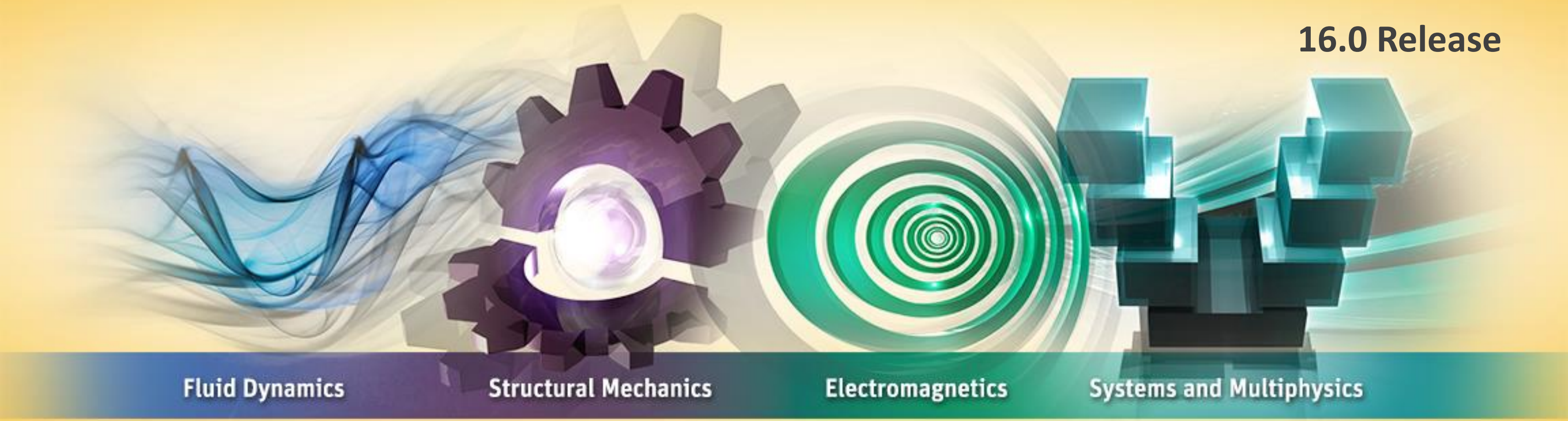
Workshop 1:
Linear Network Analysis

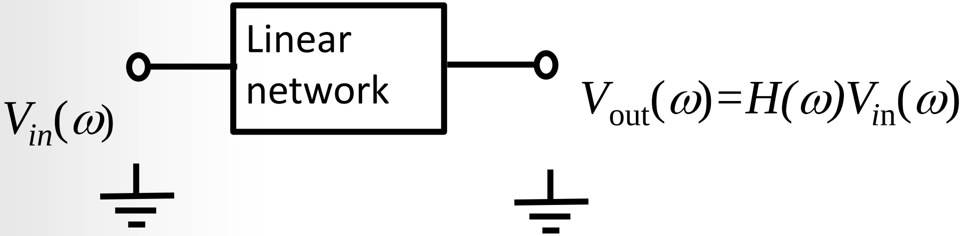
16.0 Release



Introduction to AEDT SI

Linear Network Analysis

- **Linear Network Analysis Topics to be Covered:**
  - Schematic Entry
  - W-Element Import
  - Basic Linear Analysis
  - Network Data Explorer
  - Passivity and Causality



- **Linear Network Analysis Overview**
  - Linear network analysis is used to determine the frequency dependent response of a linear network. The results can be expressed as scattering parameters (S-parameters), as well as admittance (Y) or impedance (Z) parameters. Additional user defined quantities may be derived from these network parameters .
  - It is possible to include non-linear components in a linear analysis, but these devices will be treated as if they are linear. That is, signal amplitudes are assumed to be so small that individual device transfer characteristics are linear.
  - In addition to extracting network parameters it is also possible to define sinusoidal voltage and current sources.
  - For a linear network, all output signals are proportional to the input signals as shown above.
  - The response of a linear network to a time harmonic input is always at the same frequency as the input signals.

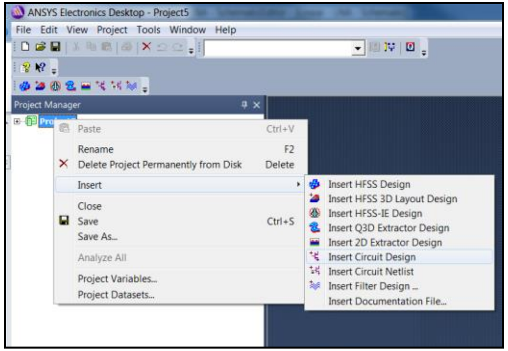
Linear Network Analysis

- **Introduction**
  - The goal of this exercise is to learn how to use the ANSYS Electronics Desktop interface to import models, create a circuit schematic, define and run analyses, create reports, and use the Data Network Explorer to examine the results. To do this, a schematic diagram of a high-speed serial channel will be created using W- and S-element models.

  - **Launching ANSYS Electronics Desktop**
    - To access ANSYS Electronics Desktop, click the Microsoft Start button, select:

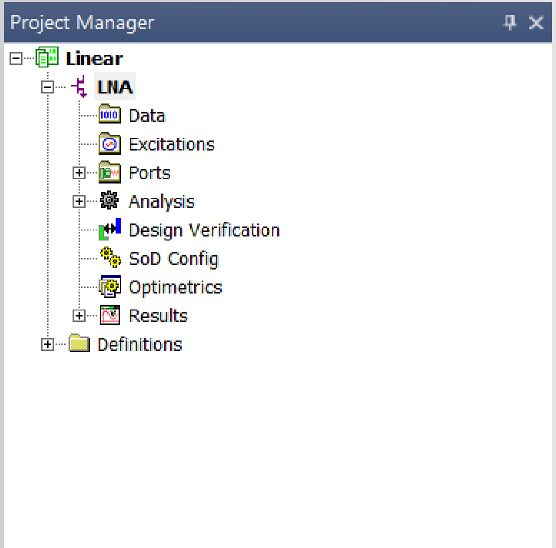
    All Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.0 > ANSYS Electronics Desktop 2015

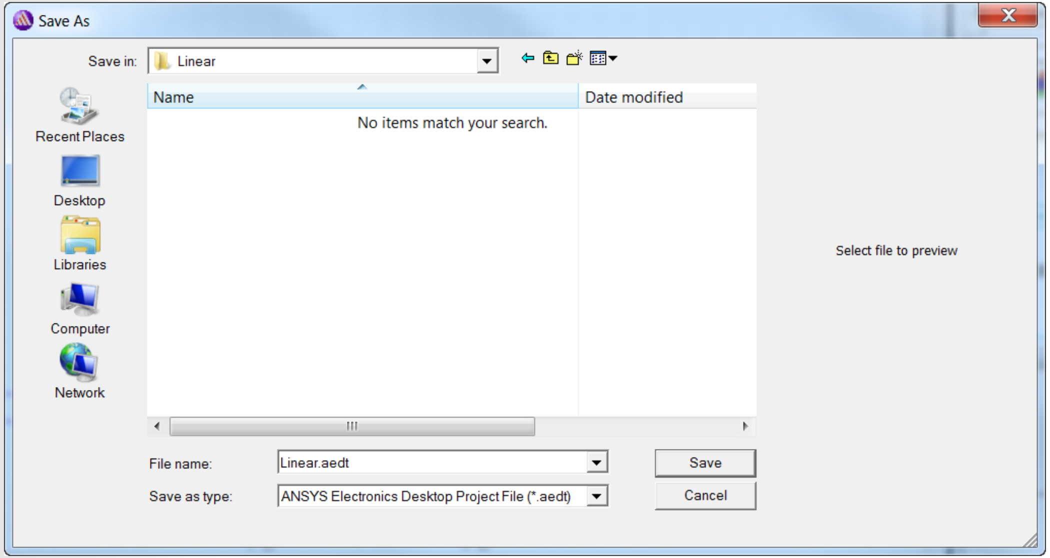
    - To open a new project:
      - In an AEDT window, Click the button on the Standard toolbar, or select the menu item ***File > New.***
      - From the Project menu, select ***Insert Circuit Design*** or **Right Mouse Click** on project folder and select **Insert Circuit Design** . A new window is created for schematic entry.
      - Click the **None** button when prompted to Choose Layout Technology.



Linear Network Analysis

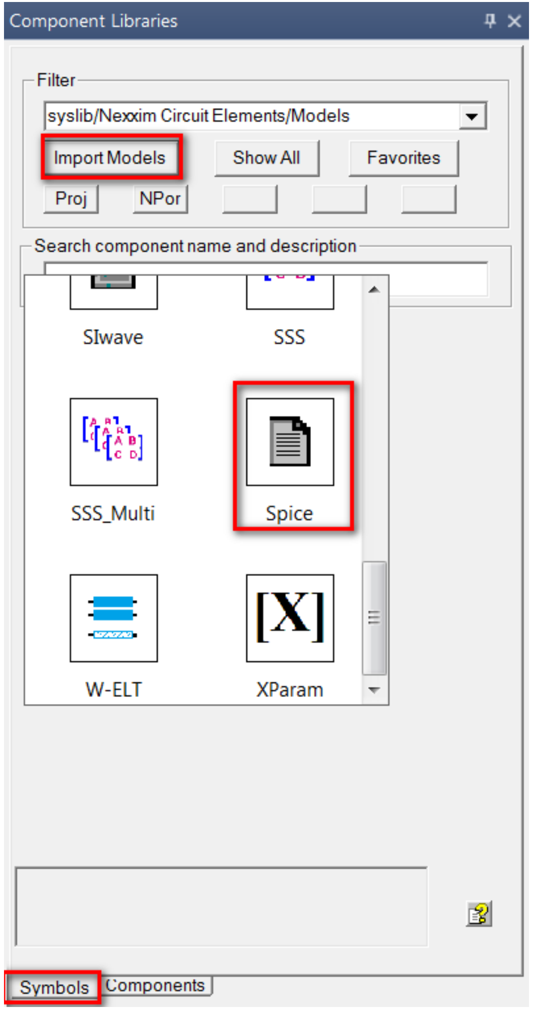
- **Save your project**
  - You will now see an empty schematic in the main Designer window.
  - **Right-click** on the design Circuit1 under the Project Manager and select **Rename** to rename the design **LNA**.
  - Select the menu item ***File > Save*** to save the project: **Linear**
    - This will create a file named **Linear.aedt** in your working directory.





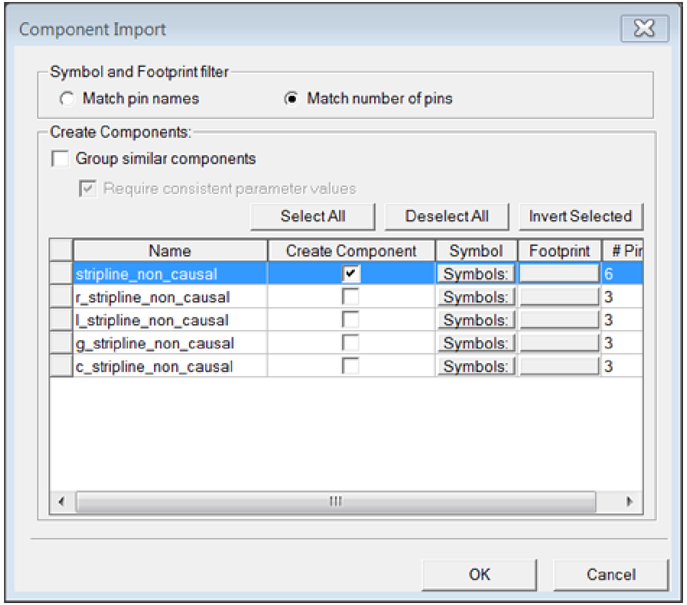
Linear Network Analysis

- **Import W-Element Model**
  - The first component that will be inserted into the schematic is a W-element model of a transmission line.
  - Go to the **Component Libraries window** and click on the button **Import Models**
  - In the **Symbols** tab select the item **Spice**
    - Browse to the folder that contains the file: **stripline_non-causal.sp** and select it
    - Be sure to use the **Project Folder** as the search path for the file



Linear Network Analysis

- **Import W-Element Model**
  - In the Component Import dialog box,
    - De-select the **Create Component** checkboxes for all models except stripline_non_causal
    - Click the **OK** button
    - Place the component on the schematic and then press **Esc**

