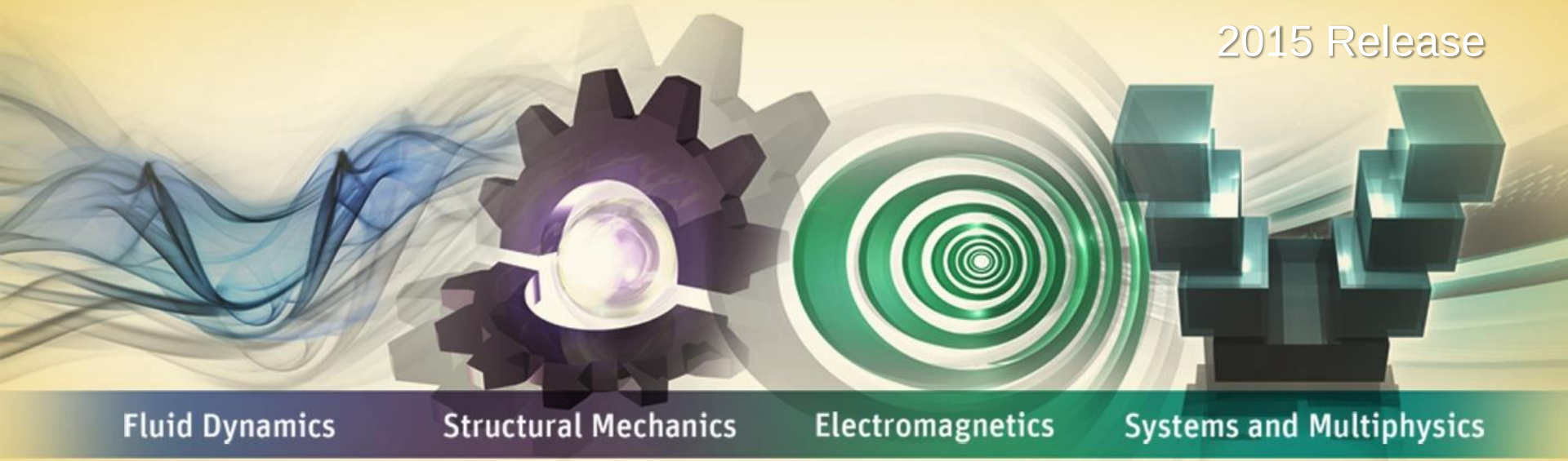
ANSYS

Realize Your Product Promise™

Workshop 7: SYZ For SI

2015 Release



Introduction to ANSYS SIwave

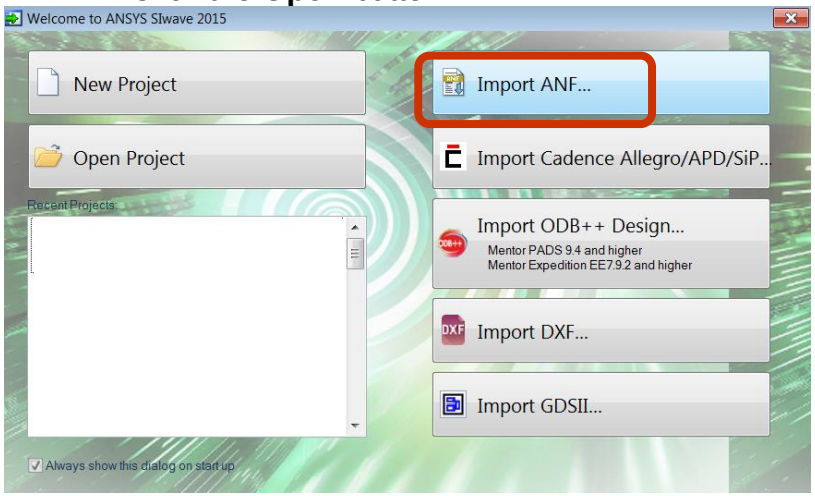
SIwave SYZ Analysis For SI

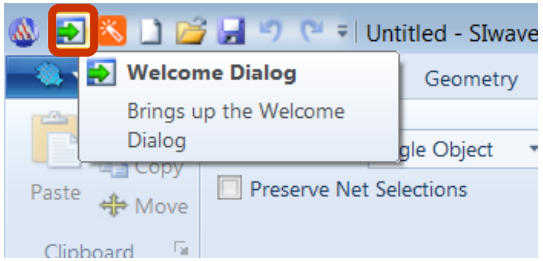
- **In this example you will learn and use the following skills:**
  - Import an Ansoft Neutral Format (ANF) file and component file (CMP) and create an SIwave project
  - Place ports on selected nets using an automated process
  - Run an SYZ simulation and plot results for transmission, reflection, and crosstalk
