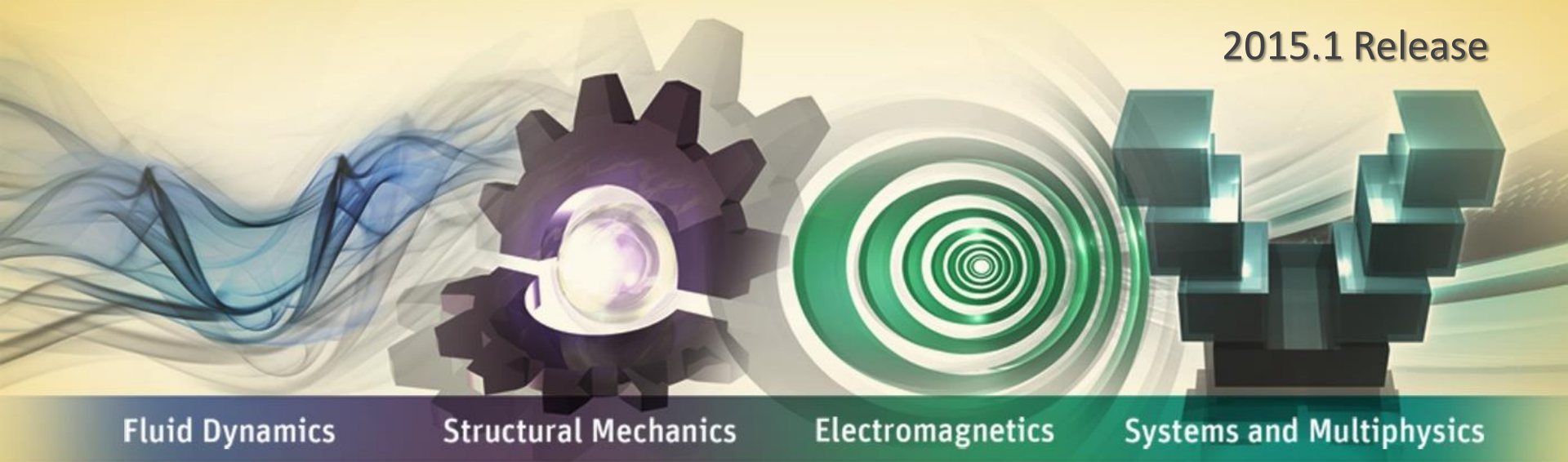


# Workshop 5: SSN

2015.1 Release



## Introduction to ANSYS SIwave

- **Simultaneous Switching Noise (SSN) Simulation**

- The example is intended to show you how to simulate SSN. Two signal nets will be driven and 1 signal net will be left quiet. We will look at the noise on the quiet signal net and power rail.
- You will learn how to:
  - Create ports using automatic port generation feature
  - Compute S-parameters
  - Compute FWS sub-circuit
  - Import IBIS model for time domain simulation in ANSYS Electronics Desktop
  - Simulate SSN with FWS sub-circuit using NEXXIM and ANSYS Electronics Desktop

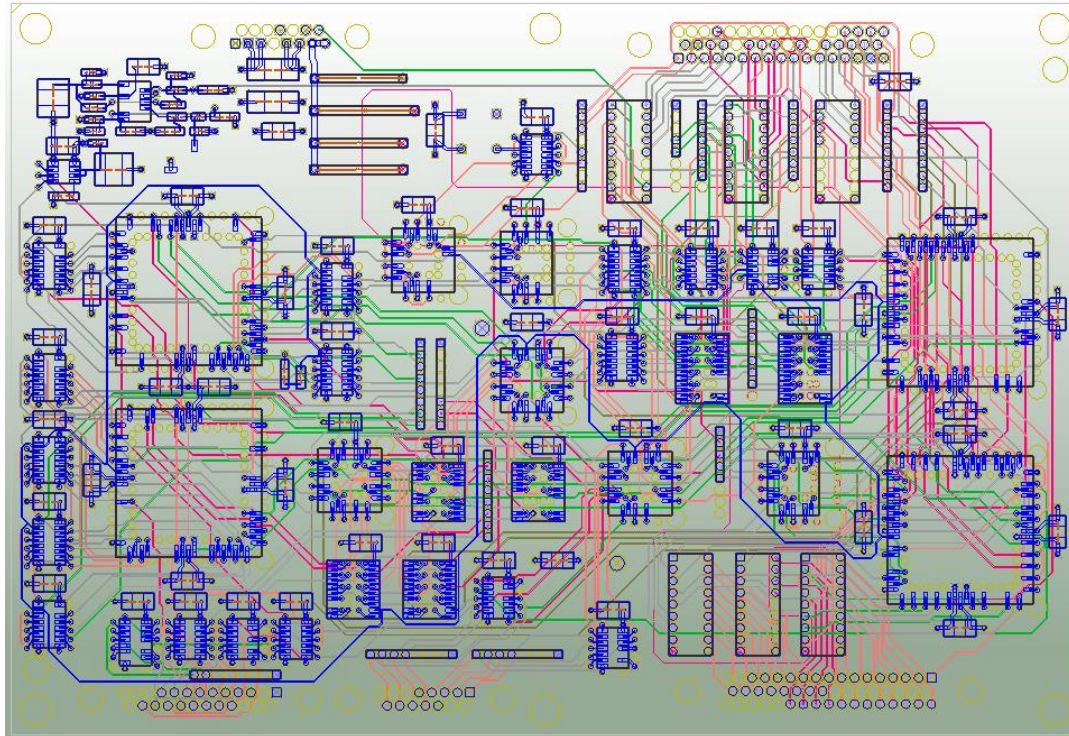
# Example – SSN on PCB

## • Starting SIwave

- To launch SIwave program, click the Microsoft **Start** Button and select: **All Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.1 > ANSYS SIwave 2015.1**

## • Open a SIwave Project

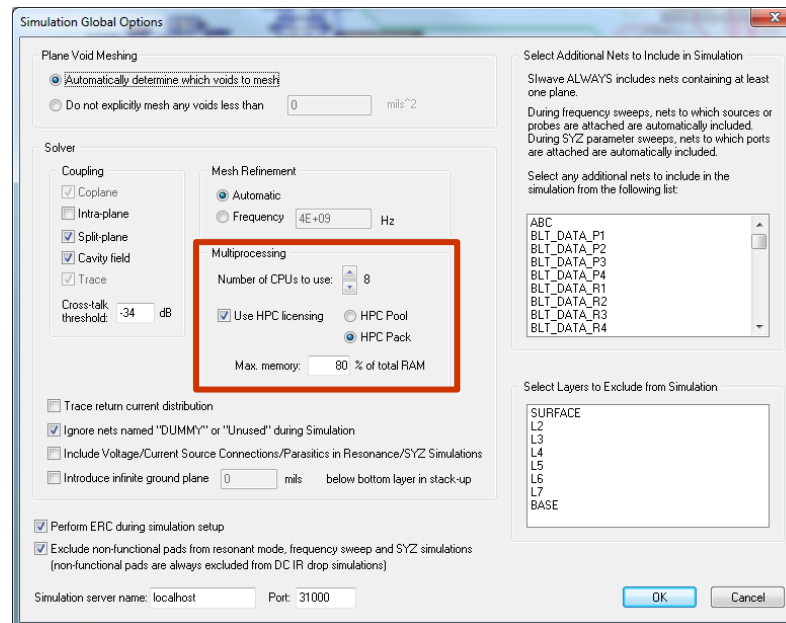
- Select the menu item **File > Open...**
  - Browse for file: **SIwave SSN Analysis.siw**,
  - Click the **Open** button



## Example – SSN on PCB

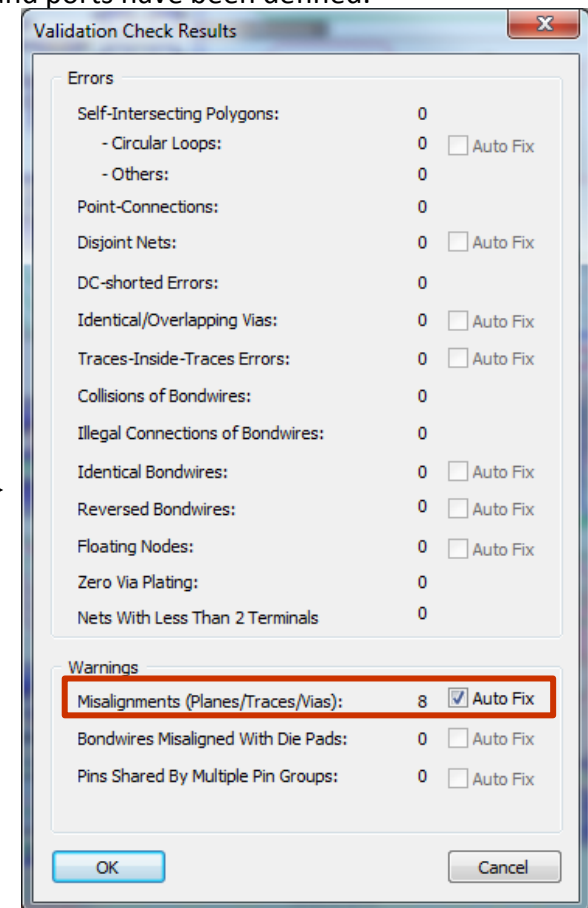
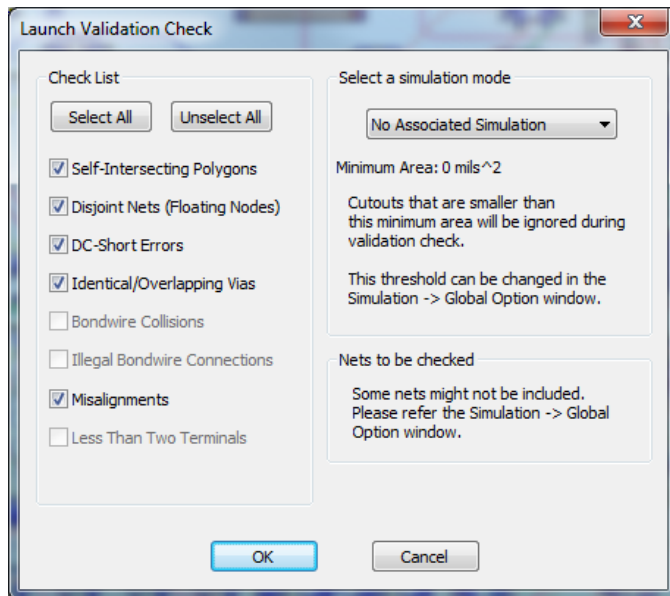
### • Setting Simulation Global Options

- Select the menu item **Simulation > SIwave > Options...**
  - Under the Plane Void Meshing: **Automatically determine which voids to mesh**
  - Under Mesh Refinement: **Automatic**
  - Ignore nets named “DUMMY” or “Unused” during simulation: ☒ **Checked**
  - Perform ERC during simulation setup: ☒ **Checked**
  - Exclude non-functional pads...: ☒ **Checked**
    - **Note:** These are the default simulation settings.
  - If you have an MP license for SIwave here set the number of CPU's to use to the available number of cores.
  - If you have an HPC Pool or HPC Pack enable these here and set the number of CPU/HPC Cores.
  - Click the **OK** button to exit and save these settings as the defaults.



## • Validation Check

- It is a good idea to do a validation check before you start working on any design in SIwave for the first time
- The Validation Check consists of self-intersecting polygons, disjoint nets, overlapping nets (dc-shortened) and nets with overlapping vias checks.
- The last thing you want to find out is that you need a new layout after all the settings and ports have been defined.
- To do a validation check:
  - Select the menu item **Tools > Validation Check**
    - Click the **OK** button to start the validation
  - There are no layout and DRC related problem with this design.
  - Notice that there are 8 misalignments warnings that will be Auto Fixed.
  - Click the **OK** button to exit the Validation Check dialog.