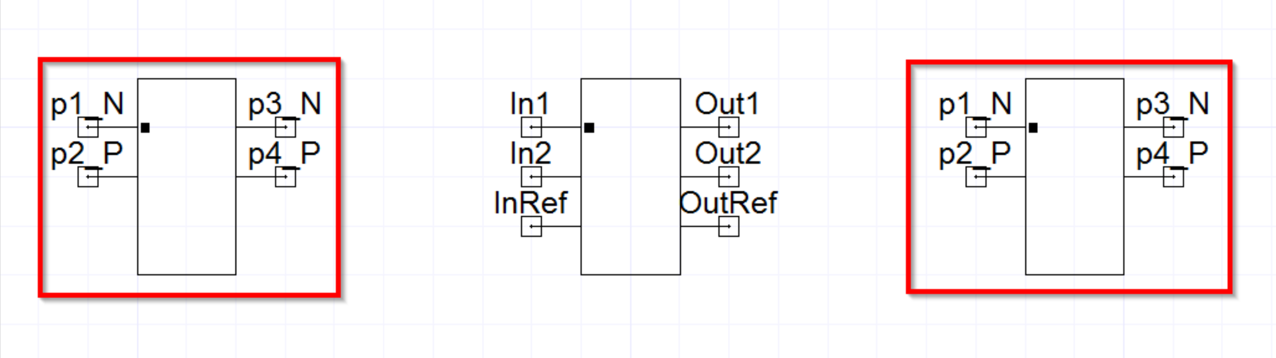


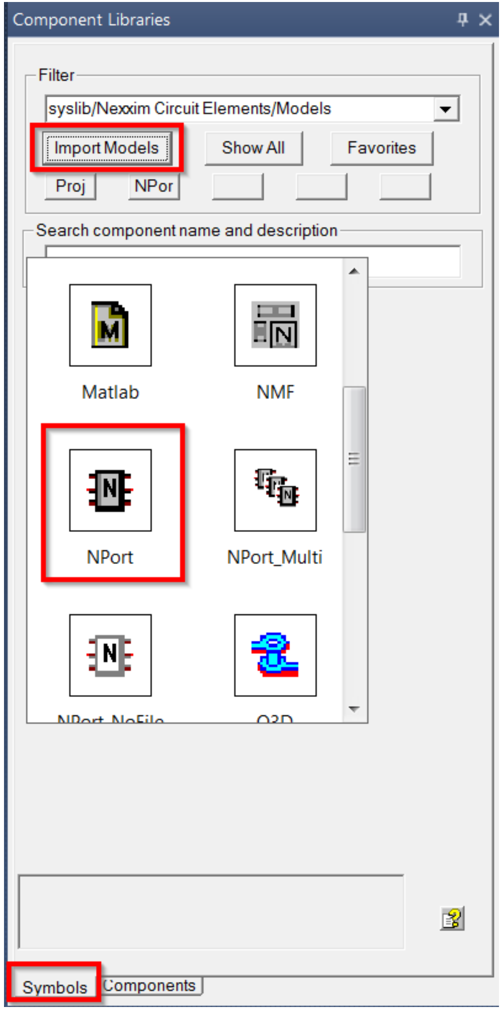
Linear Network Analysis

- **Path for linked files**
  - Files that are linked to a circuit or stored in a library (libraries will be discussed later) may use one of several search paths to locate the file.
    - **Use Path**: The path to the file is absolute and will be hardwired as-is. If this project is later opened in a different location or by another user, the absolute path may no longer be available and AEDT will be unable to find the file.
    - **PersonalLib, SysLib**: The path to the file is relative to the user's ***PersonalLib*** or ***SysLib***. These directories may be changed from the menu under ***Tools > Options > General Options***
    - **Project Folder**: The path to the file will be relative to the location of the project. Therefore, if the AEDT project is opened in another location or by another user over the network, the linked file will still be found as long as its location does not change relative to the AEDT project (i.e. the *.aedt file).

Linear Network Analysis

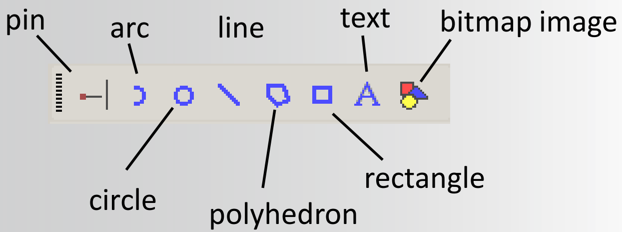
- **Import S-Parameter Model**
  - The next step is to import S-parameter data to model the via transition at either end of the W-element transmission line.
  - Go to the **Component Libraries window** and click on the button **Import Models**
  - In the **Symbols** tab select the item **NPort**
    - Browse to the folder that contains the file: **via_non-causal.s4p** and select it
    - Be sure to use the **Project Folder** as the search path for the file.
    - Place the component on the left and on the right of the previous component.
    - Press **Esc**

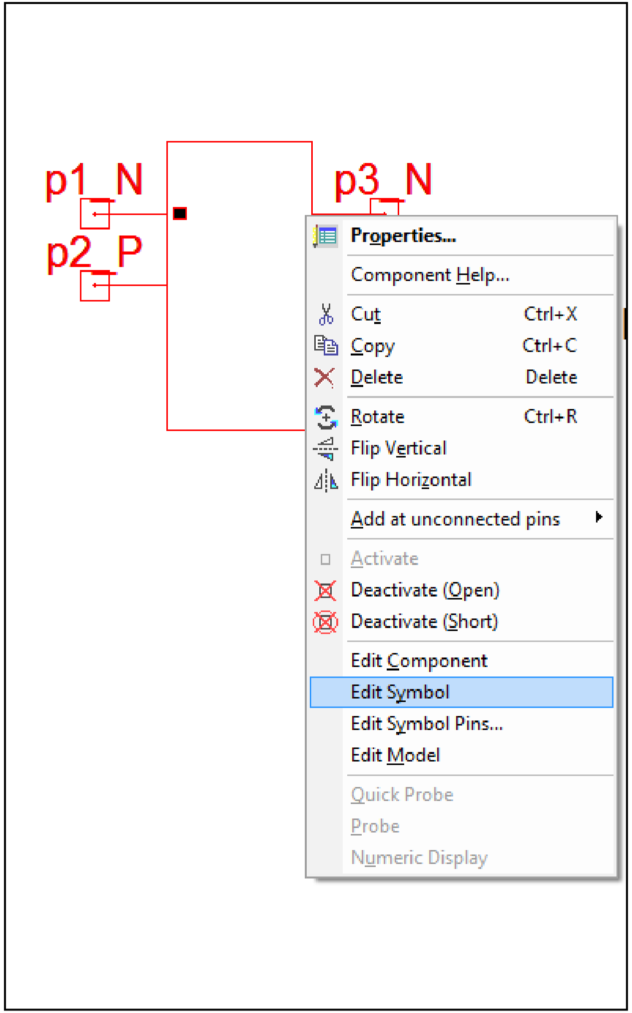




Linear Network Analysis

- **Modify the component symbols**
  - Each new component was created with a generic symbol. If many components are imported and used in the channel the schematic becomes difficult to read. Creating a custom symbol for each component helps to keep the schematic manageable.
  - To modify the generic symbol for the via model, right click on the component in the schematic and choose the option **Edit Symbol**.
  - This causes the symbol editor to be opened.
  - The following primitives are available for modifying or creating symbols:



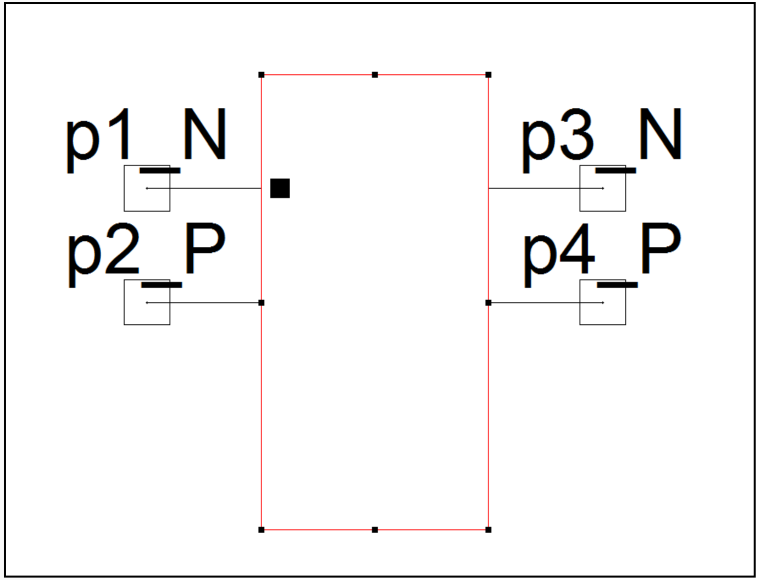


Linear Network Analysis

- **Modifying the Component Symbol (continued)**
  - The properties of any primitive can be modified in the properties window or by double-clicking with the left mouse button on the primitive. For example, the properties of a rectangle are shown here:

Properties

Name	Value	Unit	Evaluated Valu
Schematic...	2		
Color			
BorderWidth	0	mil	
BorderColor			
FillStyle	Hollow		
Center	0 , 0	mil	
Width	200	mil	
Height	400	mil	
Angle	0	deg	
Level	0		

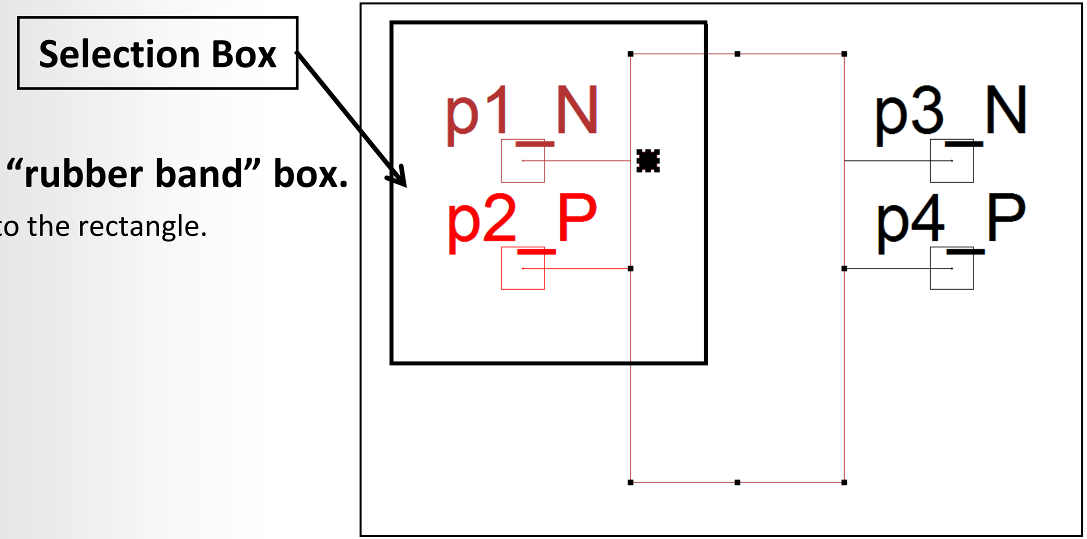


Linear Network Analysis

- **Modifying the Component Symbol (continued)**
  - Primitives can be moved the same way they are in the schematic.

    - Rotate <ctrl>-R
    - Mirror

- **Multiple objects can be selected with a “rubber band” box.**
  - The pins p1_N and p2_N are selected in addition to the rectangle.