  - Export a Full-Wave SPICE sub-circuit
  - Modify trace geometry to reduce crosstalk

- **Starting SIwave**
  - To launch SIwave, click the Microsoft **Start** Button, select:

  All Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.0 > ANSYS SIwave 2015

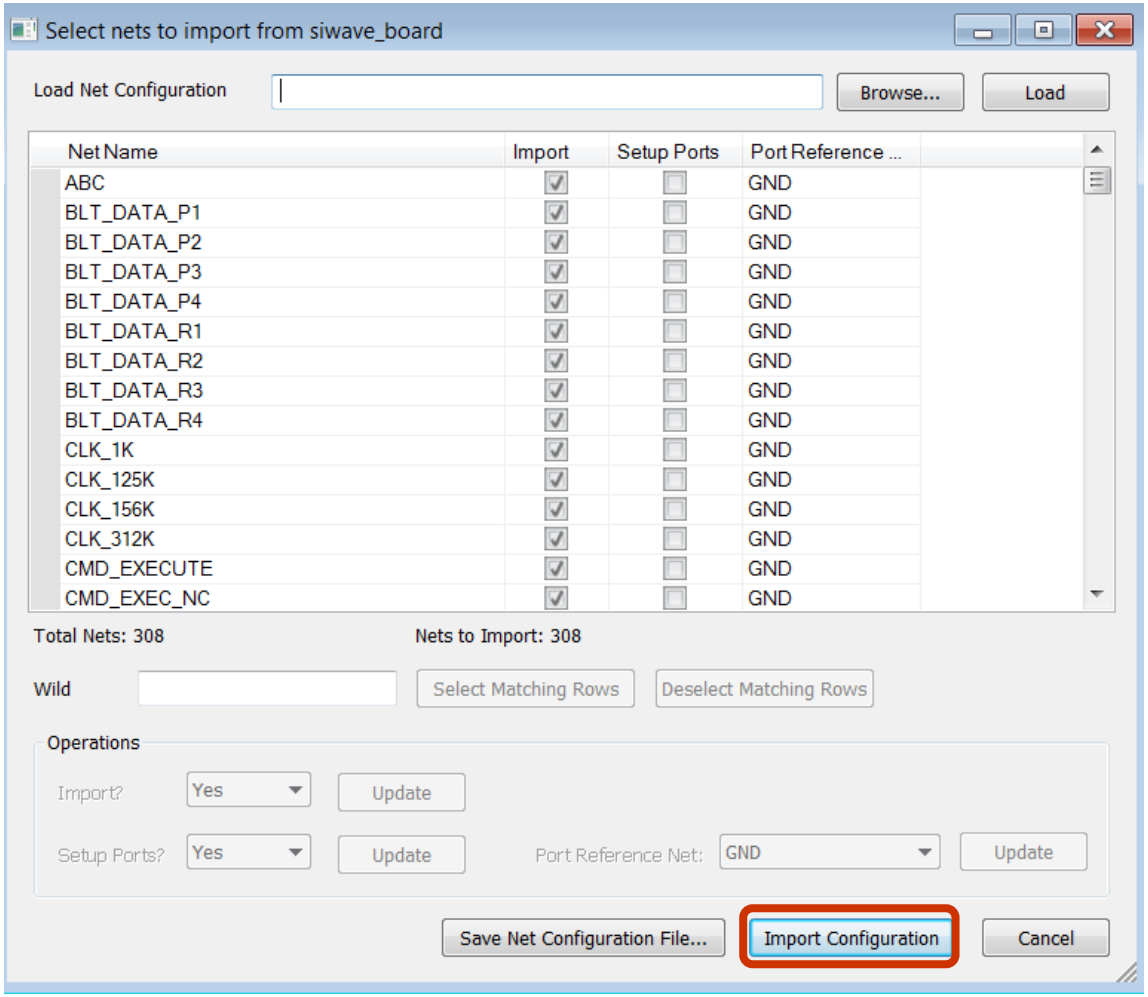
- **Import the .ANF (Ansoft Neutral File) file**
  - Click the **Import ANF**… box
    - Navigate to the training files and choose : **siwave_board.anf**
    - Click the **Open** button





