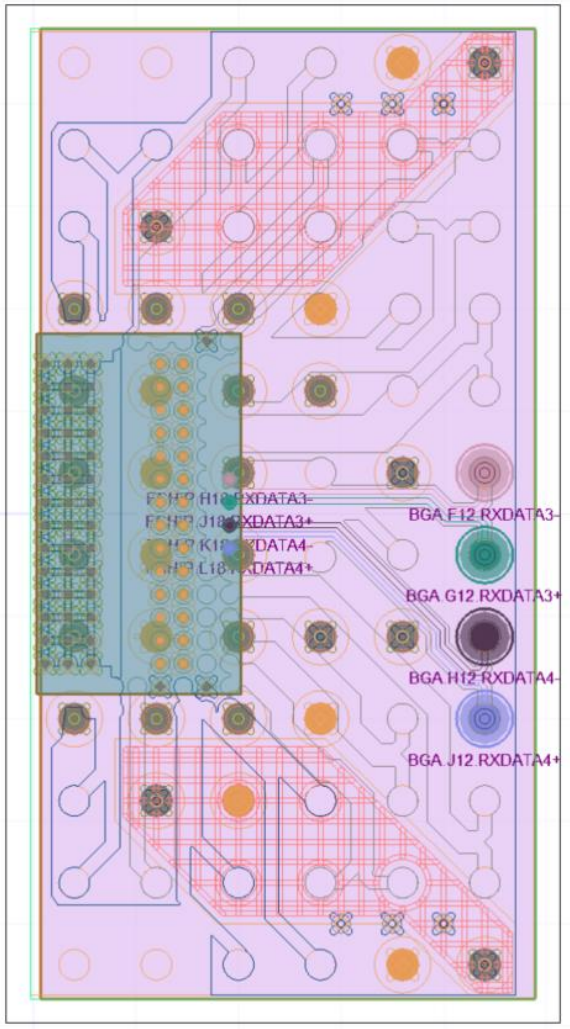
Workshop 4.1: BGA Package Differential Pairs

ANSYS HFSS 3D Layout Getting Started Course
WS4.1

Outline - HFSS 3D Layout BGA Package Differential Pairs Workshop

This HFSS 3D Layout workshop W01 starts with a PCB geometry and goes through the simulation workflow steps to simulate two differential pairs.

1.	Starting BGA geometry	hfss_layout_bga0.aedt
2.	Edit stack up and verify materials	hfss_layout_bga1.aedt
3.	Cut out subdesign	hfss_layout_bga2.aedt
4.	Add solder balls and flip chip bumps	hfss_layout_bga2.aedt
5.	Add ports	hfss_layout_bga2.aedt
6.	Define Extents	hfss_layout_bga2.aedt
7.	Add Solution Setup and Frequency Sweep	hfss_layout_bga2.aedt
8.	Simulate	hfss_layout_bga3.aedt
9.	Plot S-parameters	hfss_layout_bga4.aedt

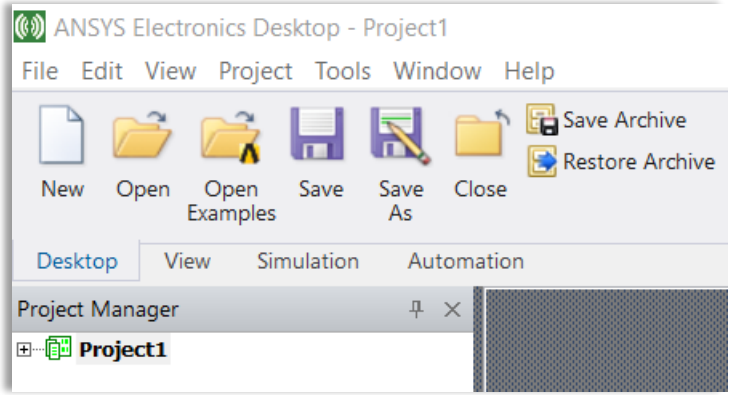


HFSS: Launching ANSYS Electronics Desktop

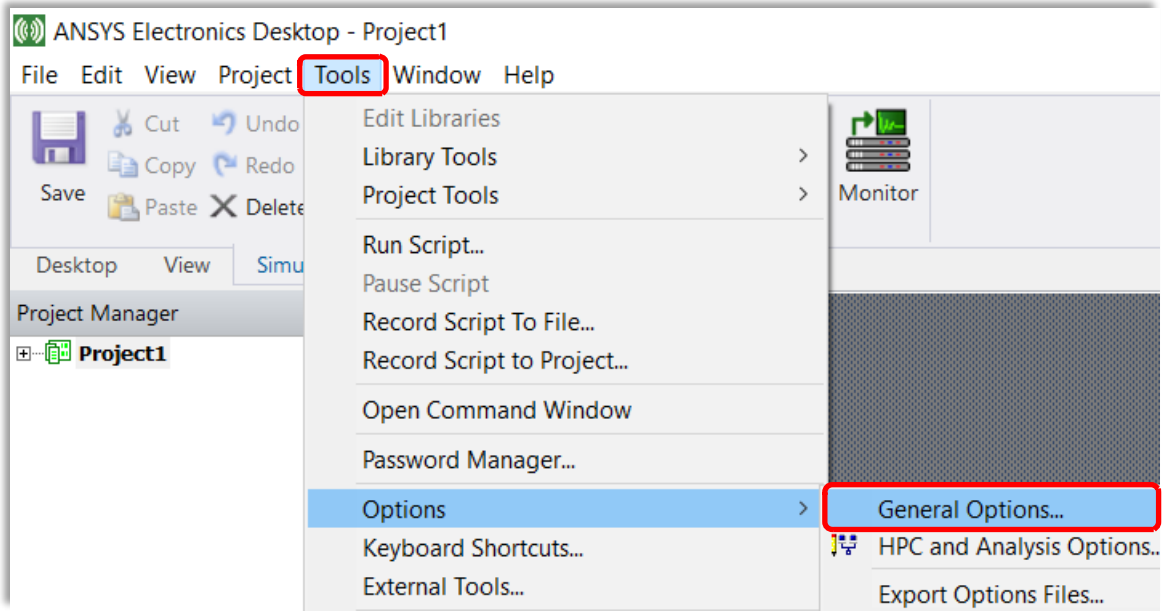
Open HFSS and Set Options (if not already set)

- A To access HFSS, click the Microsoft Start button, Select:
  Programs > ANSYS Electromagnetic Suite > ANSYS Electronics Desktop

  A new "Project1" appears under the Project Manager.
- Setting Tool Options ***(skip this slide and two more if already done.)***
  Select the menu item ***Tools > Options > General Options***

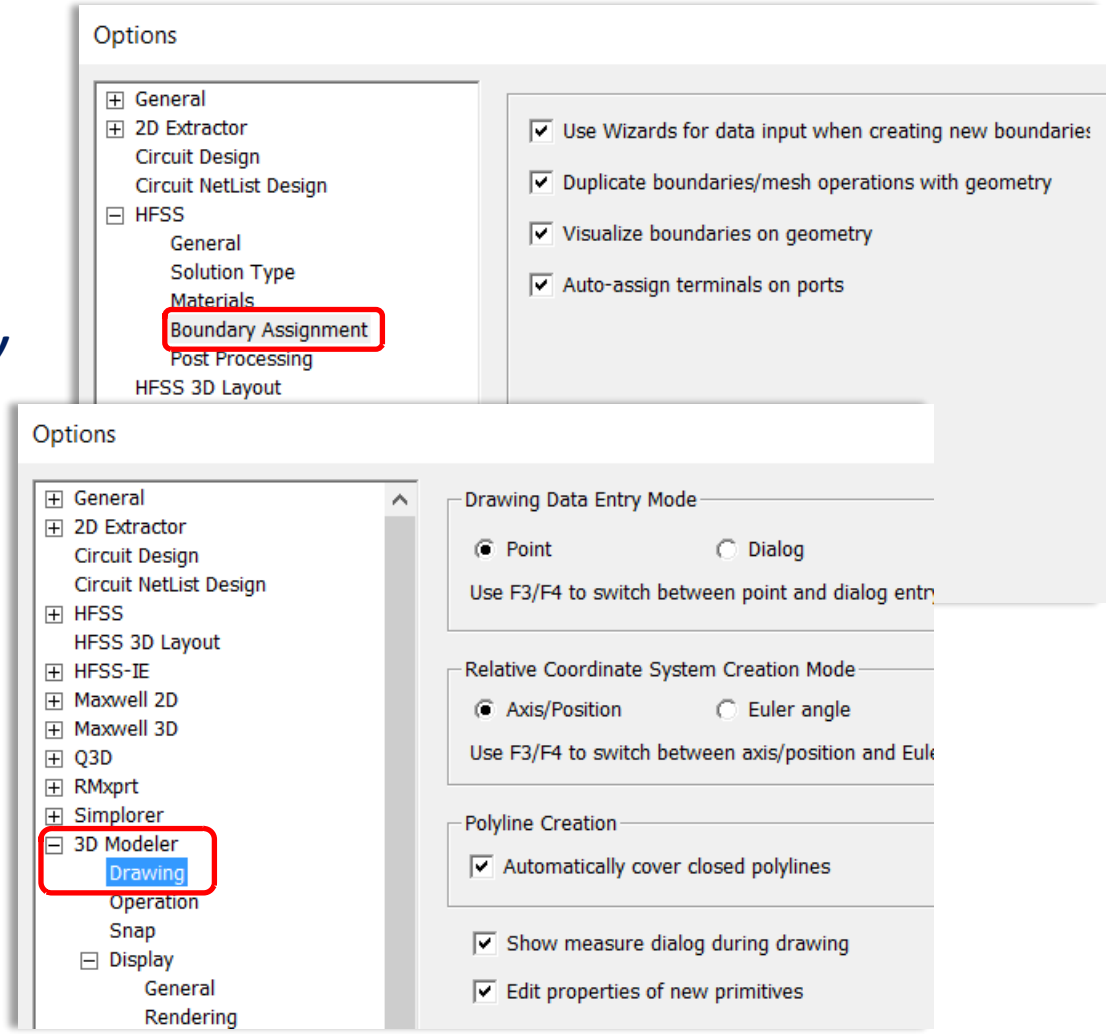


Option settings suggested here ensure that the user can consistently follow the steps in the Workshop.



Setting *Tool > Options* for HFSS 1 - Boundary and Drawing

- Selected: ***Tools > Options > General Options***
- Expand ***HFSS*** (by clicking on the + sign) and Select ***Boundary Assignment***
  - ***Check all entries***
- Expand ***3D Modeler*** **and click** ***Drawing***
  - ***Automatically cover closed polylines*****: ☑ Checked**
  - ***Edit properties of new primitives*****: ☑ Checked**



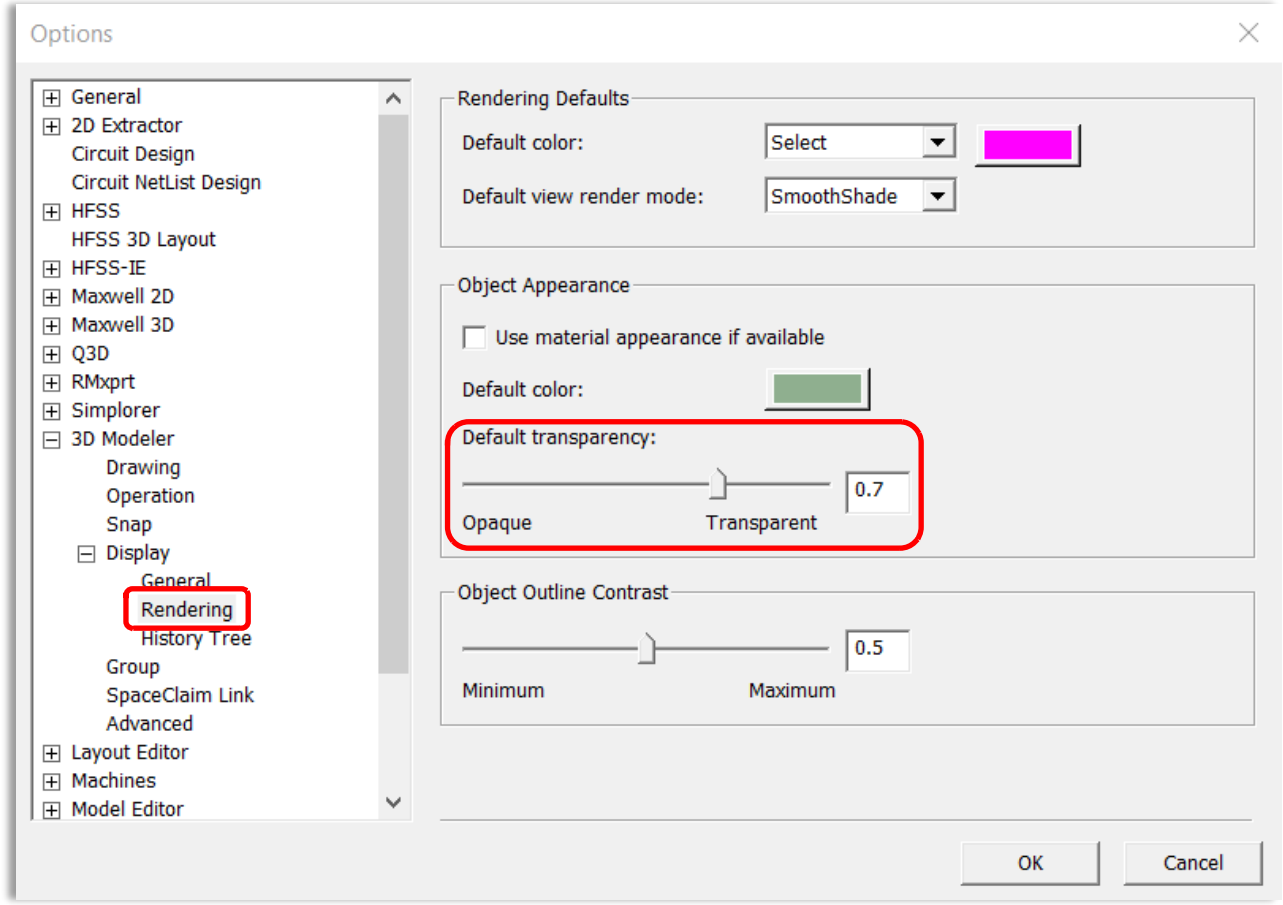
Option settings suggested here ensure that the user can consistently follow the steps in the Workshop.

Setting *Tool > Options* for HFSS 2 - Display History and Transparency

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

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

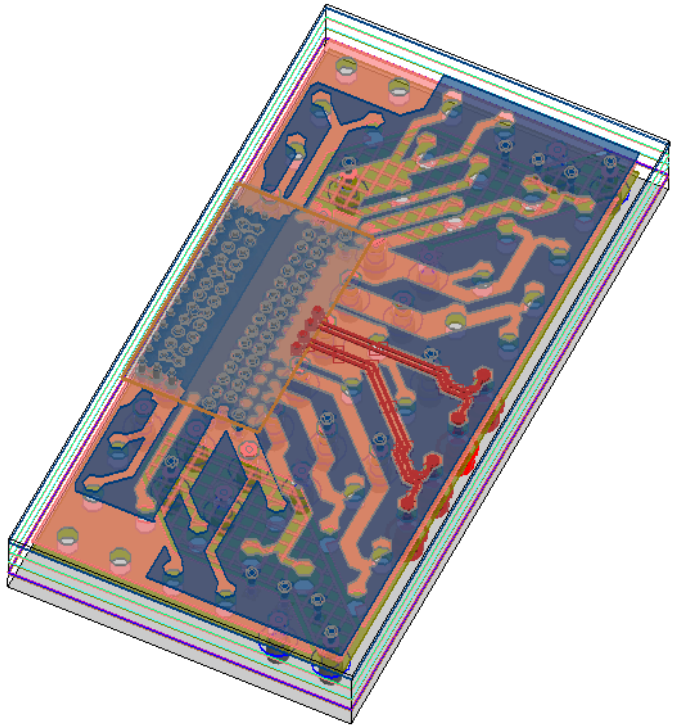
Option settings suggested here ensure that the user can consistently follow the steps in the Workshop.