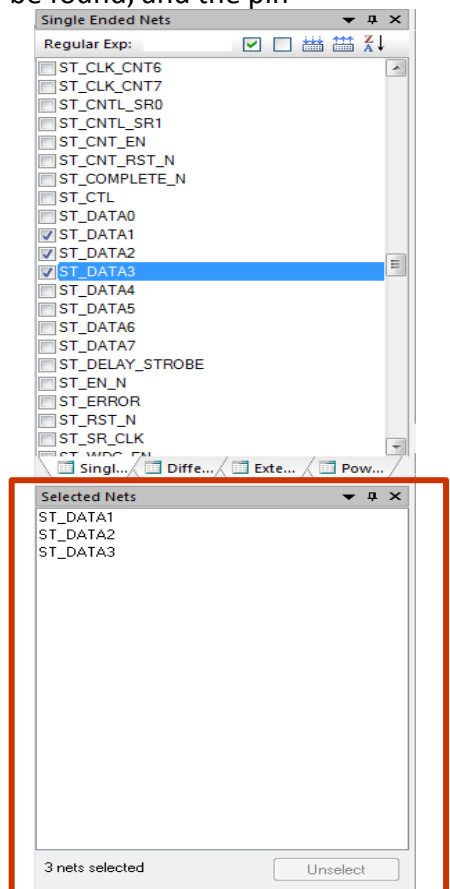
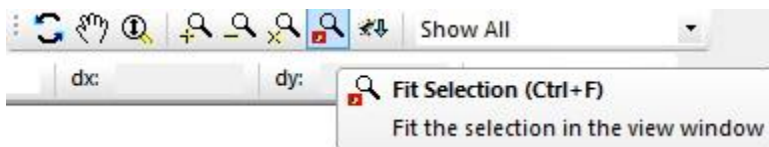


## • Searching for Component and Pin Names

- The goal of this example is to drive nets **ST\_DATA1** and **ST\_DATA2** with a stimulus, then observe noise on net **ST\_DATA3** and **VCC**, with respect to **GND**.
- Ports are needed on **ST\_DATA1**, **ST\_DATA2**, **ST\_DATA3**, and **VCC**, with **GND** as the negative reference net.
- In order to make port creation easier, the components that are connected to these nets need to be found, and the pin numbers/names associated with these nets need to be found. .
- Using the **Single Ended Nets** tab in the Workspace, select the following nets
  - **ST\_DATA1**: ☒ Checked
  - **ST\_DATA2**: ☒ Checked
  - **ST\_DATA3**: ☒ Checked

**Note:** A new workspace has been added to SIwave 16.x to show which nets are selected.

- Select the toolbar button **Fit Selection**, or **Ctrl+F**.

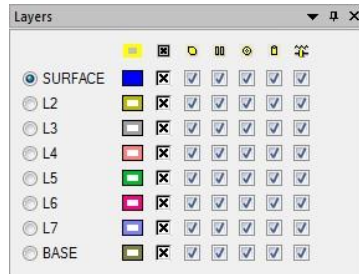




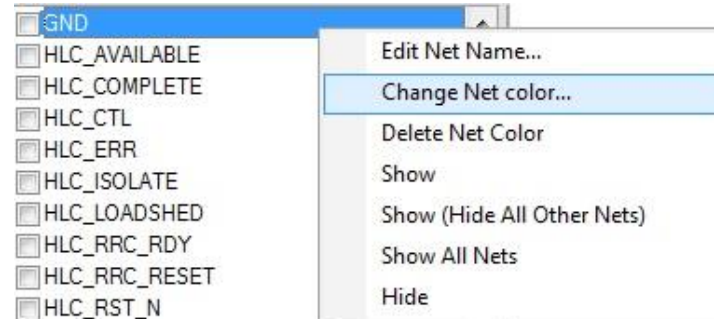
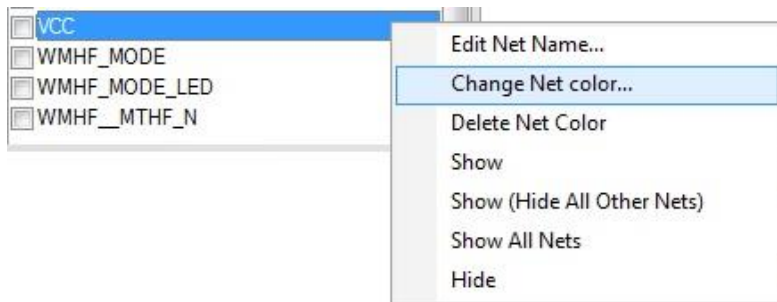
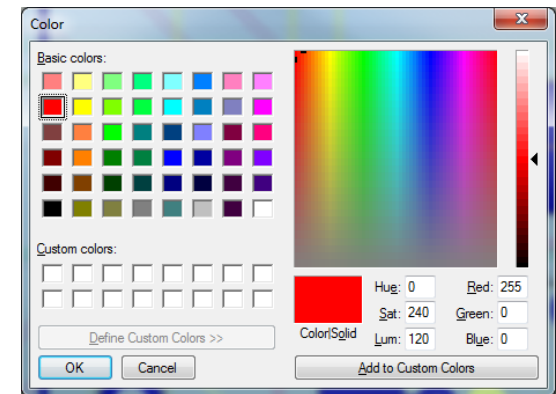
# Example – SSN on PCB

## • Searching for Component and Pin Names - continued

- Change the top layer from wire frame to solid color display
  - In the **Layers** workspace click on the color box net to **SURFACE**, to fill in the geometry on this layer.



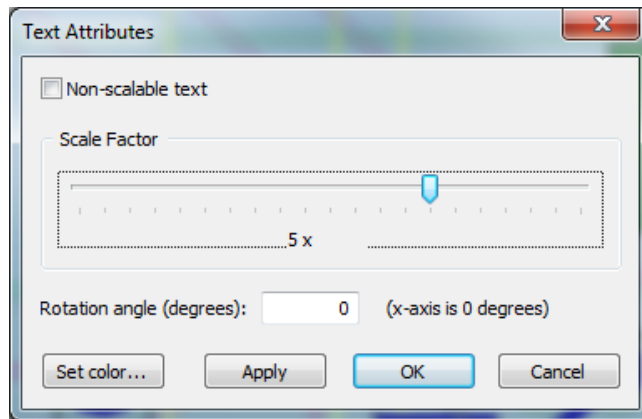
- Select the menu item **Visibility > Circuit Elements > Pin Names > On**
- From the **Single Ended Nets** Workspace
  - Highlight net **VCC**
  - Right mouse click and select **Change Net Color**
    - Click on a **red** color from the color template, and click the **OK** button
  - Highlight net **GND**
  - Right mouse click and select **Change Net Color**
    - Click on a **green** color from the color template, and click the **OK** button



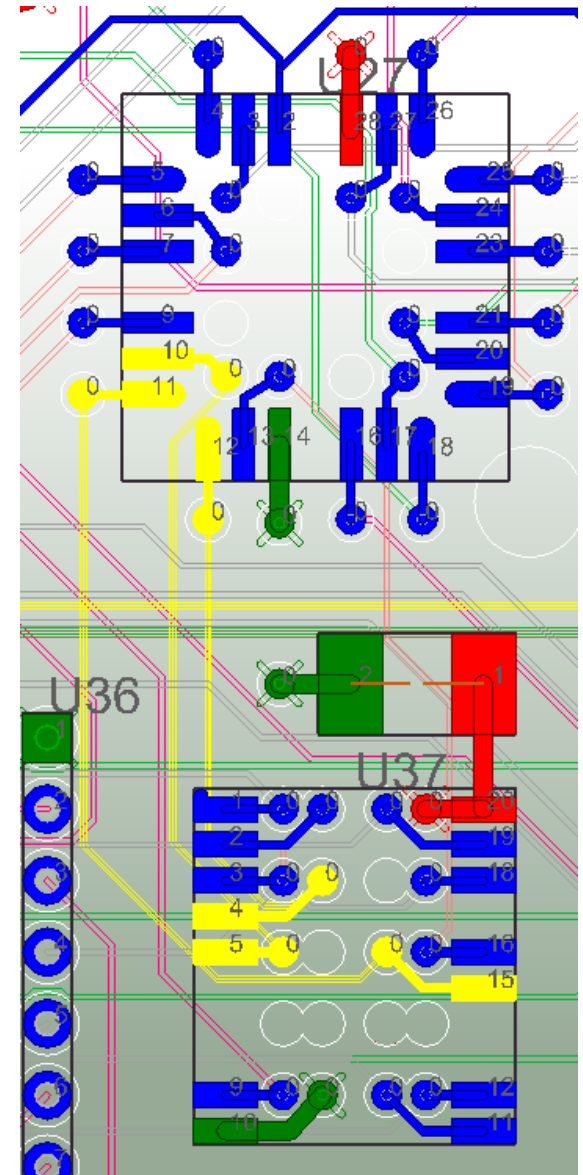
# Example – SSN on PCB

## • Searching for Component and Pin Names – continued

- If the Text Attributes for the pin names are too small to read select:
  - View > Modify Attributes > Change text Attributes...**
  - Change the slider to 5x and click **Apply** then **OK**.



- The 3 signal nets are highlighted in yellow
- VCC** pins are in red
- GND** pins are in green
- The 3 signal nets are connected to component **U27** and **U37**
  - ST\_DATA1** is connected to pins **U27-12** and **U37-4**
  - ST\_DATA2** is connected to pins **U27-11** and **U37-15**
  - ST\_DATA3** is connected to pins **U27-10** and **U37-5**
- VCC** pins are **U27-28** and **U37-20**
- GND** pins are **U27-14** and **U37-10**

