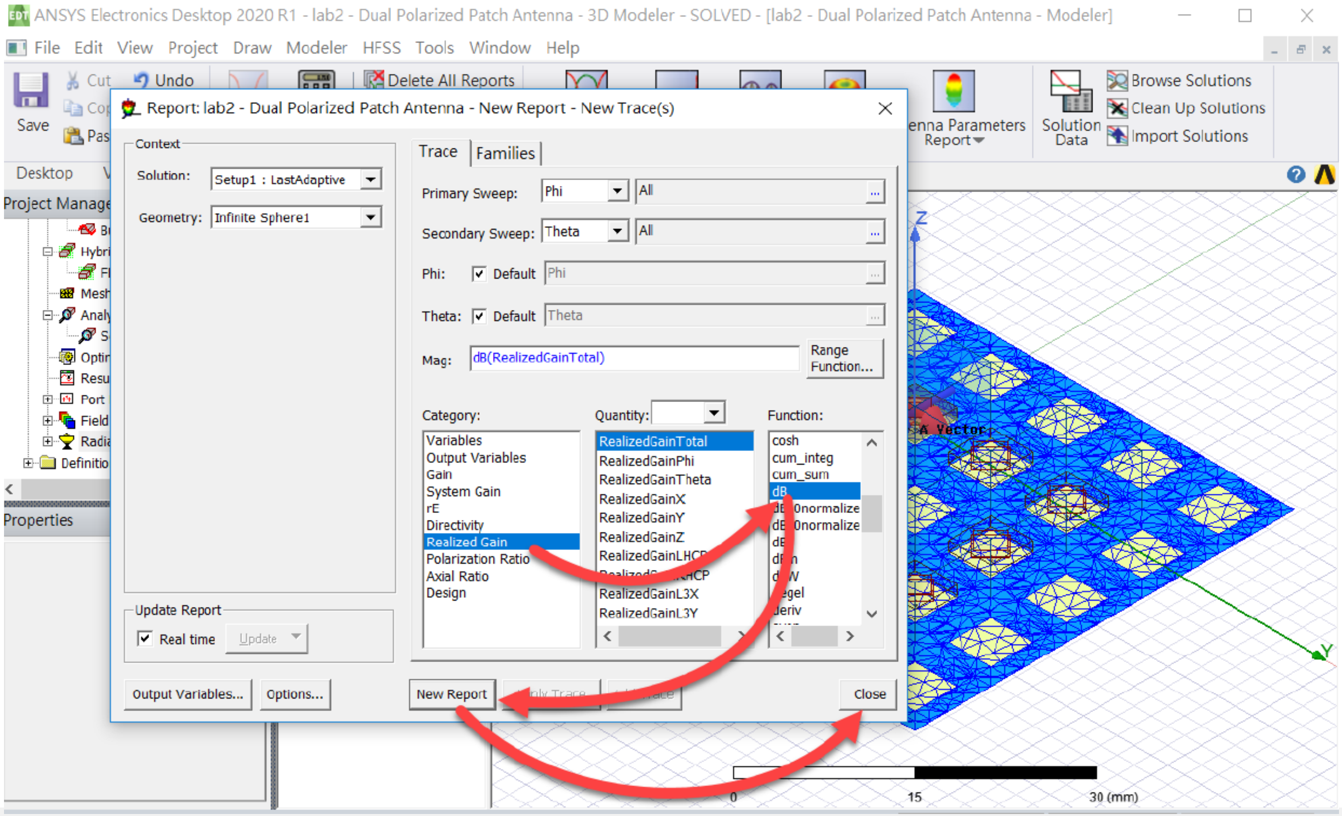


ANSYS Electronics Desktop 2020 R1 - lab2 - Dual Polarized Patch Antenna - 3D Modeler - [lab2 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Project Manager
Model
Next Behind
Selection Mode
Select Objects
Extend Selection
Go to History
Measure
View
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...
Create Mesh Plot
Name: Mesh2
Design Name: Dual Polarized Patch Antenna
Context
Solution: Setup1 : LastAdaptive
Field Type: Fields
Full Model
Done
Cancel

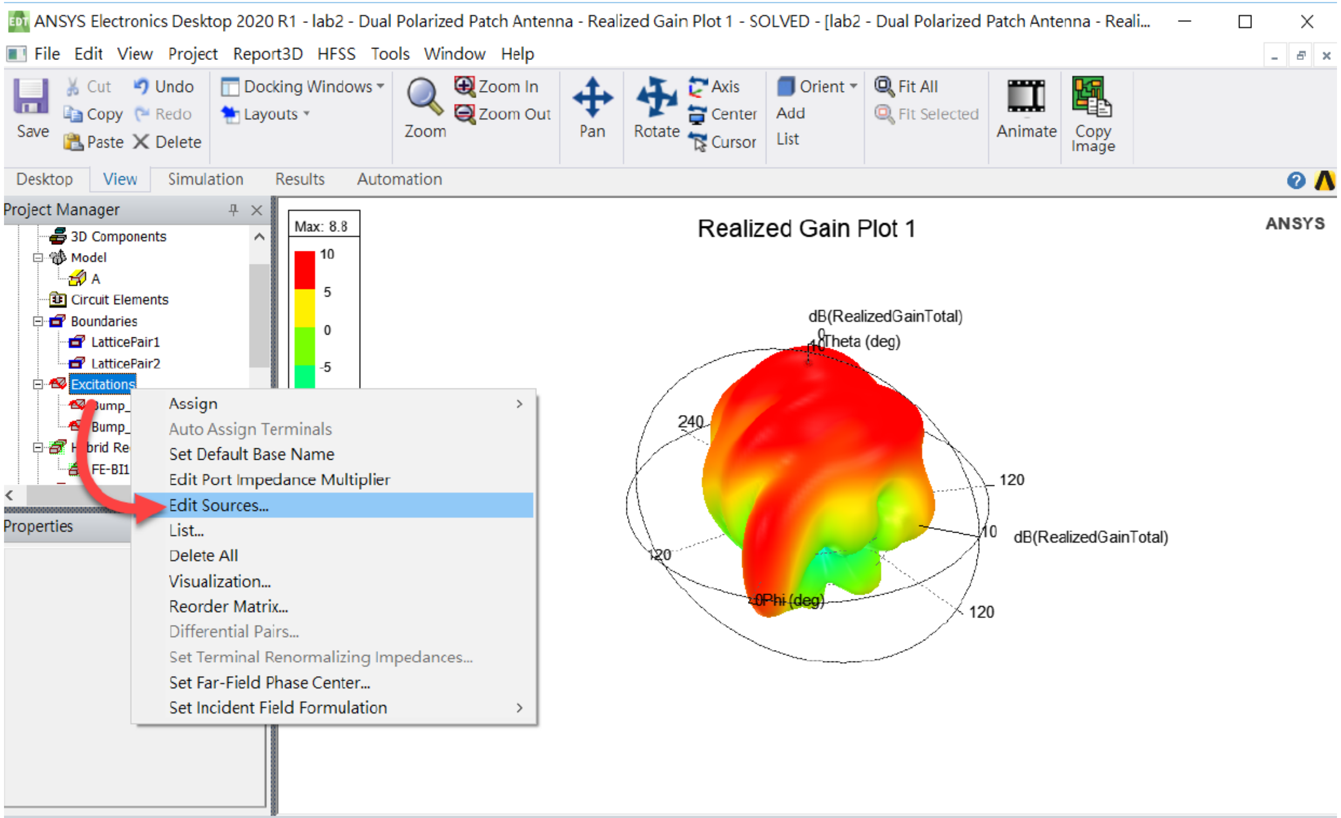
Result


ANSYS Electronics Desktop 2020 R1 - lab2 - Dual Polarized Patch Antenna - 3D Modeler - [lab2 - Dual Polarized Patch Antenna - 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
Draw Operations Measure Visibility Snap Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
Bump_V1
Hybrid Regions
FE-BI1
Mesh
Analysis
Setup1
Optimetrics
Results
Port Field Display
Field Overlays
Radiation
Definitions
Properties
Model
Solids
ant_substrate
copper
vacuum
Box1
Sheets
Coordinate Systems
Planes
Global:XY
Global:XZ
Global:YZ
Points
Lists
Z
X
Y
0
15
Mesh are the same for the same type of cell

