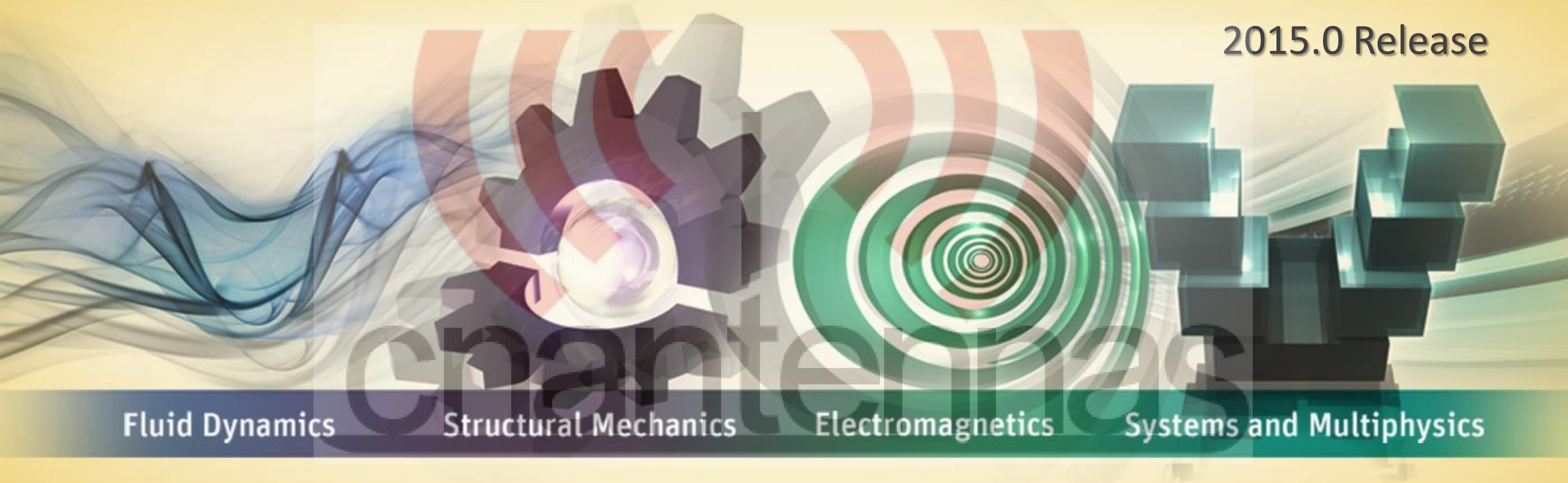
Realize Your Product Promise®

Workshop 10-1: HPC for Finite Arrays

2015.0 Release



ANSYS HFSS for Antenna Design

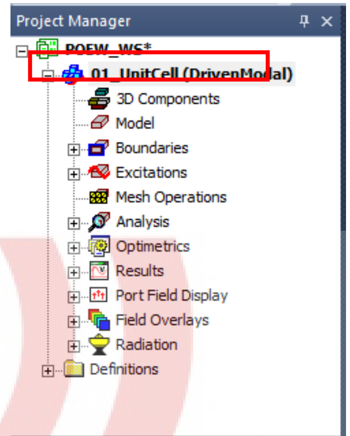
Getting Started

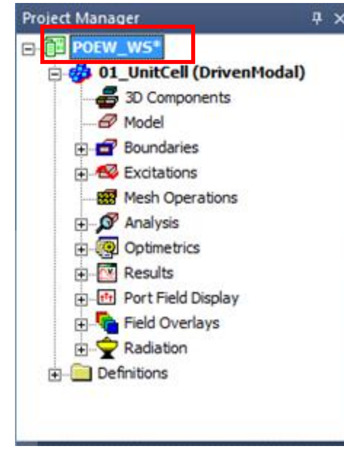
- **Launching *ANSYS Electronics Desktop 2015***
  - Select ***Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.0***
  - Select ***ANSYS Electronics Desktop 2015***.

- **Open File**
  - Select ***File > Open…*** to open **POEW_Finite_Array_WS.aedt**
    - Open the .aedt file you saved from the previous workshop: Unit Cell

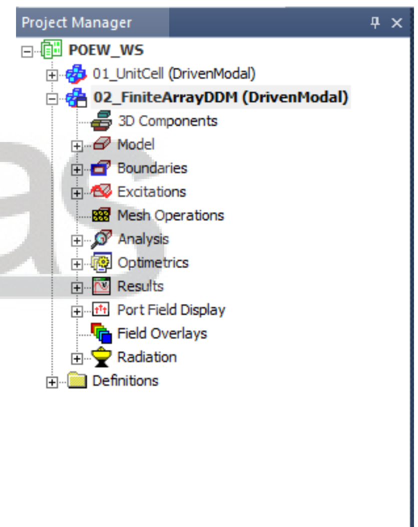
Copy/Paste Unit Cell Design

- **Copy/Paste Unit Cell Design**
  - In the Project Manager window, select: **01_UnitCell (DrivenModal)**
    - Right-click and select the menu item: ***Edit > Copy***
  - In the Project Manager window, select: **POEW_WS**
    - Right-click and select the menu item: ***Edit > Paste***



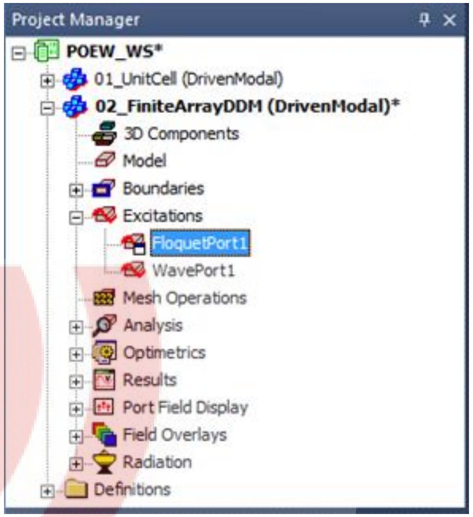


- **Rename Design**
  - In the Project Manager window, select: **01_UnitCell1 (DrivenModal)**
    - Right-click and select the menu item: ***Edit > Rename***
  - Rename design to: **02_FiniteArrayDDM**
    - In the Project Manager window, minimize the Unit Cell design and expand the Finite Array design