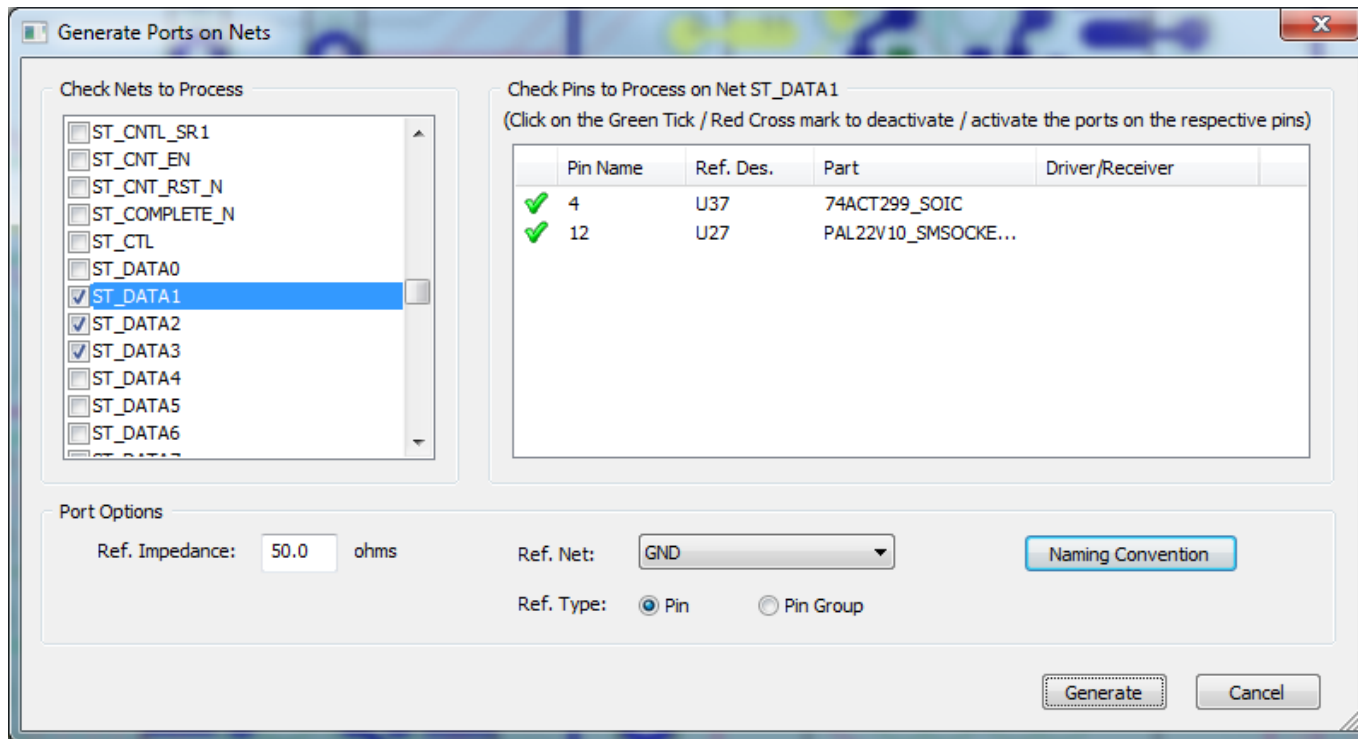




# Example – SSN on PCB

- **Ports Generation**

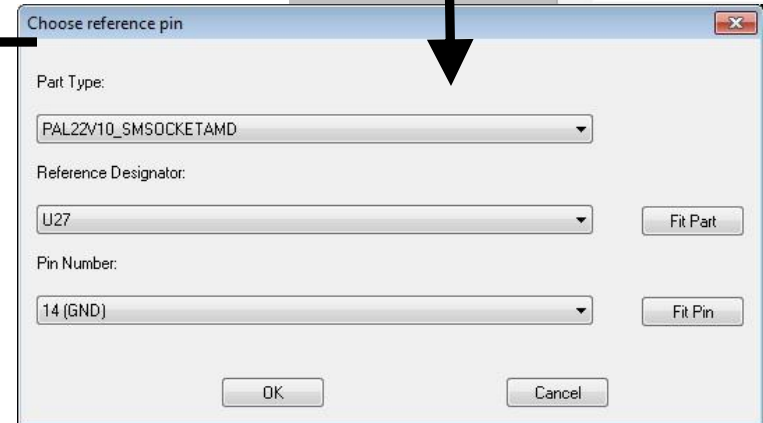
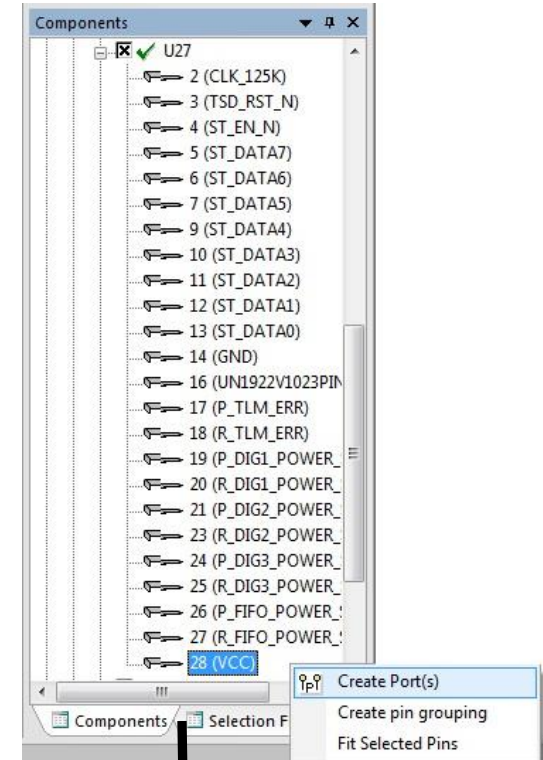
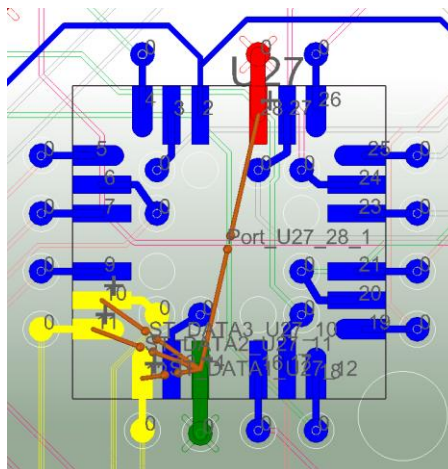
- To create automatic ports on the selected nets
  - Click **Tools > Generate Port on Selected Nets...**
  - Check **ST\_DATA1**
  - Check **ST\_DATA2**
  - Check **ST\_DATA3**
  - Click **Generate**



# Example – SSN on PCB

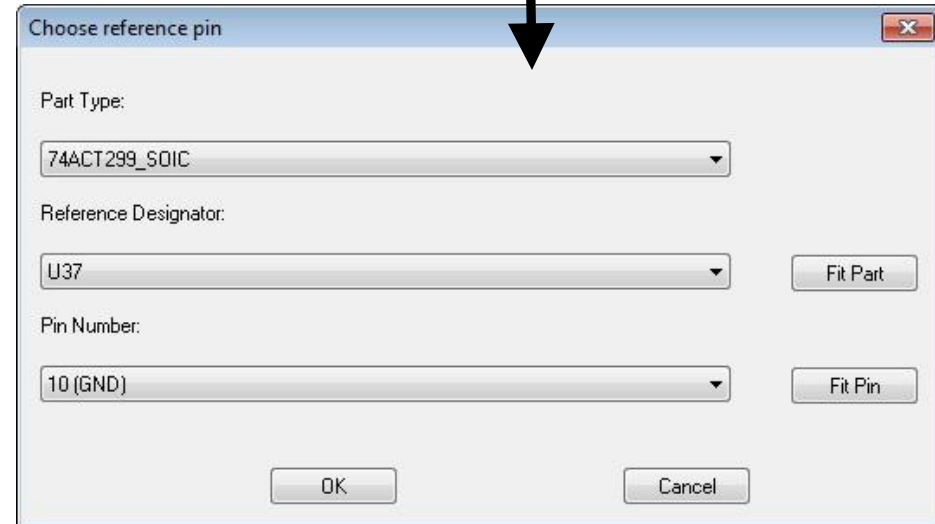
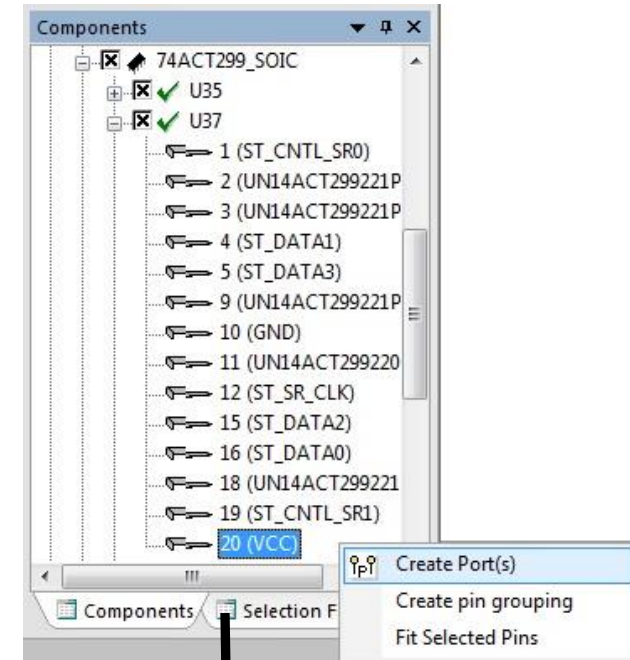
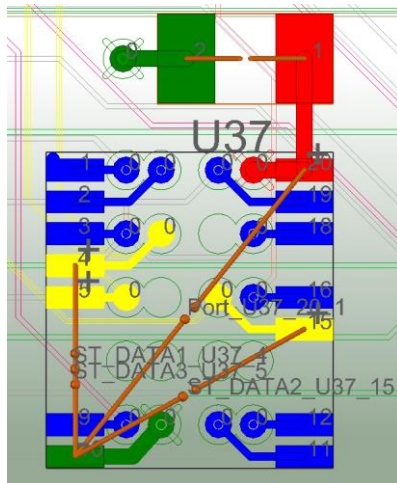
## • Ports Generation – continued

- Click on the **Components** tab in the sidebar workspace.
  - Expand **Integrated Circuits**
  - Expand **PAL22V10\_SMSOCKETAMD**
  - Expand **U27**
  - Select pin **28**
  - Right mouse click on pin name 28, select **Create Port(s)**
  - In the Choose reference pin window:
    - Part Type: **PAL22V10\_SMSOCKETAMD**
    - Reference Designator: **U27**
    - Select **14 (GND)** for the pin number
    - Click the **OK** button
  - As seen in the display area a port has been added to component U27.



## Ports Generation – continued

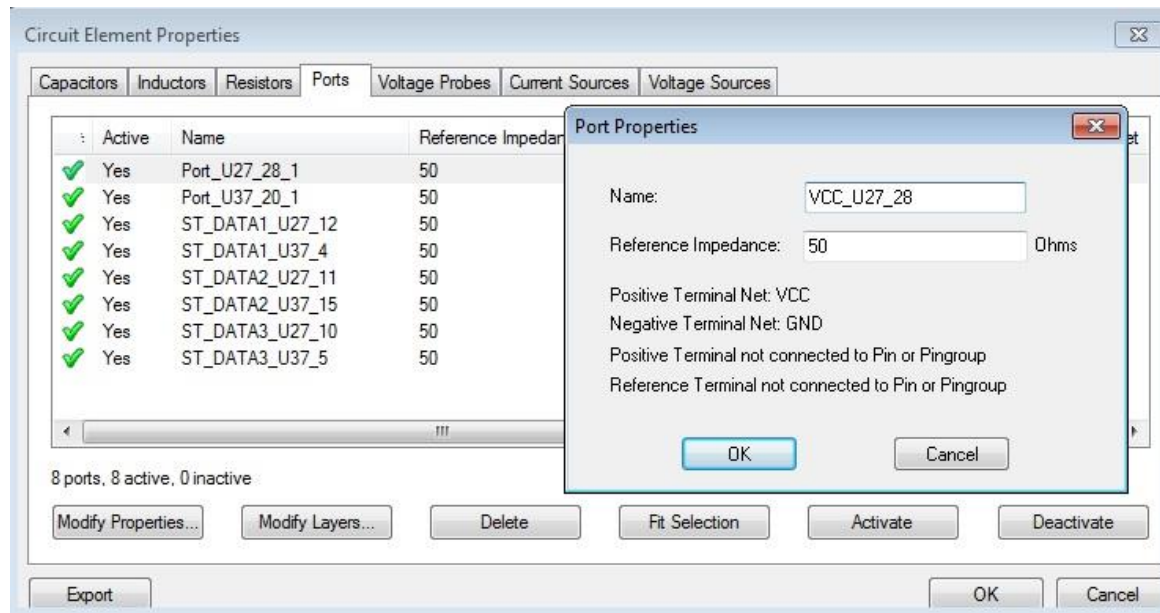
- Click on the **Circuit Elements** tab in the sidebar workspace.
  - Expand **Integrated Circuits**
  - Expand **74ACT299\_SOIC**
  - Expand **U37**
  - Select pin **20 (VCC)**
  - Right mouse click on pin name 20, select **Create Port(s)**
  - In the Choose reference pin window:
    - Part Type: **74ACT299\_SOIC**
    - Reference Designator: **U37**
    - Select **10 (GND)** for the pin number
    - Click the **OK** button
  - As seen in the display area a port has been added to component U37



## Example – SSN on PCB

- **Ports Generation – continued**

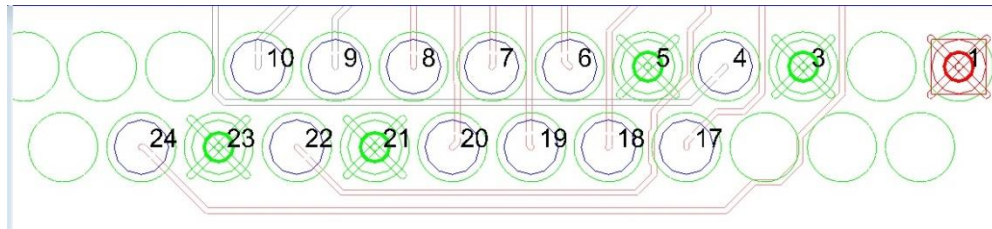
- Change the name of the ports
  - Select the menu item **Home > Circuit Element Parameters...**
  - Click tab for **Ports**
  - Highlight **Port\_U27\_28\_1**
  - Click **Modify Properties**
    - Change name to **VCC\_U27\_28**
    - Click **OK**
  - Highlight **Port\_U37\_20\_1**
  - Click **Modify Properties**
    - Change name to **VCC\_U37\_20**
    - Click **OK**
- Click **OK**.



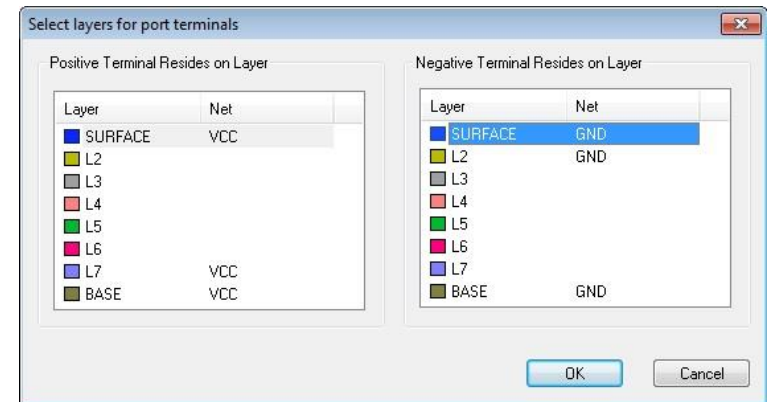
# Example – SSN on PCB

## • Create Port for VRM

- A port for the external power supply will be created at the edge of the PCB.
- Zoom in to the bottom left corner of the design. The port will be manually created on the connector J3 between pins 1 and 3