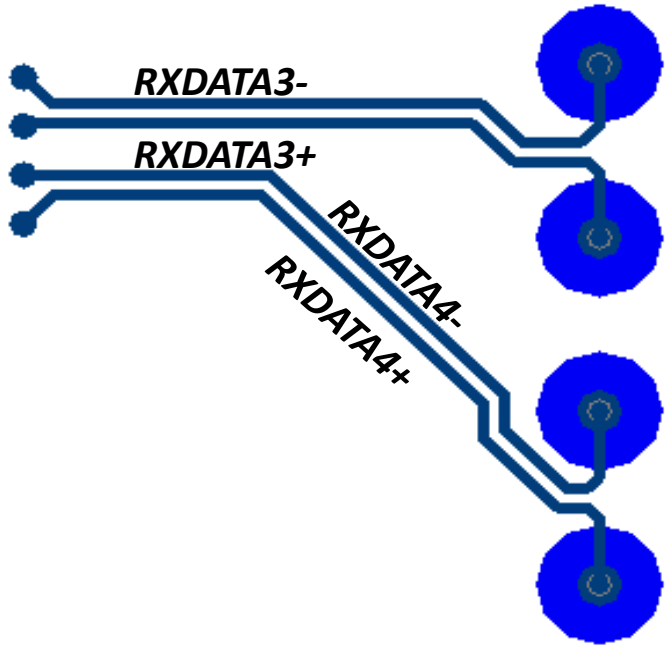


BGA Package Differential Pair Simulation with HFSS 3D Layout

From a BGA Package CAD geometry, this example shows you how to set up, simulate, and analyze two differential pairs:

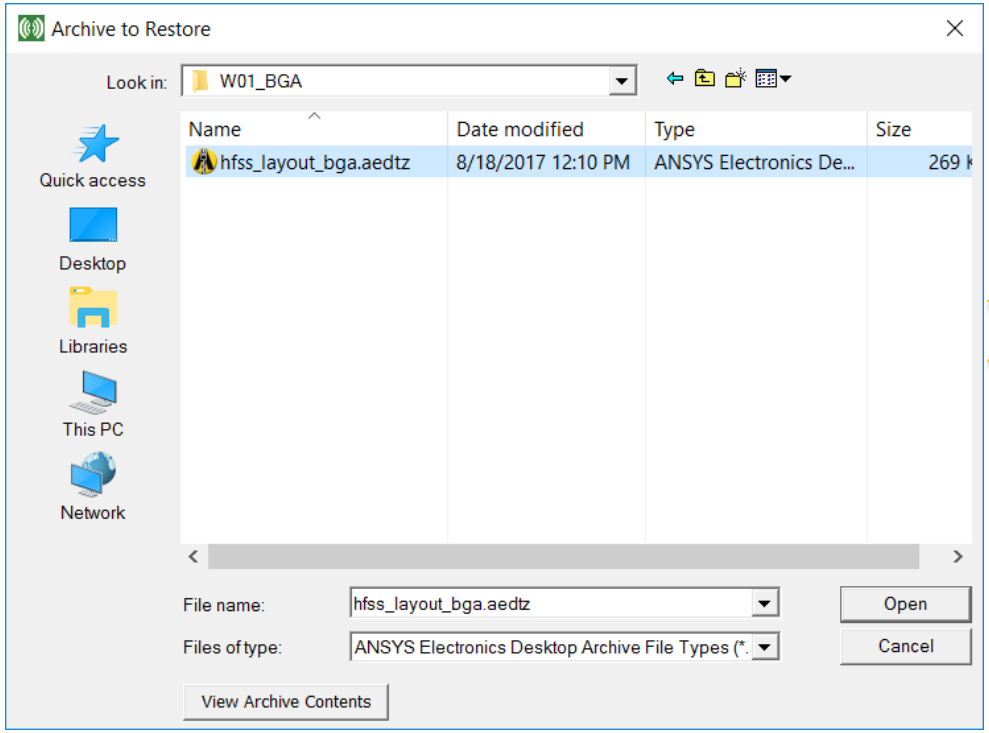
RXDATA3+, RXDATA3- *and* ***RXDATA4+,RXDATA4-***

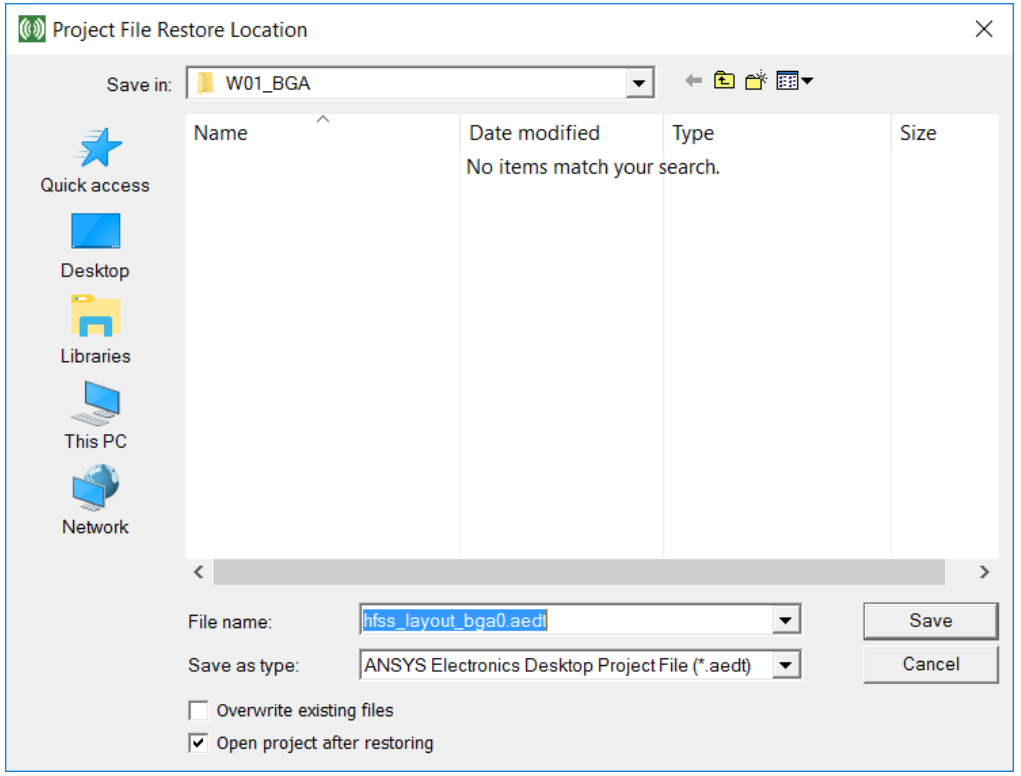




Import/Restore .aedtz Archive File with HFSS Standalone

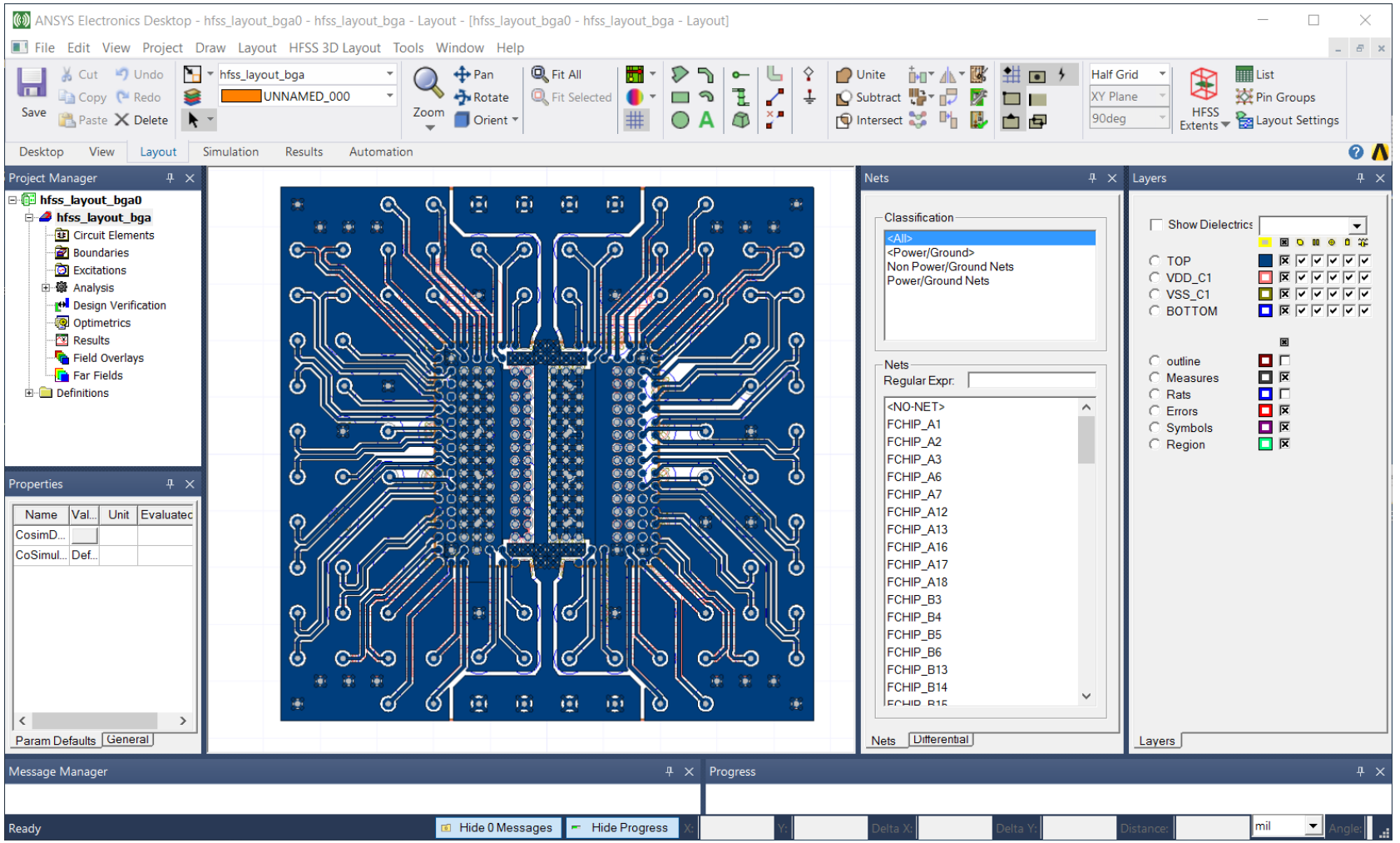
- Select the menu item ***File > Restore Archive***
  1. File name: **hfss_layout_bga.aedtz**
  2. Click the ***Open*** button
  3. ***Project File Restore Location Dialog***
     - File name: **hfss_layout_bga0.aedt**
     - ☑ Open project after restoring





- After restoring, click the ***Close*** button to close the ***Restore Archive*** pop up box.

Imported HFSS 3D Layout - hfss_layout_bga0.aedt

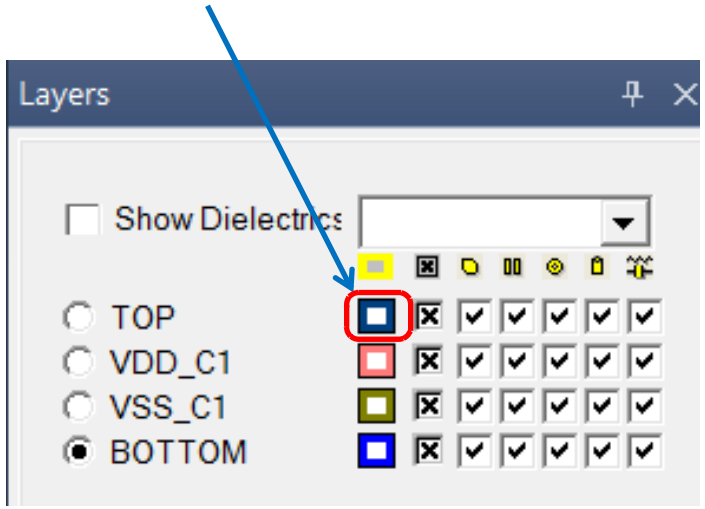


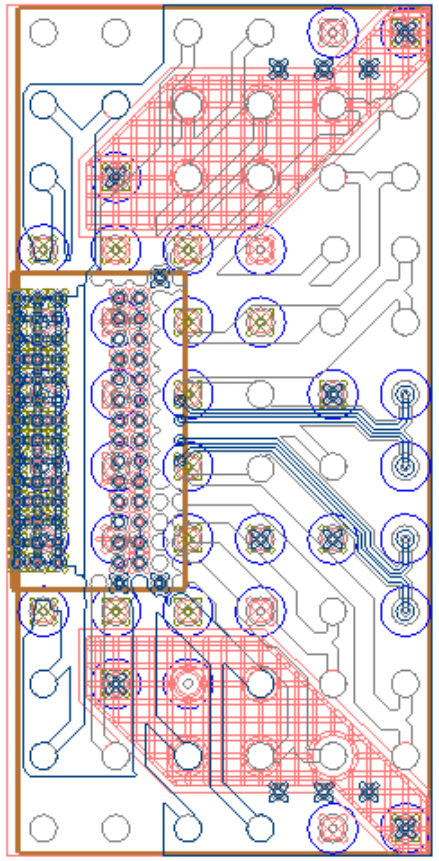
Layout

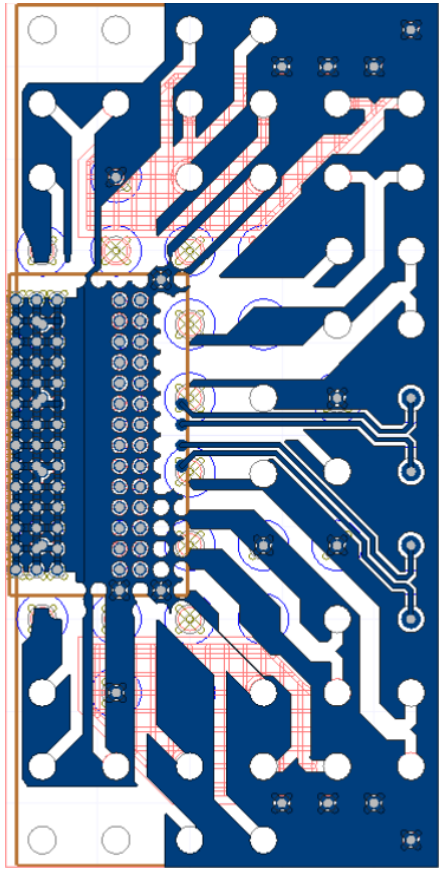
Appearance

2/27/2018 The blue, on the right hand image, comes from clicking the first "Top" box which fills in that square solid.

Toggle to change appearance







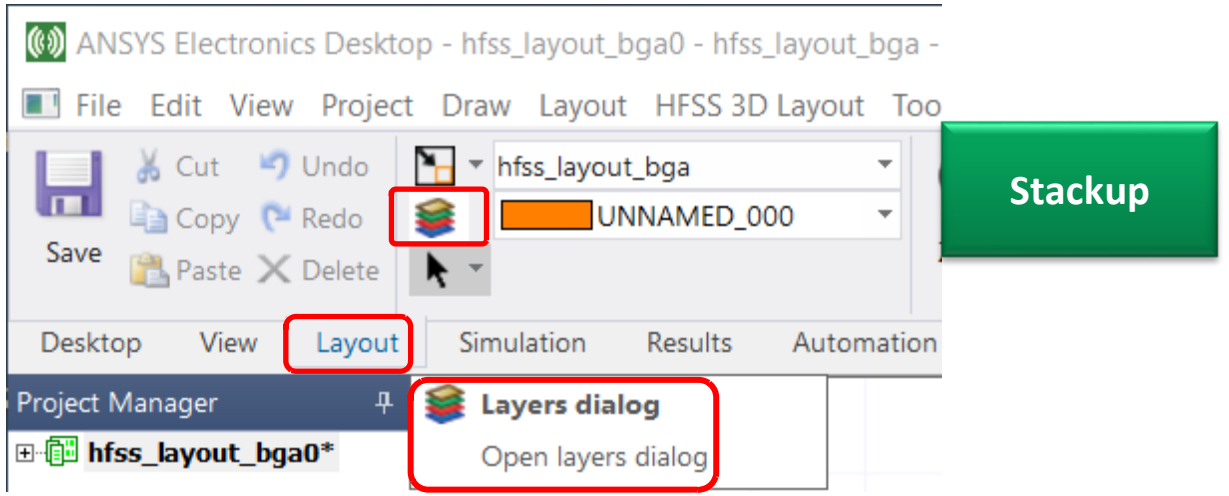
Start Editing the Stackup

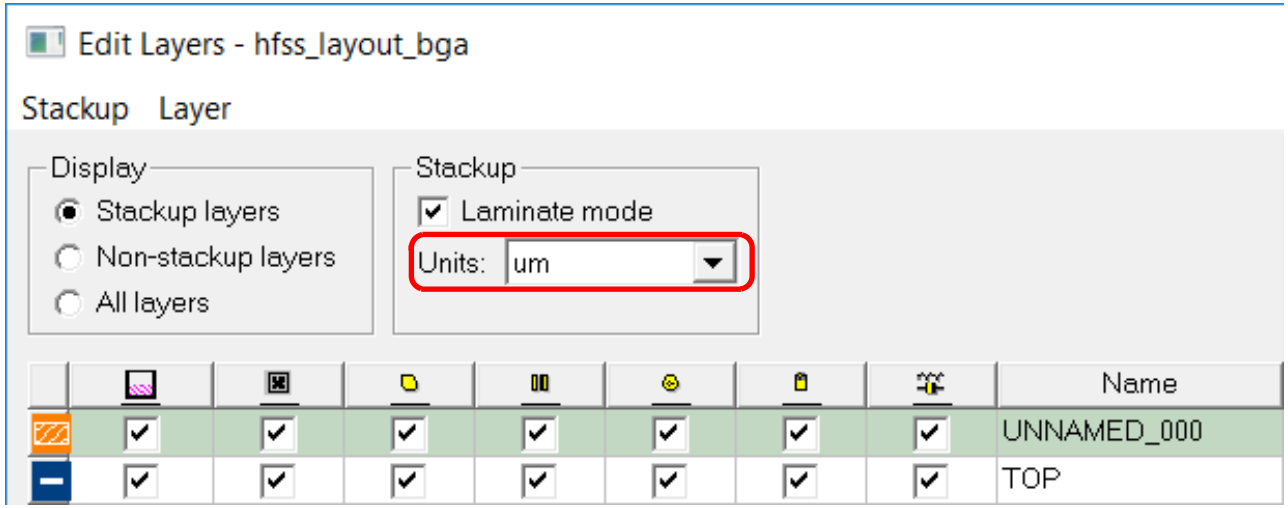
- In the ***Ribbon***, with the **Layout** tab selected, click of the ***Layers dialog*** icon to open up the ***Edit Layers*** dialog box.

This dialog box is also available from ***Layout > Layers***.

- Near the top of the ***Edit Layers*** dialog box, in the box labeled ***Stackup***, set units to microns
  - ***Units***: **um**

- By clicking on the symbols on the left hand side, rows in the stackup can be selected.