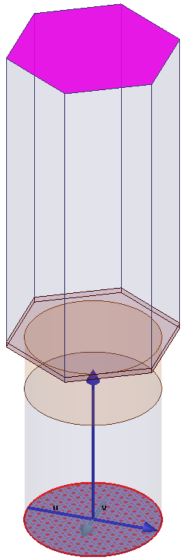
Note: We don't want to invalidate the solution of the unit cell simulation because we will want to use the mesh from this solution to solve the finite array design. By copy/pasting the unit cell design to create the finite array design, we are ensuring that the unit cell used to build the finite array is geometrically-equivalent to the one used in the unit cell simulation --- in other words, we can safely use the same mesh in both designs.

Remove Floquet Port, Add Radiation Boundary



- **Remove Floquet Port**
  - In the Project Manager window, expand the Excitations list
    - Select FloquetPort1
    - Edit > Delete

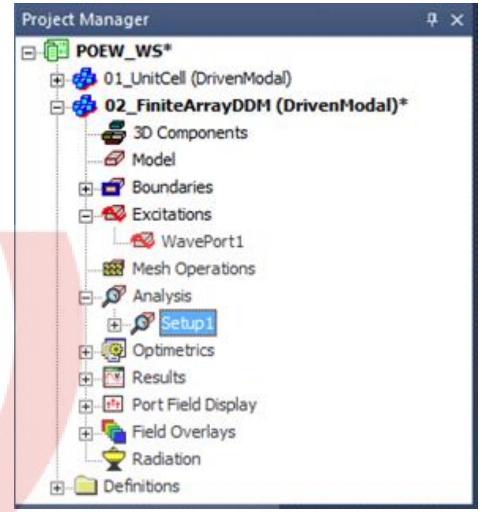
Note: When we solve the finite array, we are no longer using Floquet analysis. The radiation boundary condition assigned to the single unit cell will be used across all elements of the finite array. This could be an ABC, PML or FE-BI.

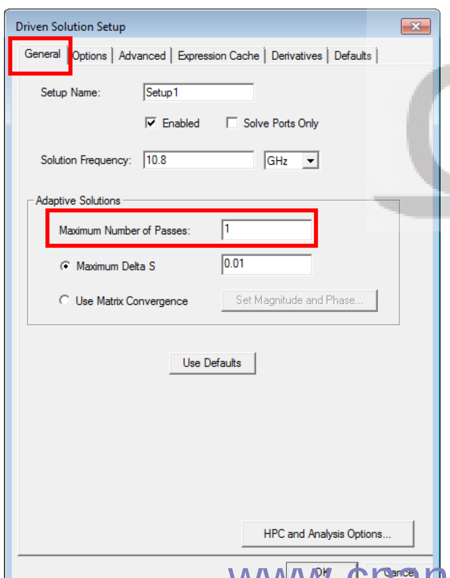


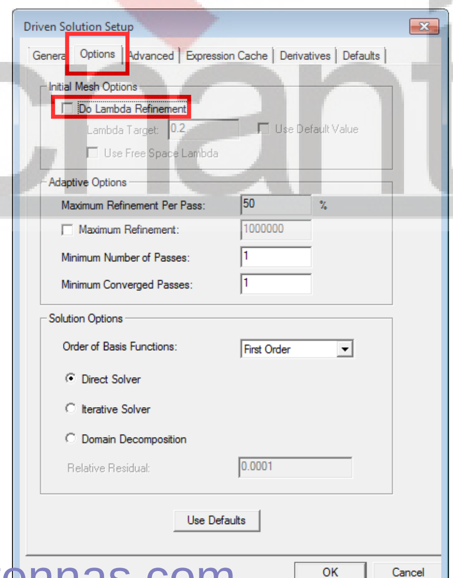
- **Add Radiation Boundary**
  - Edit > Select > Faces
  - In the 3D Modeler window, select the top face where the Floquet port was once defined
  - HFSS >> Boundaries >> Assign >> Radiation
  - Select OK

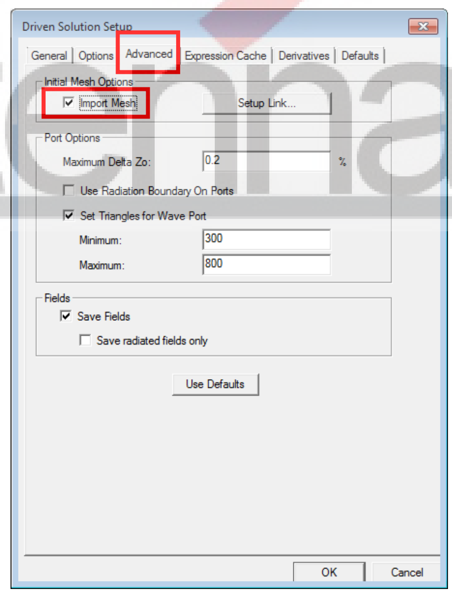
Analysis Setup

- **Driven Solution Setup**
  - In the Project Manager Window, expand the **Analysis** list
  - Double-click on **Setup1**
    - **General** tab
      - **Maximum Number of Passes: 1**
    - **Options** tab
      - Uncheck **Do Lambda Refinement**
    - **Advanced** tab
      - Check **Import Mesh**





