SI Xplorer Help

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R2
July 2025

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Table of Contents

1 - Introduction to SI Xplorer (Beta)

SI Xplorer is a standalone application based on the Layer Stackup Wizard implemented in SIwave and HFSS 3D layout. For help with Layer Stackup Wizard in the context of those applications, please consult the SIwave or HFSS 3D Layout documentation.

SI Xplorer has all of the capabilities of Layer Stackup Wizard, as well as:

- Dynamic computation and update of Z0/Zdiff value stackup parameters (signal layers only)
- Persistent view of conductor cross-section defined in the w-element panel
- Persistent view of impedance and w-element plots
- Ability to define sweeps for impedance plots
- Support for project variables
- Ability to define and edit padstacks
- Ability to define new via breakouts as independent 3D Layout designs
- EDB cell view for visualization, editing padstack instances, and defining backdrills
- Ability to run HFSS simulation setups
- Network Data Explorer workspace for visualizing results

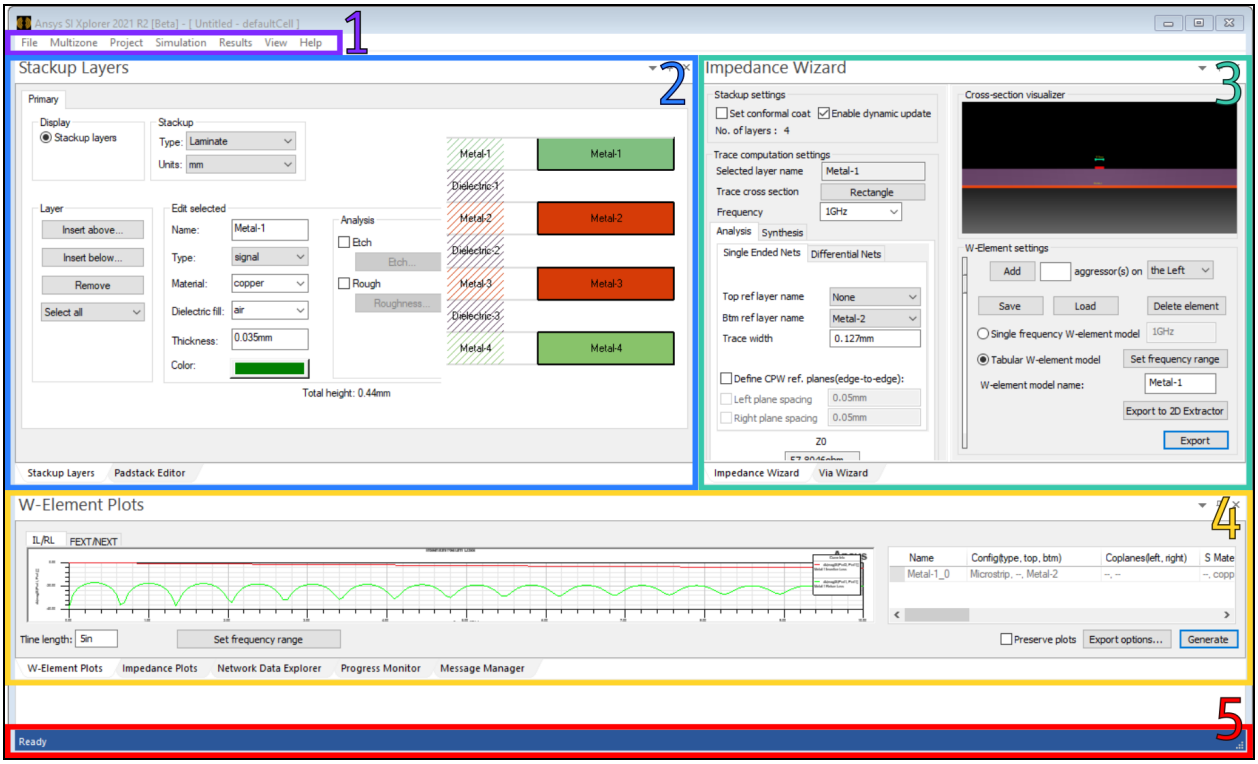
SI Xplorer also supports the use of variables.

Launching SI Xplorer

Launch SI Xplorer from its executable file `AnsysSIXplorer.exe` from the Windows or Linux installation folder:

- `.../ansys_inc/v252/AnsysEM`
- `"...\ANSYS Inc\v252\AnsysEM`

The SI Xplorer Window



The **SI Xplorer** window contains the following areas:

1. **Menu Bar** – provides access to options for importing and exporting files, defining multizone stackups, performing a simulation, viewing results, and customizing the view. See: SI Xplorer Menu Bar Options.
2. **Stackup / Padstacks** – displays a visual of the layer stackup and allows you to edit layer properties. Selecting a layer in the list also selects it for analysis in the Impedance Wizard.
3. **Impedance / Via Wizard** – use the tabs to select either the Impedance Wizard or Via Wizard.
   - The **Impedance Wizard** contains a calculator for impedance, differential impedance, and trace width/pitch. It also allows you to change w-element settings and export w-element information. Changes made in the Impedance Wizard display in the **Plots and Messages** area as **W-Element Plots** and **Impedance Plots**.
   - The **Via Wizard** allows you to define, visualize, and edit padstacks.
4. **Plots and Messages** – depending on which tab is selected, displays w-element plots, impedance plots, the Network Data Explorer, a progress monitor, or the message manager. The plots update dynamically as you make inputs, and can be exported. See: Viewing and Exporting Plots in SI Xplorer.

> **Note:**
>
> Impedance Plots only update dynamically when **Enable Dynamic Update** is selected in the **Impedance Wizard** area.

5. **Status Bar** – displays a progress bar while actions are in progress.

You can resize each area as appropriate by dragging its borders. Each area is explained in further detail in the topics below.

Loading Files into SI Xplorer

SI Xplorer can load an HFSS 3D Layout Design (*.aedt) or EDB Definition file (*.def).

> **Important:**
>
> SI Xplorer can only open *.aedt projects that were created in SI Xplorer.

From the Command Line

To open SI Xplorer with a file already loaded, pass an argument to that file's path. For Example:

```
C:\Program Files\ANSYS Inc\v252\AnsysEM\Win64\AnsysSIXplorer.exe
"C:\Users\jdoe\Documents\SAMPLEFILES\MyFile.aedt"
```

From the GUI

SI Xplorer supports the following design types:

- Ansys Electronics Desktop (*.aedt)
- EDB Definition File (*.def)

Additionally, it supports the following stackup file types:

- IPC2581 Design Files (*.xml, *.cvg)
- SIwave Control File (*.stk)
- XML Control File (*.xml)

To open an *.aedt file:

- Click **File** > **Open Project**.

To load any of the other file types:

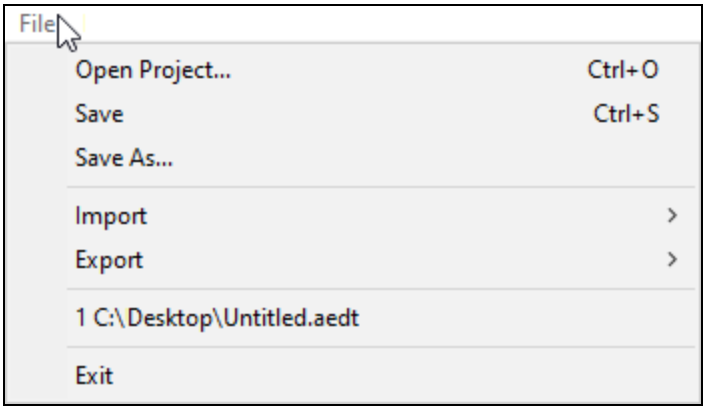
- Click **File** > **Import** > [**File Type**].

SI Xplorer Menu Bar Options

The SI Xplorer menu bar contains the following submenus:

- File
- Multizone
- Project
- Simulation
- Results
- View
- Help

File Submenu

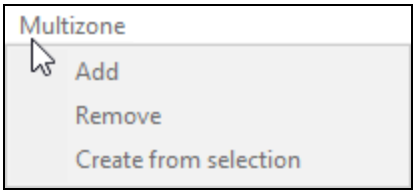


The **File** menu contains the following options:

- **Open Project** – allows you to open Ansys Electronics Desktop files (*.aedt).
- **Save** – saves the current *.aedt file.
- **Save As** – allows you to specify a name and location to save a new *.aedt file.
- **Import** – allows you to import an EDB definition file (*.edb), IPC2581 design files (*.xml, *.cvg), or layer Stackup files (*.xml, *.stk).
- **Export** – allows you to export to EDB, IPC2581, Stackup XML, Stackup STK, or Stackup CSV.
- **Recent File** – displays a list of recently opened files, if any exist.
- **Exit** – closes SI Xplorer.

Multizone Submenu

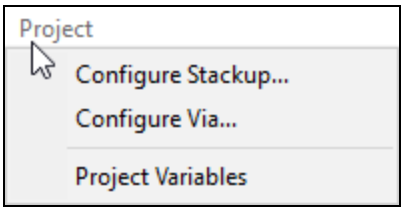
Multizone menu options become available when you select the Multizone stackup type in SI Xplorer. See: Modifying the Layer Stackup using SI Xplorer.



The **Multizone** menu includes the following options:

- **Add** – Adds a new multizone in a new sheet.
- **Remove** – Removes a multizone.
- **Create from Selection** – Creates a new multizone based on the selection.

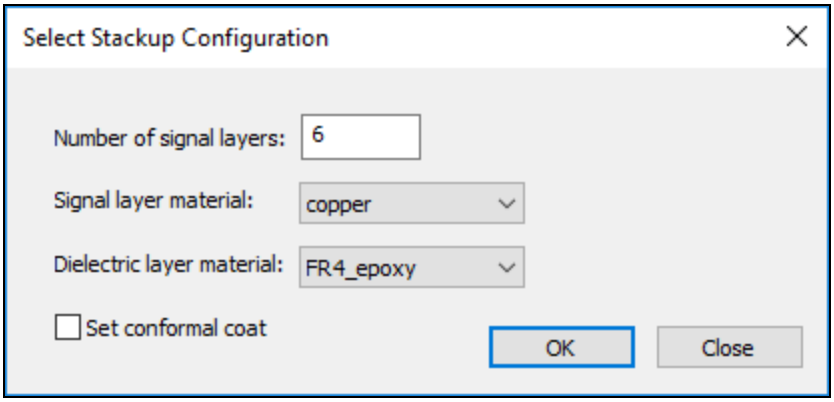
Project Submenu



The **Project** submenu contains options to **Configure Stackup**, **Configure Via**, or configure **Project Variables**.

Configure Stackup

The **Configure Stackup** menu option launches the **Select Stackup Configuration** window:



You can set the following default options:

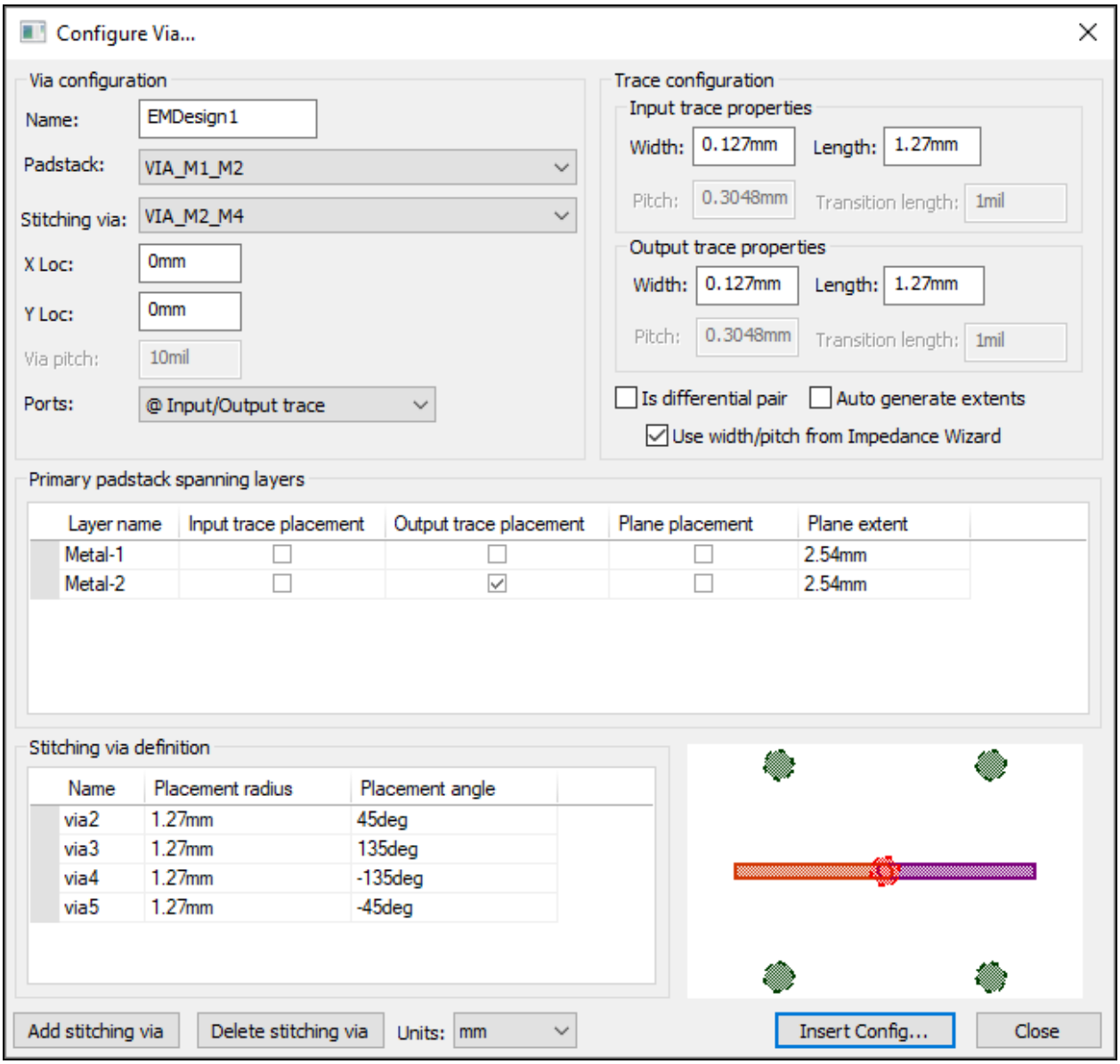
- **Number of signal layers** – enter a value.
- **Signal layer material** – select a material from the drop-down menu.

- **Dielectric layer material** – select a material from the drop-down menu.
- **Set conformal coat** – select the check box to enable.

Click **OK** to save the default options.

Configure Via

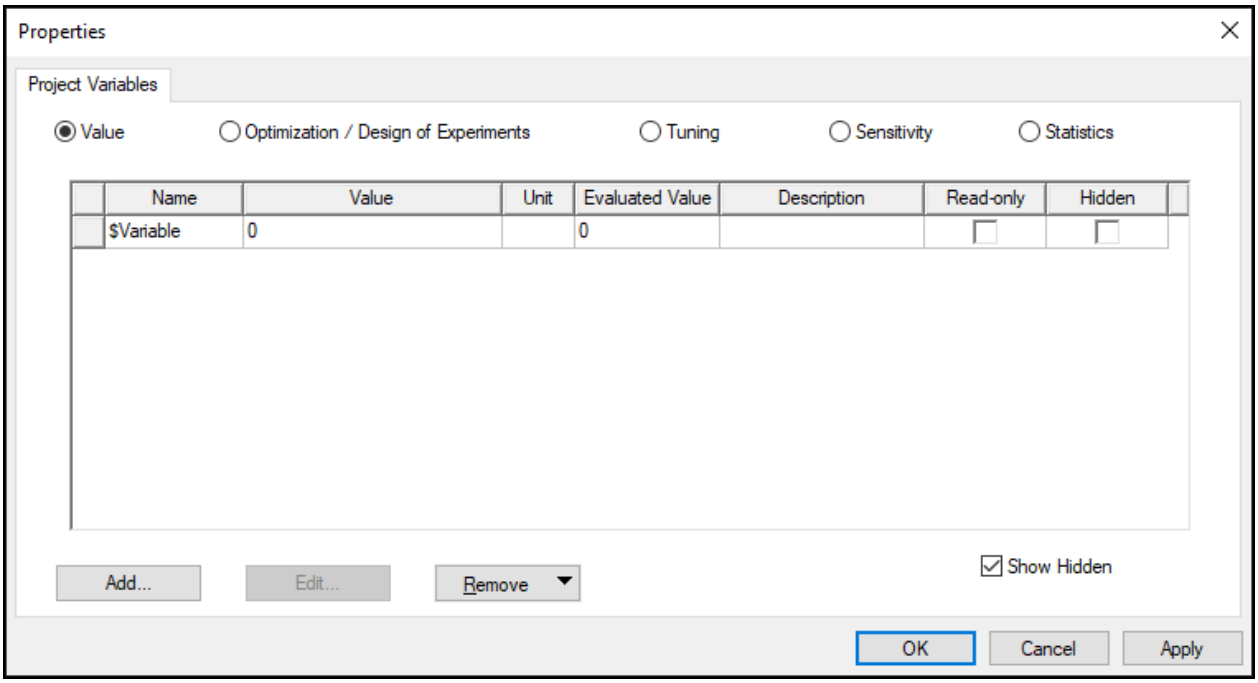
The **Configure Via** menu option launches the **Configure Via** window:



See: Configuring Vias.

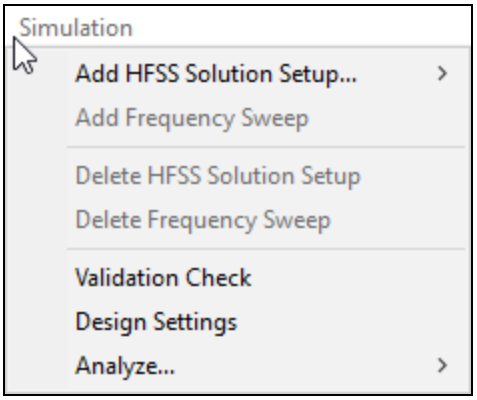
Project Variables

The **Project Variables** menu option launches the **Properties** window, on the **Project Variables** tab:



See: Setting Project Variables.

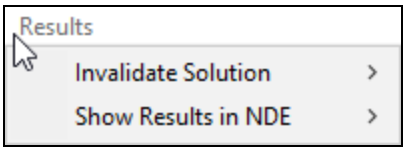
Simulation Submenu



The **Simulation** menu includes the following options:

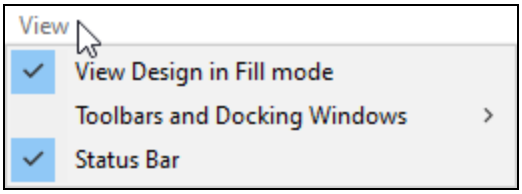
- **Add HFSS Solution Setup** – Allows you to add either an **Auto** or **Advanced** HFSS solution setup.
- **Add Frequency Sweep** – Allows you to add a Frequency Sweep to an existing HFSS solution setup.
- **Delete HFSS Solution Setup** – Removes a solution setup.
- **Delete Frequency Sweep** – Removes a Frequency Sweep.
- **Validation Check** – Verifies that the design is ready for simulation.
- **Design Settings** – Allows you to change design settings for simulation.
- **Analyze** – Runs either the **Active Setup** or **All Setups**.

Results Submenu



The **Results** menu allows you to invalidate solution results or show the results in Network Data Explorer. Either action can be applied to the **Active Setup** or to **All Setups**.

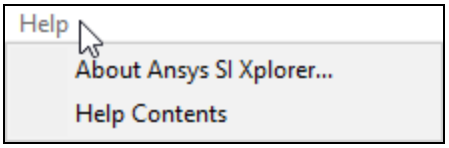
View Submenu



The **View** menu contains the following options:

- **View Design in Fill Mode** – when selected, the design displays as solid. When deselected, the design displays as wireframe.
- **Toolbars and Docking Windows** – contains the following display options:
  - **Reset Workspaces** – allows you to return SI Xplorer to its default view.
  - **Auto Toggle Workspaces** – allows you to synchronize workspaces. When you click on the Impedance Wizard, the Stackup Layers and W-element Plots workspaces become active. When you click the Via Wizard, the Padstack Editor and Network Data Explorer workspaces become active.
- **Status Bar** – toggles the status bar at the bottom of SI Xplorer.

Help Submenu



The **Help** menu allows you to retrieve software information by clicking **About Ansys SI Xplorer**.

You can also click **Help Contents** to navigate to the SI Xplorer online help.

Setting SI Xplorer Options

The SI Xplorer menu bar can be used to set a number of options.

The following topics explain these in further detail:

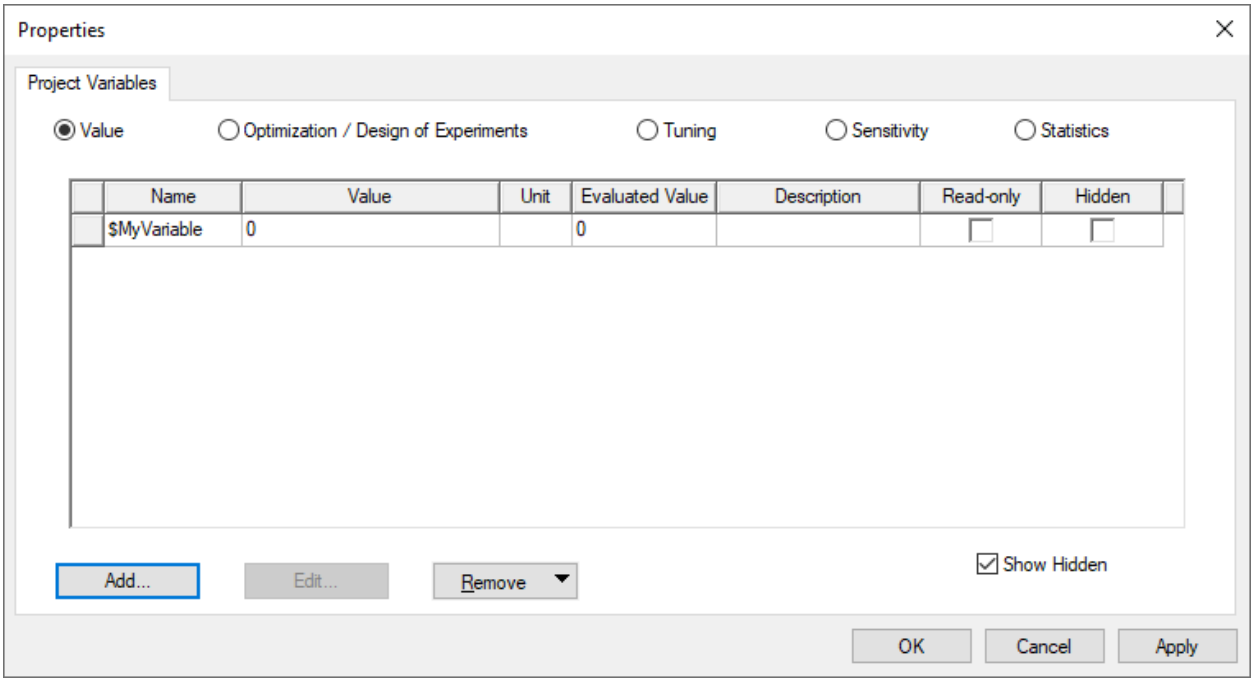
- Configuring Vias
- Setting Project Variables

Setting Project Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter. You can assign a variable to any dimension, material property, or output value in SI Xplorer. Variables are useful if you expect to change a parameter often, or expect to use the same parameter value often.

Access project variables in SI Xplorer by clicking **Project** > **Project Variables**.

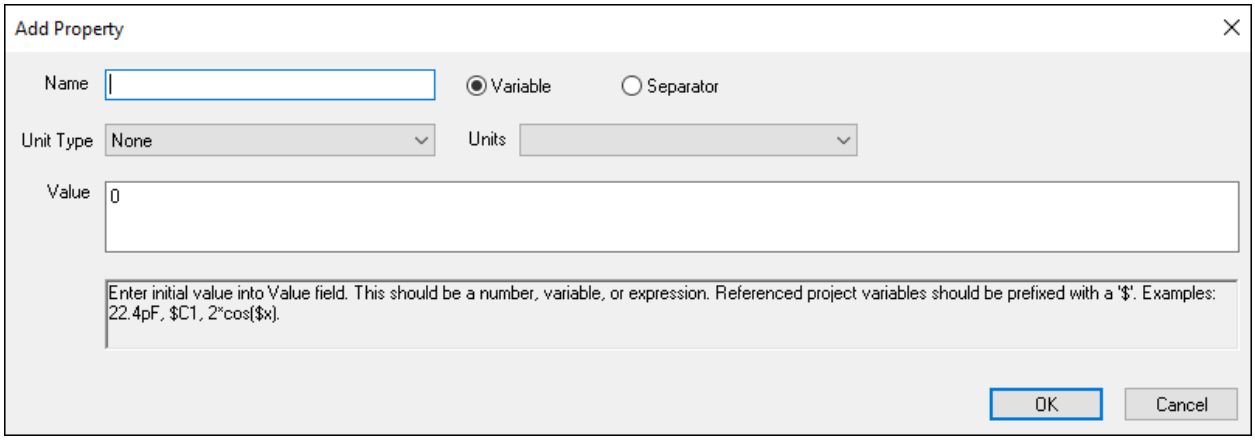
The **Properties** window appears, on the **Project Variables** tab:



Adding Project Variables

1. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



2. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_).
   - **Variable Type** – use the **Variable** and **Separator**radio buttons to select the variable type.

Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name only. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.

- **Unit Type** – for Variables, use the drop-down menu to select a type from the list (e.g., Charge, Density, Energy, et cetera). “None” is the default.

  When you select a Unit Type, the choices in drop-down menu for the Units text box adapt to that unit type. For example, selecting Length as the Unit Type causes the Unit menu to show a range of metric and english units for length. Similarly, if you select the Unit Type as Resistance, the Units drop down lists a range of standard Ohm units.
- **Units** – for Variables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends.

  > **Warning:**
  >
  > If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

3. Click **OK**.

   The new variable appears in the list. You can sort project variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.
4. If appropriate, use the check boxes to designate a variable as **Read-only** or **Hidden**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
5. Click **Apply** to apply changes.
6. Click **OK** to exit the window.

Editing Project Variables

To edit a project variable:

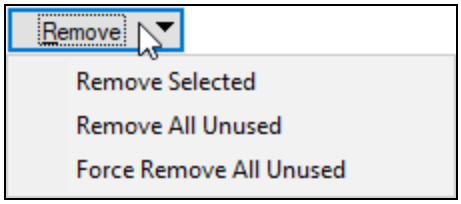
1. Access **Project Variables**.
2. Select the variable you want to edit and click **Edit**.
3. Change the properties as appropriate and click **OK**.

Removing Project Variables

To remove a project variable:

1. Access **Project Variables**.
2. Click **Remove**.

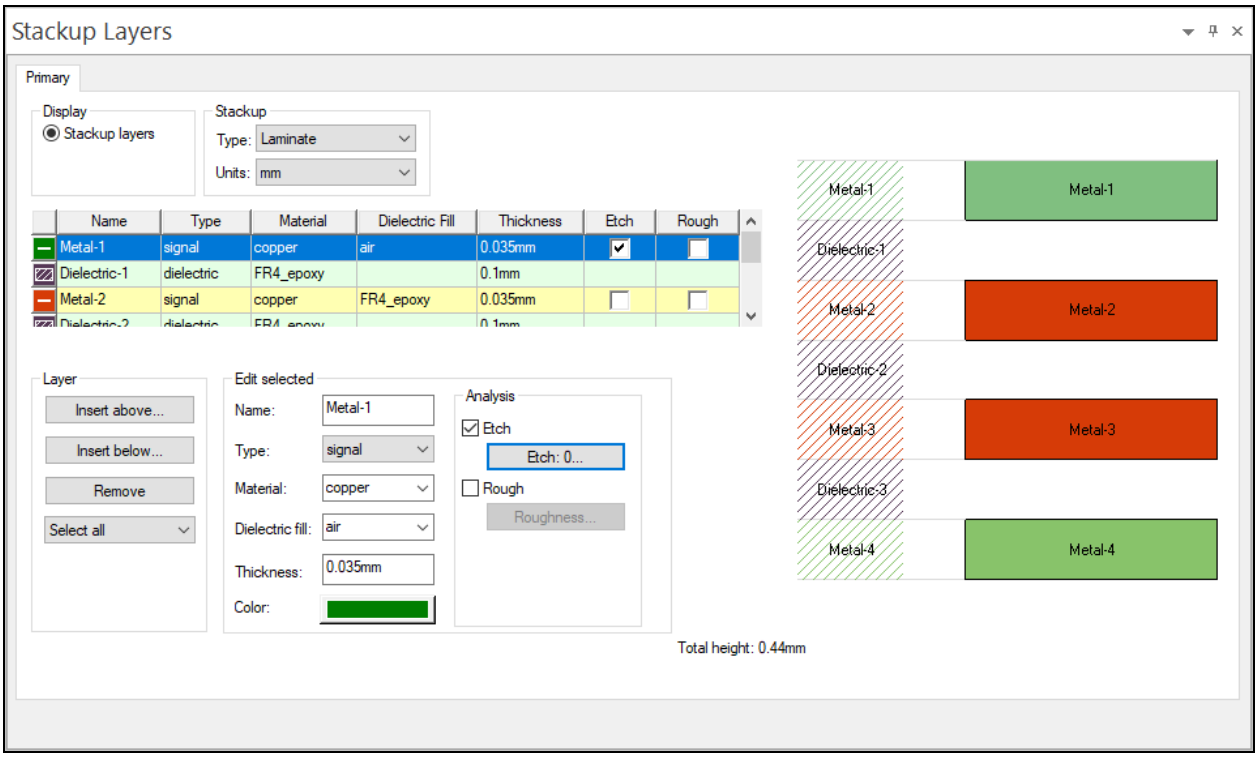
   A drop-down menu appears.



3. Remove one or more variables:
   - To remove a specific variable, ensure it is selected and click **Remove Selected**.
   - To remove all unused variables, click **Remove All Unused**.
   - To force the removal of all unused variables, including those in the project's undo/redo history, click **Force Remove All Unused**.

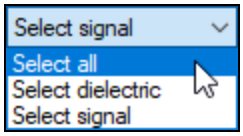
2 - Modifying the Layer Stackup

The **Stackup Layers** area of SI Xplorer allows you to add and remove layers, edit layer properties, edit HFSS etch factor, and change surface roughness.

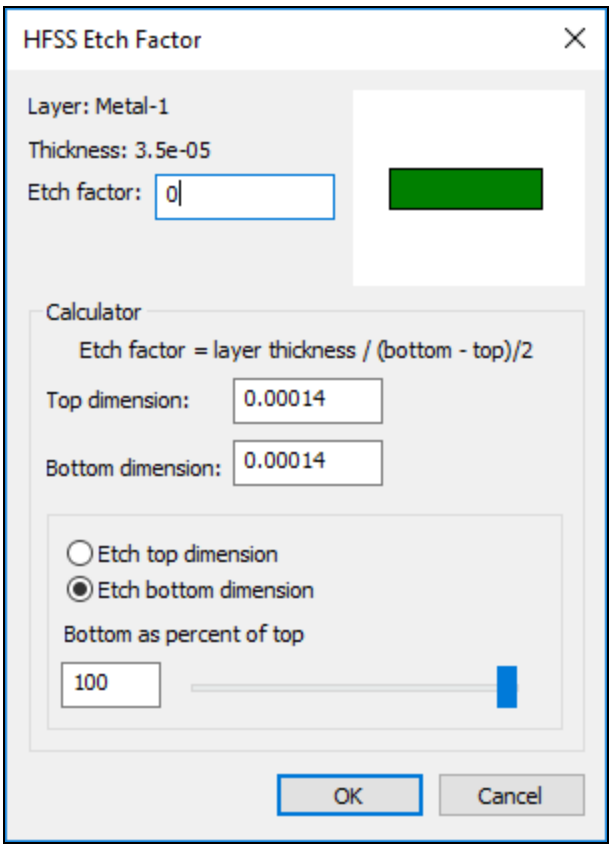


The right side displays an image of the layer stack. The left side displays options for altering it:

- **Stackup** – use the drop-down menus to change the stackup Type or Units. See: Multizone Stackup below.
- **Layers List** – allows you to select rows, highlighting them in the stack and selecting them for impedance or w-element analysis. Each cell in the table can be edited, allowing you to quickly change a layer's name, type, material, fill, and thickness. You can also use the check boxes to enable HFSS etch factor or surface roughness for a layer.
- **Layer** – contains options for inserting new layers and removing existing layers. The drop-down menu allows you to select all layers, select dielectric layers, or select signal layers. You can then bulk edit these layers' properties in the **Edit Selected** area.

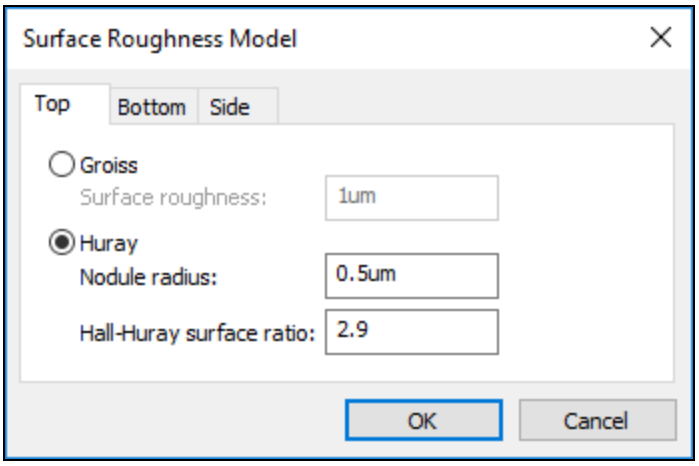


- **Edit Selected** – allows you to edit a single layer's properties, or the properties of multiple layers selected using the layer selection drop-down menu.
  - **Etch** – when **Etch** is selected for a layer, click the **Etch** button to open the **HFSS Etch Factor** window:



  See: Setting HFSS Etch Factor.

  - **Rough** – when **Rough** is selected for a layer, click the **Roughness** button to open the **Surface Roughness Model** window:



For the layer's **Top**, **Bottom**, and **Side**, select the roughness model and enter an appropriate parameters.

The **Groiss** model used within Ansys Electronics products is a modified version of the original Groiss model. It is causal.

The **Huray** model is also a causal model. See: Huray Surface Roughness Model.

Note:

HFSS Etch Factor and Surface Roughness can be edited when one or more signal layers is selected. Selecting dielectric layers disables these options.

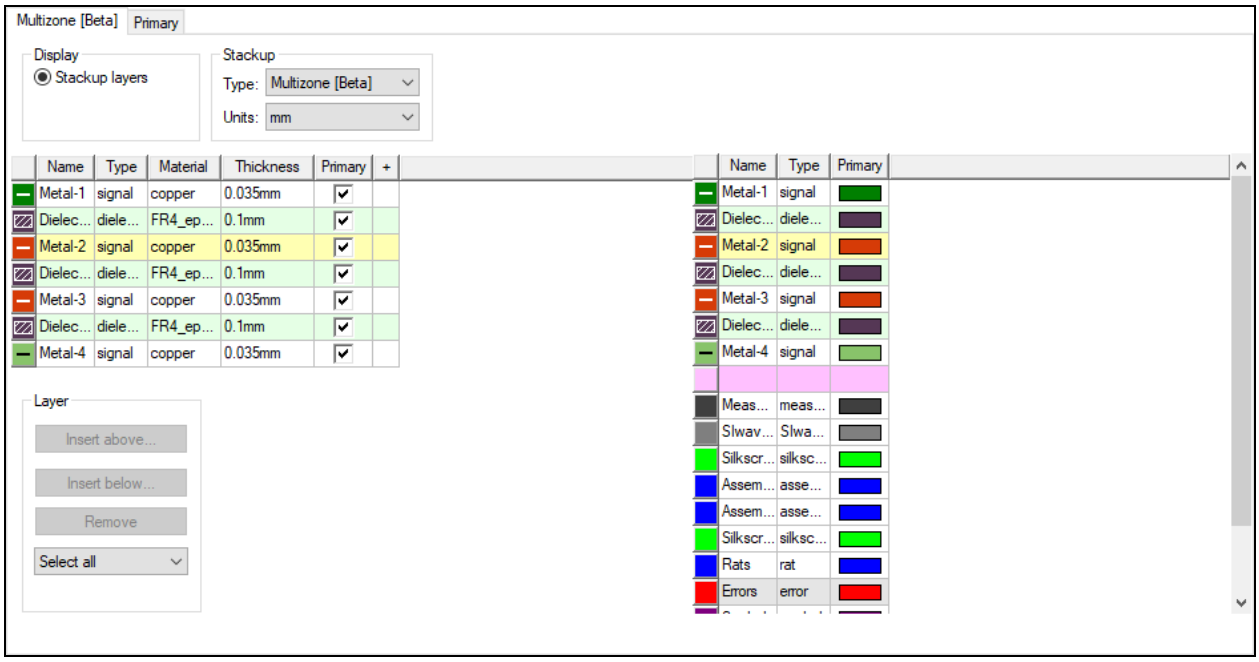
Multizone Stackup (Beta)

By default, the **Stackup Layers** area displays one tab: the **Primary** tab. SI Xplorer can also divide the layer stack into multiple zones.

To use this feature:

1. Use the **Type** drop-down menu to select **Multizone**.

   The **Stackup Layers** area updates to show the **Multizone** tab:

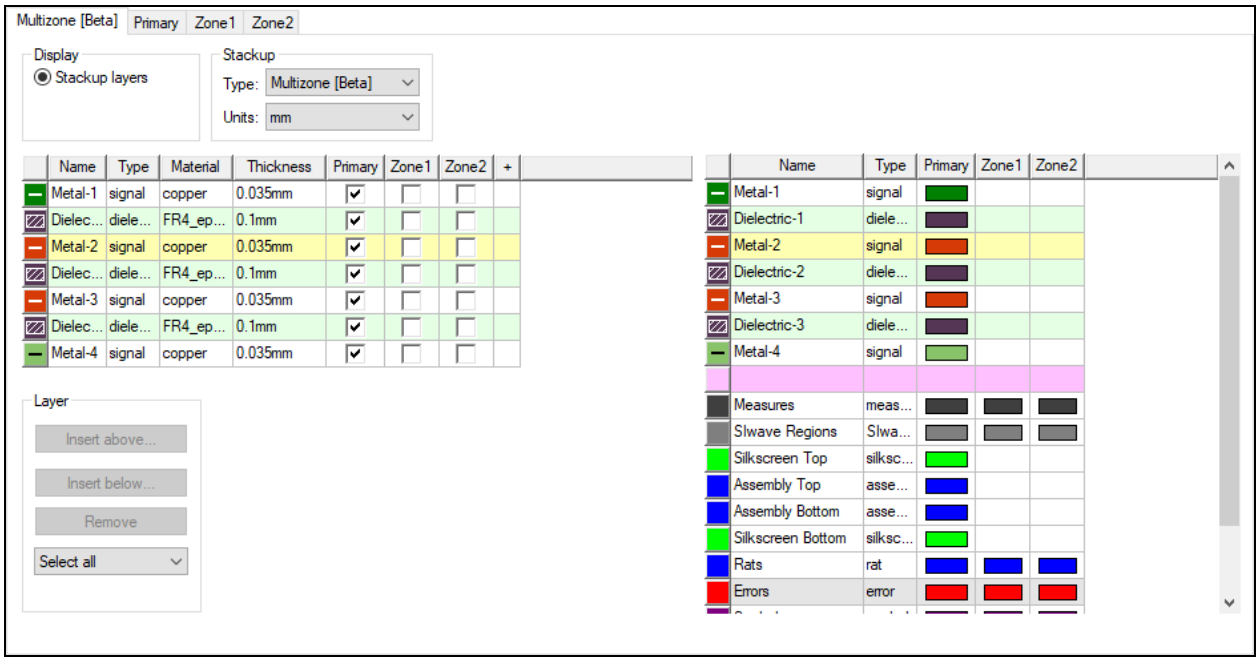


2. Add or remove zones using the **Multizone** menu on the menu bar.

   A **Zone#** tab appears for each new zone:

   The **Multizone** tab updates to include the new zones:



The left side allows you to assign layers to zones. The right side contains a color key.