Slwave SYZ Analysis For SI

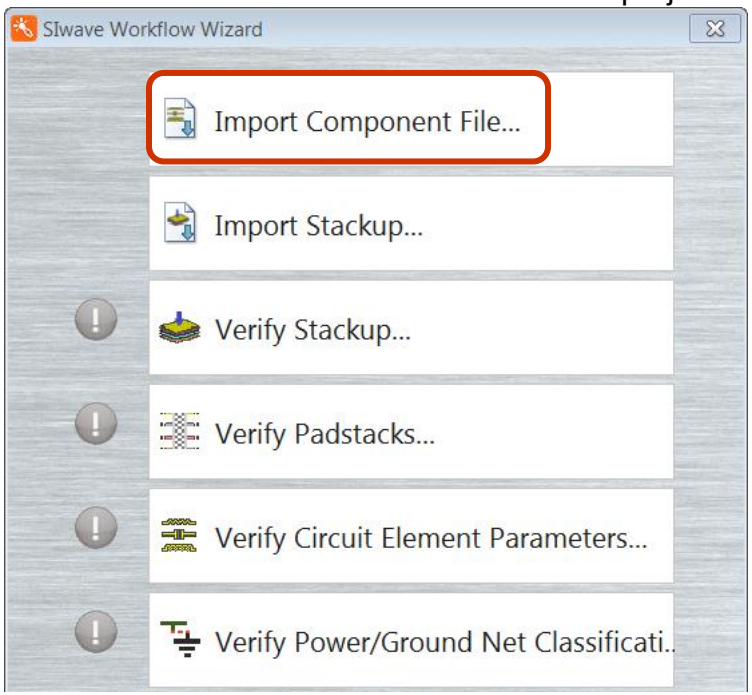
- **The Select nets box will appear. (If desired, the user can filter nets to be imported. For this example, all nets will be imported)**
  - Click the **Import Configuration** button

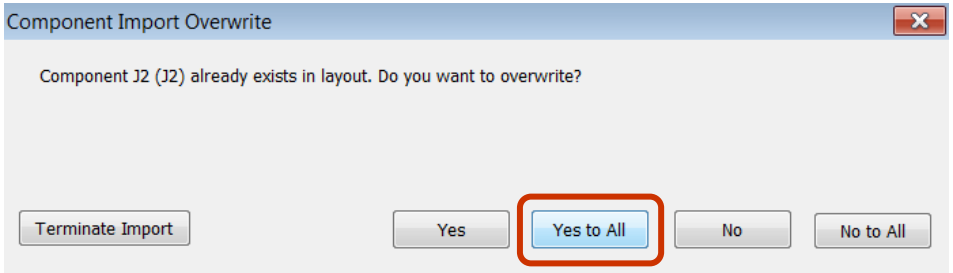


SIwave SYZ Analysis For SI

- **The SIwave Workflow Wizard will appear**

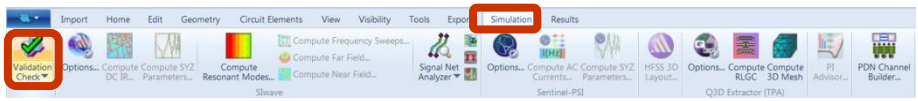
- **Import the .cmp (Ansoft Component File)**
  - Click on the **Import Component File…** button
    - Navigate to the training files and choose : **siwave_board.cmp**
    - Click the **Open** button
    - If there is a warning message click the **Yes to All** button to overwrite existing names
    - Close the Wizard and save the project

