Workshop 2.2: Busbar Inductance

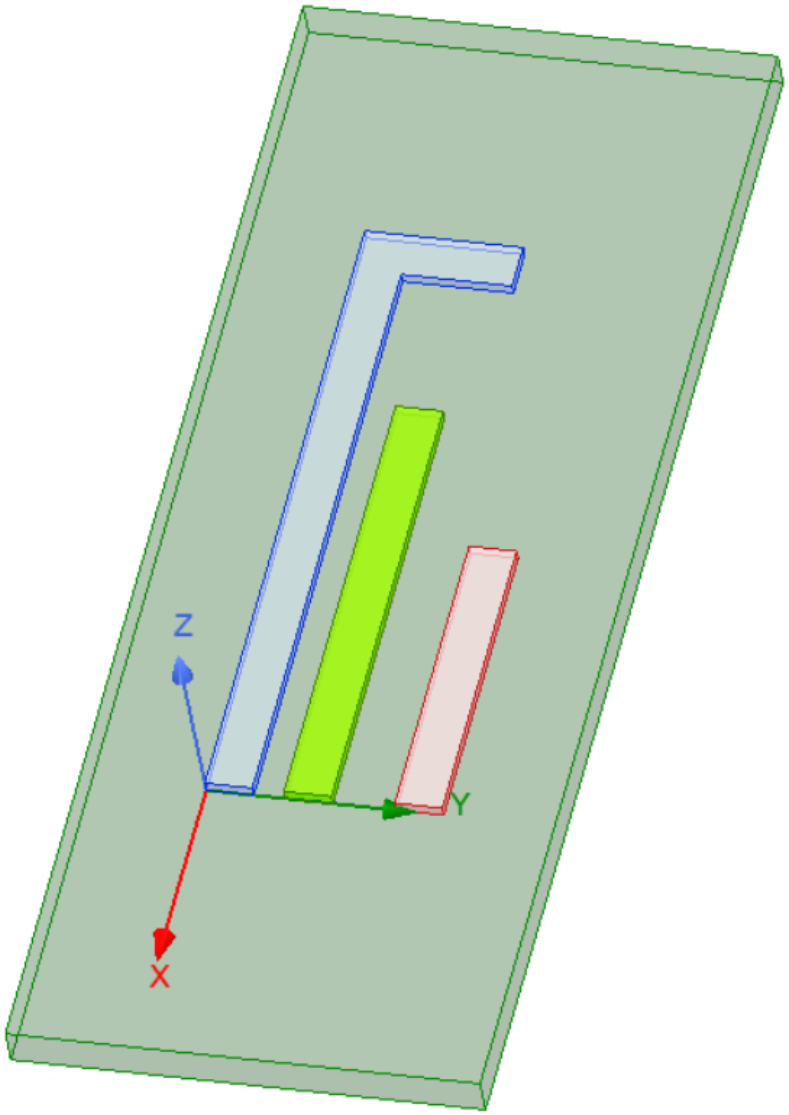
Release 2020 R1

Outline - Busbar 100 MHz Simulation Workshop

The goal of this workshop is to set up a Q3D simulation and learn the workflc

List of what this workshop will do:

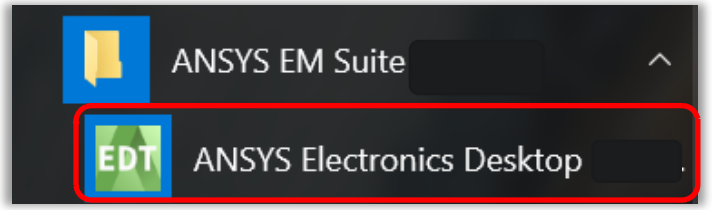
- Invoke AEDT and open archived Q3D example ***EMCDbarS0.aedtz***.
- Set up ***Nets*** - ***EMCDbarS1***
- Define sources and sinks - ***EMCDbarS1***
- Create a single-frequency analysis solution setup.
- Simulate at 100 MHz - ***EMCDbarS2***
- Plot a Data Table of DC and AC inductance
- Add a frequency sweep to the solution setup - ***EMCDbarS3***
- Resimulate
- Plot DC and AC inductance over frequency - ***EMCDbarS3***

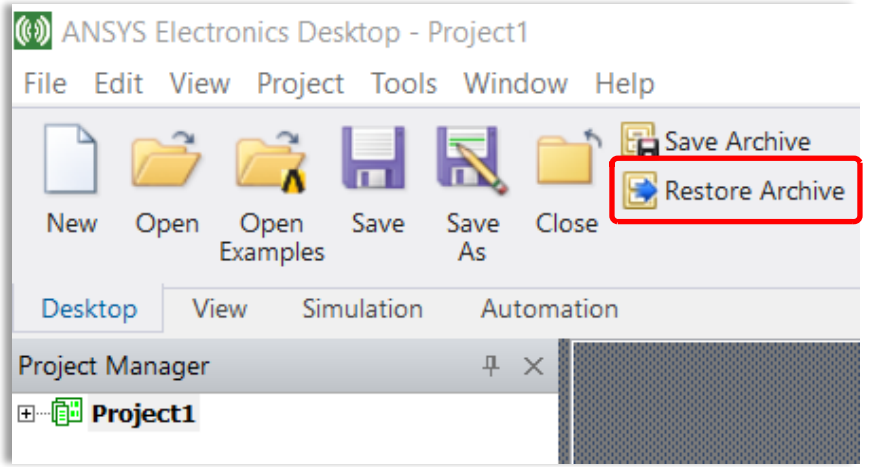


Launch AEDT - Open Q3D Project *EMCDbarS0.aedtz*

Open Q3D and the archived example project ***EMCDbarS0.aedtz …***

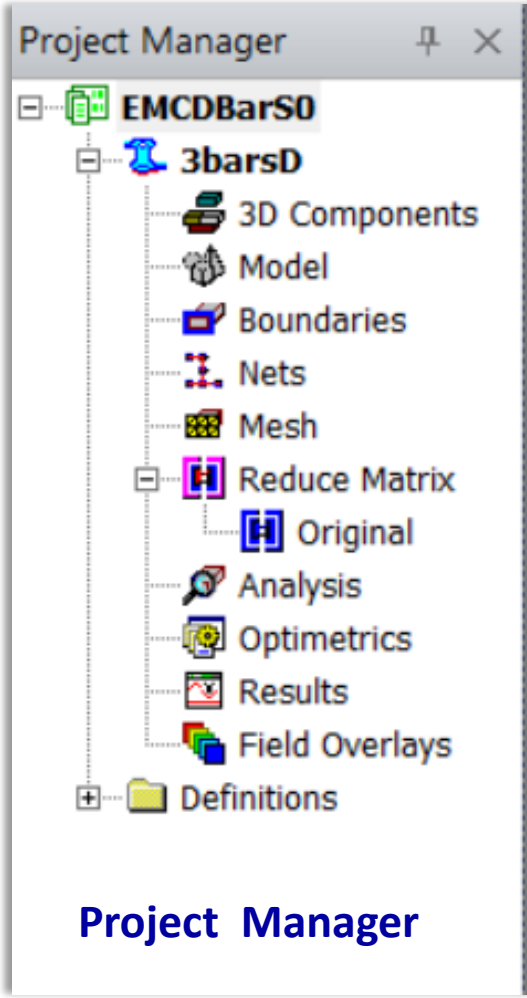
- To access Q3D, click the Microsoft Start button, select ***Programs > ANSYS EM Suite 20XXRY > ANSYS Electronics Desktop***
  A new ***Project1*** appears under the ***Project Manager***.
- In the ***Ribbon***, in the Desktop tab, select the icon: ***Restore Archive***
- Browse to the workshop directory and click on ***EMCDbarS0.aedtz***
  z refers to archive
- Select ***Open***
- When the browser changes to ***Project File Restore Location, Save*** the file ***EMCDbarS0.aedtz*** in a working directory.



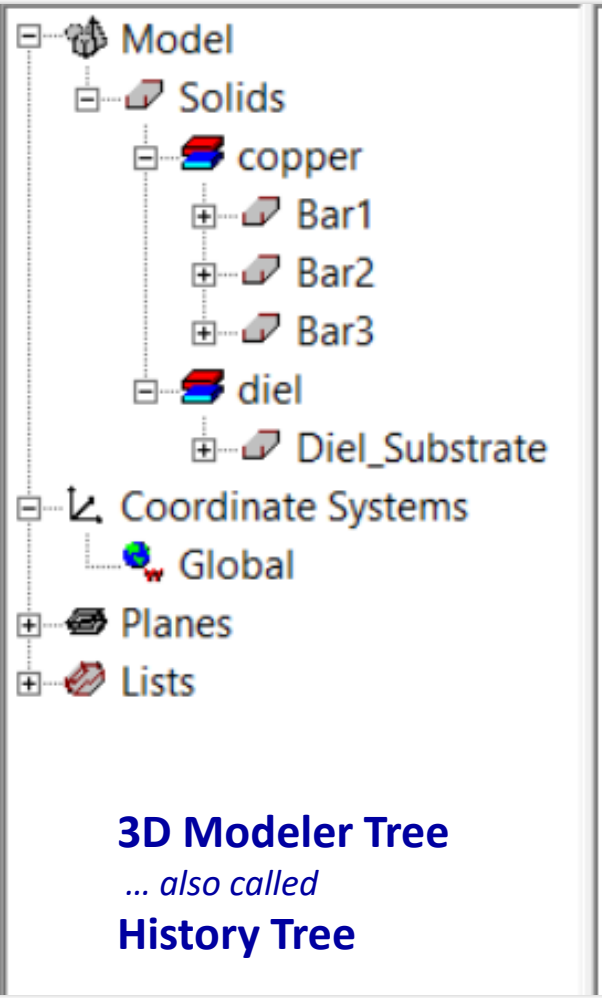


Q3D Project EMCDbarS0.aedt

There are no ***Boundaries*** nor ***Nets*** set up. There are no ***Analysis*** nor ***Results*** setups.