Plot Pattern

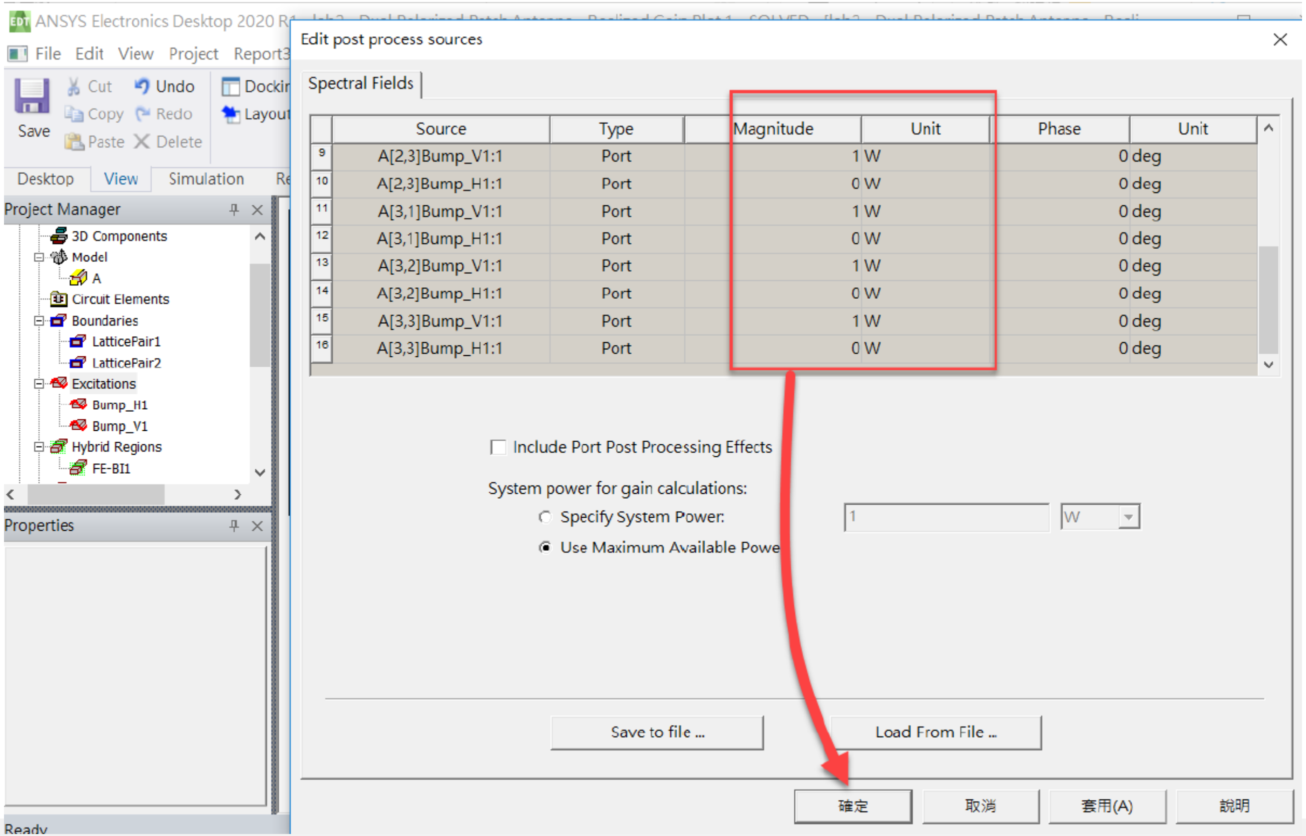


Edit Source

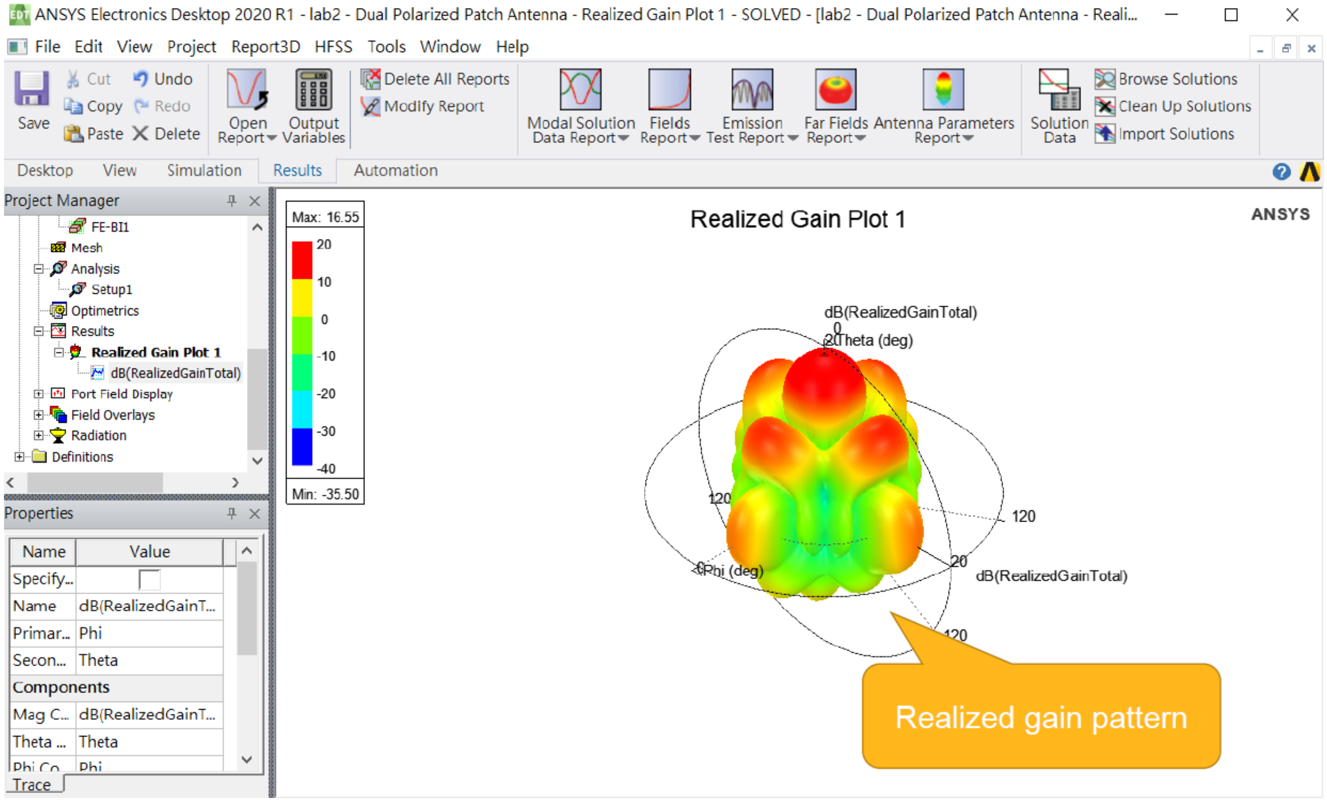


ANSYS Electronics Desktop 2020 R1 - lab2 - Dual Polarized Patch Antenna - Realized Gain Plot 1 - SOLVED - [lab2 - Dual Polarized Patch Antenna - Reali...
File Edit View Project Report3D HFSS Tools Window Help
Save
Cut
Copy
Paste
Undo
Redo
Delete
Docking Windows
Layouts
Zoom
Zoom In
Zoom Out
Pan
Rotate
Axis
Center
Cursor
Orient
Add
List
Fit All
Fit Selected
Animate
Copy Image
Desktop View Simulation Results Automation
Project Manager
3D Components
Model
A
Circuit Elements
Boundaries
LatticePair1
LatticePair2
Excitations
Properties
Assign
Auto Assign Terminals
Set Default Base Name
Edit Port Impedance Multiplier
Edit Sources...
List...
Delete All
Visualization...
Reorder Matrix...
Differential Pairs...
Set Terminal Renormalizing Impedances...
Set Far-Field Phase Center...
Set Incident Field Formulation
Max: 8.8
10
5
0
-5
Realized Gain Plot 1
ANSYS
dB(RealizedGainTotal)
Theta (deg)
240
120
120
120
10
dB(RealizedGainTotal)
Phi (deg)