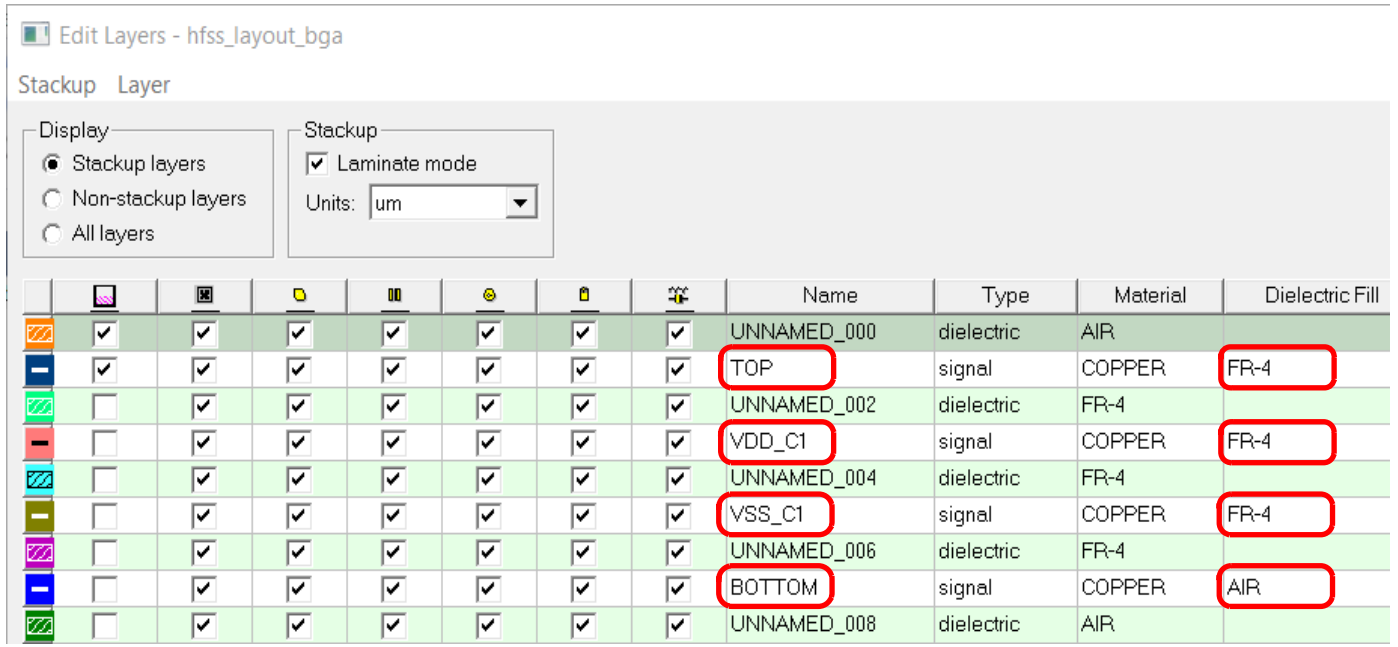




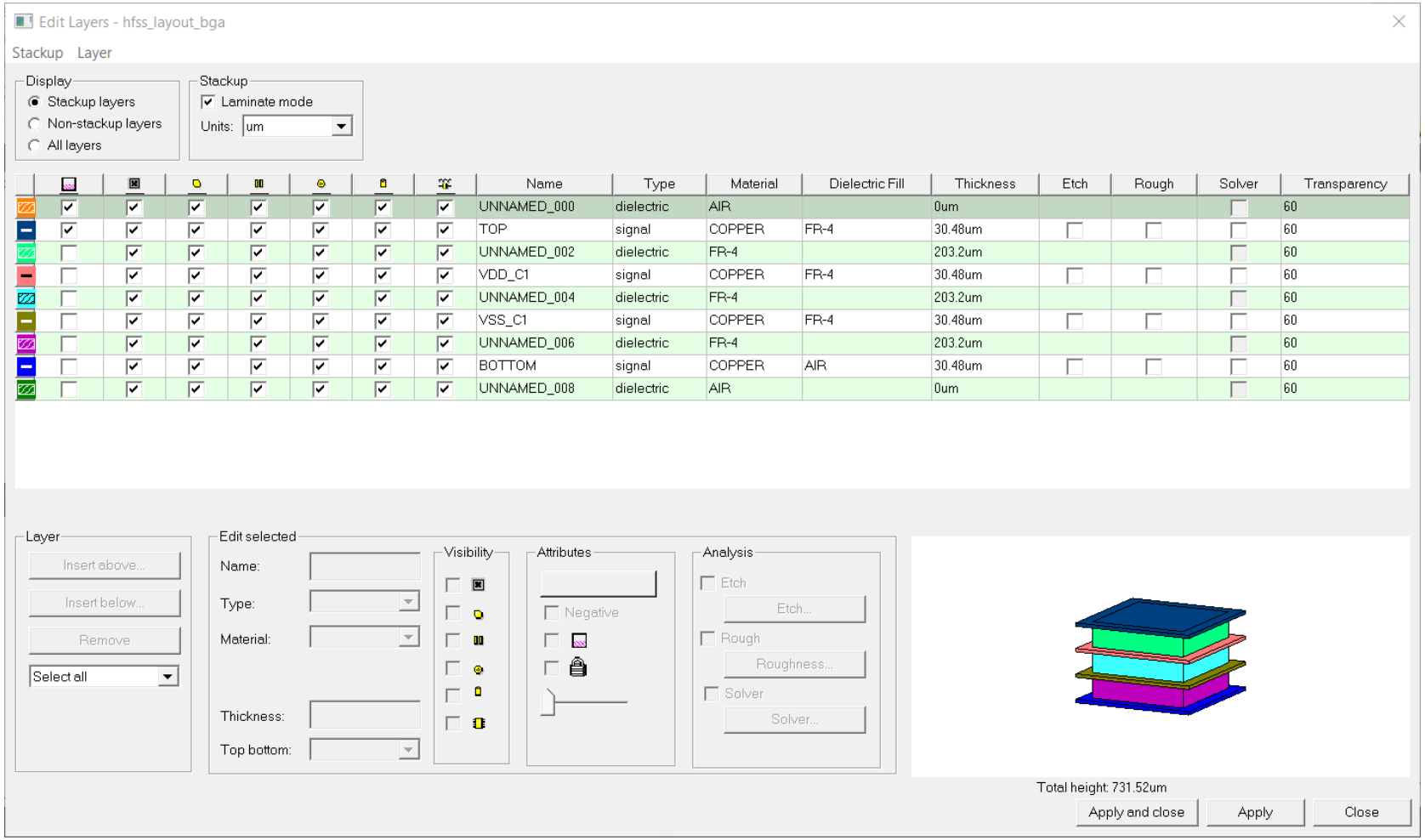
Editing Dielectric Fills in the Stackup

Stackup

- In the ***Edit Layers*** dialog box….
- Correct the ***Dielectric Fill***
  - Select the row ***Name***: **TOP**
    - In the ***Dielectric Fill*** column: from the pull-down, select **FR-4**
  - Select the row *Name:* **VDD_C1**
- ***Dielectric Fill*****:** select **FR-4**
  - Select the row ***Name*****: VSS_C1**
- ***Dielectric Fill***: select **FR-4**
  - Select the row ***Name*****: Bottom**
- ***Dielectric Fill***: select **AIR**
  - Click the ***Apply*** and ***Close*** button



Stackup - Dielectric Fill Editing Done - *hfss_layout_bga1.aedt*



Stackup

Save file as **hfss_layout_bga1.aedt**

Total height ***731.52 um***

Start Verifying Material Properties - Air

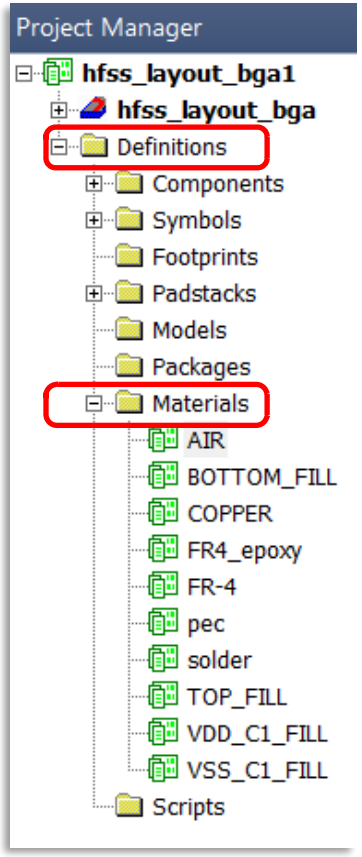
Stackup

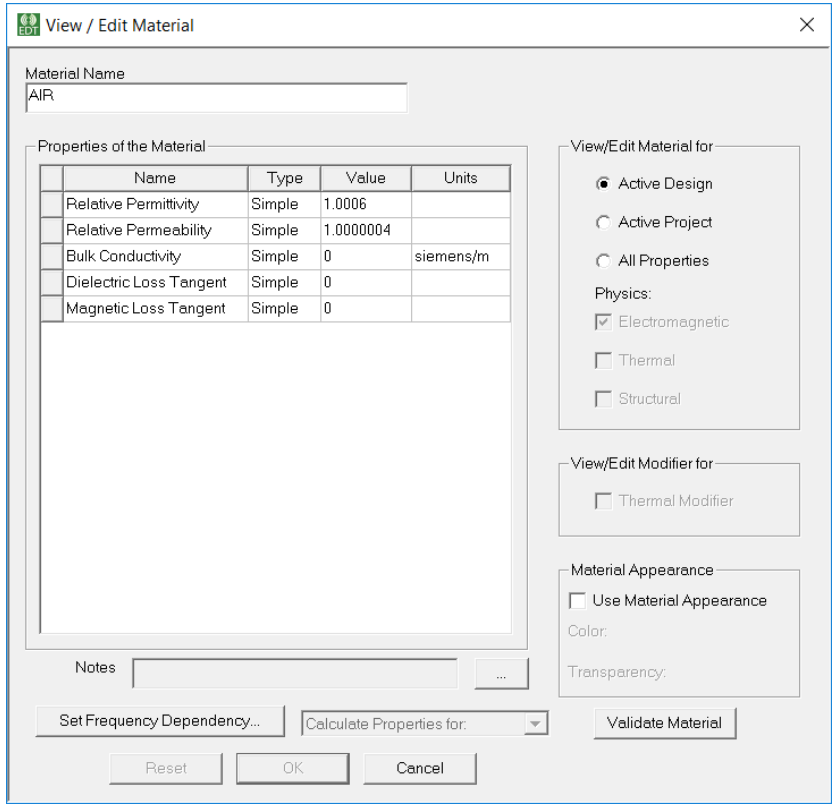
- To edit/verify the material definitions of the imported materials…
  - In the ***Project Manager***, expand the ***Definitions*** folder and then the ***Materials*** folder
  - Double-click on the material: **Air**
    - ***View/Edit Material*** dialog
      1. ***Material Name***: **Air**
      2. ***Relative Permittivity***: **1.0006**
      3. ***Relative Permeability***: **1.0000004**

Notice that the ***OK*** button only becomes available after something is edited.

- Click the ***OK*** button (if needed) to close the ***View/Edit Material*** dialog when finished.

…continued on next page



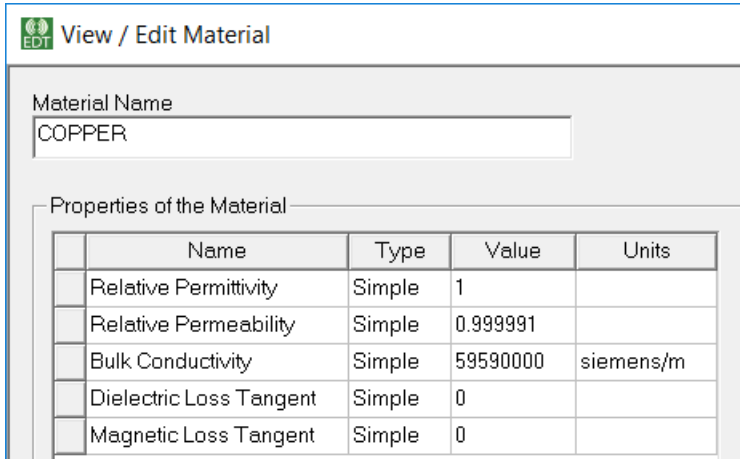


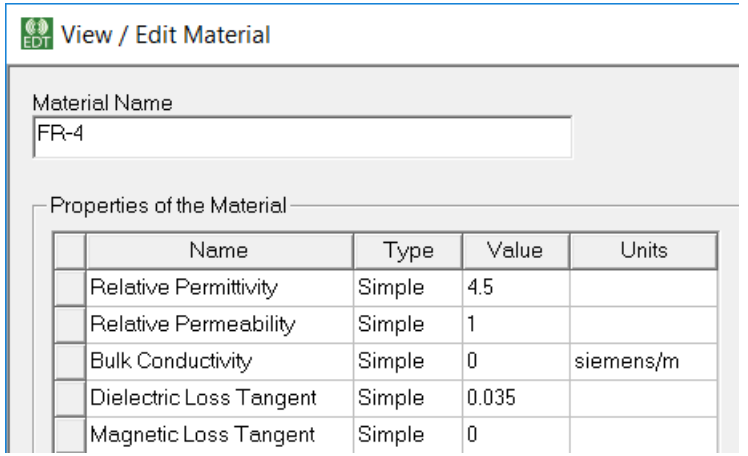
Verify Material Properties (Copper and FR-4)

…continue verifying material properties…in the ***Project Manager > Definitions folder > Materials*** folder

- Double-click on the material: **COPPER**
  - ***View/Edit Material*** dialog
  1. ***Material Name***: **COPPER**
  2. ***Relative Permittivity***: **1**
  3. ***Relative Permeability***: **0.999991**
  4. ***Conductivity***: **59590000**
  5. ***Dielectric Loss Tangent***: **0**
  6. Press the ***OK*** button

- Double-click on the material: **FR-4**
  - ***View/Edit Material*** dialog
  1. ***Material Name***: **FR-4**
  2. ***Relative Permittivity***: **4.5**
  3. ***Relative Permeability:*** **1**
  4. ***Dielectric Loss Tangent***: **0.035**
  5. Press the ***OK*** button





Stackup

Note: The ***OK*** button only becomes available after something is edited.

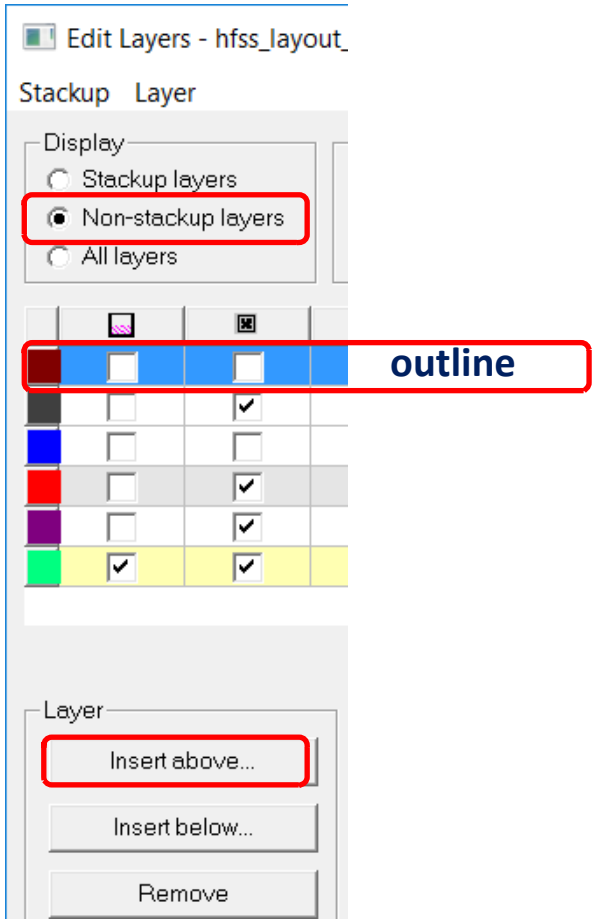
Region "User" Layer for Subdesign Cutout

Layout

In order to analyze two specific differential signal pairs, we'll cut out a portion of the overall layout into a subdesign that contains only the two desired differential pairs and the reference net(s). Net names for the differential pair include RXDATA3+, RXDATA3-, RXDATA4+, RXDATA4-.

To facilitate cutting out the subdesign, we'll draw a rectangle on a "user" layer, an extra utility layer not used for geometry in the PCB design. The user layer name in this design is ***Region***.

If there is not yet present in the layers a ***Region*** layer, follow the instructions below to create one.

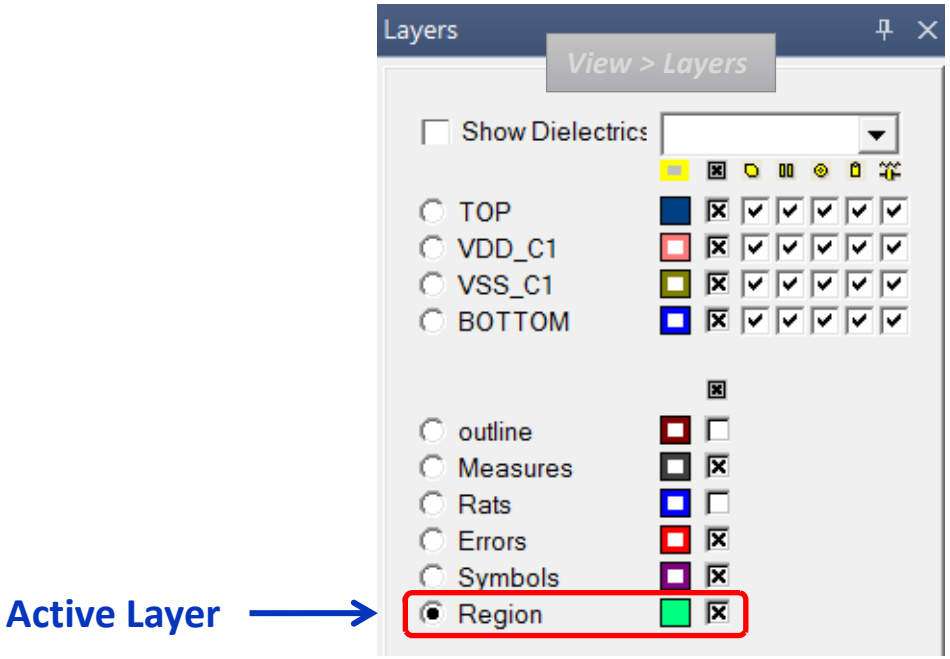


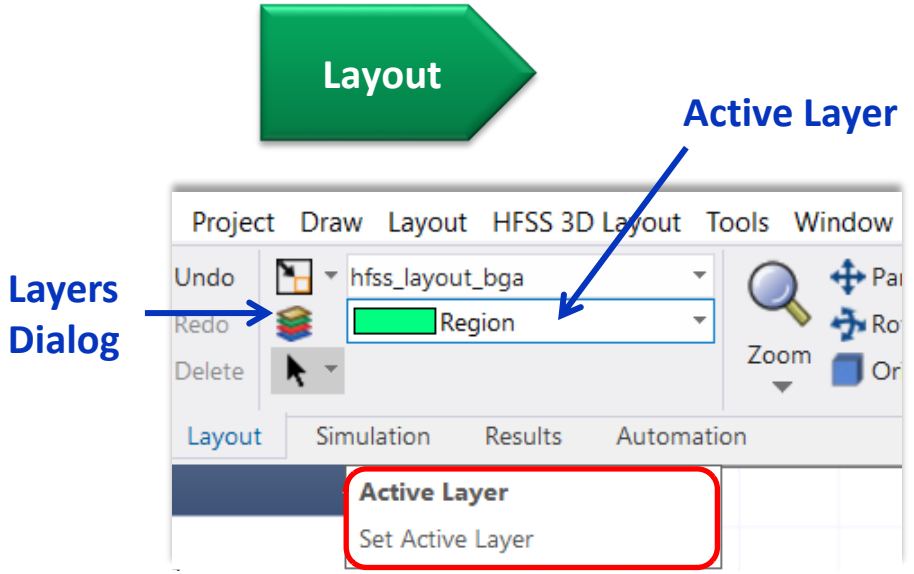
- If the ***Region*** layer is present, skip forward and make *Region* the active layer.
- To create a ***Region*** User Layer…(if the ***Region*** layer is **NOT** yet present)
  - Select the menu item ***Layout > Layers*** to bring up the Edit Layers dialog box.
    1. In the upper left ***Display*** area: ***Non-stackup layers.***
    2. Click on the (top?) row for the layer with name ***outline.***
    3. In the lower left ***Layer*** area, click ***Insert above...*** to open the ***Add Layer*** dialog.
       - ***Name*****: Region**
       - ***Type*****: User**
       - Press the ***OK*** button
    4. Press the ***Apply*** and ***Close*** button to close the Edit Layers dialog box.