  October 6, 2015

Linear Network Analysis

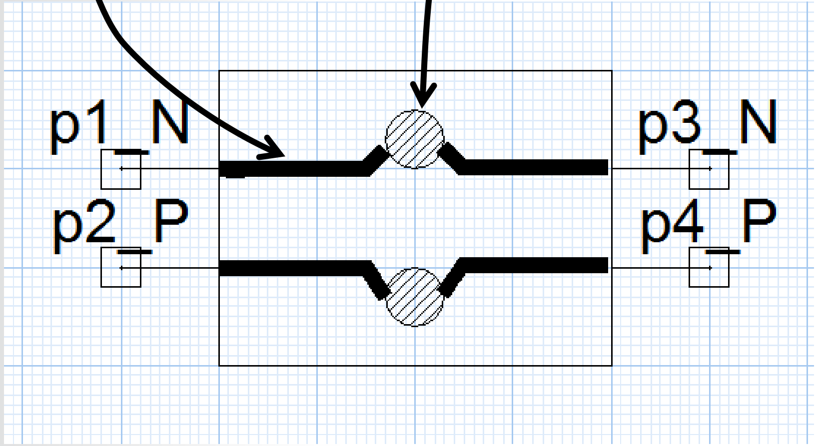
- **Modifying the Component Symbol (continued)**

Properties

Name	Value	Unit	Evaluated V...
Schema...	4		
Color			
Linewidth	8	mil	
Line Style	Solid		
Begin O...	None		
End Obj...	None		
Vertex#0	-200 , 100	mil	
Vertex#1	-30 , 100	mil	
Vertex#2	-20 , 110	mil	
Level	1		

Properties

Name	Value	Unit	Evaluated V...
Schema...	2		
Color			
Border...	0	mil	
BorderC...			
FillStyle	NE Diagonal		
Center	0 , 130	mil	
Radius	30	mil	
Level	1		

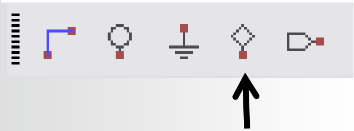


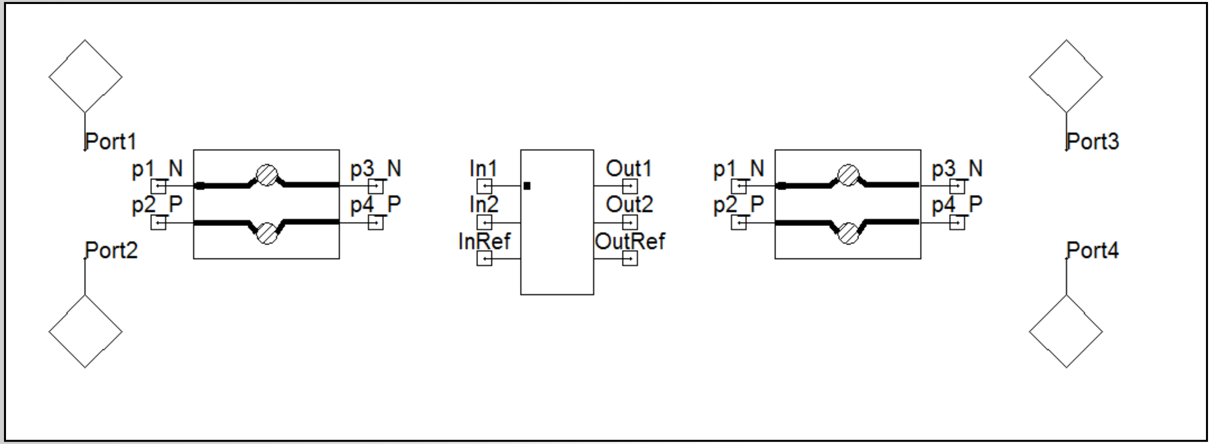
- Now try to make the symbol look like that shown here. When finished, save the changes by choosing ***Symbol > Update Project*** from the menu.

- After modifying the symbol and saving the changes, close the editor. The changes will be visible in the schematic.

Linear Network Analysis

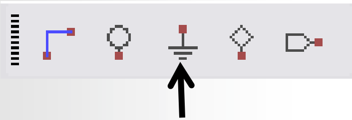
- **Placing Ports**
  - Place four ports as shown by clicking on the port icon or selecting the ***Draw > Interface Port*** menu item.
  - Note that the orientation of any symbol can be rotated while it is being placed by pressing the **R** key.
  - Click the right mouse button in the schematic and select **Finish** when done placing ports (or press the **<space bar>**)

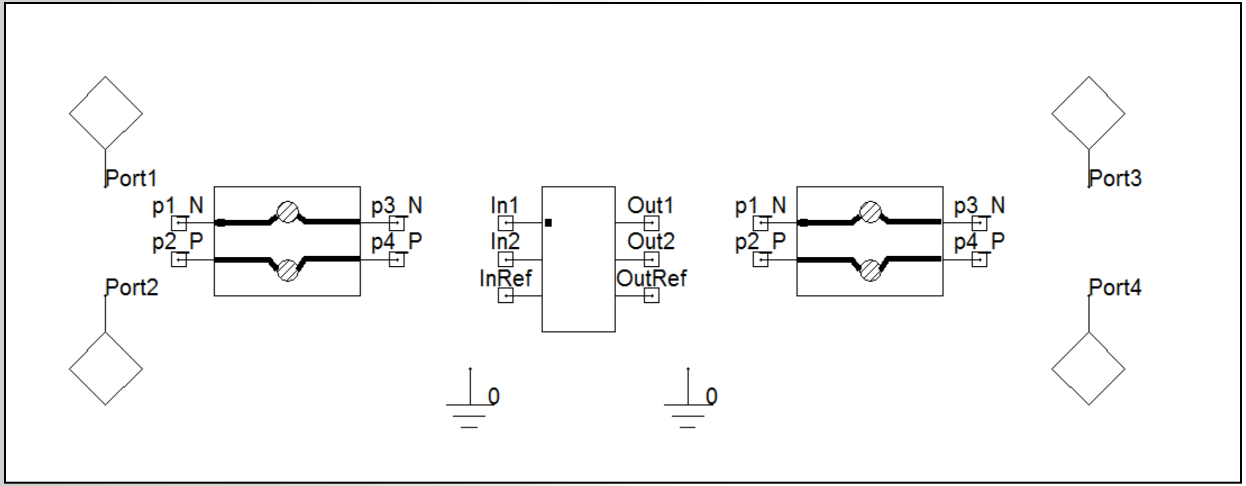




Linear Network Analysis

- **Place Grounds**
  - Place two ground connections as shown by clicking on the ground icon or selecting the ***Draw > Ground*** menu item.
  - Right-click in the schematic and select **Finish** when done placing grounds.





  October 6, 2015 ANSYS Confidential

Linear Network Analysis

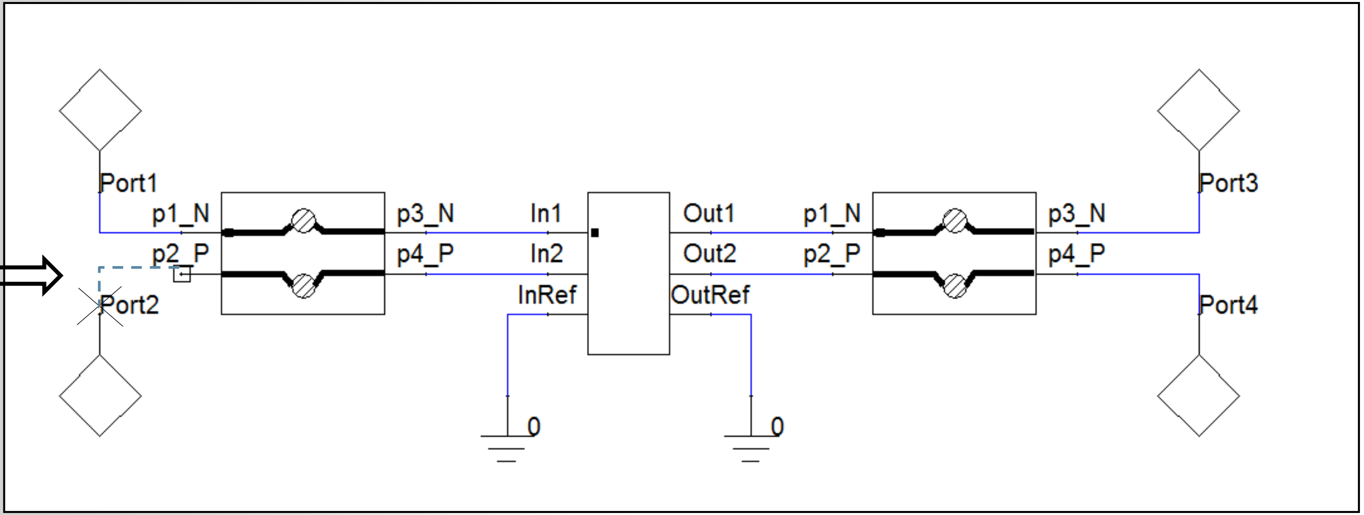
- **Wire the components**
  - Method 1:
    - – Place the cursor over a component pin
      - The cursor becomes an to indicate the wiring tool is active.
    - – Click with the left mouse button on a pin.
      - A blue “wire” appears when the cursor is moved away from the pin.
    - – Click on the second pin to make a connection.

  - Method 2:
    - – Move a component so that its pin touches another pin.
    - – An electrical connection is automatically established.
    - – The components can then be separated and the connection remains.
  - Wire the schematic as shown below (complete the connection for Port2 as well):

Linear Network Analysis

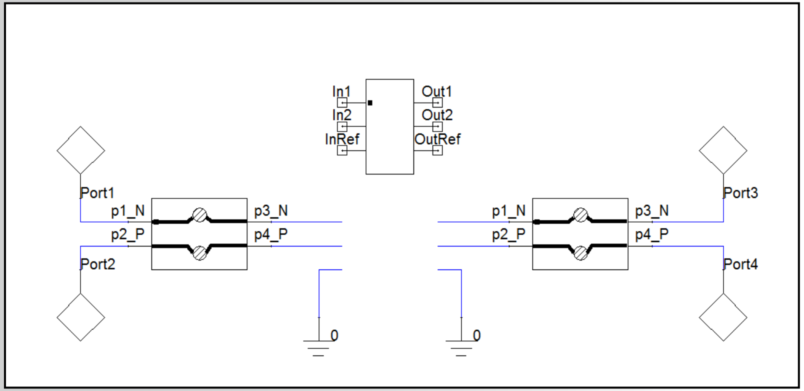
- **Wire the components**

Clicking the left mouse starts creating a wire connection



Linear Network Analysis

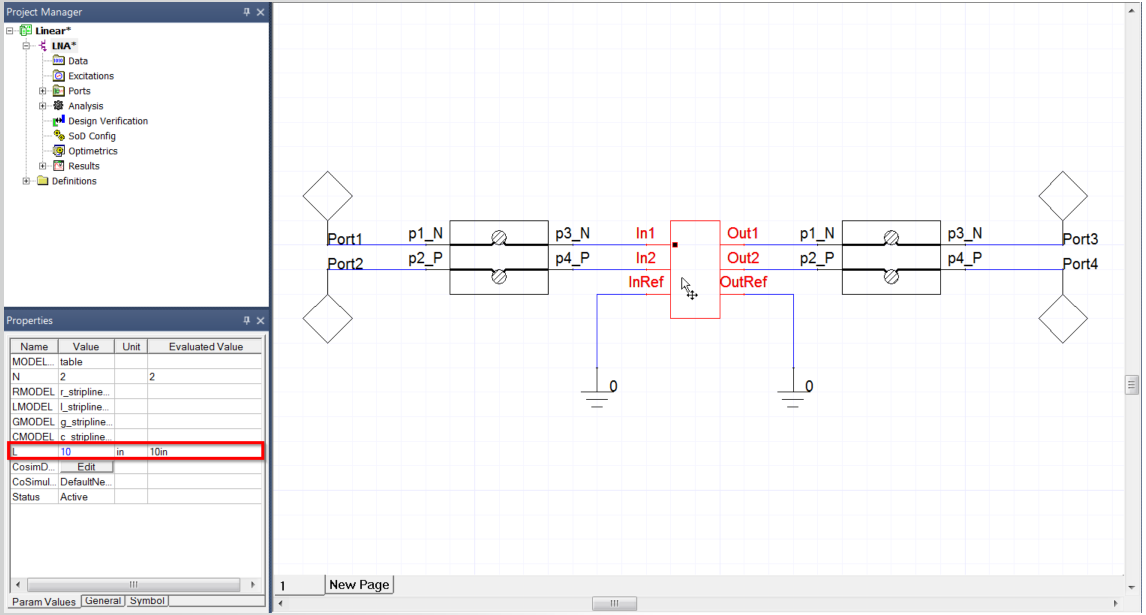
- **Working with Schematic Entry**
  - The schematic editor has several editing features that may be useful when manipulating a schematic.
  - The shortcut combination **<ctrl>-R** rotates a component while it is being moved in the schematic.
  - A new instance of any component that has already been placed in the schematic can be created by copying **<ctrl>-c** and pasting **<ctrl>-v** a selected component in the schematic.
  - The shortcut to **pan** within the schematic window is **<shift>-<left mouse button> .** Dragging the mouse across the schematic moves everything in the window with the cursor.
  - The shortcut to **zoom** within the schematic window is **<alt>-<shift>-<left mouse button>.** Dragging the mouse up or down will zoom in or out within the schematic.



Pressing **<shift>-<ctl>** while dragging components in the schematic allows components to be moved without retaining the electrical connections.

Linear Network Analysis

- **Change the length of the transmission line.**
  - The properties of any object that is selected in the Designer user interface are shown in the **Properties** window as can be seen below.
  - Select the W-element transmission line component and change the value of the variable L to 10 inches.