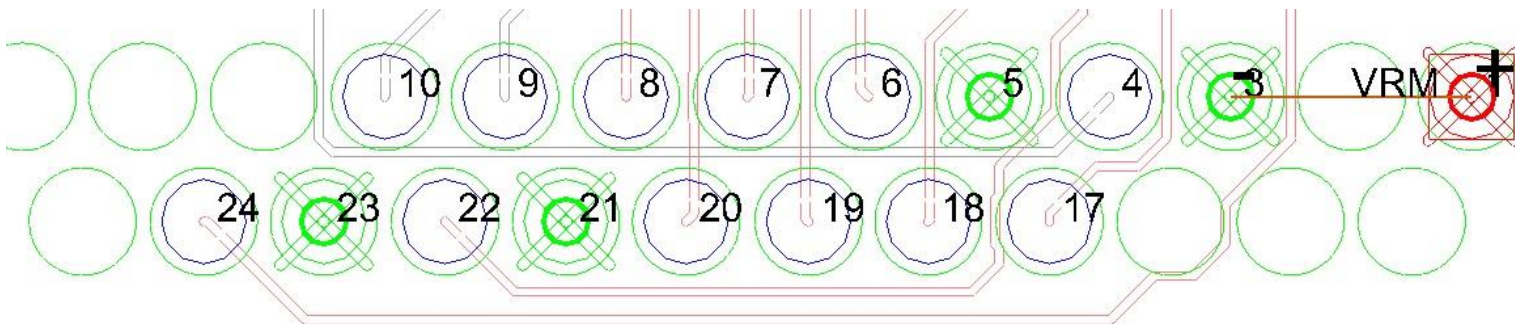
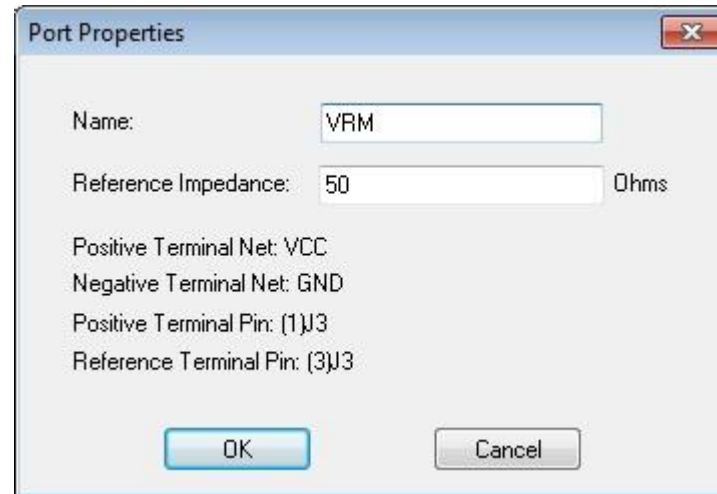


- Alternately expand the **Discrete Devices** in the **Components** tab of the sidebar workspace and expand the **JCONNECTOR\_DB2590F-1** component.
  - Right Mouse Click on **J3** and select **Fit Discrete Device**
- Select the menu item **Circuit Elements > Add Port**
  - Place the cursor on top of **pin 1** of connector **J3**, do a left mouse click when round bull's eye mouse cursor appears
  - Place mouse cursor on top of **pin 3** of connector **J3**, do a left mouse click when round bull's eye mouse cursor appears
  - In the **Select layers** for port terminals window:
    - Positive Terminal Resides on Layer: **SURFACE VCC**
    - Negative Terminal Resides on Layer: **SURFACE GND**
    - Click the **OK** button



- **Create Port for VRM – continued**

- In the **Port Properties** window:
  - Name: **VRM**
  - Reference Impedance: **50 Ohm**
  - Click the **OK** button



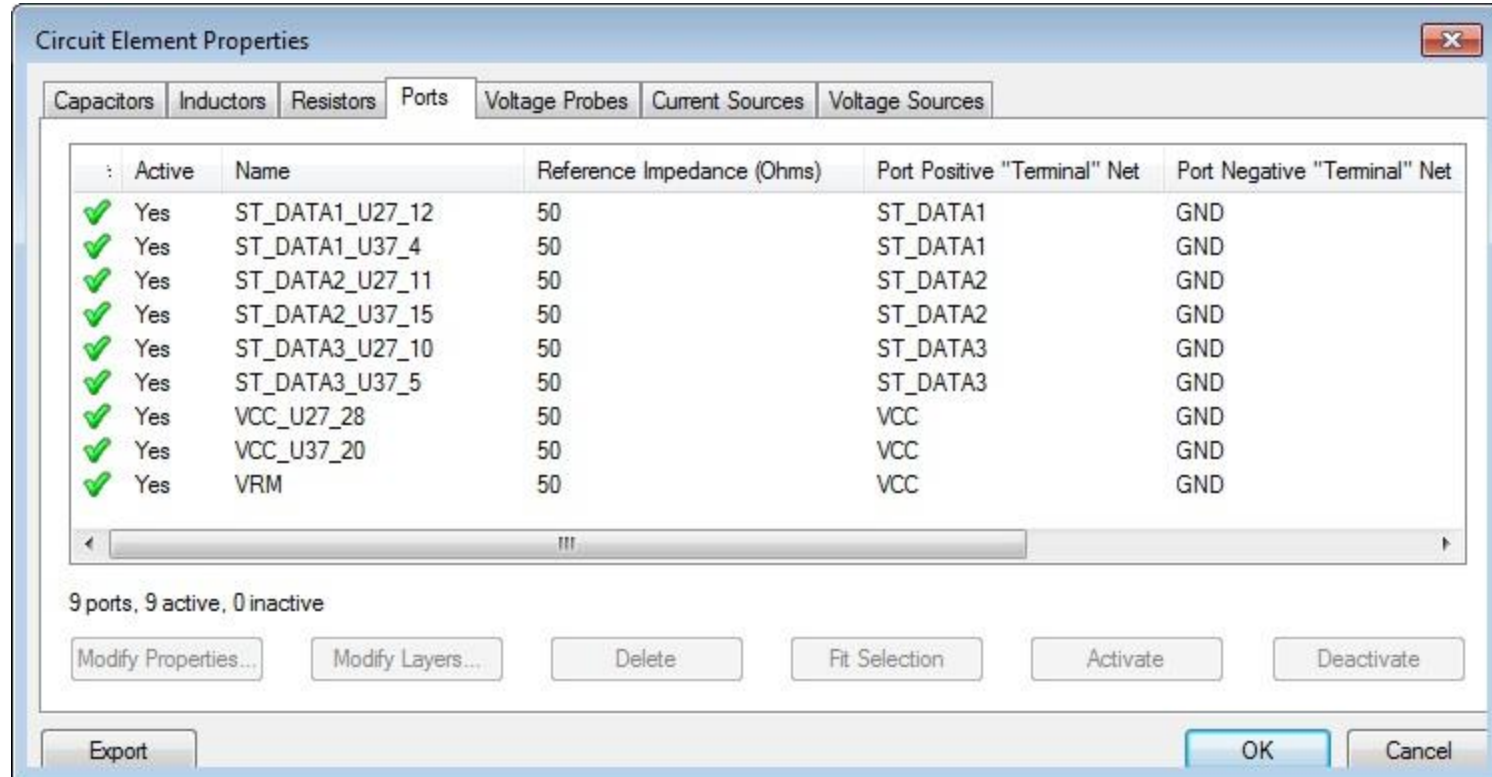
- To exit from Port creation mode Select the menu item **Circuit Elements > Port**



## Example – SSN on PCB

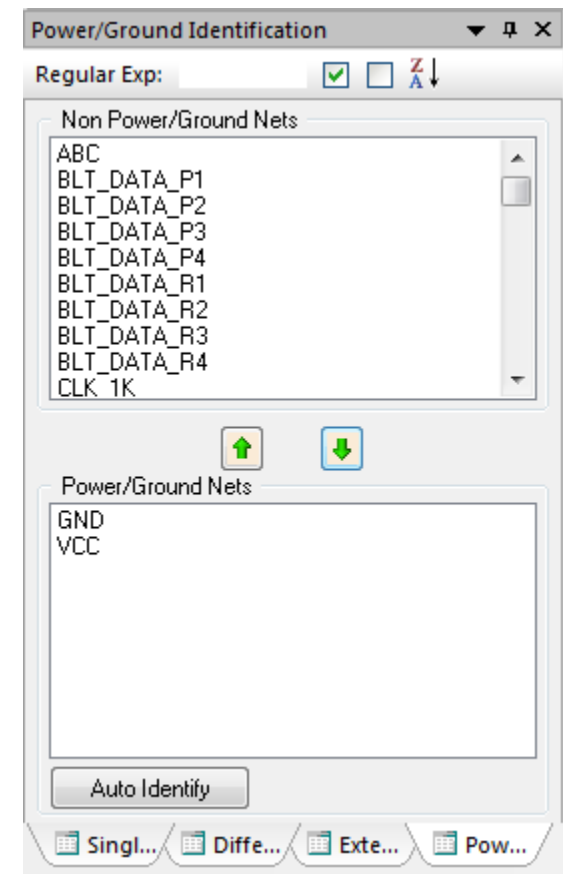
- **Verify Ports**

- Select the menu item **Home > Circuit Element Parameters**
  - Select the **Ports** tab
    - Click the column Port Positive “Terminal” Net twice to sort the column by net name beginning with ST\_DATA1
    - Verify that all 9 ports have the right reference impedance, port positive and negative nets
    - Click the **OK** button



- **Define Power/Ground Nets**

- The project in SIwave does not have the Power and Ground nets defined. To provide optimal simulation run time and accuracy the power and ground nets need to be configured.
- Navigate to the **Power/Ground Identification** workspace in the side bars.
  - Scroll through the list of nets and find the net **GND** and select it by clicking on it.
  - Press the green down arrow to define the **GND** net as a power/ground net.
  - Repeat the process for the net labeled **VCC**.



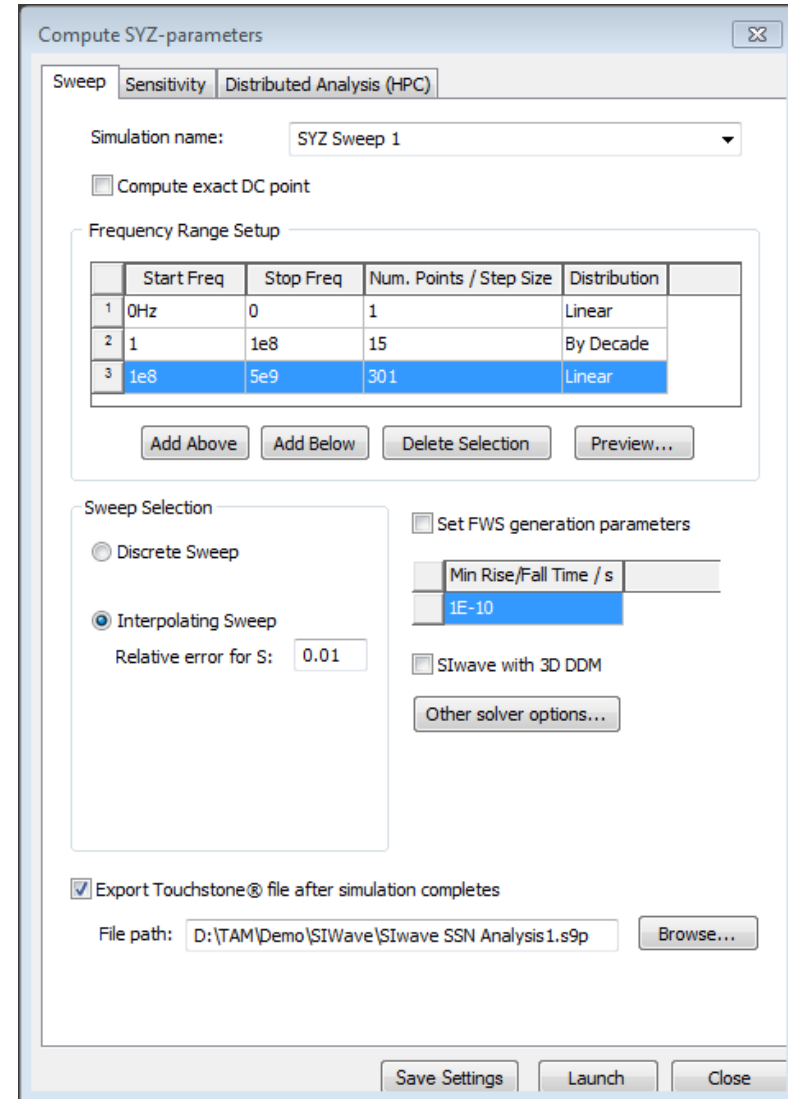
# Example – SSN on PCB

## • Save SIwave Project

- Select the menu item **File > Save As**
  - File name: **ssn**
  - Click the **Save** button

## • Computing S-Parameters

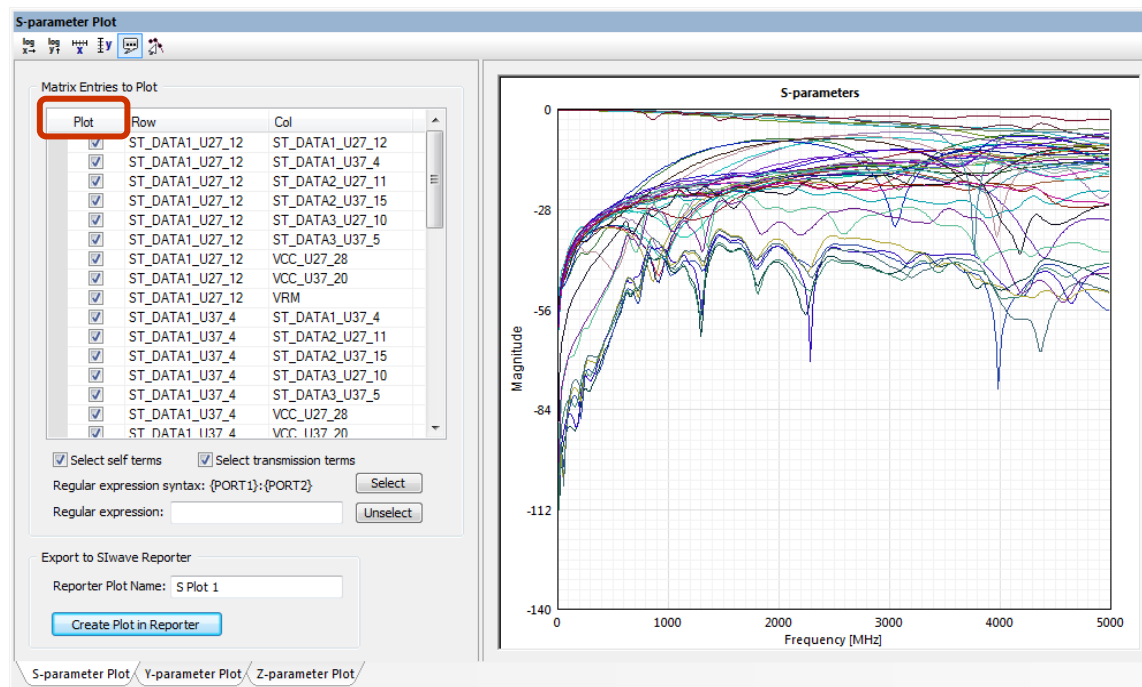
- For Full Wave Spice export, we would run the frequency sweep up to the knee frequency (Fknee), where  $F_{knee} \approx 0.5/\text{rise time}$ .
  - For a 100 ps rising edge source, the knee frequency is 5 GHz
- Select the menu item: **Simulation > Compute SYZ Parameters (SIwave)**
- Change the settings to as shown
  - To Add additional frequency range click on **Add Below**
  - Click the **Launch** button