Set Active Layer - *Region*

There are two easy ways to set the active drawing layer in HFSS 3D Layout.

- In the Ribbon, with the **Layout** tab chosen, click on the down-triangle next to layer names to choose a different active layer.
- In the Layers window, simply click on the radio button (round space) next the layer name to make it active.

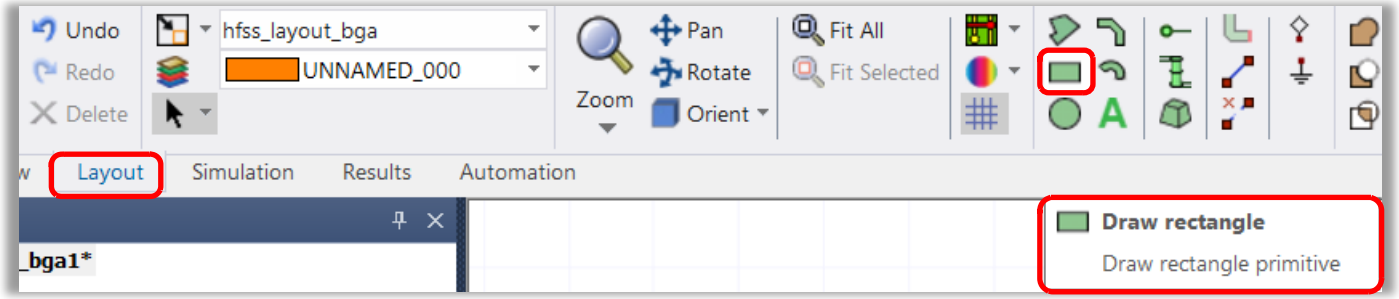




Draw Rectangle on Region Layer for Subdesign Cutout

In order to create a rectangle for the cut-out…

- In the Ribbon, with the **Layout** tab highlighted, select the ***Draw rectangle*** icon.



- Another way to access drawing a rectangle primitive is from the top pull downs ***Draw > Primitive > Rectangle.***

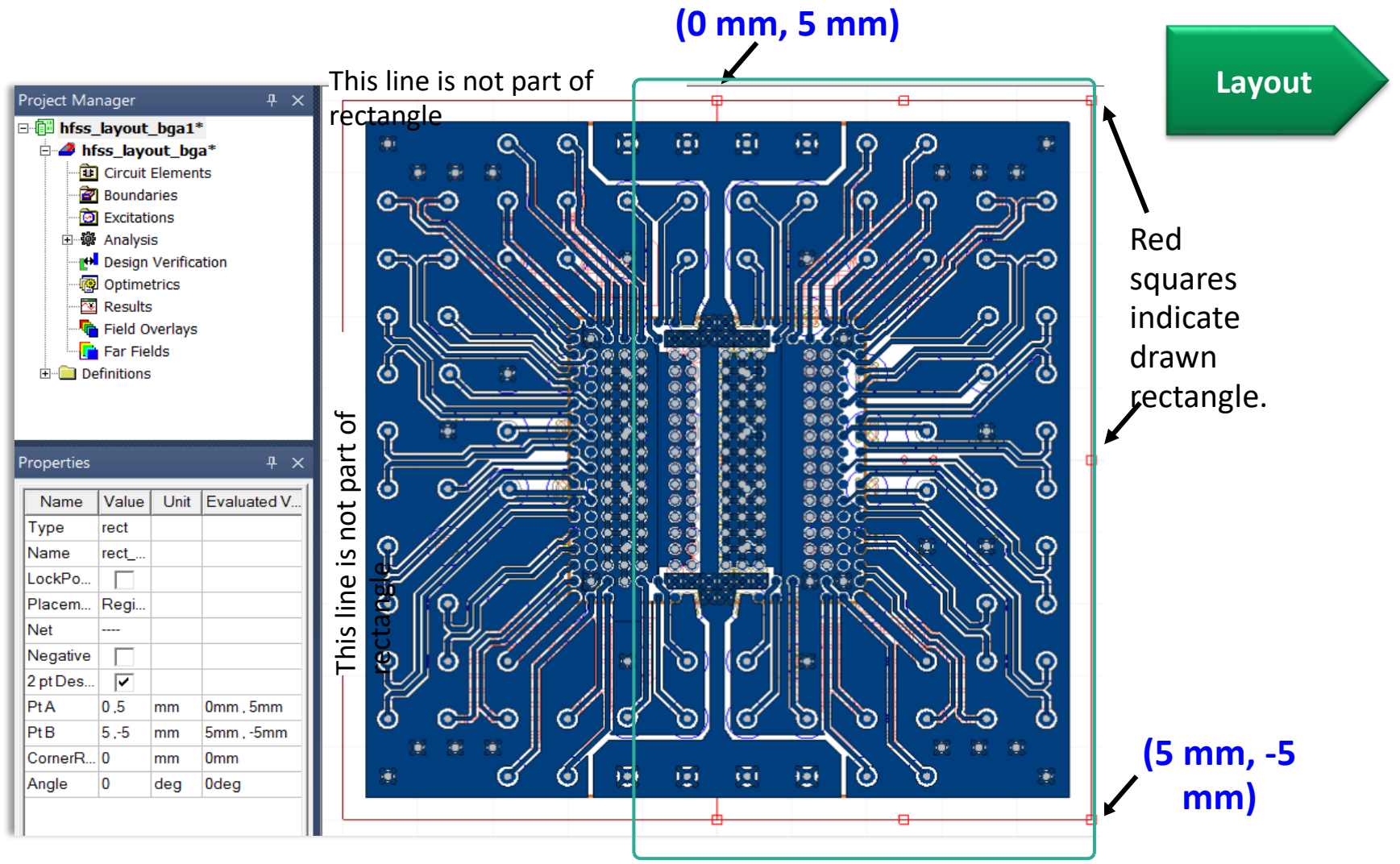
  - The first click signifies first vertex of the rectangle. The second click is for the diagonally opposite vertex.

…continued…

Finished Rectangle for Subdesign Cutout

- Move your mouse to visualize the rectangle.
- Draw the rectangle to encapsulate the right-hand half of the package model.

The coordinates are accessible in the ***Properties*** menu with the rectangle highlighted. If the initial freehand draw doesn't work, the two points
Pt A: 0, 5 mm and
Pt B: 5, -5 mm
can be entered in ***Properties***.



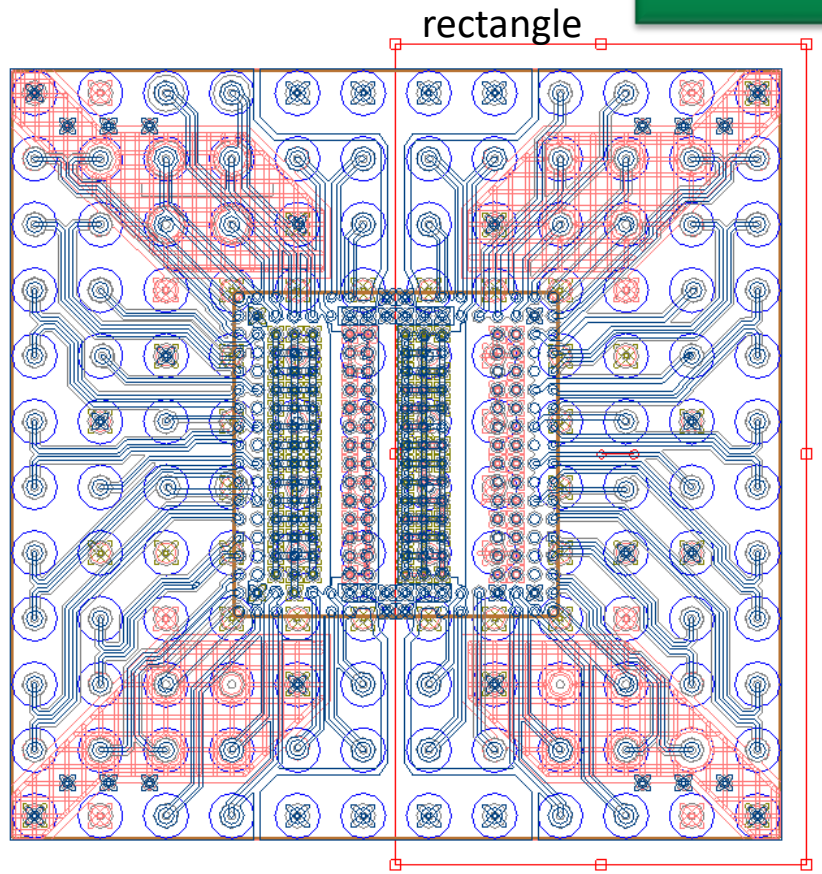
Alternative Way to Draw Rectangle

Layout

- Select the menu item ***Draw > Primitive > Rectangle***
  - Using the coordinate entry at the bottom of the windows, enter the following:
    1. Units: mm
    2. X: 0 Press the ***Tab*** key
    3. Y: 5 Press the ***Enter*** key

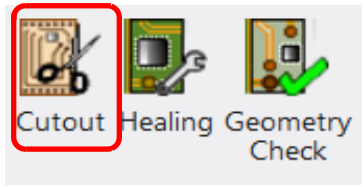
    4. Press ***Tab*** key twice
    5. Delta X: 5 Press the ***Tab*** key
    6. Delta Y: -10

    7. Press the ***Enter*** key

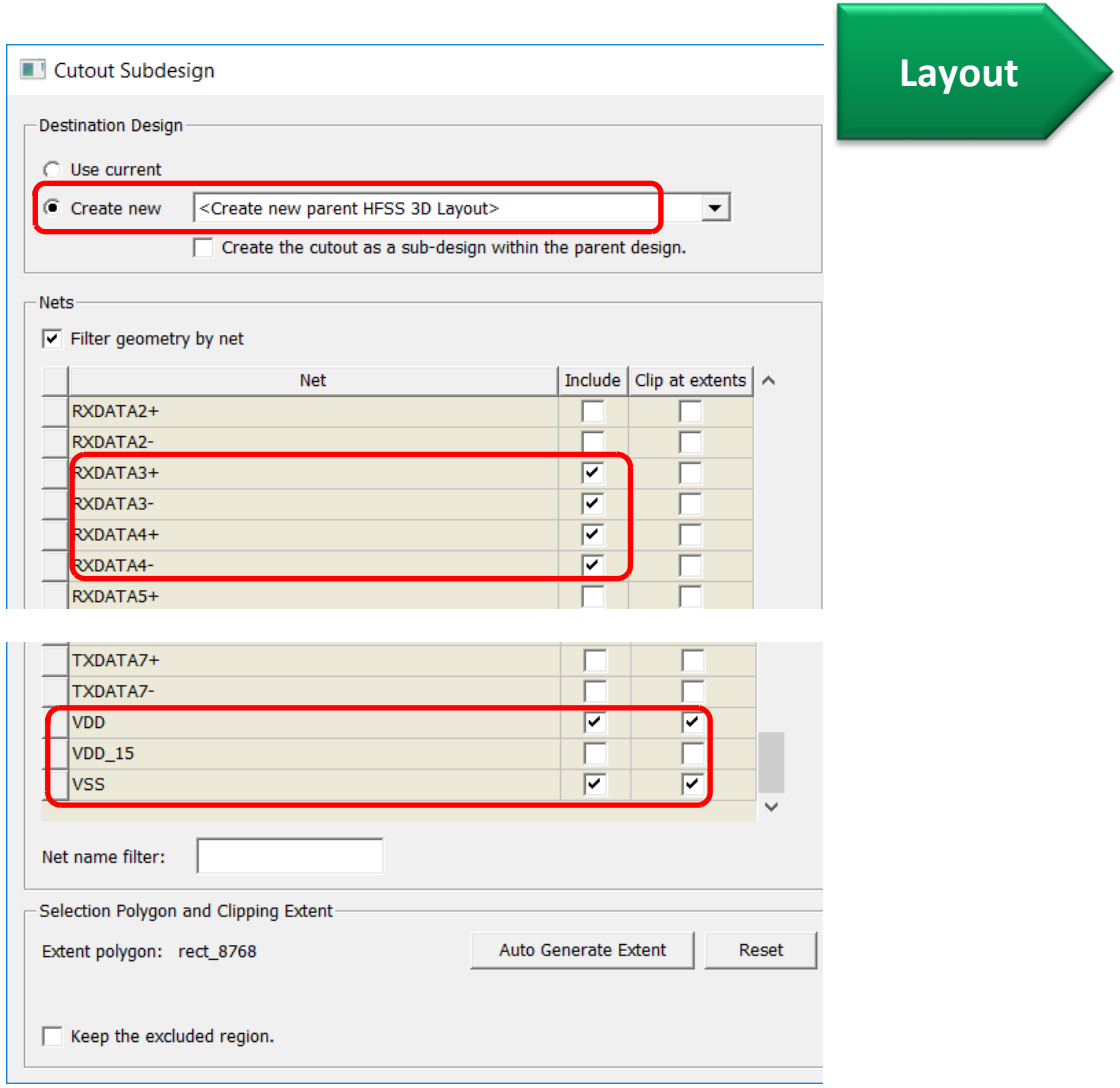


Cutout Subdesign

In order to cut out the subdesign….



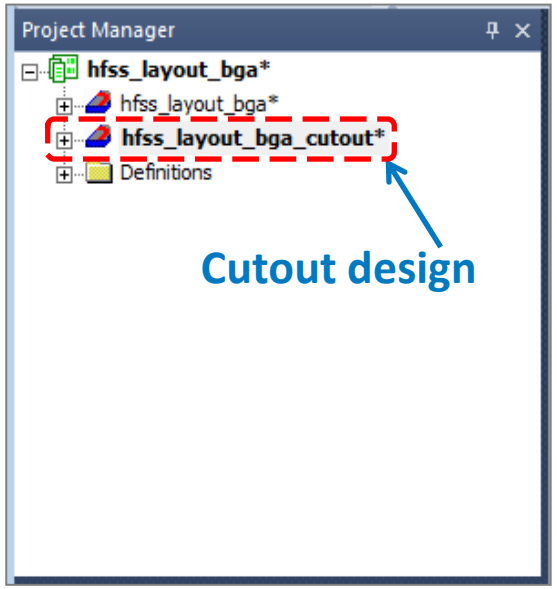
- ***With the rectangle selected***, in the Ribbon, with the **Layout** tab selected, select ***Cutout***, which brings up the ***Cutout Subdesign*** dialog box.
- This cutout function can also be accessed with the menu item ***Layout > Cutout Subdesign***
  - Check the box for ***Create new*** **<Create new parent HFSS 3D Layout>**
  - Place check in the *Include* column for the following nets:
    - **RXDATA3+, RXDATA3-, RXDATA4+, RXDATA4-, VDD, VSS**
  - Place check in the Clip at extents column for the following nets:
    - **VDD, VSS**
  - Click the ***OK*** button *to close the* ***Cutout Subdesign*** *dialog box.*

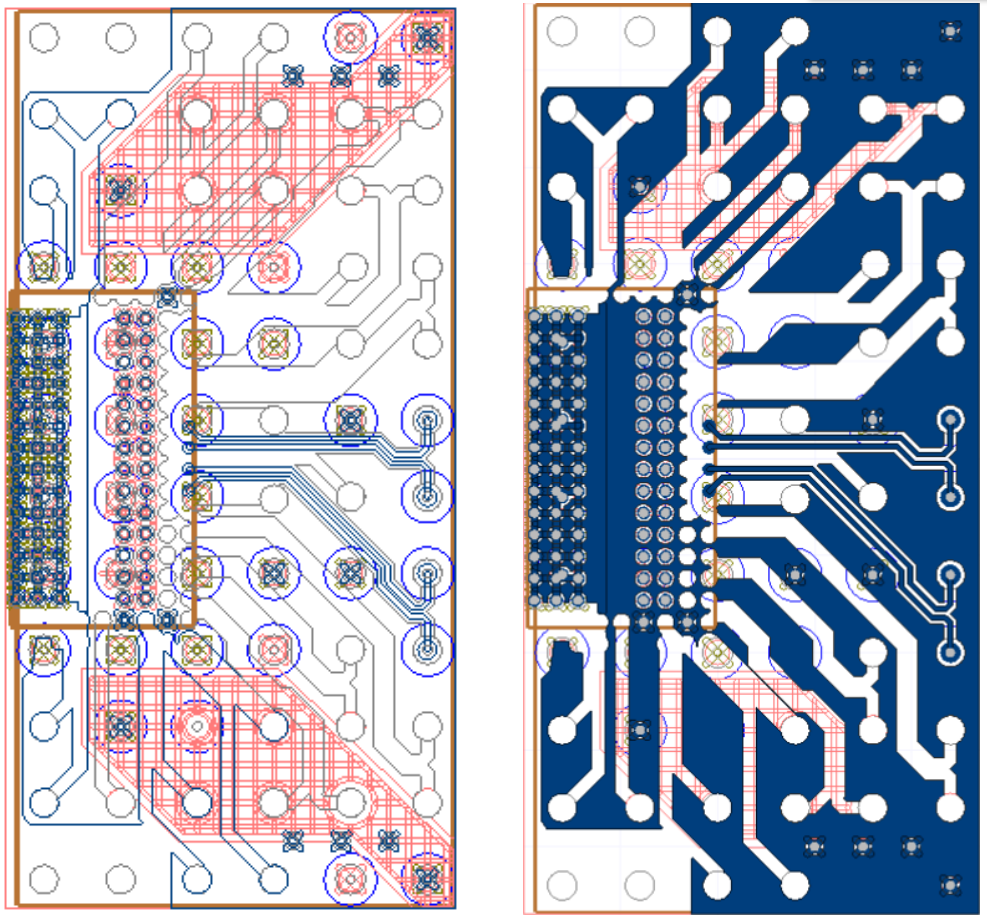


Cutout Subdesign

Layout

- A new design will be added to the project and should become the active design.
- Below is a picture showing the cutout
- Save file as ***hfss_layout_bga2.aedt***