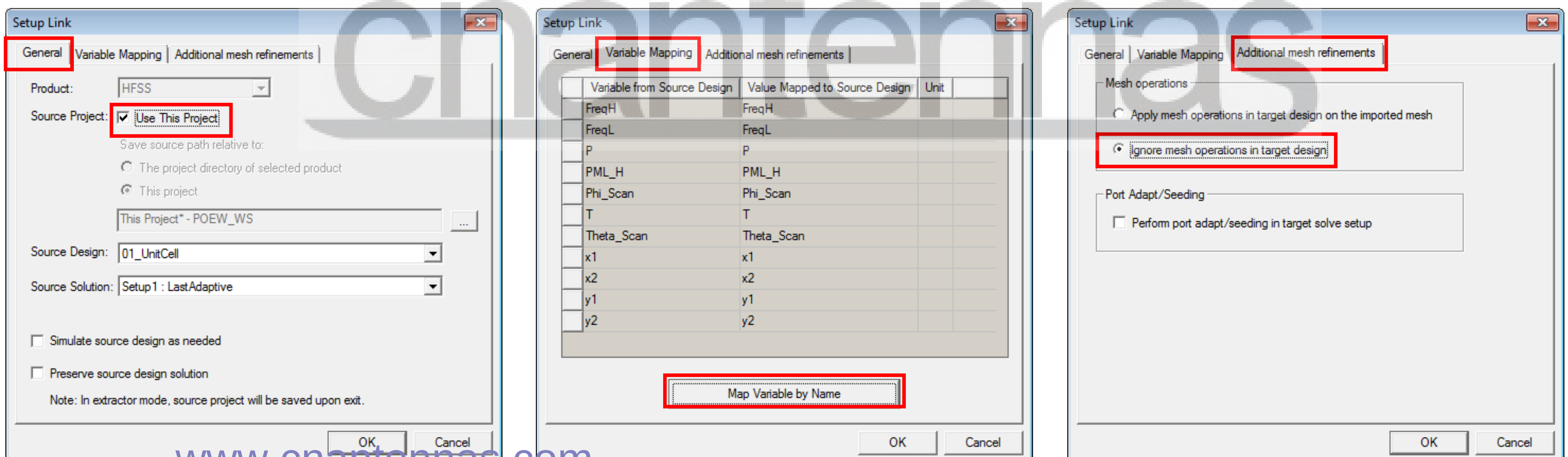




Note: These settings will ensure that absolutely no additional meshing will take place since we are importing a well-converged mesh from the unit cell model for use in the finite array solve.

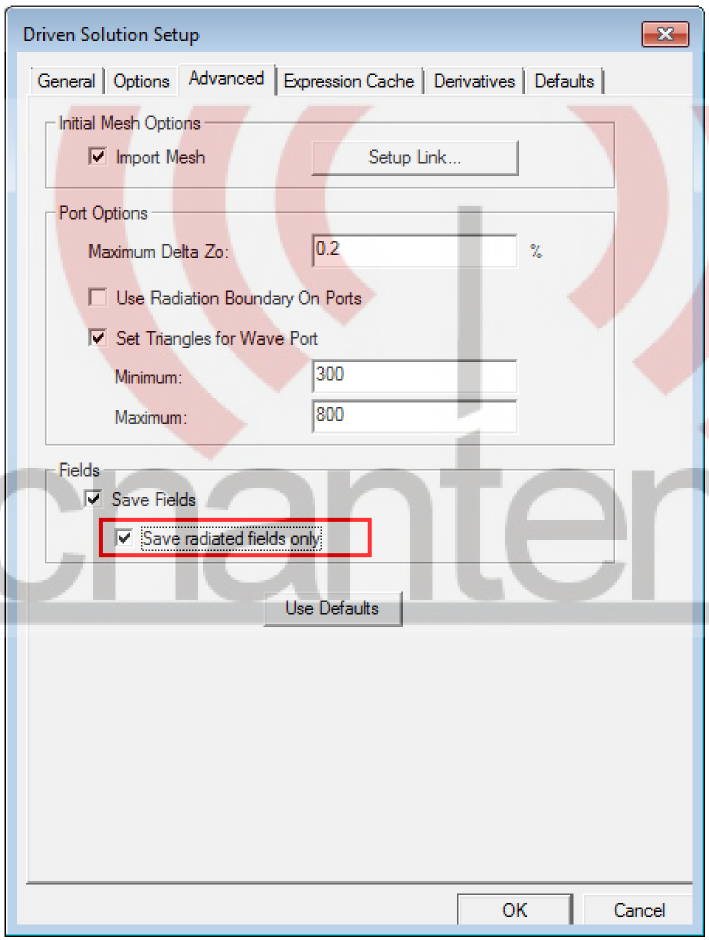
Analysis Setup

- **Import Mesh**
  - **General** tab
    - Check **Use This Project**
    - Verify:
      - **Source Design: 01_UnitCell**
      - **Source Solution: Setup1 : LastAdaptive**
  - **Variable Mapping** tab
    - Select **Map Variable by Name**
  - **Additional mesh refinements** tab
    - Select **Ignore mesh operations in target design**
  - Select **OK**