## Example – SSN on PCB

- **Open Report Editor**

- Select the menu item **Results > SYZ (SIwave) > SYZ Sweep 1 > Plot Magnitude/Phase...**
- Click the plot column header to enable the display of all curves
- Click **Create Plot in Electronics Desktop**
- When done viewing the S-Parameters, select the menu item **File > Exit**
  - Click the **Yes** button if asked to save the changes
- Click the **Close** button to close the Plot Generation Dialog



- **Exit SIwave**

- Select the menu item **File > Exit** and click **Yes** to save the changes.

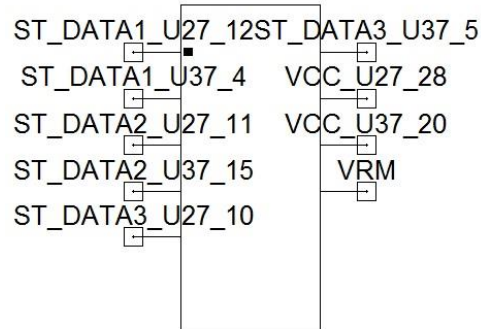
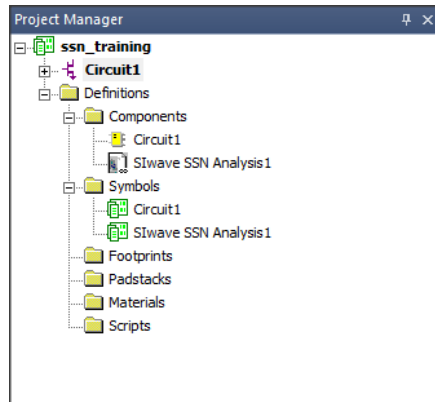
- **Getting Started**

- For this exercise, the direct link between ANSYS Electronics Desktop and SIwave will be used to import a circuit model to ANSYS Electronics Desktop . Simulation results from SIwave can also be exported in “Full Wave SPICE™” format or as raw S-parameter data in Touchstone® format for use in other 3rd party circuit simulation tools.
- Launch ANSYS Electronics Desktop, Click the Microsoft Start Button and select: **All Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.1 > ANSYS Electronics Desktop 2015.1**
  - After launching the Desktop, a new project is automatically created. Save this project to the directory where the SIwave project **ssn.siw** is located.
    - From the menu select **File > Save as**
      - Change to the directory that the ssn.siw file is stored in
      - Filename: **ssn\_training**
      - Click the **Save** button
    - Place the file **proibis.ibs** in the same directory as the ANSYS Electronics Desktop project ssn\_training.aedt. This file contains IBIS models that will be used as drivers and receivers for the SSN analysis. This file is available from ANSYS technical support personnel.
- **Create the circuit schematic**
  - Insert a new circuit schematic by selecting the menu item **Project > Insert Circuit Design**
    - Select **None** when asked to choose a Layout Technology.

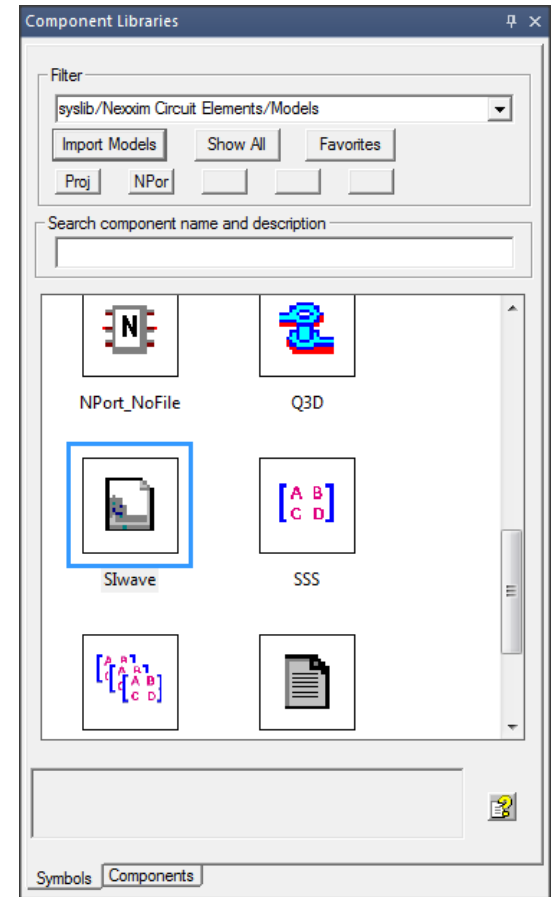
# Example – SSN on PCB

## • Add the Siwave Model

- Under the Components Libraries workspace, click **Import Models**
  - File name: **Siwave SSN Analysis.siw**
  - Click **Open**
  - Place the model on the schematic
    - Left Click to drop the symbol on the schematic
    - Click **ESC** to exit placement mode
- Expand the project tree Definitions to show Models



**Siwave dynamically linked model  
in the schematic**

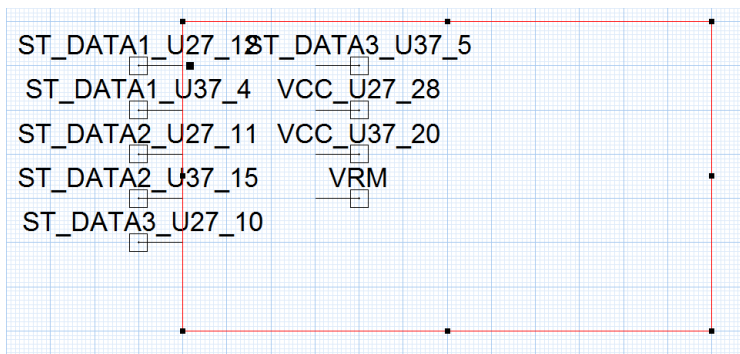




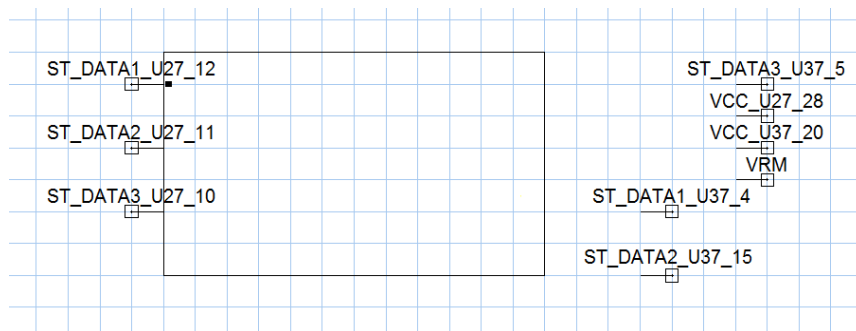
## Example – SSN on PCB

### • Modify siwave\_board symbol

- When the siwave\_board symbol was created during the import the signals were distributed onto the symbol in an unorganized way. The following steps will allow for a cleaner more organized schematic.
- In the **Project Manager** box expand the **Symbols** and double click on the **Siwave SSN Analysis 1**
- First select the symbol's box by clicking on it and selecting the middle right handle
  - Stretch the box to be 12 grid units wide.



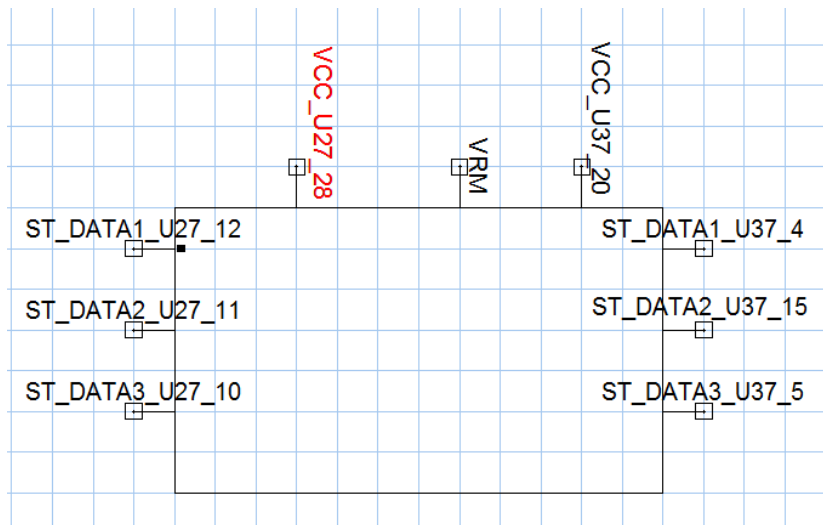
- Next the pins will be rearranged to create a better schematic flow.
- Select pins **ST\_DATA1\_U37\_4**, **ST\_DATA1\_U37\_15** and RMB and select **Flip Horizontal**



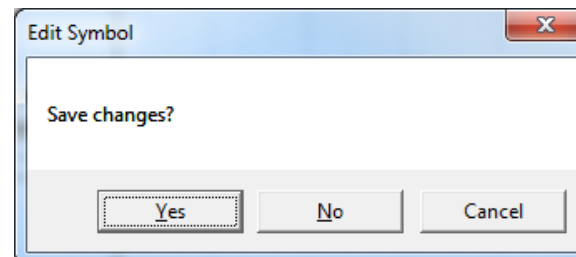
## Example – SSN on PCB

- **Modify siwave\_board symbol (Cont.)**

- Select the following pins (use Ctrl to select multiple at once):
  - VRM, VCC\_U37\_20, VCC\_U27\_28
- **RMB** and select **Rotate** (Alternately press **Ctrl-R**)
- Now drag the pins into place to look like the symbol below.



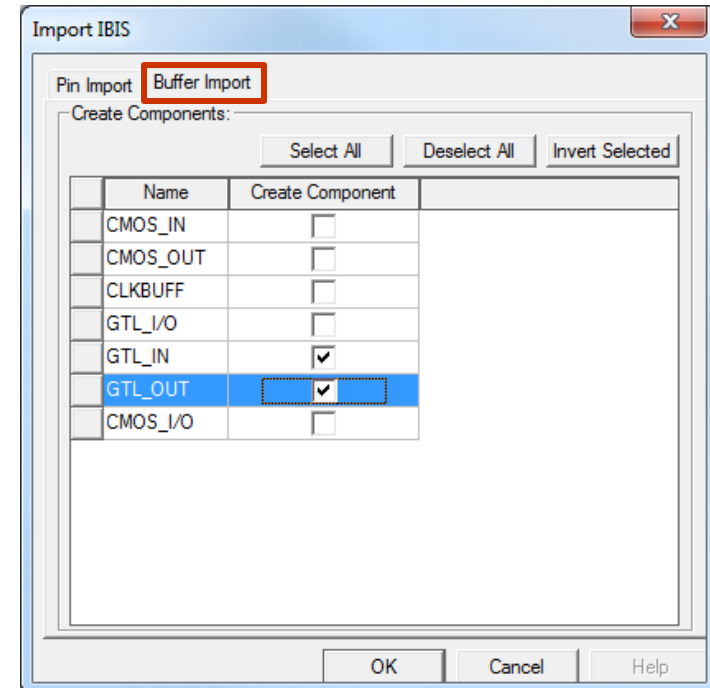
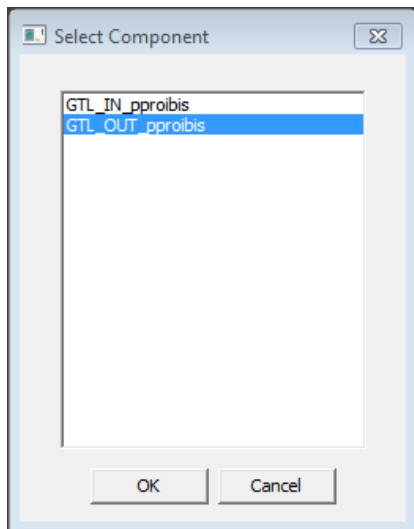
- Press the small 'x' in the upper right corner under the main program 'x' to close the symbol.
- When prompted to save the changes click **Yes**