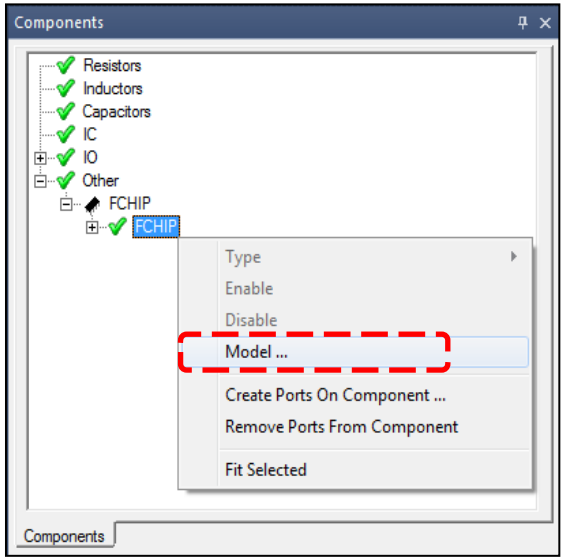


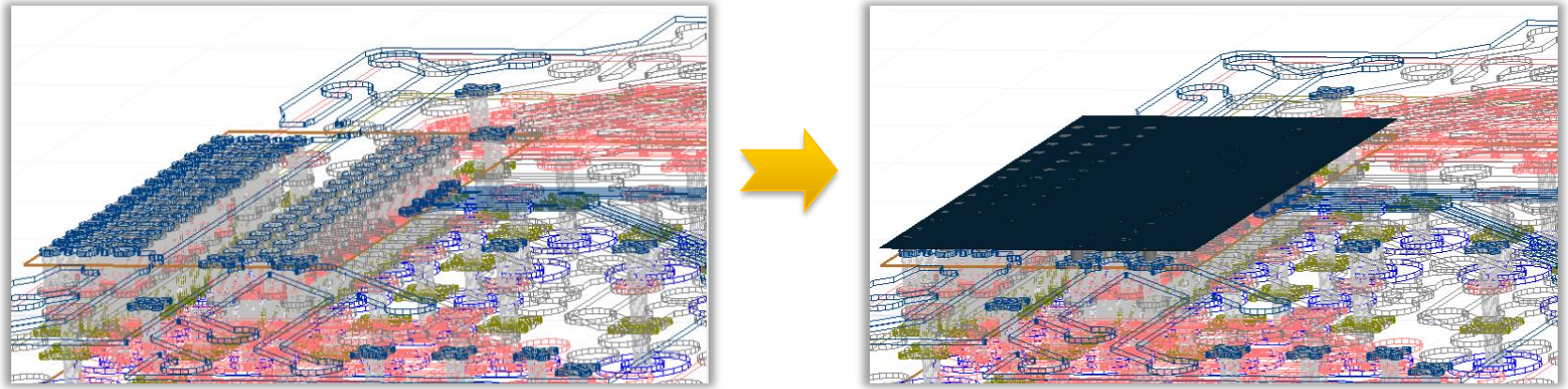


Add BGA Solder Balls/Flip Chip Bumps to FCHIP on Subdesign

- For the cutout subdesign "hfss_layout_bga_cutout", access the ***Components*** window…
  The ***Components*** window is available from ***View > Components***
- From the ***Components*** dialog window
  - Expand ***Other*** (by clicking on the + sign), expand ***FCHIP***, click on ***FCHIP*** component and Right-click
- Click on ***Model***..

…continued…



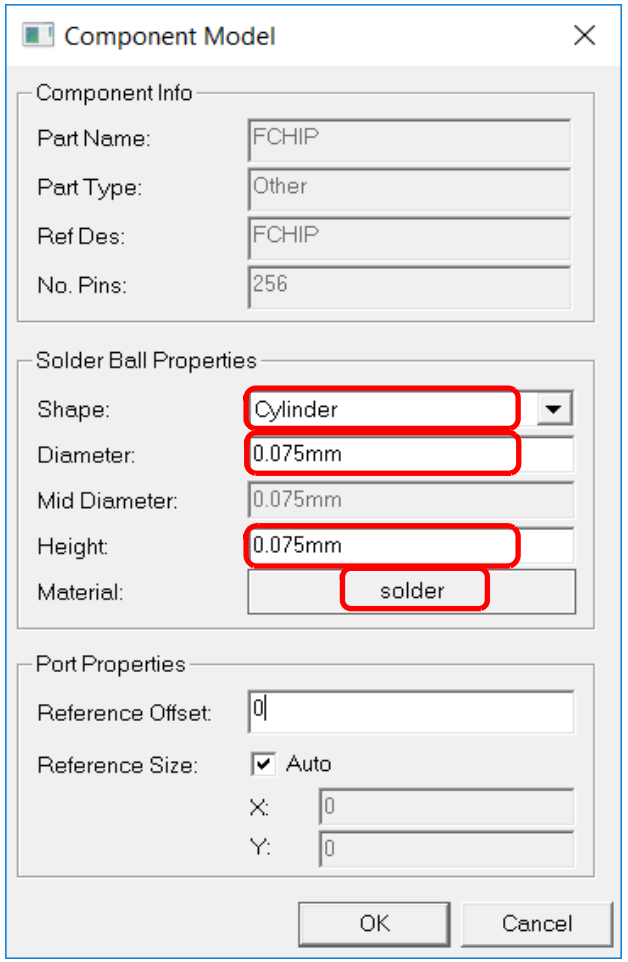


To create Flipchip bumps for the component, we define the Solder Ball properties to the FCHIP component. A reference plane for the Flipchip bumps is automatically added to the geometry.

Add BGA Solder Balls/Flip Chip Bumps to FCHIP on Subdesign

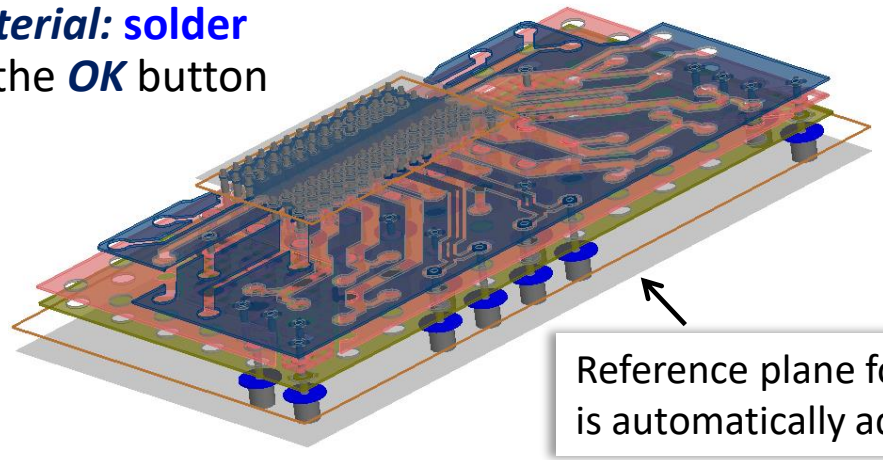
In the ***Component Model*** dialog box…

- ***Solder Ball Properties***
  - ***Shape*: Cylinder**
  - ***Diameter*: 0.075mm**
  - ***Mid Diameter*: 0.075mm**
  - ***Height*: 0.075mm**
  - ***Material*: solder**
- Click the ***OK*** button

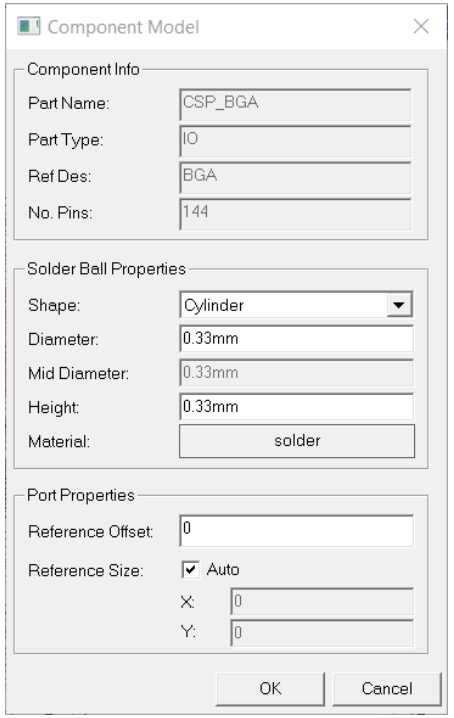


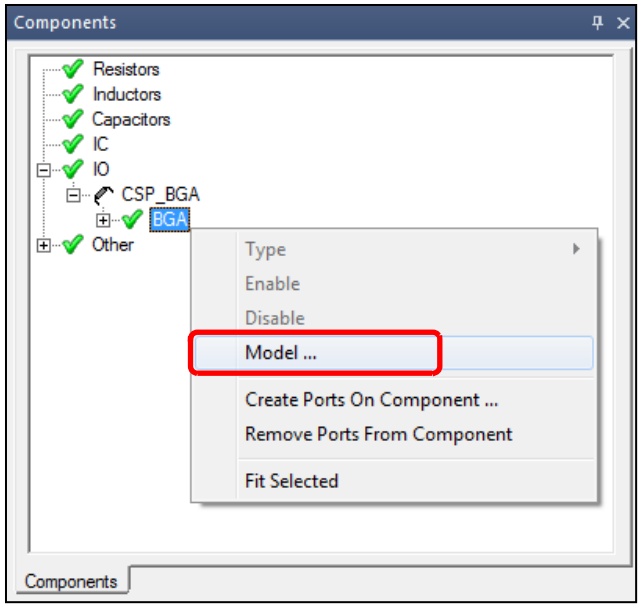
Add BGA Solder Balls/Flip Chip Bumps to CSP_BGA

- Adding BGA Solder Balls/Flipchip Bumps…
  - From the ***Component*s** dialog window
    - Expand ***IO***, expand ***CSP_BGA***, click on ***BGA*** component and right-click
  - Click on *Model …*bringing up ***Component Model*** dialog box
    1. ***Solder Ball Properties***
       - ***Shape:* Cylinder**
       - ***Diameter:* 0.33mm**
       - ***Height:* 0.33mm**
       - ***Material:* solder**
    2. Click the ***OK*** button



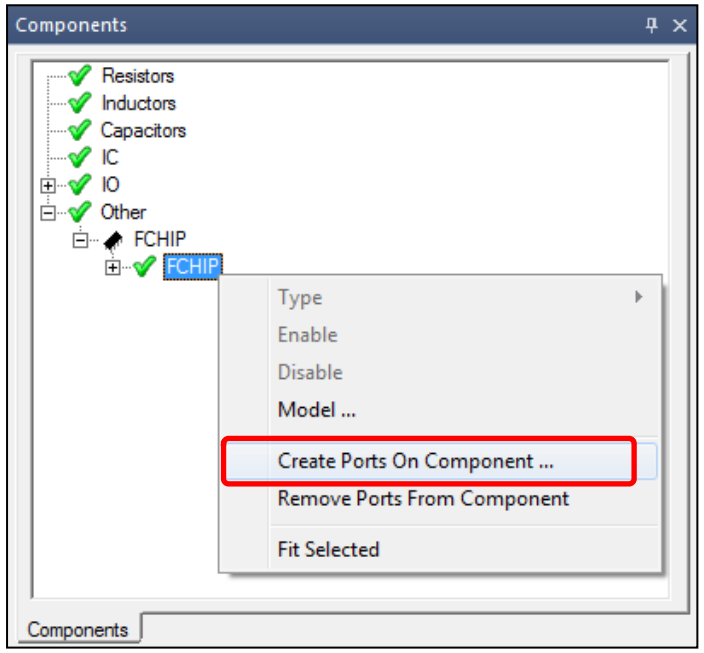
Reference plane for the BGA Solder Balls is automatically added to the geometry.

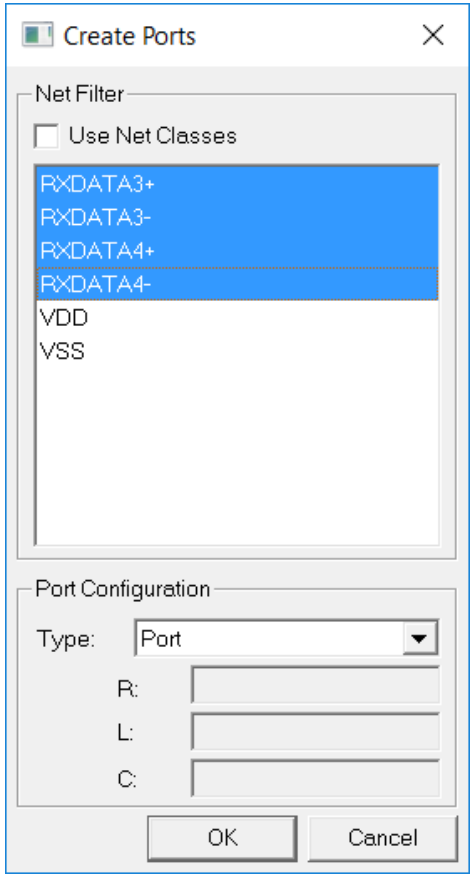


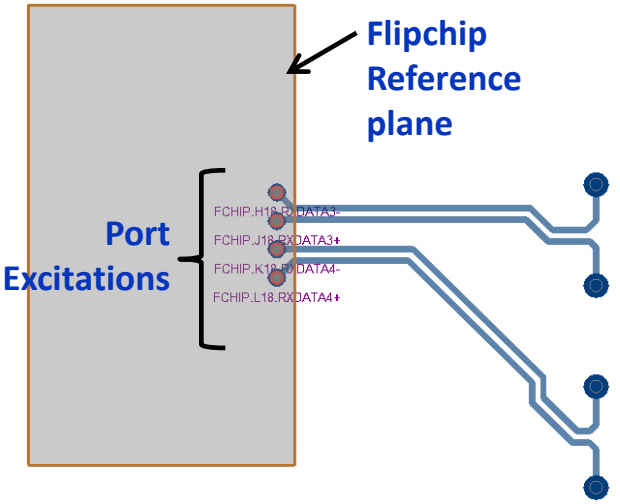


Create Ports on Diff Pairs at Flip Chip Reference Plane

- From the ***Components*** dialog window
  - Expand ***Other***, expand ***FCHIP***, click on ***FCHIP*** component and Right-click
  - Click on ***Create Ports on Component...***
    1. Select the nets: **RXDATA3+, RXDATA3-, RXDATA4+, RXDATA4-**
    2. ***Port Configuration***
       - Type: **Port**
    3. Click ***OK***

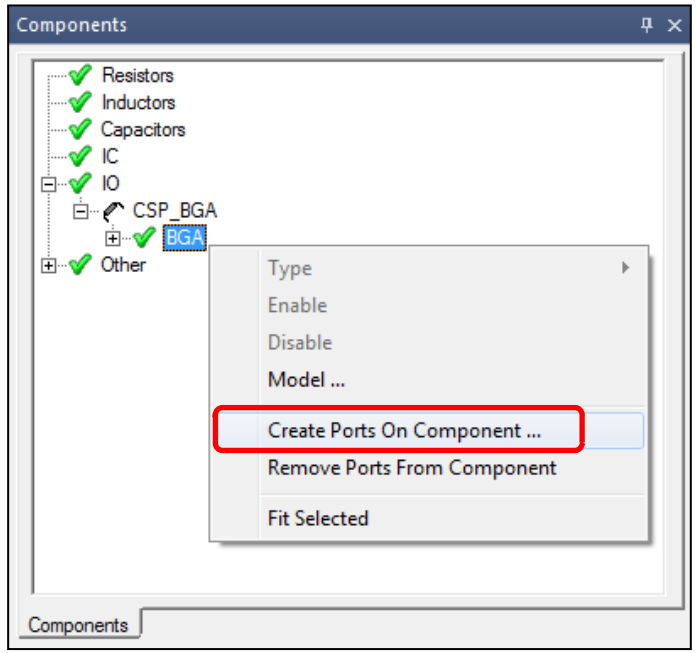


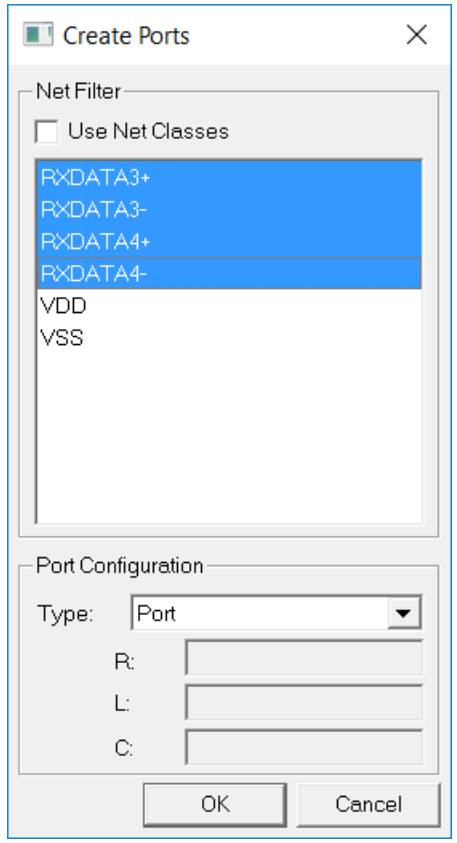


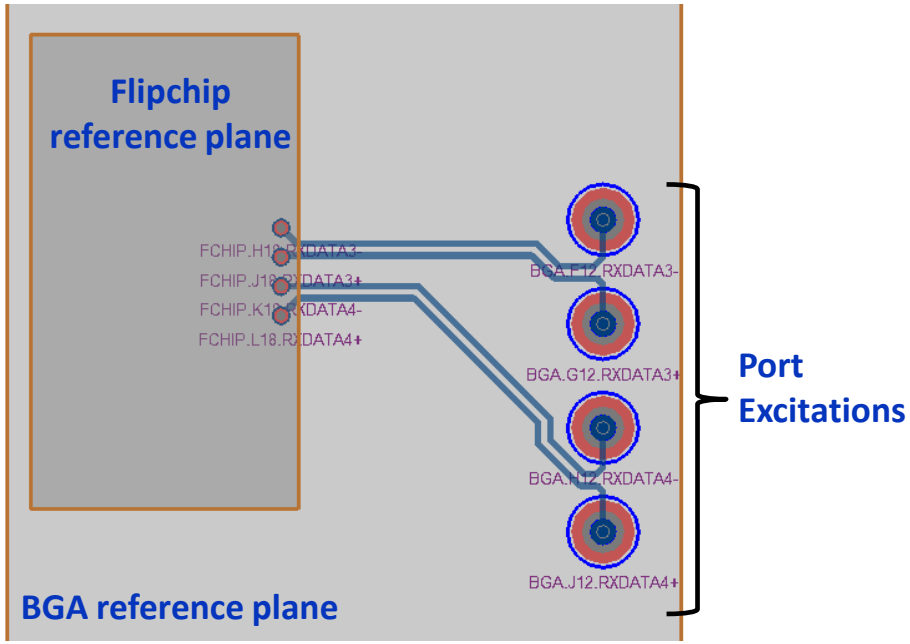


Create Ports on Diff Pair at BGA Reference Plane

- From the ***Components*** dialog window
  - Expand ***IO***, expand ***CSP_BGA***, click on ***BGA*** component and right-click
  - Click on ***Create Ports on Component...***
    1. Select the nets: **RXDATA3+, RXDATA3-, RXDATA4+, RXDATA4-**
    2. ***Port Configuration***
       - Type: **Port**
    3. Click ***OK***

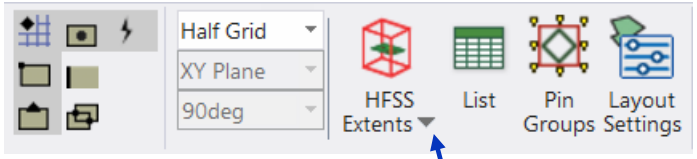






Define HFSS Extents

Using HFSS Extents, we define the computation region for HFSS. Dielectric ***Horizontal Padding*** and the dimensions of ***Airbox*** are defined. The values can be entered as absolute values (10mm) or as percentage of the maximum of either length or width of the dielectric-polygon.