Linear Network Analysis

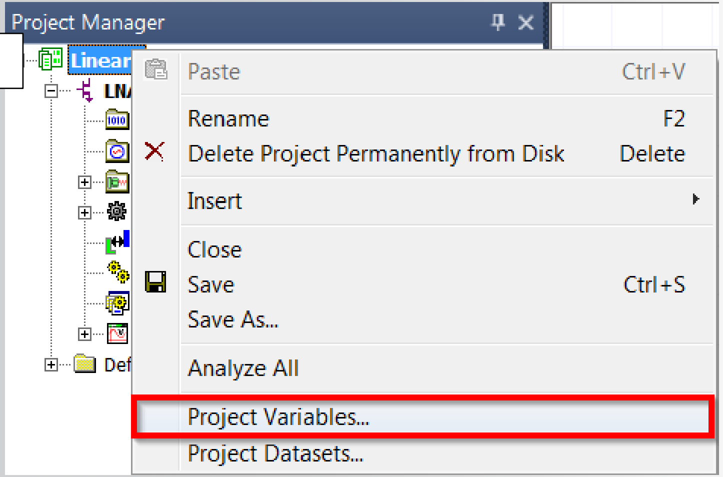
- **Defining Variables**
  - Three types of variables can be defined in AEDT. These variables differ in their scope of visibility within a project.
    - **Project variables** are available across all hierarchical levels of a project and are preceded by a dollar sign ($).
    - **Parameters** are available in the local schematic, but their values may be modified from a higher level of the hierarchy. For example, if parameters are used in a subcircuit, the parameter values are to be modified from the higher level schematic.
    - **Local variables** are available only within the local schematic and can only be modified from within the schematic where they are defined.

Linear Network Analysis

Defining variables

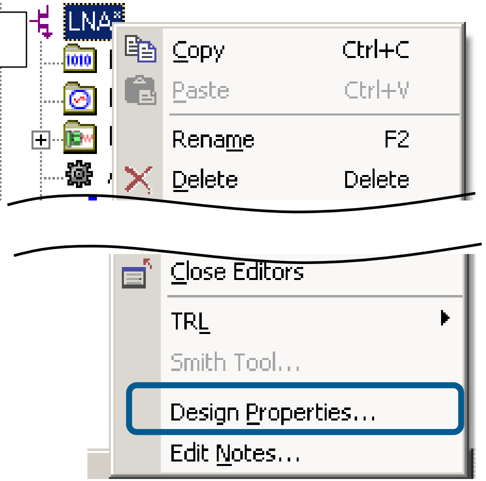
Project variables

<right mouse>



Parameters and local variables

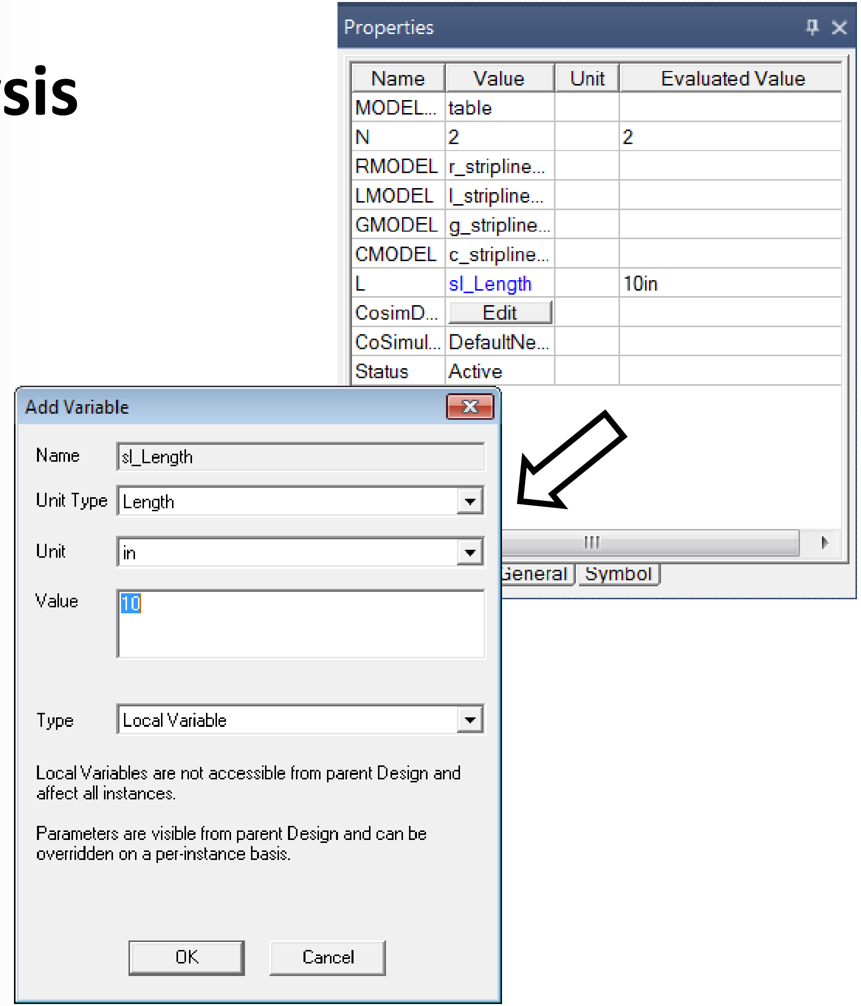
<right mouse>



Linear Network Analysis

- **Defining Variables (continued)**
  - Variables may also be defined “on the fly” by assigning a variable as a value before the variable has been defined. AEDT will query the user to enter the value and type of the variable if it has not already been defined.
  - AEDT will automatically recognize the “$” prefix as belonging to a project variable.

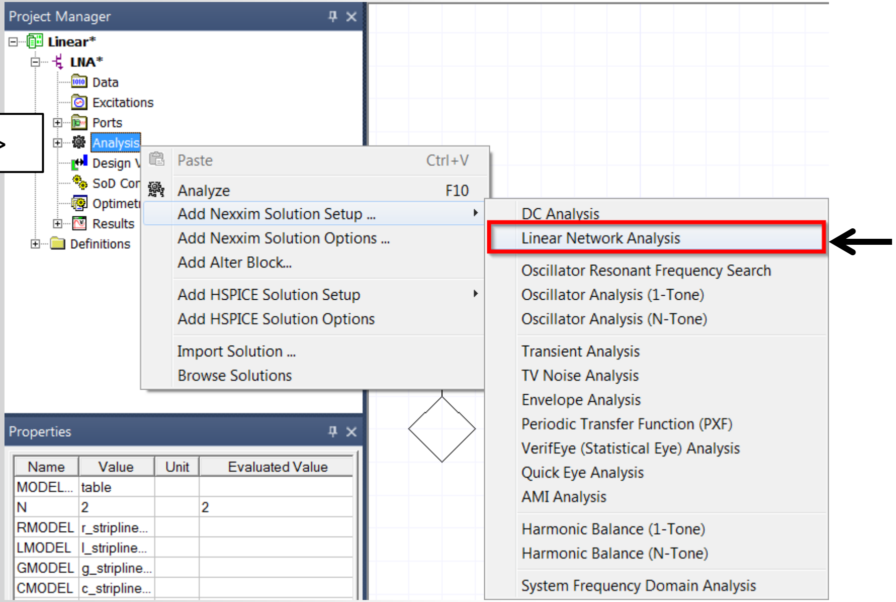
- **Define and assign a local variable**
  - Again select the transmission line component in the schematic.
  - Assign the value **sl_Length** to the component parameter **L**. Designer will ask for the value of this variable and create the definition. Choose **Local Variable** as the **Type**.
  - Assign the value **10 in** to **sl_Length** as shown here.



Linear Network Analysis

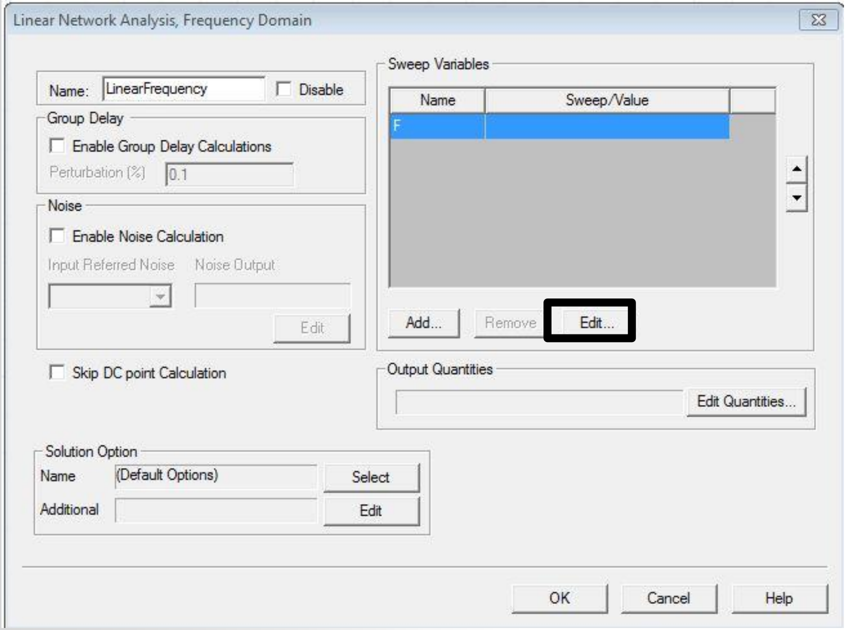
- **Specify the analysis**
  - In order to run a simulation, an analysis type and setup must be associated with the current schematic. This is done by clicking with the right mouse button on **Analysis** in the Project Manager. Select ***Add Nexxim Solution Setup… > Linear Network Analysis***
  - Note that this can also be achieved from the menu by selecting ***Circuit > Add Nexxim Solution Setup > Linear Network Analysis*** for the active design (i.e. the currently selected schematic).

<right-mouse>



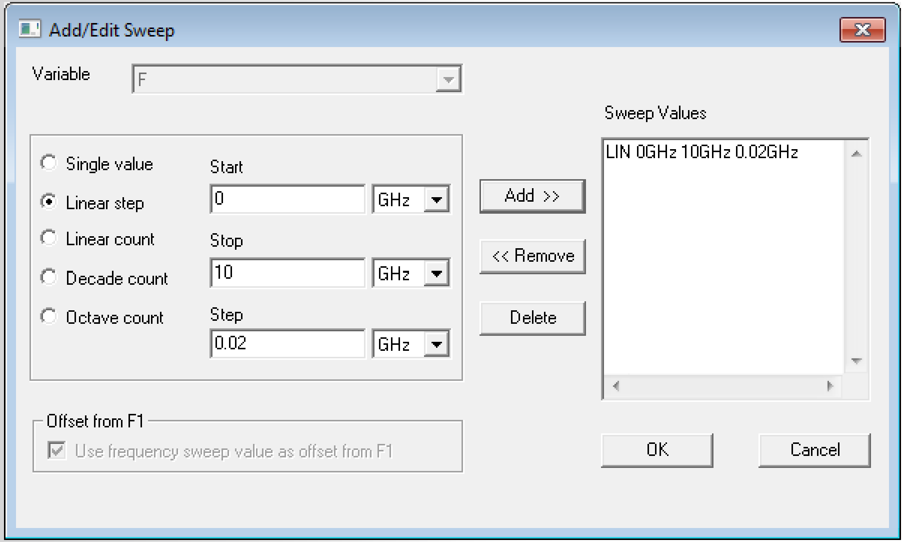
Linear Network Analysis

- **Specify the Analysis (continued)**
  - The most important part of the analysis setup is the sweep definition. A placeholder with the variable F is shown in the window, but no sweep values are defined. Press the **Edit…** button to define the frequency range and resolution for the linear analysis. The swept variable will be frequency (F)



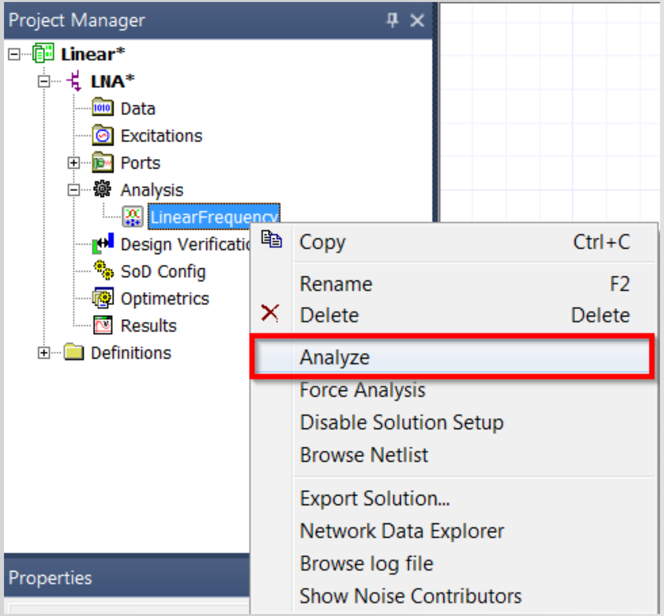
Linear Network Analysis

- **Specify the Analysis (continued)**
  - Specify a **Linear step** frequency analysis from 0 to 10 GHz with a step size of 0.02 GHz as shown below.
  - Press **Add >>** to add this sweep to the list of values to be simulated.
    - Press **OK** when the sweep values have been defined.
  - Press **OK** a second time to accept the linear frequency sweep definition.