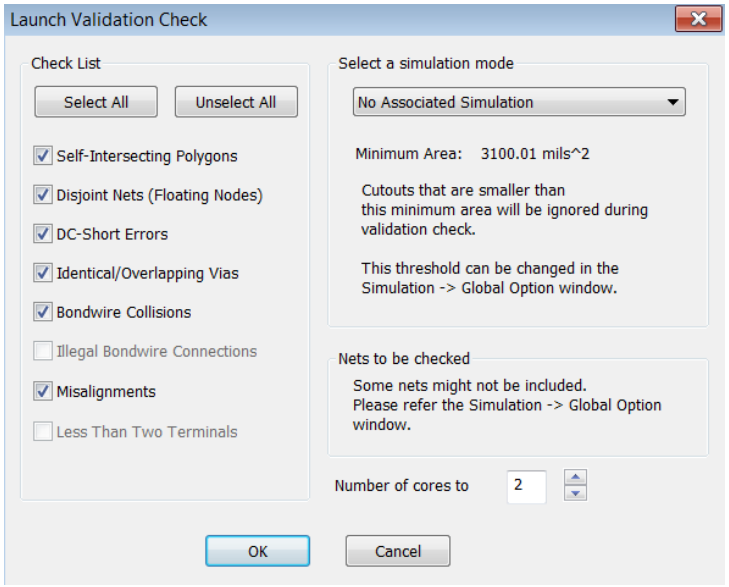


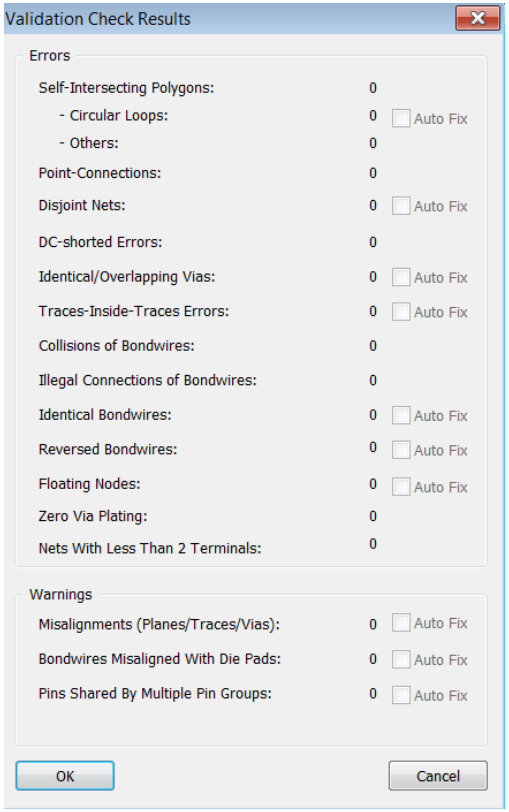


SYZ Parameters For Signal Integrity

- **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 tests for self-intersecting polygons, disjoint nets, overlapping (DC-shorted) nets and nets with overlapping vias.
  - This helps you avoid finding layout errors after all the setting up ports and other solution settings.
  - To do a validation check:
    - Go to the **Simulation** tab, select ***Validation Check***.
    - Click **OK** button to start the validation
    - There are no layout and DRC related problem with this design.
      - Repeat this process if there are some **Misalignments** to verify that the **Auto Fix** worked
    - Click **OK**.