Save Radiated Fields Only

- Check the box for **Save radiated fields only**
- Select **OK**



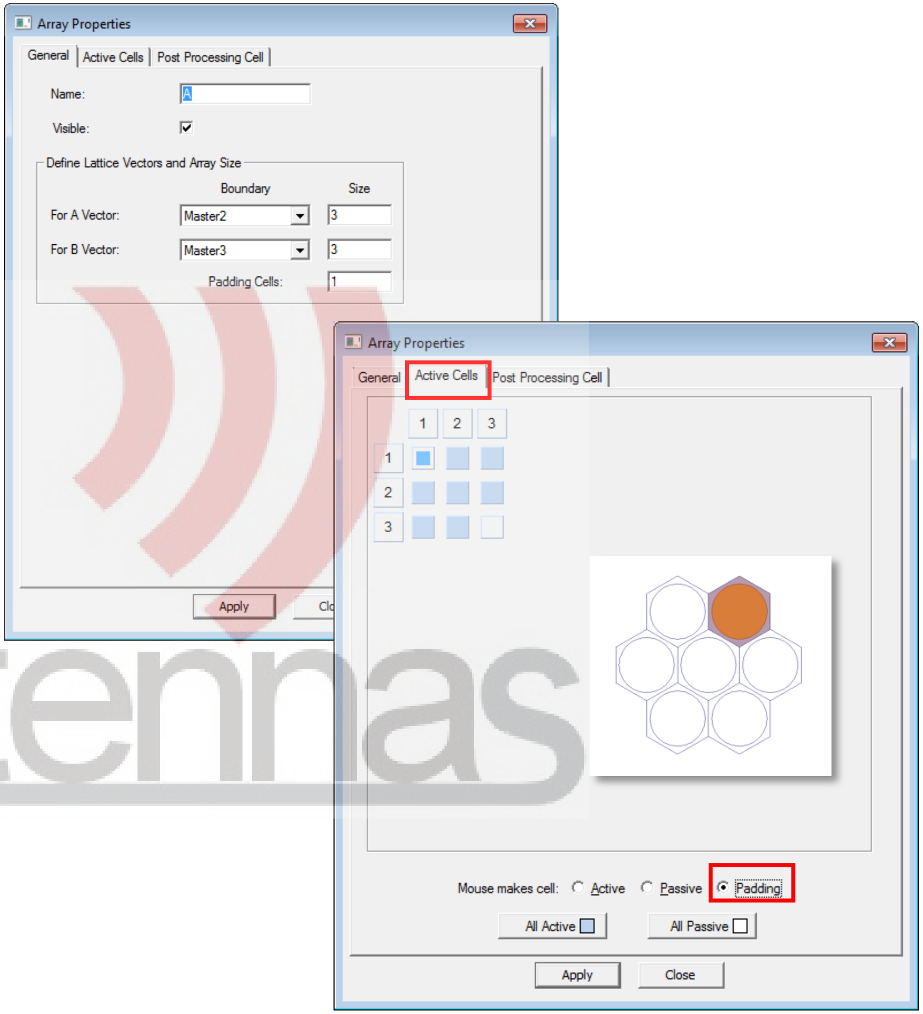
Note: For large finite arrays, solving all fields may require significant disk space. In most cases, the radiated fields are the only field solutions of interest. How much disk space will you save by doing this? One example: 9x5 patch array with 90 excitations saw a 23.6x reduction in disk space (15.6GB with saved fields vs 0.66GB with save radiated fields only option).

Create Finite Array

- ***HFSS >> Model >> Create Array…***
  - **General** tab
    - Check the box for **Visible**
    - **Define Lattice Vectors and Array Size**

	Boundary	Size
For A Vector:	Master2	3
For B Vector:	Master3	3

  - **Active Cells** tab
    - Select the radio button for **Padding**
    - Click on cells (1,1) and (3,3) to convert them from **Active** to **Padding** cells
  - Select **Apply**
  - Select **Close**



ANSYS

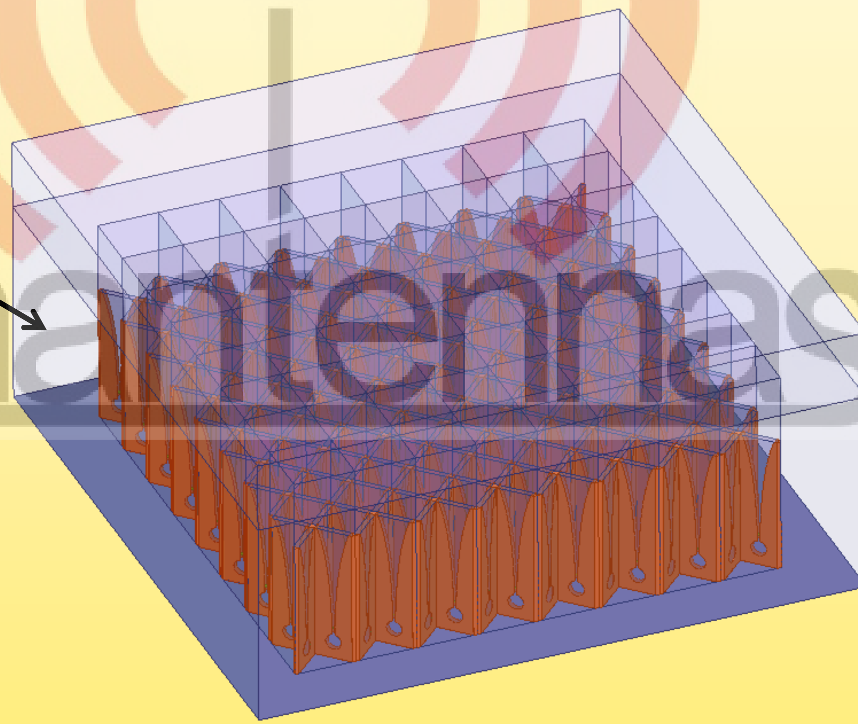
Active vs Passive vs Padding

Note:

- Active elements: the geometry of the unit cell is there, ports are active
- Passive elements: the geometry of the unit cell is there, ports are perfectly terminated
- Padding elements: the geometry of the unit cell is removed (cell is filled with air), ports do not exist