Edit Source


ANSYS Electronics Desktop 2020 R1
File Edit View Project Report3
Save Cut Copy Paste Undo Redo Delete
Desktop View Simulation
Project Manager
3D Components
Model
A
Circuit Elements
Boundaries
LatticePair1
LatticePair2
Excitations
Bump_H1
Bump_V1
Hybrid Regions
FE-BI1
Properties
Ready
Edit post process sources
Spectral Fields
Source Type Magnitude Unit Phase Unit
9 A[2,3]Bump_V1:1 Port 1 W 0 deg
10 A[2,3]Bump_H1:1 Port 0 W 0 deg
11 A[3,1]Bump_V1:1 Port 1 W 0 deg
12 A[3,1]Bump_H1:1 Port 0 W 0 deg
13 A[3,2]Bump_V1:1 Port 1 W 0 deg
14 A[3,2]Bump_H1:1 Port 0 W 0 deg
15 A[3,3]Bump_V1:1 Port 1 W 0 deg
16 A[3,3]Bump_H1:1 Port 0 W 0 deg
Include Port Post Processing Effects
System power for gain calculations:
Specify System Power:
Use Maximum Available Powe
1
W
Save to file ...
Load From File ...
確定
取消
套用(A)
說明

Result



ANSYS Electronics Desktop 2020 R1 - lab2 - Dual Polarized Patch Antenna - Realized Gain Plot 1 - SOLVED - [lab2 - Dual Polarized Patch Antenna - Reali...
File Edit View Project Report3D HFSS Tools Window Help
Save Cut Copy Paste Undo Redo Delete
Open Report Output Variables
Delete All Reports
Modify Report
Modal Solution Data Report
Fields Report
Emission Test Report
Far Fields Report
Antenna Parameters Report
Solution Data
Browse Solutions
Clean Up Solutions
Import Solutions
Desktop View Simulation Results Automation
Project Manager
FE-BI1
Mesh
Analysis
Setup1
Optimetrics
Results
Realized Gain Plot 1
dB(RealizedGainTotal)
Port Field Display
Field Overlays
Radiation
Definitions
Properties
Name Value
Specify...
Name dB(RealizedGainT...
Primar... Phi
Secon... Theta
Components
Mag C... dB(RealizedGainT...
Theta ... Theta
Phi Co... Phi
Trace
Realized Gain Plot 1
ANSYS
Max: 16.55
20
10
0
-10
-20
-30
-40
Min: -35.50
dB(RealizedGainTotal)
Theta (deg)
Phi (deg)
120
Realized gain pattern

Lab 3. Build 3D Component Array

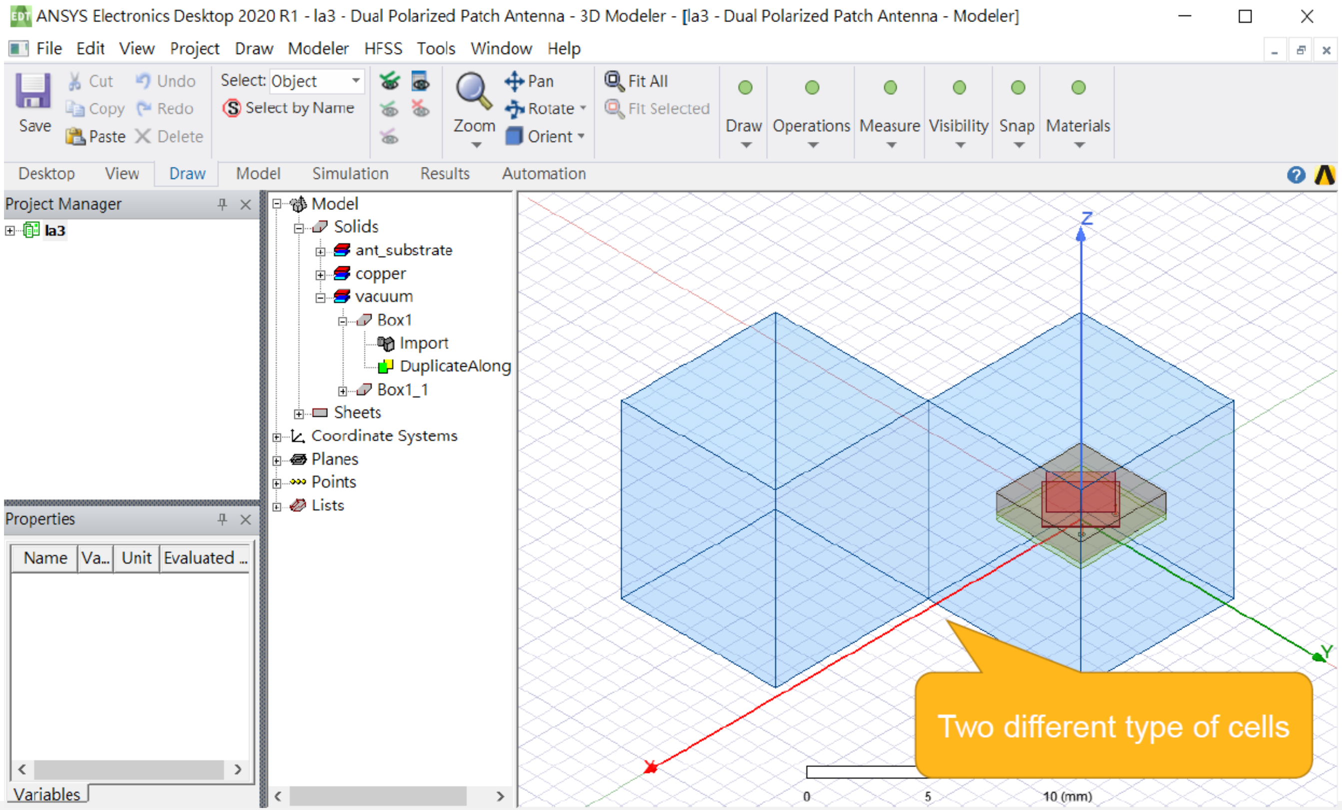
2021 R1

Engineering What's Ahead.