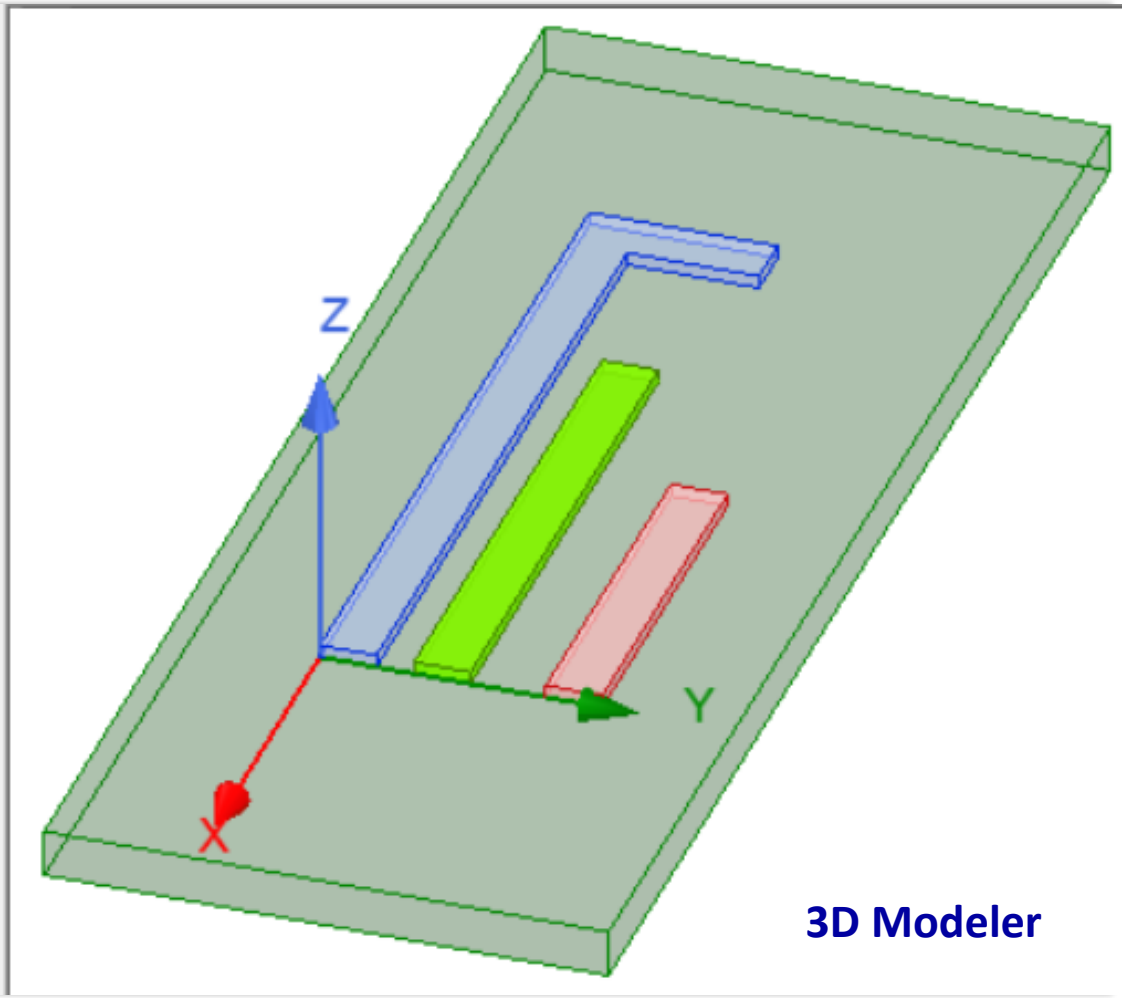
Project Manager



3D Modeler Tree

… also called

History Tree

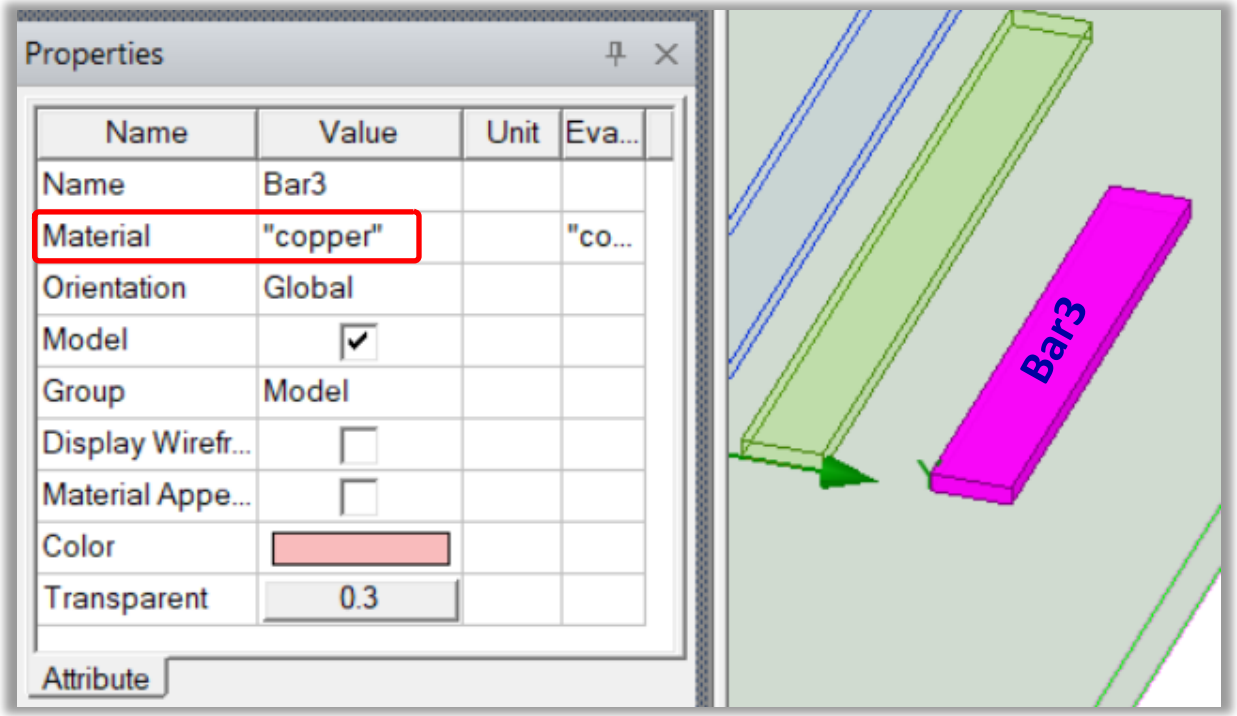


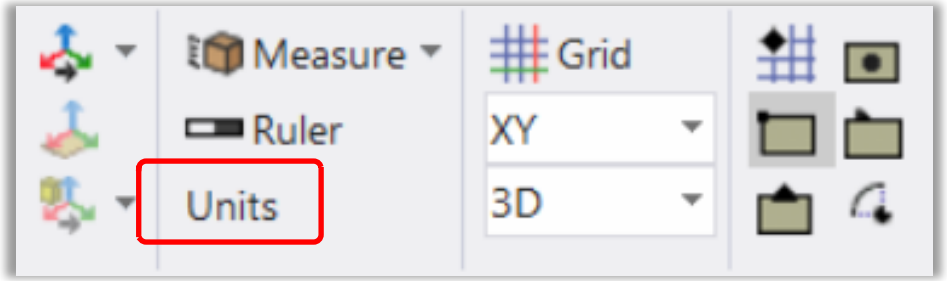
3D Modeler

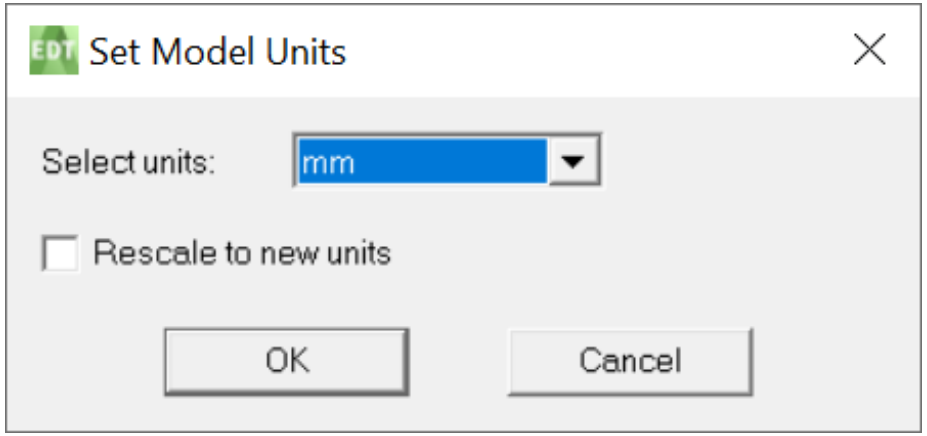
Ansys

Units mm and Copper Material - EMCDBarS1.aedt

- In the ***Ribbon***, with the Draw tab selected, click on the icon for ***Units*** to bring up the ***Set Model Units*** window.
- Verify that units are ***mm*** and close the window.
- In the ***3D Modeler***, click on **Bar3**, the shortest bar on the right, to select it.