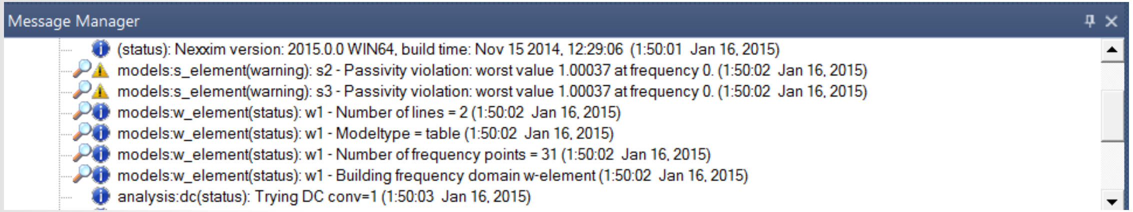


Linear Network Analysis

- **Run the Simulation**
  - The analysis setup named ***LinearFrequency*** has been added to the project. The setup is visible in the project tree when the **Analysis** tab is expanded.
  - The setup can now be run by clicking with the right mouse button on **LinearFrequency** and choosing **Analyze.**

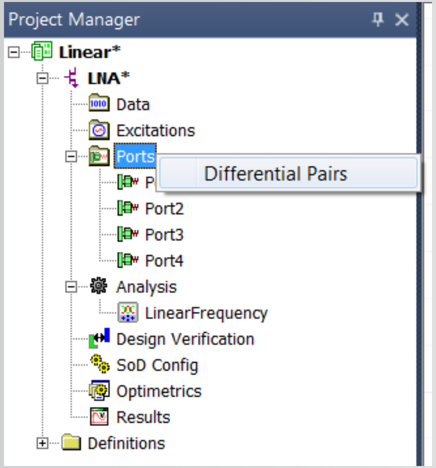


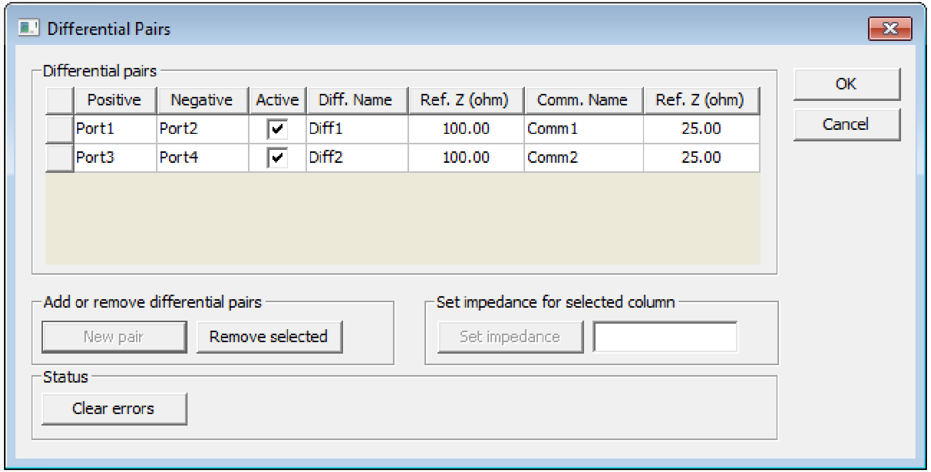
– When the analysis is launched, messages will appear in the **Message Manager** indicating simulation status, showing warnings or errors.
– In the current example, there are two warnings about passivity violation.
– Passivity will be discussed later in this exercise.



Linear Network Analysis

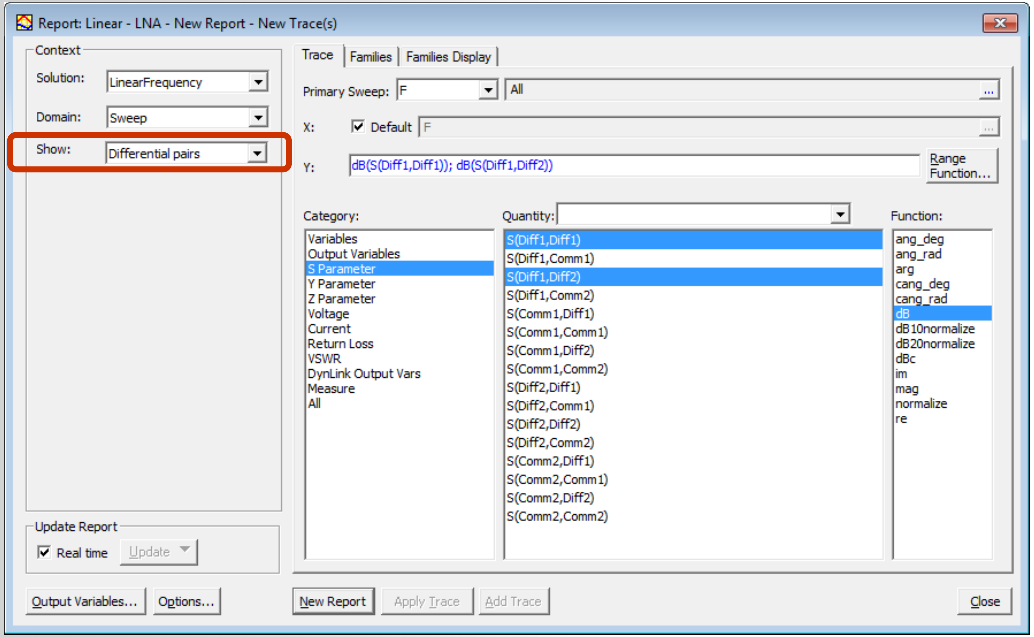
- **Post-Processing**
  - High speed serial channels typically use differential signaling to improve impedance control and noise immunity. Therefore, differential pair must be defined so the corresponding network parameters are available for post-processing.
  - Click with the right mouse button on **Ports** in the project tree and open the **Differential Pairs** definitions.
    - Press the button **New Pair**. A new differential pair is defined with ***Port1*** being positive and ***Port2*** being negative. The common and differential mode impedance are also defined.
    - Pressing the **New Pair** button a second time creates the 2nd differential pair between ***Port3*** and ***Port4***.
    - Press **OK** when finished.





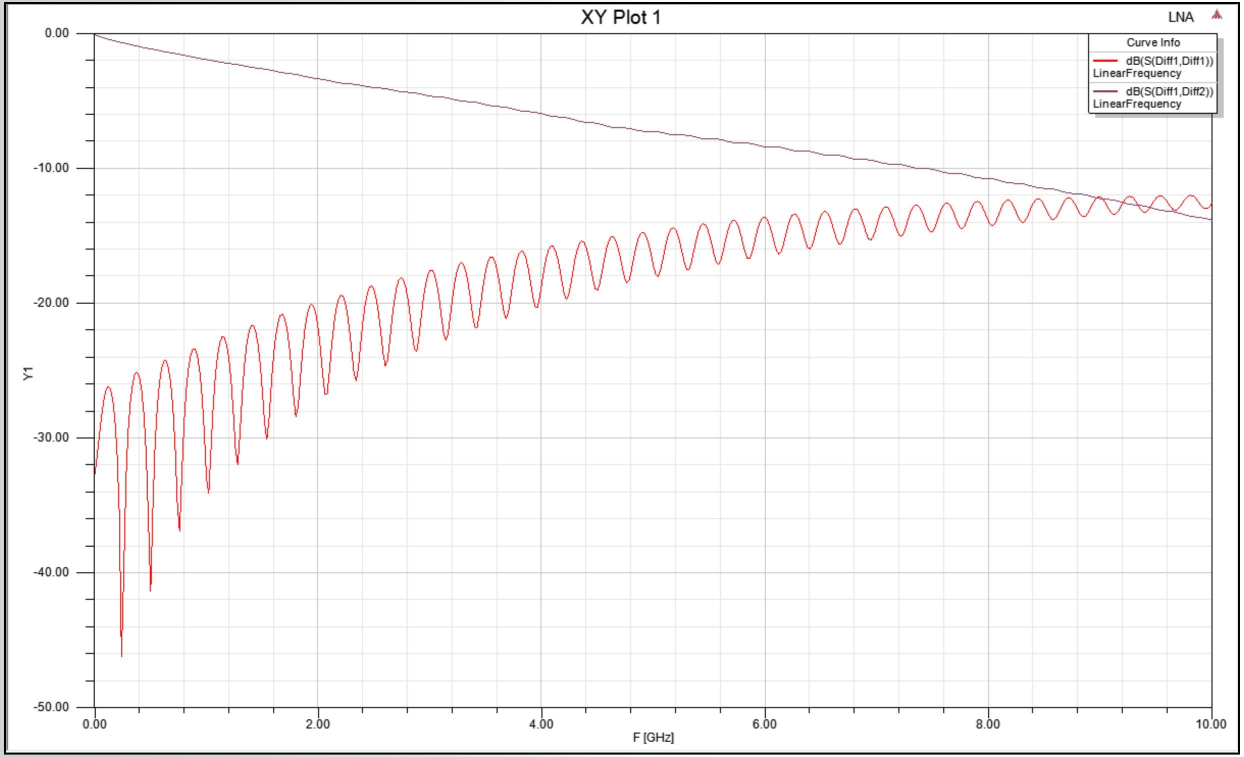
Linear Network Analysis

- **Post-Processing (continued)**
  - The S-parameters can be viewed by creating a report. From the menu select ***Circuit > Results > Create Standard Report > Rectangular Plot***
  - Select the values ***S(Diff1,Diff1)*** and ***S(Diff1,Diff2)*** from the **Quantity** column. Select the function **dB** from the **Function** column as shown below.



Linear Network Analysis

- **Results – Continue to next page**



Linear Network Analysis

- **Report Editor Dialog**
  - The report editor dialog has many options for displaying data. These are described briefly below.

  - **Context**
    - **Solution** – This is the name of the analysis setup. It is possible to define many analyses for a single schematic. If multiple analyses exist, they will all be shown in the drop down menu.
    - **Domain** – For some analysis types, there may be multiple domains for which data display makes sense. For example, plotting transient results in the frequency domain causes a windowed FFT of the time domain data to be displayed.

  - **Update Report**
    - **Real Time** ☑ – enables real time updating for reports. If this is not selected, only the most recent simulation data will be displayed.

Linear Network Analysis

- **Report Editor Dialog**
  - **Trace Tab**
    - **X (Primary Sweep)** – Select the independent variable to be plotted on the abscissa (x-axis).
    - **Y** – Specifies the dependent quantity to be associated with the ordinate (y-axis).
    - **Category** – the available categories depend on the solution type. In this exercise, S-parameters will be plotted.
    - **Quantity** – This is the specific quantity to be displayed. Holding down the **<ctrl>** key and clicking with the left mouse button on various values allows multiple quantities to be selected.
    - **Function** – This is the function to apply to selected quantity when plotting. For example **dB()** or **cang_deg()** (=continuous phase angle)
    - **Range function** – opens the **Set Range Function** dialog that allows the calculation of minimum, maximum, and other array functions to be applied. For example, if the linear analysis had been run for multiple lengths of the W-element transmission line, the maximum loss at any frequency could be plotted as a function of length.
  - **Families Tab**
    - The families tab can be used to control the display of multiple parametric analyses. That is, if the linear analysis has been carried out for multiple values of some parameter, the **Families** Tab is used to select which solution results are to be displayed.

Linear Network Analysis

- **Passivity and Causality**
  - Two important properties of models based on frequency domain data are causality and passivity. AEDT makes it possible for users to easily investigate these properties for a channel and, if necessary, correct the models so they are passive and causal for transient analysis. These features can be investigated using linear frequency domain analysis.
  - State-space model
    - The Nexxim simulator offers several methods to incorporate frequency domain models in a transient simulation. One robust method is to fit a rational polynomial transfer function, or state-space model to the frequency domain data. This model is inherently causal.