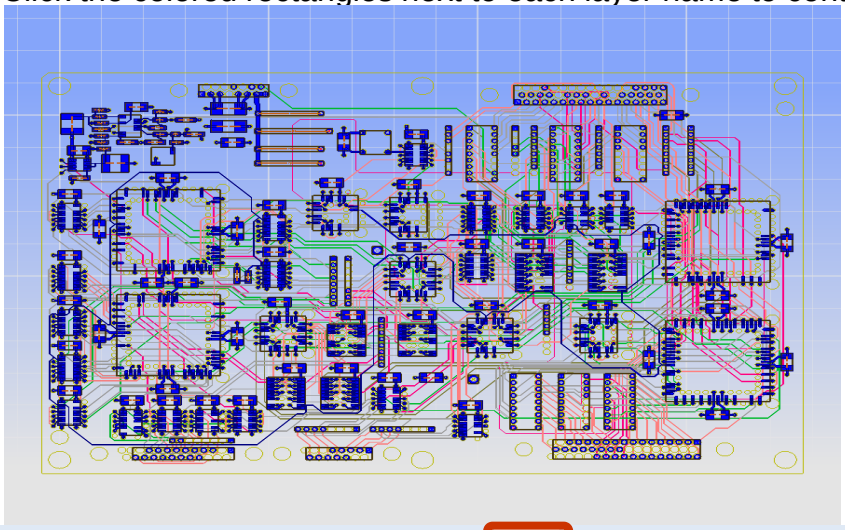


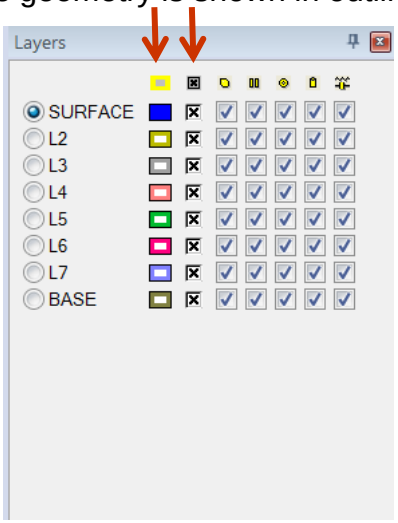


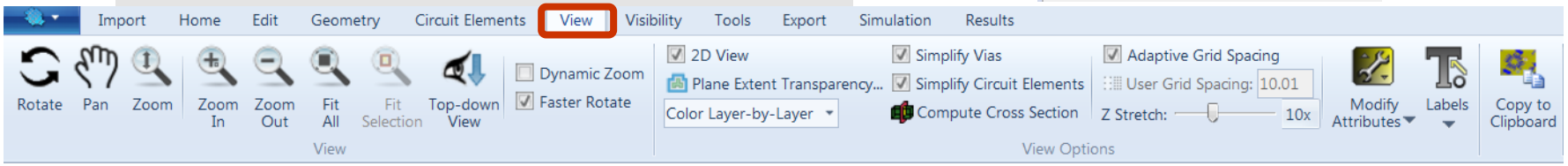


SYZ Parameters For Signal Integrity

- **Set Visibility options**
  - Using the checkboxes in the **Layers** workspace, turn on full visibility of all layers
  - Click the colored rectangles next to each layer name to control whether the geometry is shown in outline or filled form.