## Example – SSN on PCB

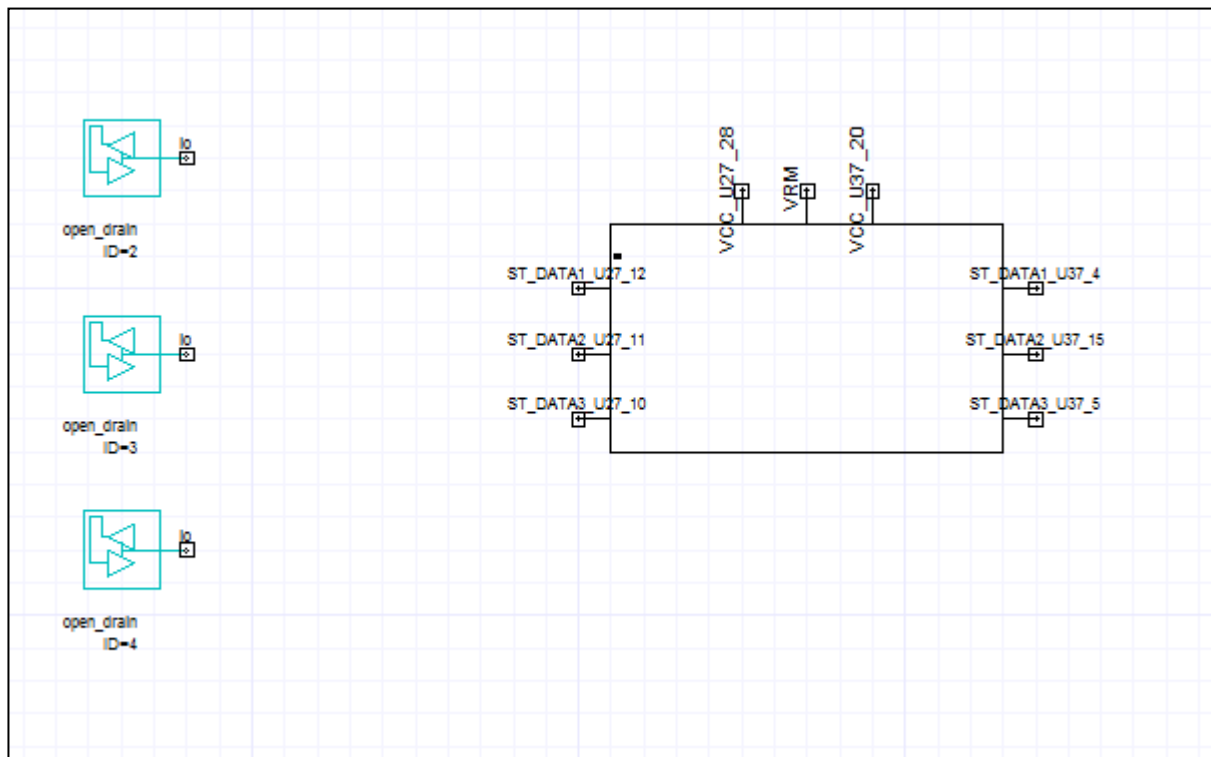
- **Import IBIS model**

- Under the Components Libraries workspace, click **Import Models**
  - Filename: **pproibis.ibs**
  - Click the **Open** button
- Import IBIS
  - In the **Buffer Import** tab
  - Click on **Deselect All**
  - Choose the components **GTL\_IN** and **GTL\_OUT**
  - Click the **OK** button
  - Highlight **GTL\_OUT\_pproibis**
  - Click OK to exit the Select Component window



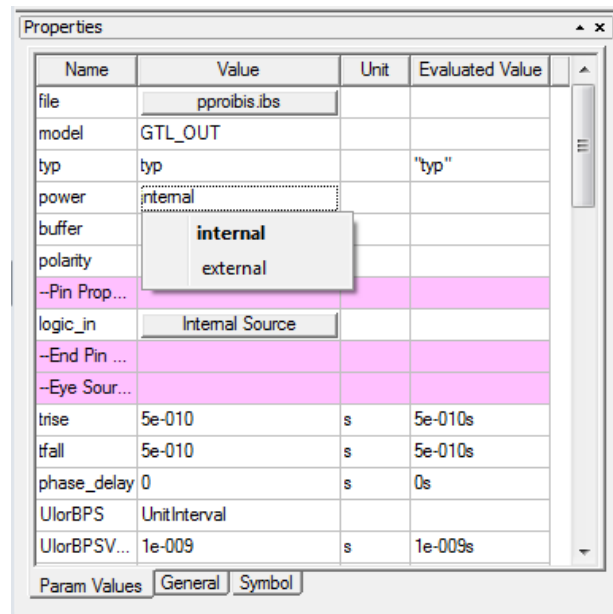
## Example – SSN on PCB

- **Placing the components**
  - Place the component 3 times to the left of the board symbol
  - Press the **ESC** key to cancel component placement
  - Select all three components and **RMB** and **Flip Horizontal**



## • IBIS Driver Settings

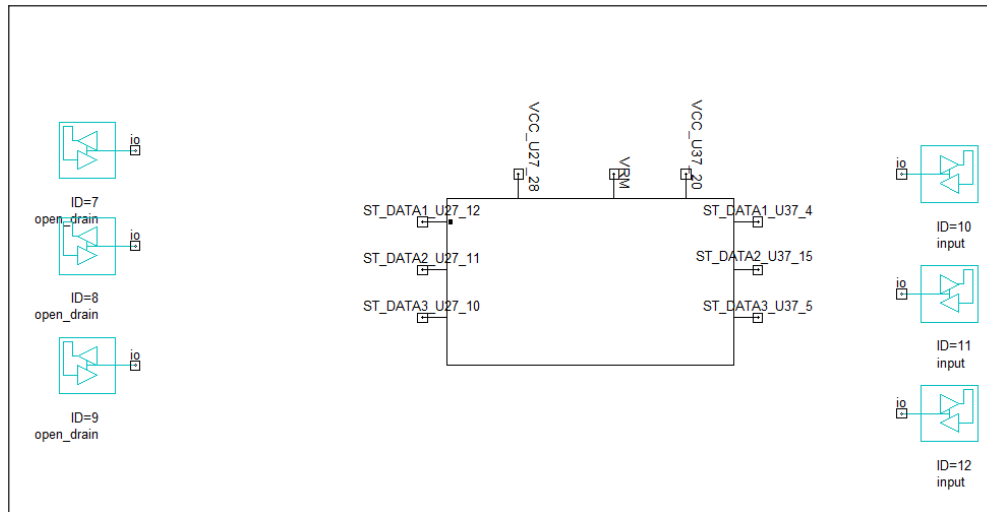
- The effect of non-ideal ground and supply nets is an important aspect of signal integrity. Hence, the IBIS drivers should draw current through the PCB and the internal power supply of the components will be turned off.
- Select all three **GTL\_OUT** instances in the schematic.
  - The properties window displays all common properties of the selected components.  
**Note** that multiple components can be selected and modified simultaneously.
  - Change the power property of the drivers to **external**. This causes the internal power source of the IBIS drivers to be turned off, requiring the connection of an external source.



# Example – SSN on PCB

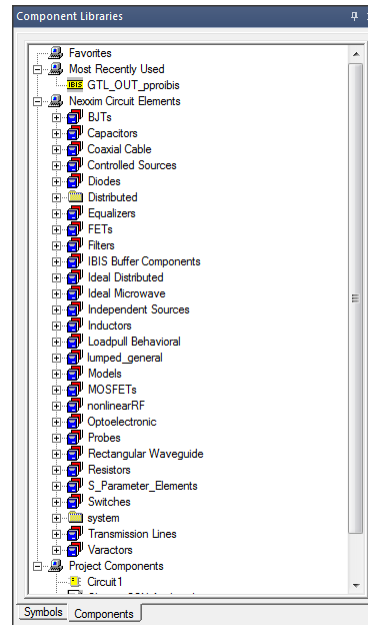
- **Placing the components**

- In the Project Manager workspace
  - Left Mouse click **GTL\_IN\_pproibis** under Components
  - Drag and drop this component onto the schematic, to the right of the SIwave model
  - Place a total of 3 components
- Change the power property of all 3 **GTL\_IN\_pproibis** to **external**. This causes the internal power source of the IBIS drivers to be turned off, requiring the connection of an external source






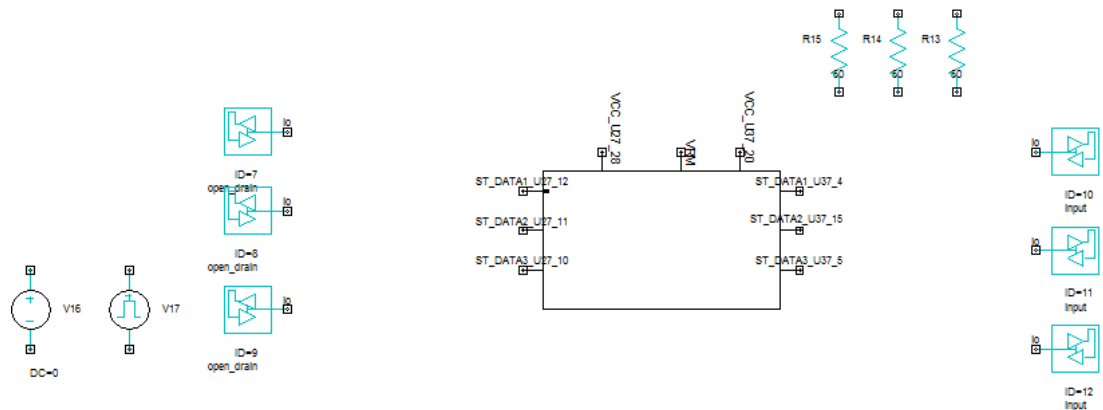
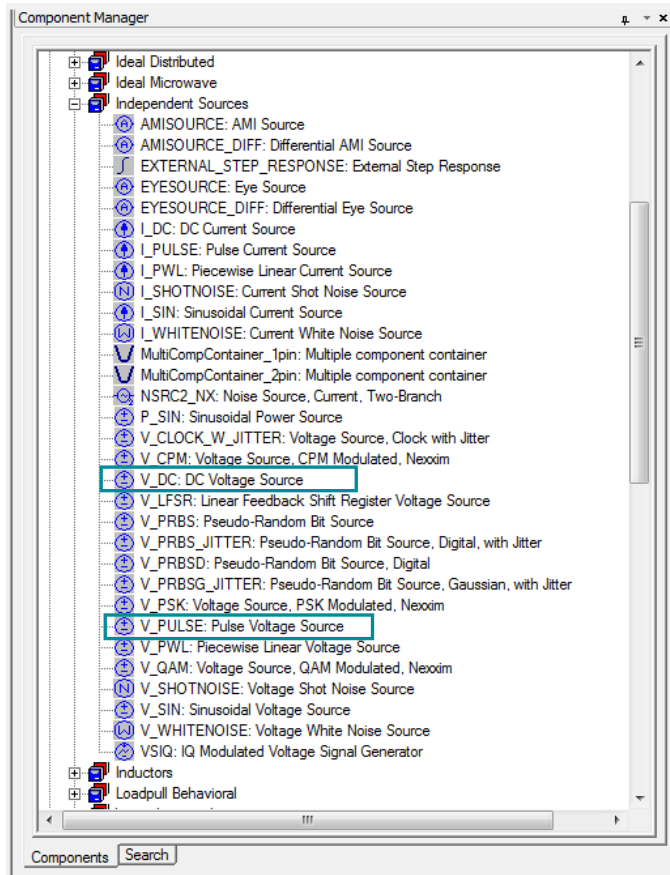
- **Creating the Schematic (continued)**



- The following additional components will be required to complete the circuit. These are available in the components tab and can be placed by double-clicking on the component in the components tab of the **Component Manager** sidebar workspace and placing instances in the schematic.
- Component placement is terminated by pressing the <ESC> key or the space bar.
  - **Resistors > RES\_: Resistor (3x).**
  - **Independent Sources > V\_DC: DC Voltage Source (1x)**
  - **Independent Sources > V\_PULSE: Pulse Voltage Source (1x)**
  - **Probes > VPROBE (10x)**

**Note:** it is probably easiest to place the probes after wiring the schematic.
- Ground connections can be added by selecting the ground symbol  from the toolbar or choosing **Draw > Ground** from the menu.