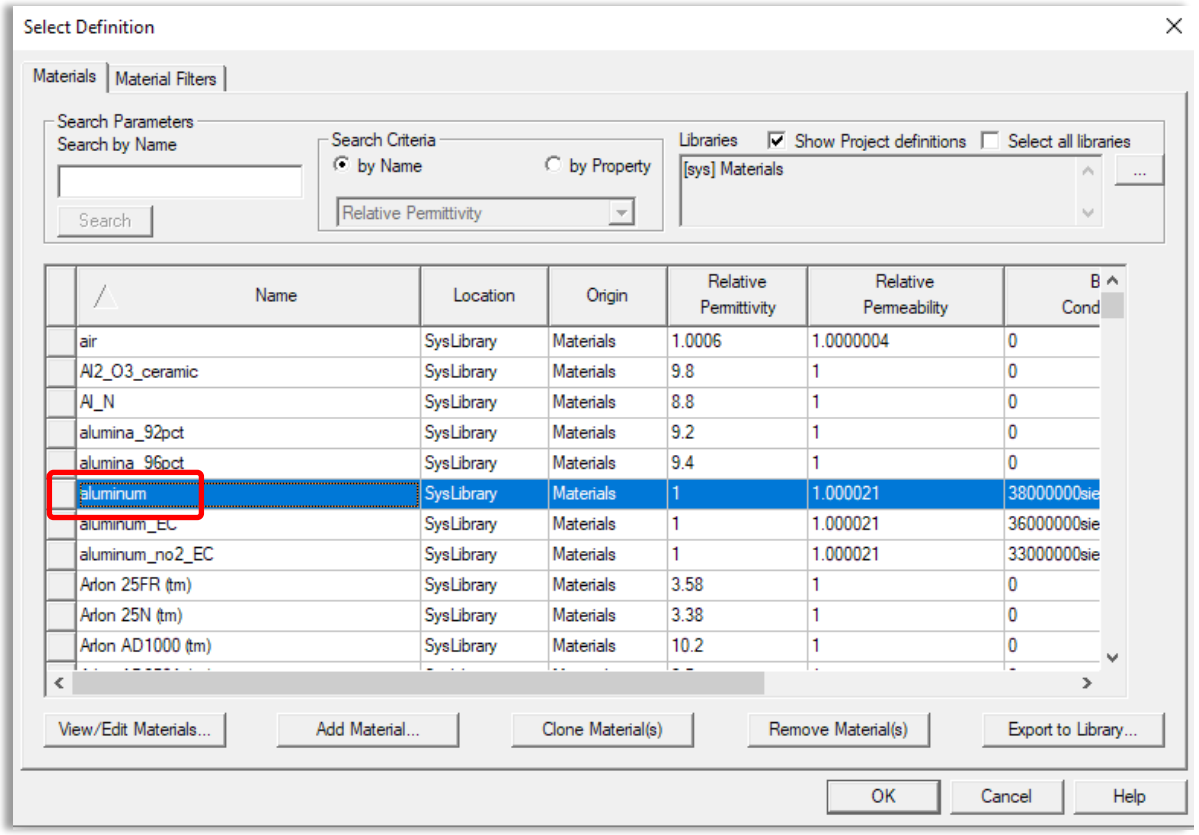


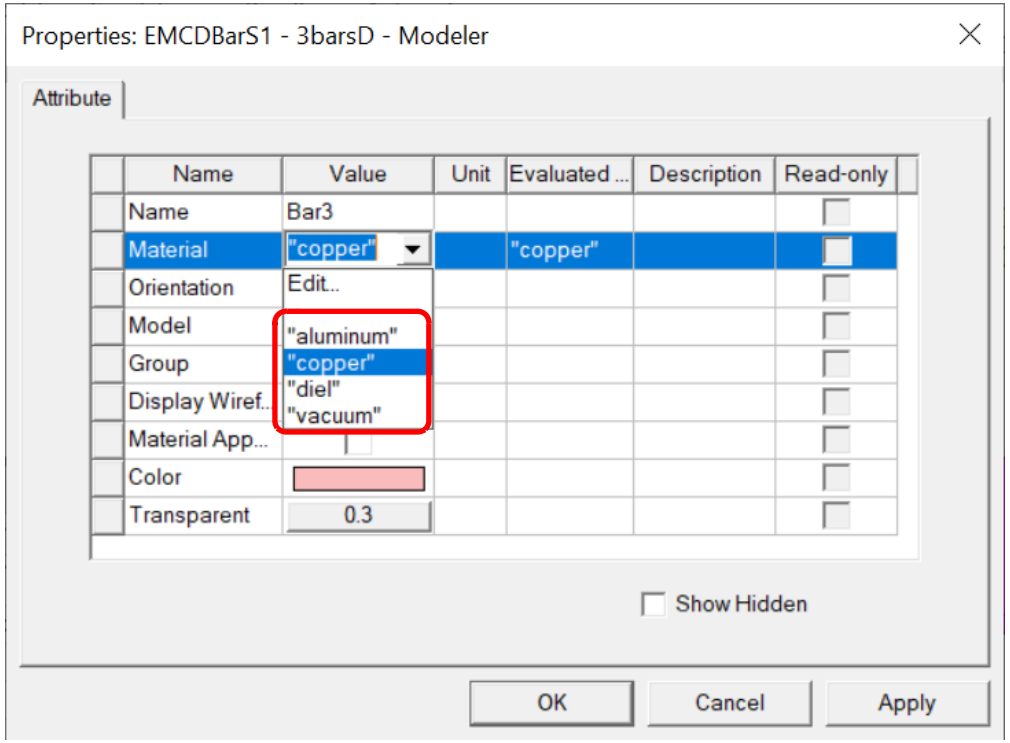
- In the lower left, in the ***Properties*** window, verify the copper material on **Bar3**.
- Save project to **EMCDBarS1.aedt**

Changing Bar3 *Material* to Aluminum - EMCDBarS1.aedt

- Still in ***Properties***, choose ***"copper"*** in the ***Material*** row and choose ***Edit…*** to bring up a ***Select Definition*** window.

  (Another way to bring up this same ***Select Definition*** window is to double-click in the ***3D Modeler Tree*** on ***Bar3.)***
- In the ***Select Definition*** window, scroll or search for ***aluminum***.





- Select ***aluminum*** and press ***OK*** to close the window.

  (If you were using the double-click method for properties, you'll have to press ***OK*** again to close the window.)

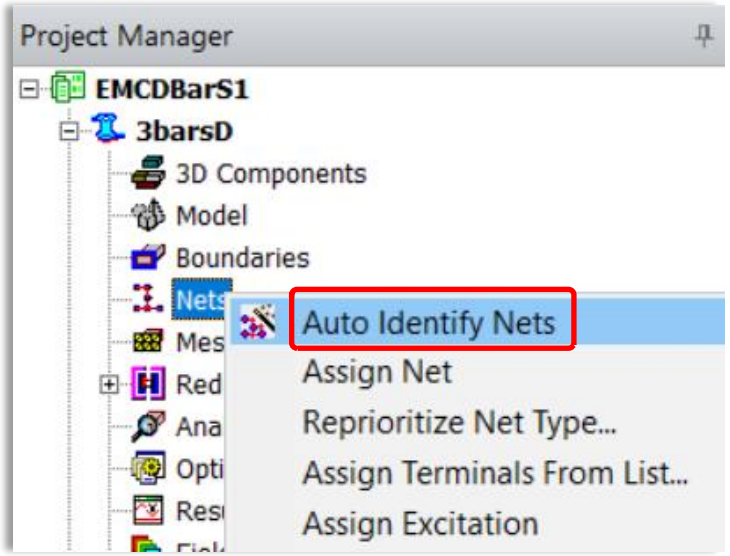
Ansys

Defining Nets - EMCDBarS1.aedt

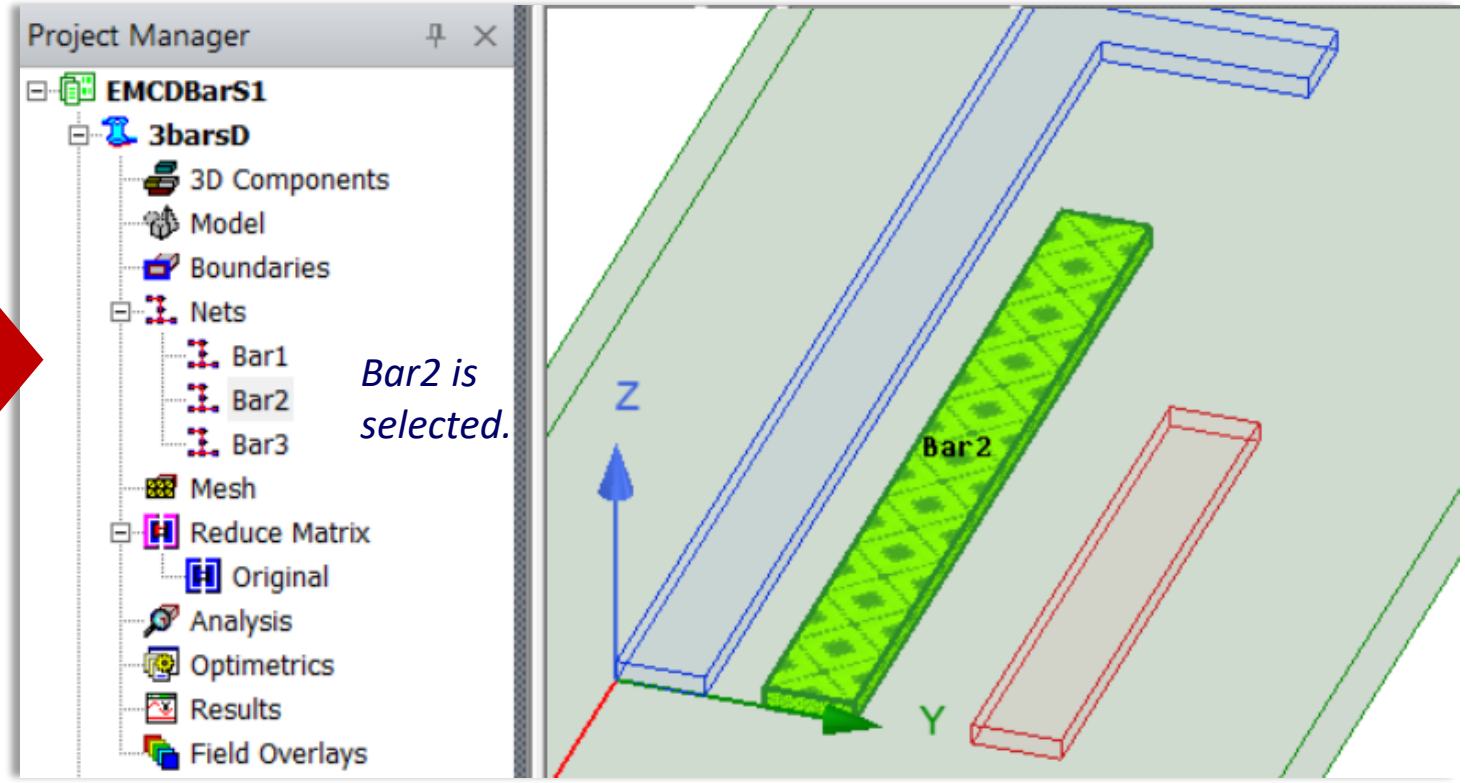
When the design geometry and all the materials are properly set, you can define ***Nets***:

- In the ***Property Manager***, right-click on ***Nets*** and select ***Auto Identify Nets.***

This project already has 3 conducting structures, so when we ***Auto Identify Nets***, 3 ***Nets*** will appear.