CYBERNET 思渤科技

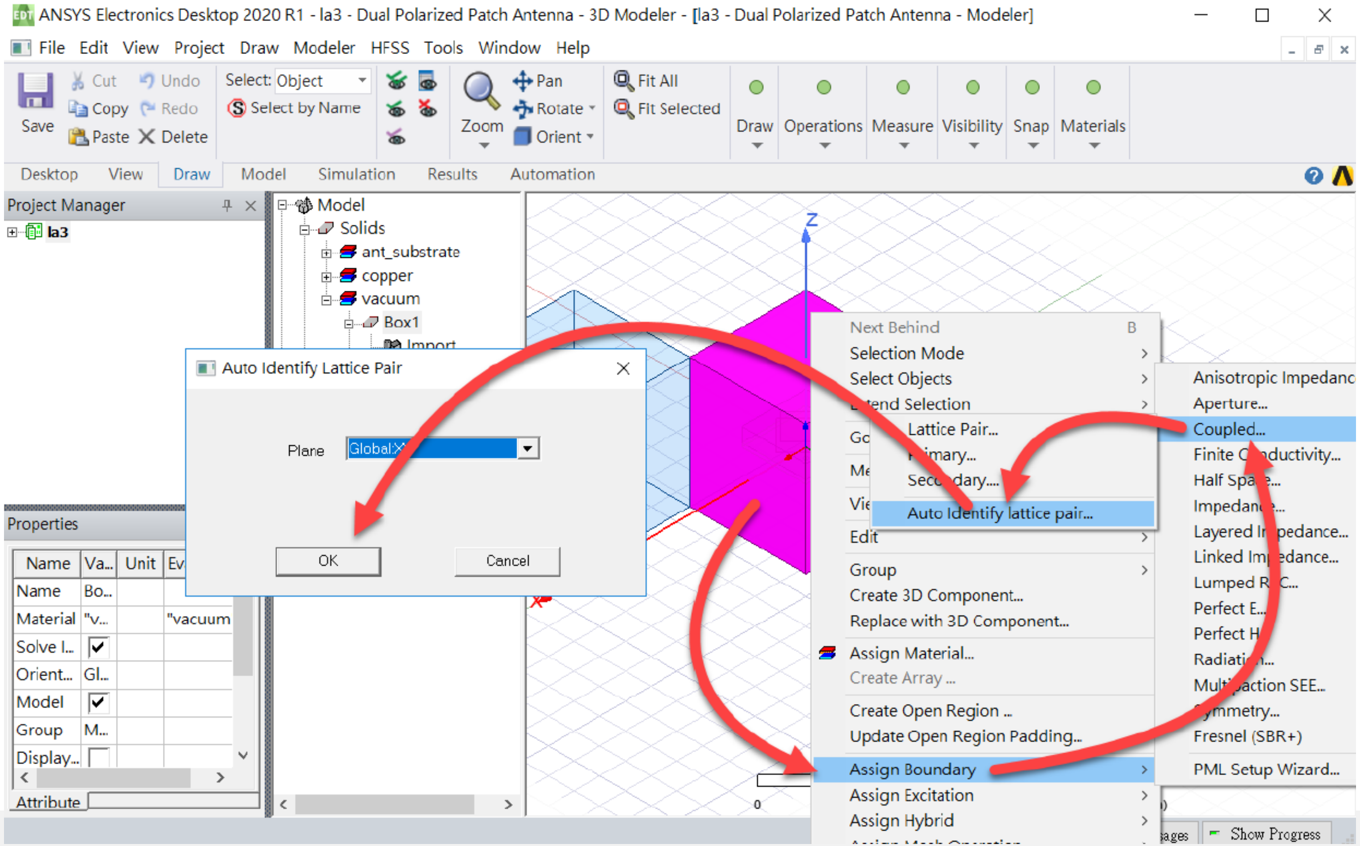
CYBERNET SYSTEMS TAIWAN

Add Air Box

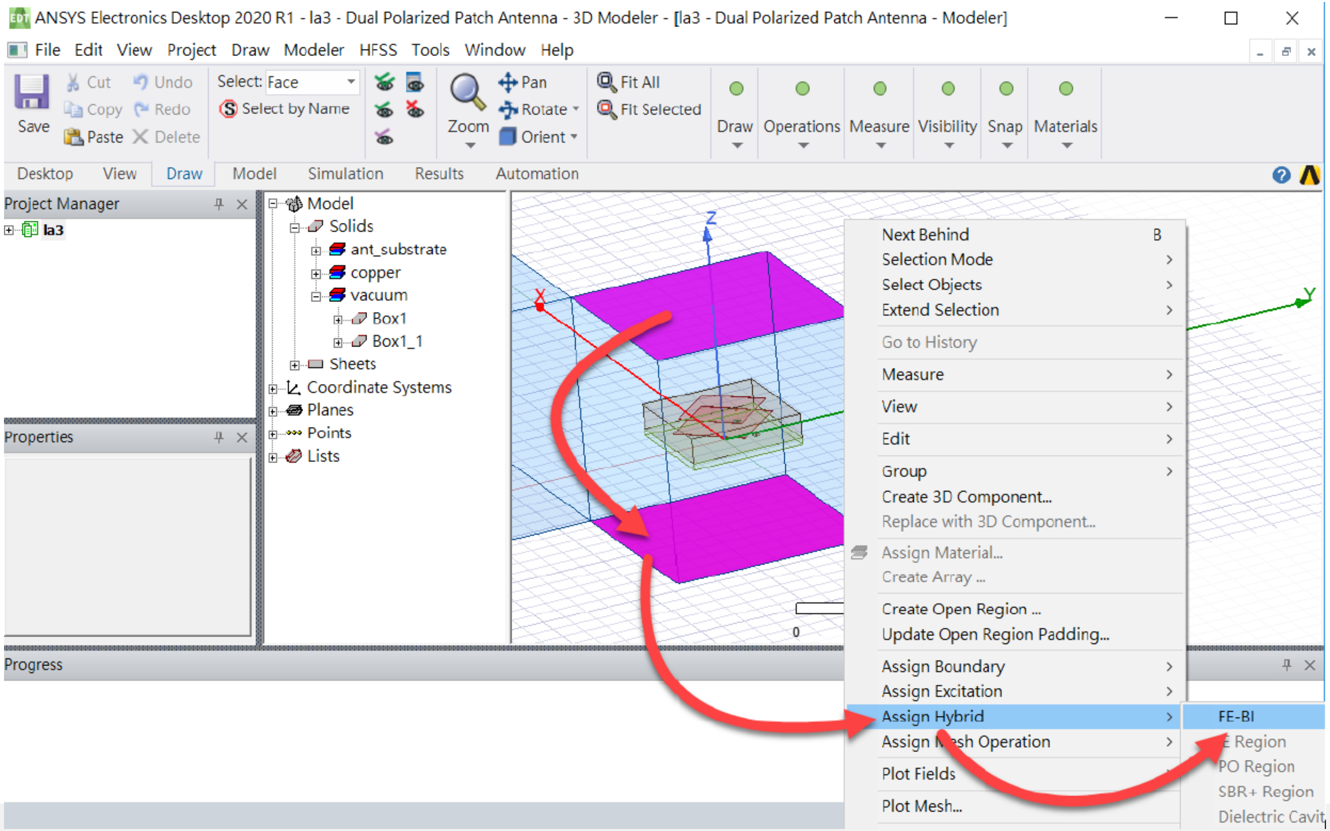


ANSYS Electronics Desktop 2020 R1 - la3 - Dual Polarized Patch Antenna - 3D Modeler - [la3 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut
Copy
Paste
Undo
Redo
Delete
Select: Object
Select by Name
Zoom
Pan
Rotate
Orient
Fit All
Fit Selected
Draw
Operations
Measure
Visibility
Snap
Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
la3
Model
Solids
ant_substrate
copper
vacuum
Box1
Import
DuplicateAlong
Box1_1
Sheets
Coordinate Systems
Planes
Points
Lists
Properties
Name
Va...
Unit
Evaluated ...
Variables
Z
Y
X
0
5
10 (mm)
Two different type of cells