3. Use the **Zone#** check boxes to assign layers to zones.

   One assigned, layers appear in their zone tab(s). Each **Zone#** tab looks and functions like the **Primary** tab.

Note:

By default, **Enable Dynamic Update** is selected in the **Impedance Wizard**.

When any change is made in the **Layer Stackup** window:

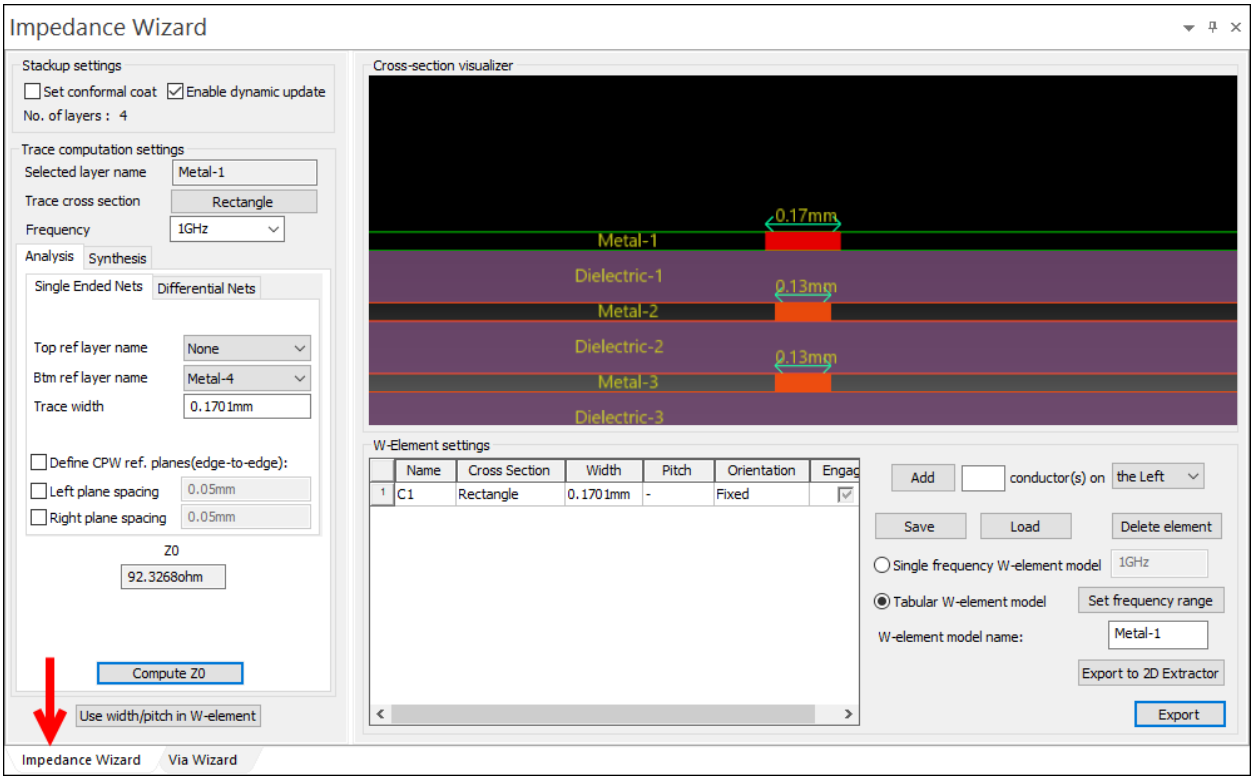
- The design updates.
- Impedance and differential impedance are recalculated.
- W-element plots are recalculated and replotted.

To disable these recalculations, deselect **Enable Dynamic Update**.

3 - Using the Impedance Wizard

The **Impedance Wizard** displays by default when SI Xplorer is opened.

If it has been closed, select **View** > **Toolbars and Docking Windows** > **Reset Workspaces**. Then, ensure the **Impedance Wizard** tab is selected:



After you have configured stackup layers, the **Impedance Wizard** allows you to:

- Calculate Impedance
- Calculate Differential Impedance
- Calculate Trace Width and Pitch
- Perform W-element Analysis

Additionally, calculations made using the Impedance Wizard can be used to:

- View Impedance Plots
- View IL/RL and FEXT/NEXT Plots

Note:

By default, **Enable Dynamic Update** is selected in the **Impedance Wizard**.

When any change is made in the **Layer Stackup** window:

- The design updates.
- Impedance and differential impedance are recalculated.
- W-element plots are recalculated and replotted.

To disable these recalculations, deselect **Enable Dynamic Update**.

Calculating Impedance (Z0)

Using SI Xplorer's **Impedance Wizard**, you can configure and perform an impedance calculation.

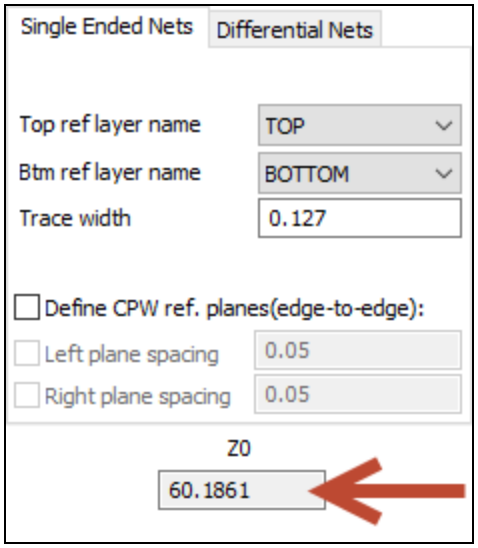
Prerequisites:

- Use the **Stackup Layers** pane to select a layer. The **Selected Layer Name** field displays the currently selected layer.
- Use the **Trace Cross Section** button to open the HFSS Etch Factor window.
- Use the **Frequency** drop-down menu to select the frequency used for impedance calculations.

To calculate impedance:

1. Select the **Analysis** tab.
2. Select the **Single Ended Nets** tab.
3. Select your reference layers.
4. From the **Trace Width** field, enter a trace width.
5. If you wish to define coplanar waveguide reference planes, select **Define CPW ref. planes (edge-to-edge)** to enable additional options.
   - You may select **Left plane spacing**, **Right plane spacing**, or both.
   - For W-element export, selecting **Left plane spacing** means that aggressors may only be added to the right of a victim trace; any traces to the left are removed. Selecting **Right plane spacing** means that aggressors may only be added to the left of a victim trace; any traces to the right are removed.
6. Click **Compute Z0**.

7. The **Z0** field updates to show the result.



Calculating Differential Impedance (Zdiff)

Using SI Xplorer's **Impedance Wizard**, you can configure and perform a differential impedance calculation.

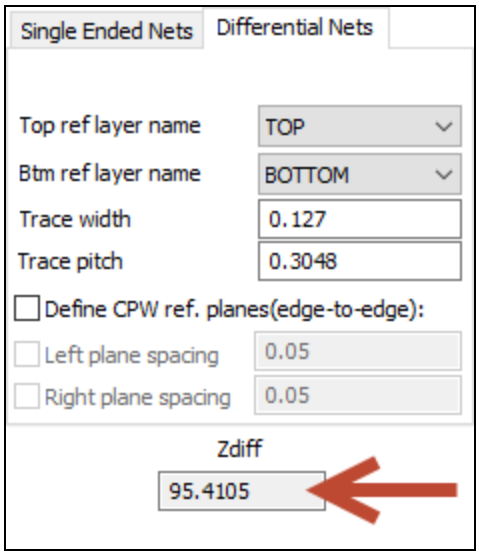
Prerequisites:

- Use the **Stackup Layers** pane to select a layer. The **Selected Layer Name** field displays the currently selected layer.
- Use the **Trace Cross Section** button to open the HFSS Etch Factor window.
- Use the **Frequency** drop-down menu to select the frequency used for differential impedance calculations.

To calculate differential impedance:

1. Select the **Analysis** tab.
2. Select the **Differential Nets** tab.
3. Select your reference layers.
4. From the **Trace Width** and **Trace Pitch** fields, enter a trace width and pitch.
5. If you wish to define coplanar waveguide reference planes, select **Define CPW ref. planes (edge-to-edge)** to enable additional options.
   - You may select **Left plane spacing**, **Right plane spacing**, or both.
   - For W-element export, selecting **Left plane spacing** means that aggressors may only be added to the right of a victim trace; any traces to the left are removed. Selecting **Right plane spacing** means that aggressors may only be added to the left of a victim trace; any traces to the right are removed.

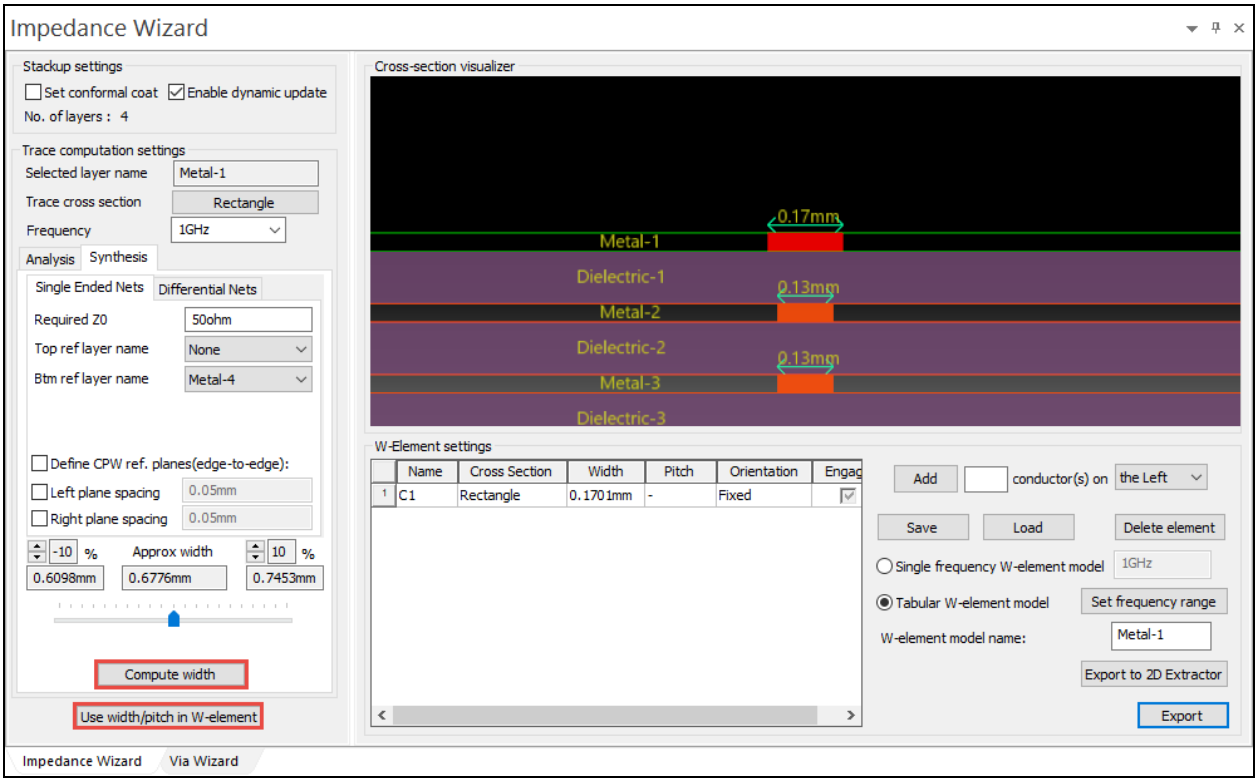
6. Click **Compute Zdiff**.
7. The **Zdiff** field updates to show the result.



Calculating Trace Width and Pitch

Using SI Xplorer's **Impedance Wizard**, you can configure and perform a trace width or trace pitch calculation.

You can then push that information to the W-Element Settings area for analysis.



Prerequisites:

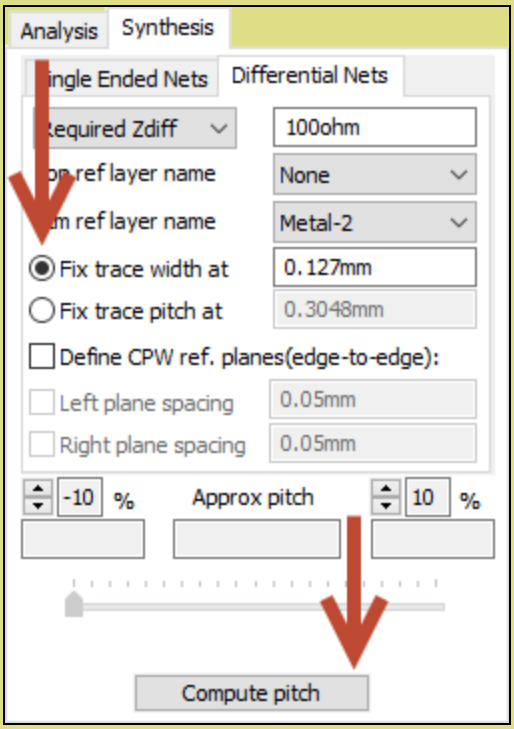
- Use the **Stackup Layers** pane to select a layer. The **Selected Layer Name** field displays the currently selected layer.
- Use the **Trace Cross Section** button to open the HFSS Etch Factor window.
- Use the **Frequency** drop-down menu to select the frequency used for trace width calculations.

To calculate trace width or pitch:

1. Select the **Synthesis** tab.
2. Select the **Single Ended Nets** or **Differential Nets** tab.
3. Use the drop-down menus to select the required impedance or differential impedance (in ohms), as well as reference layers.
4. If you are using the **Differential Nets** tab, select *either* **Fix trace width at** or **Fix trace pitch at** and enter a value.

Important:

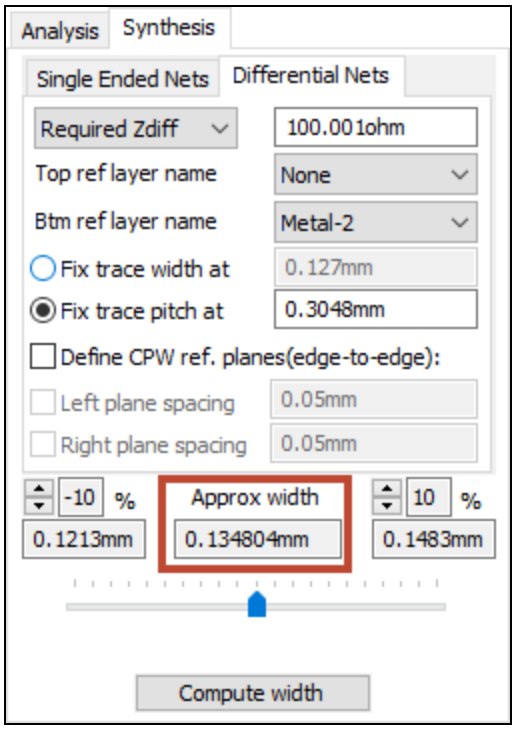
By default, **Fix Trace Pitch** is selected. Selecting **Fix Trace Width** changes the calculator so that it computes pitch instead of width.



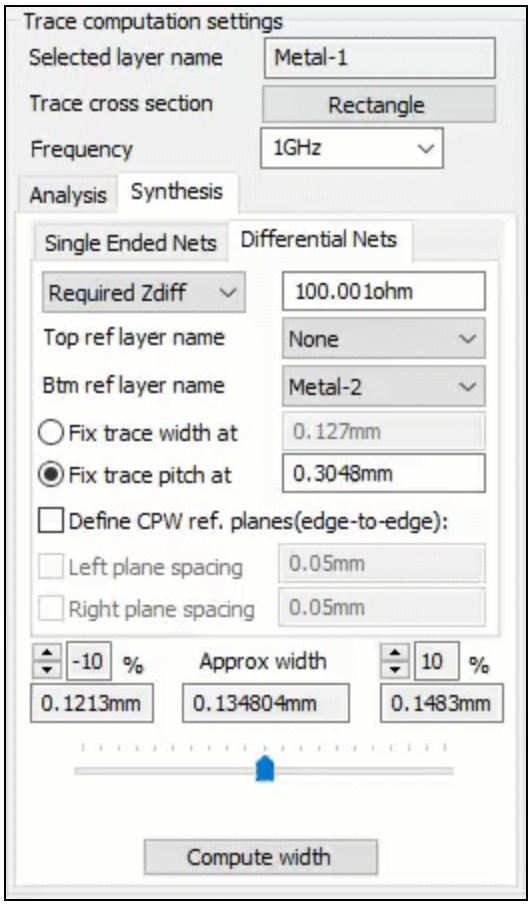
5. If you wish to define coplane wave reference planes, select **Define CPW ref. planes (edge-to-edge)** to enable additional options.
   - You may select **Left plane spacing**, **Right plane spacing**, or both.
   - For W-element export, selecting **Left plane spacing** means that aggressors may only be added to the right of a victim trace; any traces to the left are removed. Selecting **Right plane spacing** means that aggressors may only be added to the left of a victim trace; any traces to the right are removed.
6. To perform the calculation, click **Compute width**.

   The green progress bar shows the calculation's progress.

   When the calculation is complete, the **Approx width** area updates to show the calculated result.



7. Use the slider above the progress bar to update trace width as well as the required Zdiff value.



To use a calculated width/pitch in w-element analysis:

- Click **Use width/pitch in w-element**:

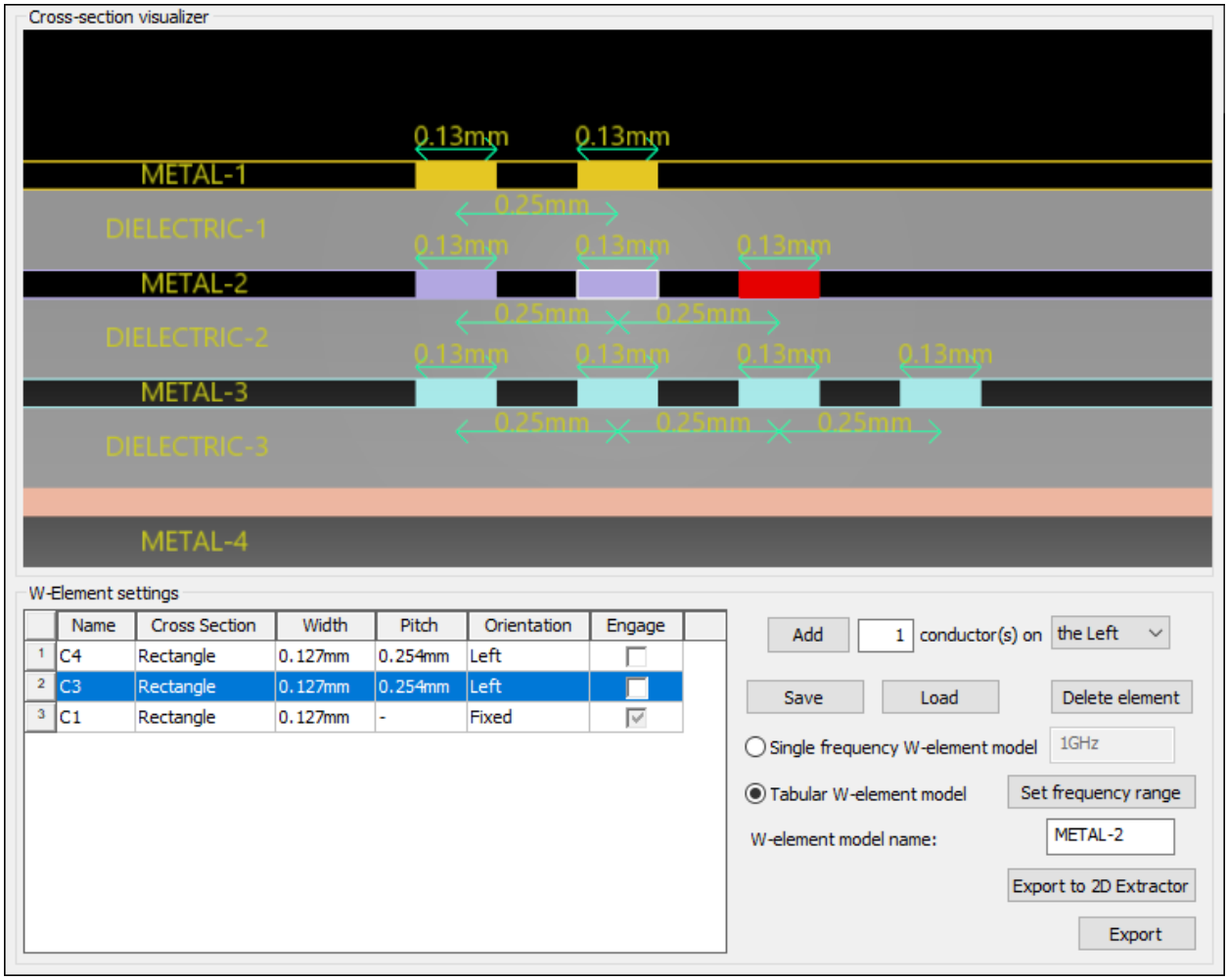
W-element Analysis

W-elements are a distributed model for transmission lines used in certain HSPICE-compatible circuit simulators. For designs containing traces and planes, SI Xplorer can export the W-element in either single frequency or tabular format.

After you have used SI Xplorer's **Impedance Wizard** to calculate impedance, differential impedance, or trace width/pitch, the **Cross-section Visualizer** area allows you to set up aggressor traces around a victim trace and export the corresponding w-element.

To analyze and export the W-element:

1. Click an element in the **Cross-section visualizer** or select it from the list in the **W-Element Settings** area.



The following options are available:

- **Add** – Enter the number of aggressors to add, and use the drop-down menu to select whether to add them on the **right**, **left**, or on **both sides**. Then click **Add**.

> **Note:**
>
> If you previously enabled CPW settings during trace width, impedance, or differential impedance calculation, your options will be limited.
>
> - If you selected **Left plane spacing**, aggressors may only be added to the right of a victim trace.
> - If you selected **Right plane spacing**, aggressors may only be added to the left of a victim trace.

- **Delete Element** – Click to remove an element.
- **Single Frequency / Tabular Model** – Select **Single frequency** and enter the frequency in Hz, or select **Tabular** and click **Set frequency range** to set up a range of frequencies.
- **W-Element Model Name** – If appropriate, enter a new model name for the w-element.
- **Export to 2D Extractor** – generates a VB script to export the selected layer's cross-section to 2D Extractor. Microstrips, stripline and conformal coats are supported. Generating this script launches a prompt asking if you want to run the script in Electronics Desktop. Consult the Q3D Extractor/2D Extractor help for instructions on running an analysis in 2D Extractor.

You can also double-click a value within the **W-Element Settings** list to change width and pitch.

2. Make your appropriate selections, then click **Export**.

   The **Select Spice File to Export** window appears.

   > **Note:**
   >
   > If you selected **Tabular W-element model** but did not specify the frequency range, the **Frequency Sweep Dialog** appears. You must specify the frequency range before you can save.

3. Browse to a save location, name your file, and click **Save**.

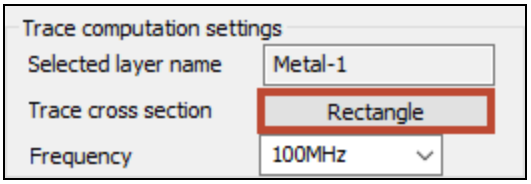
   The file is saved in Spice (*.sp) format.

Setting HFSS Etch Factor

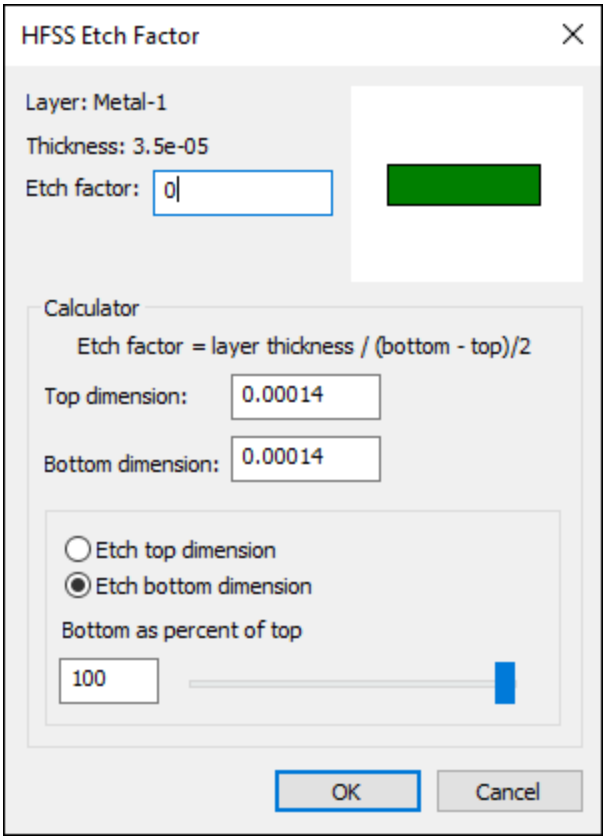
HFSS geometry allows for the presence of an etch factor in the design and modeling of different substrate layers. The specific etch factor that applies to a particular layer is usually available from the manufacturer. But in general, the etch factor is calculated by taking the ratio of etch depth to undercut.

When using the **Impedance Wizard** to calculate impedance, differential impedance, or trace width/pitch, change the **HFSS Etch Factor** settings.

Access these options by clicking the button next to **Trace cross section**. The button name reflects the current shape of the cross section:



The **HFSS Etch Factor** window appears:



The **HFSS Etch Factor** window acts as a calculator. Update any value and the other values, as well as the cross section diagram, will update accordingly.

When the calculator opens, it is populated with top and bottom dimensions that correspond to the default etch factor. These dimension values are representational and do not correspond to a particular geometry in the layout.

The **Etch Factor** value may be entered as a number or may be parameterized as a project variable. If the input is a number, the calculator controls are available; otherwise, the calculator controls are disabled. The top and bottom dimensions can be changed directly using their edit controls or they can be modified using the slider. The slider sets the value of one dimension to a percentage of the other. As the top and bottom dimensions are changed, both the etch factor and the layer drawing are modified to reflect the change.

When you click **OK**, the etch factor is applied to the selected layer.

4 - Configuring Padstacks

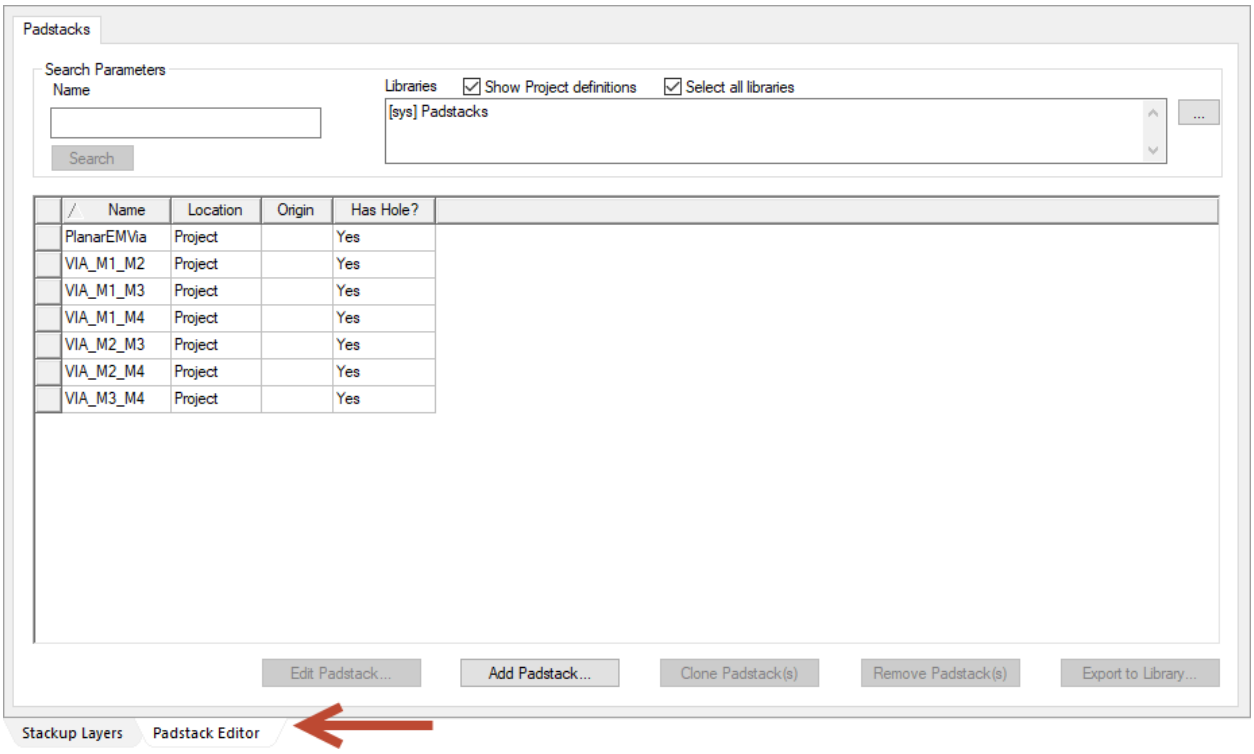
Padstacks describe the stacking structure of shapes associated with pads and vias. They enable different shapes on different layers. You can easily create and edit padstacks across all board types. Changes to the saved format and functionality are minimal, since the padstacks are independent of the number of layers in the design.

Padstacks consist of:

- Drill information
- Plane layer information – used for calculating the heat and iso shapes used on plane layers.
- Inner layer information – used for assigning different shapes for used/unused component pads.

The **Padstack Editor** displays in the same pane as **Stackup Layers**.

If that pane has been closed, select **View** > **Toolbars and Docking Windows** > **Reset Workspaces**. Then, ensure the **Padstack Editor** tab is selected:



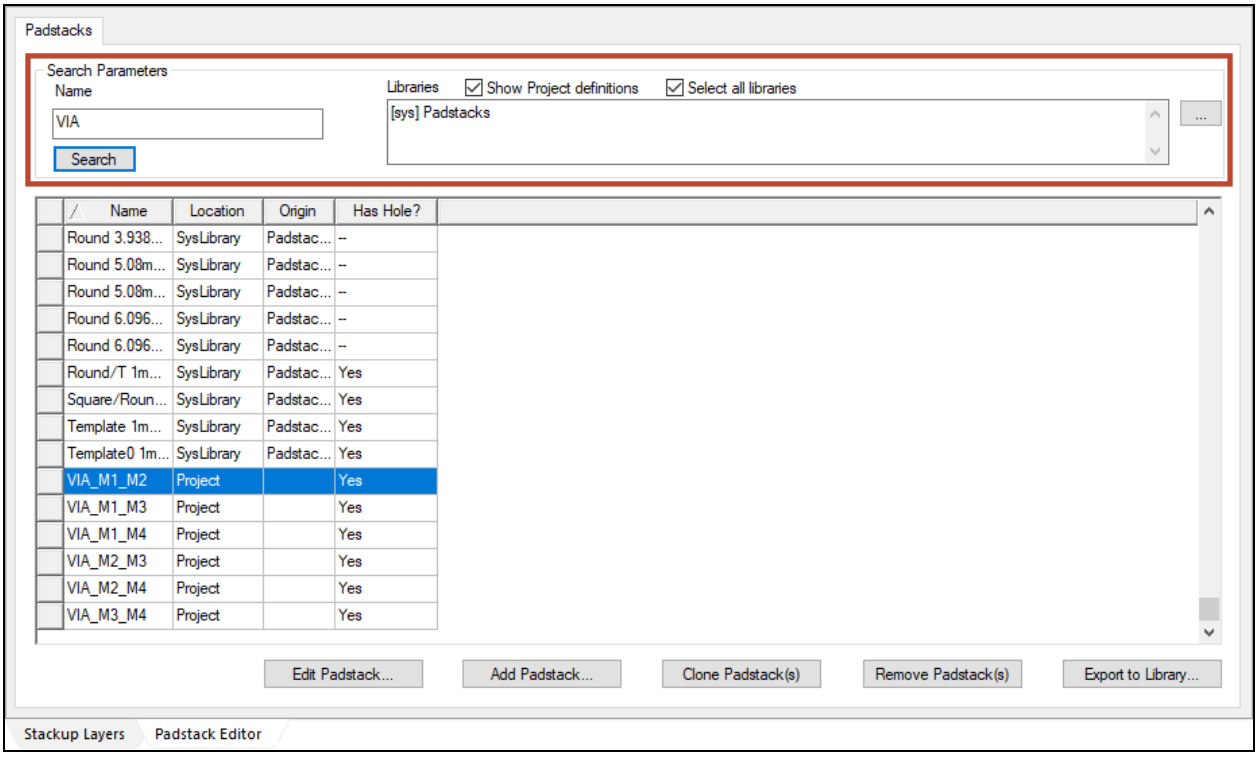
The **Padstack Editor** consists of the following areas:

- **Search Parameters** – allows you to search for a padstack by name.
- **Padstack List** – lists padstacks in the selected library or libraries.

- **Padstack Options** – buttons at the bottom of the pane allow you to add, edit, clone, or remove padstacks. You can also use them to access padstack settings.

Searching Padstacks

Use the **Search Parameters** area of the **Padstack Editor** to find specific padstacks.



To find a padstack:

1. Click the ellipses button (**...**) to select one or more libraries from the list at the right.

   If appropriate, enable the **Select all libraries** check box to view padstacks in all available libraries.
2. From the **Name** field, type a search query.
3. Click **Search**.

   The list of padstacks scrolls to where the search is found, and highlights the first row meeting the search criteria.

Modifying Padstacks

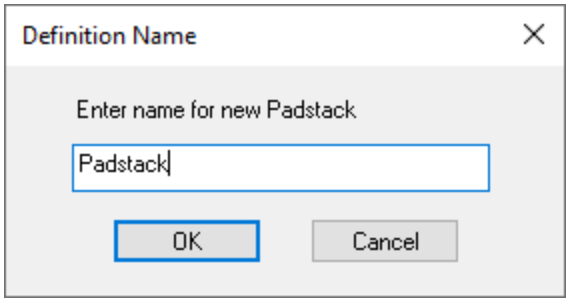
Use the buttons at the bottom of the **Padstack Editor** to add, edit, remove, or clone padstacks.

Adding Padstacks

To add a padstack:

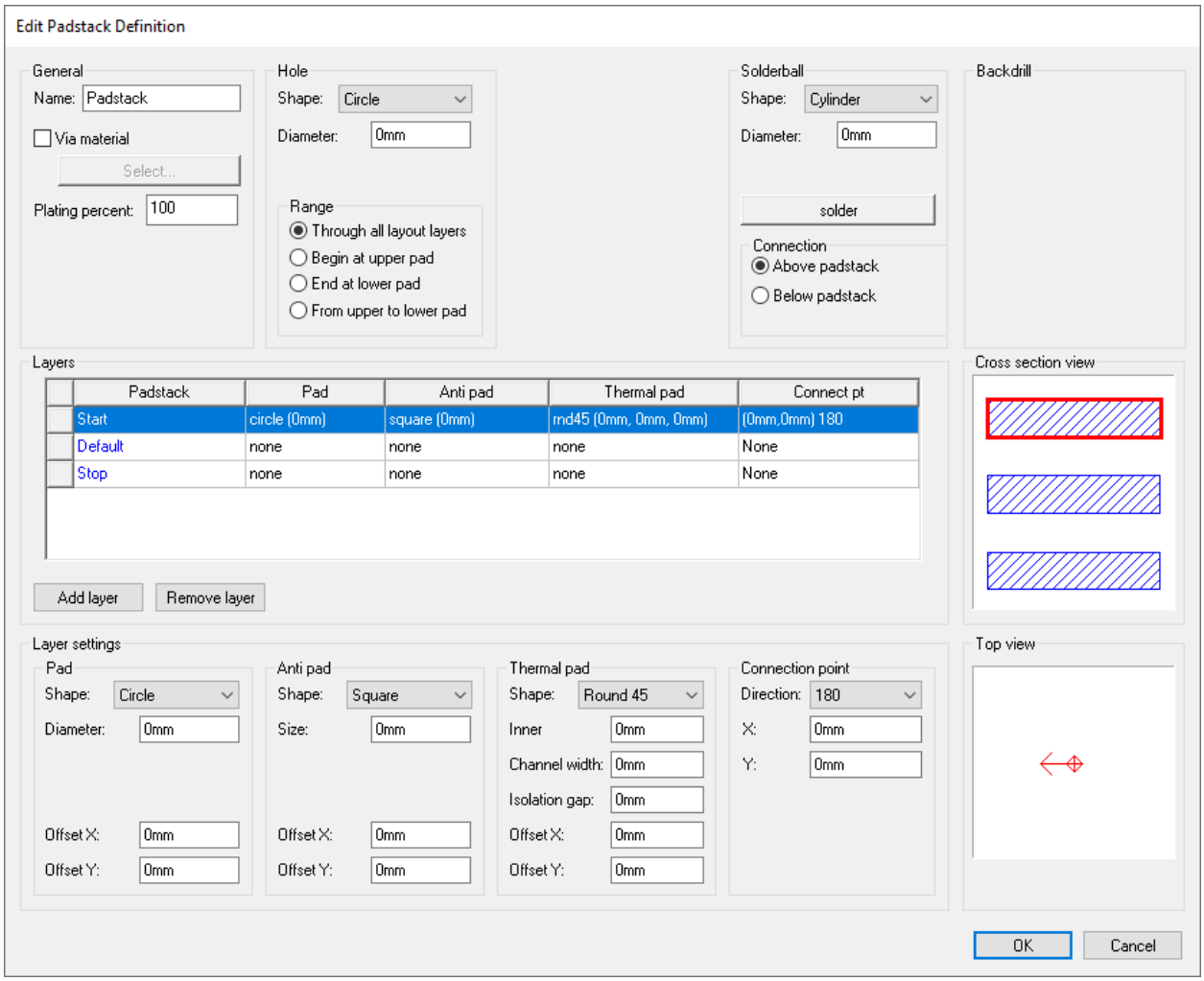
1. Click **Add Padstack**.

   The **Definition Name** window appears.



2. Enter a name for the padstack.
3. Click **OK**.

   The **Edit Padstack Definition** window appears.