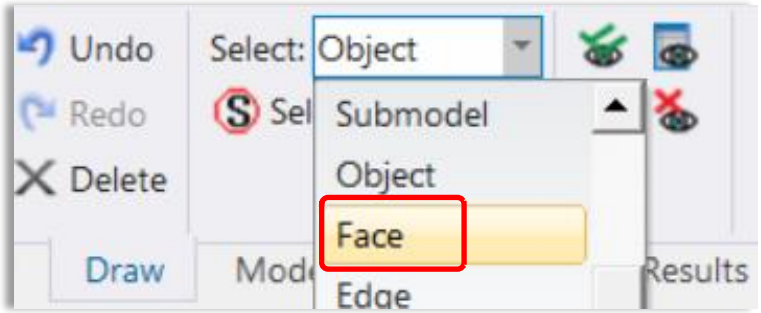
Reminder: If we only want to solve for capacitance, we don't need to define ***Sources*** *and* ***Sinks****.*



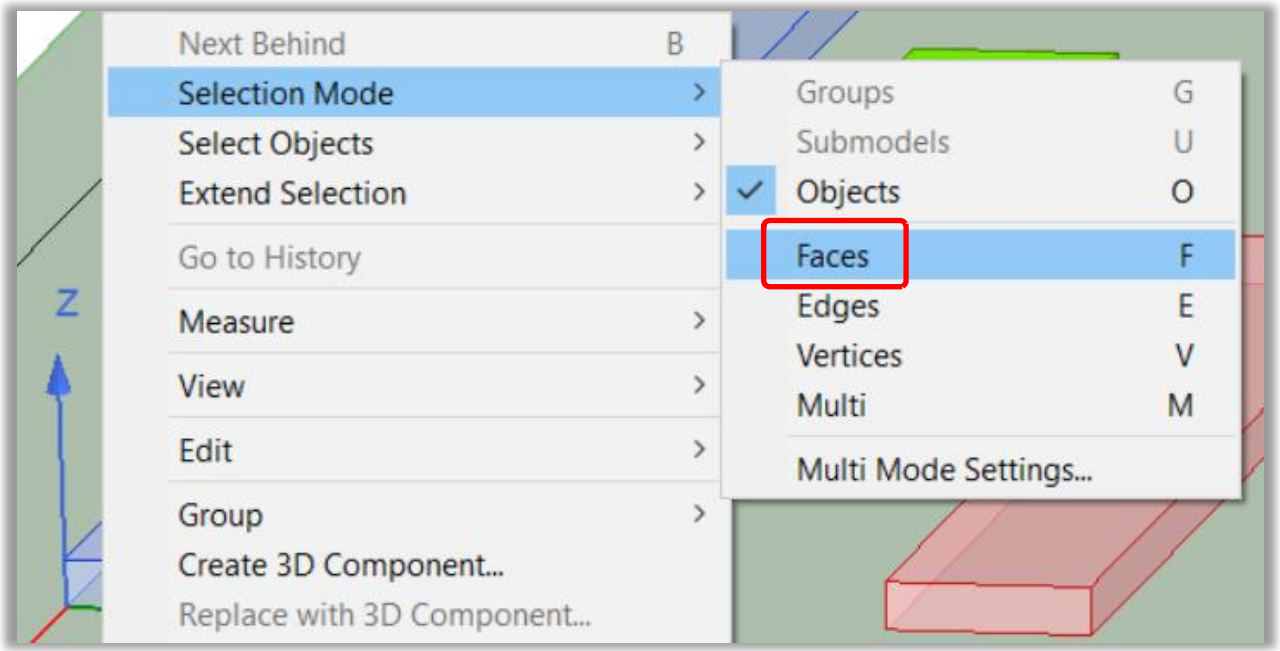
Ansys

Selecting *Faces* for *Terminals* - EMCDBarS1.aedt

In this next section, we assign a current ***Source*** on one side of each bar and a current ***Sink*** on the other side of each bar. ***Terminals*** can be directly applied to geometry faces.



- Enter ***Face*** selection mode one of three ways:
  - In the ***Ribbon***, in the Draw tab, change the ***Select*** option to ***Face***.
  - In the ***3D Modeler*** window, right-click and choose ***Selection Mode > Faces***
  - Press ***F*** … the hot key for invoking ***Faces*** select mode.

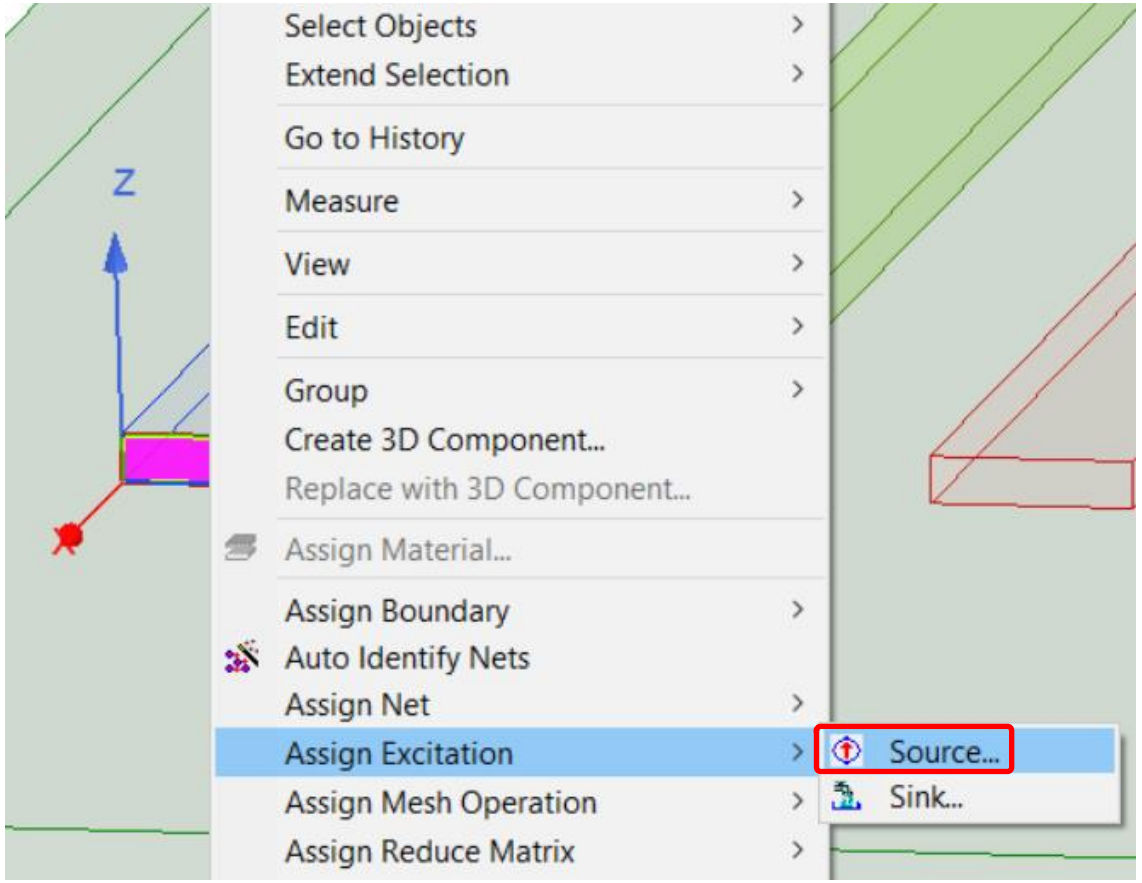


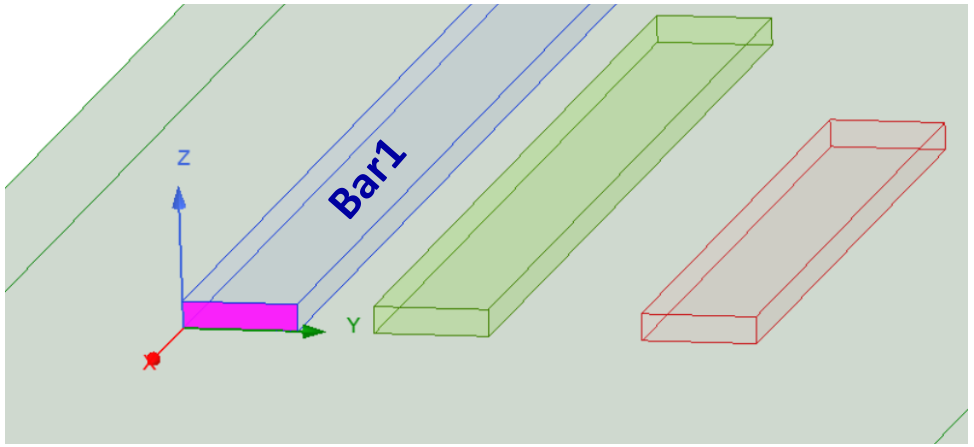
*Additional information: Q3D offers a boundary condition called **Thin Conductor**. This boundary can be applied to a sheet. If this is done, then you can click the edge and apply a **Terminal** to the edge.*

Ansys

Adding *Terminals* to *Net* Faces - Source1

- In ***Face Select*** mode, click the **Bar1** face, (on the left-hand side), at the front of the model looking towards negative **X**.



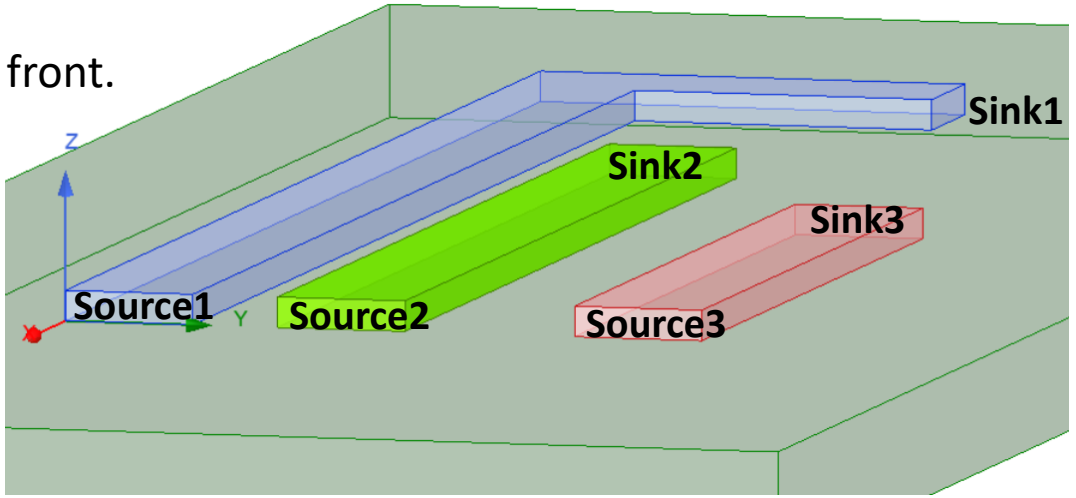


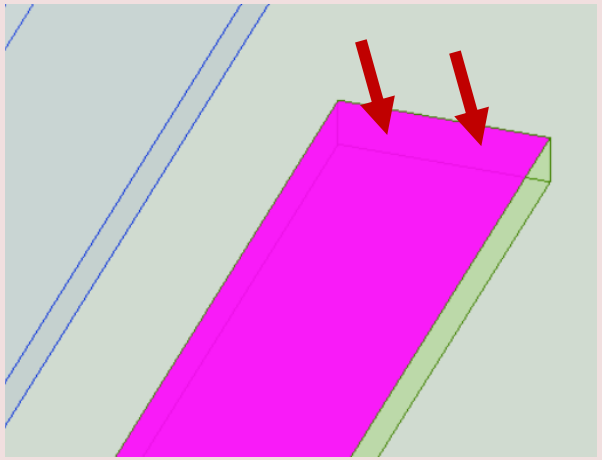
- In the ***3D Modeler***, right-click and select ***Assign Excitation > Source,*** which brings up a ***Source*** dialog box***.***

  (An alternative is pull-down menu ***Q3D Extractor > Nets > Assign Excitation.***)
- In the ***Source*** dialog box, keep the default ***Source 1*** and click on ***OK.***

Assigning *Terminals* to All *Nets* - EMCDBarS1

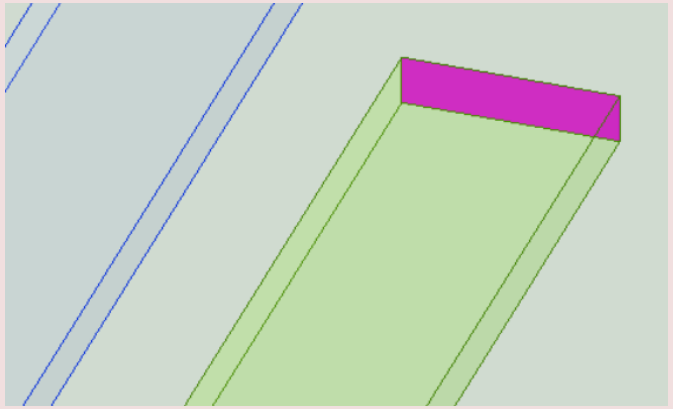
- Repeat this process of assigning two more ***Sources*** to the front.
- Assign ***Sinks*** to the back faces of all three bars.
- Save project **EMCDBarS1**.