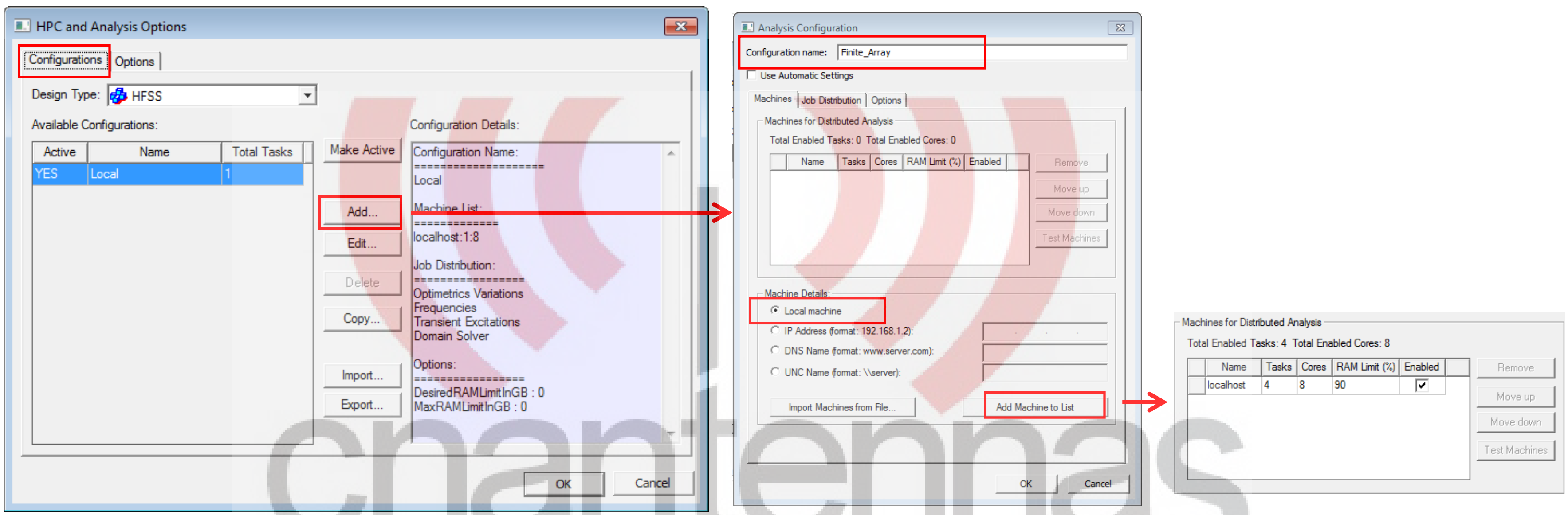
In the finite array simulation, padding elements border the entire array. These are not visible to the user. When comparing the finite array DDM results to an explicitly drawn array of the same configuration, one must remember to include the air padding buffer around the model as shown below.

Air buffer region mimics DDM



High Performance Computing (HPC)

- ***Tools>>Options>>HPC and Analysis Options***
  - Select the Configurations tab
  - Add a finite array HPC configuration by selecting the “Add…” button

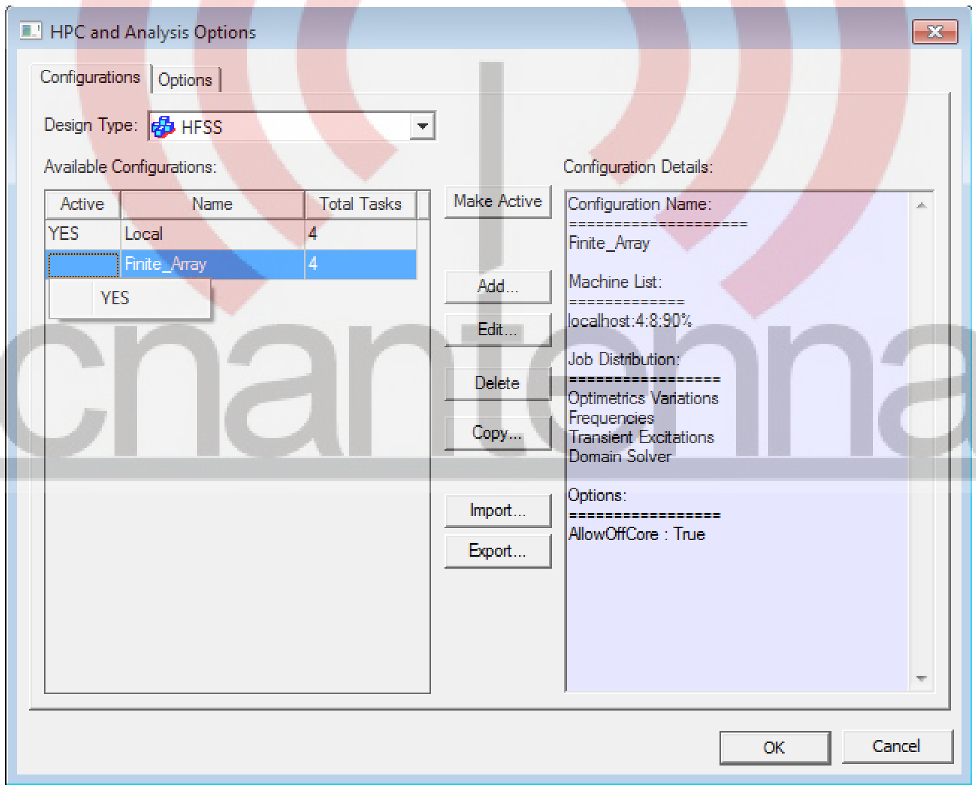


- **Analysis Configuration**
  - Configuration name: Finite_Array
  - Machine Details:
    - **Local machine**
    - **Number of Tasks: 4**
    - **Total Cores: 4 or greater (list total number of cores in your local machine)**
  - Select **Add Machine to List**
  - In the **Job Distribution** tab, make sure that **Domain Solver** is checked
  - Select **OK**

Note: Can add more machines and distribute the tasks across machines if local machine does not have enough memory to solve the whole problem. See slide 16 for more information for how to choose number of tasks.