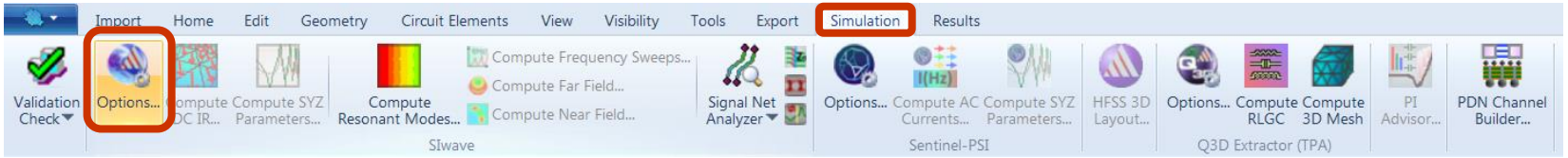




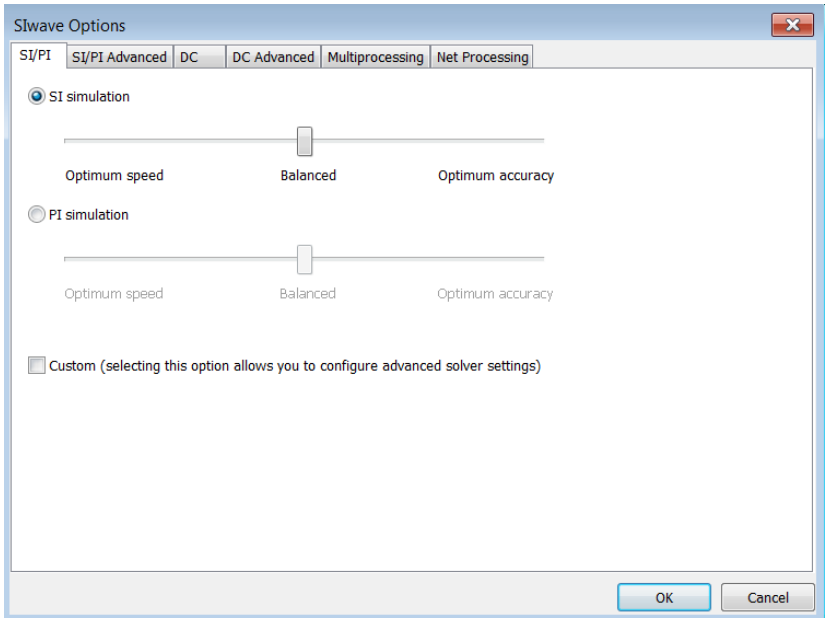
  - Go to the **View** tab
    - Check **2D View** to view the project as a 2-dimensional entity
    - Check ***Faster Rotate***
    - Uncheck ***Dynamic Zoom***
    - Check ***Simplify Circuit Elements***
    - Check ***Simplify Vias***
  - **Notes:** Selecting these options for large designs reduces memory used by SIwave and improve the response time.

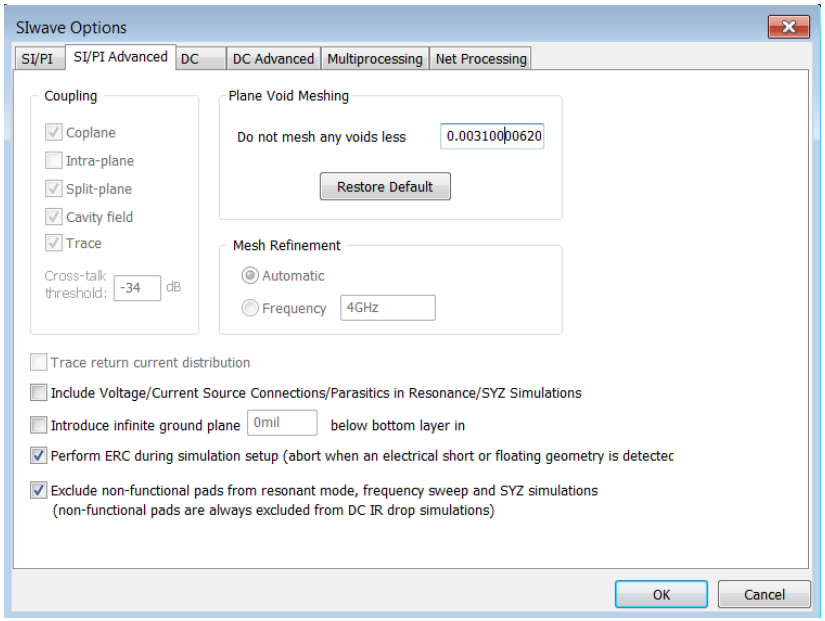
SYZ Parameters For Signal Integrity

- **Setting Simulation Global Options**
  - From the **Simulation** tab, Select ***Options***



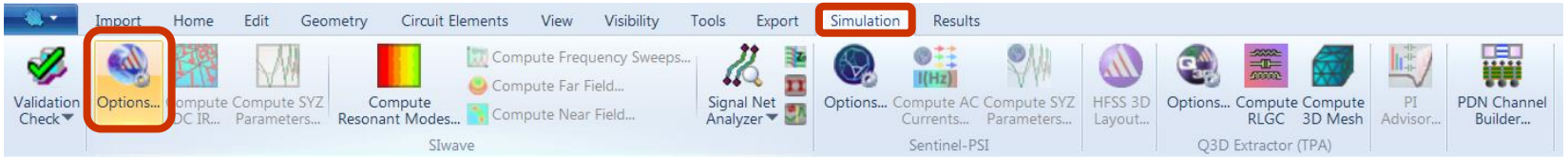
  - Under ***SI/PI*** tab, let the slider bar of ***SI simulation*** to ***Balanced***.
    - Note : To configure advanced solver settings Check ***Custom***.
  - Under ***SI/PI Advanced*** tab, look at the solver settings
    - Perform ERC during simulation setup: ☑ **Checked**
    - Exclude non functional pads : ☑ **Checked**