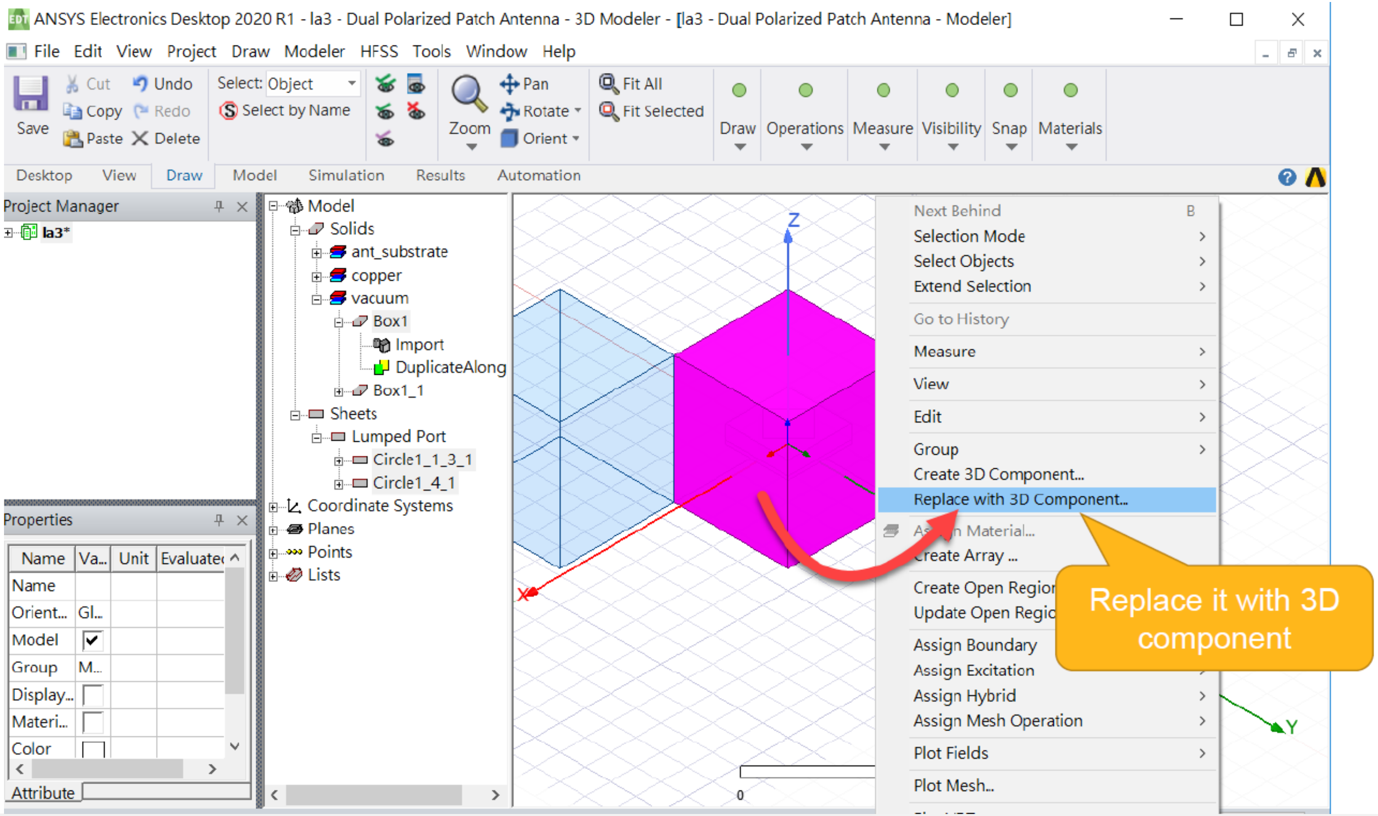
Add lattice Boundary in Antenna Box



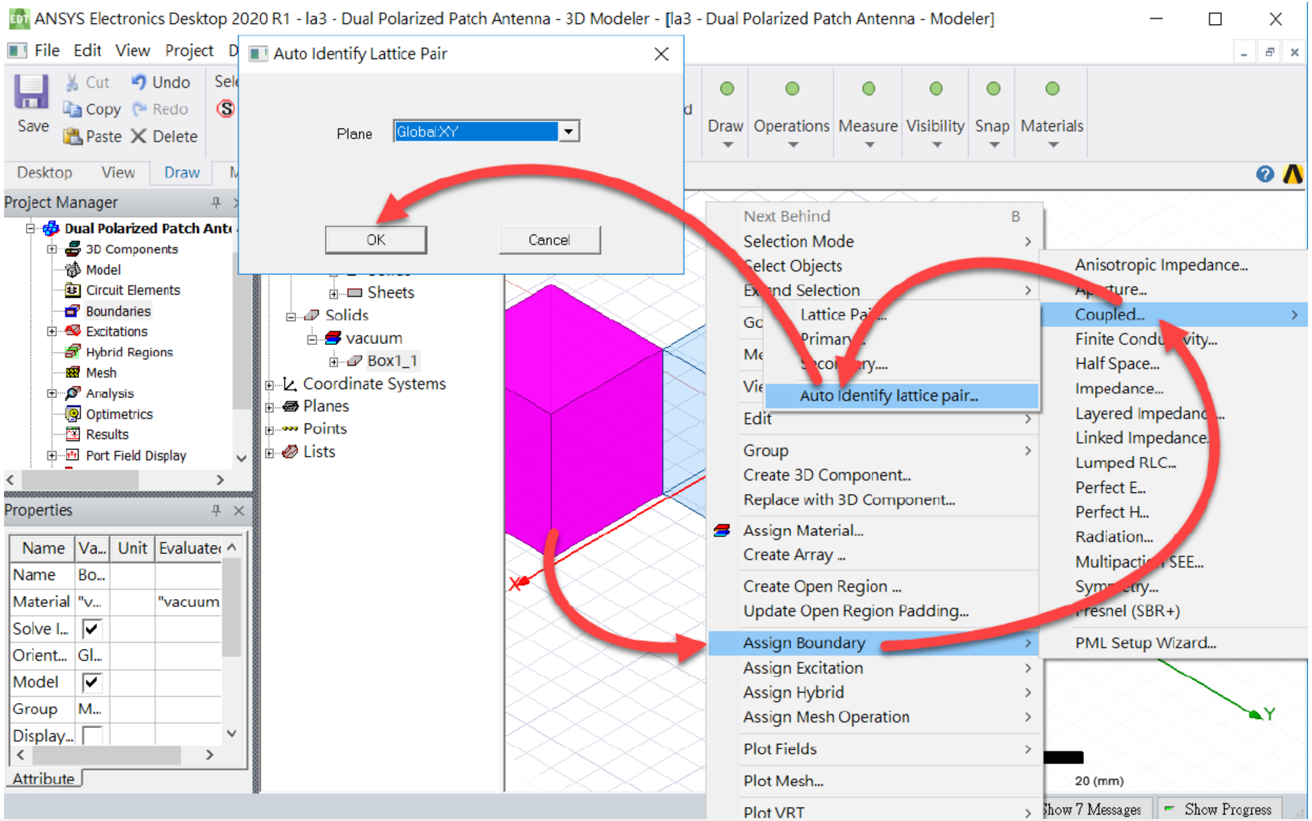
Add FEBI Boundary



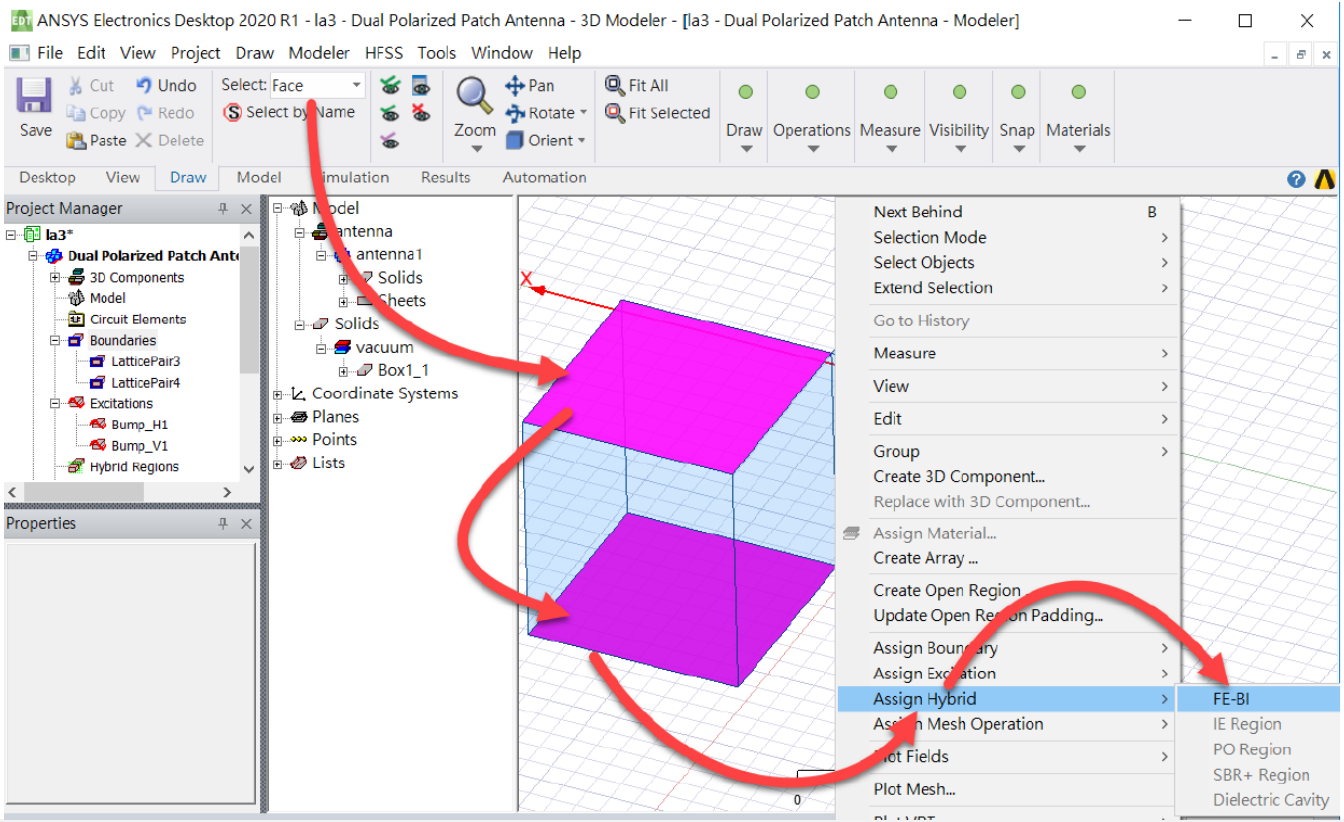
Replace 3D Component



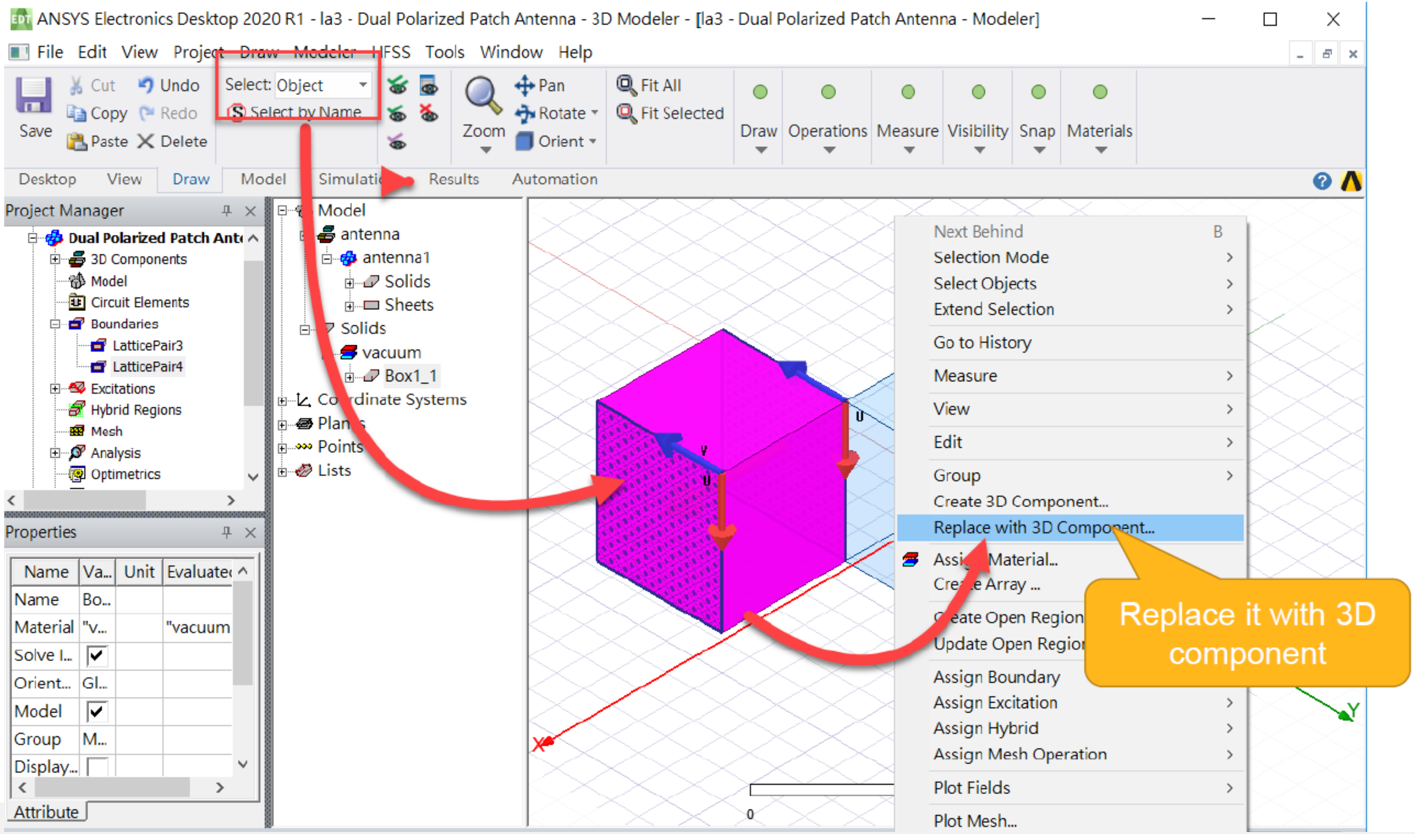
Add lattice Boundary in Air Box



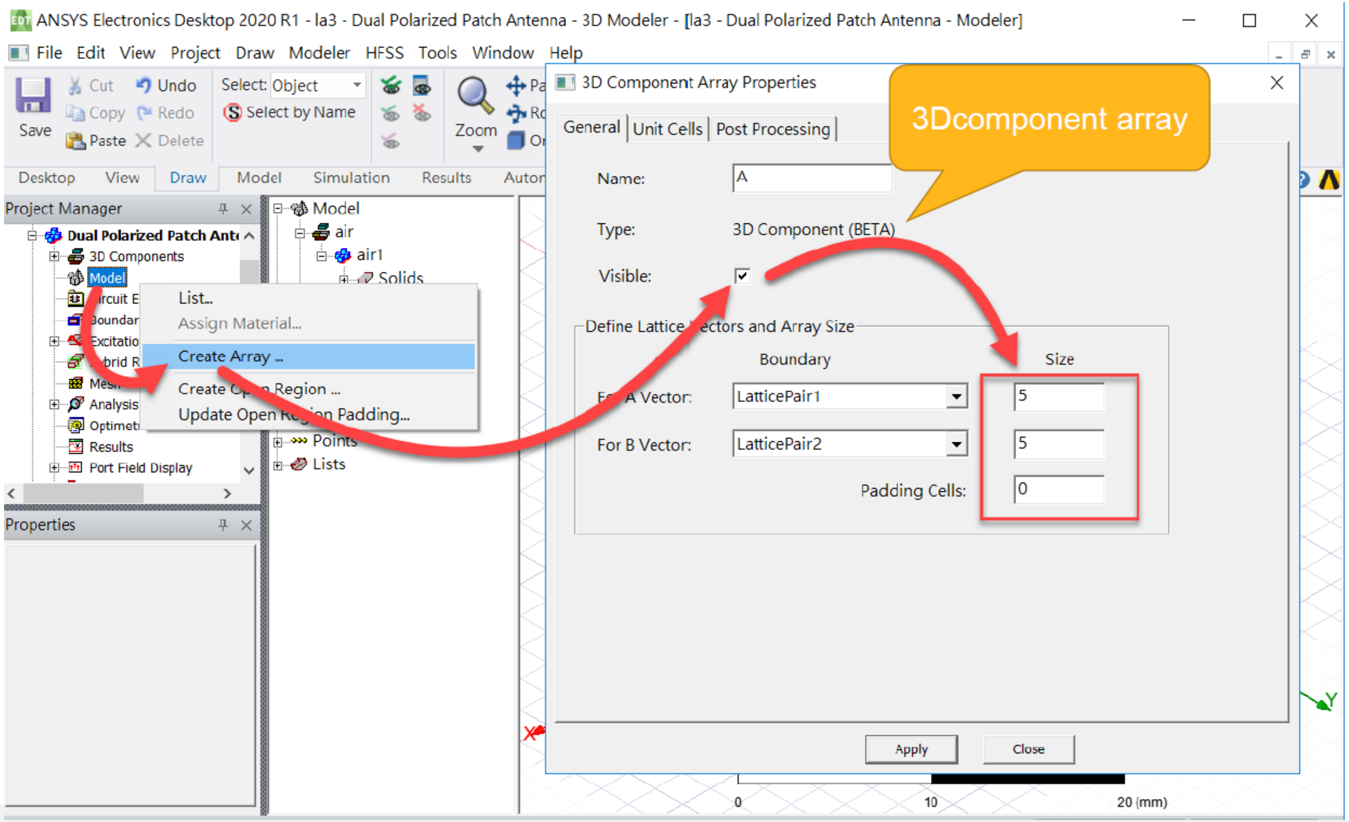
Add FEBI Boundary



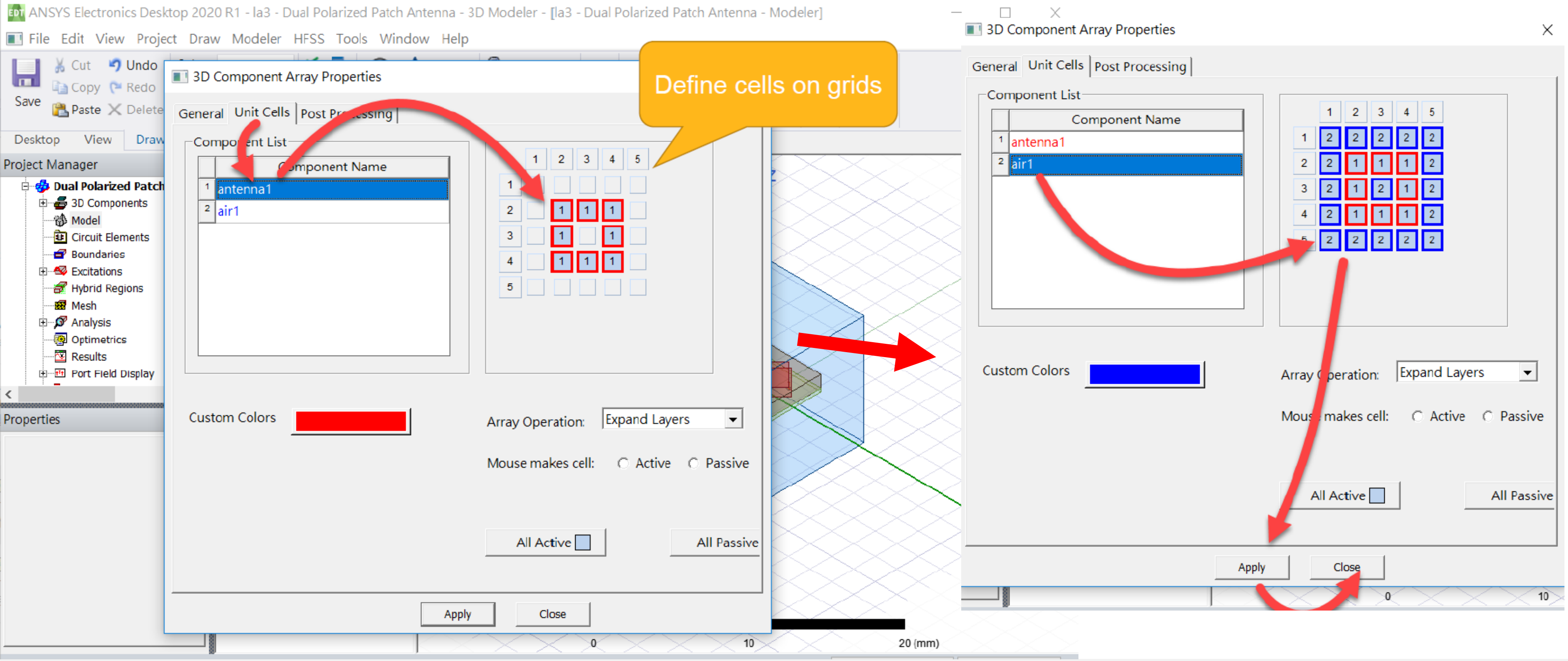
Replace 3D Component



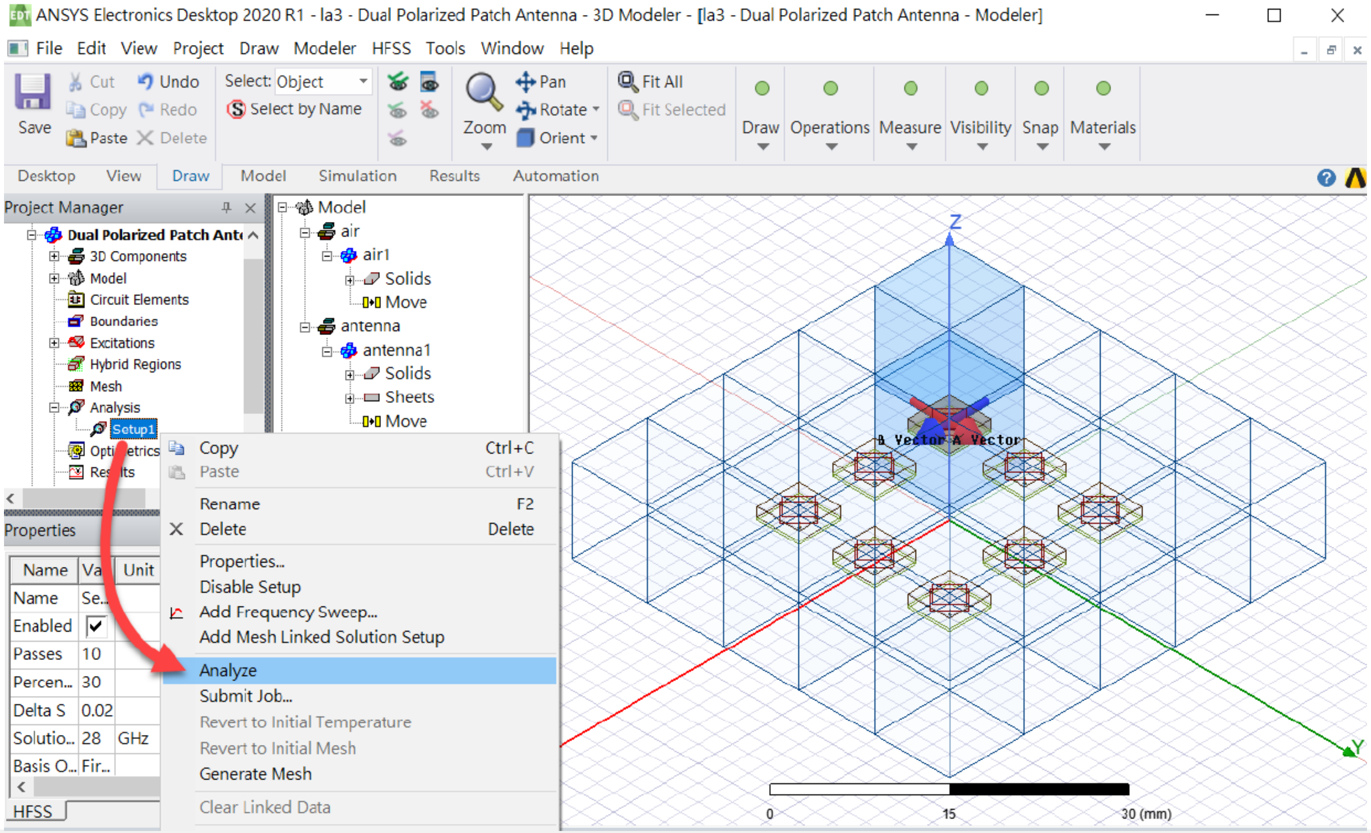
Create Array



Build Array in 3D Component Array GUI

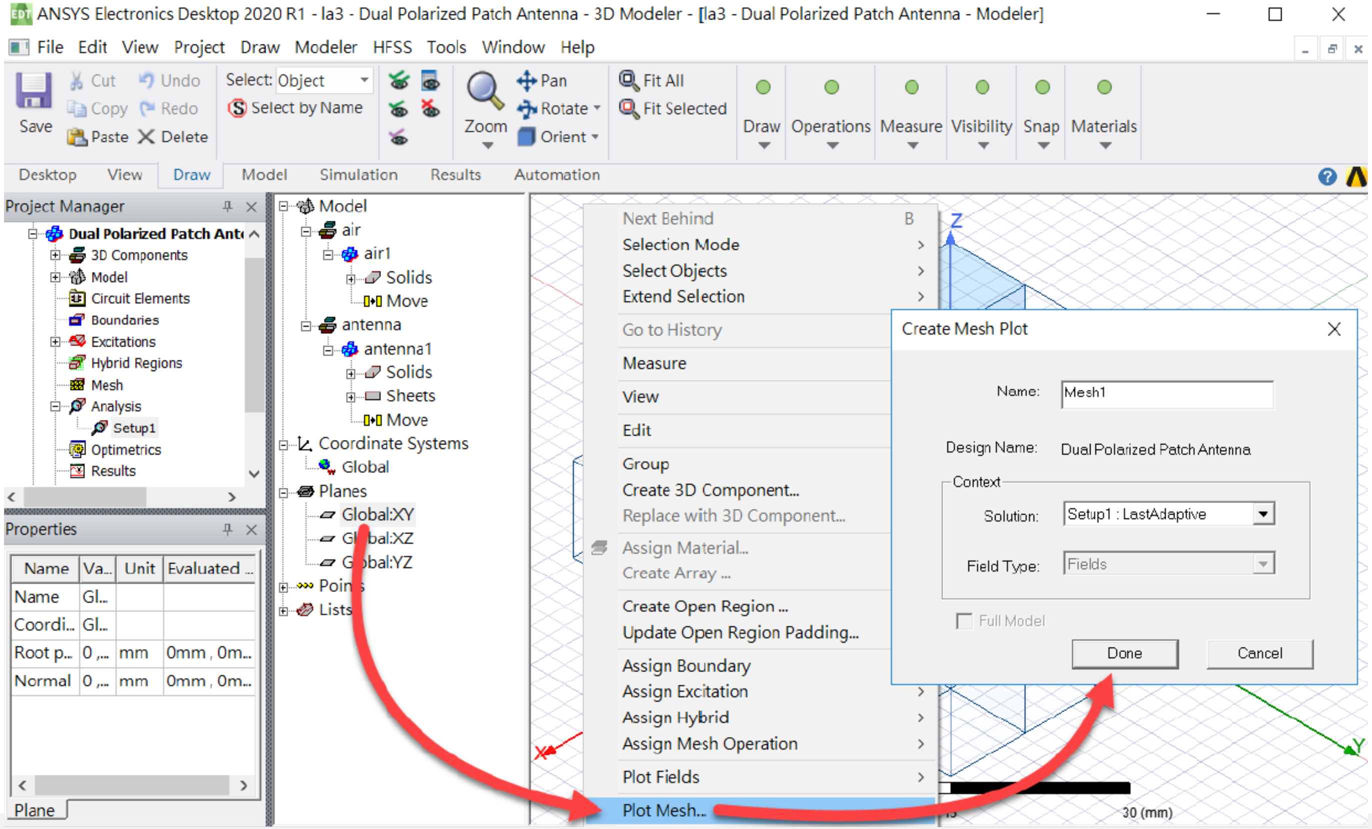


Anslyze



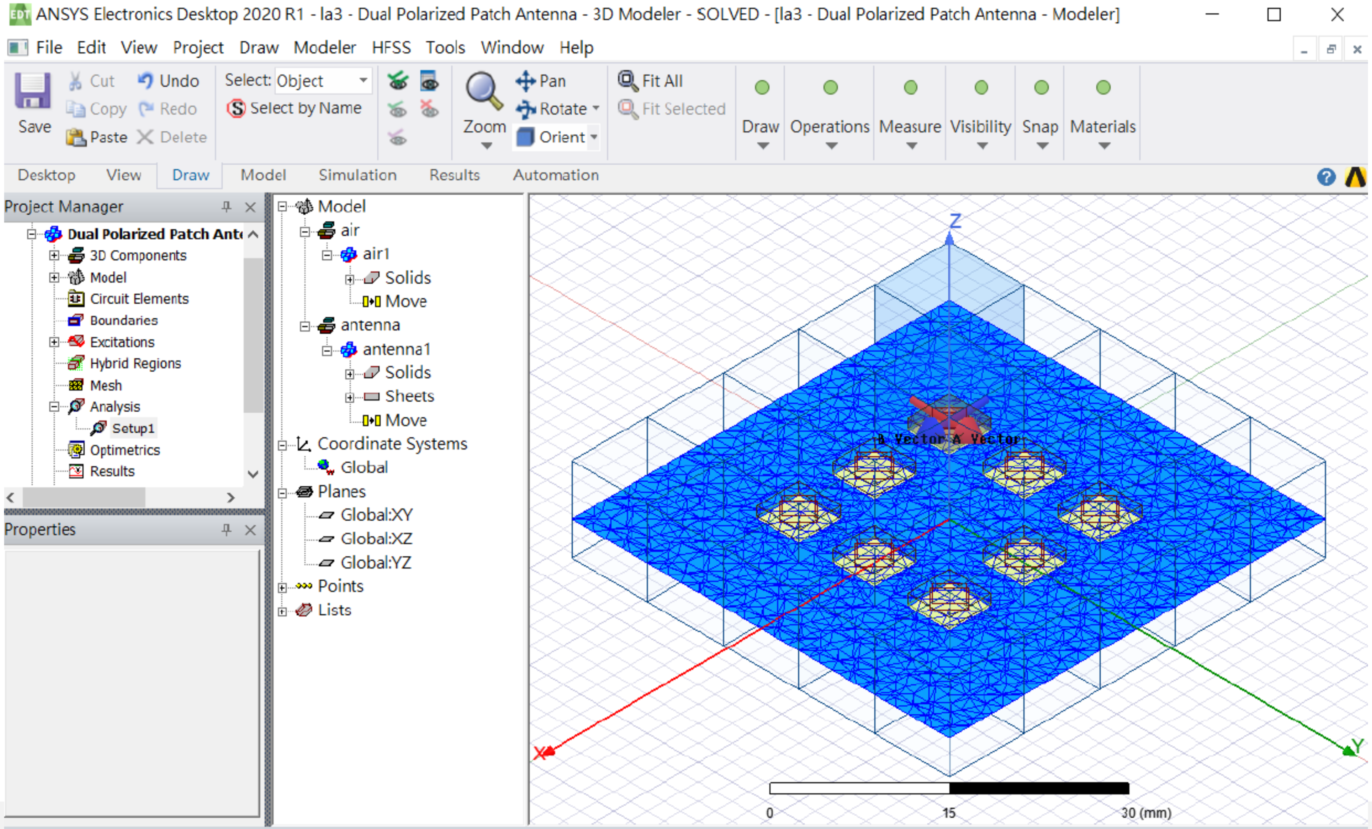
ANSYS Electronics Desktop 2020 R1 - la3 - Dual Polarized Patch Antenna - 3D Modeler - [la3 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut Undo
Copy Redo
Paste Delete
Select: Object
Select by Name
Zoom
Pan
Rotate
Orient