Finite_Array HPC Configuration

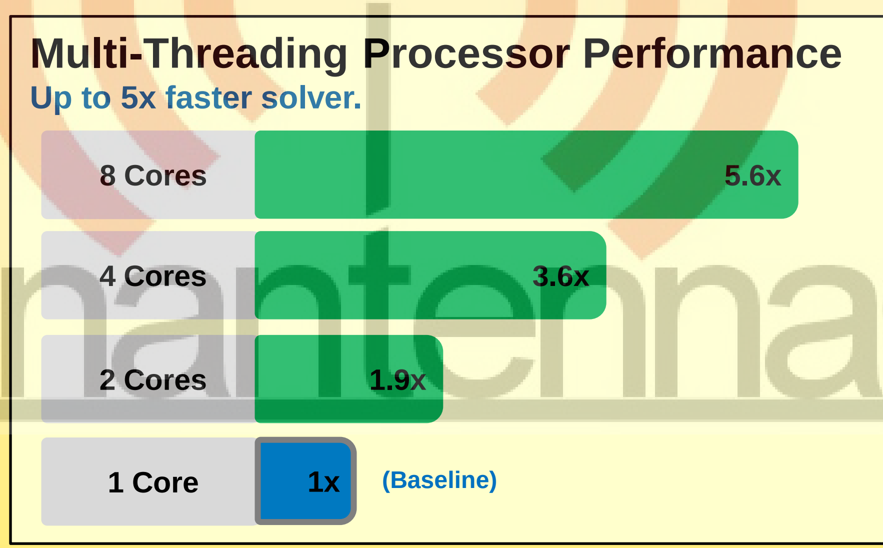
- **HPC Cont'd**
  - Select the row labeled as **Finite_Array** from the **Available Configurations** list
  - In the Active column, select the empty cell and toggle to **YES**
  - Select **OK**



ANSYS

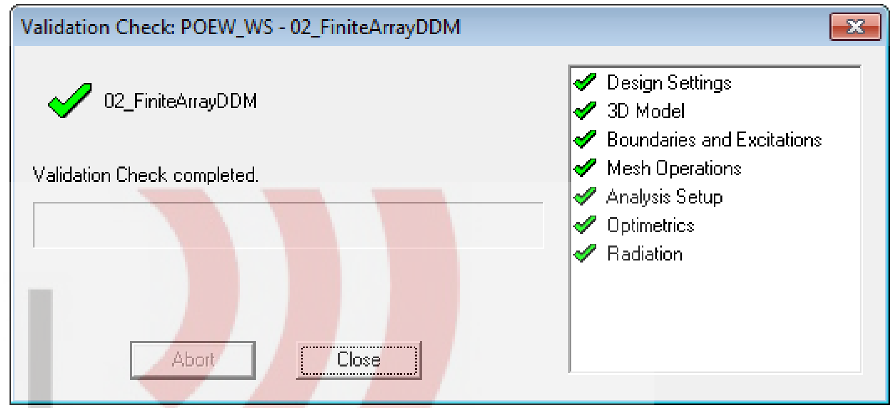
Why 4 Tasks?

Note: Finite array distributed domain method splits the problem into "domains" and solves the different domains in parallel for a more efficient (less RAM, less time) solve. This requires at least 3 tasks: one task will be the "head node" and the other two tasks are domains. 4 tasks are defined here: one "head node" and 3 domains. The number of tasks assigned will distribute evenly across the available cores. Should I assign 3 tasks (where two domains will be solved in parallel, each using 3 cores per domain) or 4 tasks (where three domains will be solved in parallel, each using 2 cores per domain)? **Our general recommendation for domain distribution problems is to assign 2-4 cores per task whenever possible while maximizing the number of domains you solve in parallel. Anymore than 4 cores per task may amount to some speed-up in solve time, but not as much speed-up as what is seen between having 1 core per domain vs 2 cores per domain (i.e., approx 2x).**

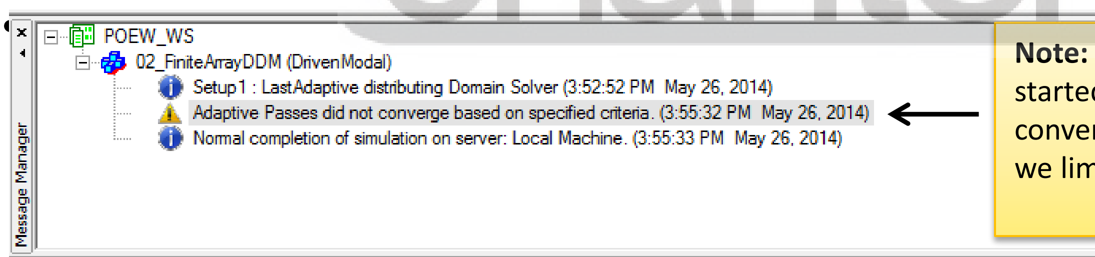


Save and Analyze

- ***File >> Save***
- ***HFSS >> Validation Check***
  - Select **Close**



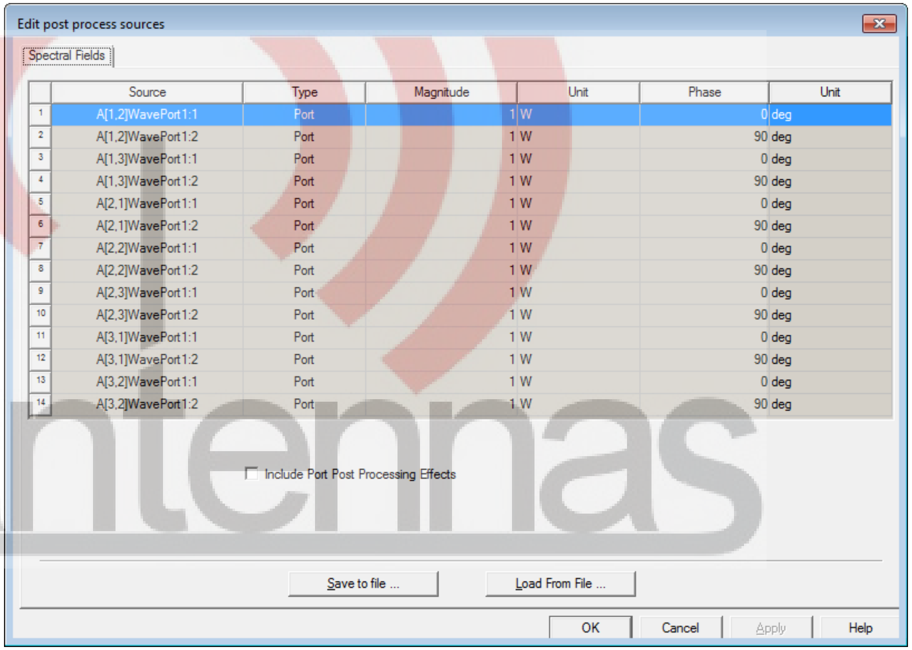
- ***Note: Before running the simulation disable or delete the optometric setup.***
- ***HFSS >> Analyze All***



Note: This warning message is ok! Remember we started this simulation by importing a well-converged mesh from the unit cell model and then we limited number of adaptive passes to 1.