4. Enter an appropriate settings for the new padstack and click **OK**. See: Padstack Settings.

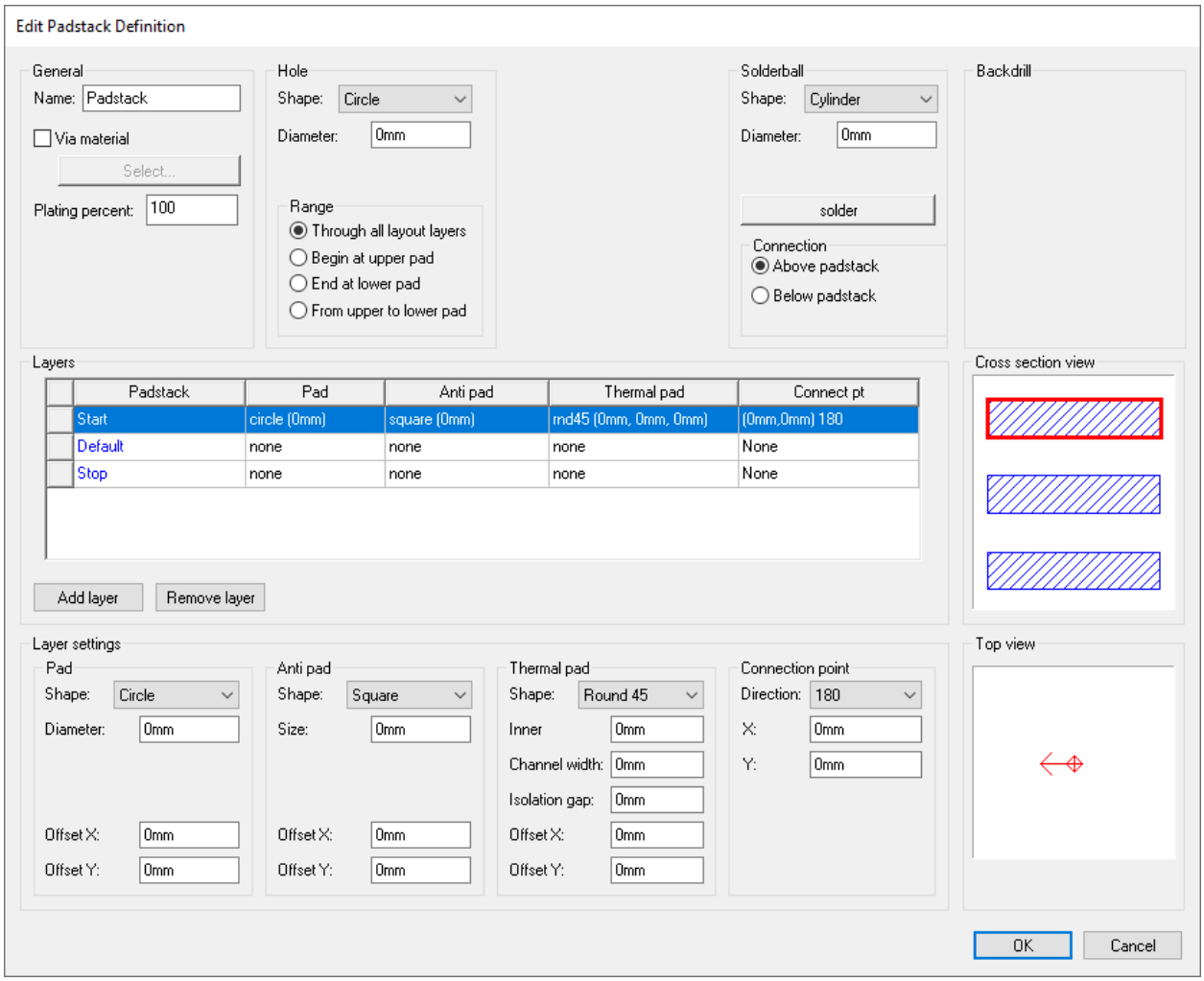
   The new padstack appears in the Padstack Editor's list of padstacks.

Editing Padstacks

To edit an existing padstack:

1. Select a padstack from the list of padstacks.
2. Click **Edit Padstack**.

   The **Edit Padstack Definition** window appears.



3. Enter an appropriate settings and click **OK**. See: Padstack Settings.

Cloning Padstacks

To clone one or more existing padstacks:

1. Select the padstack(s) from the list of padstacks.
   To select more than one padstack, use **Ctrl+click**.
2. Click **Clone Padstack(s)**.
   For each padstack selected, the **Edit Padstack Definition** window appears.
3. If appropriate, update each padstack's settings.

4. For each padstack, click **OK** to confirm settings.

   When you have verified the settings for all cloned padstacks, the list updates to include the cloned padstacks.

Removing Padstacks

> **Important:**
>
> This action cannot be undone.

To remove one or more existing padstacks:

1. Select the padstack(s) from the list of padstacks.

   To select more than one padstack, use **Ctrl+click**.
2. Click **Remove Padstack(s)**.

   An alert appears, warning you that there is no way to undo this action.

   > **Note:**
   >
   > You cannot remove padstacks from the system library.
3. Click **Yes** to continue.

   The padstack(s) are deleted.

Padstack Settings

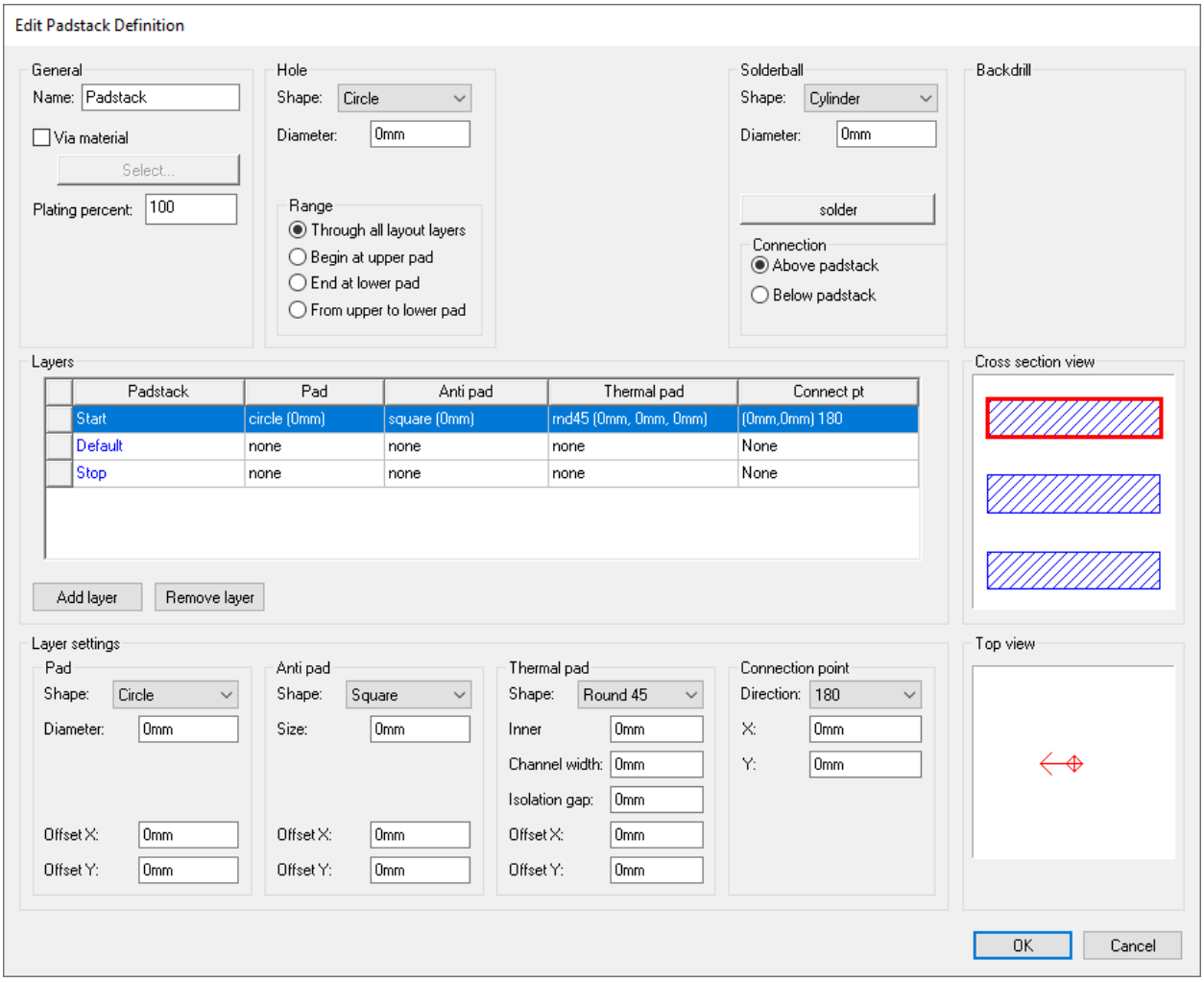
The **Edit Padstack Definition** window allows you to change the settings for a new or existing padstack.

For more information about adding and editing padstacks, see: Modifying Padstacks.

To access the **Edit Padstack Definition** window:

1. Select the **Padstack Editor** tab to view the **Padstack Editor**.
2. Select a padstack from the list of padstacks and perform one of the following actions:
   - Right-click the padstack and select **Edit Padstack**, or
   - Click the **Edit Padstack** button.

   The **Edit Padstack Definition** window appears.



This window contains the following areas for changing settings:

- **General** – allows you to set a padstack name, via material, and plating percentage.
- **Hole** – allows you to set a hole shape, dimensions, and range (depth). The dimensions available for editing depend on the shape of the hole.
- **Solderball** – allows you to set a solderball shape, dimensions, material (default: solder), and connection point (above or below the padstack). The dimensions available for editing depend on the shape of the solderball.
- **Backdrill** – [[???]]
- **Layers** – a list of layers in the padstack. You can add or remove a layer using an appropriate buttons, or change a layer's settings in the **Layer Settings** area below.
- **Layer Settings** – activated only when a layer is selected in the **Layers** list. See: Pad, Antipad, Thermal Pad, and Connection Point below.

Additionally, it displays the following visualizations:

- **Cross Section View** – displays all definition layers and their pads from a side view. Layers selected in the **Layers** list are highlighted in red. Hovering the cursor over a layer displays its name as a tooltip.
- **Top View** – displays the pads of selected layers from a top down perspective.

Pad Settings

Pad settings vary based on the shape of pad chosen.

Pad shape choices are:

- **None** – no pad.
- **Circle** – specify the diameter, offset x, and offset y.
- **Square** – specify the size, offset x, and offset y.
- **Rectangle** – specify the x size, y size, offset x, and offset y.
- **Oval** – specify the x size, y size, corner radius, offset x, and offset y.
- **Bullet** – specify the x size, y size, corner radius, offset x, and offset y.
- **N-Sided Polygon** – specify the size, number of sides, offset x, and offset y.
- **Polygon** – specify the shape, offset x, and offset y. See: Polygon Editor.

Antipad Settings

Antipad settings vary based on the shape of antipad chosen.

Antipad shape choices are:

- **None** – no antipad.
- **Circle** – specify the diameter, offset x, and offset y.
- **Square** – specify the size, offset x, and offset y.
- **Rectangle** – specify the x size, y size, offset x, and offset y.
- **Oval** – specify the x size, y size, corner radius, offset x, and offset y.
- **Bullet** – specify the x size, y size, corner radius, offset x, and offset y.
- **N-Sided Polygon** – specify the size, number of sides, offset x, and offset y.
- **Polygon** – specify the shape, offset x, and offset y. See: Polygon Editor.

Thermal Pad Settings

Thermal pad settings vary based on the shape of thermal pad chosen.

Thermal pad shape choices are:

- **None** – no thermal pad.
- **Polygon** – specify the shape, offset x, and offset y. See: Polygon Editor.
- **Round45** – specify the inner size, channel width, isolation gap, offset x, and offset y.
- **Round90** – specify the inner size, channel width, isolation gap, offset x, and offset y.
- **Square45** – specify the inner size, channel width, isolation gap, offset x, and offset y.
- **Square90** – specify the inner size, channel width, isolation gap, offset x, and offset y.

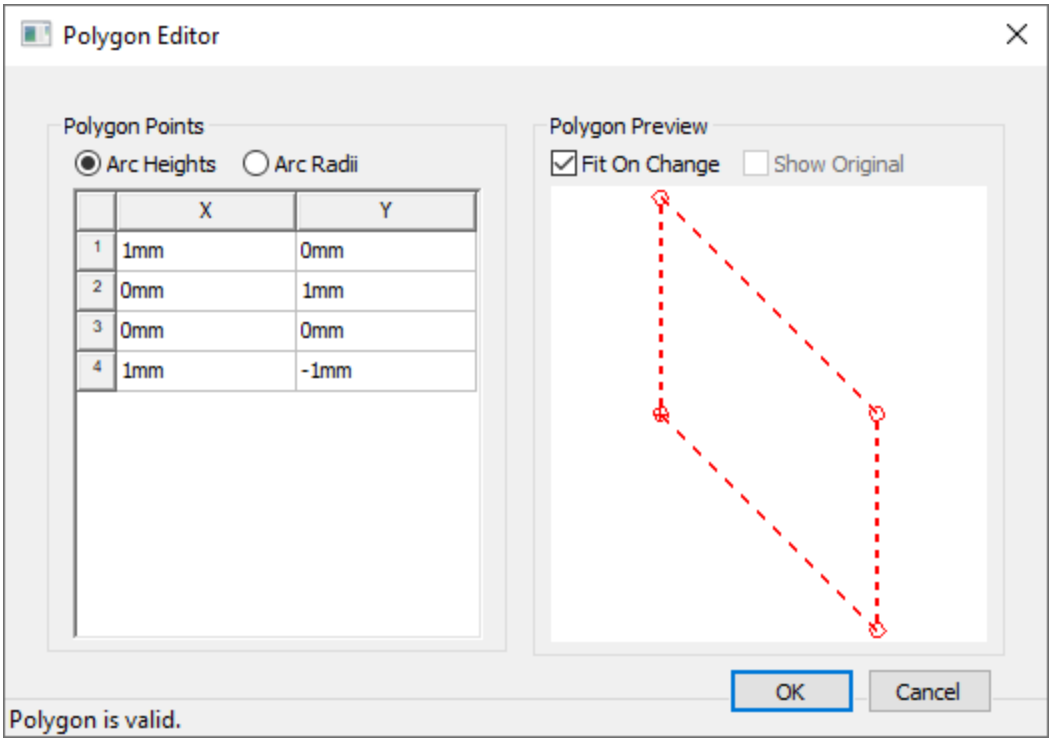
Connection Point Settings

For the **Connection Point**, select a **Direction**.

- **None** – No direction.
- **Any** – specify x and y, for any direction.
- **[# in Degrees]** – specify x and y.

Polygon Editor

The **Polygon Editor** appears when you specify that a pad, antipad, or thermal pad is polygonal in shape.



The window is divided into two sections:

- **Polygon Points** – lists the points of the polygon. Right-click in this area to add points, then edit the values in the X and Y columns.
- **Polygon Preview** – displays a preview image of the polygon. Right-click in this area to zoom in, zoom out, fit the polygon to the view window, or change the display attributes.

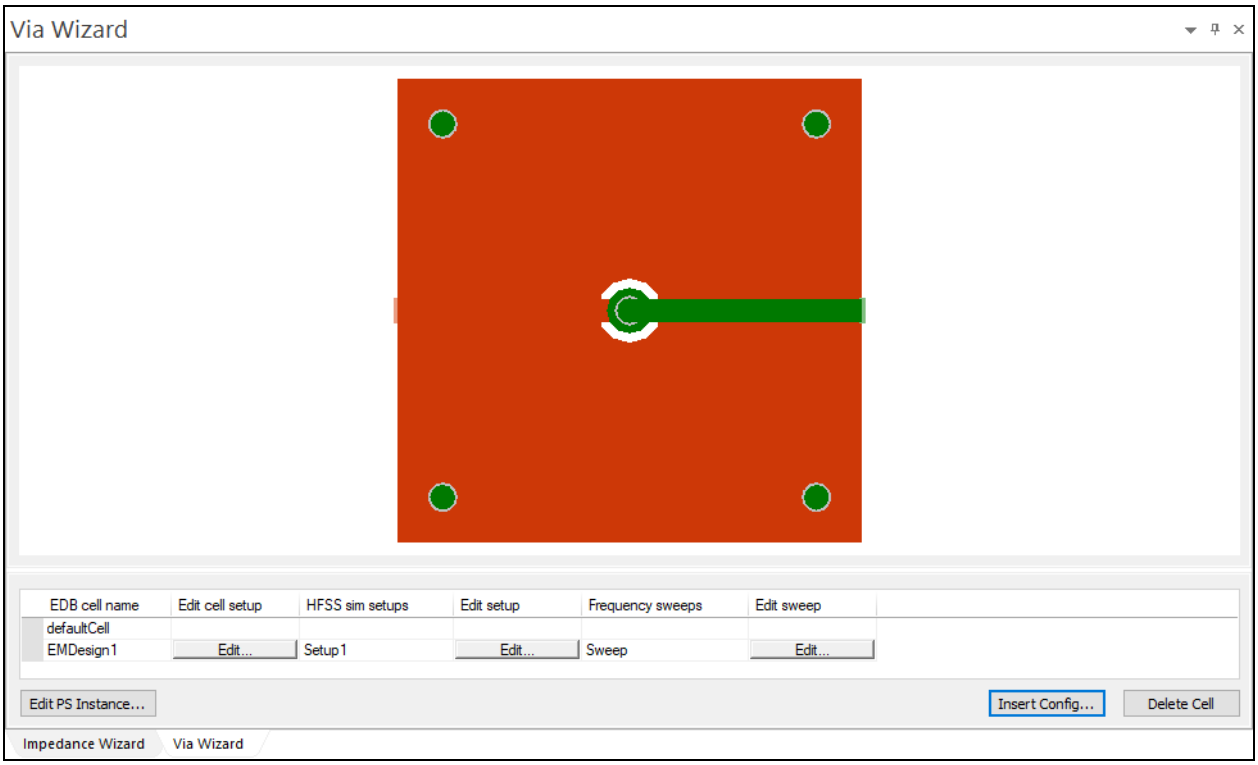
5 - Using the Via Wizard

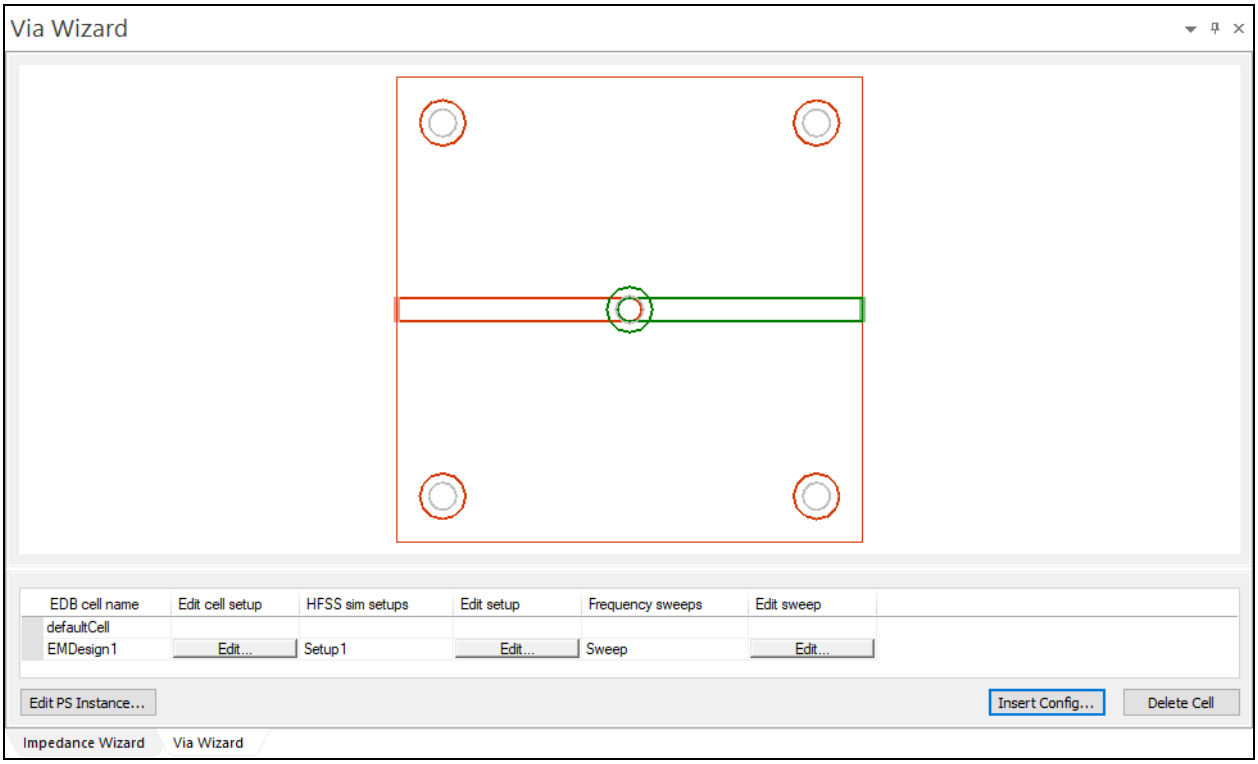
Traces are separate conducting lines that make up an important part of the PCB layout.

Vias consist of a hole cut in the PCB to route a trace vertically from one layer to another, as well as the vertical pieces of conductor that make connections between layers. If a via is too big, it does not have sufficient room to route signal traces. If vias are made smaller, more traces can be drawn.

The Via Wizard pane allows you to configure and visualize via breakouts.

The Via Wizard's visualization can be either solid or wireframe. To toggle between these views, select **View** > **View Design in Fill Mode**.





The process for using Via Wizard generally follows these steps:

- Configure padstacks
- Configure vias and visualize via breakouts
- Add backdrills, if appropriate
- Set the design up for HFSS simulation, if appropriate
- Post-process results in Network Data Explorer

Configuring Vias

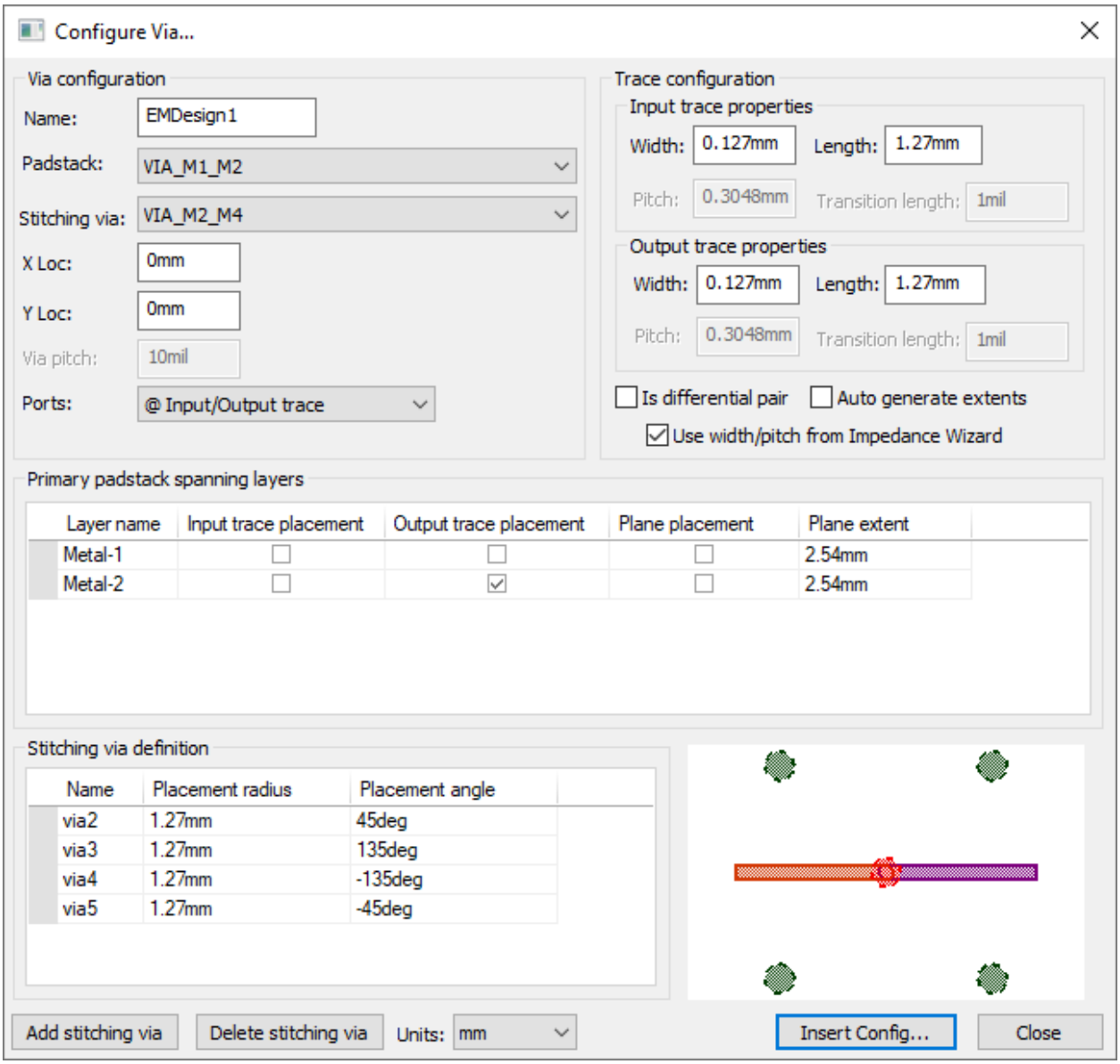
Traces are separate conducting lines that make up an important part of the PCB layout.

Vias consist of a hole cut in the PCB to route a trace vertically from one layer to another, as well as the vertical pieces of conductor that make connections between layers. If a via is too big, it does not have sufficient room to route signal traces. If vias are made smaller, more traces can be drawn.

Launch the **Configure Via** window in Via Wizard one of the following ways:

- Select **Project** > **Configure Via**.
- From the **Via Wizard** pane, click **Insert Config**.
- From the **Via Wizard** pane, edit an existing via configuration by clicking the **Edit** button.

The **Configure Via** window appears.



Specify the settings as appropriate:

- **Name** – enter a name for the via.
- **Padstack** – select a padstack from the list. New stacks can be added from the Padstack Editor.
- **Ground Via** – select a ground via from the list.
- **X Loc** – enter a value and unit.
- **Y Loc** – enter a value and unit.

- **Via Pitch** – specify the distance between two adjacent vias. A smaller via pitch introduces more mutual coupling capacitance.

  > **Note:**
  >
  > The **Via Pitch** setting is only available when **Is Differential Pair** is selected.

- **Ports** – determine whether to insert ports at the input trace, output trace, both traces, or neither.

From the **Trace Configuration** area:

- Specify the width and length for both input and output traces.
- If applicable, specify the pitch and transition length for input and output traces.

  > **Note:**
  >
  > **Pitch** and **Transition Length** options are only available when **Is Differential Pair** is selected.

- If applicable, use the check boxes to:
  - Specify a differential pair.
  - Select **Auto Generate Extents**. This is used to automatically compute the reference plane extents based on the length of traces above or below the said planes.
  - Use the width/pitch calculated in Impedance Wizard. See: Calculating Trace Width and Pitch.

From the **Stitching Via Definition** area:

- **Add Stitching Via** – opens a dialog box that allows you to specify the radius and angle for a new stitching via. If you need to remote a stitching via, select it from the list and click **Delete Stitching Via**.

Click **Insert Config** to insert the configuration for use in Via Wizard.

Adding Backdrills

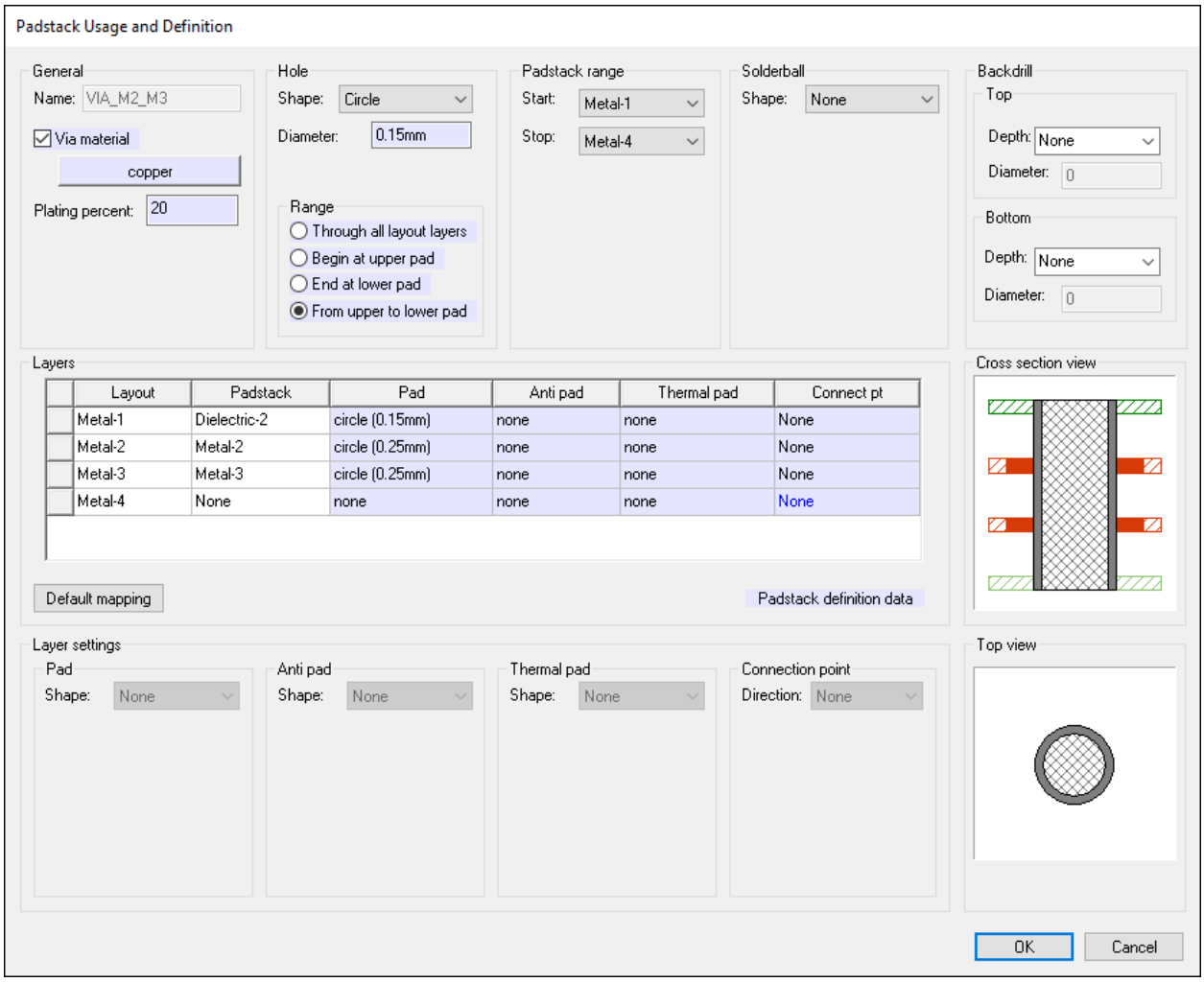
Unused portions of vias (known as "stubs") can create significant signal integrity disturbances. Backdrilling is a technique that removes these stubs.

To add a backdrill in SI Xplorer:

1. Insert a via configuration.
2. From the **Via Wizard**, click a port or padstack in the diagram to select it.

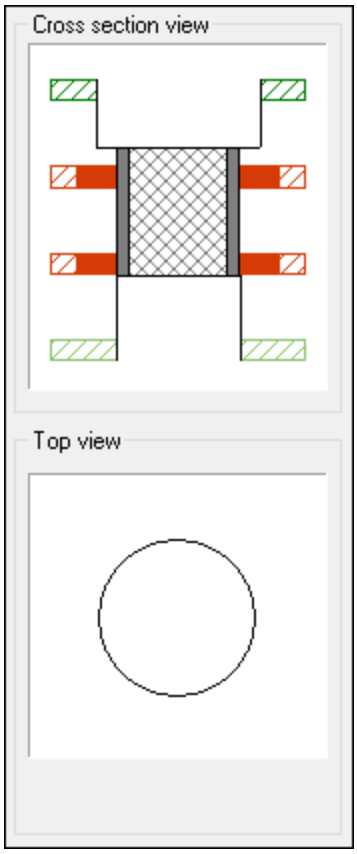
3. Click **Edit PS Instance**.

   The **Padstack Usage and Definition** window appears.



4. Use the options in the **Backdrill** area to define a backdrill:
   - **Top** – use the drop-down menu to select the depth of the backdrill from the top layer.
   - **Bottom** – use the drop-down menu to select the depth of the backdrill from the bottom layer.
   - **Diameter** – for either top or bottom, specify the backdrill's diameter.

   The **Cross Section View** and **Top View** areas update to display the backdrill.



5. Click **OK**.

6 - Running Simulations

Performing a simulation in SI Xplorer involves the following steps:

1. Edit Design Settings for Simulation.
2. Add an HFSS Solution Setup.
3. Run a Validation Check.
4. Click **Simulation** > **Analyze**.

   The **Progress Monitor** displays simulation progress.

   The **Message Manager** alerts you of important developments, such as errors.

Editing Design Settings for Simulation

Before running a simulation, you may want to adjust the design settings:

- Click **Simulation** > **Design Settings**.

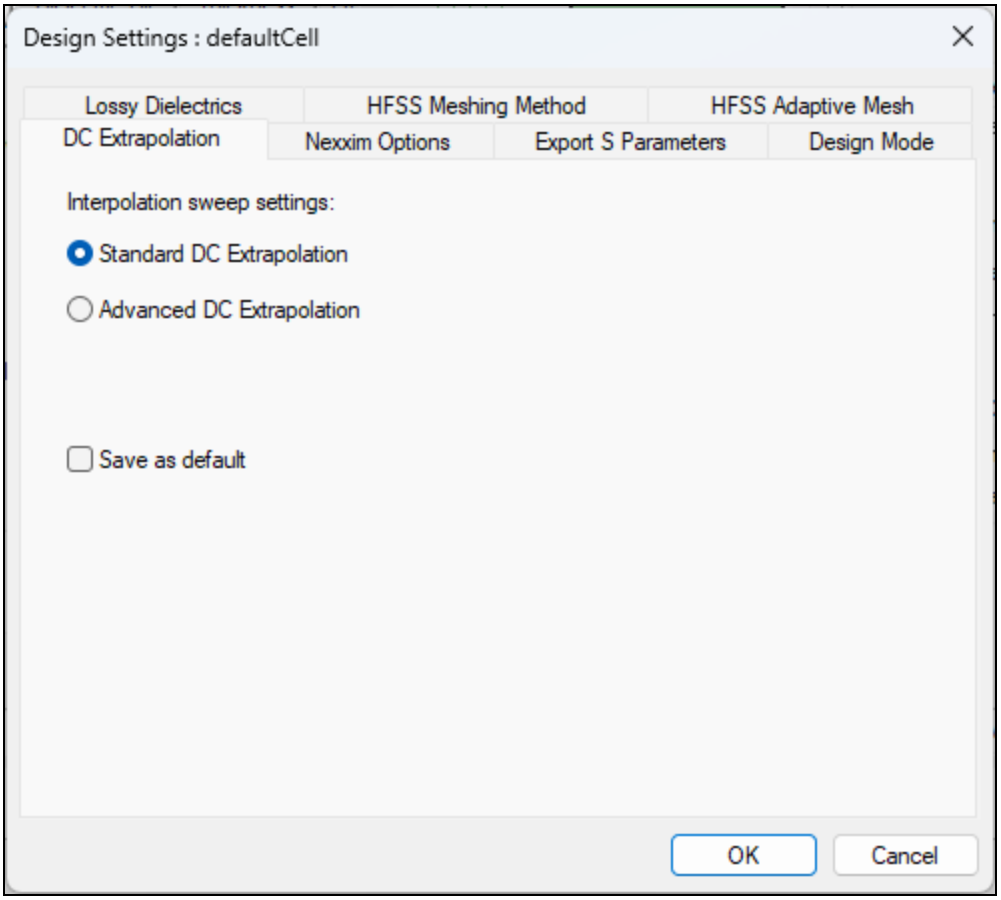
The **Design Settings** window appears. It consists of six tabs:

- DC Extrapolation
- Nexxim Options
- Export S Parameters
- Design Mode
- Lossy Dielectrics
- HFSS Meshing Method
- HFSS Adaptive Mesh

Adjust the settings as appropriate and click **OK** to apply. On any tab, you may select **Save as default** to preserve your settings on that tab as the default.

The sections below describe the available settings in greater detail.

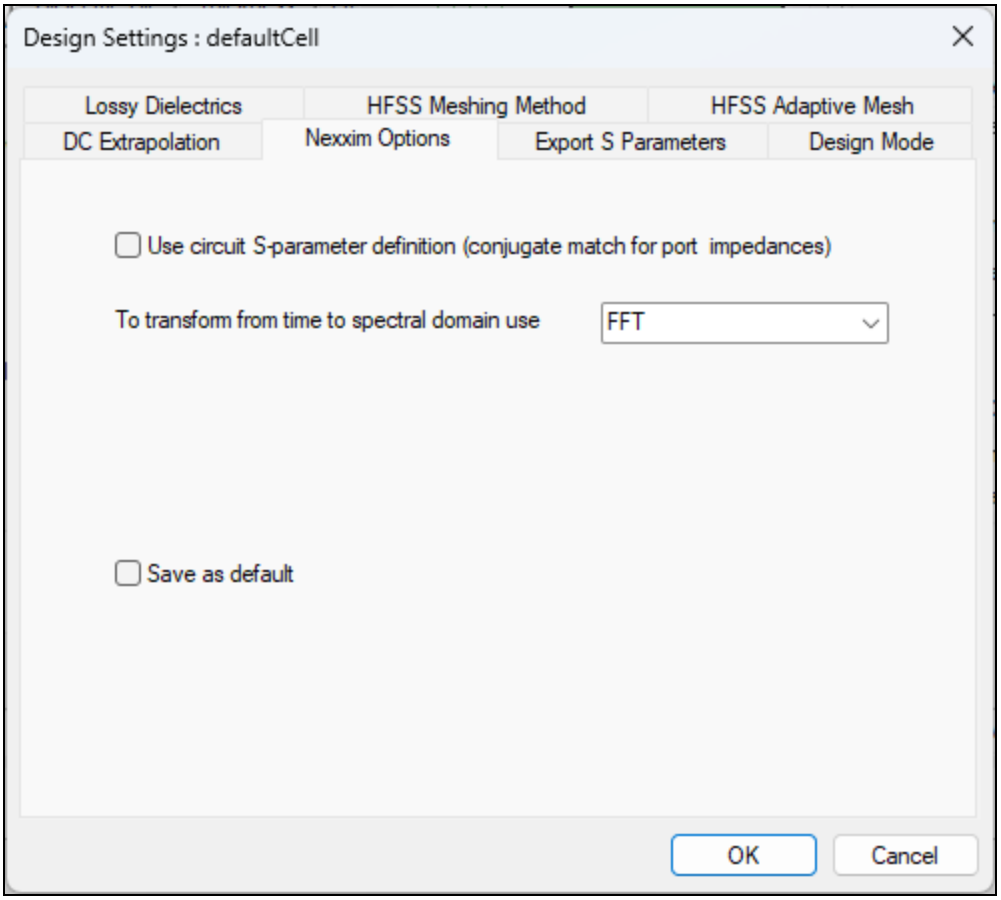
DC Extrapolation Settings



The **DC Extrapolation** tab allows you to select one of **Standard DC Extrapolation** or **Advanced DC Extrapolation**.

> **Important:**
>
> This setting changes the "ground rules" of the modeler and may have unexpected results.