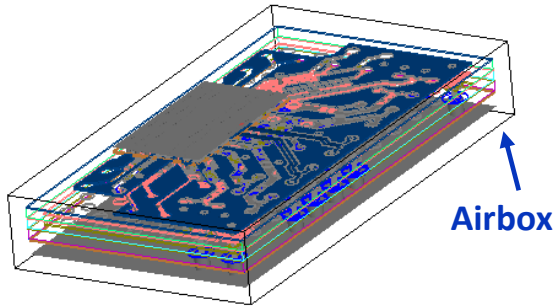


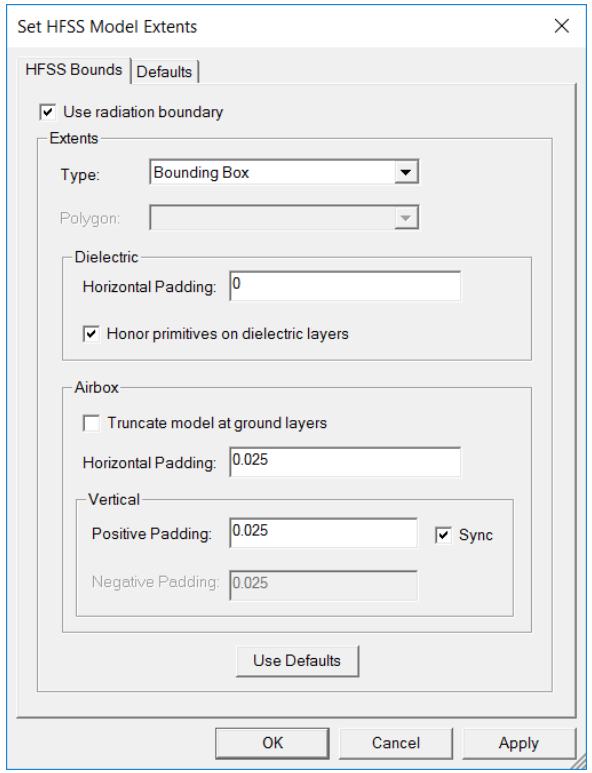
- In the Ribbon, with the **Layout** tab highlighted, click on the triangle by HFSS Extents, and select Edit…

  Another way to access the ***Set HFSS Model Extents*** dialog box is to Select the menu item ***HFSS 3D Layout > HFSS Extents.***

  - ***Dielectric***
    1. Horizontal Padding: **0**
  - ***Airbox***
    1. Horizontal Padding: **0.025**
    2. Vertical Padding: **0.025**
    3. Sync: ☑
  - Click the ***OK*** button

Draw HFSS Air Box





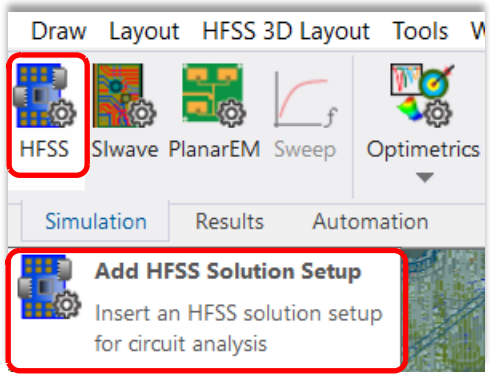
- To visualize the HFSS extents:
  - Select the menu item ***Layout > Draw HFSS Air Box***
  - Select the menu item ***View > Rotate***

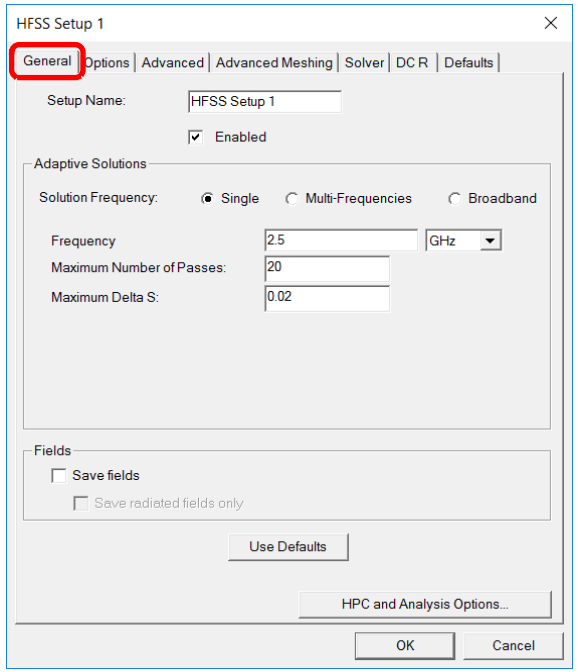
Solution Setup

- Creating an HFSS Solution Setup
  - In the Ribbon, with the **Simulation** tab highlighted, click on HFSS to bring up the HFSS Setup dialog box.

    Another way to access ***HFSS Solution Setup*** is select the menu item ***HFSS 3D Layout > Solution Setup > Add HFSS Solution Setup***

    1. Click the ***General*** tab:
       - ***Setup Name***: **HFSS Setup1**
       - ***Solution Frequency***: **Single**
       - ***Frequency***: **2.5 GHz**
       - ***Maximum Number of Passes***: **20**
       - ***Maximum Delta S***: **0.02**
       - ☐ Save fields
    2. Click the ***OK*** button
- The frequency sweep setup dialog will automatically appear…*continued*



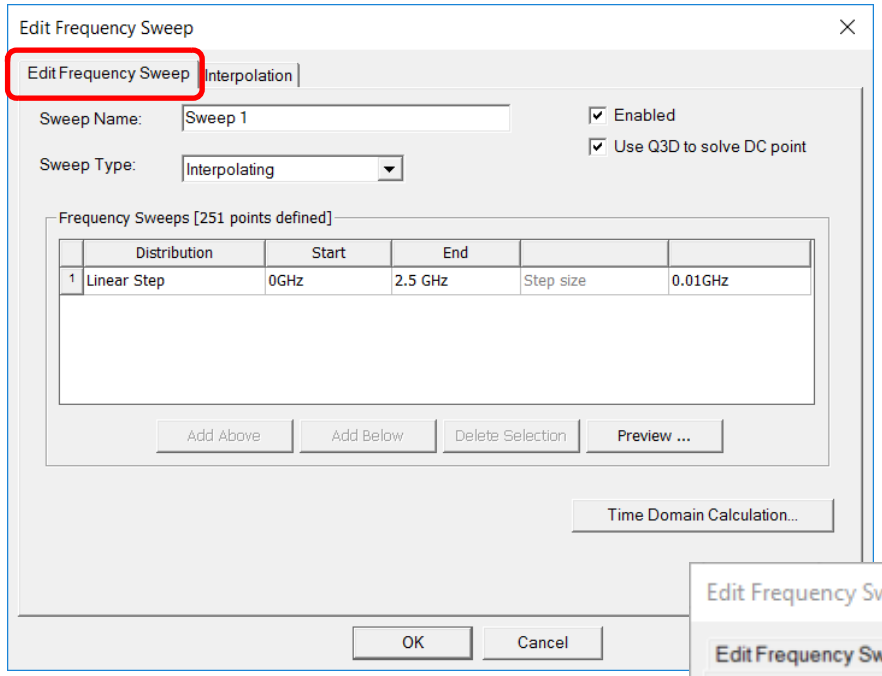


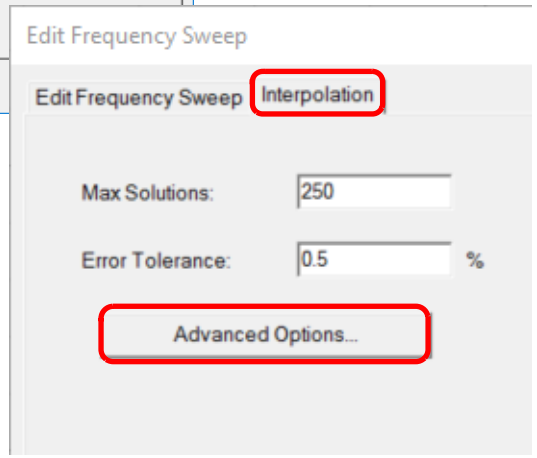
Edit Frequency Sweep

In the ***Edit Frequency Sweep*** dialog box…

Edit Frequency Sweep tab…

- ***Name*: Sweep 1**
- **☑ Use Q3D to solve DC point**
- ***Sweep Type*: Interpolating**
  - ***Type*: Linear Step**
  - ***Start*: 0 GHz**
  - ***End*: 2.5 GHz**
  - ***Step*: 0.01 GHz**





Solve Setup

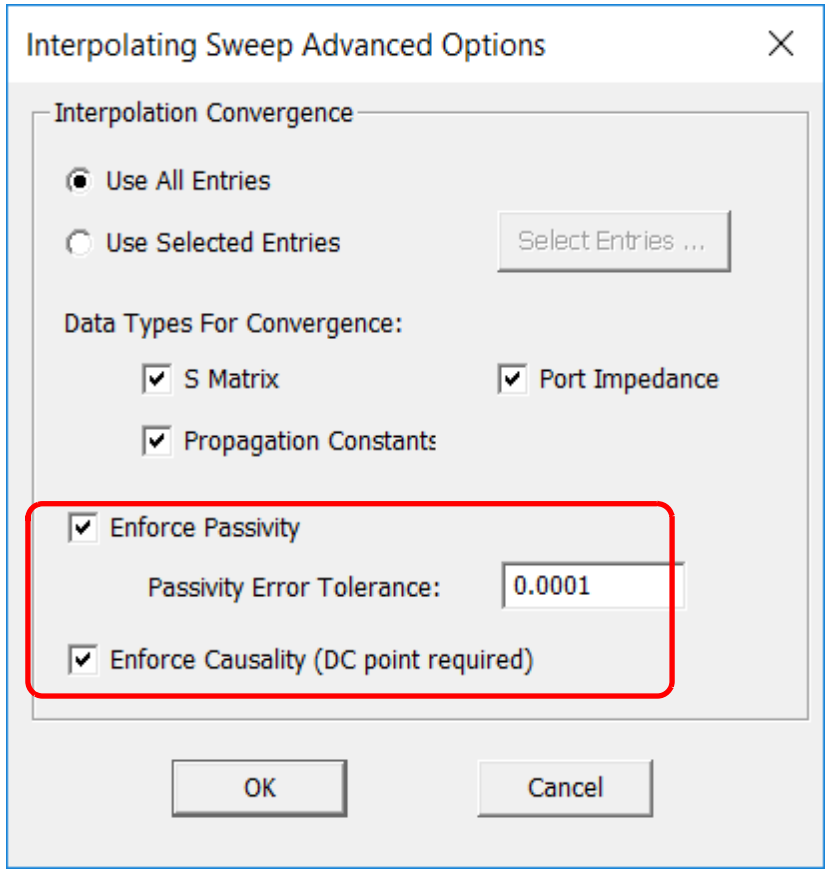
- Click on the Interpolation tab…
  - ***Relative error*: 0.5%**
- Click on ***Advanced Options*** button to open up the ***Interpolating Sweep Advanced Options*** dialog box…continued

Frequency Sweep - Interpolating Sweep Advanced Options

Solve Setup

In the ***Interpolating Sweep Advanced Options*** dialog box…

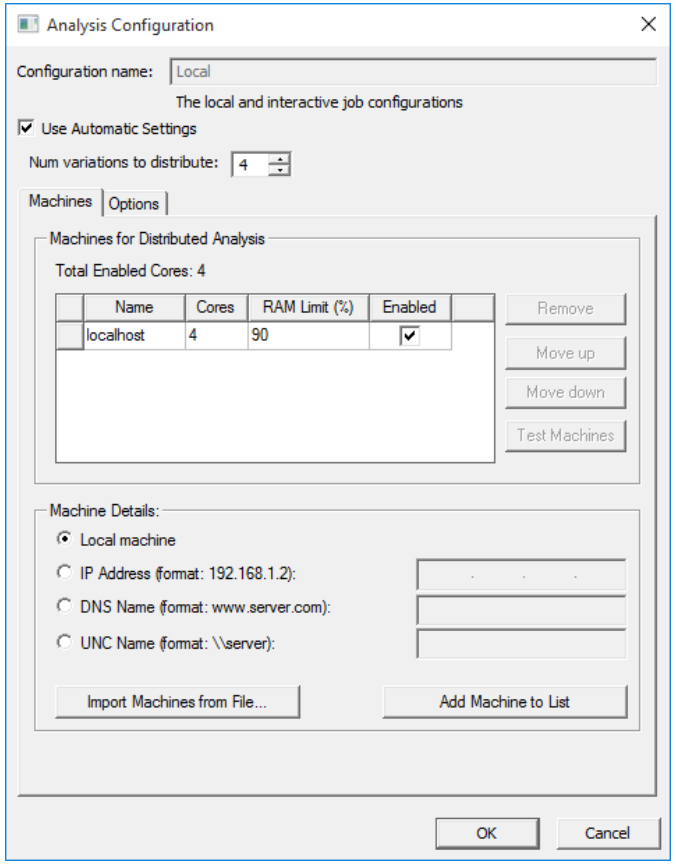
- ☑ ***Enforce causality*** (DC point required)
- ☑ ***Enforce passivity***
- Press the *OK* button to close ***Interpolating Sweep Advance Options*** dialog box.
- Press the ***OK*** button to close ***Edit Frequency Sweep*** dialog box.



HPC Setup

Setting up High Performance Computing (HPC)…

- Select the menu item ***Tools > Options > HPC and Analysis Options***
  - Design Type: **HFSS 3D Layout Design**
  - Select the ***Active Configuration***: **Local**
  - Click the ***Edit*** button
  - Analysis Configuration Dialog
    1. Check the option for Use Automatic Settings
    2. Click the Machines tab
       - ***Name: localhost***
       - ***Cores: 4***
       - ***RAM Limit (%): 90***
       - ***Click the OK button***
    3. Click the ***OK*** button



Note: The ANSYS training computers are typically configured with a single quad core processor. The configuration shown here will use 4 cores for multi-threading.

Save, Validate, and Analyze HFSS Project - *hfss_layout_bga3.aedt*

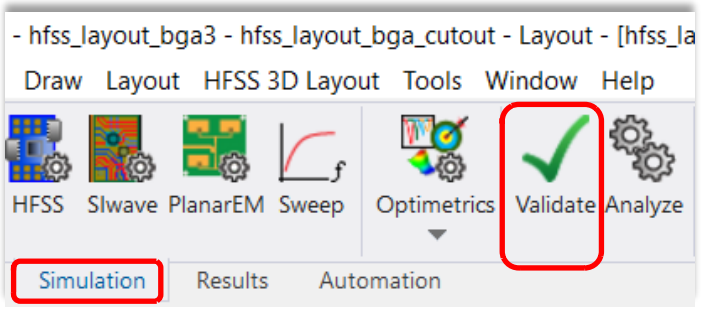
- Click the ***Validate*** green check mark in the ribbon (with **Simulation** chosen) to validate the project.
- Select ***File > Save As*** and save project renaming file as: ***hfss_layout_bga3.aedt***

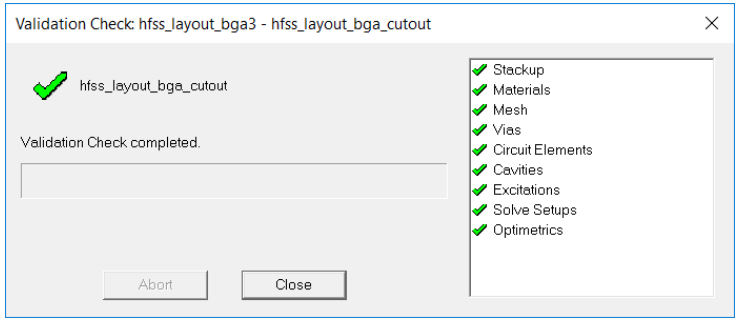
  Keep all HFSS workshop simulation files; future workshops may continue with these files.
- Click on ***Analyze All*** in the ribbon to start the HFSS simulation.