Fit All
Fit Selected
Draw
Operations
Measure
Visibility
Snap
Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
Dual Polarized Patch Ant
3D Components
Model
Circuit Elements
Boundaries
Excitations
Hybrid Regions
Mesh
Analysis
Setup1
Optimetrics
Results
Model
air
air1
Solids
Move
antenna
antenna1
Solids
Sheets
Move
Copy Ctrl+C
Paste Ctrl+V
Rename F2
Delete Delete
Properties...
Disable Setup
Add Frequency Sweep...
Add Mesh Linked Solution Setup
Analyze
Submit Job...
Revert to Initial Temperature
Revert to Initial Mesh
Generate Mesh
Clear Linked Data
Properties
Name Va Unit
Name Se...
Enabled
Passes 10
Percen... 30
Delta S 0.02
Solutio... 28 GHz
Basis O... Fir...
HFSS
Z
Y
0
15
30 (mm)

Plot Mesh



ANSYS Electronics Desktop 2020 R1 - la3 - Dual Polarized Patch Antenna - 3D Modeler - [la3 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut
Copy
Paste
Undo
Redo
Delete
Select: Object
Select by Name
Zoom
Pan
Rotate
Orient
Fit All
Fit Selected
Draw
Operations
Measure
Visibility
Snap
Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
Dual Polarized Patch Ante
3D Components
Model
Circuit Elements
Boundaries
Excitations
Hybrid Regions
Mesh
Analysis
Setup1
Optimetrics
Results
Properties
Name
Va...
Unit
Evaluated ...
Name Gl...
Coordi... Gl...
Root p... 0 ,... mm 0mm , 0m...
Normal 0 ,... mm 0mm , 0m...
Plane
Model
air
air1
Solids
Move
antenna
antenna1
Solids
Sheets
Move
Coordinate Systems
Global
Planes
Global:XY
Global:XZ
Global:YZ
Points
Lists
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...
Create Mesh Plot
Name:
Mesh1
Design Name:
Dual Polarized Patch Antenna
Context
Solution:
Setup1 : LastAdaptive
Field Type:
Fields
Full Model
Done
Cancel
Z
Y
X
30 (mm)

Result



ANSYS Electronics Desktop 2020 R1 - la3 - Dual Polarized Patch Antenna - 3D Modeler - SOLVED - [la3 - Dual Polarized Patch Antenna - Modeler]
File Edit View Project Draw Modeler HFSS Tools Window Help
Save
Cut
Copy
Paste
Undo
Redo
Delete
Select: Object
Select by Name
Zoom
Pan
Rotate
Orient
Fit All
Fit Selected
Draw
Operations
Measure
Visibility
Snap
Materials
Desktop View Draw Model Simulation Results Automation
Project Manager
Dual Polarized Patch Ant
3D Components
Model
Circuit Elements
Boundaries
Excitations
Hybrid Regions
Mesh
Analysis
Setup1
Optimetrics
Results
Properties
Model
air
air1
Solids
Move
antenna
antenna1
Solids
Sheets
Move
Coordinate Systems
Global
Planes
Global:XY
Global:XZ
Global:YZ
Points
Lists
Z
X
Y
0
15
30 (mm)

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