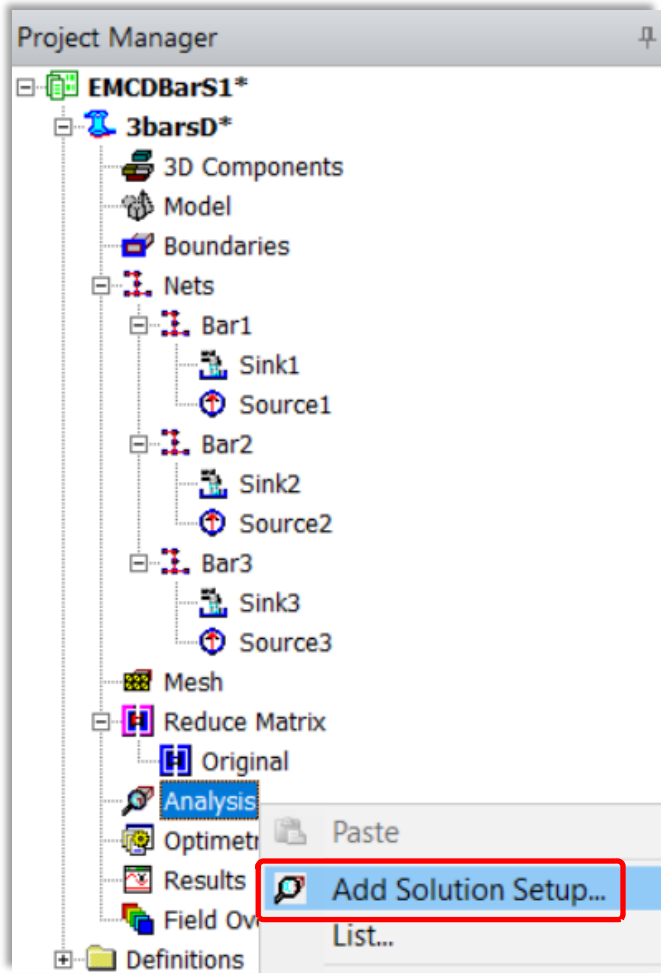


For the ***Sinks***, that are on the back of the geometry, you can click where the desired face is, and when the top faces gets selected, click the ***B*** button (***Next Behind***) to select the desired face.

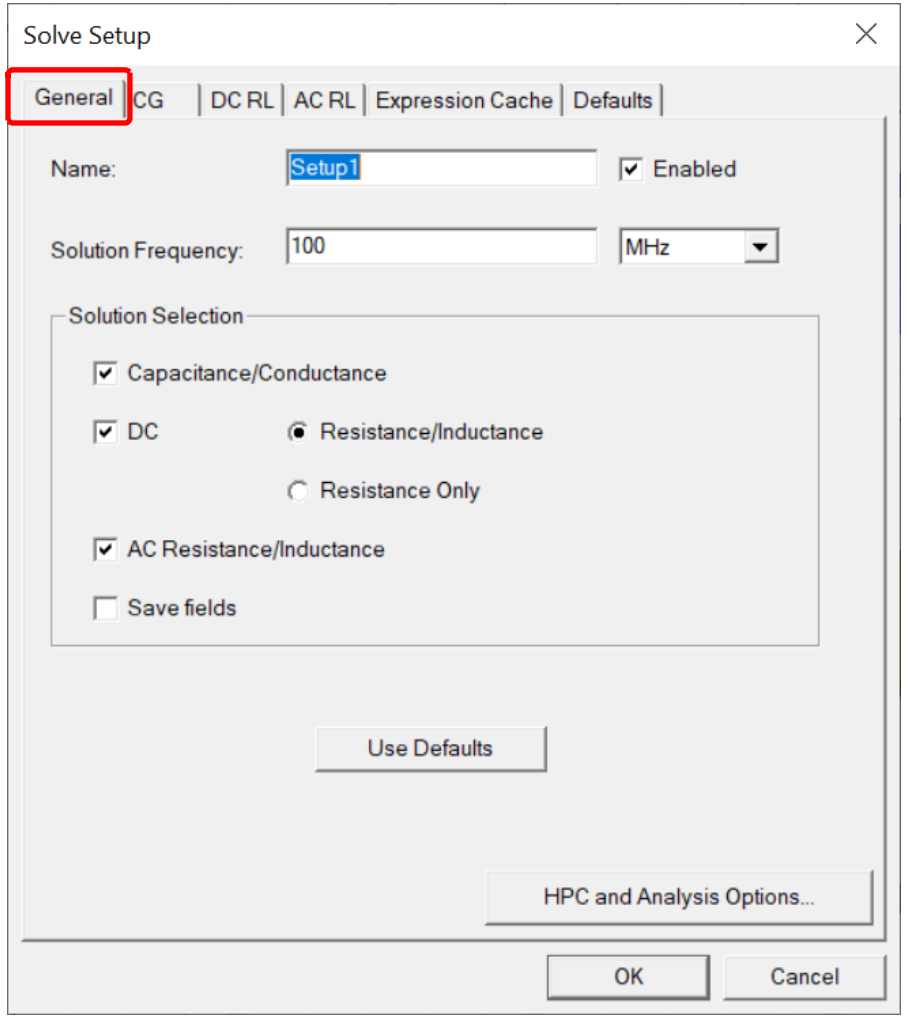
See Appendix at the end for other useful hot keys.



Adding a *Solution Setup* - EMCDBarS1



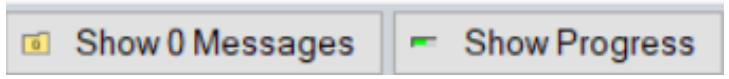
- In the ***Project Manager***, right-click on ***Analysis*** and select ***Add Solution Setup …*** to bring up the S***olve Setup*** dialog box.
- In the ***Solve Setup*** window, in the ***General*** tab, set:
- ***Solution Frequency***: **100 MHz**
- Leave all the boxes checked ☑
- Save project **EMCDBarS1**.



Another way to access Solution Setup is from the top pull downs:
Q3D Extractor -> Analysis Setup -> Add Solution Setup.

Validate and Analyze at 100 MHz - EMCDBarS2

- Save the project as **EMCDBarS2**.
- In the ***Ribbon***, in the Simulation tab, click the icon for ***Validate*** with the **green** check mark.
- Make sure that the ***Progress*** and ***Message Manager*** windows are showing. If not, click the icons at the bottom of the interface to open those windows.



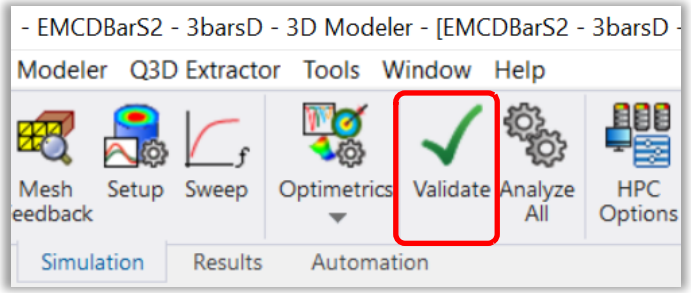
- Just to the right of Validate, click on ***Analyze All*** to start the Q3D simulation.

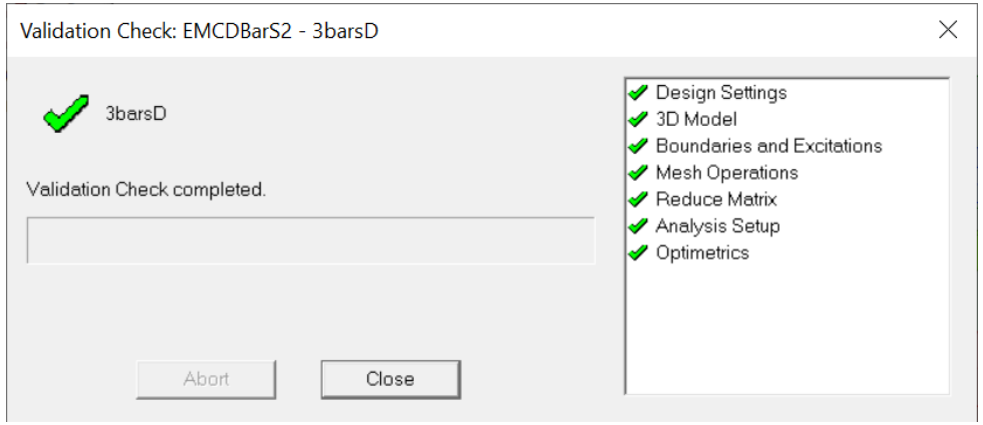
  The ***Validation Check*** and ***Analyze All*** operations are also available from the ***Q3D Extractor*** pull-down at the top of the graphical user interface (GUI).

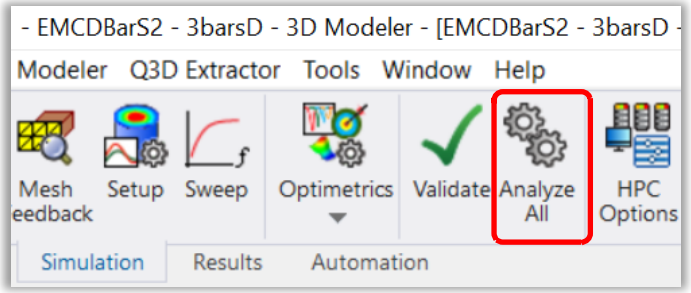
- Watch the ***Progress Window*** and the ***Message Manager*** to see when the simulation finishes.

  Keep all Q3D workshop simulation files; future workshops may continue with these files or refer to results.

- When the simulation finishes, save the project again.