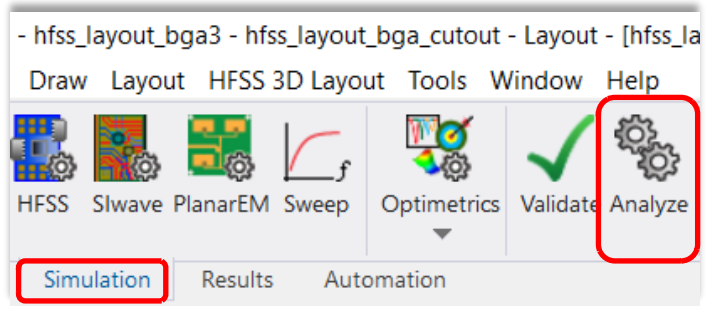
  The ***Validation Check*** and ***Analyze All*** operations are also available from the ***HFSS*** pull-down at the top of the graphical user interface (GUI).
- Save ***hfss_layout_bga3.aedt*** when the simulation finishes.

Click on **Show Messages** and **Show Progress** in the bottom right of the GUI in order to see the simulation progress.

Break time? This simulation might take a few minutes.

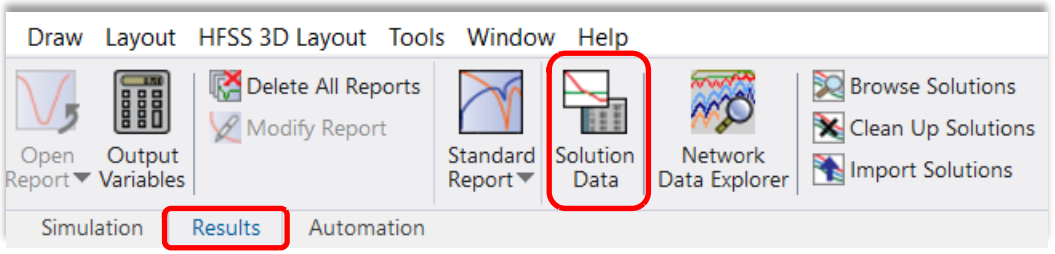






Matrix Data in Solution Data

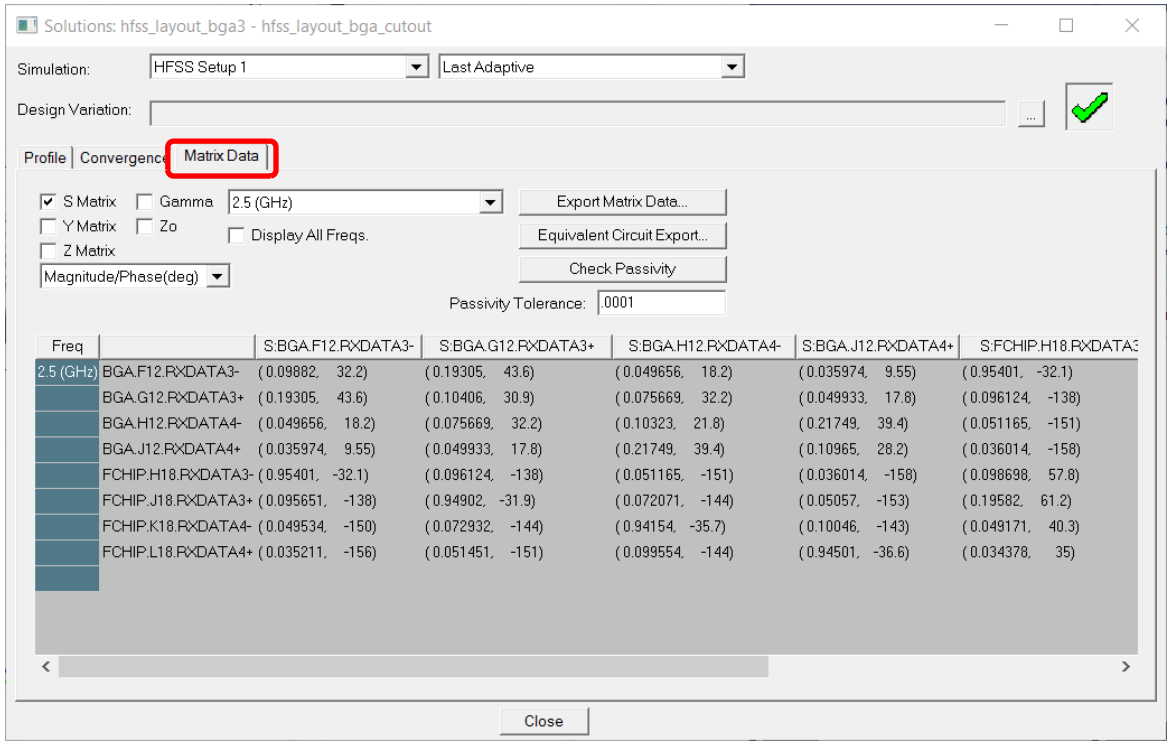
- In the Ribbon, with the **Results** tab highlighted, select ***Solution Data***, which brings up the ***Solutions*** dialog box.



This ***Solutions*** dialog box is also available by right-clicking in the ***Project Manager,*** under ***Analysis***, on ***HFSS Setup 1*** and choosing ***Matrix Data.***

And..

…from the top pull down menus ***HFSS 3D Layout > Results > Matrix Data***



- One can click on the ***Convergence tab***, shown in next slide
- Press the ***Close*** button when you are finished.

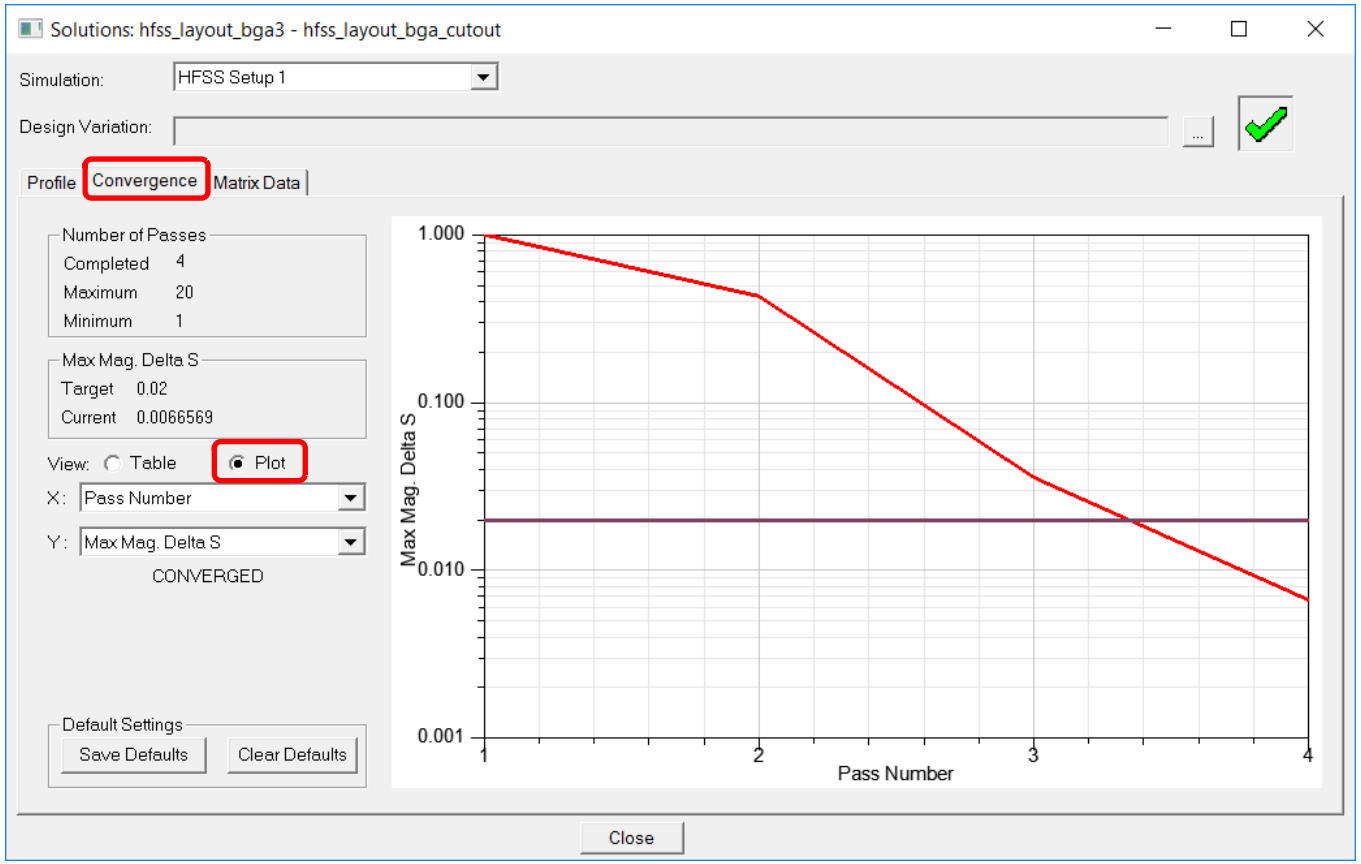
Convergence Plot in Solution Data

- In the ***Solutions*** dialog box, click on the ***Convergence*** tab.

This ***Solutions*** dialog box is also available by right-clicking in the ***Project Manager,*** under ***Analysis***, on ***HFSS Setup 1*** and choosing ***Convergence***.

And..

…from the top pull down menus ***HFSS 3D Layout > Results > Matrix Data or Profile…*** and then click on the ***Convergence*** tab.



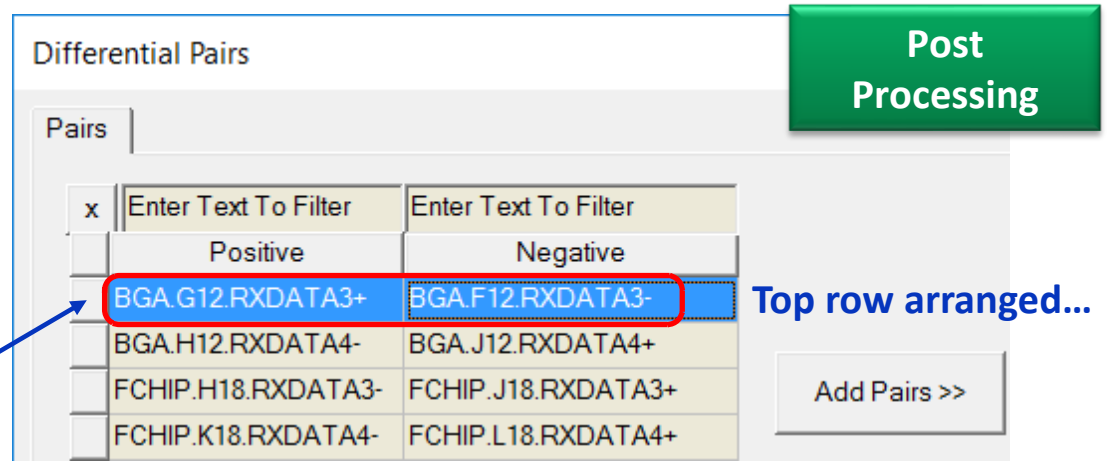
- The default view is for convergence is Table. Select the Plot radio button to view a graphical representations of the convergence data

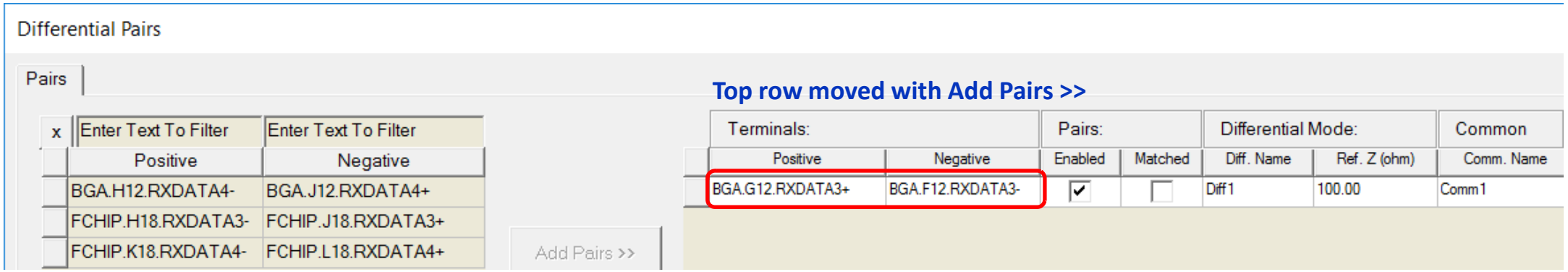
- Press the ***Close*** button when you are finished.

Select Differential Pairs

Creating Differential Pairs - Step 1

- In the ***Project Manager***, under ***Excitations***, right-click and select ***Differential Pairs…***
- This ***Differential Pairs*** dialog box is also available from the top pull-down menus: ***HFSS 3D Layout > Differential Pairs***
- Using the left mouse button, click and drag **BGA RXDATA3** signals to match the ***Positive*** and ***Negative*** headings and each other as shown here on the right.
- Using Ctrl-click or even Shift-click for vertical selection, highlight both **BGA RXDATA3** signals. This makes the ***Add Pairs >>*** button active.
- Click on the ***Add Pairs >>*** button. *…continued….*



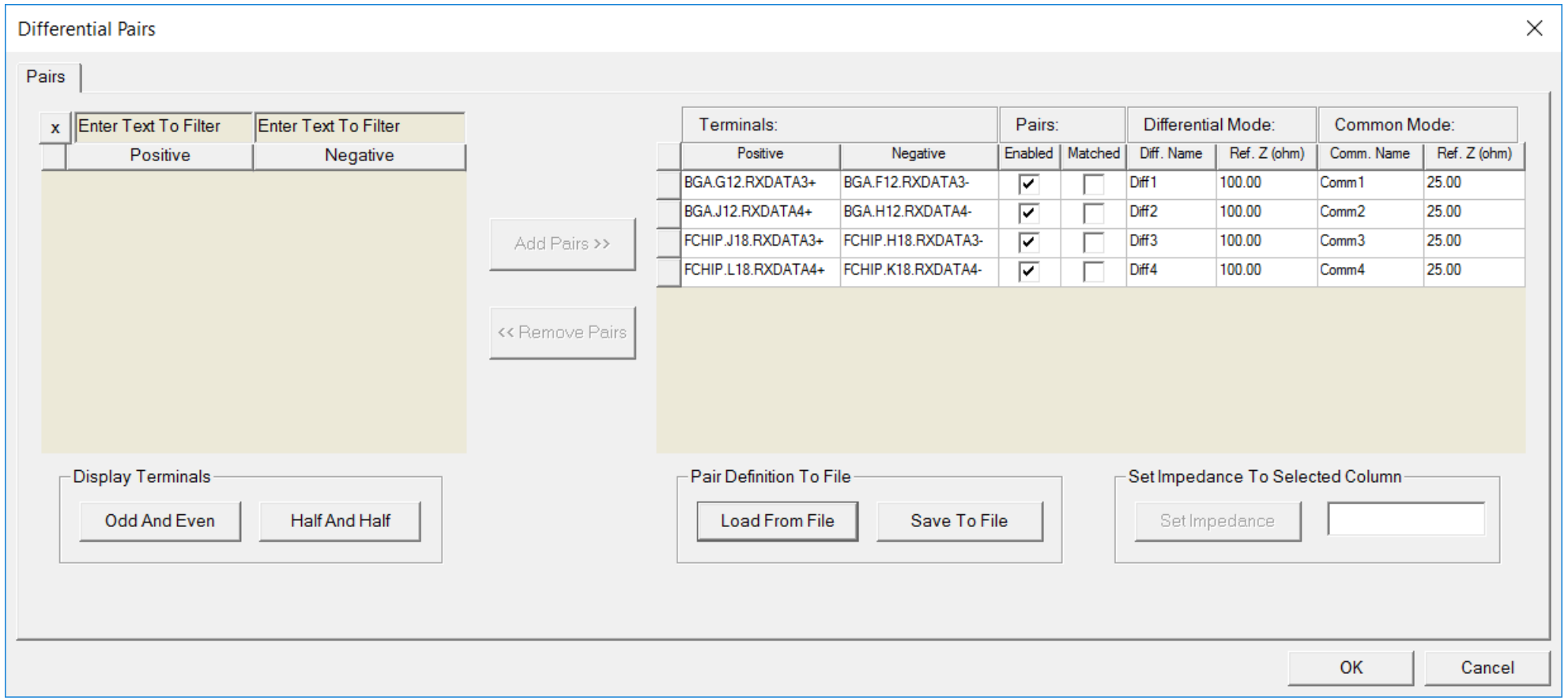


Finish Adding Differential Pairs

Creating Differential Pairs - Step 2

Post Processing

- In the ***Differential Pairs…*** dialog box, continue adding differential pairs.
- Click ***OK*** when finished.



Viewing Results - Starting 2D Report for Differential Pairs

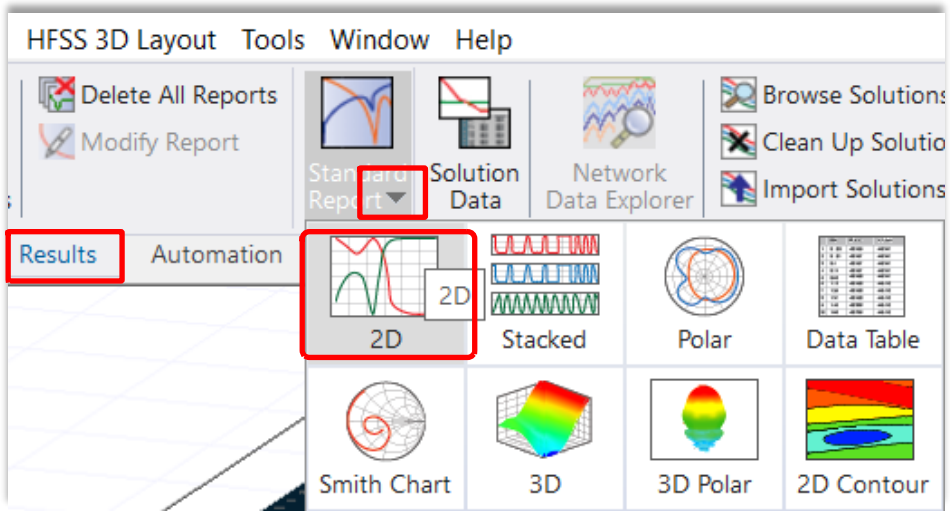
Creating Reports…

- From the Ribbon, with the **Results** tab selected, click on the down triangle by ***Standard Report*** and select ***2D***.