- Creating the Schematic (continued)

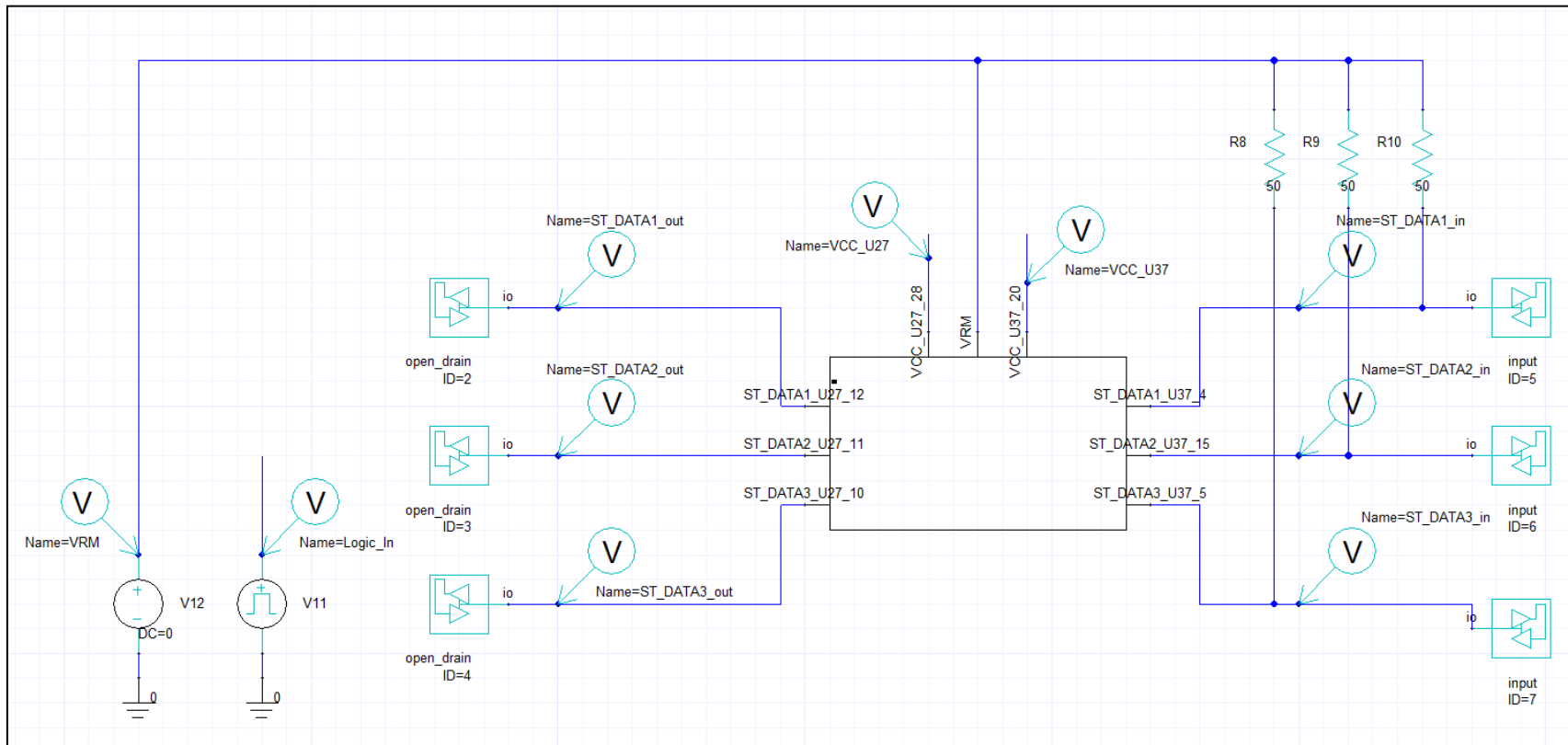


General components can be accessed from the Components tab of the Project Manager. Double clicking the component makes it available for placement in the schematic.


# Example – SSN on PCB

## • Creating the Schematic (continued)

- The wired schematic is shown here. Electrical connections can now be made by selecting the wire drawing tool from the toolbar or by choosing **Draw > Wire** from the menu.
- See the following page for tips and tricks to use while wiring the schematic.
- Include wire stubs from the **VCC\_U27\_28**, **VCC\_U37\_20**, and the **V\_Pulse** pins



- **Tips and tricks while working with the schematic:**

- A component that has been placed in the schematic can be rotated by selecting the component and choosing Draw > Rotate from the menu (shortcut = <Ctrl>-r)
- When a component has been wired into the schematic, selecting and dragging this component will also move the wires and may change connectivity.
- The undo history is retained back to the last time the file was saved. Undo can be accessed from the menu by selecting edit > undo (shortcut <Ctrl>-z)
- A wired component can be moved without modifying the connecting wires by pressing <Ctrl>-<shift>, selecting the component, and dragging it in the schematic.
- Holding <Ctrl> and selecting a component creates a new copy of that component and places it wherever the cursor is located when the left mouse key is released.
- The entire schematic can be fit into the view using View > Fit Drawing (shortcut <Ctrl>-d)
- The zoom and pan shortcut keys are the same in Electronics Desktop as they are in SIwave:
  - <alt>-<shift> mouse click (zoom)
  - <shift> mouse click (pan)
- Component can be mirrored vertically or horizontally using these toolbars  or from the menu using Draw > Flip Vertical or Flip Horizontal

# Example – SSN on PCB

## • Creating the Schematic (continued)

- Name the voltage probes as indicated in the schematic. The probe names are repeated here for clarity.
  - VCC\_U27, VCC\_U37, VRM
  - ST\_DATA1\_out, ST\_DATA2\_out, ST\_DATA3\_out
  - ST\_DATA1\_in, , ST\_DATA2\_in, ST\_DATA3\_in
  - Logic\_in
- Assign the value DC = **1.5V** to the DC voltage source.
  - Select the DC source in the schematic
  - With the DC source selected, assign the value **1.5V** to the property DC in the properties window.
- Change the resistance values for all 3 resistors to 25 Ohms.
  - Select all 3 resistors by holding the <Ctrl> key while selecting them in the schematic.
  - Modify the value of R in the properties window to 25

Name	Value	Unit	Evaluated V...
ACMAG	nan		nan
ACPHASE	0		0
DC	1.5	V	1.5V
CosimDefinition	Edit		
CoSimulator	DefaultNetlist		
Status	Active		
Info	V_DC		

Name	Value	Unit	Evaluated V...
R	25		25
CosimDefinition	Edit		
CoSimulator	DefaultNetlist		
Status	Active		
Info	RES_		

Param Values General Symbol

# Example – SSN on PCB

## • Creating the Schematic (continued)

- Define Pulse Source
  - Select the pulse source component in the schematic by double clicking on it.
  - Enter the pulse properties as show below in the component properties window.

**Note:** When entering the time values type them into the field as “10ns” and the fields will populate correctly

V8:V\_PULSE Properties: ssn\_training - Circuit1

Parameter Values | General | Symbol | Property Displays

☒ Value ☐ Statistics

	Name	Value	Unit	Evaluated Value	Description	Callback	Override
	ACMAG	nan		nan	AC magnitude for small-signal analysis (...)	...	<input type="checkbox"/>
	ACPHASE	0		0	AC phase for small-signal analysis	...	<input type="checkbox"/>
	V1	0		0	Initial and final voltage (Volts)	...	<input type="checkbox"/>
	V2	1		1	Pulse voltage (Volts)	...	<input checked="" type="checkbox"/>
	TD	0		0	(Positive) delay time to start of upramp (s...)	...	<input type="checkbox"/>
	TR	10	ps	10ps	Risetime from V1 to V2 (seconds)	...	<input checked="" type="checkbox"/>
	TF	10	ps	10ps	Falltime from V2 to V1 (seconds)	...	<input checked="" type="checkbox"/>
	PW	20	ns	20ns	Pulse width (V2 hold time) (seconds)	...	<input checked="" type="checkbox"/>
	PER	40	ns	40ns	Period of repetition for trapezoidal pulse ...	...	<input checked="" type="checkbox"/>

☐ Show Hidden

OK Cancel

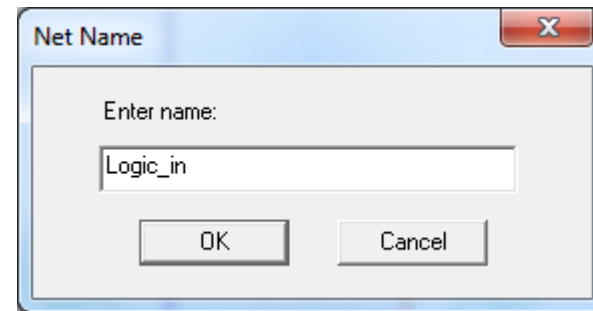
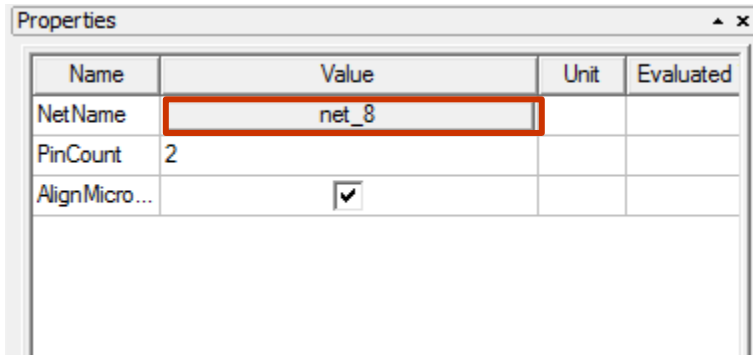
- If you haven't already done so, save the project.



## Example – SSN on PCB

- **Creating the Schematic (continued)**

- Define the nets used internally in the IBIS models
- Select the wire stub coming out of the **V\_PULSE** source and in the properties window click on the NetName property and assign the name **Logic\_in** to the net.

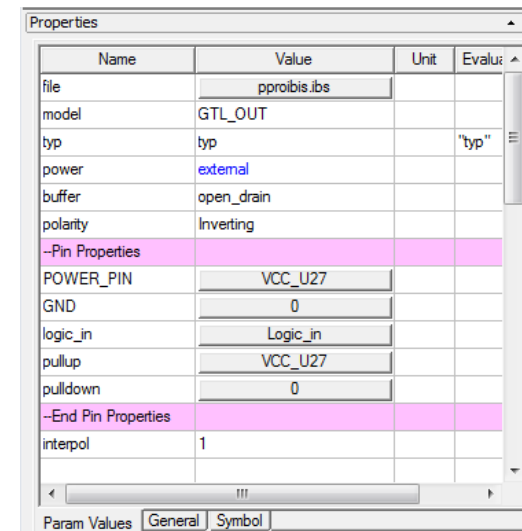
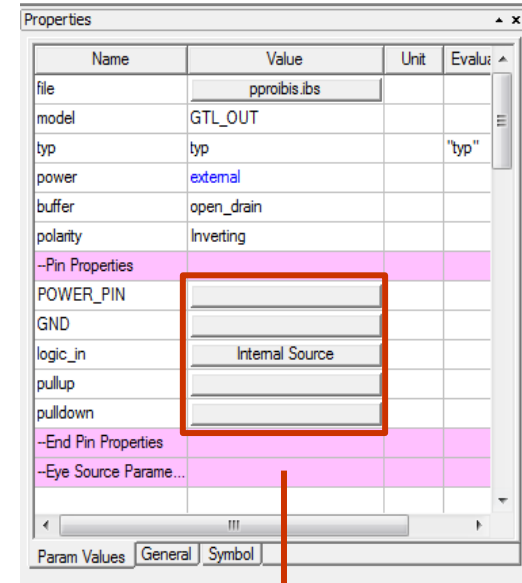
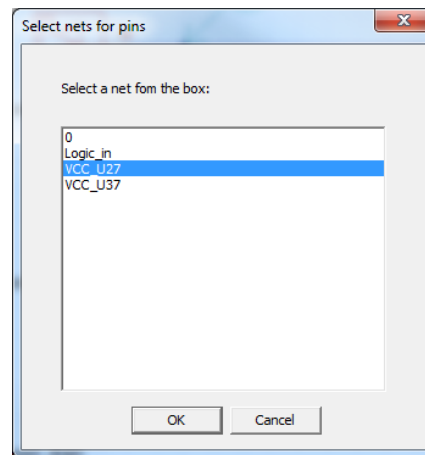


- Repeat this process for the wire stubs on **VCC\_U27\_28** and **VCC\_U37\_20** giving them the net names **VCC\_U27** and **VCC\_U37** respectively.