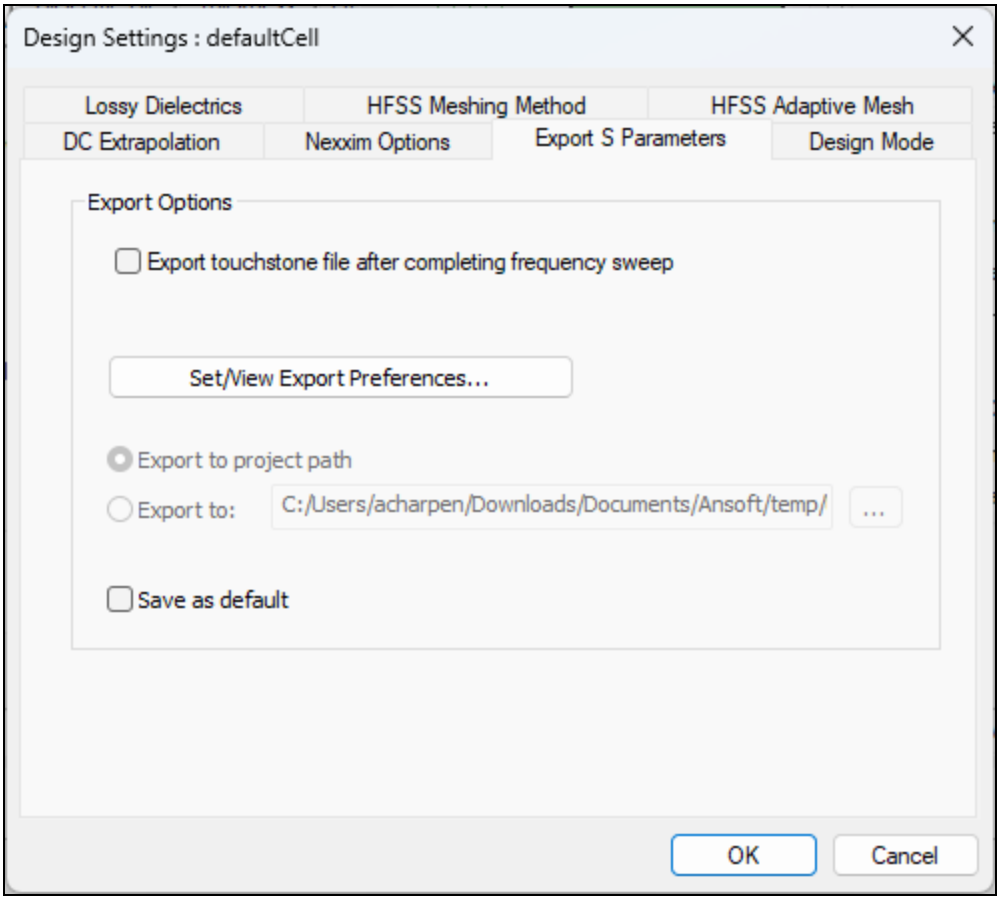
Nexxim Options



The **Nexxim Options** tab allows you to select:

- Whether to **Use circuit S-parameter definition (conjugate match for port impedances)**
- Method used to transform time to spectral domain (FFT, Fourier Integration, or Fourier Transform)

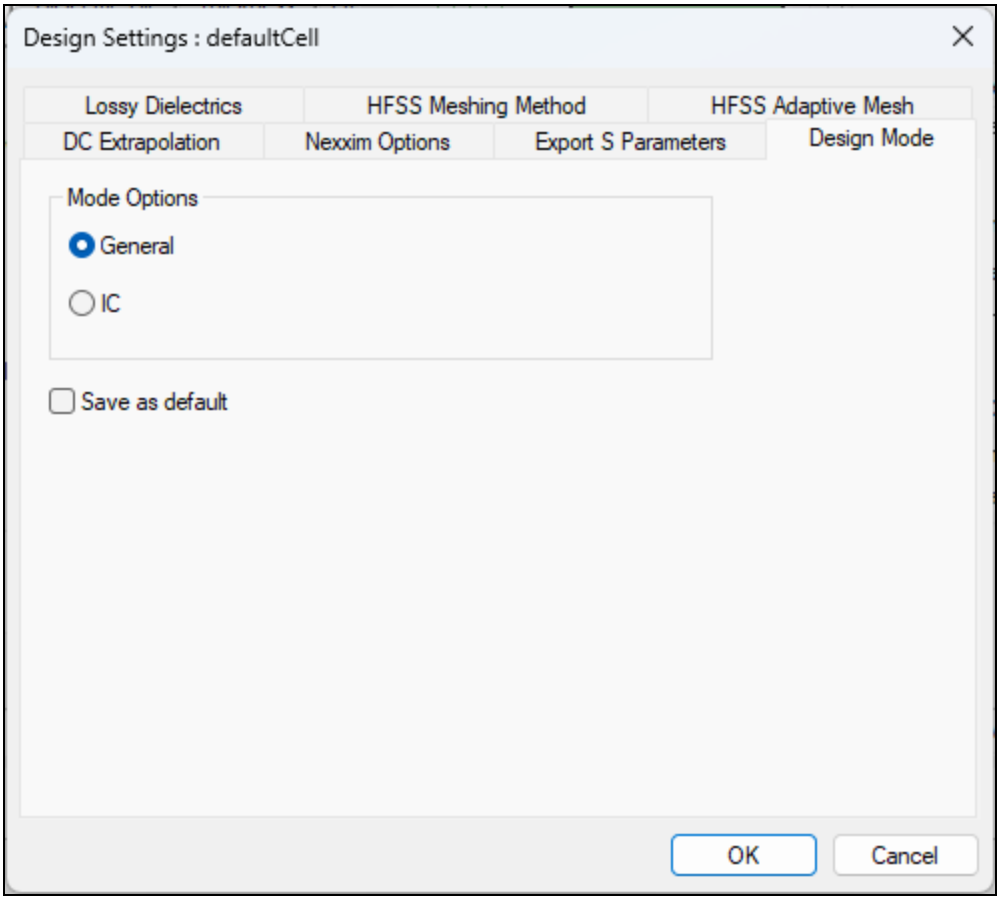
Export S Parameters Settings



The **Export S Parameters** tab allows you to:

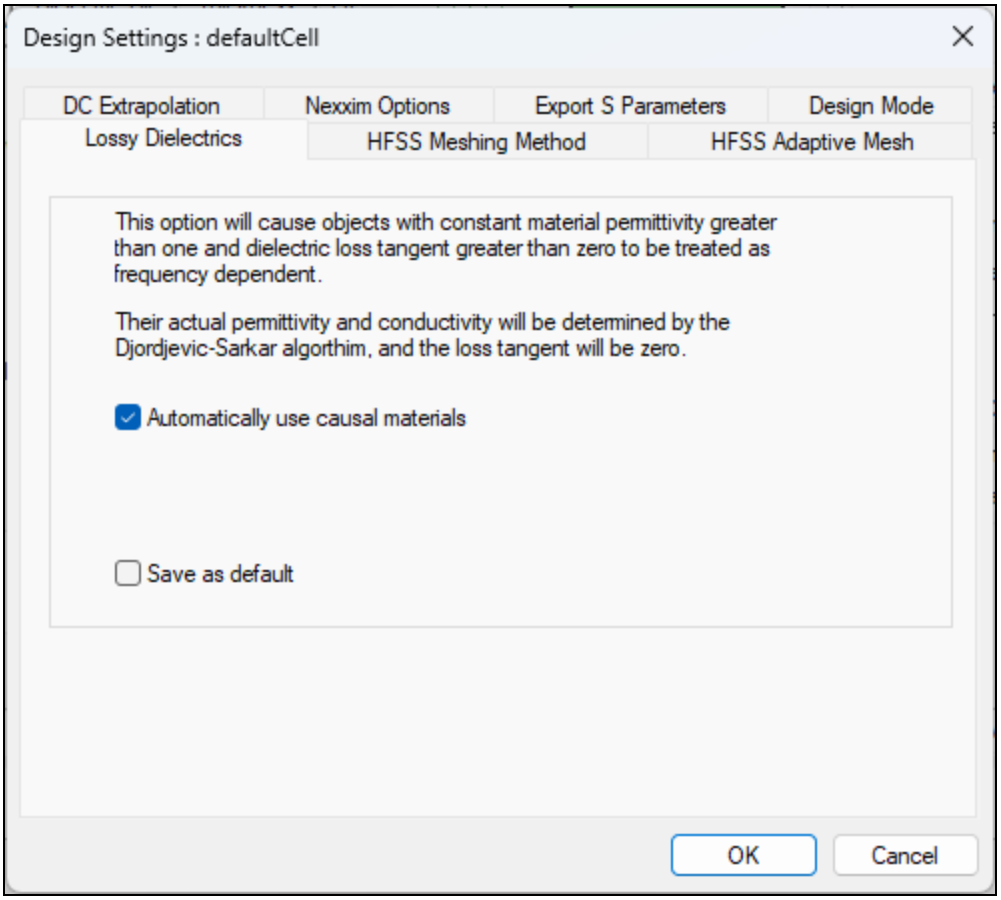
- **Set/View Export Preferences** – opens the **SYZ Data Export Options** window, from which you can export Touchstone or Broadband.
- **Export touchstone file after completing frequency sweep** – select **Override** and use the asterisk **(...)** button to select a destination other than the default.

Design Mode



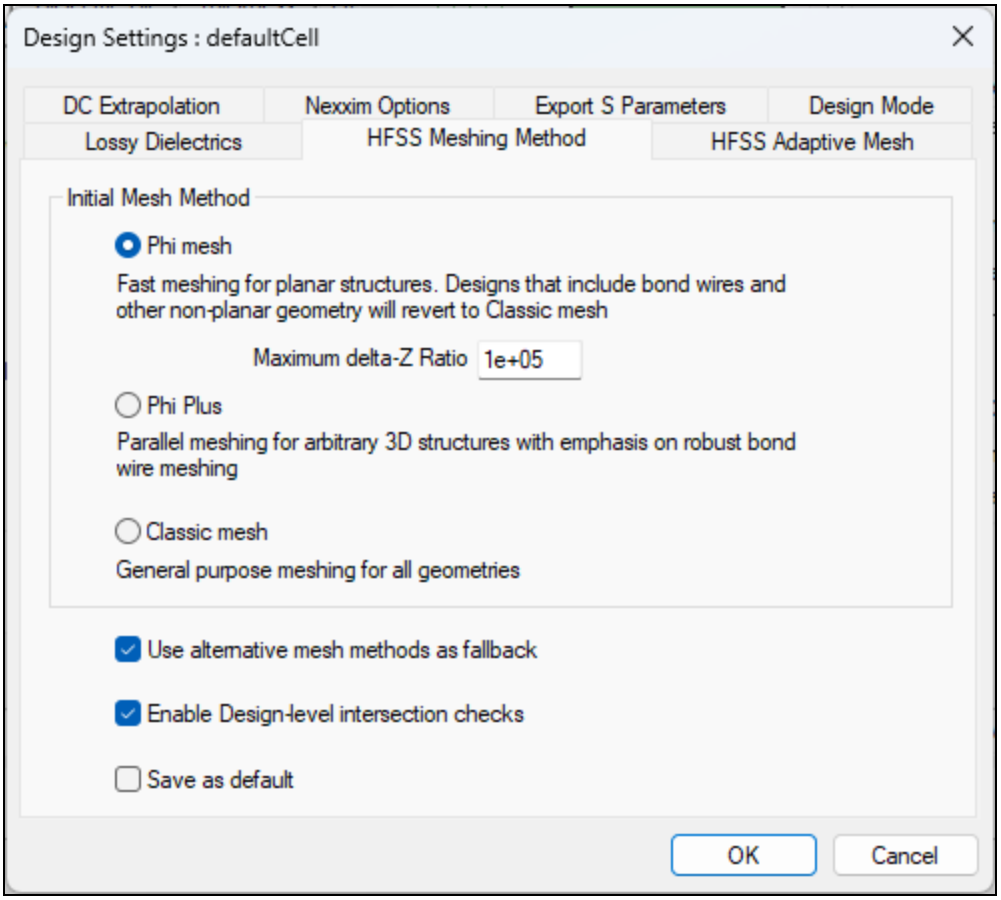
The **Design Mode** tab allows you to select **General** or **IC** (i.e., Integrated Circuit) mode.

Lossy Dielectrics Settings



The Lossy Dielectrics tab allows you to **Automatically use causal materials**. This causes objects with constant material permittivity greater than one and dielectric loss tangent greater than zero to be treated as frequency dependent. Their actual permittivity and conductivity will be determined by the Djordjevic-Sarkar algorithm, and the loss tangent will be zero.

HFSS Meshing Method Settings



The **HFSS Meshing Method** tab allows you to select one of three meshes:

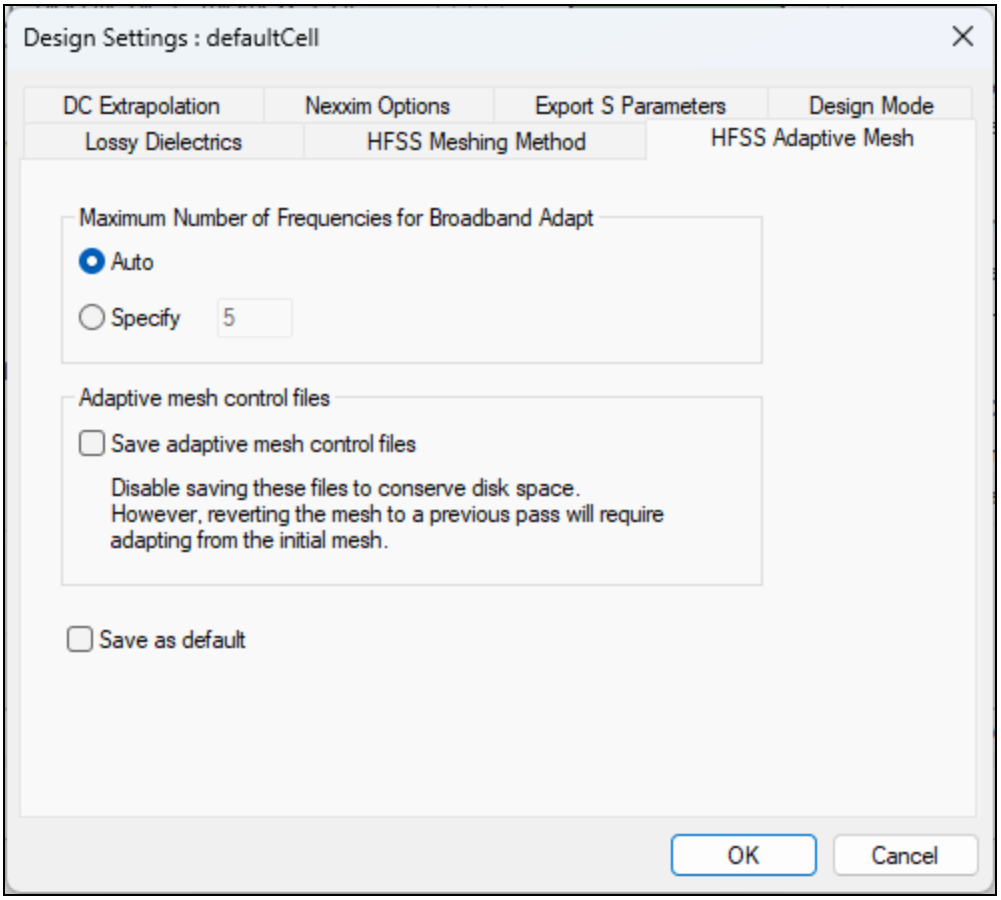
- **Phi Mesh**
- **Phi Plus**
- **Classic Mesh**

Additionally, you can select:

- **Use alternative mesh methods as fallback**
- **Enable Design-level intersection checks**

Consult the HFSS help for additional information about HFSS meshing.

HFSS Adaptive Mesh Settings



The **HFSS Adaptive Mesh** tab allows you to adjust settings for:

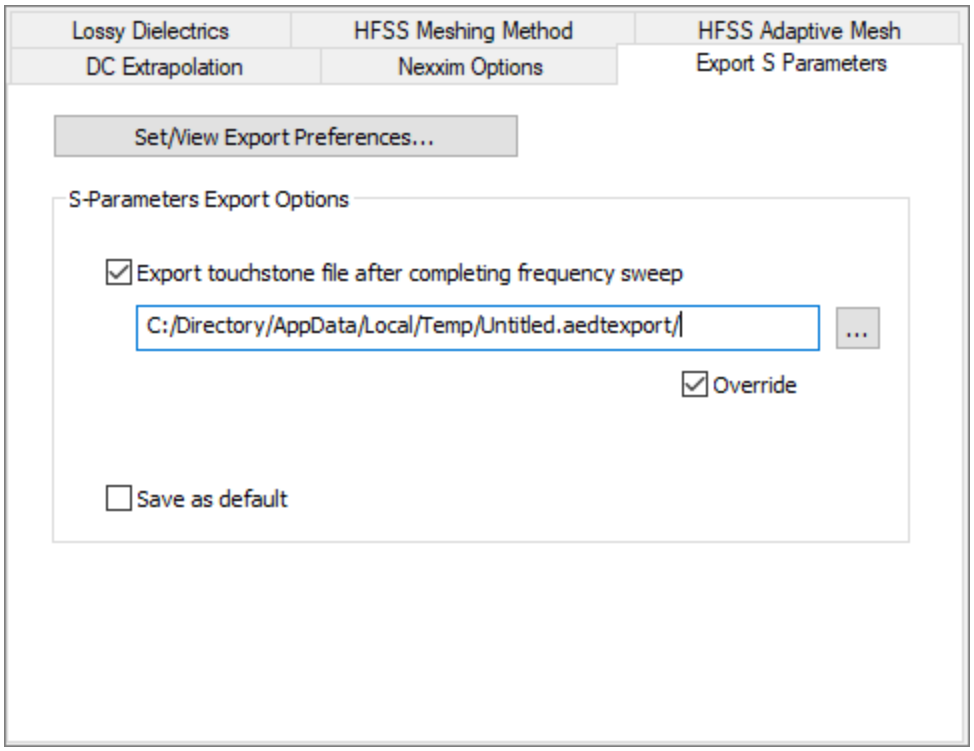
- Maximum Number of Frequencies for Broadband Adapt:
  - **Auto** – select this radio button to have the maximum number automatically defined.
  - **Specify** – select this radio button and enter a value for the maximum number of frequencies.
- **Save Adaptive Mesh Control Files** – this option is disabled by default to preserve disk space.

SYZ Data Export Options

To export S parameter data in either Touchstone or Broadband format:

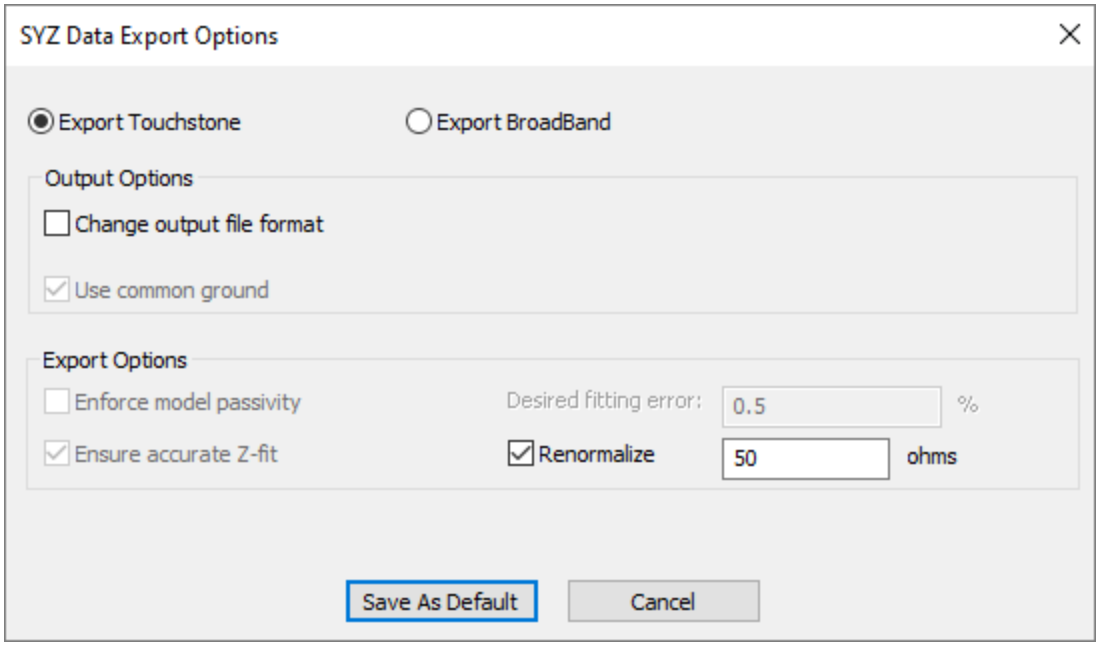
1. From the **Design Settings** window, select the **Export S Parameter** tab.

   The **Export S Parameter** tab appears.



2. Click **Set/View Export Preferences**.

   The **SYZ Data Export Options** window appears.



3. Select either **Export Touchstone** or **Export Broadband**.

   The available **Output Options** and **Export Options** vary based on your selection.
4. Click **Save As Default** to save your settings, or click **Cancel** to exit.

   You are returned to the **Design Settings** window.
5. If appropriate, select **Override** and specify a file destination other than the default.
6. Click **OK** to exit the window.
7. Run a simulation.

   When it completes, the exported file appears in the destination you specified (or the default).

Adding an HFSS Solution Setup

There are two ways to add an HFSS solution setup:

- HFSS Auto Setup
- HFSS Advanced Setup

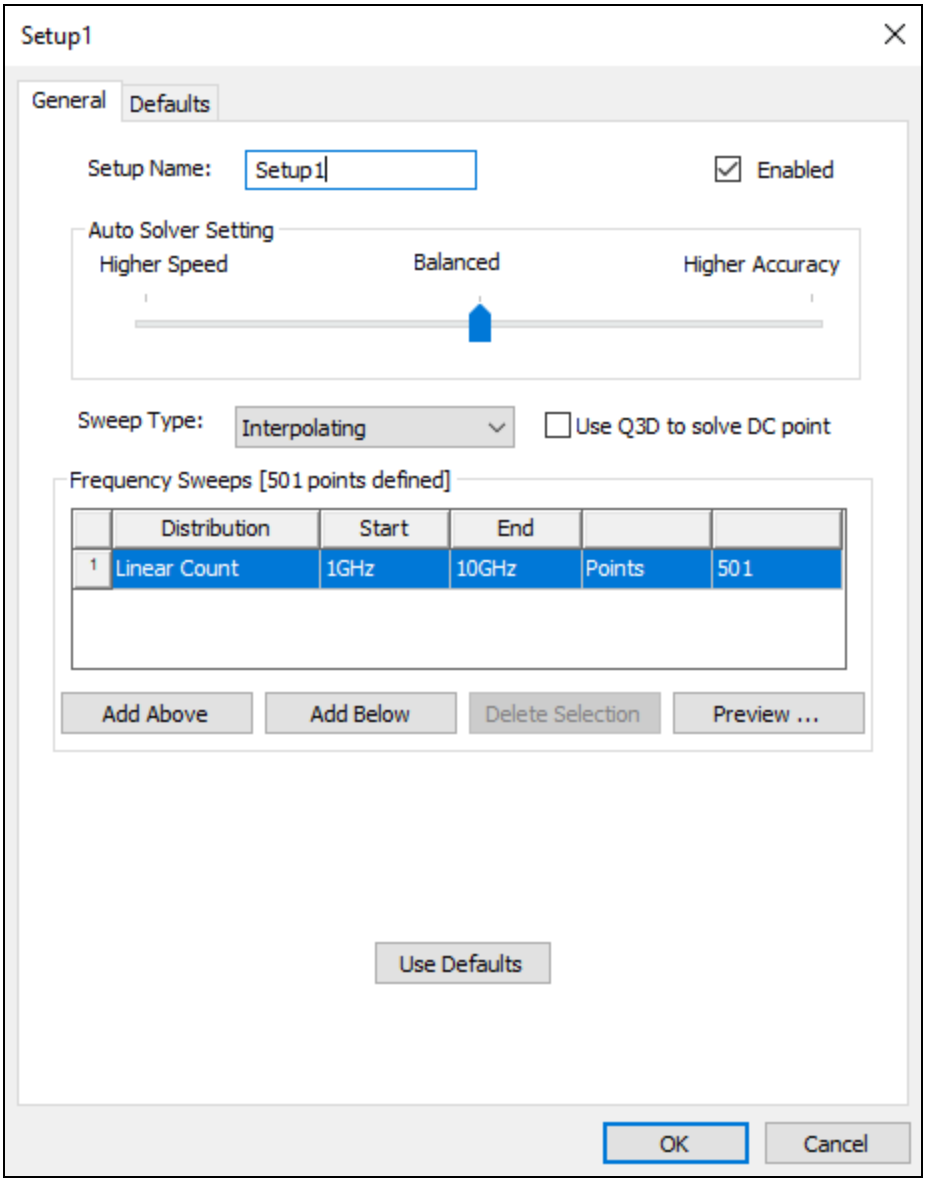
> **Important:**
>
> In addition its own settings, SI Xplorer respects any **HPC and Analysis Options** selected for the **HFSS 3D Layout** design type in Ansys Electronics Desktop.

HFSS Auto Setup

To add an HFSS Auto solution setup:

1. Click **Simulation** > **Add HFSS Solution Setup** > **Auto**.

Auto setup features a simplified window with a slider bar that you can adjust from **Higher Speed** (which sacrifices accuracy to achieve optimum speed) to **Balanced** (which is more accurate but faster than the highest accuracy setting) to **Higher Accuracy** (which takes the time to ensure optimal accuracy).



The auto setup includes a single sweep, for which you can edit the Distribution, Start, and End. The design must include one or more ports.

Use the **Sweep Type** drop-down menu to select either:

- **Discrete** – generates field solutions at specific frequency points in a frequency range.
- **Interpolating** – estimates a solution for an entire frequency range using an adaptive rational function interpolation method. Using an Interpolating Fast sweep will result in faster analysis, however the current distribution values cannot be saved.

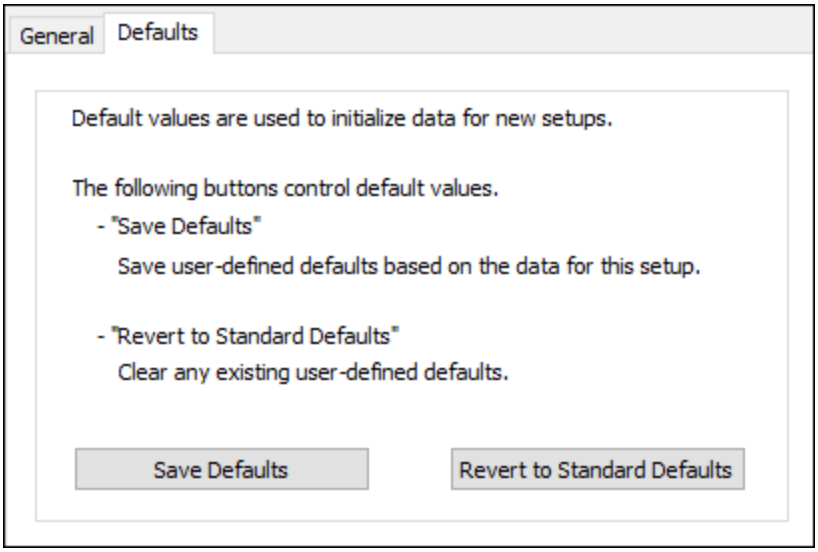
When **Use Q3D to solve DC point** is selected, the HFSS solver invokes Q3D Extractor to explicitly solve for a DC point. The mesh from HFSS is used for the Q3D extraction, with sources and sinks defined based on HFSS port assignment. Matrix reductions are automated to integrate the Q3D results with HFSS.

Use the **Frequency Sweeps** area to specify sweeps in terms of **Distribution** type, which can be:

- Linear Step
- Linear Count
- Log Scale
- Single Point
- Single Point Sweep, which adds a set of 10 Single Point Sweeps defaulting from 1 GHz to 10 GHz in increments.

To add more sweeps (including mixed sweep types), use the **Add Above** and **Add Below** buttons. This feature provides flexibility (e.g., you can define sweeps with log scale at lower frequencies and linear step at higher frequencies).

If appropriate, use the **Defaults** tab to save your settings as the default or revert to the original settings.



HFSS Advanced Setup

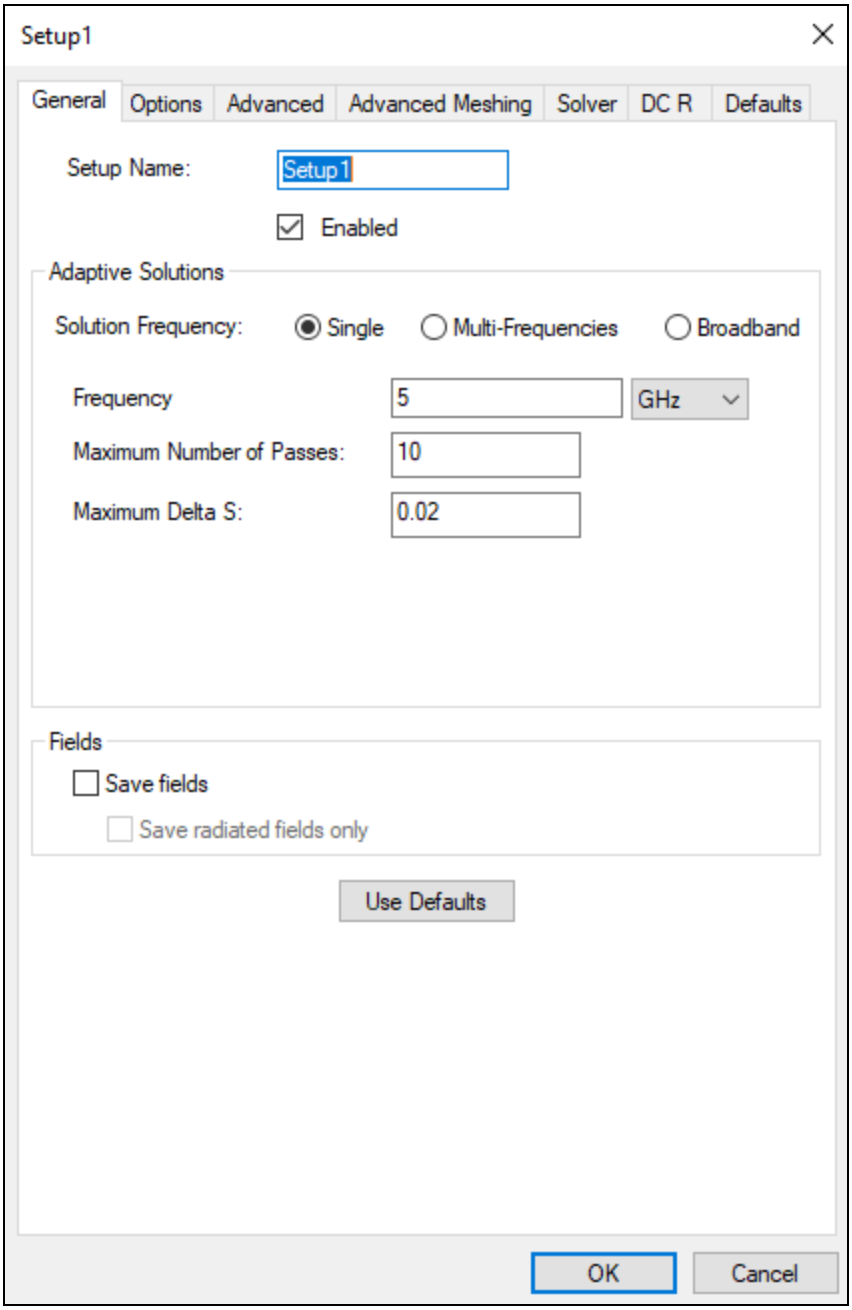
To add an HFSS Advanced solution setup:

1. Click **Simulation** > **Add HFSS Solution Setup** > **Advanced**.

The settings for **Advanced** setup are divided into eight tabs:

- **General** – general solution settings.
- **Options** – settings for lambda refinement, adaptive analysis and solution, order of basis, and whether to enable the use of solver domains.
- **Advanced** – settings for defeaturing a mesh, which model type to use, and vias.
- **Advanced Meshing** – settings for curve approximations and layer alignment.
- **Solver** – port settings and configurations for modeling.
- **DC R** – settings for the DC Resistance and Inductance analysis.
- **Defaults** – enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

HFSS Advanced Setup – General Tab

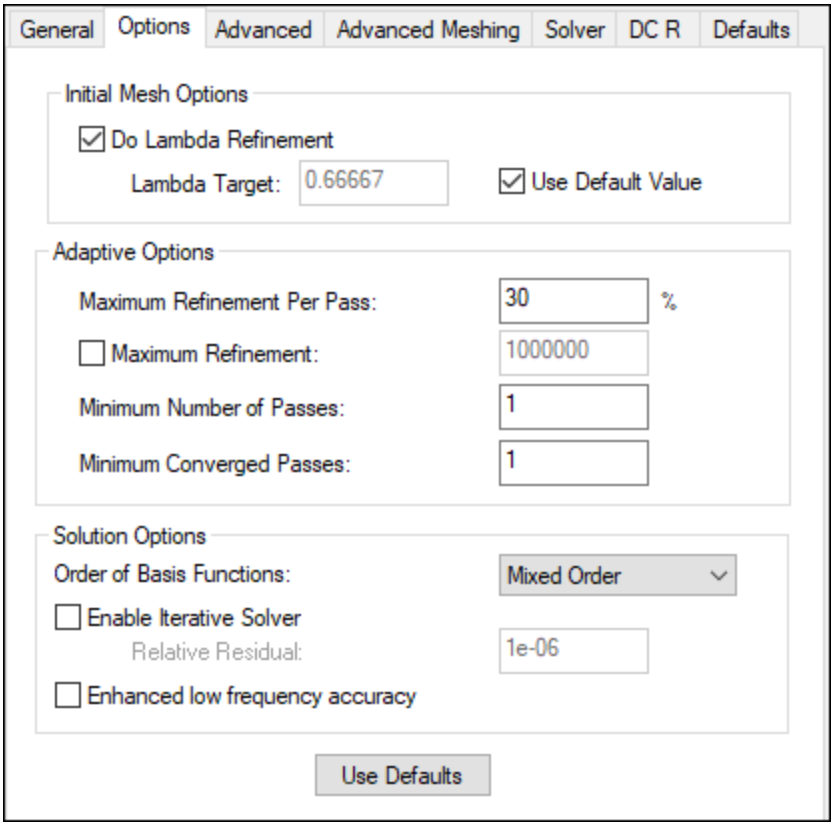


The **General** tab allows you to set the following:

- **Setup Name** – Name of the setup. It must be unique.
- **Enabled** – If not enabled, this setup cannot be analyzed and will be not included in a general "Analyze" of all setups.

- **Solution Frequency** – Specify the frequency and units at which to generate the solution. If a frequency sweep is solved, an adaptive analysis is performed only at the solution frequency. The following fields change according to your selection:
  - **Frequency** – The adaptive frequency for refinement.
  - **Low Frequency** – The smallest adaptive frequency for refinement for a **Broadband**.
  - **High Frequency** – The largest adaptive frequency for refinement for a **Broadband**.
  - **Maximum Number of Passes** – The maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues until the convergence criteria are reached.
  - **Maximum Delta S** – The magnitude of the change of the S-parameters between two consecutive adaptive passes. This is a stopping criterion for the adaptive solution. If the magnitude of the change of all S-parameters are less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.
  - **Output Var.** – A button labeled **Add** to add output variables to the refinement criteria for that frequency or **Edit** to edit the output variables for that refinement criteria. Selecting this option brings up the **Advanced Mesh Convergence** dialog box or displays the error message 'You must have at least one output variable defined' if you have no output variables defined in your design. Users may receive the error 'You must have at least one output variable defined that returns a single real value (as opposed to complex, etc)'.
    - **Output Variable** – The name of the output variable.
    - **Include** – A check box for whether to include the value of this output variable in the mesh convergence calculations.
    - **Max Delta** – The maximum amount of change appropriate in the output variable between adaptive passes. It is a stopping criterion for the adaptive solution. If the output variable changes from one iteration to the next by an amount that is less than Max Delta, the adaptive solution.
  - **Add** – Adds a new frequency to the table.
  - **Remove** – Removes a selected frequency from the table.
- **Fields** – Click **Save fields** to post process or plot fields. This is only available for Discrete Sweep Types. To restrict the fields saved to radiated fields, click **Save radiated fields only**.
- **Use Defaults** – The target defaults depend on the Order of Basis function selections (e.g., for Driven solutions and a First Order basis function, the default target is 0.3333. HFSS refines the mesh until most element lengths are approximately one-third wavelength).

HFSS Advanced Setup – Options Tab



The **Options** tab allows you to set the following:

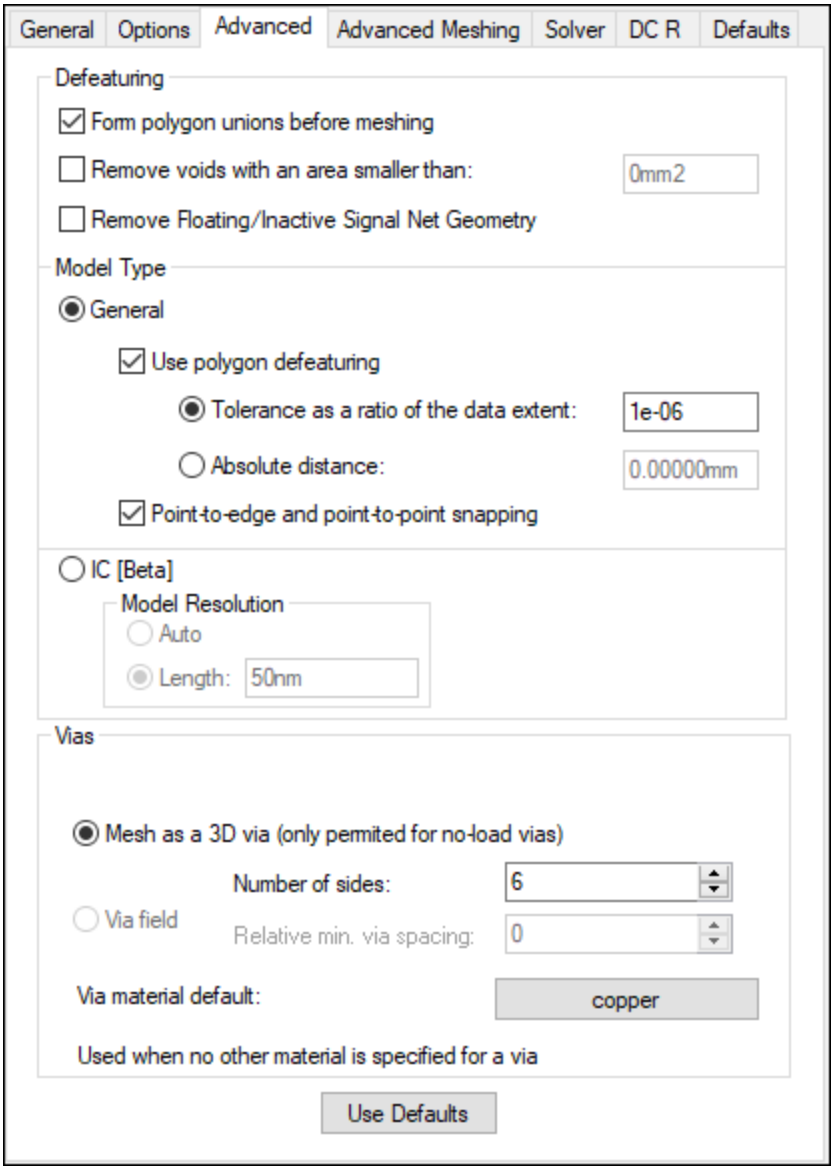
- Initial Mesh Options:
  - **Do Lambda Refinement** – Lambda refinement is the process of refining the initial mesh based on the material-dependent wavelength.
  - **Lambda Target** – The fraction of the wavelength that determines the refined tetrahedra edge length (e.g., if the value is 0.1 then a edge length of 0.1 is used as the refinement target).
  - **Use Defaults** – The target defaults depend on the Order of Basis function selections (e.g., for Driven solutions and a First Order basis function, the default target is 0.3333; HFSS refines the mesh until most element lengths are approximately one-third wavelength.).
- Adaptive Options:
  - **Maximum Refinement Per Pass** – determines how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error is refined. The value is a percentage.