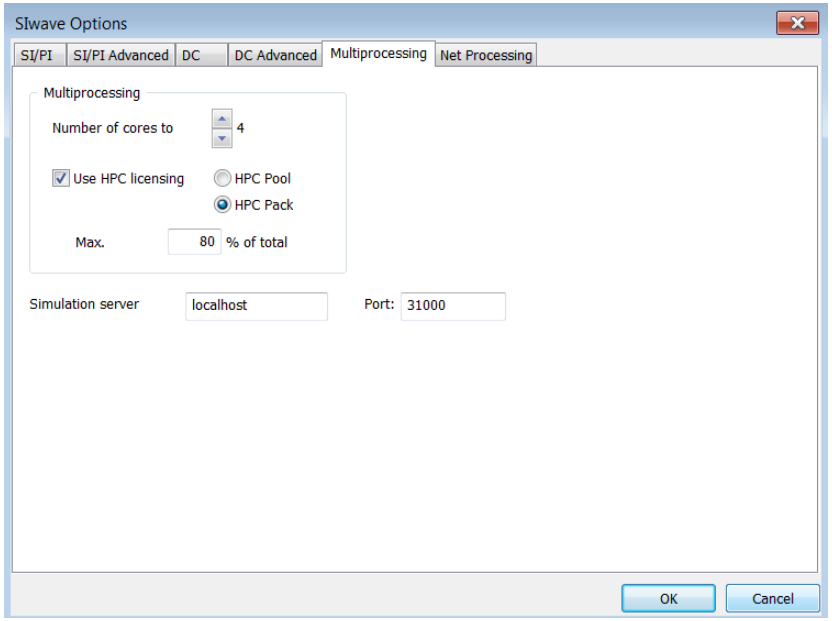


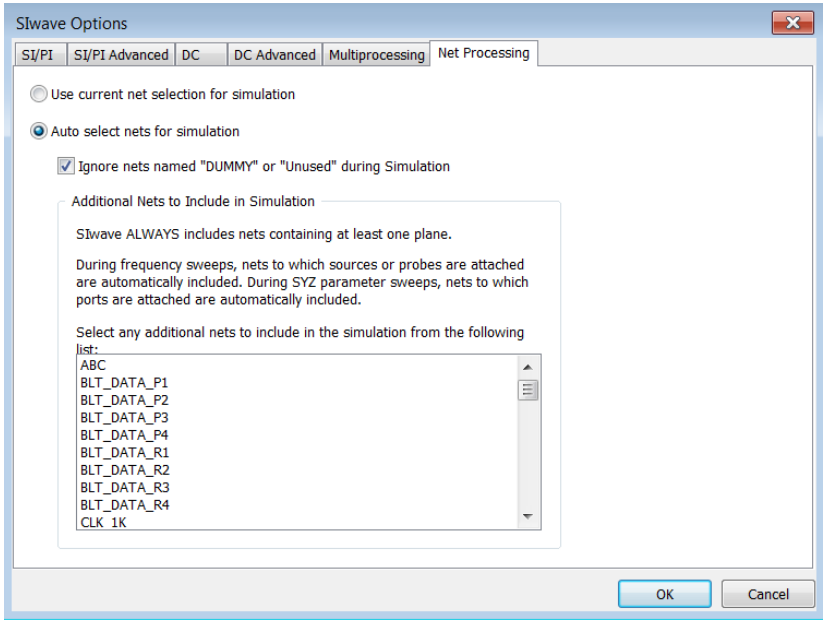
SYZ Parameters For Signal Integrity

- **Setting Simulation Global Options**
  - From the **Simulation** tab, Select ***Options***