# Example – SSN on PCB

## • Creating the Schematic (continued)

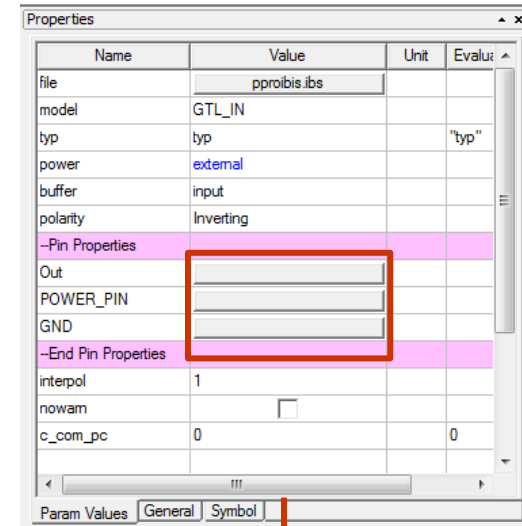
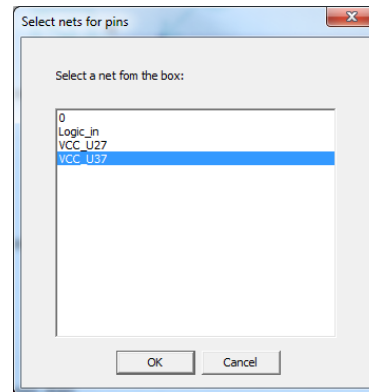
- Select the three output IBIS model components **GLT\_OUT\_pproibis**
- In the properties menu press the grey box next to **POWER\_PIN**
  - Assign **VCC\_U27** to the pins
  - Press **OK**
- Select the **GND** box
  - Assign **0** to the pin
  - Press **OK**
- Select the **pullup** box
  - Assign **VCC\_U27** to the pins
  - Press **OK**
- Select the **pulldown** box
  - Assign **0** to the pins
  - Press **OK**
- Select the **logic\_in** box
  - Select the **Select Net** radio button
  - Assign **Logic\_in** to the pins
  - Press **OK**
- Click on an open area of the schematic to deselect the parts
- Select the IBIS part connected to **ST\_DATA3\_U27\_10**
- Select the **logic\_in** box
  - Select the **Select Net** radio button
  - Assign **0** to the pins
  - Press **OK**



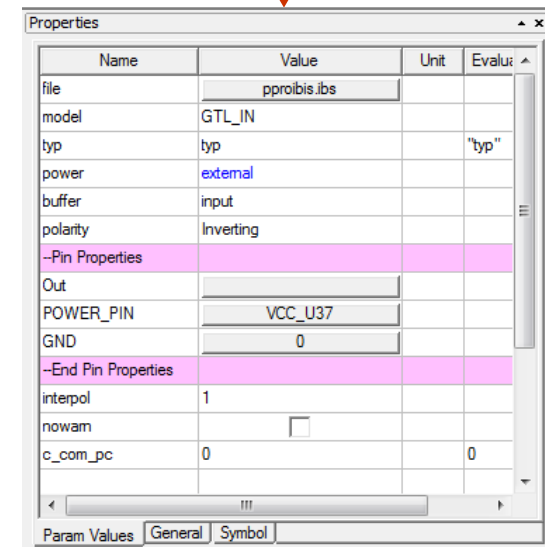
# Example – SSN on PCB

## • Creating the Schematic (continued)

- Select the three output IBIS model components **GLT\_IN\_pproibis**
- In the properties menu press the grey box next to **POWER\_PIN**
  - Assign **VCC\_U37** to the pins
  - Press **OK**
- Select the **GND** box
  - Assign **0** to the pins
  - Press **OK**



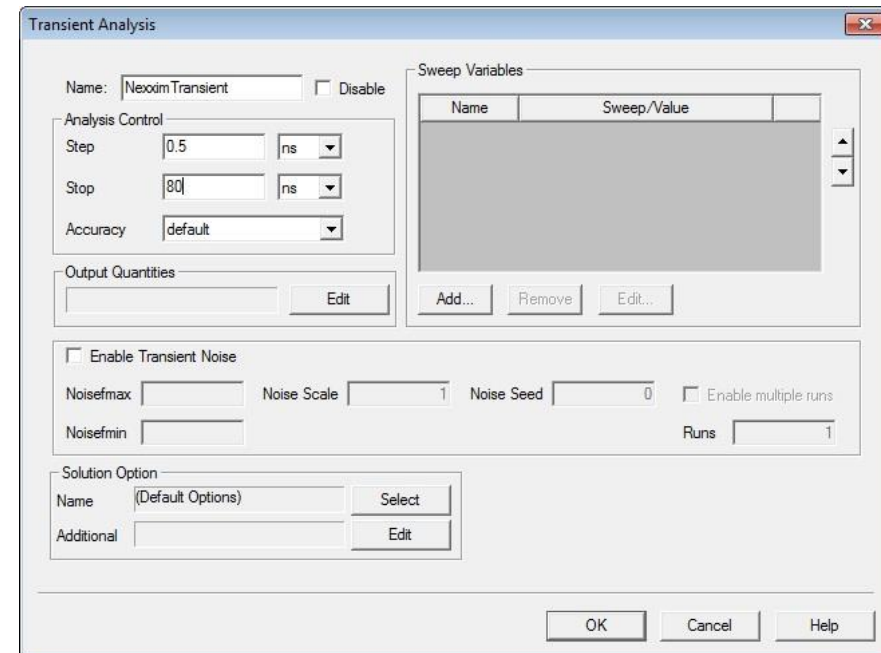
- Click on an open area of the schematic to deselect the parts



# Example – SSN on PCB

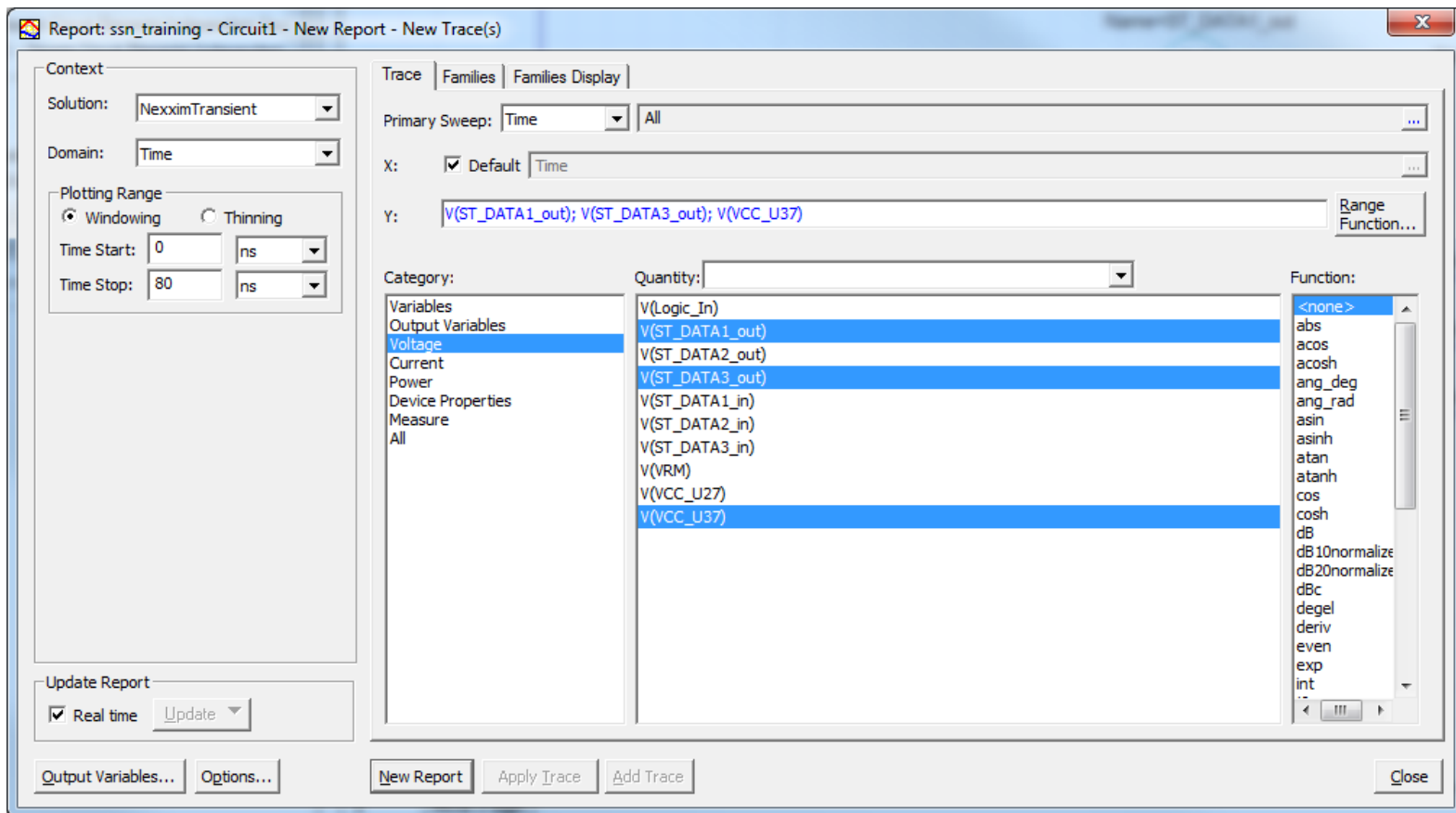
## • Analysis Setup

- To create an analysis setup:
  - Select the menu item **Circuit > Add Nexxim Solution Setup > Transient Analysis**
  - Analysis Setup Window:
    - **Analysis Type: Transient Analysis**
    - **Analysis Control**
      - **Step: 0.5 ns**
      - **Stop: 80 ns**
    - Click **OK**
  - Select the menu item **Circuit -> Analyze**
    - The progress bar displays the progress of the simulation.



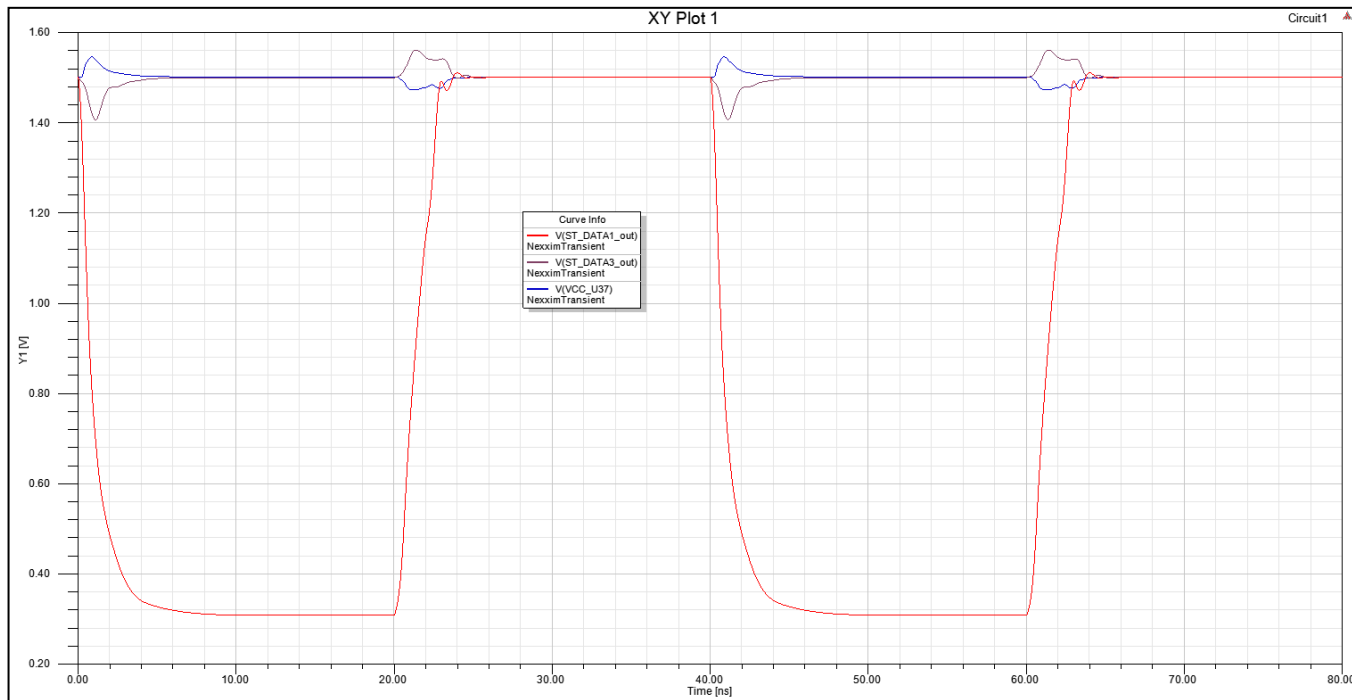
## Example – SSN on PCB

- **Plot Aggressor, Victim and power supply waveforms**
  - To create a report:
    - Select the menu item **Circuit > Results > Create Standard Report > Rectangular Plot**
    - Hold the <Ctrl> key to select multiple voltage signals as shown here:



## Example – SSN on PCB

- **Plot Aggressor, Victim and power supply waveforms (cont.)**
  - Press the button **New Report** when the desired traces have been selected. The following plot should now be visible.



- Discussion
  - SSN is often a concept associated with cross-talk that occurs when signal traces have ideal reference planes. In this example, the IBIS drivers draw current through the power supply of the board and create voltage ripple between the supply plane and ground. This noise is transferred to the signal traces because the signal reference planes are no longer ideal.
  - The classical crosstalk between the traces is included in this analysis as well and also contributes to the signal noise. An additional exercise would be to provide the IBIS drivers with current directly from the DC source and view the resulting cross-talk.