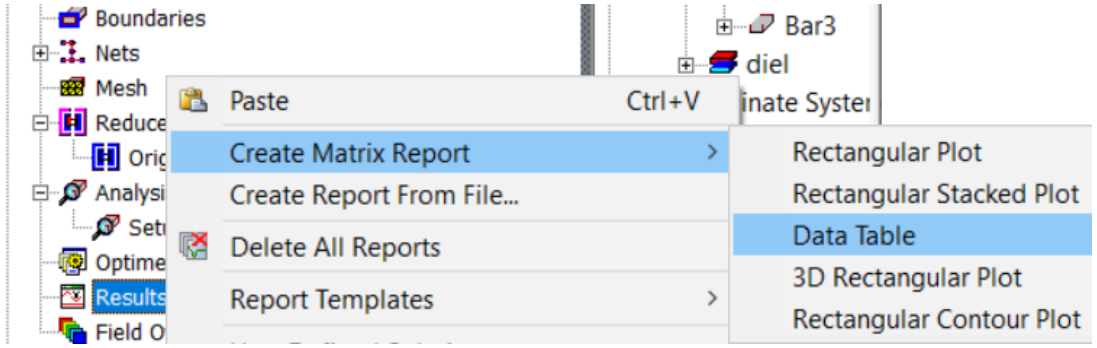


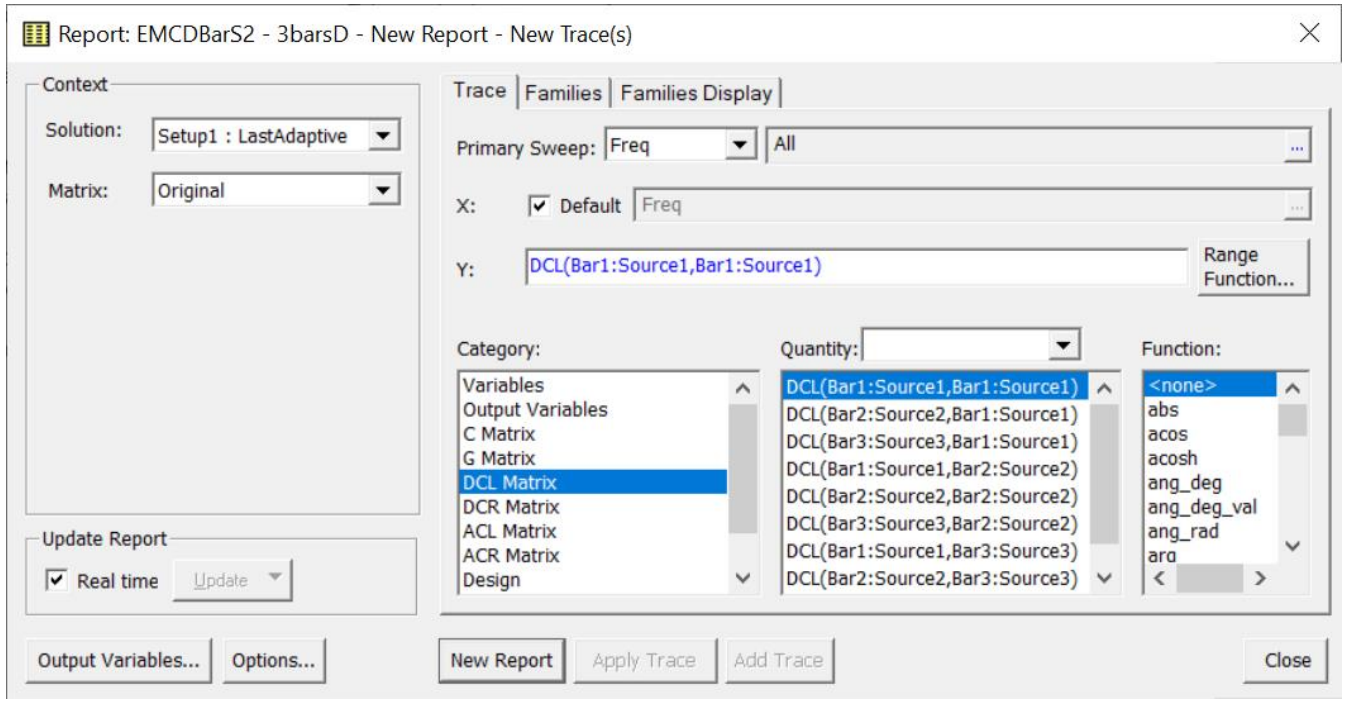


Ansys

Plot Single Frequency Inductance Results - EMCDBarS2.aedt

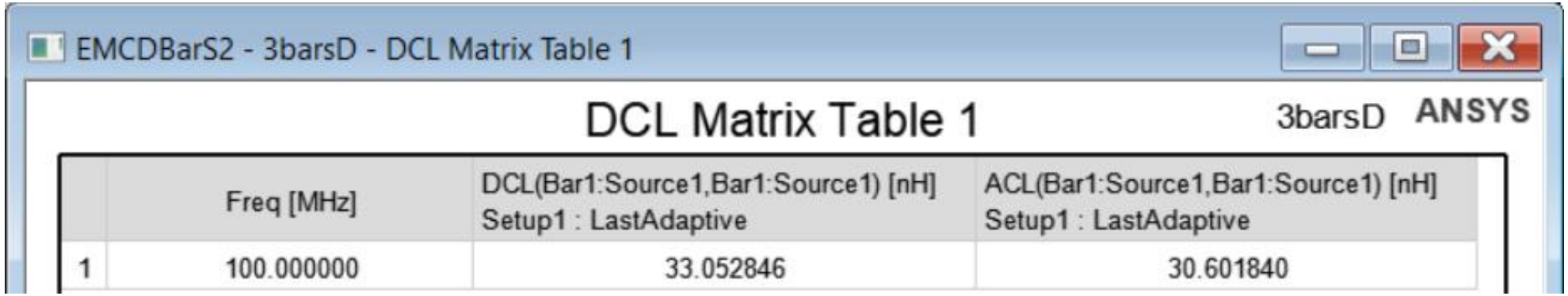
- Plotting DC Inductance ***Data Table***…
  - In the ***Project Manager***, right-click ***Results***
  - Select ***Create Matrix Report->Data Table***
  - Under ***Category***, select ***DCL Matrix***
  - Click **DCL(bar1:Source1,bar1:Source1)**
  - Press ***New Report*** (… and leave the dialog open)
- Plotting AC Inductance ***Data Table***…
  - Still in the ***Report*** menu window …
  - Under ***Category***, select ***ACL Matrix***
  - Click ***ACL(bar1:Source1,bar1:Source1)***
  - Press ***Add Trace***
- Click ***Close*** to close the ***Report*** dialog box.
- Save the project **EMCDBarS2**.





Comparing DC Inductance to AC Inductance - EMCDBarS2.aedt

The DC inductance is 33 nH, while the AC inductance is 30.6 nH for the self-inductance of Net **Bar1**.


EMCDBarS2 - 3barsD - DCL Matrix Table 1

DCL Matrix Table 1

3barsD ANSYS

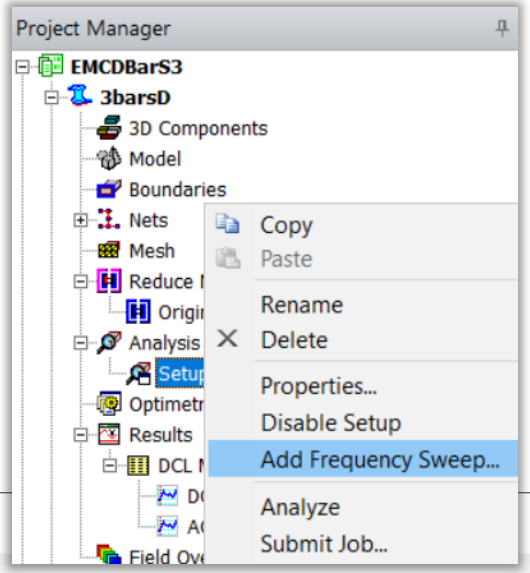
	Freq [MHz]	DCL(Bar1:Source1,Bar1:Source1) [nH] Setup1 : LastAdaptive	ACL(Bar1:Source1,Bar1:Source1) [nH] Setup1 : LastAdaptive
1	100.000000	33.052846	30.601840

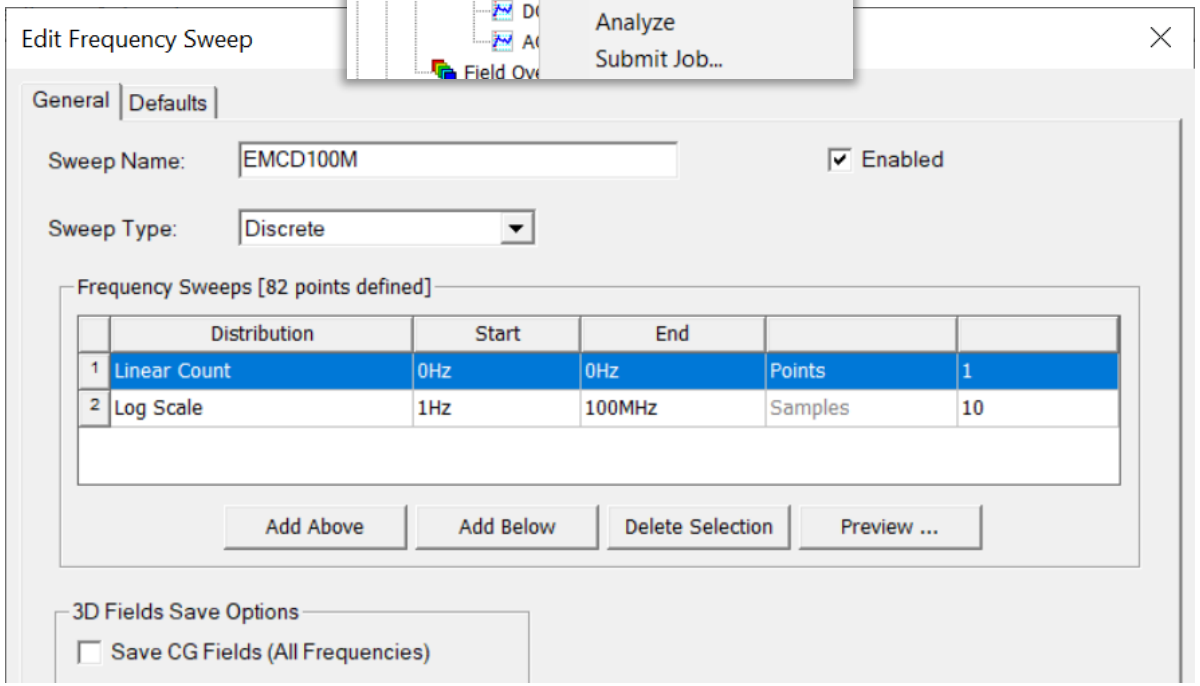
Why?