Edit Sources

- ***HFSS >> Fields >> Edit Sources***
  - In the **Magnitude** column, change all 0 values to 1
  - In the **Phase** column, change every other value from 0 to 90
  - Select **Apply**
  - Select **Close**

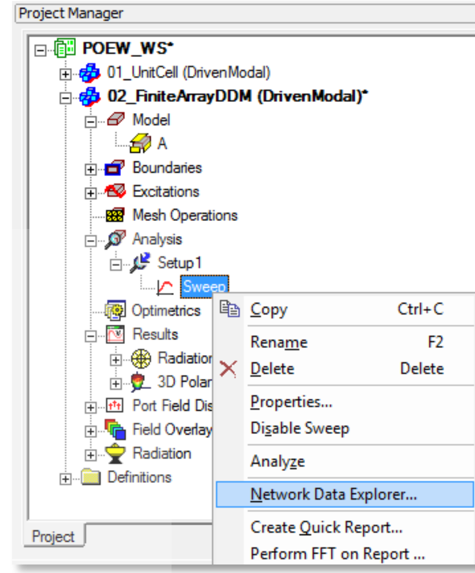


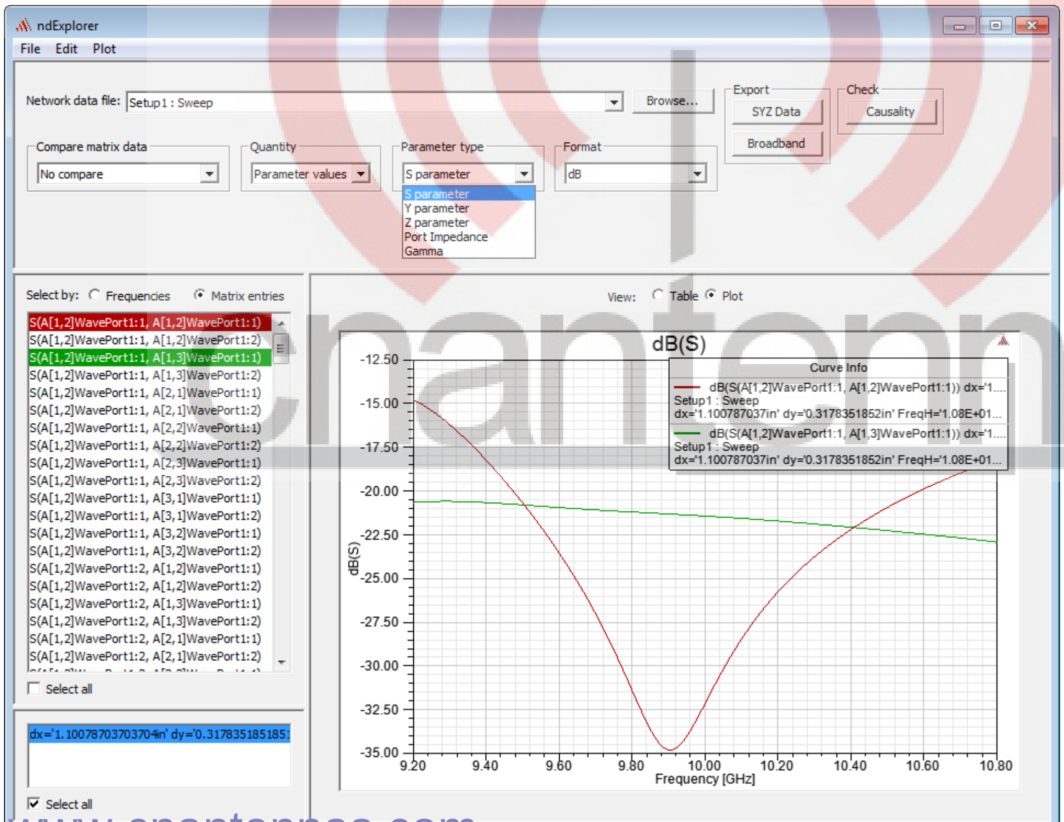
Note: The default Solution Type in HFSS is Network Analysis (***HFSS >> Solution Type***). This solution type solves for every possible excitation so editing the magnitude and phase at each port is a post-processing step that scales the field solutions accordingly. To solve for only one excitation scheme (much faster solve for finite arrays with a large number of ports), select ***HFSS >> Solution Type*** and toggle the radio button from **Network Analysis** to **Composite Excitation.** In a Composite Excitation solve, the above Edit Sources dialogue will no longer be a post-processing window – changes to the excitations will require a re-solve for fields to update.

ANSYS

Network Data Explorer

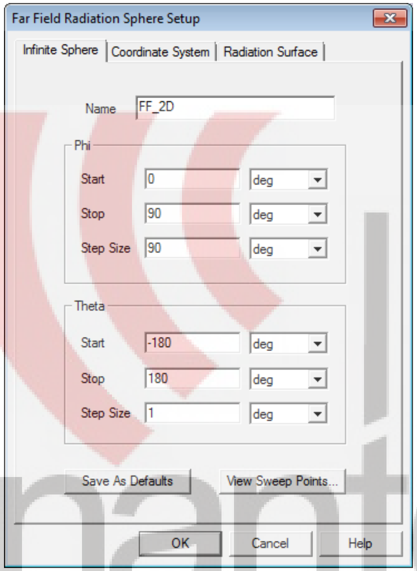
- **SYZ Results**
  - In the Project Manager window, expand the **Analysis** list and expand the list under **Setup1**
  - Right-mouse-click on **Sweep >> Network Data Explorer**
  - View results, export results (select **Broadband** to view export options)