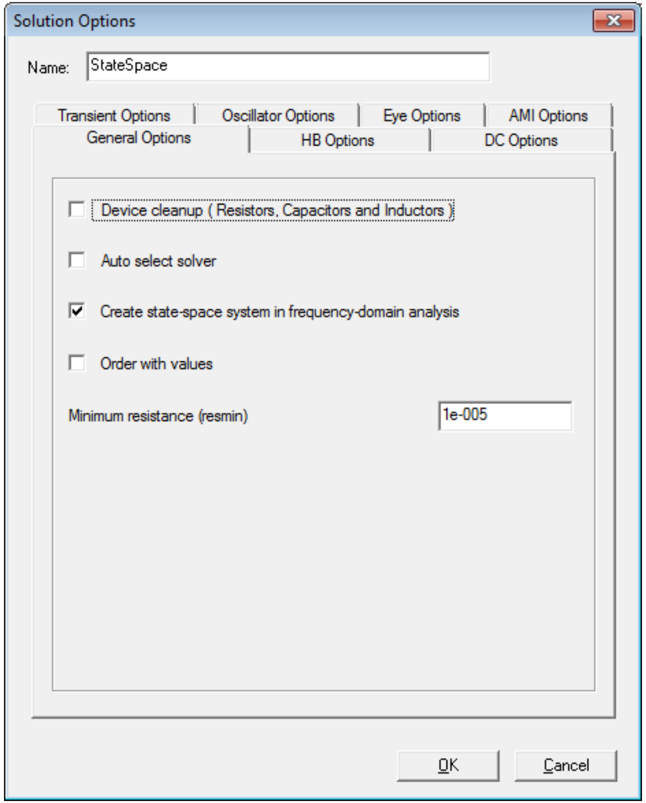
S-domain representation of a rational polynomial transfer function from port *i* to port *j*. The transfer poles are at p_n, the tranfer zeros at z_q. The variable *s* is the complex frequency *j(2pf)*

$$H_{ij}(\omega) = h_0 \frac{(s - z_Q)(s - z_{Q-1})\ldots(s - z_1)}{(s - p_N)(s - p_{N-1})\ldots(s - p_1)}$$

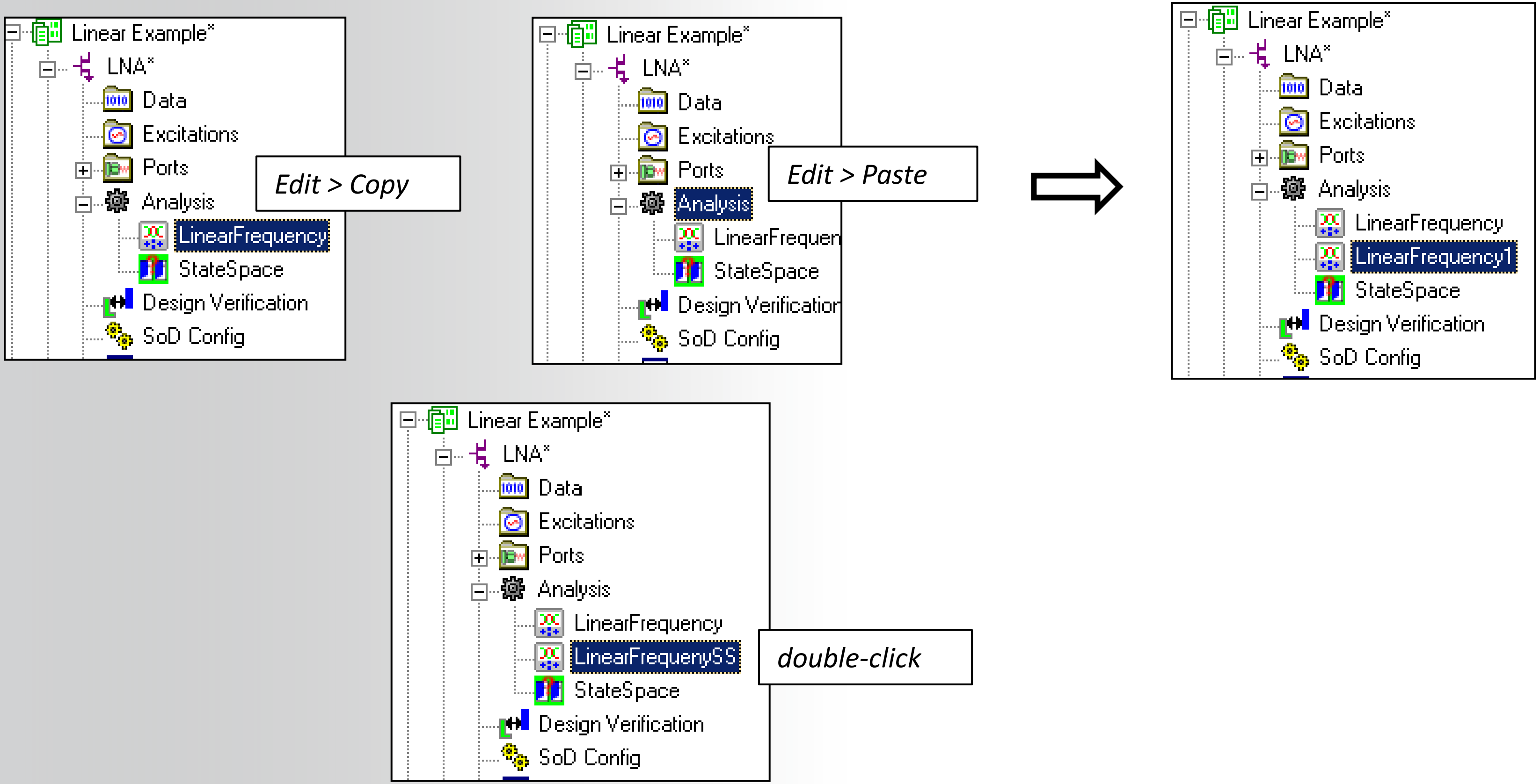
  - The following section describes how investigate the state-space model generated by Nexxim using linear frequency domain analysis.

Linear Network Analysis

- **Causality in Linear Network Analysis**
  - Nexxim can be forced to run the linear analysis using a state space model by adding this requirement as an option. From the menu choose ***Circuit > Add Nexxim Solution Options…***
  - In the ***General Options*** tab choose ***Create state-space system in frequency-domain analysis.***
  - Name this definition of **Solution Options** *StateSpace* as shown here**.**
  - Press **OK** to accept the definition.
  - Create a new analysis definition by copying the setup named ***LinearFrequency*** and pasting it into the **Project Manager** as shown here.
  - Rename the new setup to *LinearFrequencySS*
  - Now modify the properties of the newly defined analysis setup by double clicking with the left mouse button on the setup.

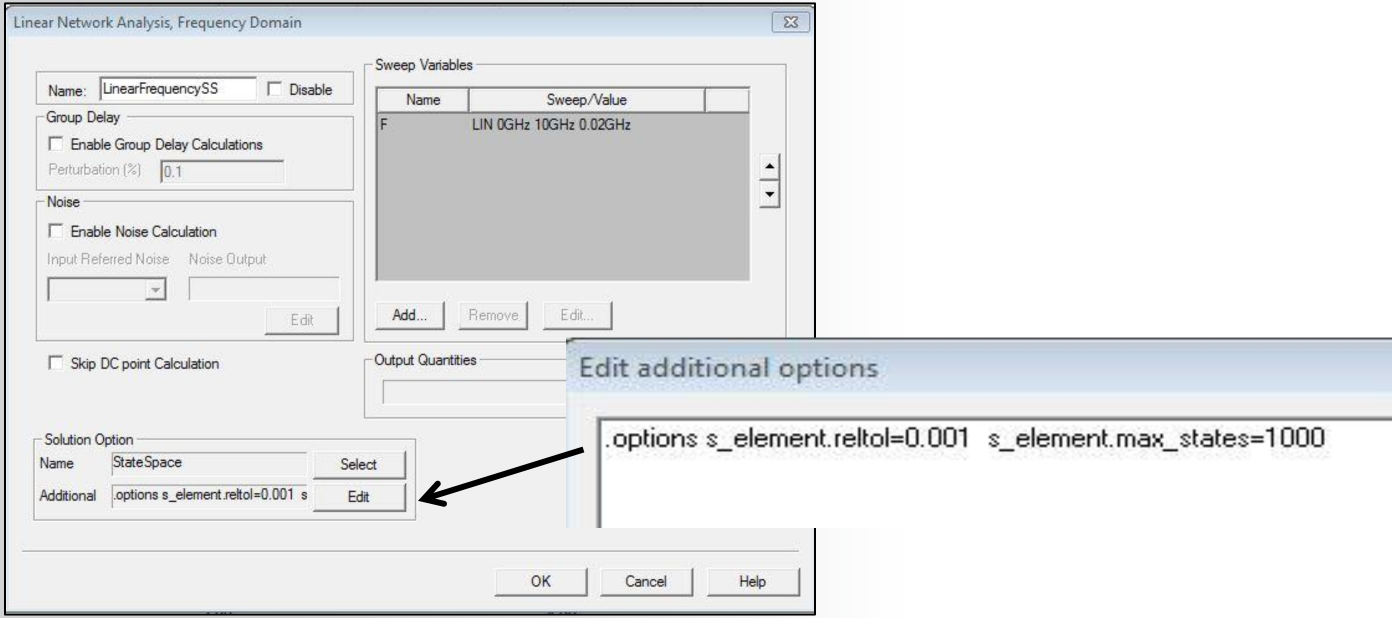


Linear Network Analysis



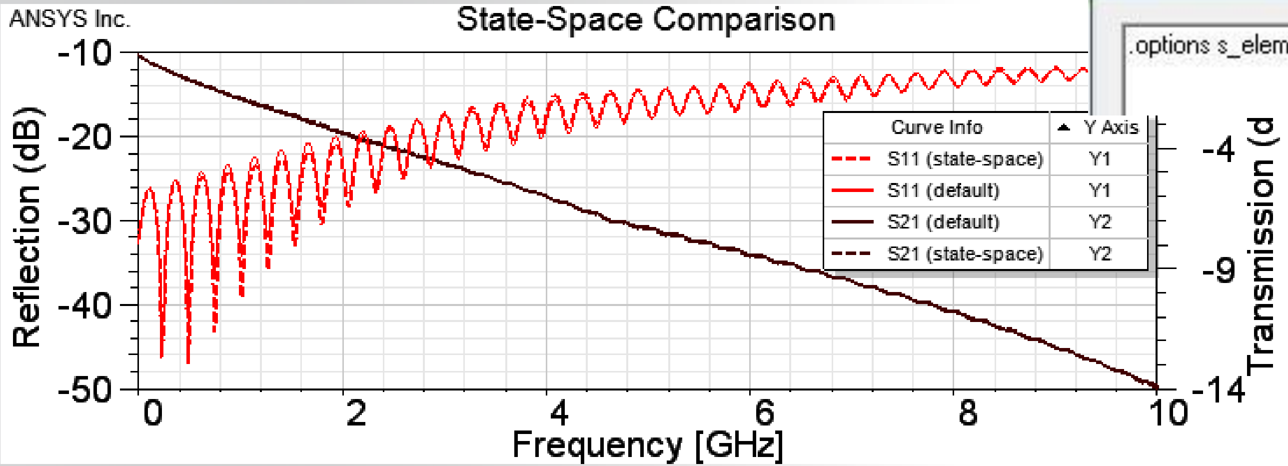
Linear Network Analysis

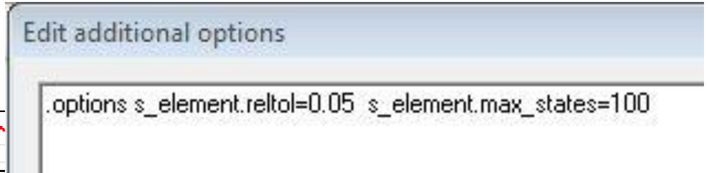
- **Causality in Linear Network Analysis (continued)**
  - Insert the name of the new options definition under **Solution Options**.
  - Under **Additional Options** enter the fitting tolerance (*reltol*) and the maximum number of states to be used for the state-space function (*max_states*).
    - **.options s_element.reltol=0.001 s_element.max_states=1000**



Linear Network Analysis

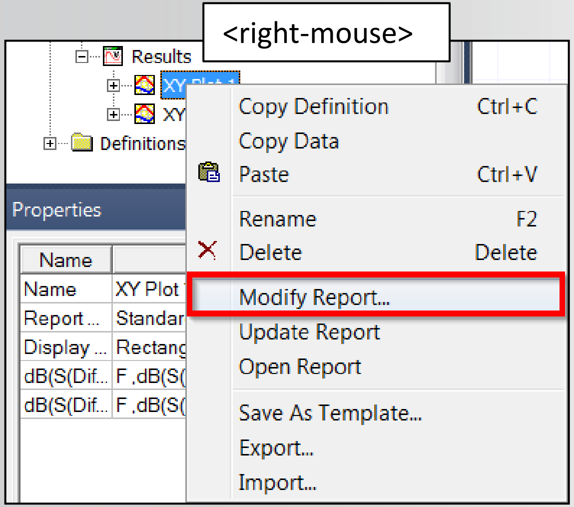
- **Causality in Linear Network Analysis (continued)**
  - Run both analysis and compare the results. Note that the analysis *LinearFrequencySS* seems to pause while the message **Loading Data…** is displayed in the progress window. This happens while the state-space fitting occurs. When the s-parameter data has many poles and zeros, or the model is not causal, this step may take several minutes.
  - The default settings for the state-space model lead to s-parameters that are indistinguishable from the original data, provided the frequency domain data is nearly causal.
  - Try reducing the number of states to 100 and relaxing the tolerance setting to 5% in the analysis setup *LinearFrequencySS* to see how the results change.