- If we think of current traveling through a wire at DC, we would see that current flows on the surface and inside of the wire. Current density would be uniform on the cross section of the wire.
- At higher frequencies, current travels mostly on the surface of the conductor, which is realized in the skin effect.
- This leads directly to a reduction in inductance as the oscillations of a wave increase in frequency. (The inductance around a wire decreases as the radius of the wire increases. Increase in frequency moves current to the outer edges of a conductor. If we consider this similar to having a larger wire, then we expect the inductance to go down as frequency goes up.)

Add Frequency Sweep and Resimulate - EMCDBarS3

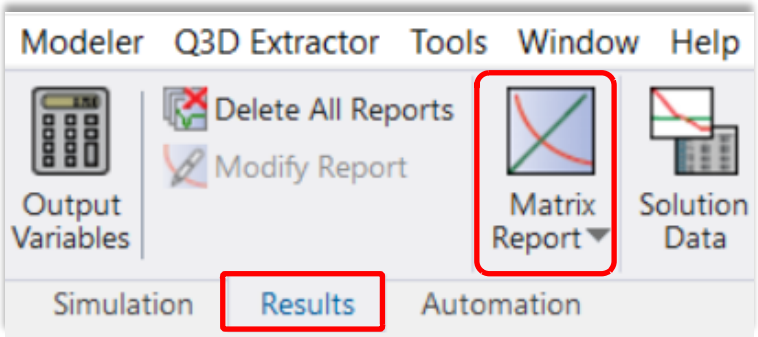
- Save the project to **EMCDBarS3**.
- In the ***Project Manager***, expand ***Analysis***.
- Right-click on **Setup1** and select ***Add Frequency Sweep…*** to bring up the ***Edit Frequency Sweep*** window.
- Set ***Sweep Name***: to **EMCD100M**
- Set ***Sweep Type*** to ***Interpolating***
- For the first ***Frequency Sweep*** line 1 set:
  - ***Distribution***: ***Linear Count***
  - ***Start***: **0Hz** ***End***: **0Hz** ***Point***s: **1**
- Click ***Add Below*** to get a second line
- For the second ***Frequency Sweep*** line 2 set:
  - ***Distribution***: ***Log Scale***
  - ***Start***: **1Hz** ***End***: **100MHz** ***Point***s: **10**
- Click ***OK*** to close the ***Edit Frequency Sweep*** window.
- Save the project.
- Click on ***Analyze*** to resimulate the project.
- Save the project again when the simulation ends.

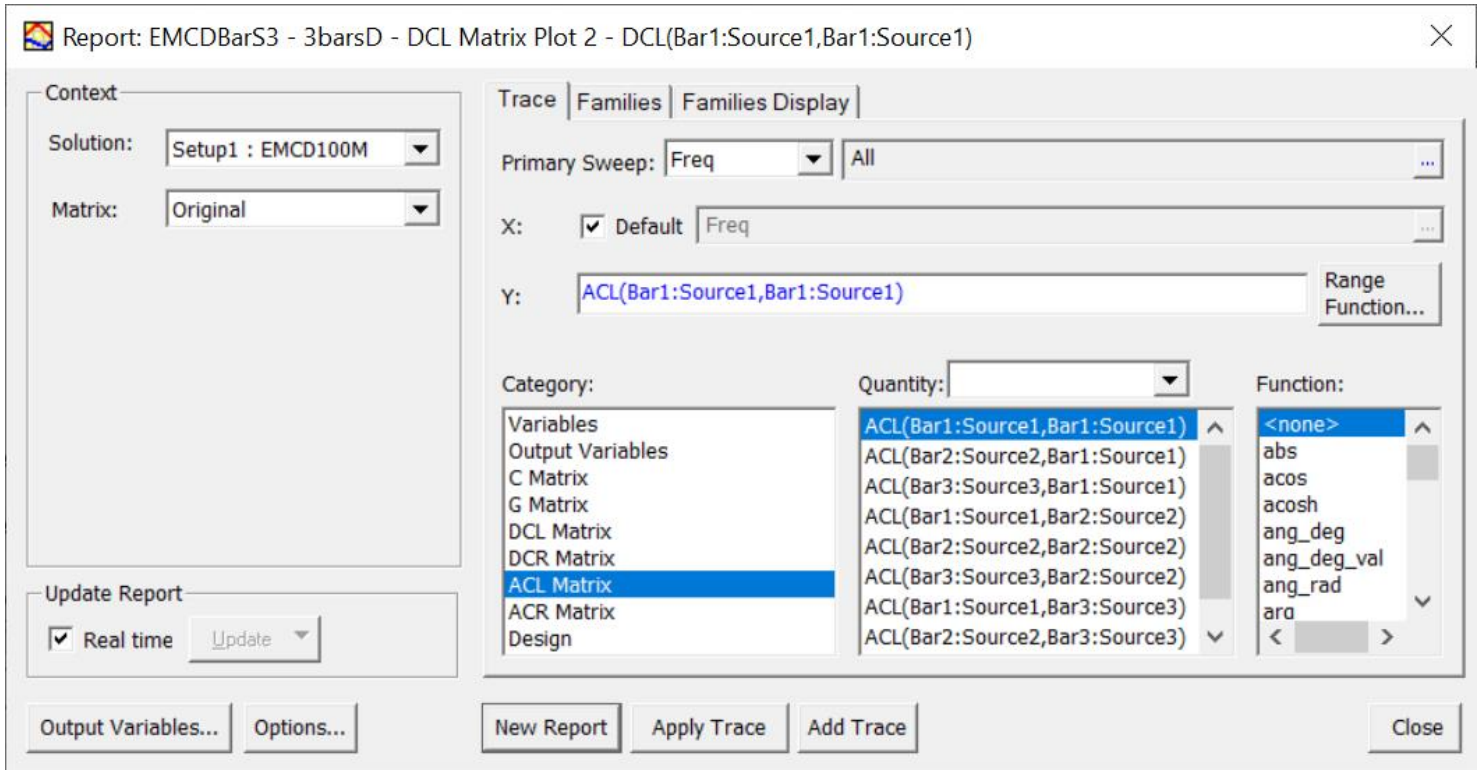




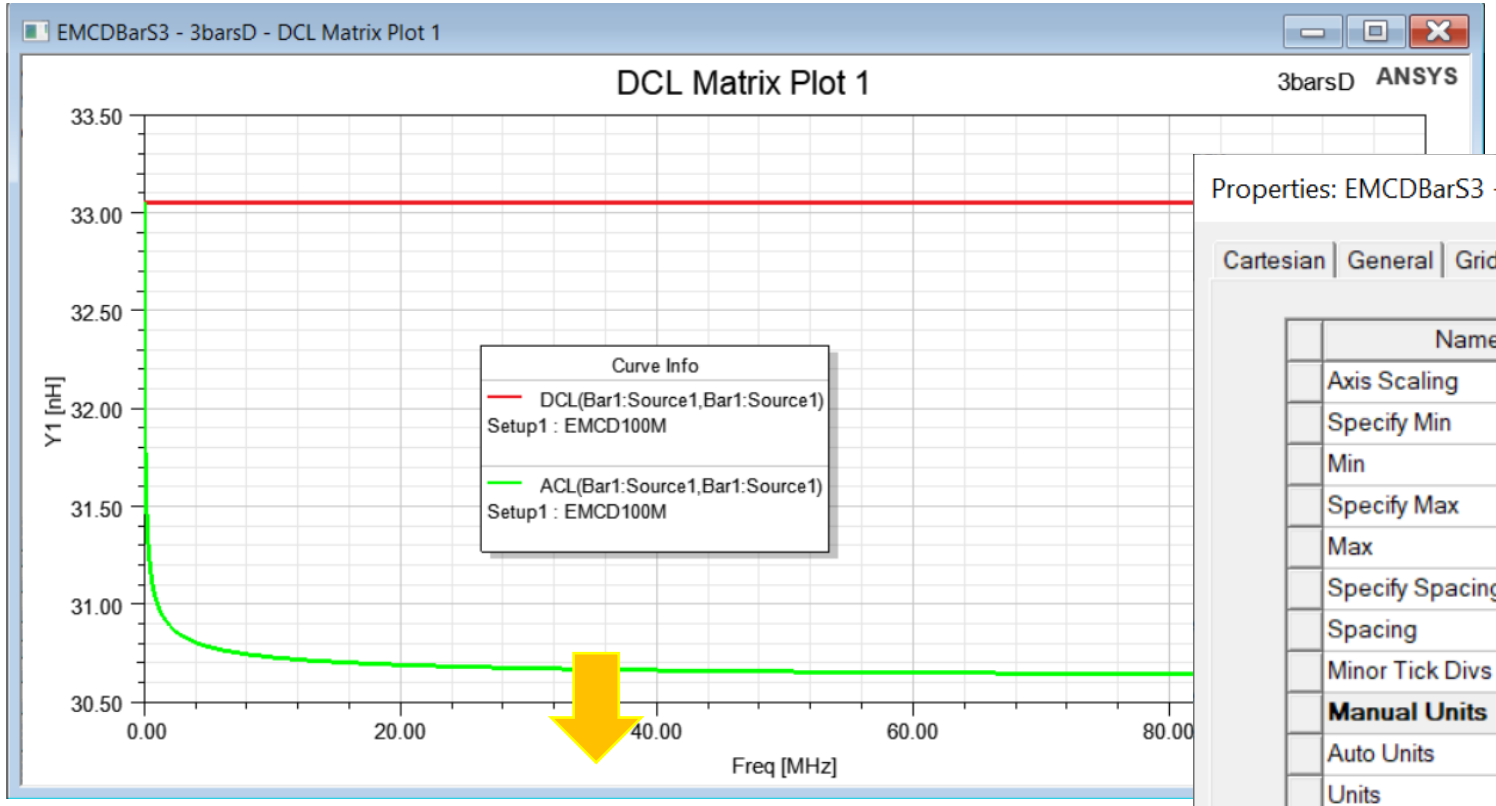
Plotting Inductance Over Frequency - EMCDBarS3