- **Maximum Refinement** – The maximum number of tetrahedra that can be added during an adaptive pass. By default, this unchecked, to that there is no maximum. If you enable the Maximum Refinement, the initial value is 1000000.
- **Minimum Number of Passes** – The maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution. If the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues unless the convergence criteria are reached.
- **Minimum Converged Passes** – An adaptive analysis does not stop unless the minimum number of converged passes has been completed. The convergence criteria must be met for at least this number of passes before the adaptive analysis stops.

- Solution Options:
  - **Order of Basis Functions** – The order of the basis functions HFSS uses to interpolate field values from nodal values. The Zero order option is useful when a model requires a mesh that produces more than 100,000 tetrahedra, but the model size is small compared to wavelength. The higher order options solve progressively more unknowns for each tetrahedra. Mixed order uses higher order where more accuracy is required, and lower order where fields are weaker.
  - **Enable Iterative Solver** – The iterative solver provides an alternative to the multi-frontal solver when a matrix is well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also provide a savings in the solution time for large simulations. When this option is enabled, HFSS automatically invokes the iterative solver when it decides that the matrix is conditioned well enough to take advantage of the iterative approach. HFSS uses the multi-frontal solver if the matrix does not meet this requirement.
  - **Relative Residual** – The residual measures the convergence of the iterative solver. The default value is 1E-4.
  - **Enhanced low frequency accuracy** – When enabled, the solver is tuned to reliably solve low frequencies for designs that only contain lumped ports as sources or a combination of lumped and circuit ports. In addition, for designs with only lumped ports and/or circuit ports, interpolating sweeps are tuned to solve more low frequency points in order to accurately represent very low frequency results.

    > **Note:**
    >
    > This requires more RAM in order to solve the entire solution vector for interpolation sweeps and for discrete sweeps not saving fields.

HFSS Advanced Setup – Advanced Tab



Use the **Advanced** tab to set the following:

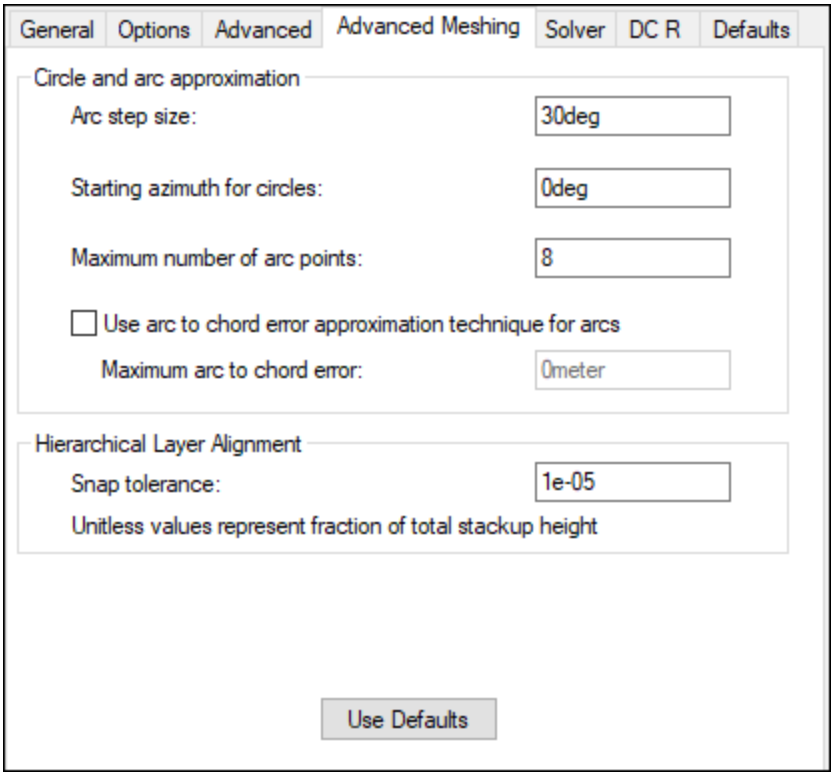
- Defeaturing Options:
  - **Form polygon unions before meshing** – specifies whether all objects are combined or “unioned” before meshing takes place. This can simplify the mesh; it can, however, also remove internal boundaries that may be desirable. “Unioning” is a complex operation and sensitive to almost coincident edges; in some instances, a union may produce undesirable results. Inspecting the mesh is a simple way to verify the operation.

  - **Remove voids with an area smaller than <value>** – simplifies meshing by removing small voids.
  - **Remove Floating/Inactive Signal Net Geometry** – approximates a mesh by removing inactive geometry.
- Model Type Options:
  - **General** – this model offers standard mesh processing.
    - **Use polygon defeaturing** – removes very close points, points that do not contribute to the geometry of an edge (e.g., collinear), and very thin intrusions. When used with "unioning", it can be helpful in healing geometry that is not snapped together.
      - **Tolerance as a ratio of the data exent** – tolerance value specified as a ratio of the overall data extent. This value must be very small.
      - **Absolute distance** – tolerance specified as an absolute value (e.g., 0.0001mm).

      > **Note:**
      >
      > From the **Vias** area, vias may be modeled as a simple 'wirebond' or as a 3D ribbon or solid. The values specified in the analysis setup only apply to vias that do not have specific property overrides; all properties explicitly specified for a via take precedence over these values.

    - **Point -to-edge and point-to-point snapping** – snaps points to edges and points to points if they are within a minimum dimensional tolerance. This tolerance is based on geometry extents.
  - **IC [Beta]** – this model type optimizes mesh processing to simulate on-chip structures.
- Vias Options:
  - **Mesh as a 3D via** – vias are modeled using rectangles to form a 3D ribbon or solid.
    - **Number of sides** – number of sides to use when creating the 3D representation. If less than 3, the via is modeled as a flat ribbon.
  - **Via Field** – this option is disabled in SI Xplorer.
  - **Via material default** – when a material has not been specified in the padstack definition and there is no override in the properties for a via, then this is the material associated with the via mesh. It is initially set to copper.

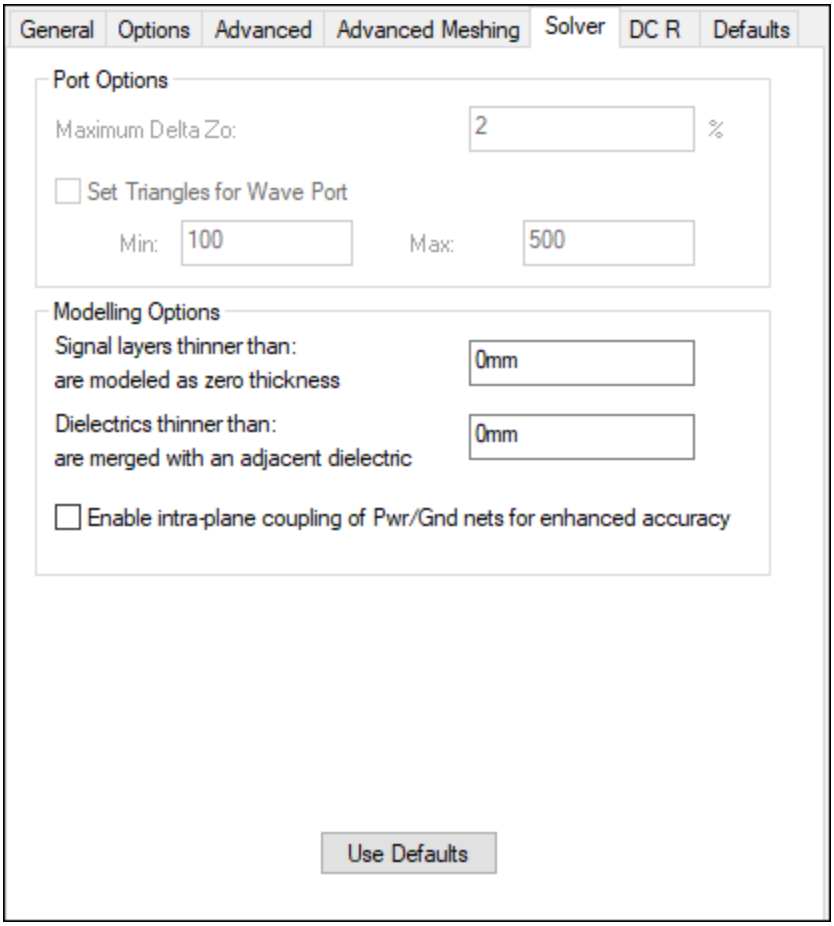
HFSS Advanced Setup – Advanced Meshing Tab



Use the **Advanced Meshing** tab to set the following:

- **Circle and arc approximation** – circles and arcs must be replaced with straight edge approximations before meshing. There is no attempt to match areas. Points are simply introduced on the original arc at the locations specified by the following parameters.
  - **Arc setup size** – the angular interval below which mesh points are added.
  - **Starting azimuth for circles** – specifies the location of the first mesh point or circles.
  - **Maximum number of arc points** – the maximum number of mesh points on a given arc segment.
  - **Use arc to chord error approximation technique for arc** – the resulting straight edge approximation will not deviate from the original arc by more than the specified error.
    - **Maximum arc to chord error** – maximum distance permissible between a straight edge approximation (the chord) and the original arc.
- **Hierarchical Layer Alignment** – aligns layers across hierarchical boundaries.
  - **Snap tolerance** – When layers across hierarchical boundaries are within this unitless tolerance, they are aligned to the same elevation. This value represents a fraction of total stackup height.

HFSS Advanced Setup – Solver Tab



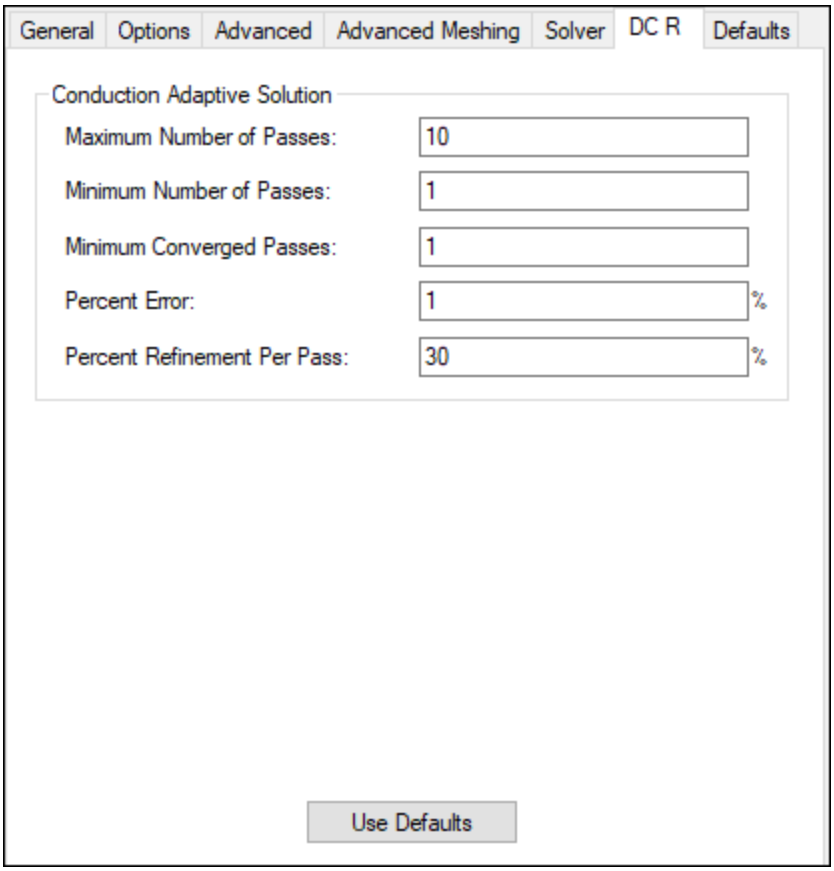
Use the **Solver** tab to set the following:

- Port Options:
  - **Maximum Delta Zo** – specify Zo as a target percentage. The default is 2%.
  - **Set Triangles for Wave Port** – controls the number of triangles used for a wave port. For designs with lumped ports, this option is not active. Higher numbers of triangles would not benefit a solution setup in this case. Set values for the **Min** and **Max** number of triangles.
- Modeling Options:
  - **Signal layers thinner than <value> are modeled as zero thickness** – Negative signal layers in the design which are thinner than the given value are modeled as being infinitely thin in order to reduce simulation time.
  - **Dielectrics thinner than <value> are merged with an adjacent** – Dielectric layers in the design which are thinner than the given value are merged into an adjacent dielectric.

Material properties are accumulated using a weighted average.

- **Enable intra-plane coupling of Pwr/Gnd nets for enhanced accuracy**

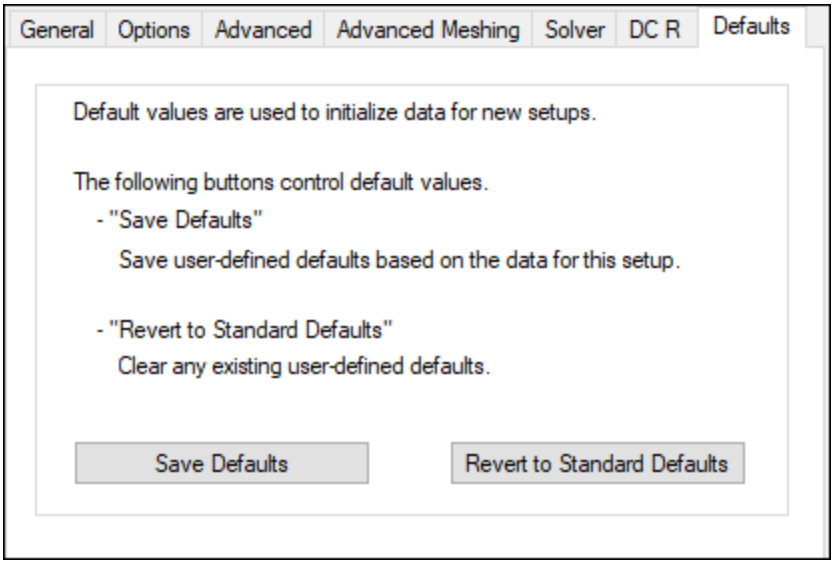
HFSS Advanced Setup – DC R Tab



Use the **DC R** tab to set the following:

- Maximum Number of Passes
- Minimum Number of Passes
- Minimum Converged Passes
- Percent Error
- Percent Refinement Per Pass

HFSS Advanced Setup – Defaults Tab



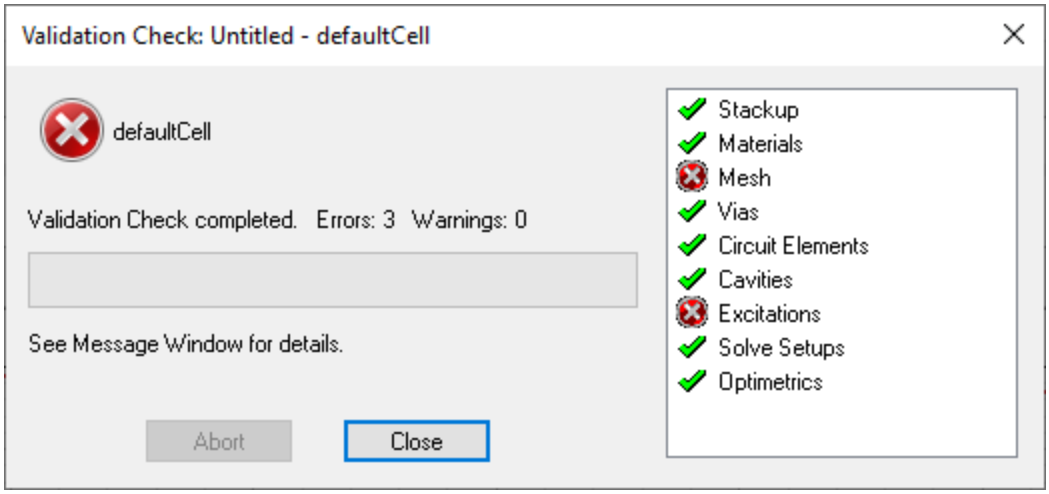
Use the **Defaults** tab to save your settings as the default or revert to the original settings.

Running a Validation Check

To run a validation check:

1. Click **Simulation** > **Validation** check.

   A progress bar displays the validation check progress. When the check has completed, the number of **Errors** and **Warnings** display:



A green check mark appears next to items that have passed validation. A red X appears next to items that have failed validation.

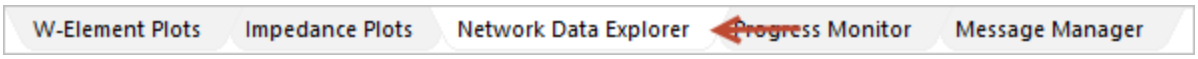
Additional details may be available in the **Message Manager**.

2. Click **Close** to exit.

7 - Network Data Explorer

Network Data Explorer provides visualization, analysis, and manipulation tools for network data.

To access Network Data Explorer, click the **Network Data Explorer** tab in the **Plots and Messages** window:

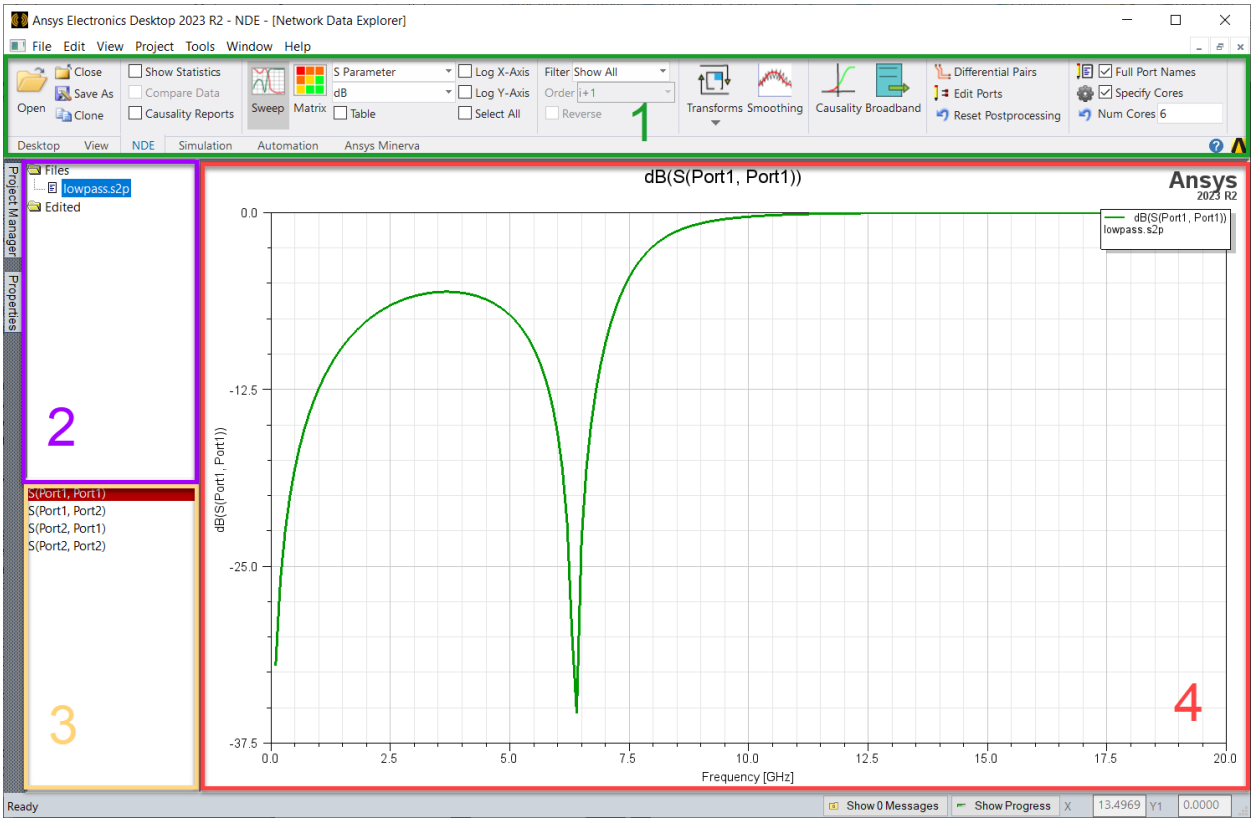


Network Data Explorer Overview

The Network Data Explorer window is divided into the following panes:

1. **Network Data Explorer Ribbon** – allows you to perform Network Data Explorer's functions.
2. **Data Selection Pane** – allows you to select a network data file.
3. **Cell and Frequency Selection Pane** – allows you to narrow your selection.
4. **Data View Pane** – displays data in table or plot format.

The panes are shown in the following figure. Additional information about each pane follows.



Network Data Explorer Ribbon

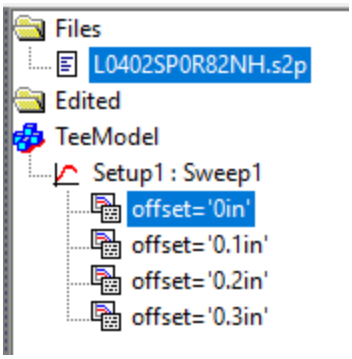
From the ribbon, you can control:

- **Plotting** – determine how the data is displayed. Click the **Sweep** or **Matrix** icon to select a plot type.
- **Quantity** – select the type of quantity to display (parameter values, matrix statistics, or causality plots).
- **Parameter Type** – choose parameters for display (S, Y, or Z parameters, Port Impedance, or Gamma).
- **Format** – decide the display function to apply to the data (e.g., magnitude, phase, dB, real, or imaginary).
- **Export** – export either SYZ data (*.s1p, *.ts, *.nmf, *.tab, *.m, *.cit) or Broadband data (*.sp).
- **Causality Checking** – check causality.
- **Cores** – enable or disable multithreading.
- **Post-Processing** – choose between terminal data and differential pairs, if your design includes differential pairs.

See: Network Data Explorer Ribbon.

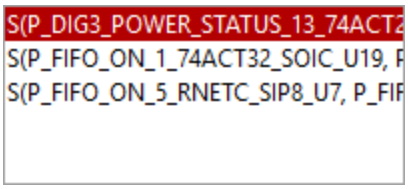
Data Selection Pane

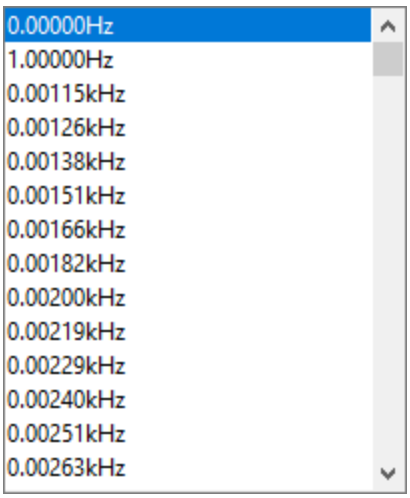
This pane allows you to view and compare various data sets. Original data sets appear under **Files**. Click one of these to see the data set as it was when it was opened. Altered data sets (data sets that have been smoothed, transformed, or changed in some way) are listed under **Edited**.



Cell and Frequency Selection Pane

This pane allows you to choose to choose which cells or frequencies to display.





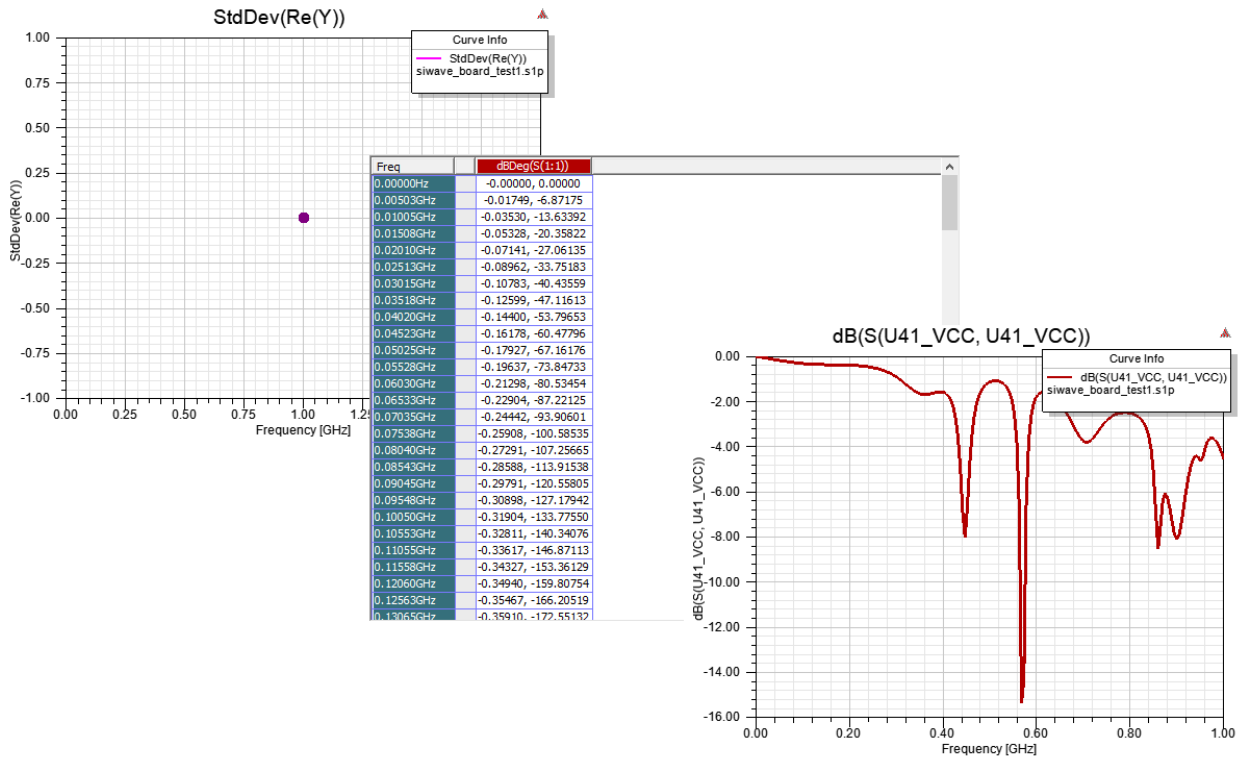
To select all frequencies or cell entries, use the **Select All** check box on the ribbon.

This pane can also list available variations. Selecting a variation activates it and affects table, plot, and statistics displays. When displayed by frequency, the entire matrix is presented in the Data View pane for each selected frequency. When displayed by matrix cell, the data for the individually chosen cells is shown across all frequencies. Use the **Select All** check box to select all variations.

> **Warning:**
>
> Given the volume of data in many network data sets, choosing **Select All** may take a considerable time to generate, especially in Plot view.

Data View Pane

The Data View Pane displays either table or plot data, depending on your selections.



Loading Data into Network Data Explorer

You can launch **Network Data Explorer** from within several Ansys products. From the Project Manager, open <project> > **Analysis** > **Setup***n* > **Sweep***n*. Right click on the sweep and click **Network Data Explorer** in the shortcut menu. The current solution data is automatically loaded and ready for viewing. Otherwise, you must load a data file into Network Data Explorer.

> **Note:** When solution data that is loaded into NDE is modified and resimulated in another Ansys product, the NDE data automatically updates.

You can import the following file types:

- Touchstone Format (*.s*p)
- Touchstone 2 Format (*.ts, *.sp)
- Citifile (*.cit)
- Neutral Format (*.nmf)
- State Space File (*.sss)

> **Note:**
>
> When this type of file is loaded, Network Data Explorer regenerates s-parameter data based on the file.
>
> You can compare the regenerated s-parameters to the original data.

To import a file into Network Data Explorer, either drag and drop an analysis from the Project Manager into Network Data Explorer or

1. On the **NDE** ribbon, click **Open**. An **Open** window opens.
2. Navigate to and select a file.
3. Click **Open**. The file appears in the **Files** tree.

The file browser allows you to open multiple files at a time. However, the displayed data always corresponds to the data set indicated in the Network Data Selection pane. Click the file you want in that pane to switch between data sets.

Exporting Data from Network Data Explorer

Network Data Explorer allows you to export data to a variety of different file formats.

In this section, you will learn about:

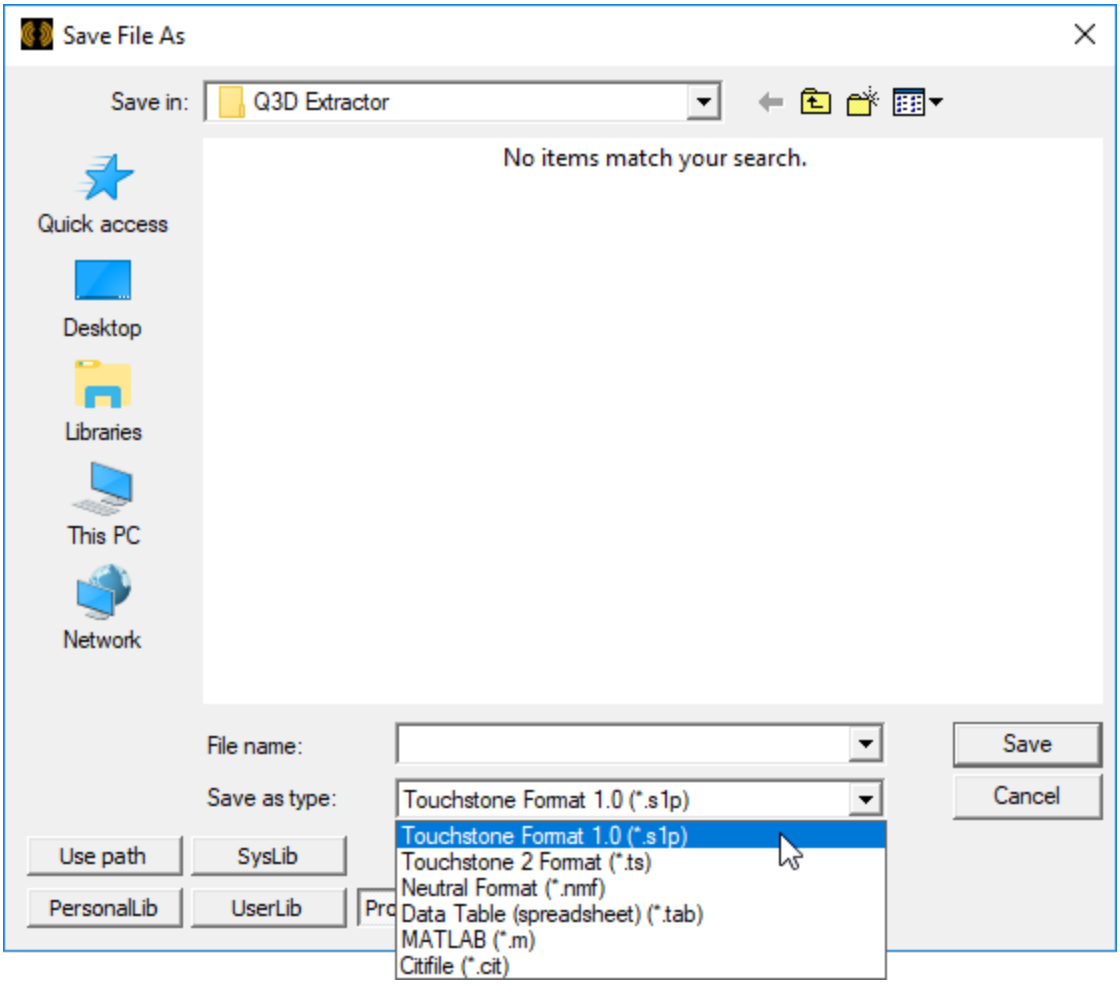
Exporting SYZ Data

Exporting Macro Model

Creating an NPort Model

Exporting SYZ Data

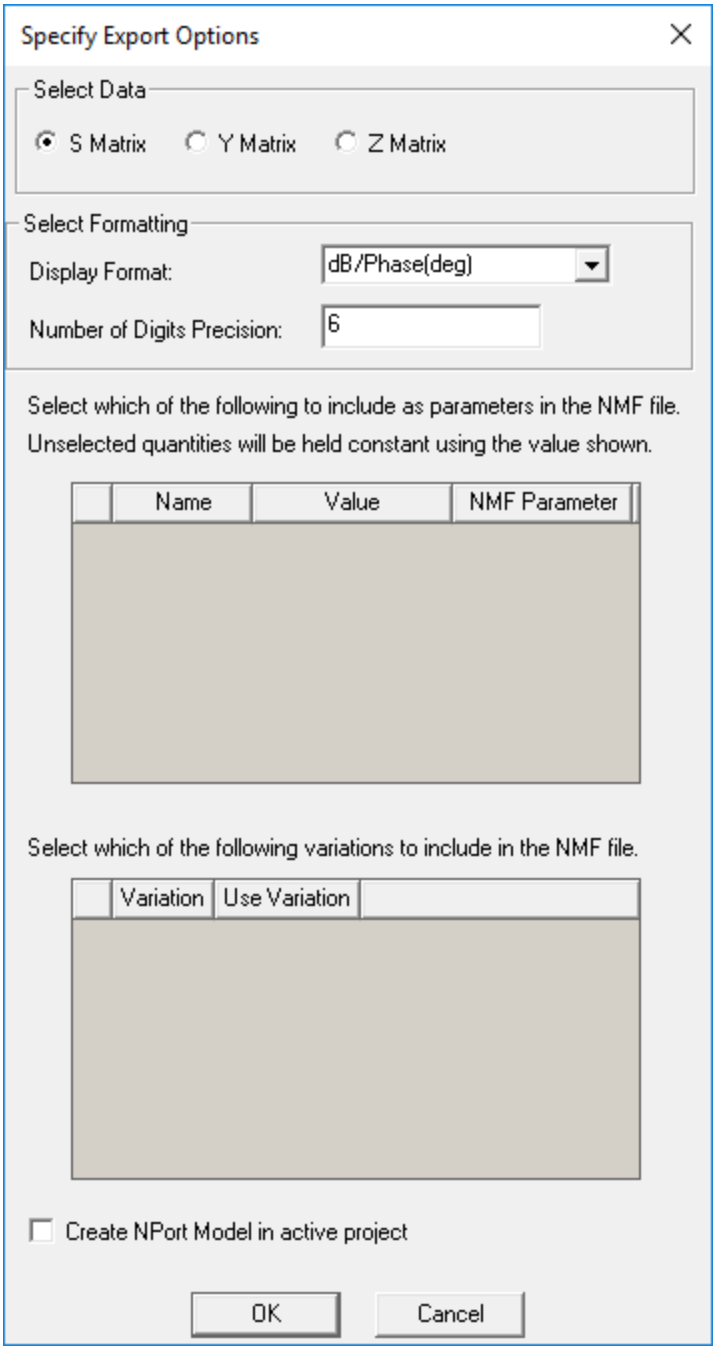
To export SYZ data from within Network Data Explorer, click the **Save As** icon on the **NDE** ribbon. The **Save File As** window appears.

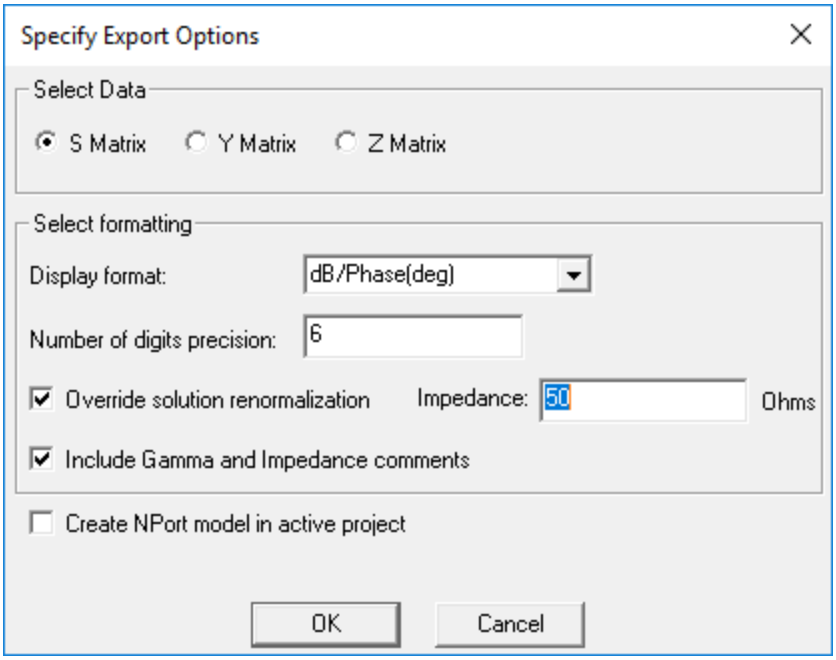


You can export data in any of six file types:

- Touchstone Format 1.0 (*.s*p)
- Touchstone 2 Format (*.ts)
- Neutral Format (*.nmf)
- Data Table Spreadsheet (*.tab)
- MATLAB (*.m)
- Citifile (*.cit)

Select a file type and name for export. A **Specify Export Options** window appears.