  The Report dialog box can also be accessed from the ***Project Manager***, under ***Results***, by right-clicking on *Results* and selecting ***Create Standard Report > Rectangular Plot***

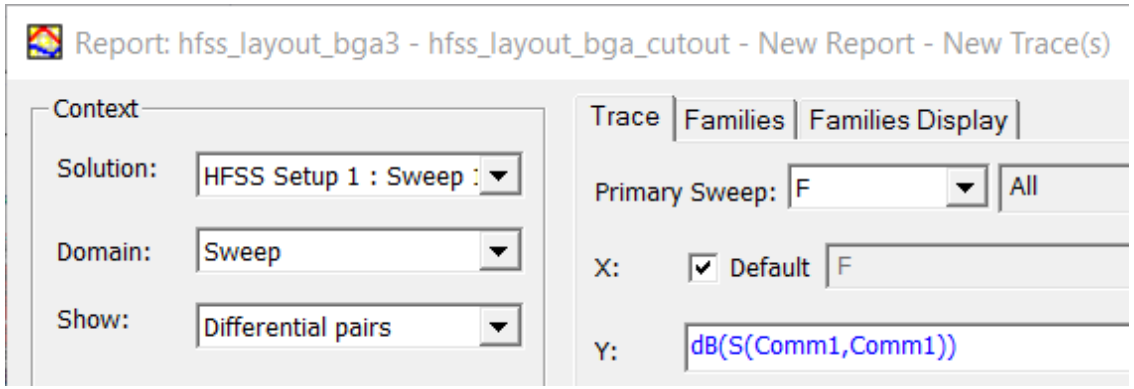
  Or

  …from the top pull-down menus…select the menu **item** ***HFSS 3D Layout > Results > Create Standard Report> Rectangular Plot***



- In the ***Report*** dialog box…
  - ***Solution***: **HFSS Setup1:Sweep 1**
  - ***Domain***: **Sweep**
  - Show: **Differential pairs**
    1. Category: **S Parameter**

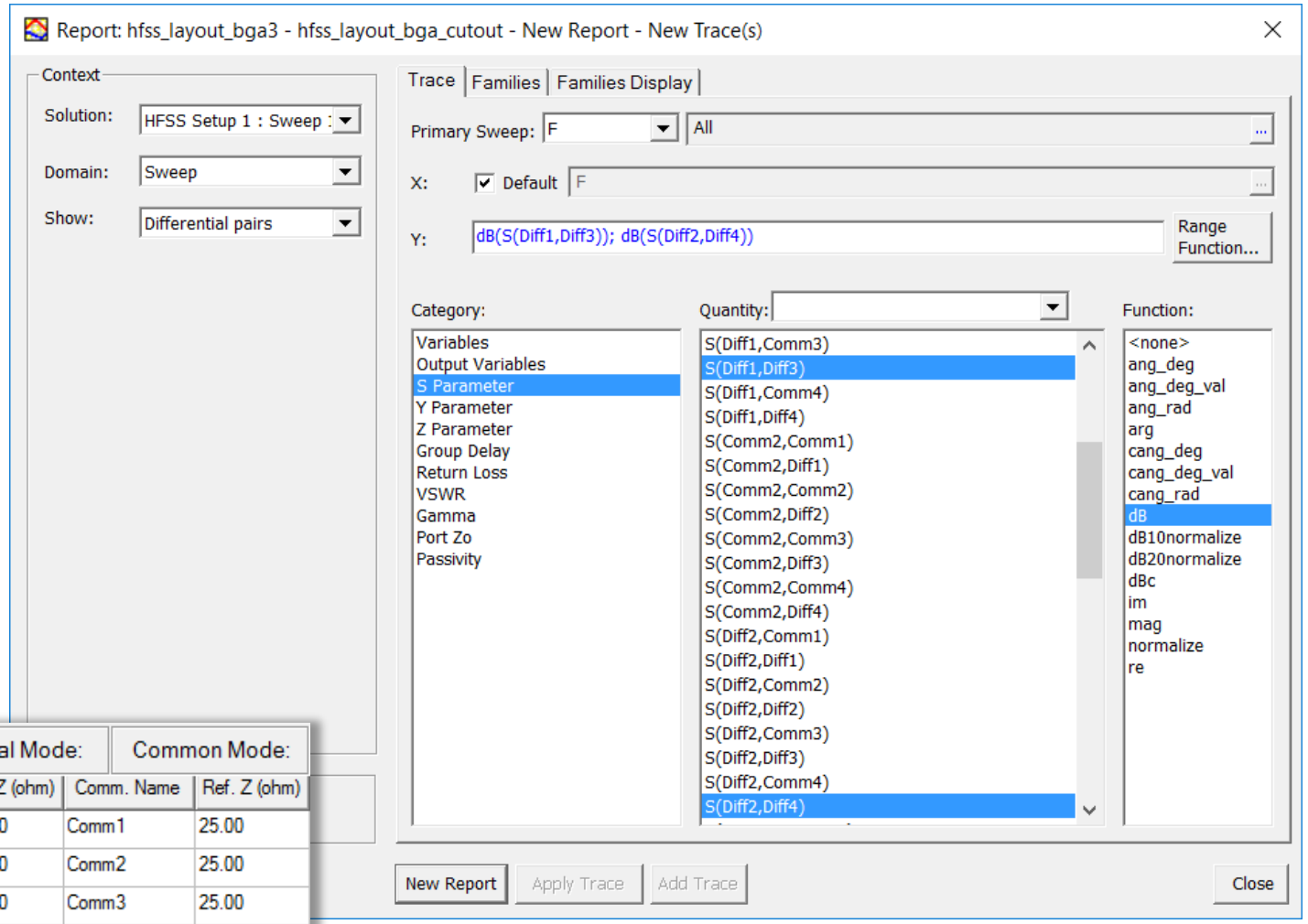
  …continued…



Viewing Results - Transmission Differential Pair S-Parameters

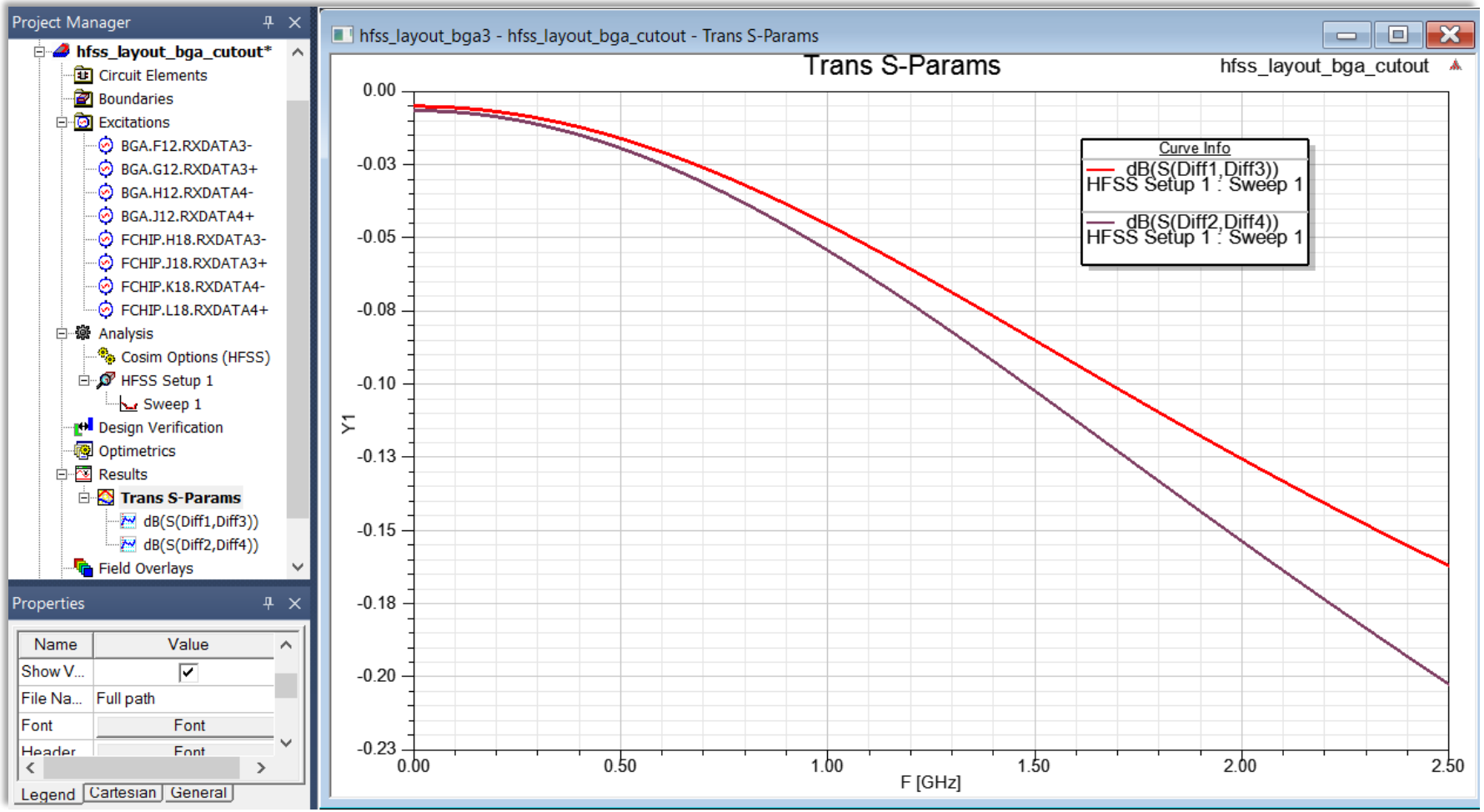
In the ***Report*** dialog we want to pick out the transmission pairs. We can refer to our differential pair definitions as needed…

- ***Quantity***: transmission terms **S(Diff1,Diff3), and S(Diff2,Diff4)**
- ***Function*: dB**
- Click ***New Report*** button and then ***Close***.



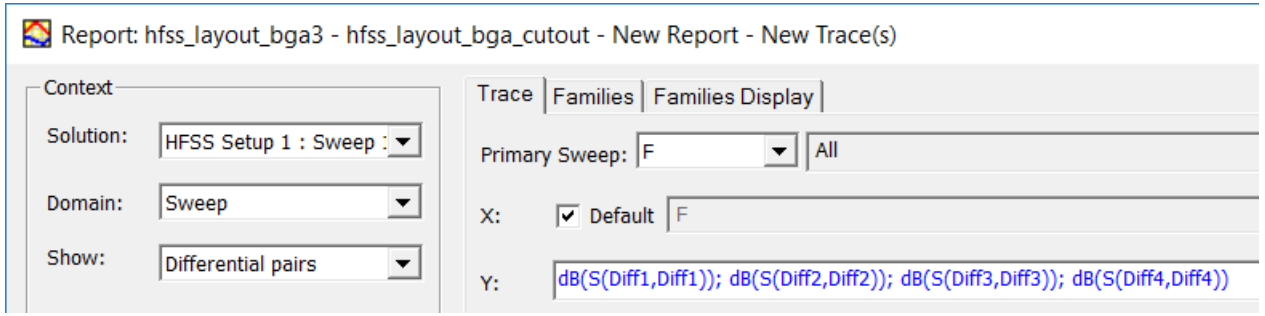
Terminals:		Pairs:		Differential Mode:		Common Mode:	
Positive	Negative	Enabled	Matched	Diff. Name	Ref. Z (ohm)	Comm. Name	Ref. Z (ohm)
BGA.G12.RXDATA3+	BGA.F12.RXDATA3-	☑	☐	Diff1	100.00	Comm1	25.00
BGA.J12.RXDATA4+	BGA.H12.RXDATA4-	☑	☐	Diff2	100.00	Comm2	25.00
FCHIP.J18.RXDATA3+	FCHIP.H18.RXDATA3-	☑	☐	Diff3	100.00	Comm3	25.00
FCHIP.L18.RXDATA4+	FCHIP.K18.RXDATA4-	☑	☐	Diff4	100.00	Comm4	25.00

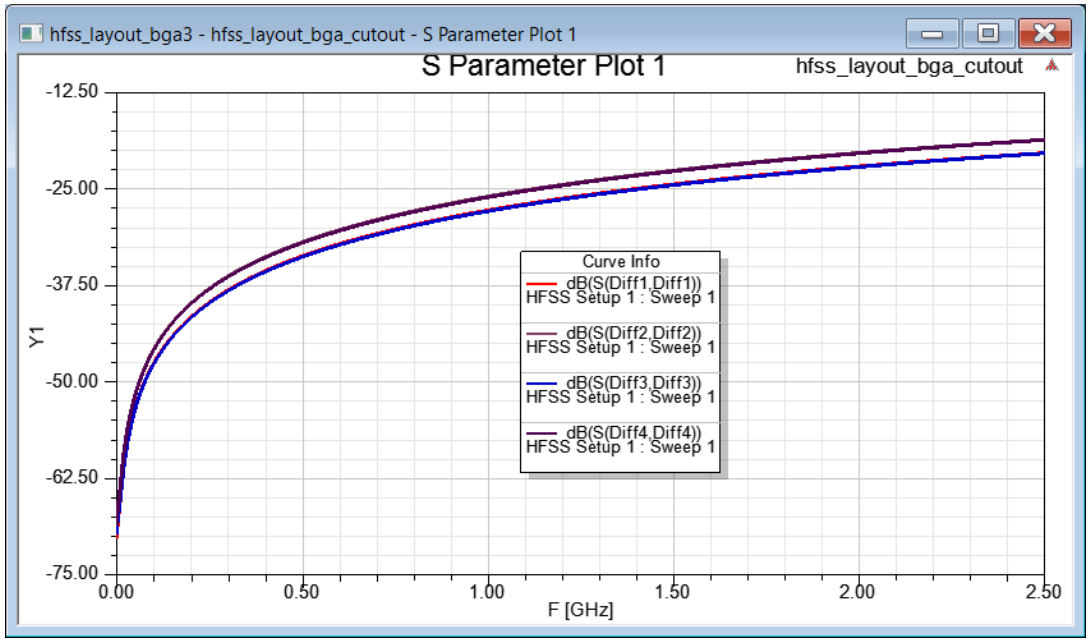
Viewing Differential Pair Transmission S-Parameter Plot



Viewing Results - Reflection Differential Pair S-Parameters

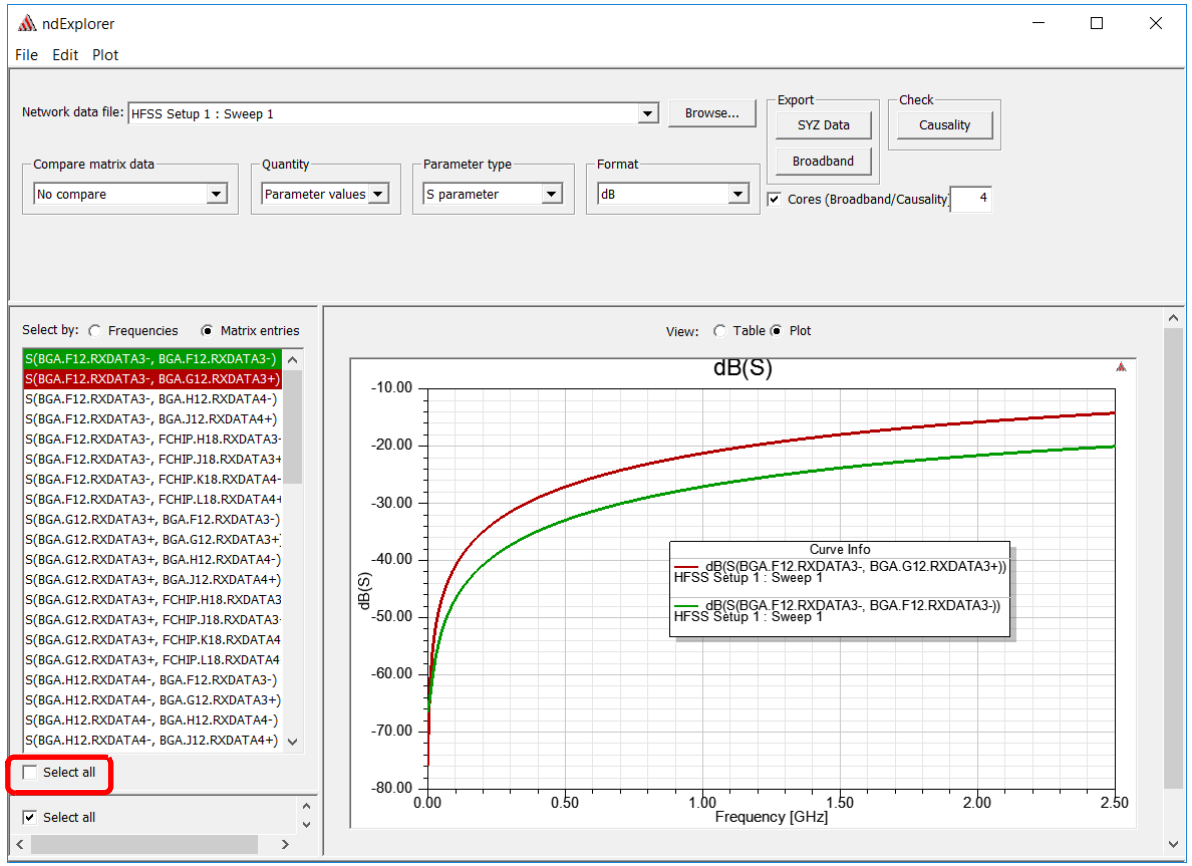
- Create a second Report…
  - ***Solution***: **HFSS Setup1:Sweep 1**
  - ***Domain***: **Sweep**
  - ***Show***: **Differential pairs**
  1. ***Category***: **S Parameter**
  2. ***Quantity***: Click the pull-down and select only self reflection terms **S(Diff1,Diff1), S(Diff2,Diff2), S(Diff3,Diff3), S(Diff4,Diff4)**
  3. *Function*: **dB**
  4. Click ***New Report*** button and then ***Close.***





View Results in Network Data Explorer

- Explore results using ***Network Data Explorer***
  For terminal data with large port-count, ***Network Data Explorer*** provides an efficient and dynamic mechanism for investigating results
  - In the ***Project tree***, under ***Analysis***, expand to show Sweep, then right-click on the ***Sweep1*** entry and select ***Results > Network Data Explorer***
  - Set Format: ***dB***
  - Check ***Select all*** or choose signals individually.
  - Save ***hfss_layout_bga4.aedt*** when the simulation finishes




Ansys