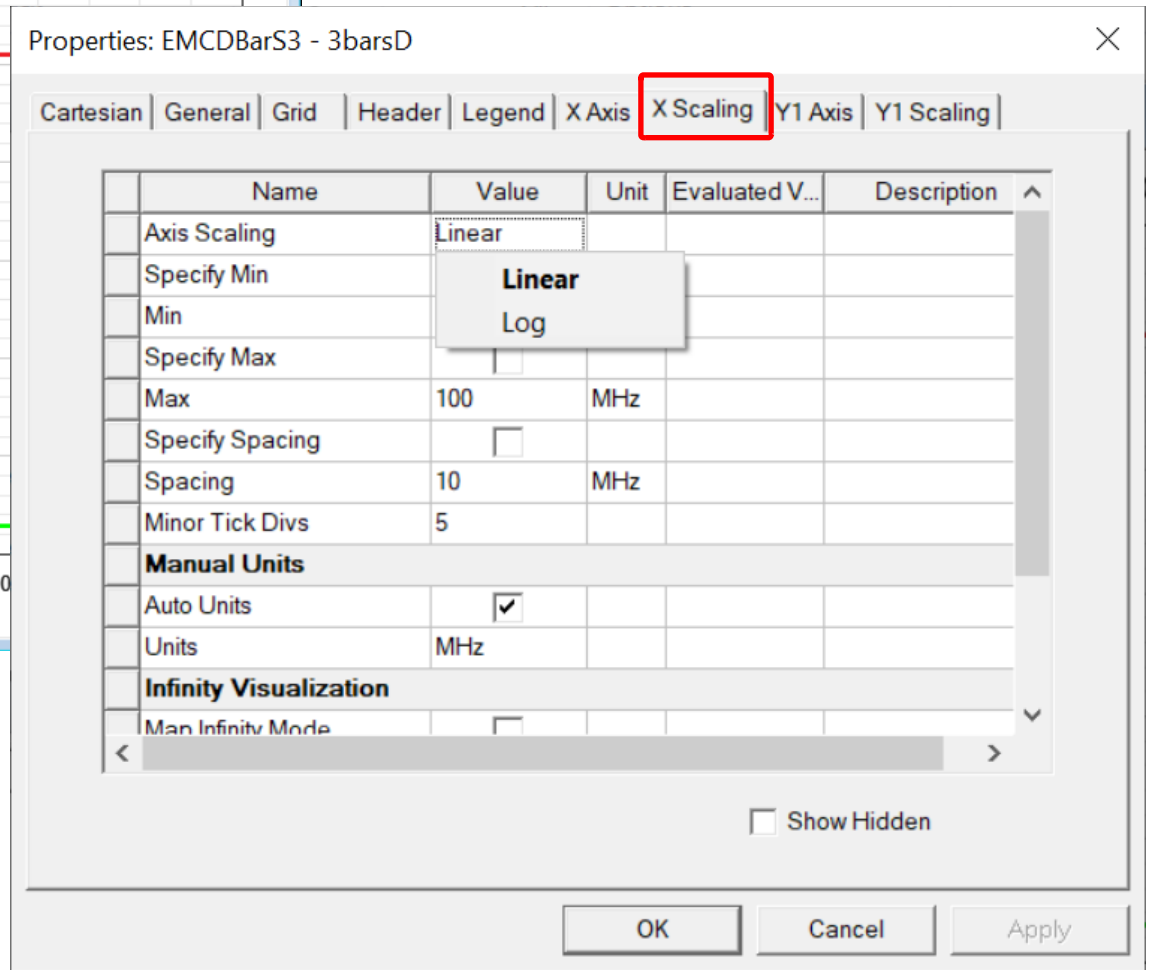
- In the ***Ribbon***, with the Results tab selected, click on the icon for ***Matrix Report*** and select ***2D*** to bring up the ***Report*** dialog window.
- In the ***Report*** window first plot ***DC Inductance***
  - Under ***Category***, select ***DCL Matrix***
  - Click ***DCL(bar1:Source1,bar1:Source1)***
  - Click ***New Report*** …keep the window open
- Next plot ***AC Inductance*** on the same plot
  - Under ***Category*** select ***ACL Matrix***
  - Click ***ACL(bar1:Source1,bar1:Source1)***
  - Click ***Add Trace***
- Click ***Close*** to close the ***Report*** window.





Initial Plot - Change X Axis Scalling to *Log*

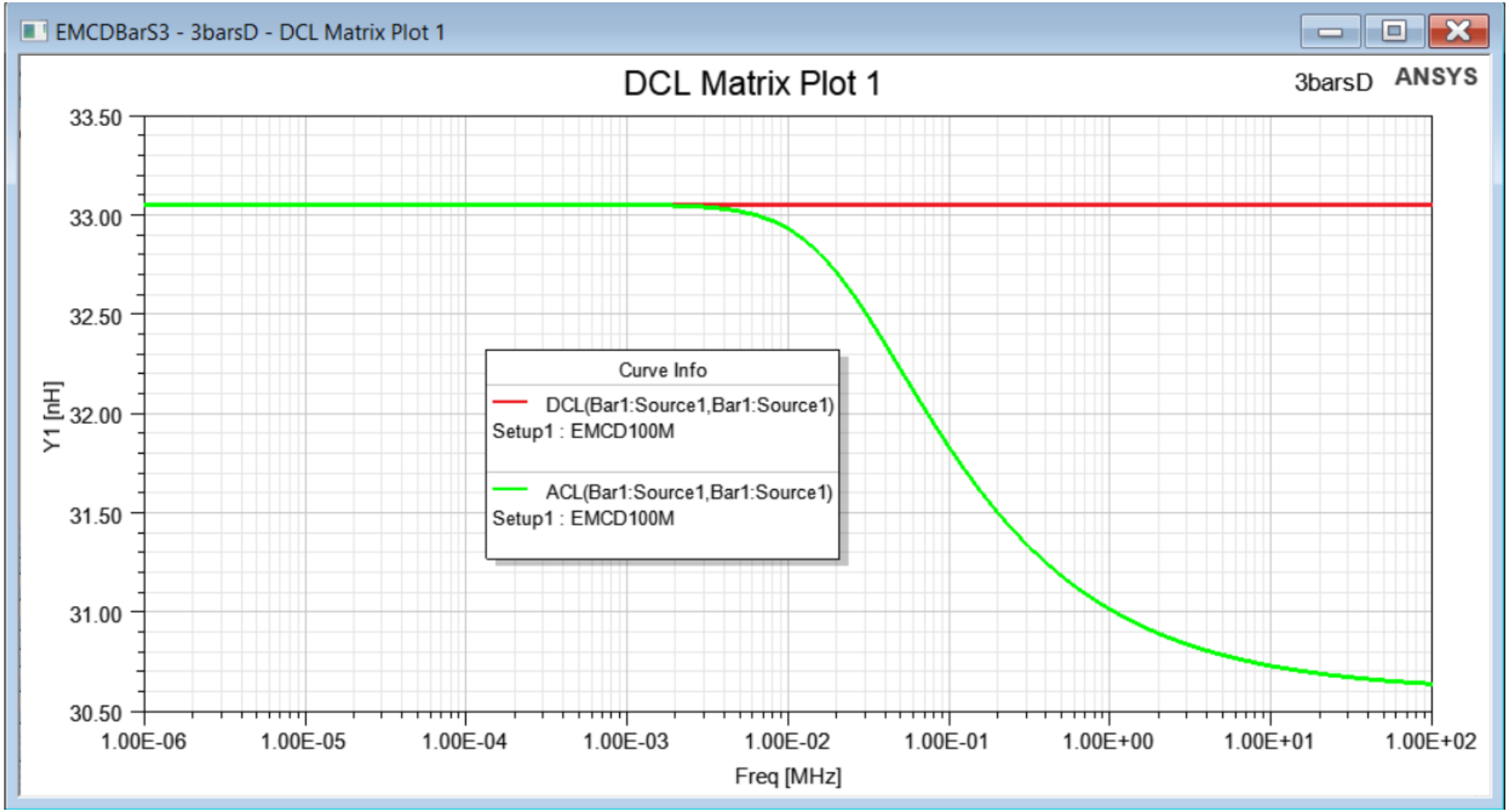




- In the initial plot of ***DCL*** and ***ACL***, double-click the X axis where the numbers appear
- Click the ***X Scaling*** tab
- Change ***Axis Scaling*** to ***Log***
- Press ***OK***

Final *DCL* & *ACL* Plot versus Frequency

As discussed with the single frequency simulation, the inductance goes down as frequency goes up.



Appendix

Electronics Desktop 3D Model Editor Keyboard Shortcuts

General Shortcuts

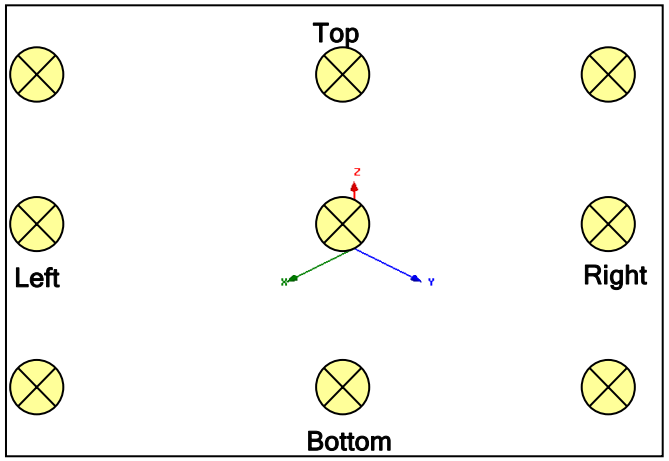
- *F1*: Help
- *F1 + Shift*: Context help
- *F4 + CTRL*: Close window
- *CTRL + C*: Copy
- *CTRL + N*: New project
- *CTRL + O*: Open...
- *CTRL + S*: Save
- *CTRL + P*: Print...
- *CTRL + V*: Paste
- *CTRL + X*: Cut
- *CTRL + Y*: Redo
- *CTRL + Z*: Undo
- *CTRL + 0*: Cascade windows
- *CTRL + 1*: Tile windows horizontally
- *CTRL + 2*: Tile windows vertically

- ***Alt + Double Click Left Mouse Button at points on screen***: Sets model projection to standard isometric projections (see diagram to the right).
- ***Alt + Double Click Right Mouse Button at points on screen***: give the nine opposite projections.

3D Modeller Shortcuts

- *B*: Select face/object behind current selection
- *F*: Face select mode
- *O*: Object select mode
- *Hold X*: Curser movement restricted to x direction
- *Hold Y*: Curser movement restricted to y direction
- *Hold Z*: Curser movement restricted to z direction
- *CTRL + A*: Select all visible objects
- *CTRL + SHIFT + A*: Deselect all objects
- *CTRL + D*: Fit view

Predefined View Angles



- ***CTRL + Left Mouse Click***: Shifts the local coordinate system temporarily
- ***SHIFT + Left Mouse Button***: Drag
- ***Alt + Left Mouse Button***: Rotate model
- ***Alt + SHIFT + Left Mouse Button***: Zoom in / out
- ***F3***: Switch to point entry mode (i.e. draw objects by mouse)
- ***F4***: Switch to dialogue entry mode (i.e. draw object solely by entry in command and attributes box.)
- ***F6***: Render model wire frame
- ***F7***: Render model smooth shaded