Depending on the type of export file, different options appear. However, all file types allow you to:

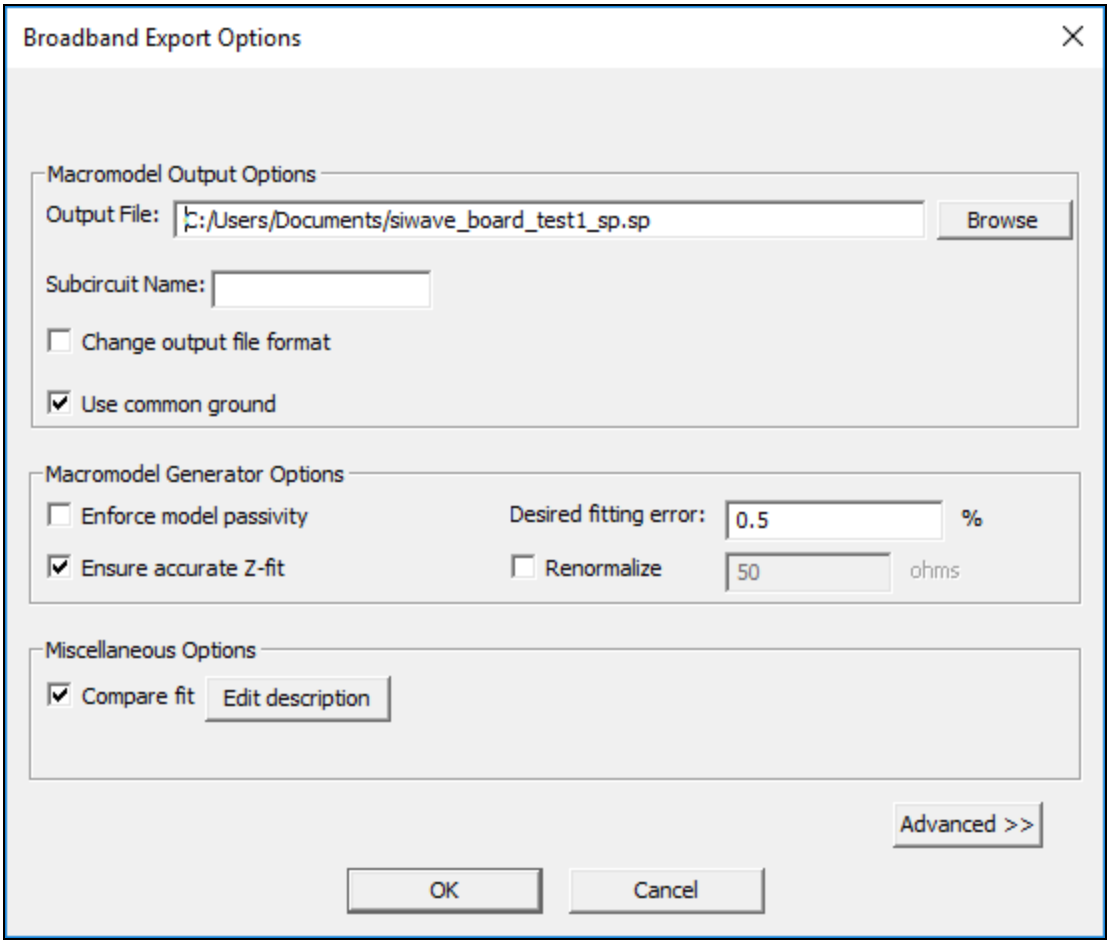
- Select from **S Matrix**, **Y Matrix**, and **Z Matrix** data.
- Select the **Display Format**.

Some types allow you to create an NPort Model in the active project.

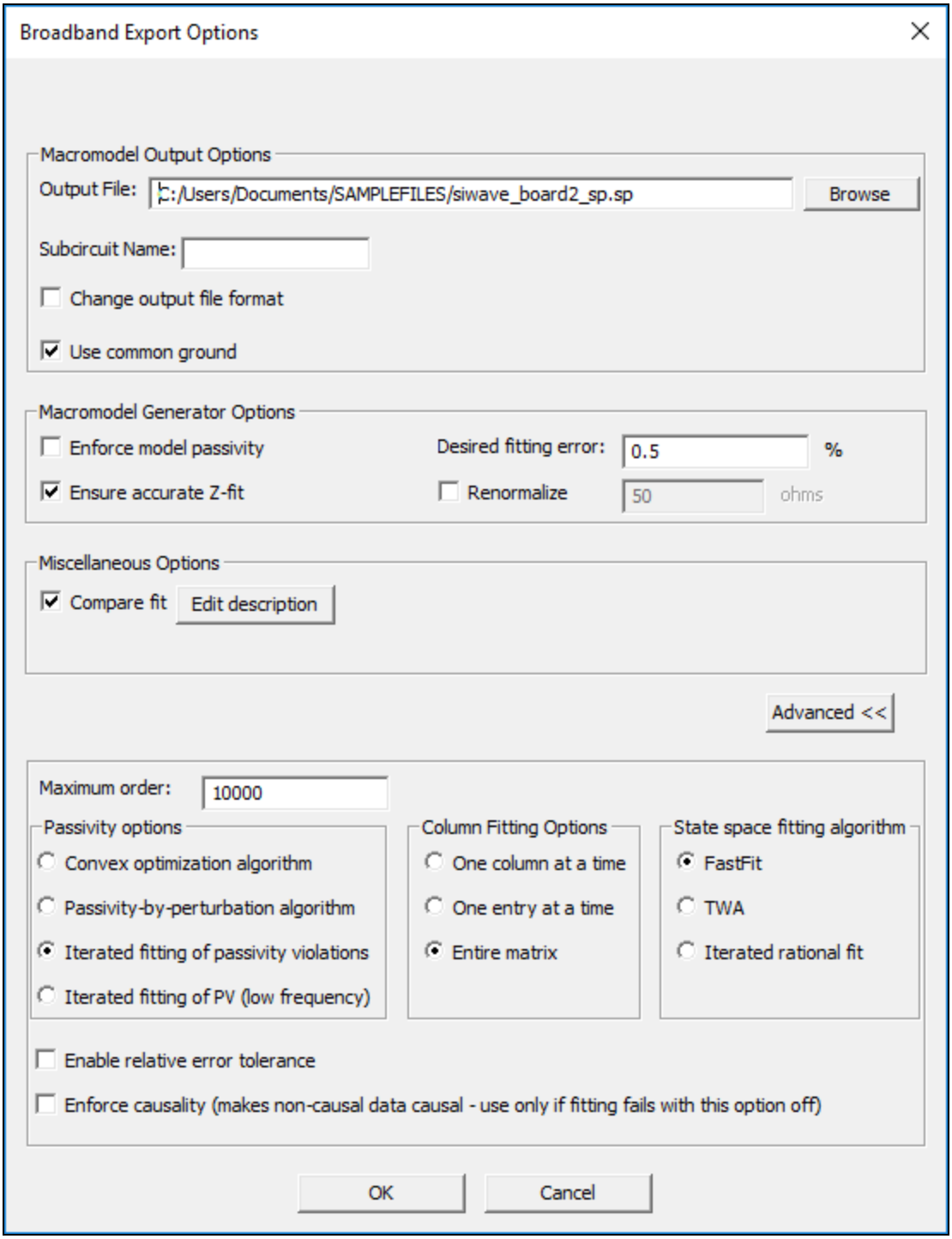
Exporting Macro Model

Network Data Explorer lets you export macro model data. To export data, click the **Broadband** icon on the **NDE** ribbon.

The **Broadband Export Options** window appears.



Click **Advanced** to view all options.



Macromodel Output Options include:

- **Output File** – Allows you to choose the name and location of the file.
- **Subcircuit Name** – Use this field to name the subcircuit.

- **Change Output File Format** – Check this box to open a submenu allowing you to select a new output format.
- **Use Common Ground** – Check this box to use common ground. When this option is on, ports are referenced to ideal ground (node 0). When this option is off, extra ports are generated to provide the reference levels. Common grounding is ideal when the pins are physically near to each other and ideal ground is suitable. For distant connections and circuits with non-ideal reference levels such as differential pairs, common grounding is not used.

> **Note:**
>
> - R and L values may be quite sensitive to the values of the S-parameters. This is an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
> - Since resistances of power cables is typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to get multiplied by 5000 times when the R and L values are computed.
> - As a general rule, for high power applications a reference impedance of 1 ohm is probably a better choice than 50 ohms.

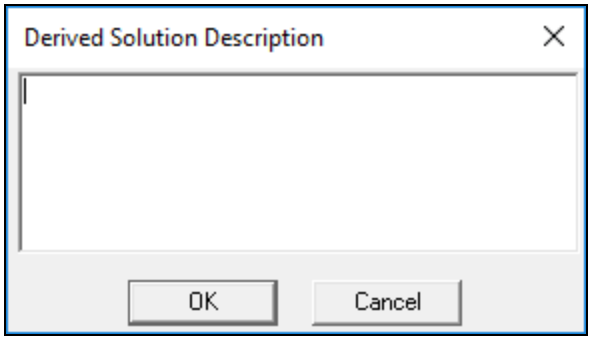
Macromodel Generator Options include:

- **Enforce Model Passivity** – Check this box to enforce passivity.
- **Ensure Accurate Z-fit** – Check this box when state-space fitting of Y-parameters or Z-parameters does not produce an accurate fit.
- **appropriate Fitting Error** – Allows you to select the value at which rational fitting fails (if the fitting error exceeds this value).
- **Renormalize** – Check this box to renormalize using the specified impedance item. 50 ohms is the default setting, but you can type a different value.

> **Note:**
>
> - R and L values may be sensitive to S-parameter values. This presents an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
> - Since resistances of power cables are typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to be multiplied by 5000 when the R and L values are computed.
> - For high-power applications, a reference impedance of 1 ohm is generally a better choice than 50 ohms.

Miscellaneous Options include:

- **Compare Fit** – When this box is checked, the original and derived solution will be available for comparison. You can click **Edit Description** to open the **Derived Solution Description** window and add a text description to better identify the export.



Advanced Options include:

- **Maximum Order** – Allows you to specify the number of poles. See Note below.
- **Passivity Options** – If you enabled **Enforce Model Passivity**, this area allows you to select the passivity enforcement method.
  - **Convex optimization algorithm** – guarantees a passive state-space realization, but is very slow and memory-intensive. Not practical for numbers of ports beyond ten.
  - **Passivity-by-perturbation algorithm** – designed for systems with a large number of ports. Less accurate than the Convex optimization method.
  - **Iterated fitting of passivity violations (IFPV)** (default) – less accurate than other algorithms but more suitable for larger numbers of ports.
  - **Iterated fitting of PV (low frequency)** – similar to IFPV while improving the fit to “Z” at DC and low frequencies. A better choice of passivity enforcement when the fit to corresponding “Z-data” is important such as power delivery, EMI/EMC applications.

> **Note:**
>
> For a more detailed explanation on any of the passivity options, consult the Circuit help.

- **Column Fitting Options** – This area allows you to choose how poles are matched to columns:
  - **One Column at a Time** – The set of poles will be shared across all entries of a single column.
  - **One Entry at a Time** – Each entry will be fitted using a separate set of poles.

- **Entire Matrix** – The set of poles will be shared across all entries of the matrix being fitted.

> **Note:**
>
> - Typically, using the same set for all entries is adequate, and yields the most compact models. However, if all the entries of the matrix have completely unrelated transfer functions, it may be better to fit them using separate pole sets.
> - The options **One column at a time** and **One entry at a time** do not work when either **Ensure accurate Z-fit** or FastFit is used.

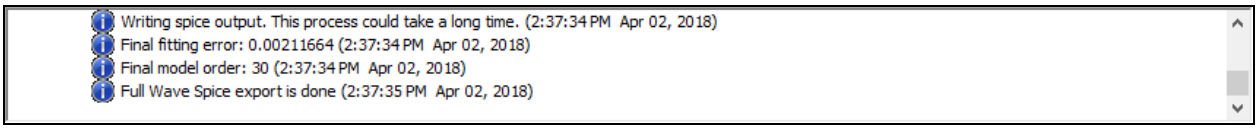
- **State Space Fitting Algorithm** – Allows you to select FastFit, TWA, Iterated rational fitting.
  - **FastFit** (default) – FastFit is the ANSYS-proprietary method for state-space fitting. Network Data Explorer uses FastFit for calculating the state-space matrices from the network data. The FastFit algorithm for state-space fitting is an alternative to the Tsuk-White algorithm (TWA) and Iterated Rational Fitting (IRF) methods. FastFit is generally as accurate as TWA, but is significantly faster than both TWA and IRF. It also aims to fit the lower frequencies with higher fidelity.
  - **TWA** – The Tsuk-White Algorithm is an ANSYS-proprietary method for fitting a state space model to extracted s-parameter data. It uses techniques based on Singular Value Decomposition (SVD) to quickly determine required number of poles for fitting a model.
  - **Iterated Rational Function** – The IRF fitting approach takes a matrix of S-parameter data and, for each matrix entry, tries a succession of different pole-zero approximations (increasing the number of poles used at each iteration) until it can find an acceptable fit to the data. For broad frequency sweeps and large numbers of excitations, this process can be time consuming because of all the iterations and is not guaranteed to produce a good fit to the data. It is retained as a fallback if the TWA algorithm fails.
- **Enable Relative Error Tolerance** – Allows you to enable relative error tolerance, which works best with TWA fitting.

> **Note:**
>
> The **Enable Relative Error Tolerance** option works best with the TWA fitting algorithm, is not recommended for use with iterated rational fitting, and is disabled when either FastFit or **Ensure accurate Z-fit** is used.

- **Enforce Causality** – Allows you to make non-causal data causal. Use this option only if fitting fails without it.

> **Note:**
>
> Broadband models are built from a rational-function approximation of the data. The fidelity of this approximation can be controlled by setting the Maximum order (number of poles).

Click **OK** to begin the export. The **Messages** pane details the export process.

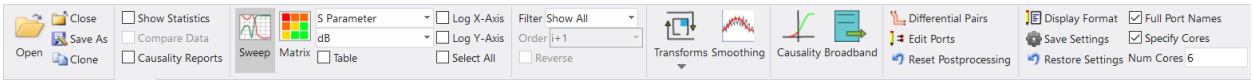


Comparing Original S-Parameters with Exported S-Parameters

If **Compare Fit** was checked during export, the **Data Selection** pane updates to list both the original and exported solution *and* the **Compare** checkbox is checked.

Network Data Explorer Ribbon Actions

The Network Data Explorer ribbon provides access to many of Network Data Explorer's functions.



In this section, you will learn about:

Setting Display Format

Displaying Full Port Names

Saving or Restoring Default Settings

Smoothing All Frequencies

Cell Filtering

Editing Port Properties

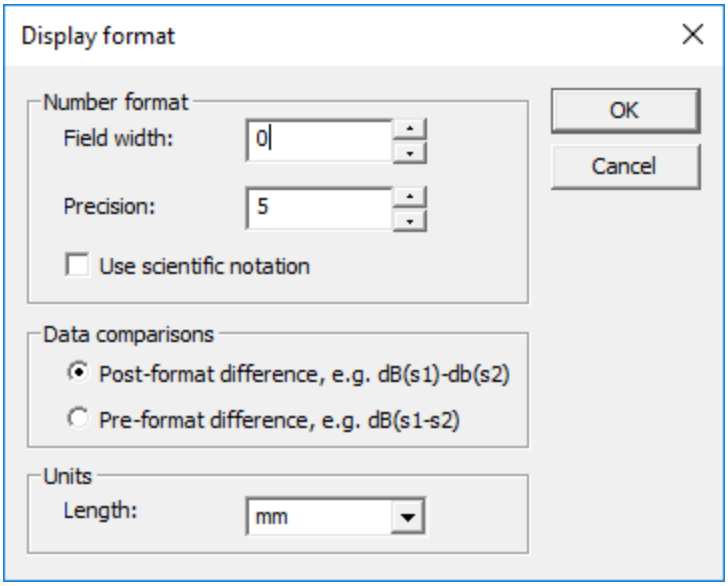
Defining Differential Pairs and Displaying Mixed Mode Parameters

Network Data Explorer – Setting Display Format

The **Display Format** window affords additional control over the display of values in Network Data Explorer.

To access this window from the ribbon, click **Display Format** ( ).

The **Display Format** window appears.

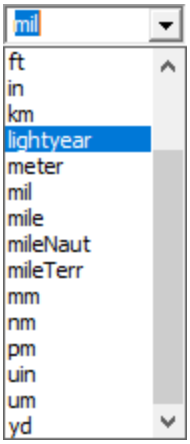


The **Number Format** options allow you to specify **Field Width** (the minimum number of characters used to display a number) and **Precision** (the number of decimals to display). You can also check the **Use scientific notation** check box, if appropriate.

The Data Comparisons options allow you to choose Post-format difference or Pre-format difference.

- **Post-format difference** – when comparing datasets, subtract values after applying the formatting function (dB, magnitude, etc.); the values displayed will be the difference between the magnitude, dB, and so on.
- **Pre-format difference** – when comparing datasets, subtract values before applying the formatting function (dB, magnitude, etc.); the values displayed will be the magnitude, dB, etc., of the complex difference.

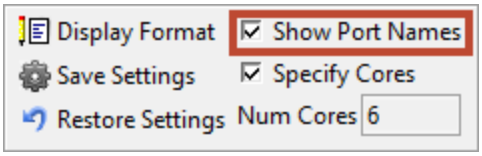
The **Units** option allows you to specify the **Length** unit (the unit used to display and interpret length values). The default is mm.



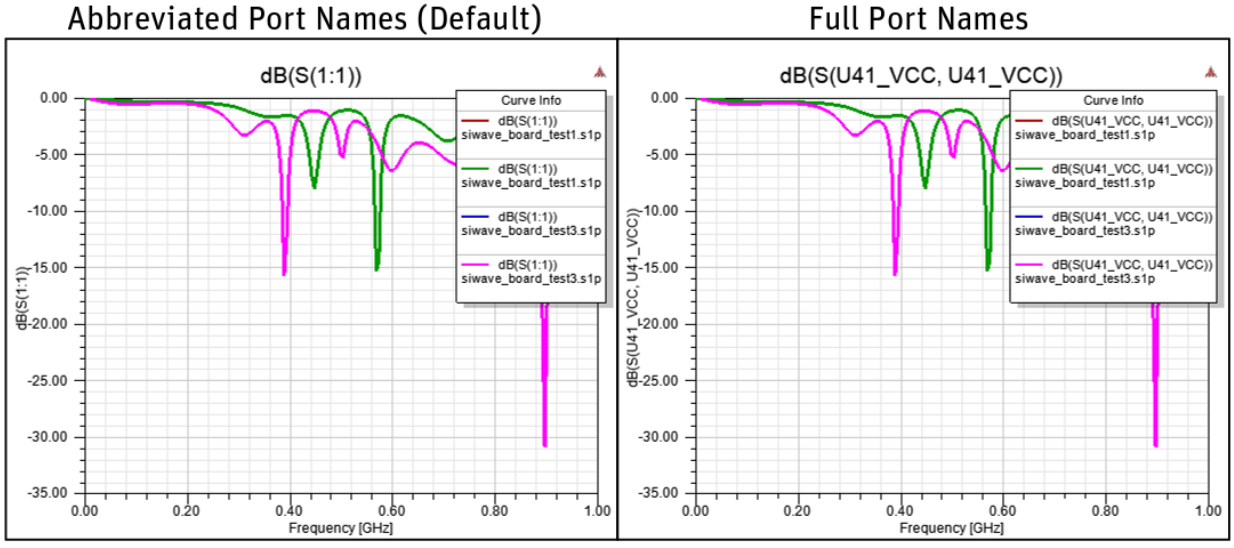
Network Data Explorer – Displaying Full Port Names

By default, port names in Network Data Explorer are displayed in an abbreviated form: P1, P2, etc. This applies to both the Data Selection pane and the Data View pane.

To change this so that full port names are displayed, select **Show Port Names** on the ribbon:



The following figure shows the difference in display for a plot.

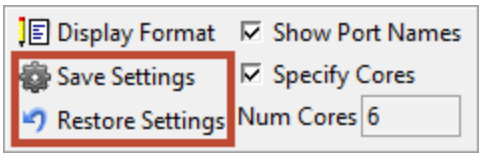


> **Note:**
>
> Tool-tips always display the full port name.

Network Data Explorer – Saving or Restoring Default Settings

To save field settings as the default, click **Save Settings**. The next time Network Data Explorer is opened, the chosen settings will be selected by default.

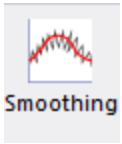
To restore the original defaults, click **Restore Settings**. The next time Network Data Explorer is opened, the original settings will be selected by default.



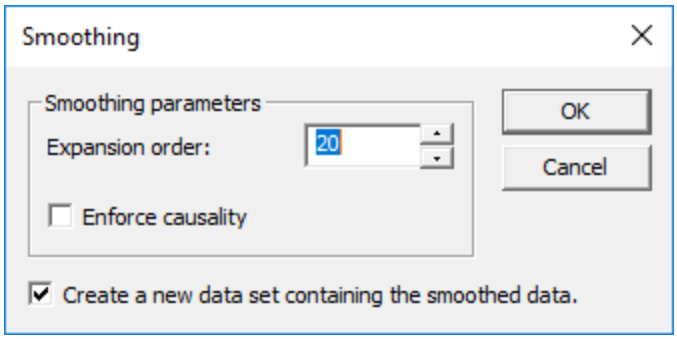
Smoothing

To access data smoothing options:

1. Click the **Smoothing** icon on the **NDE** ribbon.

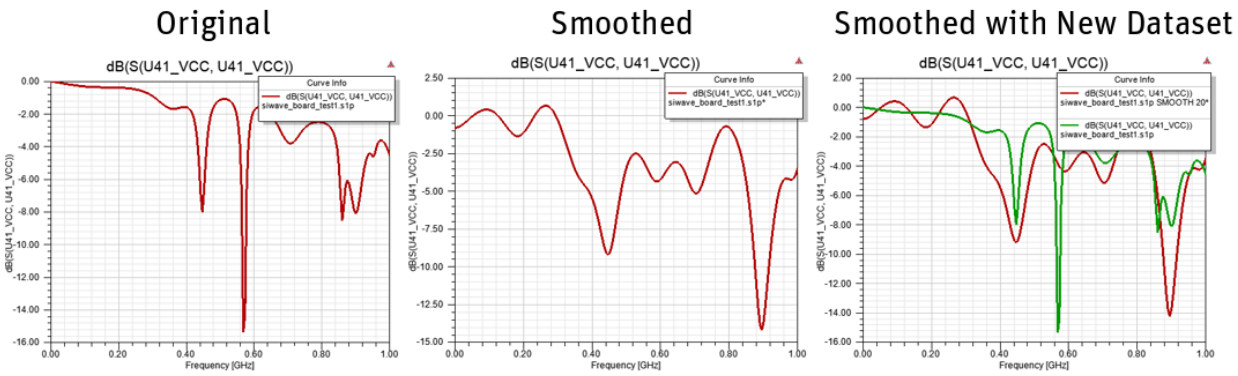
   The **Smoothing** window appears.

2. The **Smoothing Parameters** area allows you to choose the **Expansion Order**. This can be any discrete value between 1 and 150.
3. If appropriate, check the **Enforce Causality** check box.

4. If appropriate, check the **Create a new data set containing the smoothed data** check box. If selected, the smoothed data appears alongside the original data.
5. Click **OK**.

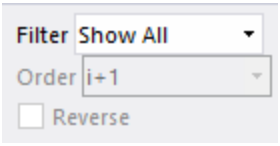
The Data View Pane updates. The following image shows a plotted result with **Create a new data set containing the smoothed data** unchecked and checked.



A least-squares polynomial fit of the specified order is used to interpolate new data points for the magnitude and phase components of the S-parameters.

Cell Filtering

The cells available in the data selection pane may be restricted using cell filtering. The **Cell Filtering** control are located on the **NDE** ribbon.



Cell filtering is modeless, and filters are immediately applied to the cell list. Filtering remains in effect when the window has been closed.

For an n-port model with a total of 2n pins in the standard arrangement, the choices are:

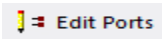
- **Show all** – display all available cells. There are n-squared choices.
- **Return loss** – show S(i, i). There are n choices.
- **Insertion loss** – show S(i, i+1). There are n choices.
- **Lower triangle** – show S(i, j) for all j < i. There are n(n-1)/2 choices.

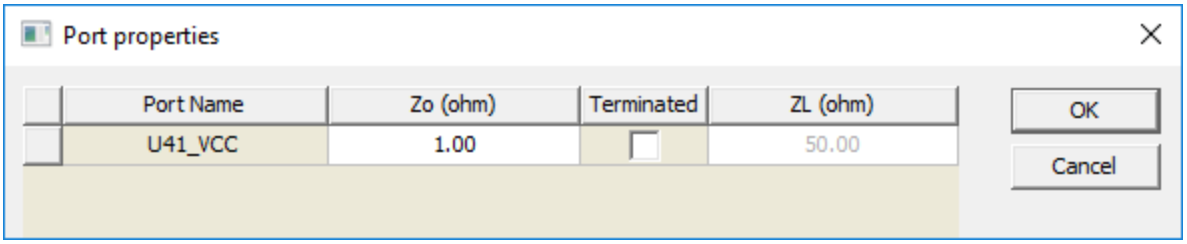
Three pin arrangements are recognized:

- S(i, **i+1**) and its **Reverse order**, S(i, i-1)
- S(i, **i+n**) and its **Reverse order**, S(i, i-n)
- S(i, **2*n-i+1**) and its **Reverse order**, S(i, 2*n-i-1)

Changing Port Properties and Reducing Matrix Size

The normalization impedance, termination, port order, gamma values, and de-embedding distance may all be edited from the **Port Properties** window.

To access these options, click **Edit Ports**  on the **NDE** tab. The **Ports Properties** window opens.



> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. Users must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Ports appear in a table. Click a column heading to sort by that column. Click within a cell to edit the port property:

- **Zo (ohm)** and **ZL (ohm)** – specify Impedance values. Accepted syntaxes are:
  - real (e.g., 50)
  - real + imag i (e.g., 50+5i)
  - imag i (e.g., 5i).
- **Terminated** – use the check box to terminate a port. Terminated ports are eliminated from the matrix, reducing the matrix size. Existing data sets with mismatching port numbers will no longer be available for data comparisons.
- **De-Embedding** – this column appears only if gamma values are available. Default units can be changed from the Set Display Format window.

To reorder ports, click and drag a row to a new location.

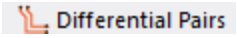
To save changes, click **OK**.

Displaying Mixed-Mode Parameters using Differential Pairs

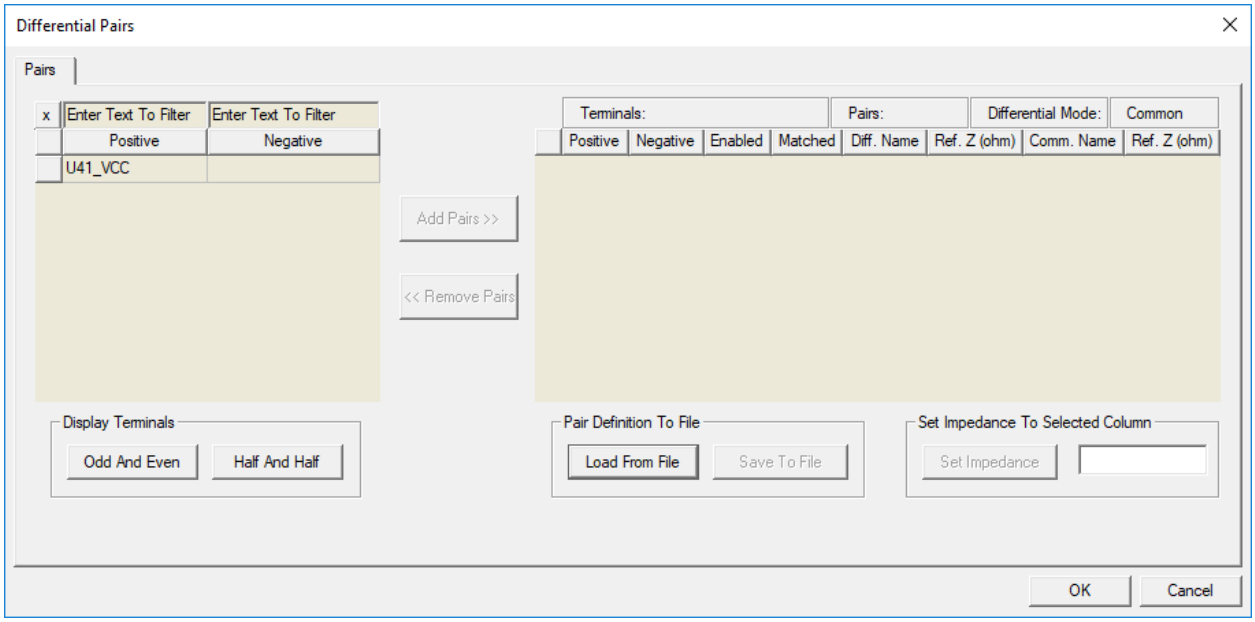
Network Data Explorer displays mixed-mode parameters when differential pairs are both defined and activated.

To define differential pairs:

1. Select existing ports.
2. Open Network Data Explorer (**Tools** > **Network Data Explorer**).
3. On the **NDE** ribbon, select **Differential Pairs**.

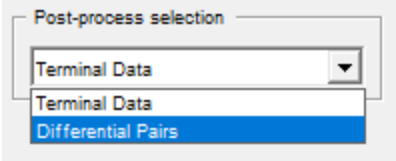


The **Differential Pairs** window appears.



4. Select a pair from the list on the left and click **Add Pairs**.
5. Click **OK**.
6. To disable all differential pairs, click the **Enabled** column header in the **Differential Pairs** dialog to deselect all pairs.

For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. Users must make port/differential pair changes directly in the HFSS Design and re-import the solution. When the HFSS Driven Terminal Design has differential pairs defined, NDE shows this drop down menu and you can change between showing reports for Terminals or Differential Pairs.Select **Differential Pairs** from the **Post-process selection** field to view mixed-mode parameters.



Note:

The Network Data Explorer **Edit** menu option **Reset All Port Properties** deactivates all pairs, but it does not clear the differential pair settings. And since **Reset All Port Properties** also clears reference impedances and terminations, it should not be used when users want to disable all differential pairs.

Data View Pane Context Menus

The Data View pane presents different right-click menu options, depending on the context. Some commands are the same as those on the NDE ribbon. Others appear only in the context menus.

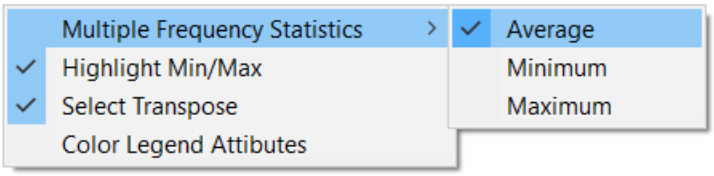
Network Data Explorer commands unique to the context menus are:

- Multiple Frequency Statistics
- Highlight Min/Max
- Select Transpose
- Color Legend Attributes
- Matrix Entries Plot Menu

Multiple Frequency Statistics

The **Multiple Frequency Statistics** menu option determines the statistical composite to display when multiple frequencies have been selected for the matrix display. The statistical data is always the first matrix displayed, followed by matrices for each individual frequency. The **Multiple Frequency Statistics** option also indicates the data used in the colored matrix plot when multiple frequencies have been selected.

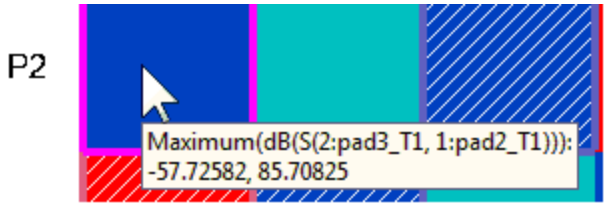
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



The menu options are:

- **Average** – display the average of the matrix values across selected frequencies.
- **Minimum** – display the minimum matrix values across selected frequencies.
- **Maximum** – display the maximum matrix values across selected frequencies.

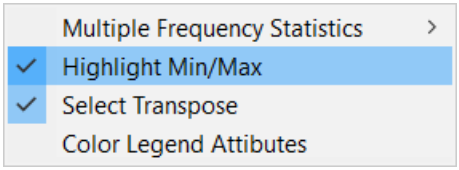
The selected information also appears in a tool-tip when you hover the cursor over a cell.



Highlight Min/Max

The **Highlight Min/Max** menu option determines whether the minimum and maximum matrix entries should be highlighted in the matrix table and color plot view.

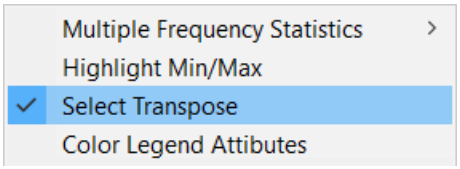
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



Select Transpose

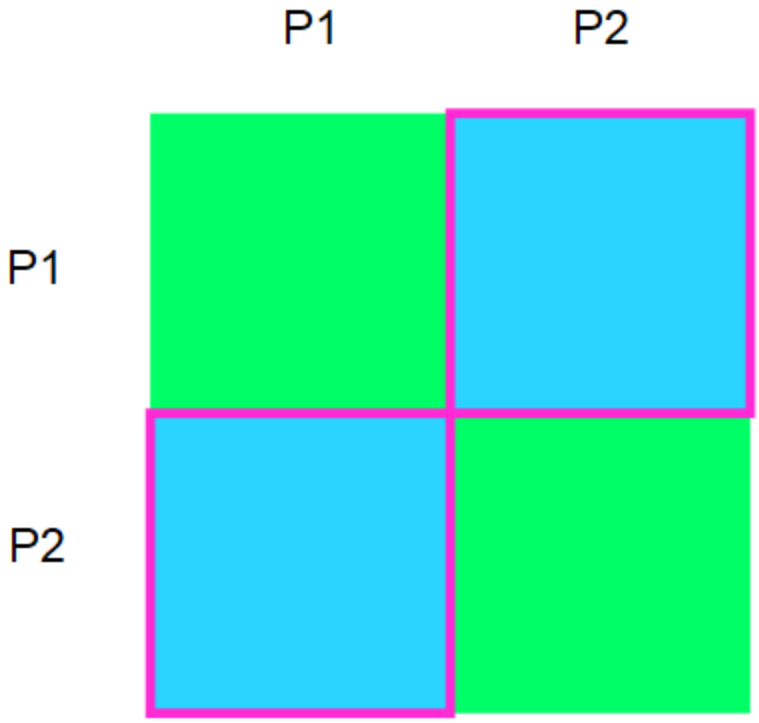
The **Select Transpose** menu option determines whether transpose cells are highlighted along with selected cells.

This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



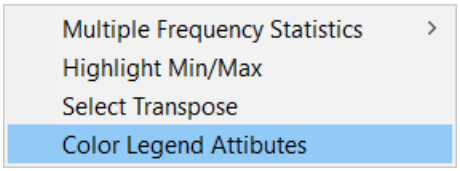
Transpose cells are highlighted in Table or Plot view, as shown below.

0.01000GHz	P1	-57.10340, -81.97841	-57.72581, 85.70836	-0.00213, -0.18916
	P2	-57.72582, 85.70825	-57.09556, -81.98749	-66.27113, 99.16916
	P3	-0.00213, -0.18916	-66.27110, 99.16941	-57.10246, -81.95892
	P4	-66.27112, 99.16942	-0.00213, -0.18929	-57.72556, 85.71465

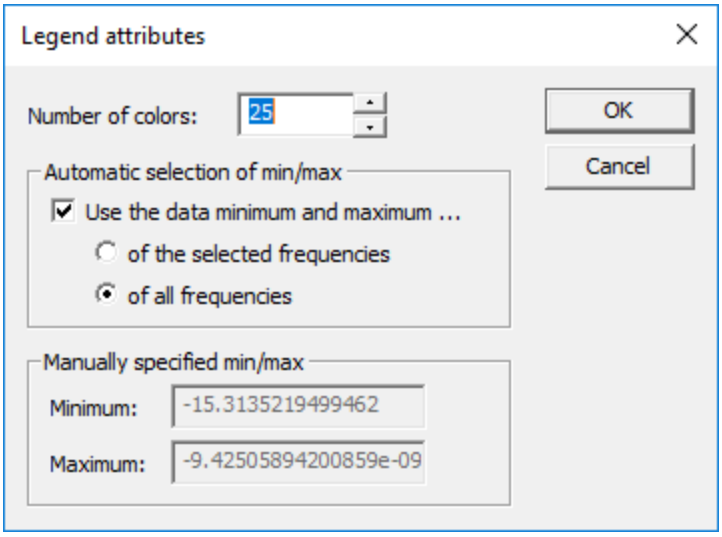


Color Legend Attributes

The **Color Legend Attributes** menu option allows you to change the granularity of the color scheme and the value range for plots. This right click menu option appears in the **Matrix** plot.



Alternatively, double-click the matrix plot's legend to open the **Legend Attributes** window.



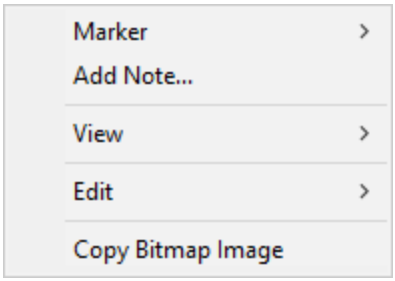
Options include:

- **Number of colors** – allows you to select number of color entries in the legend (the number of divisions between the start/end of the data range). This can be set to any discrete number between 1 and 50.
- **Automatic selection of min/max** – check the **Use the data minimum and maximum** check box to automatically select the data range using the minimum and maximum values from either selected frequencies or all frequencies in the data set.
- **Manually specified min/max** – when the when the range is not automatically determined, these fields permit users to manually enter hard values (e.g., for S parameter data magnitude data, you could enter a minimum of 0 and a maximum of 1).

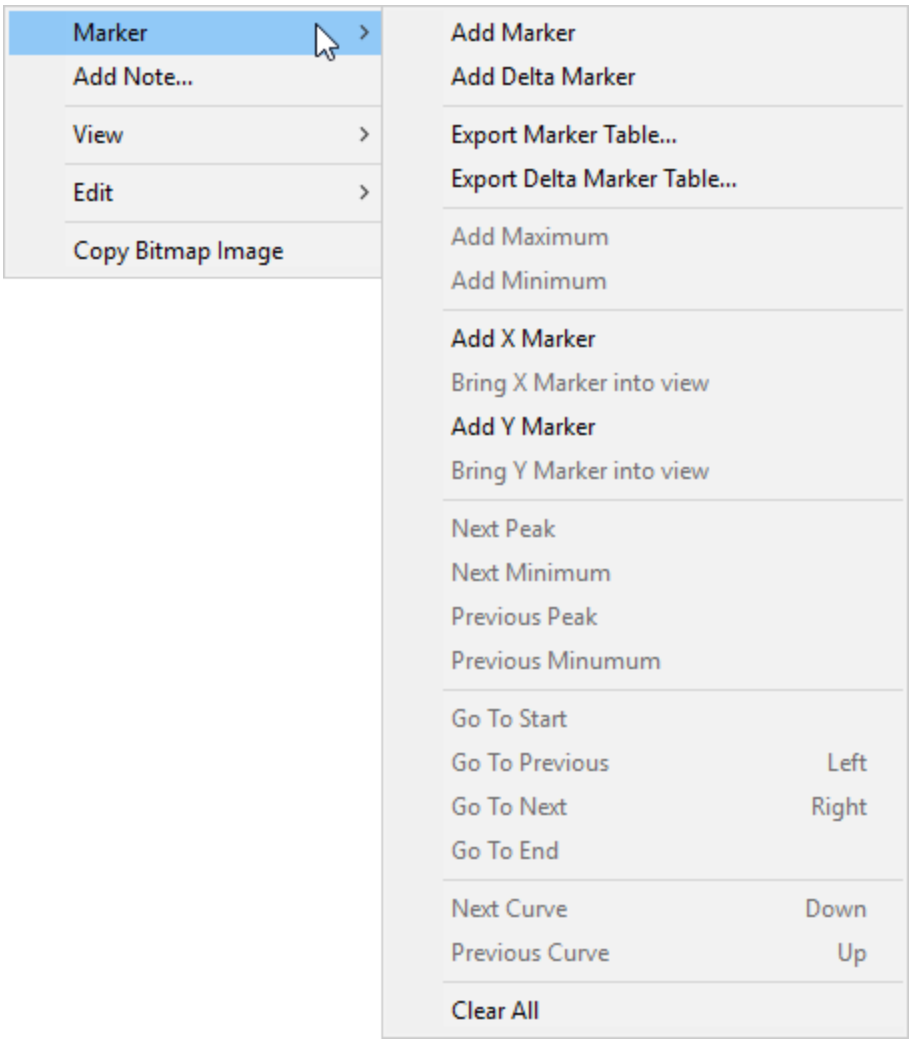
Using a standard range across all frequencies permits you to quantitatively compare plots, and ndExplorer retains legend settings for each data-type and display-format pair.