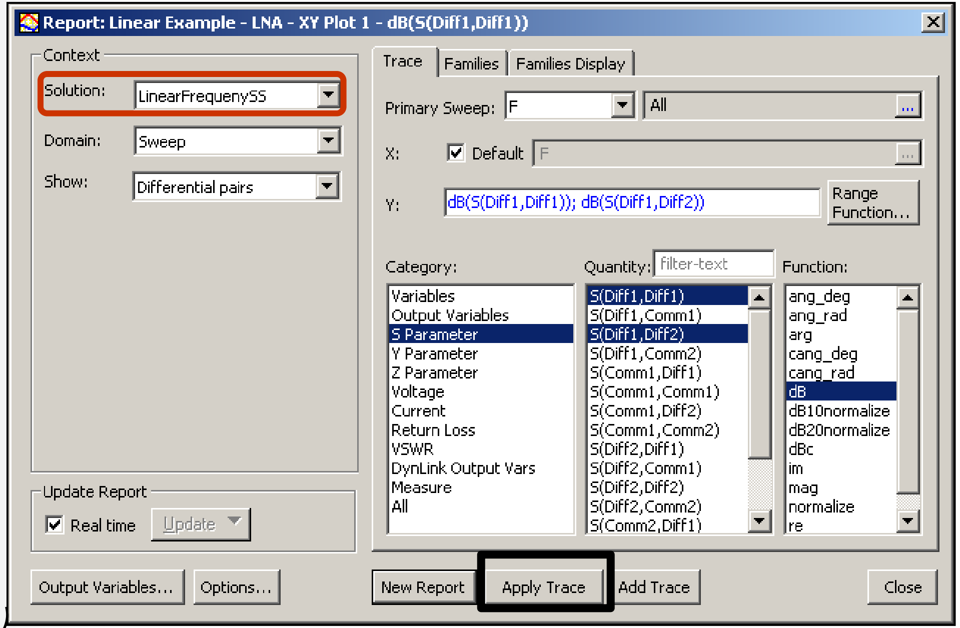




Linear Network Analysis

- **Compare Results**
  - Results from the second analysis *LinearFrequencySS* can be added to an existing plot by clicking on the plot definition with the left mouse button and choosing **Modify Report…**

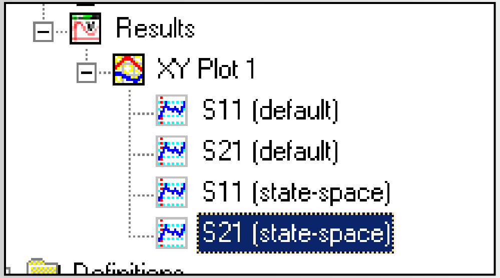


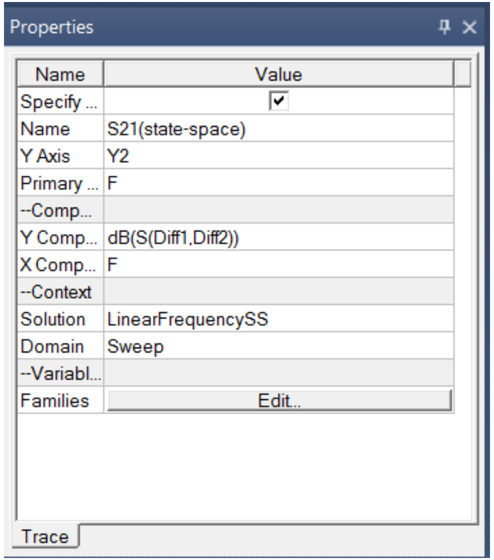


- As when a new report is created, select the desired **Solution** (*LinearFrequencySS*), **Quantity** *S(Diff1,Diff1)* and *S(Diff1,Diff2)* and **Function** (**dB**).
- Press **Add Trace** to add the new data to the existing plot.

Linear Network Analysis

- **Modifying Report Traces**
  - It is helpful to change the color or style of the traces to help distinguish them in the report.
  - Modify the names of the traces by selecting them in the project tree. When a trace is selected in the project tree, the properties displayed in the Properties window may be modified.
    - Change the names of the traces as shown here by checking the Specify Name box in the properties window and typing the name.
    - Display the transmission s-parameters (S21) on the **Y2-Axis**.





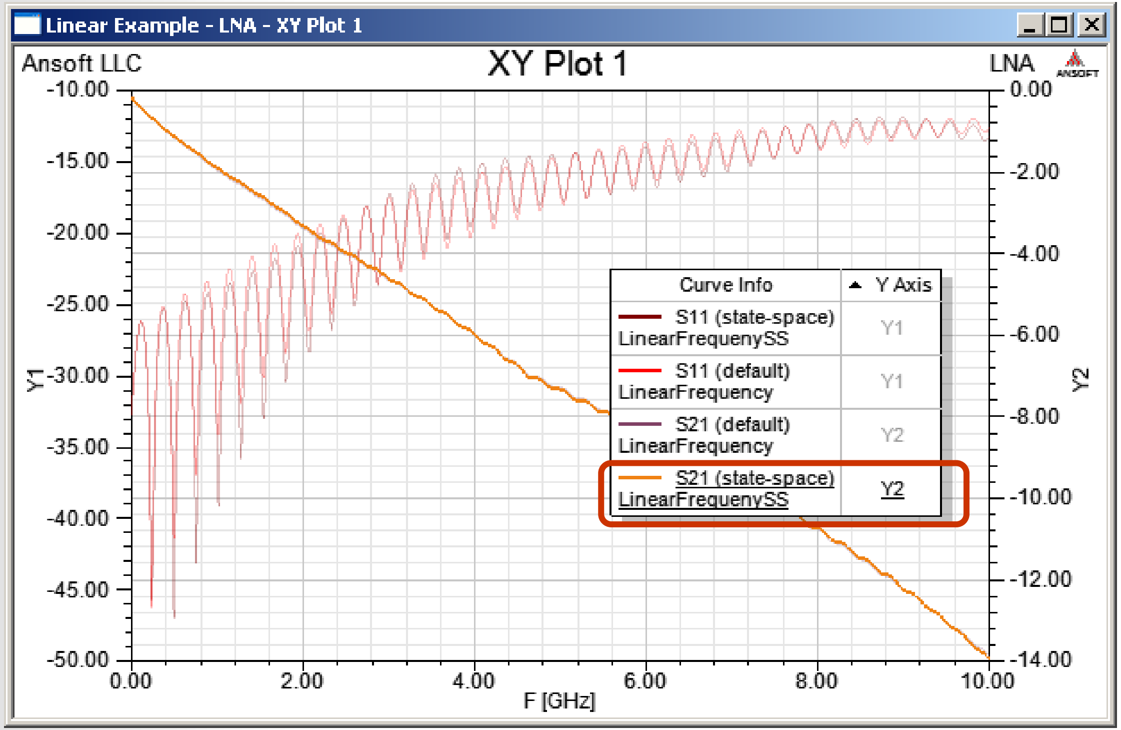
Linear Network Analysis

- **Displaying Data**
  - Individual traces can also be modified by selecting the trace in the report or legend and modifying values in the properties window.

Properties

Name	Value
Name	S21(state-space):Curve1
Color	
Line Style	Solid
Line Wi...	1
Trace T...	Continuous
Show S...	☐
Symbol ...	15
Symbol ...	Circle
Fill Sym...	☐
Symbol ...	
Show Ar...	☐

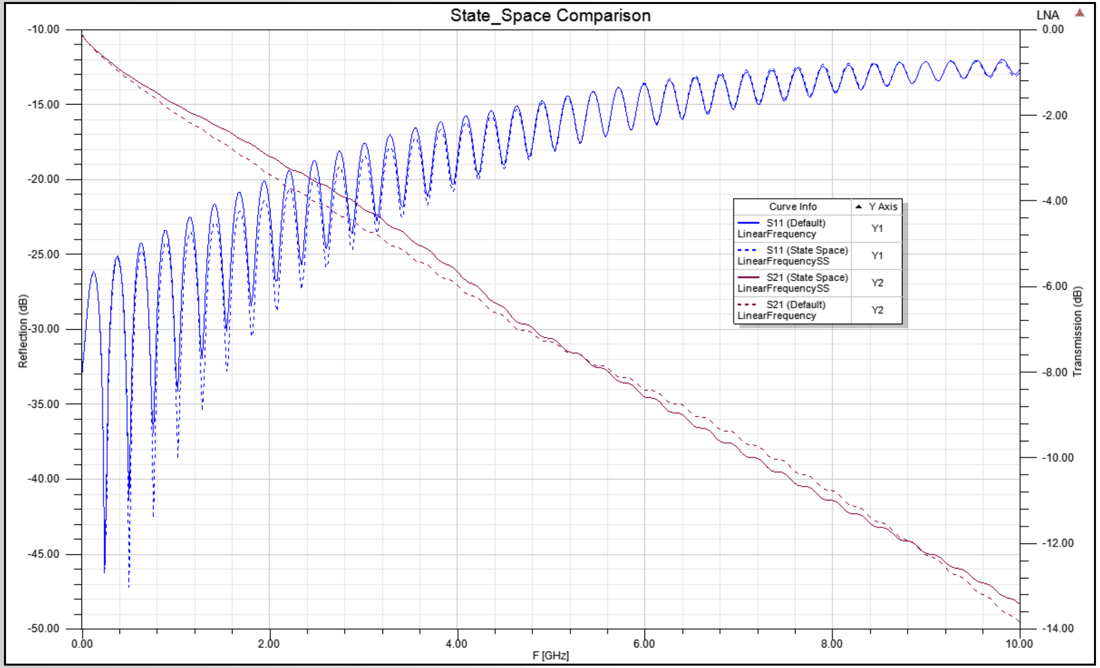
Attributes | Cartesian | General



  October 6, 2015

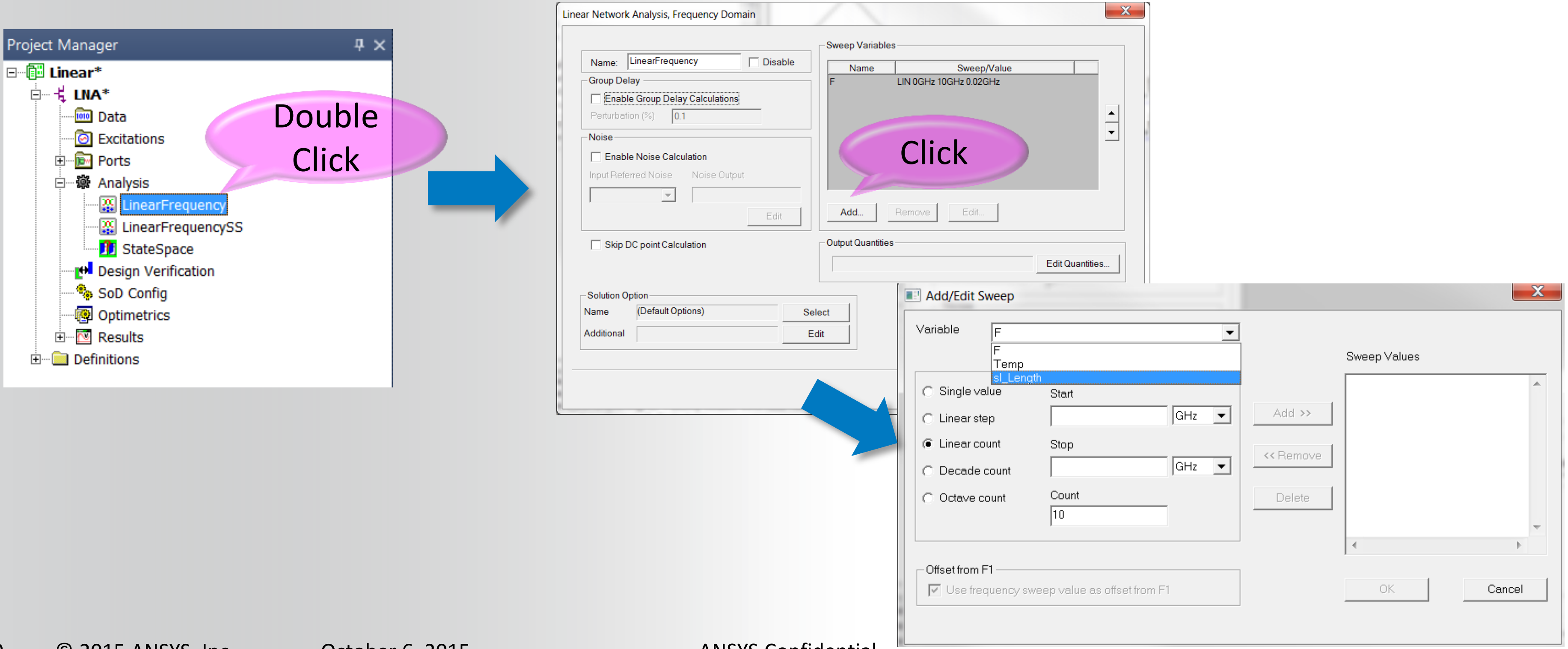
Linear Network Analysis

- **Displaying Data**
  - By selecting traces, axis labels and legends, the appearance of the plot can be customized as desired.