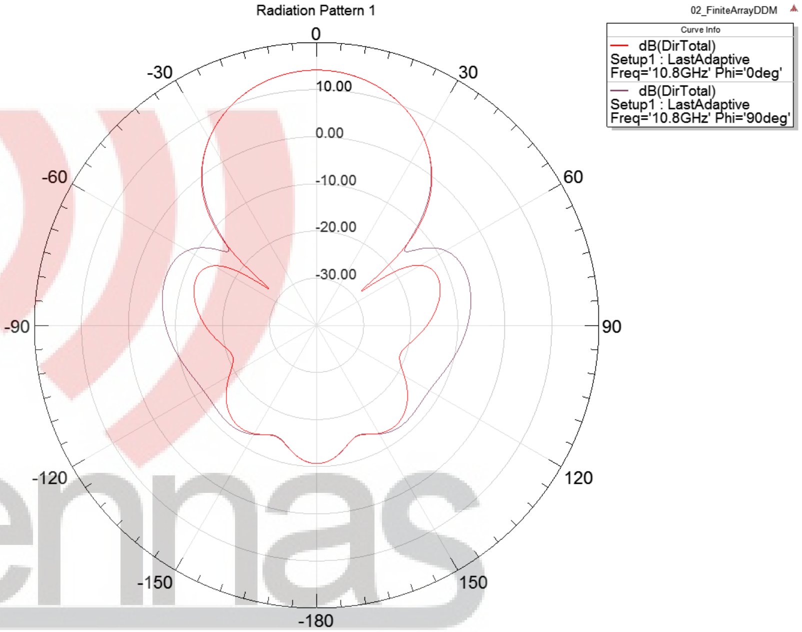




2D Radiation Pattern

- **Radiation Setup**
  - ***HFSS >> Radiation >> Insert Far Field Setup >> Infinite Sphere…***
    - **Name: FF_2D**
    - Phi
      - **Start: 0deg**
      - **Stop: 90deg**
      - **Step: 90deg**
    - Theta
      - **Start: -180deg**
      - **Stop: 180deg**
      - **Step: 1deg**
  - Select **OK**

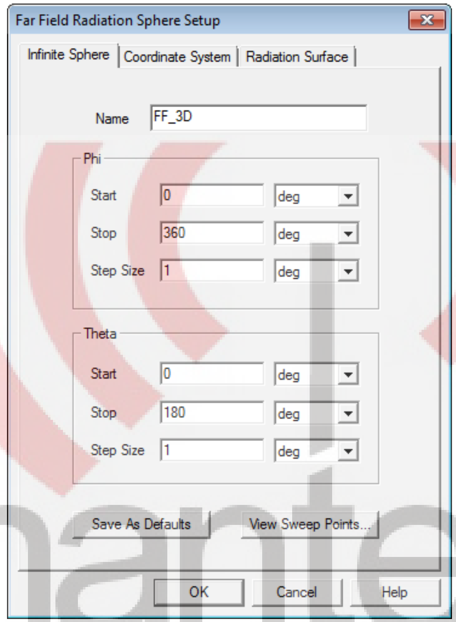


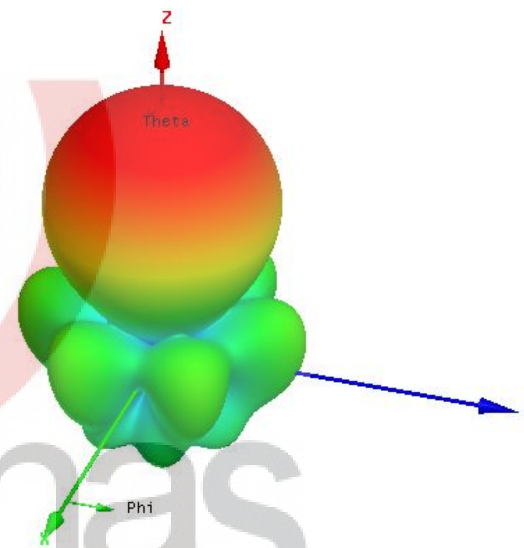


- **Total Directivity**
  - ***HFSS >> Results >> Create Far Field Report >> Radiation Pattern***
  - **Category: Directivity**
  - **Quantity: DirTotal**
  - **Function: dB**
  - Select **New Report**
  - Select **Close**

3D Polar Plot

- **Radiation Setup**
  - ***HFSS >> Radiation >> Insert Far Field Setup >> Infinite Sphere…***
    - **Name: FF_3D**
    - Phi
      - **Start: 0deg**
      - **Stop: 360deg**
      - **Step: 1deg**
    - Theta
      - **Start: 0deg**
      - **Stop: 180deg**
      - **Step: 1deg**
  - Select **OK**





- **Total Gain**
  - ***HFSS >> Results >> Create Far Field Report >> 3D Polar Plot***
  - In the Context field, select **FF_3D** from the Geometry drop-down menu
  - **Category: Gain**
  - **Quantity: GainTotal**
  - **Function: dB**
  - Select **New Report**
  - Select **Close**

Overlay Plot

- ***Window >> 2 POEW_WS - 02_FiniteArrayDDM - Modeler***
- **Relative Coordinate System**
  - ***Modeler >> Coordinate System >> Create >> Relative CS >> Offset***
  - In the data entry fields at the bottom of the HFSS working window, enter the following (use the **Tab** key to move from field to field):
    - **X: 0**
    - **Y: -550**
    - **Z: 0**
    - Click the **Enter** key on your keyboard to finish data entry
- **Modify Radiation Setup**
  - In the Project Manager window, expand the **Radiation** list
  - Double-click on **FF_3D**
  - In the **Coordinate System** tab, select **Use local coordinate system**
  - Select **RelativeCS1** from the drop-down menu
  - Select **OK**
- ***HFSS >> Fields >> Plot Fields >> Radiation Field***
  - Check the box for **Visible**
  - **Transparency: 0.3, Scale: 0.8**
  - Select **Apply**
  - Select **Close**