Matrix Entries Plot Menu

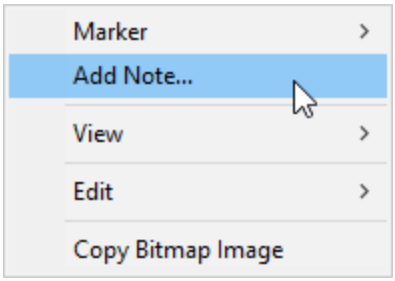
On the **Matrix** plot, several new right-click menu options appear.



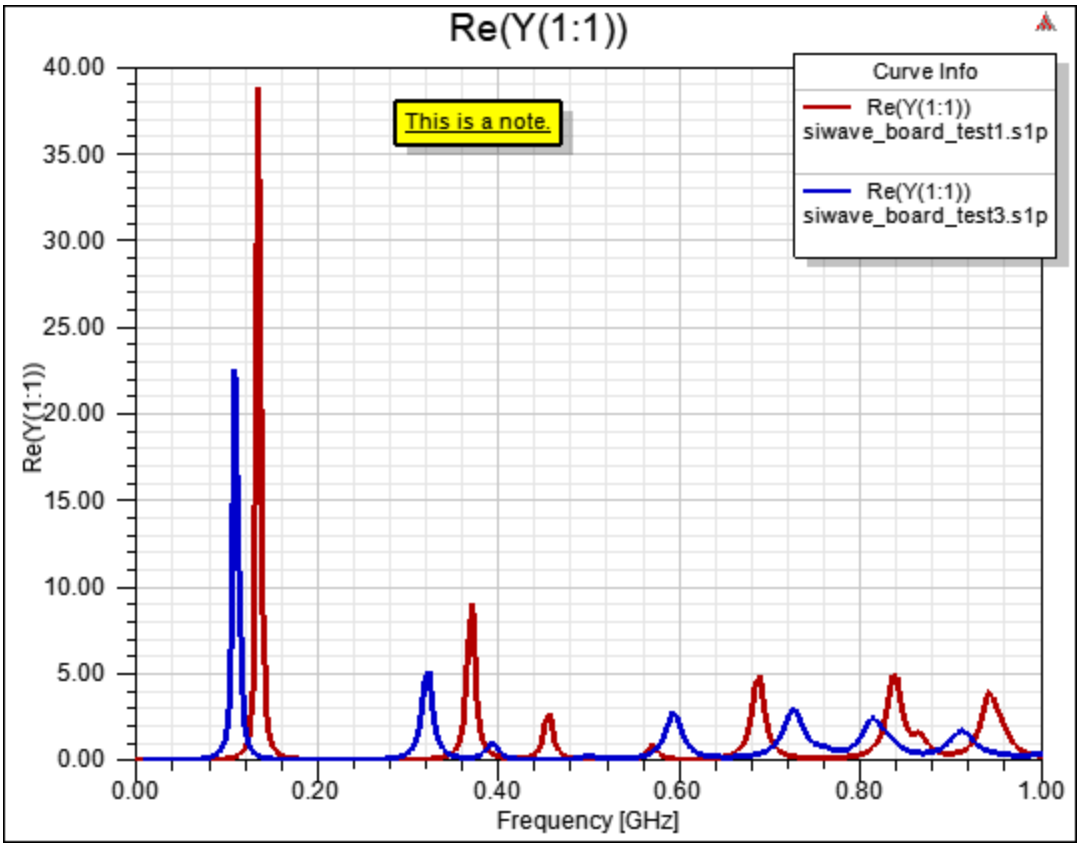
The **Marker** sub-menu provides commands for adding markers to plots.



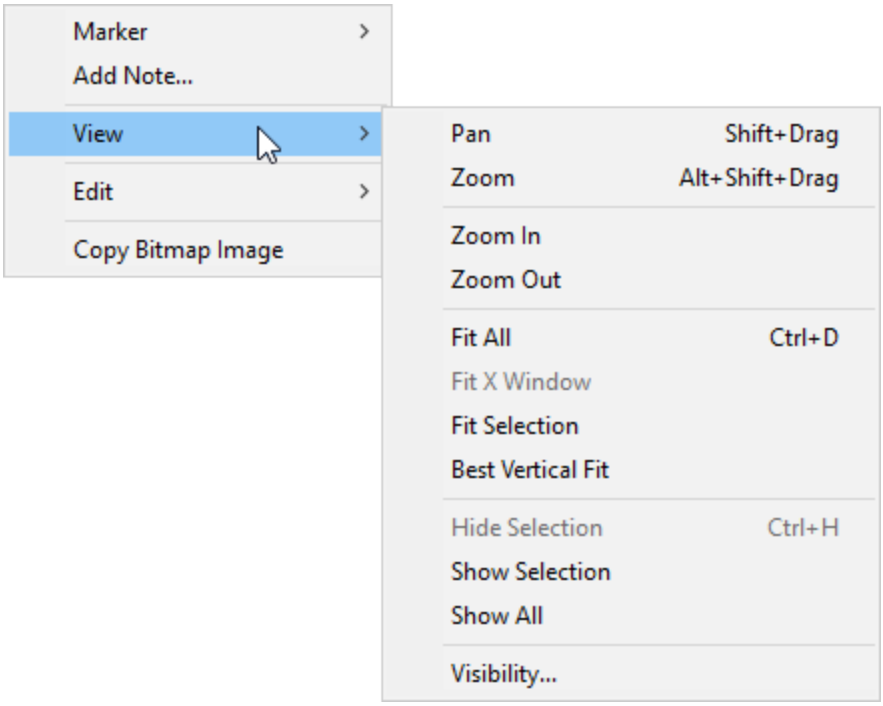
The **Add Note** menu option allows you to add a note to the plot.



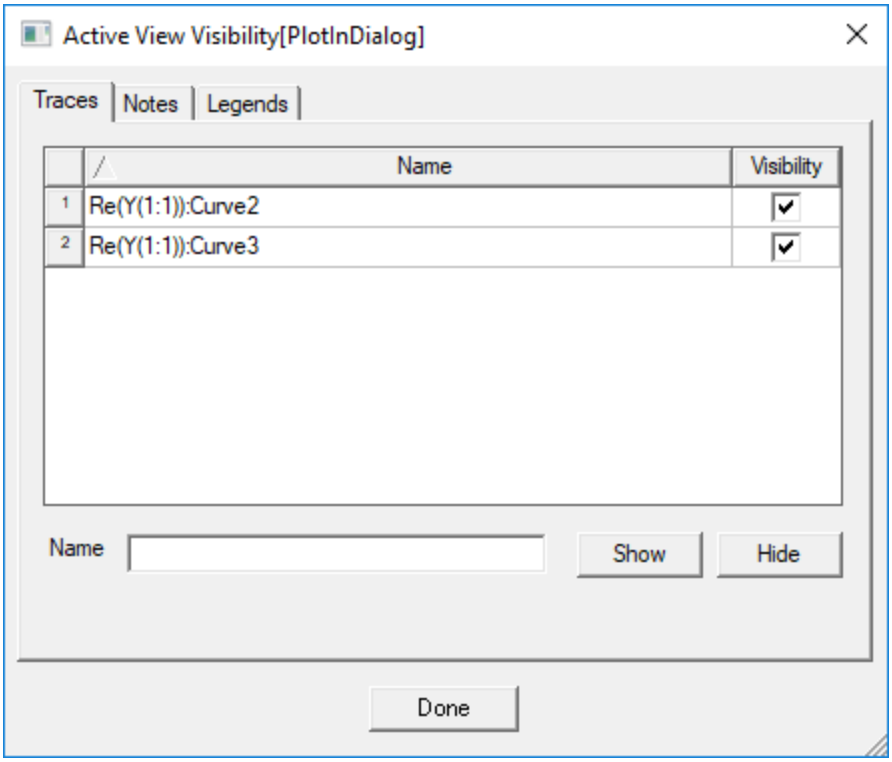
The note appears at the location you right-clicked. You can click and drag the note to a new location, or double-click the note to change its color and font.



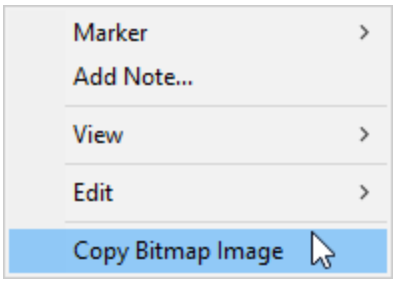
The **View** sub-menu provides commands for viewing, panning, zooming, and fitting elements to your plot.



Click **Visibility** to open the **Active View Visibility** window, where you can select the visibility of traces, notes and legends.



The **Copy Bitmap Image** menu option copies the plot to your clipboard. You can then paste it into a graphics editor.



Exploring Network Data and Modifying the Display

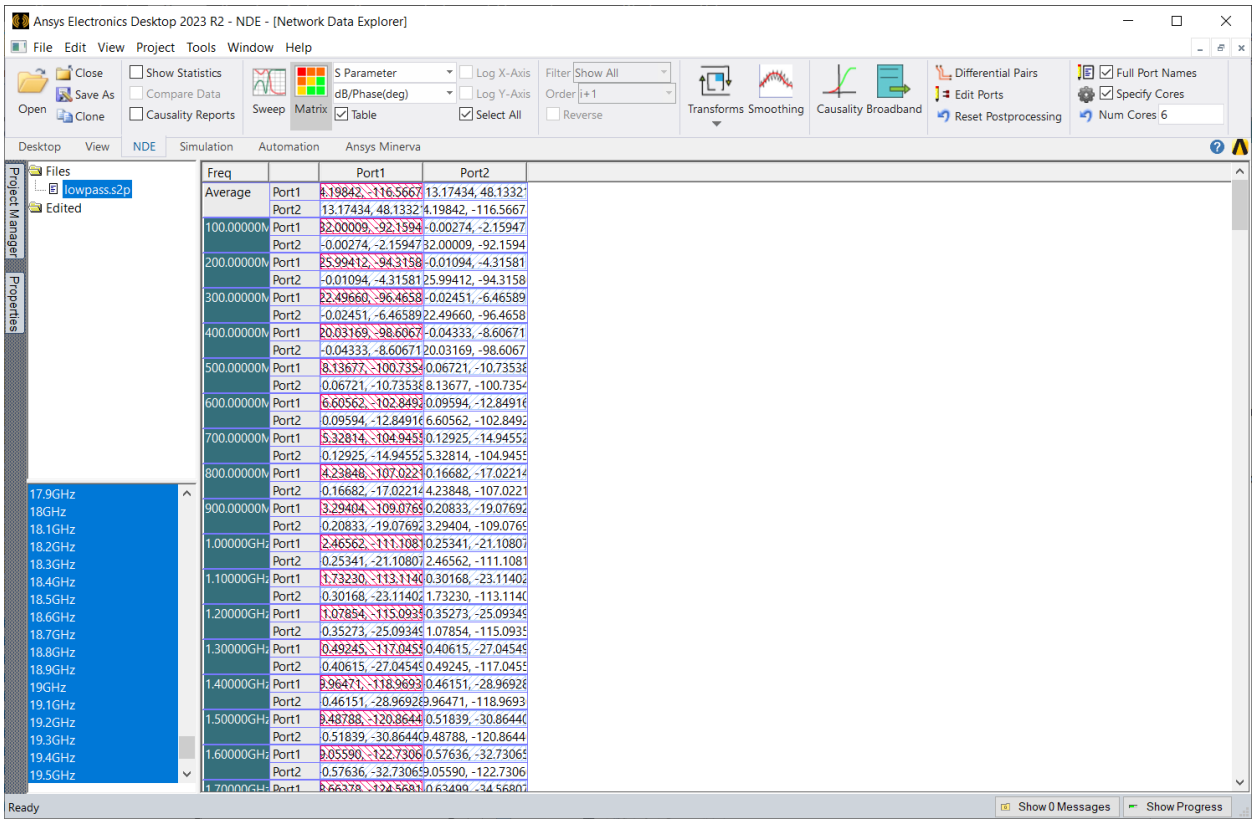
Network Data Explorer allows you to view data and modify various aspects of the display, including color plots, color coding, viewing across frequencies, and displaying individual statistics. This section provides examples of Network Data Explorer capabilities:

- Viewing the S, Y, or Z Matrix for a Frequency
- Viewing a Color-coded Matrix Plot
- Displaying a Cell Graph Across All Frequencies
- Displaying Matrix Statistics by Frequency
- Displaying Individual Statistics for All Frequencies
- Creating a Statistics Plot
- Comparing Variations

Viewing a Matrix Table

To view the S, Y or Z matrix:

1. On the **NDE** ribbon, click **Matrix**.
2. Click the **Table** check box.
3. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
4. Choose a format from the drop-down menu (e.g., dB or Mag).
5. From the Cell and Frequency Selection pane, select frequencies to display or click the **Select All** check box on the **NDE** ribbon to select all frequencies. The S, Y, or Z matrix displays.



6. Hover over a cell to see more information in a tool tip.
7. Right click to see a right click menu with other commands.

Maximum values are highlighted in red stripes. If Select Transpose is enabled, transposes are highlighted in red stripes as well.

Minimum values are highlighted in blue stripes.

Selected cells appear in solid blue. If Select Transpose is enabled, the transpose is highlighted in blue as well.

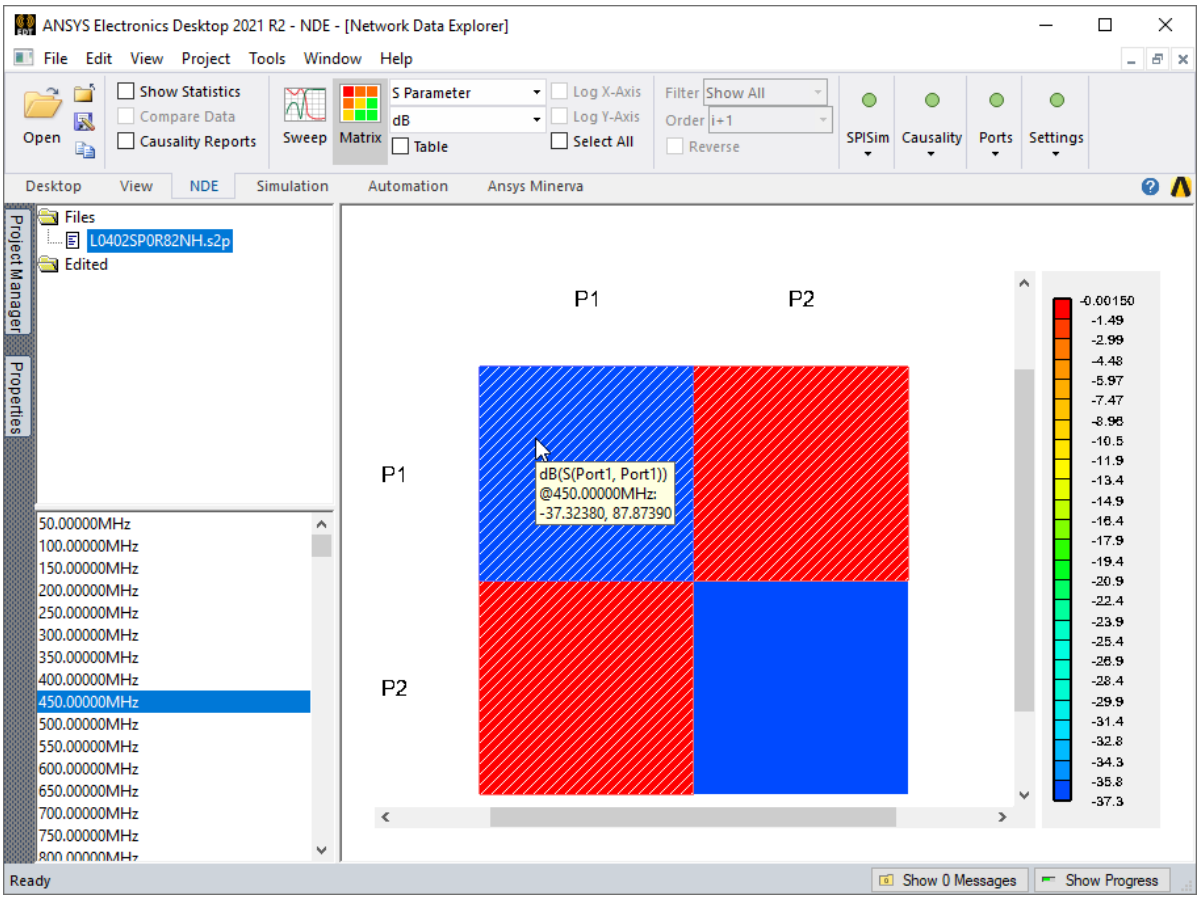
Double-clicking a cell switches to a matrix cell view, in which values for all frequencies for that cell are displayed. The double-clicked frequency is highlighted with solid red shading

Complex values are compared using their modulus. When multiple frequencies are selected, the data display depends on the Multiple Frequency Statistics setting.

Viewing a Color-coded Matrix Plot

1. On the **NDE** ribbon, click **Matrix**.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.

3. From the Cell and Frequency Selection pane, select the frequencies you want to plot or click the **Select All** check box on the **NDE** ribbon to select all frequencies.
4. Hover over a cell to see more information in a tool tip.
5. Right click to see a right click menu with other commands.



Matrix values display in a color-coded grid. If the selected **Format** is a complex value, only the real component is used to determine the display color. When multiple frequencies or variations are selected, the data display depends on the Multiple Frequency Statistics setting. Maximum values are highlighted in red. Minimum values are highlighted in dark blue.

Hover the cursor over any cell to view information about it.

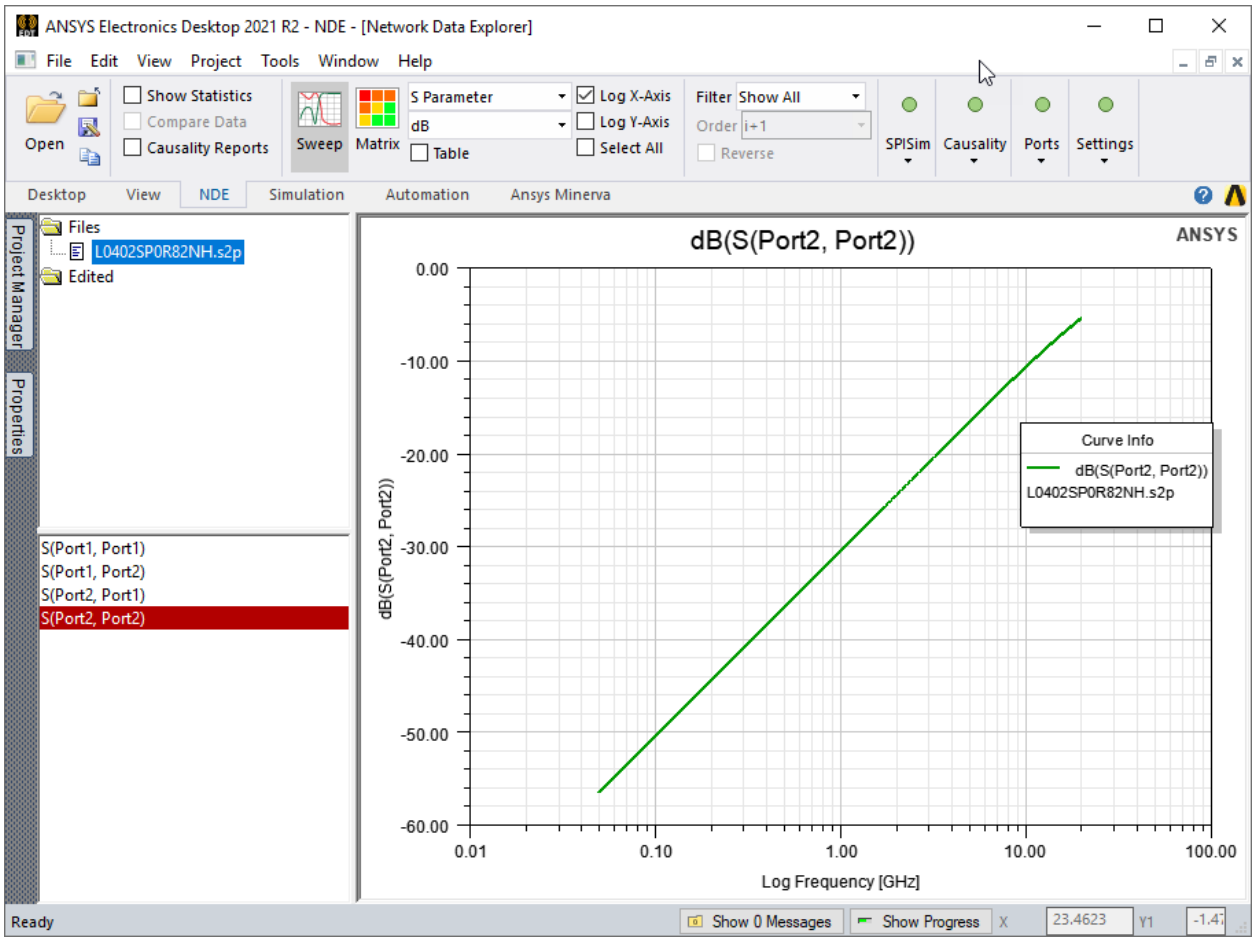
Click any cell to select it. Selected cells appear with a pink outline. If Select Transpose is enabled, the transpose is selected in pink as well.

Double-click any cell to view a matrix cell plot in which all frequency values for that matrix cell are displayed as a graph.

Displaying a Cell Graph Across All Frequencies

Network Data Explorer can plot a cell across all frequencies.

1. On the **NDE** ribbon, click **Sweep**. The Data View Pane updates if appropriate.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, or **Z parameter**.
3. From the Cell and Frequency Selection pane, select cells to display or click the **Select All** check box on the **NDE** ribbon to select all cells.
4. To add a log scale to the X-axis, click **Log X-Axis** on the **NDE** ribbon.



5. To add a log scale to the Y-axis, click **Log Y-Axis** on the **NDE** ribbon.
6. Right click to see a right click menu with other commands.

Displaying Matrix Statistics by Frequency

Network Data Explorer can display various statistical measurements.

1. On the **NDE** ribbon:
   a. Click **Matrix**.
   b. Click **Table**.

c. Click **Show Statistics**.

2. From the Cell and Frequency Selection pane, select frequencies to display.

Freq		Average	Minim...	Maxim...	StdDev	NTI	Passivity
50.00000MHz		-28.20530	-56.41050	-0.00010	28.20520	2	1.00000
100.00000MHz		-25.19490	-50.38960	-0.00020	25.19470	2	1.00000
150.00000MHz		-23.43390	-46.86750	-0.00030	23.43360	2	1.00000
200.00000MHz		-22.18450	-44.36850	-0.00050	22.18400	2	1.00000
250.00000MHz		-21.21535	-42.43000	-0.00070	21.21465	2	0.99999
300.00000MHz		-20.42350	-40.84620	-0.00080	20.42270	2	1.00000
350.00000MHz		-19.75405	-39.50700	-0.00110	19.75295	2	0.99999
400.00000MHz		-19.17415	-38.34700	-0.00130	19.17285	2	1.00000
450.00000MHz		-18.66265	-37.32380	-0.00150	18.66115	2	1.00001

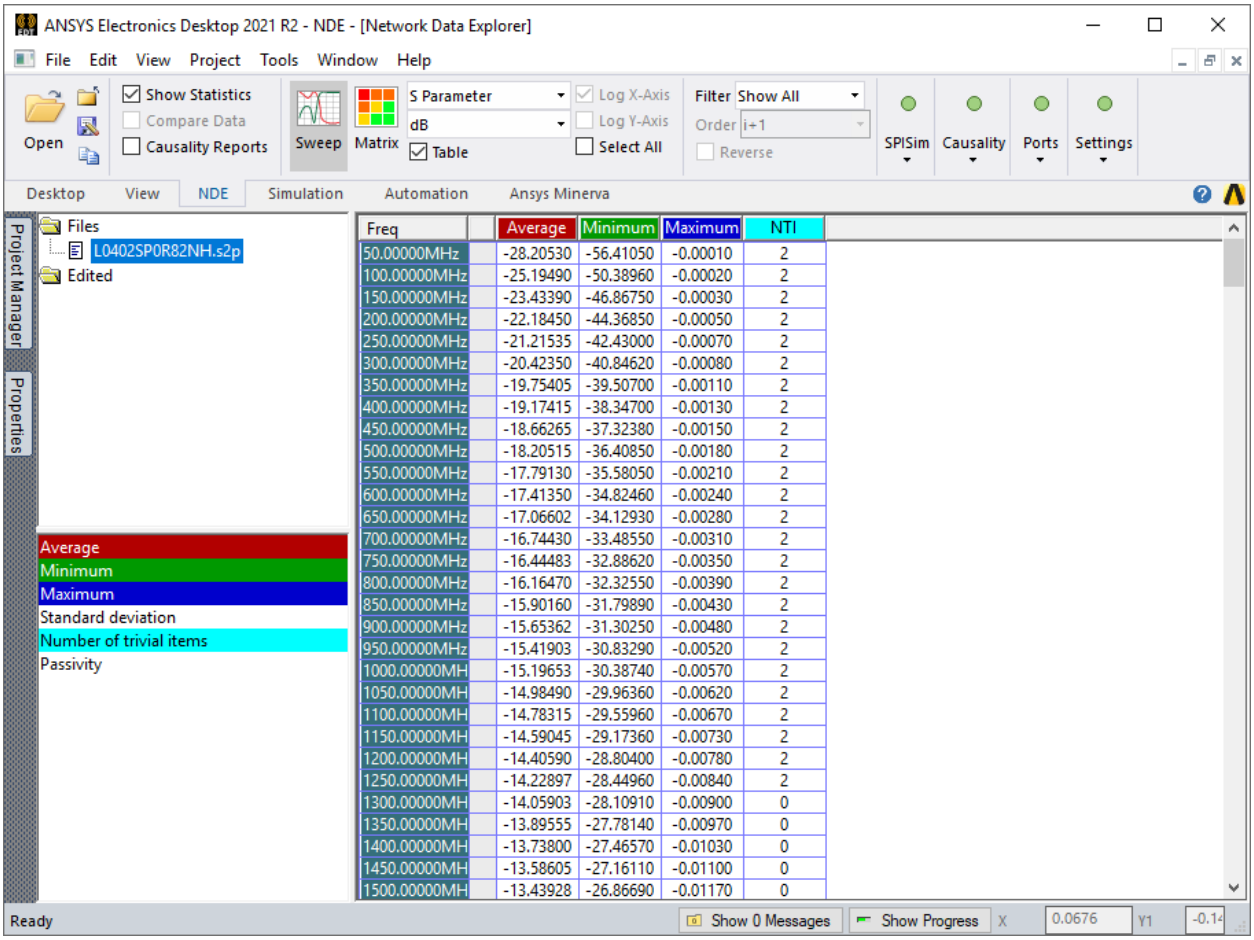
Click a column header to sort data by that column.

Hover the cursor over a cell to view information about it.

Only real (not complex) data formats are offered for statistical analysis. **Passivity** is only available for S-parameter data (comparisons inactive). **NTI** refers to the number of trivial items; for S-parameters, this includes all zeros and ones; for all other data (and data comparisons), only zeros are counted as trivial. The minimum value for each column is highlighted in blue; the maximum is highlighted in red.

Displaying Individual Statistics for All Frequencies

1. On the **NDE** ribbon:
   a. Click **Sweep**.
   b. Click **Table**.
   c. Click **Show Statistics**.
2. From the Cell and Frequency Selection pane, select statistics to display. The information displays in a table in the Data View pane.



Selected statistics are displayed for all frequencies.

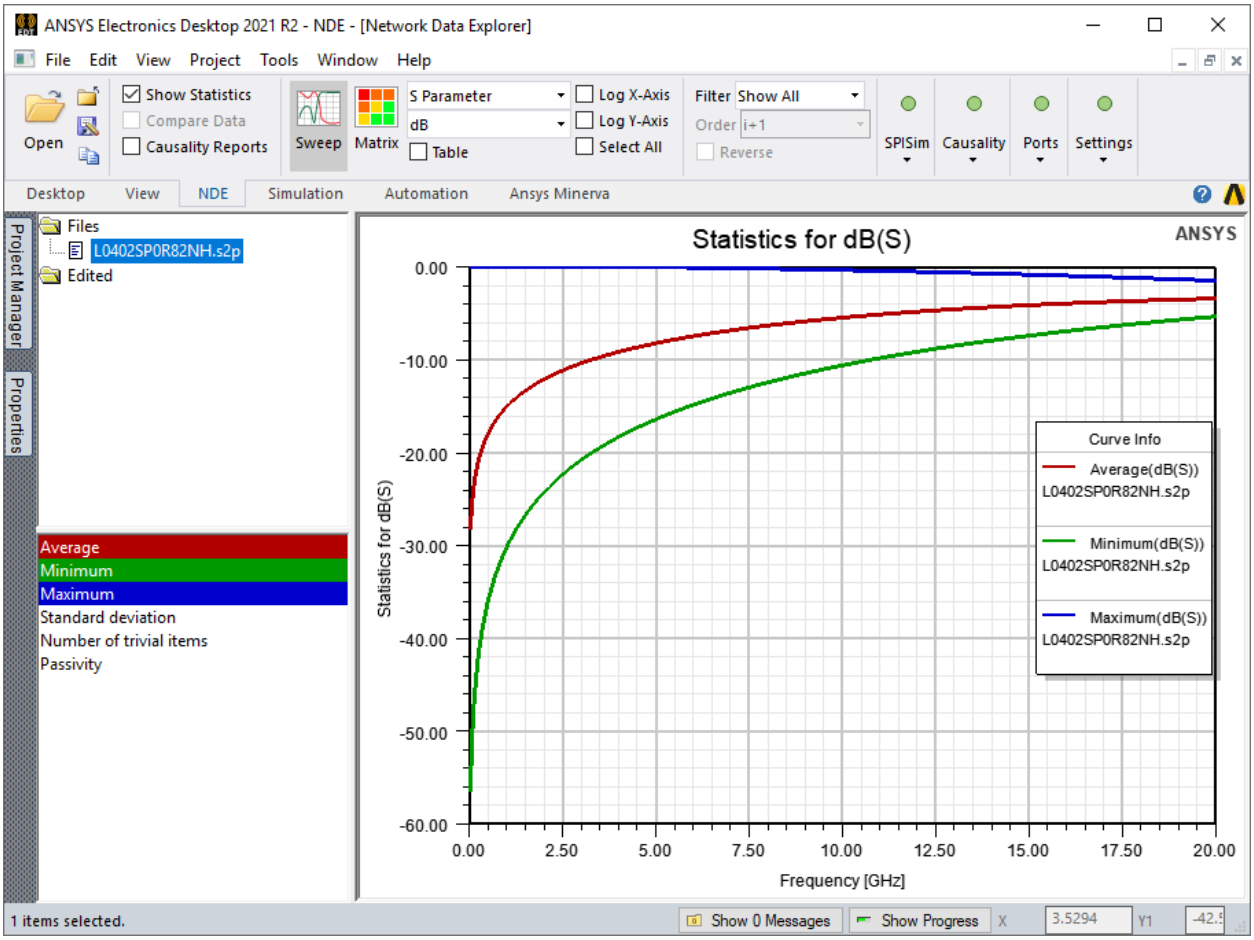
Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

Creating a Statistics Plot

Network Data Explorer can display a graph of selected statistical measures across all frequencies.

1. On the **NDE** ribbon:
   a. Click **Sweep**.
   b. Click **Show Statistics**.
2. From the Cell and Frequency Selection pane, select statistics to display. The selected statistics are plotted.



Hover the cursor over a statistic to view more information about it.

Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

Comparing Network Data

Network Data Explorer can compare variations for two network data sets that are the same size.

1. On the **NDE** ribbon, click **Sweep**.
2. From the Network Data Selection pane, select exactly two data sets.
3. On the **NDE** ribbon, click **Compare Data**.
4. From the Cell and Frequency Selection pane, select which cells you want to compare . For each value along the X-axis, the Y-axis values are subtracted, one from the other, to create the comparison plot. The second selected data is subtracted from the first selected data.
5. Optionally, check **Show Statistics** to show values applied to all cells and all frequencies

It is not possible to compare a data set against itself *unless the data set has been cloned.* Then, you can compare the original data set to the clone.

Traces for a given cell or statistical measure are displayed for all data sets; you can use tool tips to distinguish between them.

If a single cell or statistical measure is displayed, different colors are used for each data trace. If multiple cells or statistical measures are selected, a single color is used for all data traces for each cell or statistical measure.

In a data comparison, traces are shown for all selected data sets. This is true for both cell and statistical traces.

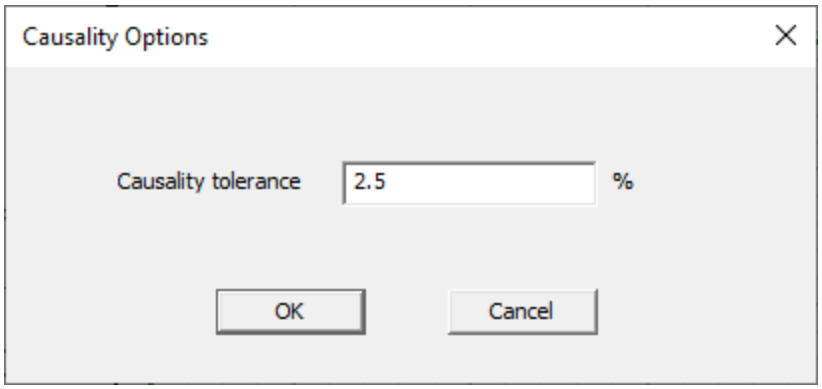
If **dB** format is shown, you can either subtract values before or after applying the dB function. See: Display Format to make this choice.

Causality Checking and Plots

Network Data Explorer can perform a causality check on S-parameter data from any source (solution or file), and provide plots of the results in various formats.

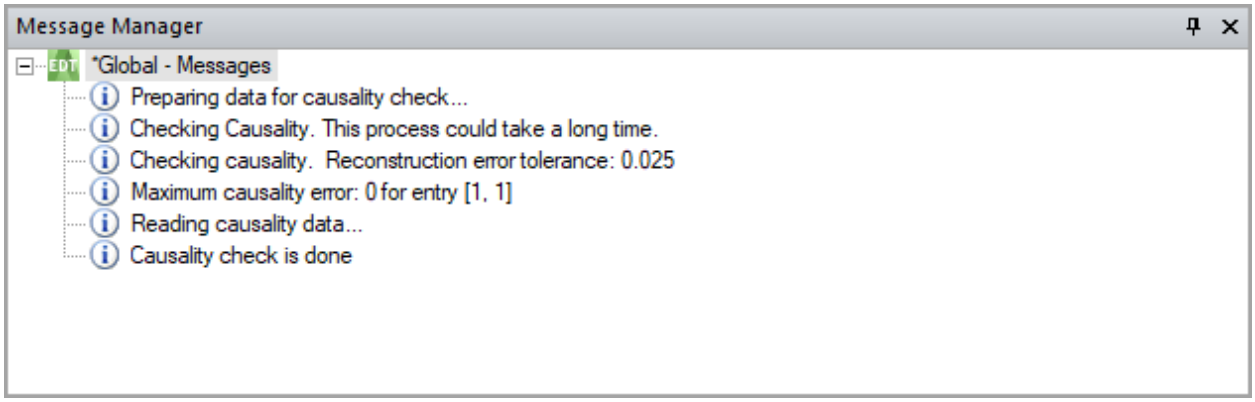
When S-parameter data is loaded into Network Data Explorer, the **Causality** button is enabled.

1. On the **NDE** ribbon, click **Causality**. The **Causality Options** dialog box opens.



2. Enter a **Causality tolerance** and click **OK** to start the causality check. Depending on the size of the S-parameter data, the causality check may take several minutes to complete.

The check's status displays in the Message Manager.

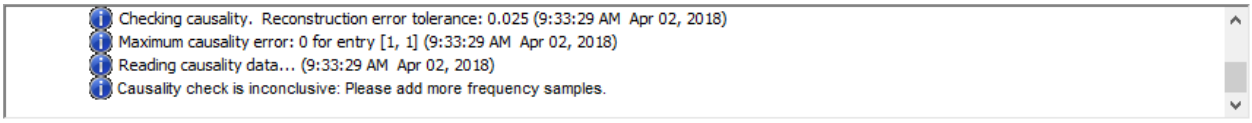


When the check completes, the Message Manager updates to display a summary of results.

Reconstruction Error Tolerance – Causality of a frequency response is determined by calculating the generalized Hilbert transform of the data at all frequencies. A causal frequency response is equal to its generalized Hilbert transform. The reconstruction error is the difference between the tabulated data and its transform at a given frequency. The message shows the maximum reconstruction error tolerance for a causal frequency response. The default tolerance of 0.01 is equal to the state-space fitting tolerance.

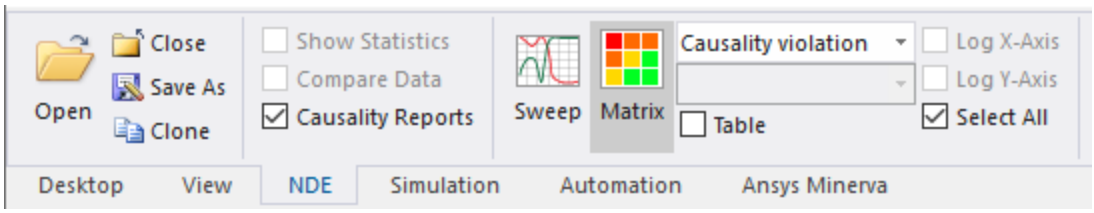
Maximum Causality Error – The maximum causality error for all port pairs and all frequencies, along with the matrix indices (port numbers) where the maximum noncausality occurs. A noncausal response is one where all matrix entries can be conclusively analyzed, and at least one entry exceeds the causality tolerance. The maximum reconstruction appears first, followed by port numbers in brackets (e.g., [port number, port number]). When all results are conclusive but no matrix reconstruction error exceeds the tolerance, the maximum causality error is reported as zero, and no matrix entry is listed.

If the data does not contain enough frequency points to determine whether the data is or is not causal, the Messages Pane will note an inconclusive result. Network Data Explorer will also report the data set as inconclusive if any cells are inconclusive, even if other entries exhibit causality violations.