Linear Network Analysis

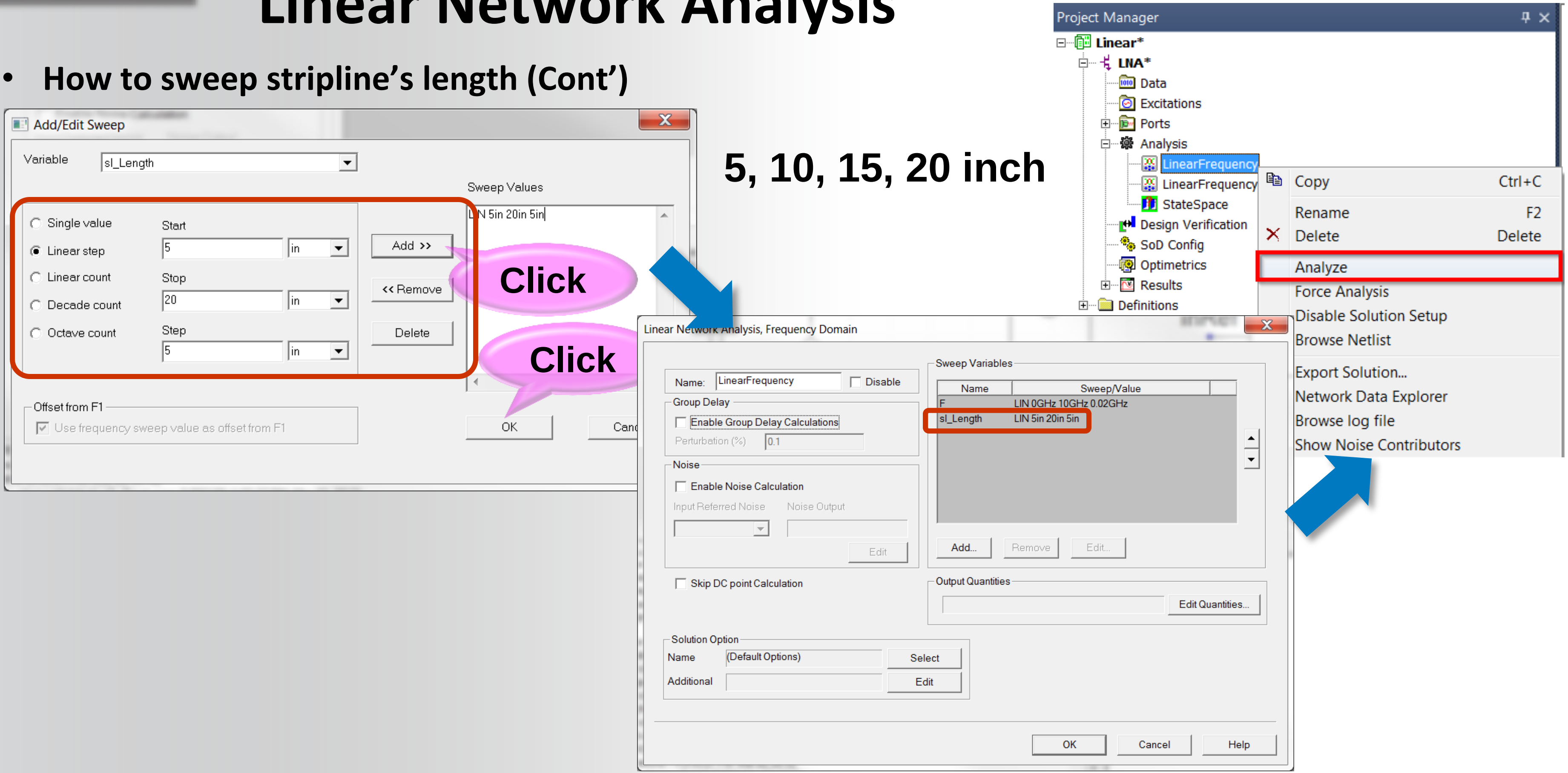
- **How to sweep stripline's length**
  - You already defined a local variable regarding stripline's length at WS1-20 slide.



Linear Network Analysis

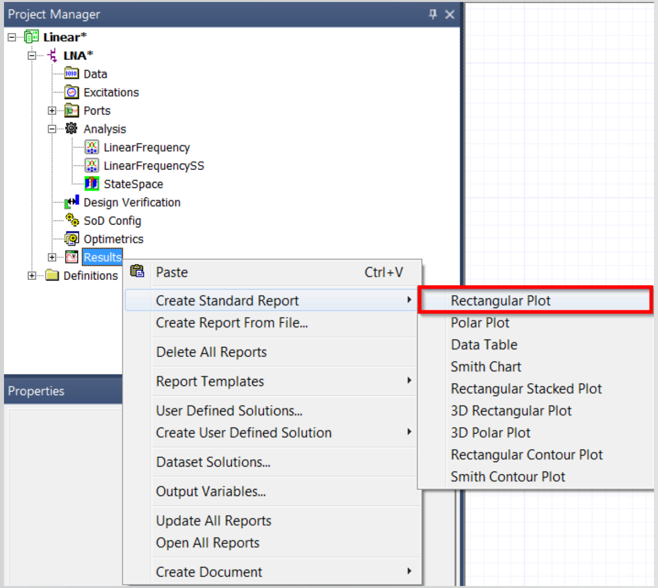
- **How to sweep stripline's length (Cont')**

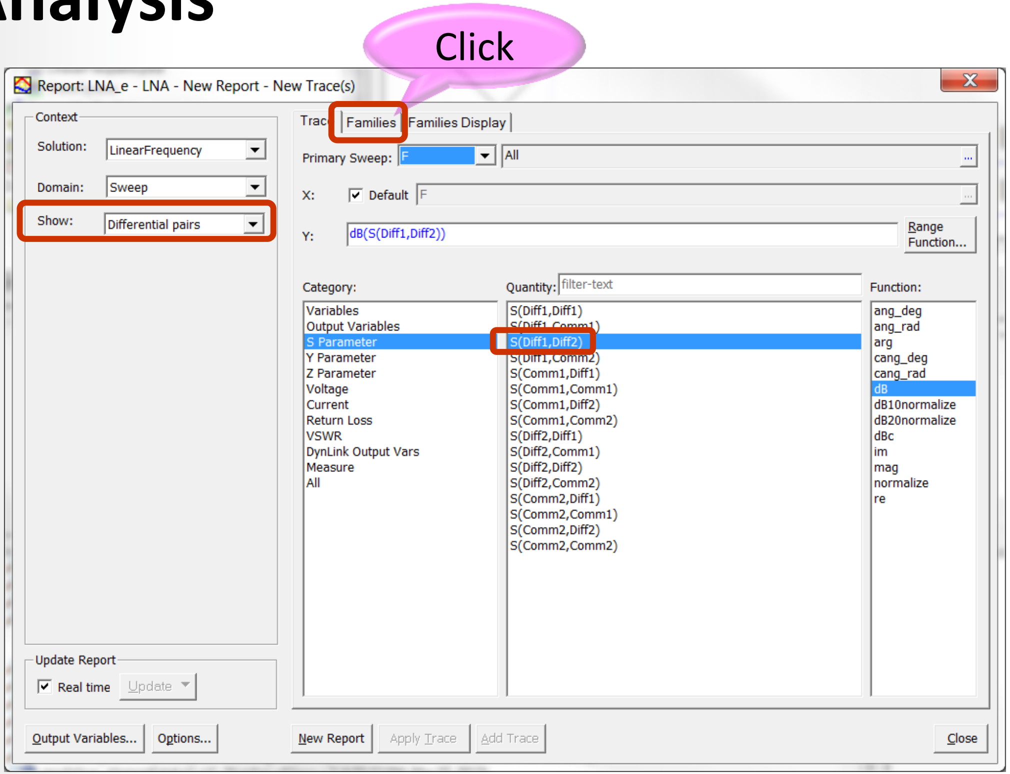
5, 10, 15, 20 inch



Linear Network Analysis

- **How to display the swept data**

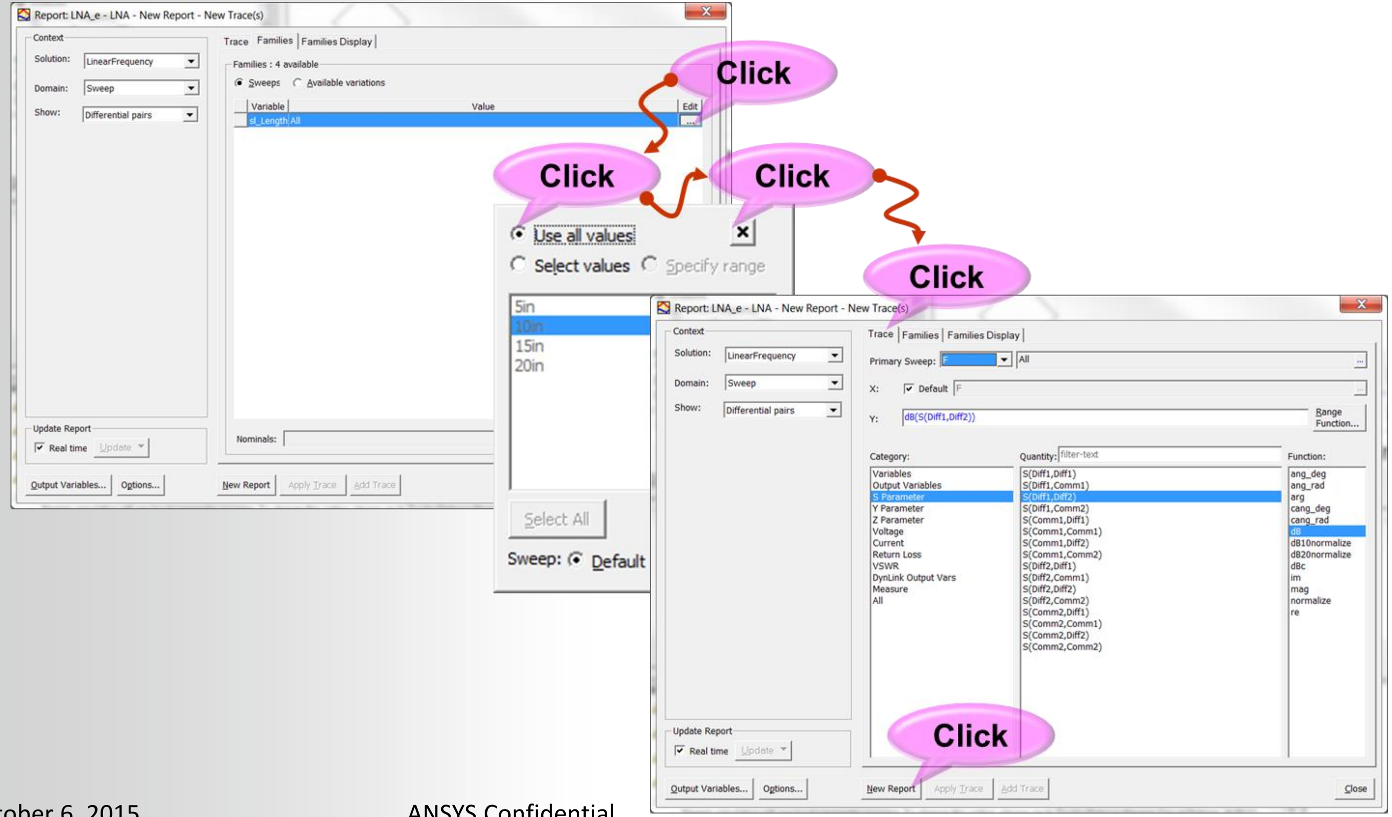




Display differential insertion loss depending on lengths.

Linear Network Analysis

- **How to display the swept data (Cont')**



Linear Network Analysis