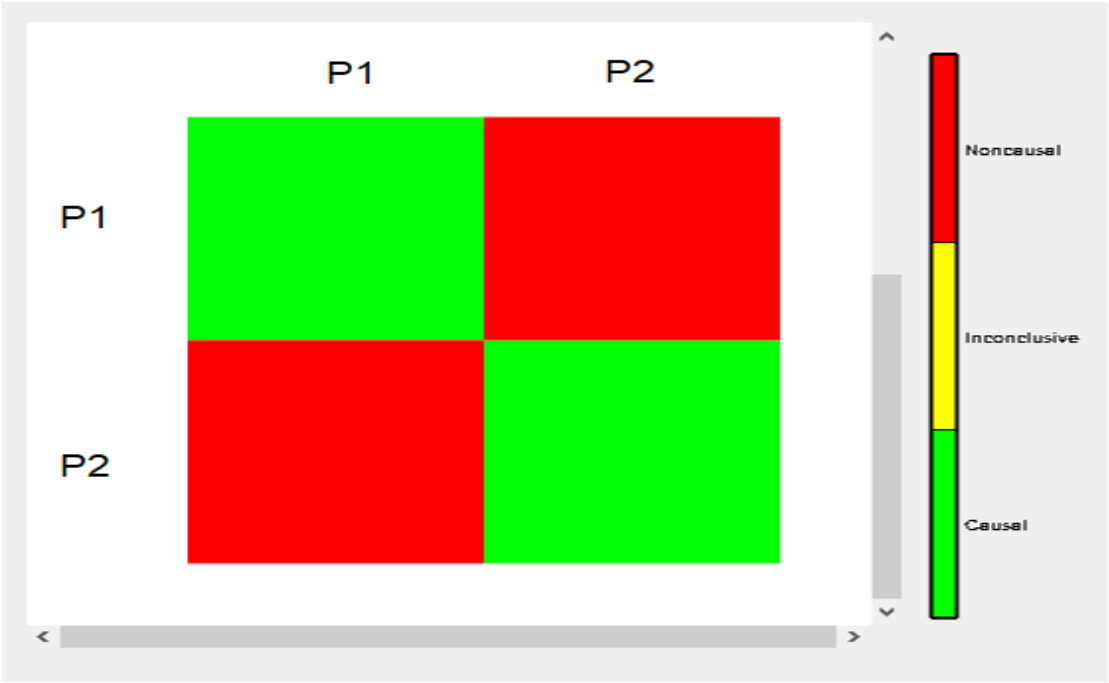


To see the plotted results of the causality check:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Matrix** icon or select **Causality violation** in the **Parameter** type drop-down menu.



3. A rectangular plot displays, with dimension N x N and color-coding to indicate the causality status of each port pairing.



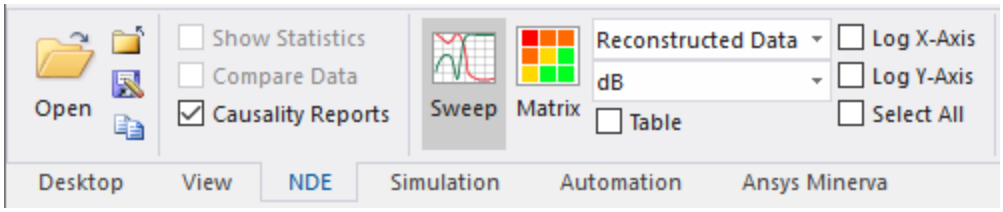
In this plot, the cells go from (Port 1, Port 1) at the upper-left area to (Port N, Port N) at the lower-right corner. The result shows the causality over all frequencies in the data. In this example, the matrix is asymmetric, so that S12 is noncausal, while S11, S21, and S22 are causal.

To see the details for each frequency, click the **Table** check button on the **NDE** ribbon.

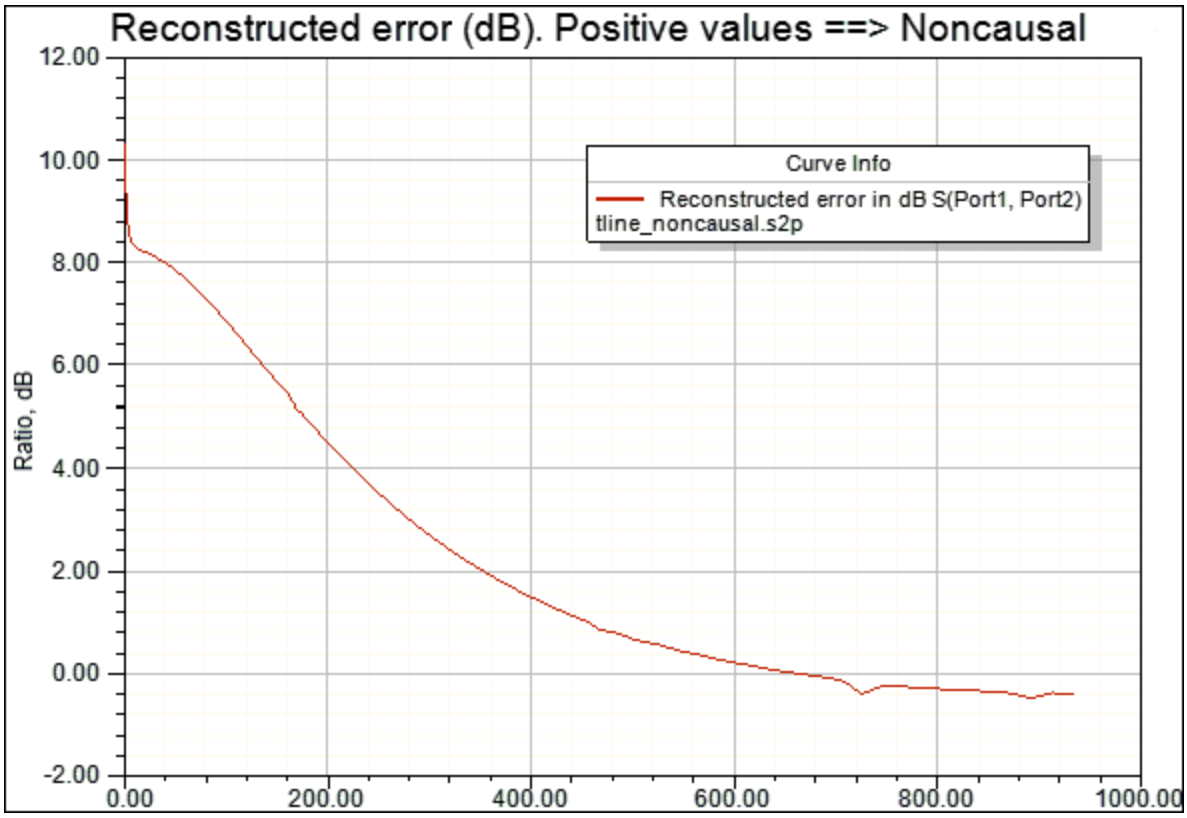
Freq		U41_VCC	U41_VCC
0.00000Hz	U41_VCC	0.00003, 0.00000	0.00018, 0.00000
0.00503GHz	U41_VCC	0.00076, 0.00000	0.00041, 0.00000
0.01005GHz	U41_VCC	0.00048, 0.00000	0.00043, 0.00000
0.01508GHz	U41_VCC	0.00044, 0.00000	0.00035, 0.00000
0.02010GHz	U41_VCC	0.00037, 0.00000	0.00035, 0.00000
0.02513GHz	U41_VCC	0.00032, 0.00000	0.00032, 0.00000
0.03015GHz	U41_VCC	0.00027, 0.00000	0.00030, 0.00000
0.03518GHz	U41_VCC	0.00023, 0.00000	0.00027, 0.00000
0.04020GHz	U41_VCC	0.00019, 0.00000	0.00024, 0.00000
0.04523GHz	U41_VCC	0.00015, 0.00000	0.00020, 0.00000
0.05025GHz	U41_VCC	0.00012, 0.00000	0.00017, 0.00000
0.05528GHz	U41_VCC	0.00008, 0.00000	0.00013, 0.00000
0.06030GHz	U41_VCC	0.00006, 0.00000	0.00009, 0.00000
0.06533GHz	U41_VCC	0.00002, 0.00000	0.00005, 0.00000
0.07035GHz	U41_VCC	0.00000, 0.00000	0.00001, 0.00000
0.07538GHz	U41_VCC	0.00003, 0.00000	0.00003, 0.00000
0.08040GHz	U41_VCC	0.00005, 0.00000	0.00007, 0.00000
0.08543GHz	U41_VCC	0.00007, 0.00000	0.00011, 0.00000
0.09045GHz	U41_VCC	0.00008, 0.00000	0.00015, 0.00000
0.09548GHz	U41_VCC	0.00010, 0.00000	0.00018, 0.00000
0.10050GHz	U41_VCC	0.00011, 0.00000	0.00022, 0.00000
0.10553GHz	U41_VCC	0.00012, 0.00000	0.00025, 0.00000
0.11055GHz	U41_VCC	0.00013, 0.00000	0.00028, 0.00000
0.11558GHz	U41_VCC	0.00014, 0.00000	0.00031, 0.00000
0.12060GHz	U41_VCC	0.00015, 0.00000	0.00034, 0.00000
0.12563GHz	U41_VCC	0.00015, 0.00000	0.00036, 0.00000
0.13065GHz	U41_VCC	0.00015, 0.00000	0.00037, 0.00000

To plot the reconstructed frequency response generated by the causality checker:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Sweep** icon or select **Reconstructed Data** in the **Parameter** type drop-down menu.
3. From the **Format** drop-down menu, select an appropriate format. **dB** is selected by default.

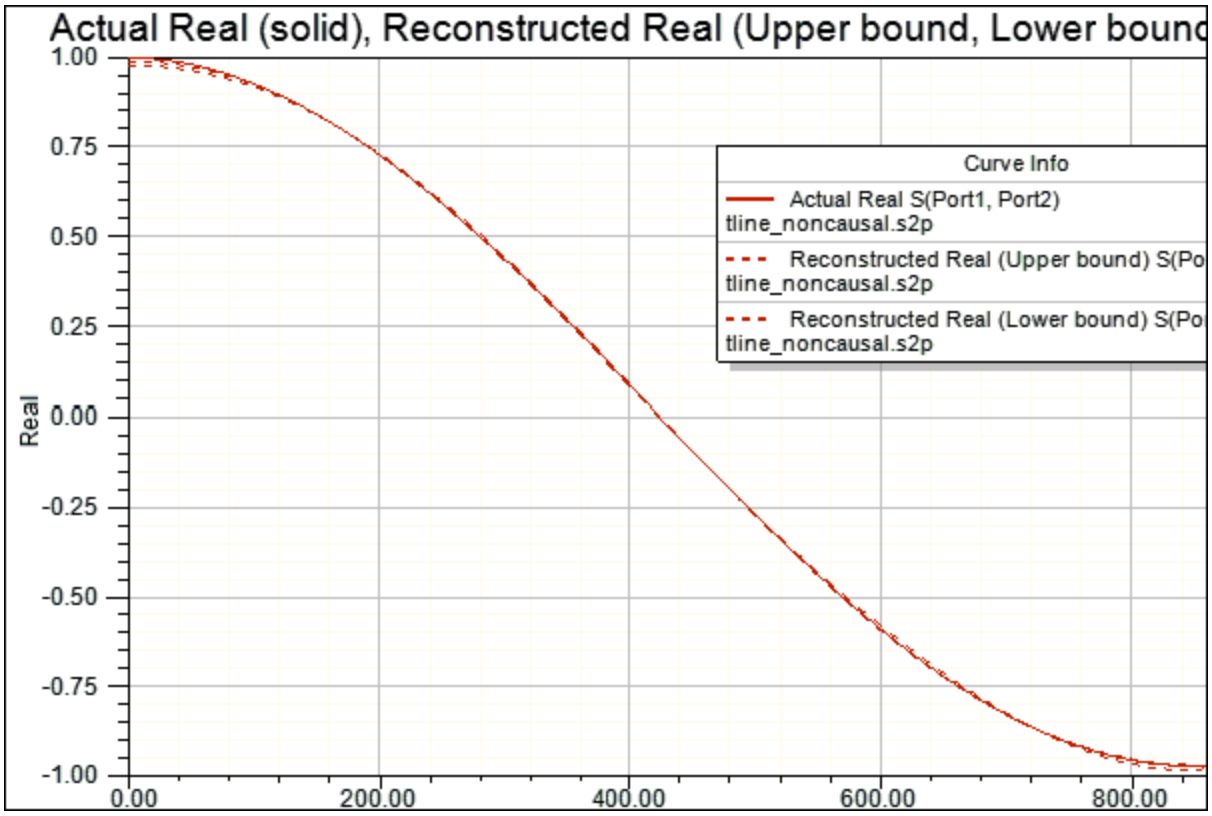


The plot appears, showing the reconstruction error at each frequency divided by the tolerance.



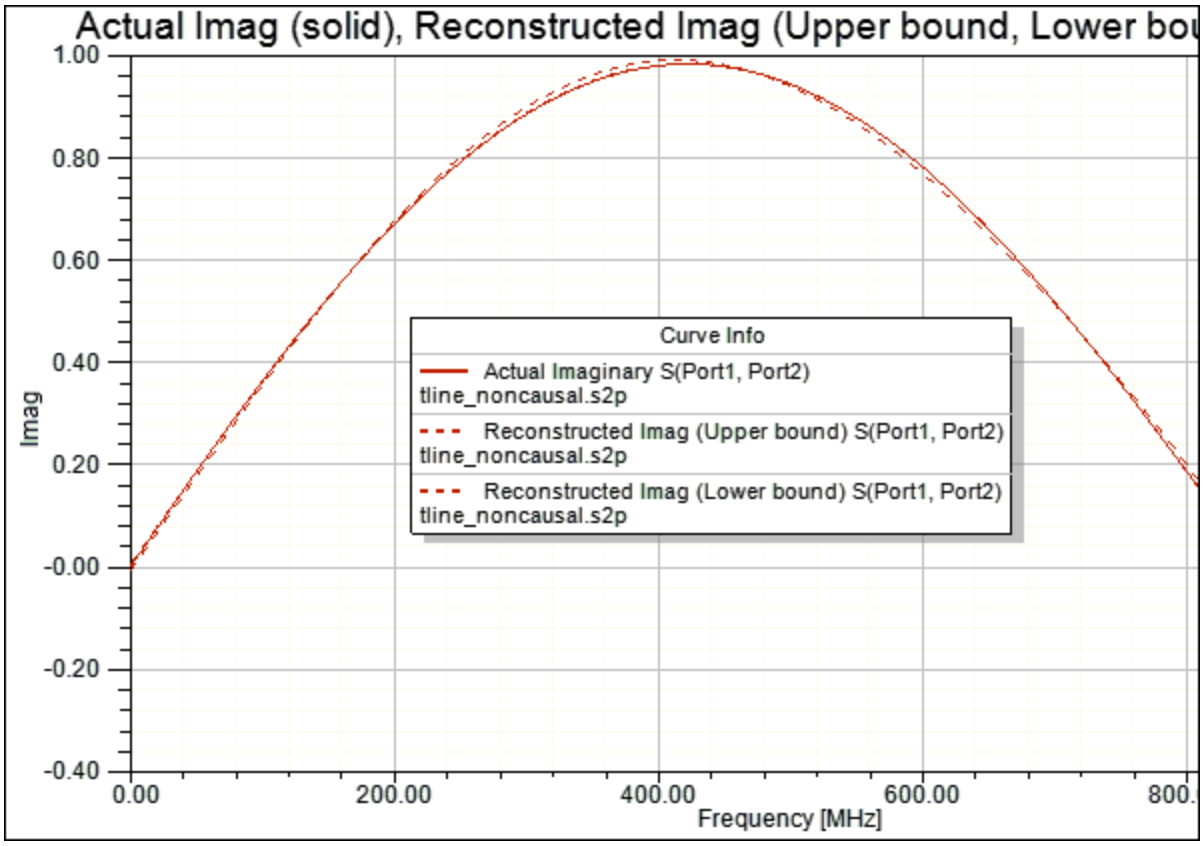
The reconstruction error ratio for parameter S12 is positive for frequencies less than about 680MHz, indicating a broad range of noncausal behavior.

4. To compare the real part of the reconstructed data to the real part of the actual data, set the **Format** to **Real**.



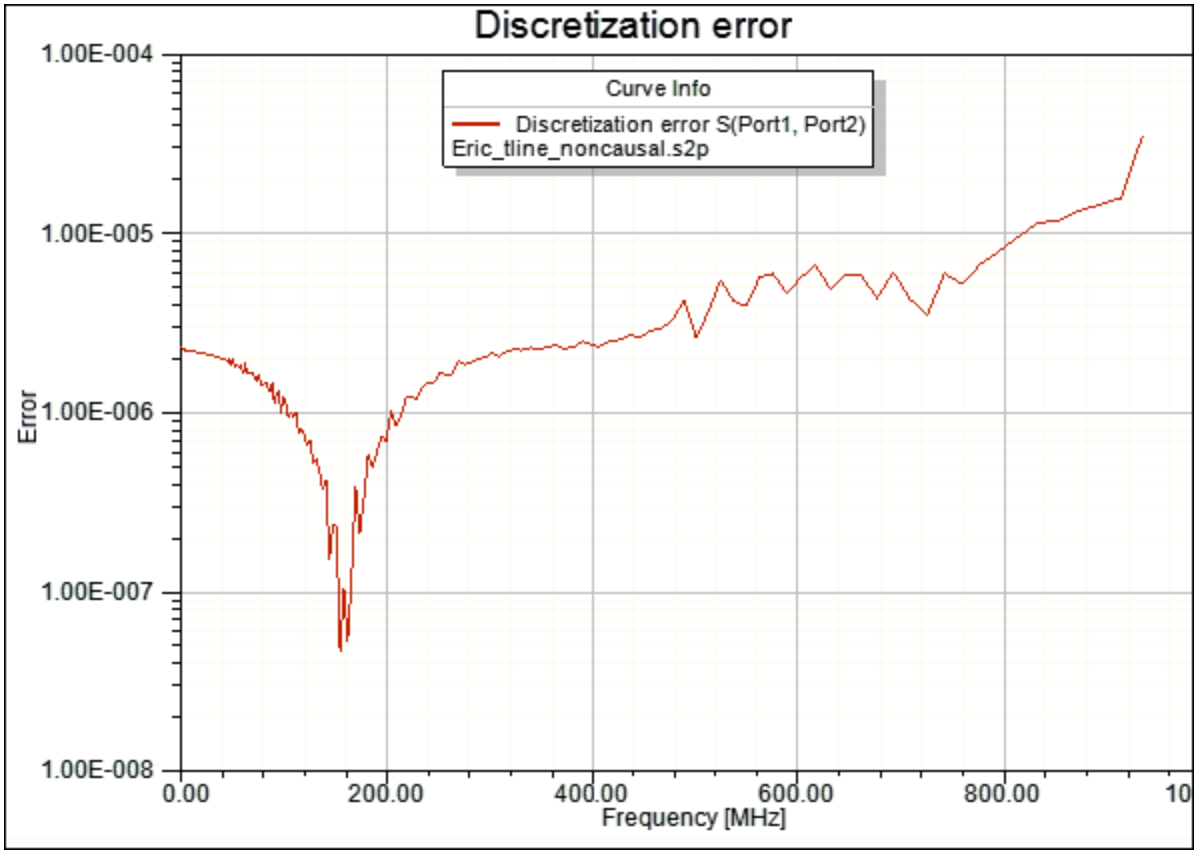
For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

5. To compare the imaginary part of the reconstructed data to the imaginary part of the actual data, set the **Format** to **Imaginary**.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

6. To view the frequency-dependent discretization error, set the **Format** to **Discretization**.



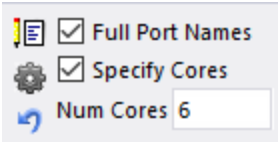
The discretization error is the error that is due to the fact that the data are available only at discrete frequencies rather than for a continuous spectrum. A discretization error near or greater than the causality tolerance renders the causality check inconclusive. Data at more frequencies could reduce the discretization error and render the analysis conclusive. This set of data exhibits low discretization errors (<<0.01) at all frequencies, and the causality check is conclusive (conclusively noncausal in this example).

Multithreading

By default, multithreading (execution on multiple cores) is enabled for Causality Checking and Macro Model export. Multithreading saves significant time in the Causality Check calculation, and improves the time for other state-space fitting operations. See Technical Notes.

The **Cores** field in the Network Data Explorer Control Pane defaults to half the number of cores detected on your computer.

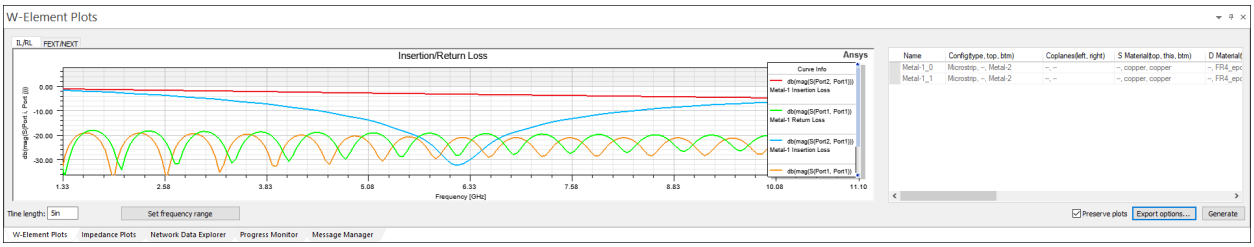
For best performance, disable hyper-threading on your computer. When hyper-threading is enabled, the number of cores includes the physical cores and an equal number of logical cores. With hyper-threading disabled, the display shows only half the number of physical cores.



To disable multithreading, uncheck the check box.

8 - Viewing and Exporting Plots in SI Xplorer

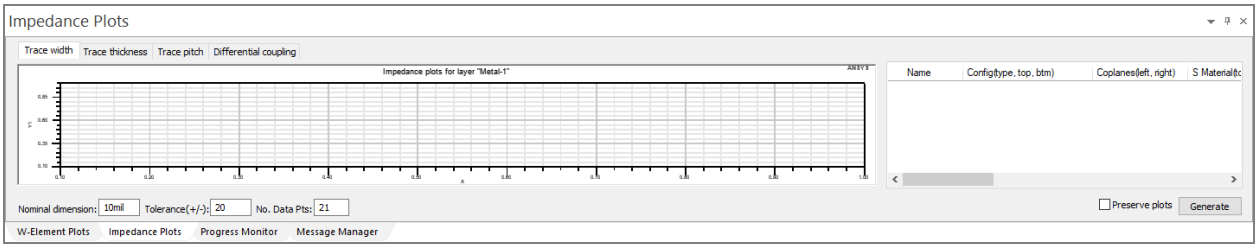
After you use the Impedance Wizard, results appear in the **Plots and Messages** pane:



Viewing and Exporting Impedance Plots

Prerequisites:

- Use the Impedance Wizard to calculate trace width/pitch, impedance, or differential impedance.



Impedance plots update as you change inputs in the **Impedance Wizard** pane and as you change any settings underneath the plot. The table to the right summarizes the configuration. To display more than one dataset, enable **Preserve Plots** to keep any plotted data displayed even after you make a new calculation.

> **Important:**
>
> To see dynamic plot changes, **Enable Dynamic Update** must be enabled in the **Impedance Wizard** pane.

For more details about these plots, see: Calculating Impedance, Calculating Differential Impedance, and Calculating Trace Width/Pitch.

To export an impedance plot from SI Xplorer:

1. Use the tabs at the top to select a plot for display.
2. Click **Generate**.

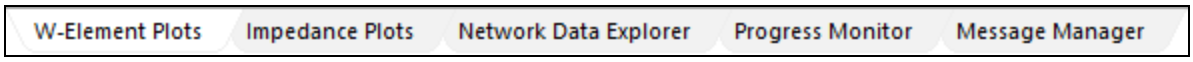
Viewing and Exporting Insertion/Return Loss and Time Domain Crosstalk Plots

Prerequisites:

- Use the Impedance Wizard to calculate trace width/pitch, impedance, or differential impedance.

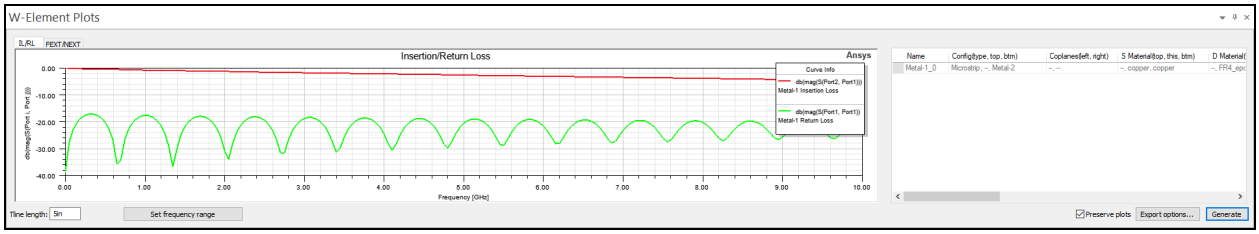
To calculate and plot IL/RL/FEXT/NEXT:

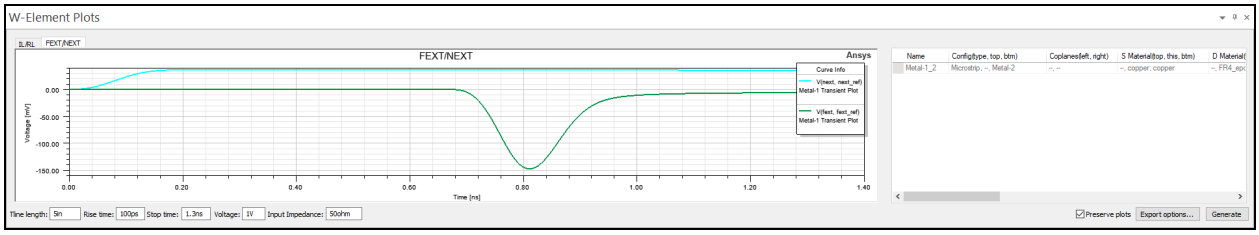
1. From the **Plots** window, select the **W-Element Plots** tab:



2. Use the **IL/RL** and **FEXT/NEXT** tabs to choose a plot:

The selected plot displays:





3. Adjust the settings as appropriate:

   For IL/RL plots, you can adjust the **Tline Length** or click **Set frequency range** to change the frequency range of the plot.

   For FEXT/NEXT plots, you can adjust the **Tline length** (trace length); edit the **Rise time**, **Stop time**, and peak **Voltage** of the driver attached to the aggressor net; or modify the

driver **Input Impedance** value. This impedance is also used to terminate all transmission lines in the specified cross-section.

Selecting the **Preserve Plot** check box allows you to display previous plots while computing additional plots.

If you wish to export plot data, click **Export Options**:

- For IL/RL plots, you can choose **Export w-element schematic to circuit** (in *.aedt format) or **Export touchstone file**.
- For FEXT/NEXT plots, you can choose **Export w-element schematic to circuit** (in *.aedt format) or **Export .csv file**.

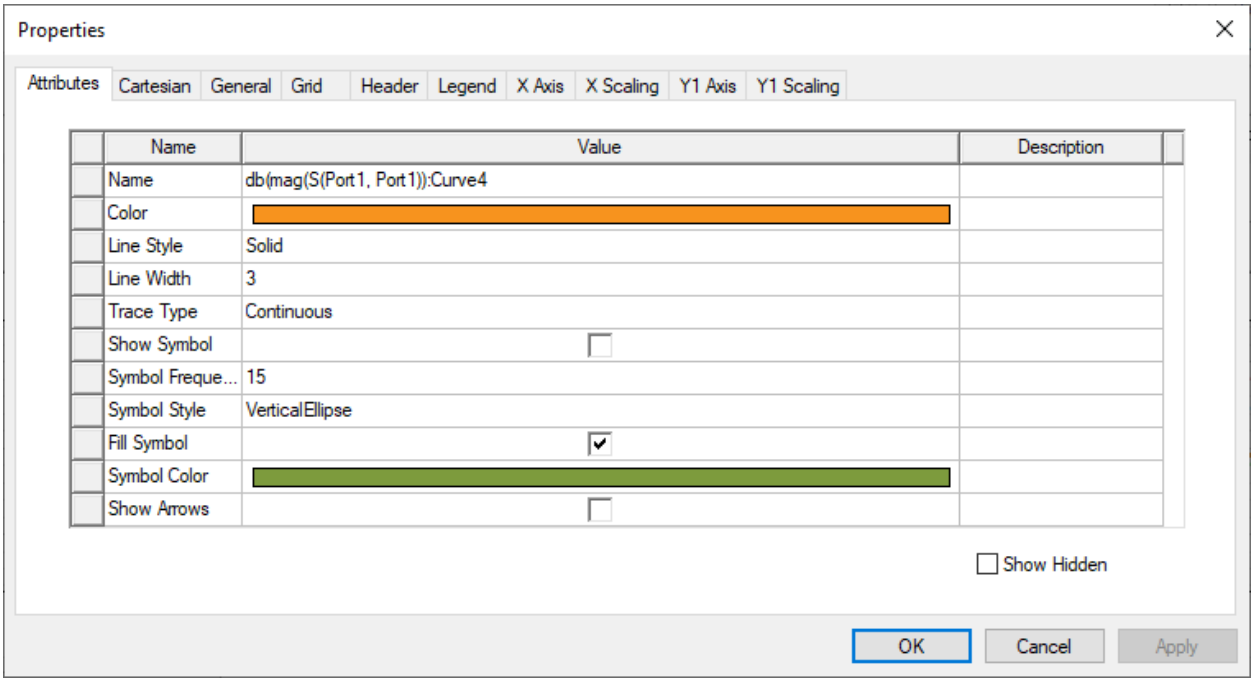
If you change values and need to regenerate the plot, click **Recompute**.

Right-clicking within the plot enables additional plotting actions.

Changing Plot Display Options

You can change the way SI Xplorer displays plots.

Double-clicking within the plot enables opens the **Properties** window on the relevant tab.



Tabs include:

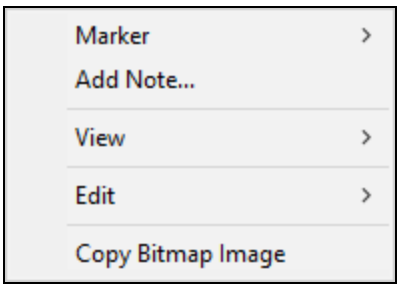
- **Attributes** – display options for the selected plot element. This tab only displays when a plot element is selected.
- **Cartesian** – display options for the coordinate system.

- **General** – display options for the plot's background colors and number formatting.
- **Grid** – display options for grid visibility and coloring.
- **Header** – display options for the header's font and formatting.
- **Legend** – display options for the legend's background colors, grid, and formatting.
- **X Axis** – display options for the X axis.
- **X Scaling** – X axis display options for minimum value, maximum value, and scaling.
- **Y Axis** – display options for the Y axis.
- **Y Scaling** – Y axis display options for minimum value, maximum value, and scaling.

Each tab allows you to specify display units, colors, fonts, and scaling for the relevant plot element. Select **Show Hidden** on any tab to view additional options for that tab.

Additional Plotting Actions

Right-clicking a plot in SI Xplorer enables options for adding markers and notes, changing the display, and editing settings.



Menu options include:

- **Marker** – Submenu containing marker options.
  - **Add Marker** – transforms the cursor into a crosshair, allowing you to click and place a marker at an appropriate point. When you are finished placing markers, right-click and select **End Marker Mode**. Markers appear on the plot and in a table showing their coordinates.
  - **Add Delta Marker** – transforms the cursor into a crosshair, allowing you to click and place a marker at an appropriate point. You must then click a second point, and a table appears showing deltas and slopes. When you are finished placing markers, right-click and select **End Marker Mode**.
  - **Export Marker Table** – exports the marker coordinate table in *.csv or *.tab format.
  - **Export Delta Marker Table** – exports the delta marker table in *.csv or *.tab format.
  - **Add Maximum** – places a marker at the maximum value along the X-axis.
  - **Add Minimum** – places a marker at the minimum value along the X-axis.

- **Add X Marker** – adds up to 10 movable markers as vertical lines rising from the X axis. Each added marker has its own color and editable properties. To move an X marker, click on the X label and drag it to an appropriate location. The label at the bottom of the line gives the X coordinate, and flag on the vertical line identifies the Y coordinate.
- **Bring X Marker Into View** – when one or more X markers is not visible, this option allows you to bring them into view.
- **Add Y Marker** – adds up to 10 movable markers as horizontal lines extending from the Y axis. Each added marker has its own color and editable properties. To move a Y marker, click on the Y label and drag it to an appropriate location. The label at the bottom of the line gives the Y coordinate, and flag on the vertical line identifies the X coordinate.
- **Bring Y Marker Into View** – when one or more Y markers is not visible, this option allows you to bring them into view.
- **Next Peak** – selects the next high point along the X-axis, before the next dip.
- **Next Minimum** – selects the next lowest point along the X-axis, after the next high point.
- **Previous Peak** – selects previous next high point along the X-axis, before the previous dip.
- **Previous Minimum** – selects the previous lowest point along the X-axis, after the previous high point.
- **Go to Start** – selects the first marker on the plot, along the X-axis.
- **Go to Previous** – selects the previous marker on the plot, along the X-axis.
- **Go to End** – selects the last marker on the plot, along the X-axis.
- **Next Curve** – selects the next curve on the plot, based on the order in the legend.
- **Previous Curve** – selects the previous curve on the plot, based on the order in the legend.
- **Clear All** – removes all markers from the plot.

- **Add Note** – Allows you to add a note to the plotting area. You can click and drag the note to your appropriate location.

  Select the note and press the **Delete** key to remove it.

- **View** – Submenu containing view options.
  - **Pan** – Changes the cursor to move the plot within the window. Right-click and deselect **Pan** to exit this mode.
  - **Zoom** – Changes the cursor to zoom the entire plot in or out within the window. Right-click and deselect **Zoom** to exit this mode.
  - **Zoom In / Zoom Out** – Changes the cursor to a crosshair. Click a point in the plot to zoom in to it or out from it. Right-click and deselect **Zoom In** or **Zoom Out** again to exit this mode.
  - **Fit All / Fit X Window / Fit Selection / Best Vertical Fit** – Automatically scales the plot to an appropriate size.

  - **Show / Hide Selection / Show All** – When a dataset is selected on the plot, **Show / Hide Selection** allows you to hide or show that dataset. Select **Show All** to return to the original plot displaying all datasets.
  - **Visibility** – Opens a window where you can set the visibility of traces, notes, and legends.
  - **Animate** – This functionality is not yet present in the plot window.
- **Edit** – Submenu containing edit options.
  - **Copy Settings** – Copies the display settings in text format for use in scripts.
  - **Reset Settings** – Resets the plot window to its original view, including scale, colors, etc.
  - **Delete** – Allows you to delete a marker.
  - **Properties** – Opens the **Properties** window, where you can change display options.
- **Copy Bitmap Image** – Copies a bitmap image of the plot to the clipboard for pasting into a graphics editor or other compatible software.

9 - Technical Notes

This section contains technical notes on the following topics:

Huray Surface Roughness Model

Surface roughness can increase conductor power losses by more than two times the loss associated with smooth conductors. In SIwave, a modification of the standard real-valued Huray surface roughness model is implemented to accurately account for these losses in both the frequency and time domains.

This modification fixes a causality problem with the original Huray model by adding some frequency-dependent reactance to the roughness-enhanced surface impedance. The purpose is to provide a complex-valued causal analytic model that exactly matches the loss results of the original model but is also suitable for time domain computations. The original Huray model was limited to calculations in the frequency domain and did not account for the effects of roughness on the phase of the transmitted signal. These limitations have been eliminated. The enhanced causal model provides accurate broad-band modeling of the losses in very rough copper foils (typically used in printed circuit board manufacturing), as did the original model, but also accurately predicts increased phase delay due to the surface roughness. because of its suitability for time domain computations.

Essentially, the smooth surface impedance (Z_{smooth}) is increased by a complex “Huray” factor (H_c), a function of frequency, given by the following formula:

$$H_c\left(j\omega\right) = 1 + \frac{K}{1+\left(\frac{j2\omega}{\omega_0}\right)^{-1/2}}$$

where:

- ω is the frequency in radians/second
- ω_0 is a corner frequency:

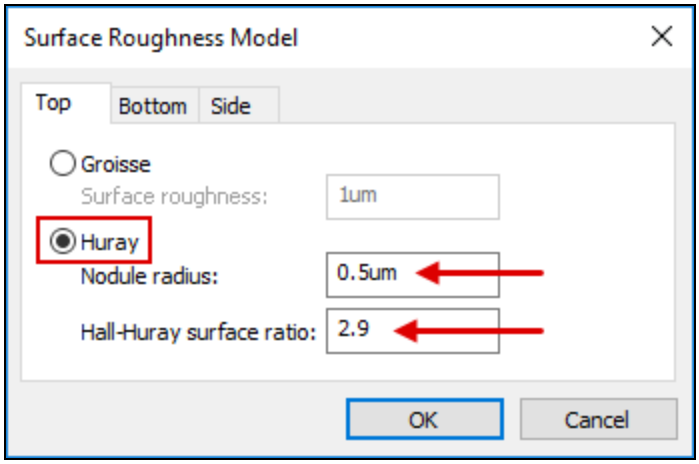
$$\omega_0 = \frac{2}{a^2\mu\sigma}$$

- K is a derived constant used to simplify the equation:

$$K = \frac{3}{2} * SR = \frac{6\pi a^2 N}{A_f}$$

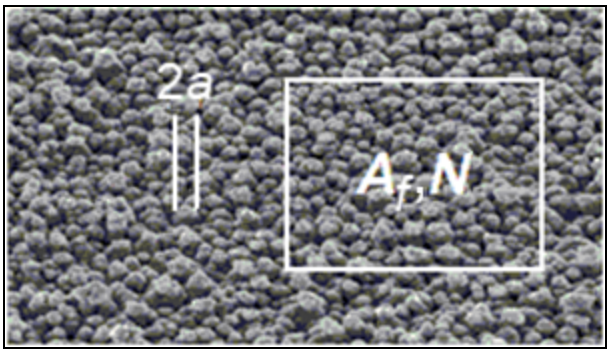
- a is the radius of a spherical nodule

> **Note:**
> a and SR are the two parameters you must enter in the *Surface Roughness Model* dialog box when you select the *Huray* option:



- μ is the permeability of free space = 4π * 1e-7 H/m
- σ is the electrical conductivity in Siemens/m
- N is the number of nodules
- A_f is a unit cell area

The following image of a magnified rough copper surface helps to clarify the a, A_f, and N parameters previously defined:



Note:

For more information about this complex causal Huray model – its necessity, background, derivation, proof of accuracy, and consistency with the original real-valued model – see the following article:

- J. Eric Bracken, ANSYS Inc., "*A Causal Huray Model for Surface Roughness*", DesignCon 2012, January 2012

This article is available from various online sources.

Once the Huray surface roughness factor is determined, the rough surface impedance (Z_{rough}) is determined as follows:

$$Z_{\text{rough}} = H_c * Z_{\text{smooth}}$$

References:

1. P.G Huray, S.G Pytel, S.H Hall, F. Oluwafemi, R.I. Mellitz, D. Hua, and P. Ye, "Fundamentals of a 3-D "Snowball" Model for Surface Roughness Power Losses", 11th Annual IEEE SPI Proceedings, May 13 – 16, 2007.
2. S.H Hall, S.G. Pytel, P.G Huray, D. Hua, A. Moonshiram, G. Brist, and E. Sijercic, "Multi-GHz, Causal Transmission Line Modeling Methodology with a Hemispherical Surface Roughness Approach", IEEE Transactions on Microwave Theory and Techniques, December 2007 pp 2614 – 2624.
3. S.G Pytel, P.G Huray, A. Moonshiram, S.H Hall, R.I Mellitz, G. Brist, F. Oluwafemi, H.M Meyer, L. Walker, and M. Garland, "Analysis of Copper Treatments and the Effects on Signal Propagation", 58th Annual IEEE ECTC, May 26 – 30, 2008, pp 1144 – 1149.
4. S.G Pytel, "Multi-gigabit data signaling rates for PWBs including dielectric losses and effects of surface roughness", PhD. Dissertation, University of South Carolina, 2007.
5. P.G Huray, O. Oluwafemi, J. Loyer, E. Bogatin, and X. Ye; "Impact of Copper Surface Texture on Loss: A Model That Works", DesignCon 2010, February 1 - 4, 2010.
6. J. Eric Bracken, ANSYS, Inc., "A Causal Huray Model for Surface Roughness", DesignCon 2012, January 2012

Index