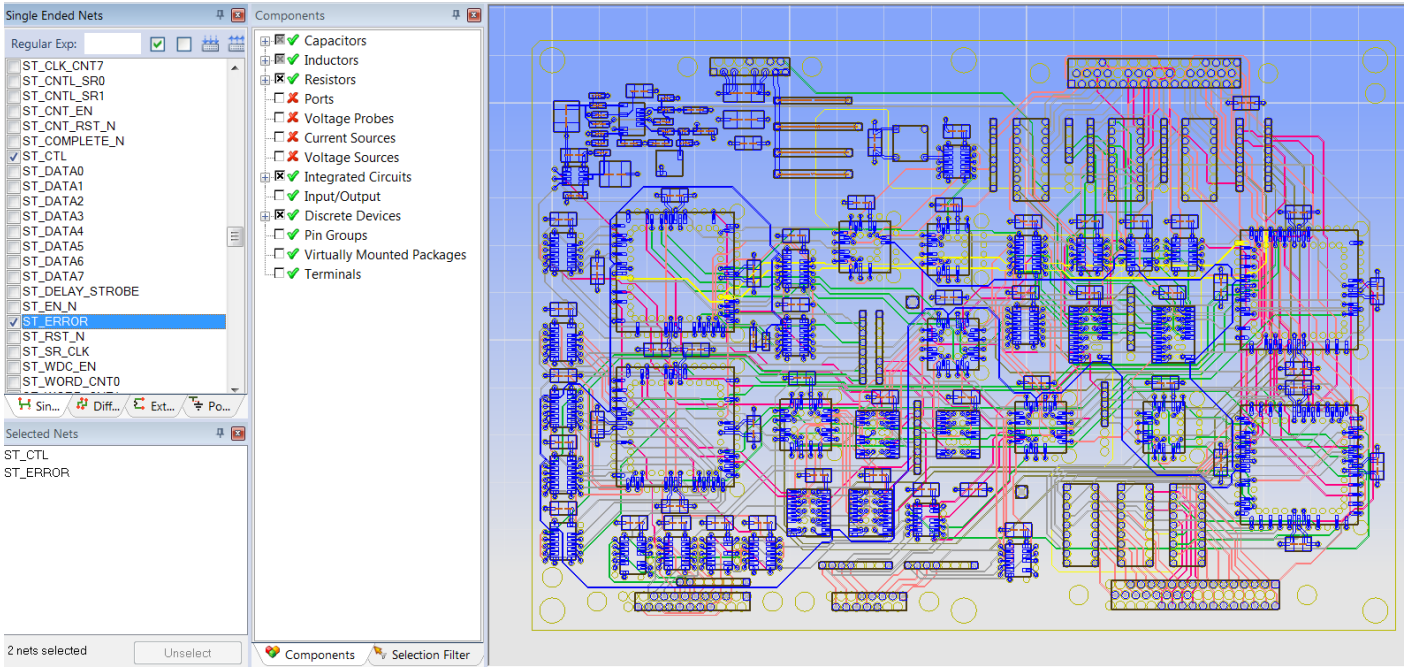
  - Under ***Multiprocessing*** tab , set the number of cores you want to use
    - Check **Use HPC Licensing** to **HPC Pack** or **HPC Pool** if you have HPC licenses
  - Under ***Net Processing*** tab
    - Ignore nets named “DUMMY” or “Unused” during Simulation: ☑ **Checked**
  - Click **OK** button





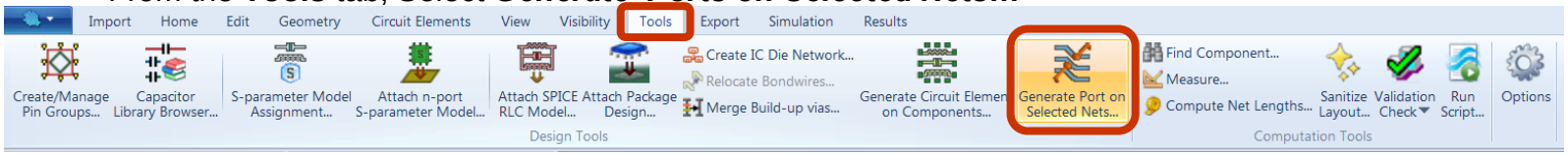
SYZ Parameters For Signal Integrity

- **Add ports to the nets of interest**
  - Ports are elements that enable the entry and exit of electromagnetic energy in a design. We will add ports at both ends of two coupled traces.
  - In the **Single Ended Nets** workspace, scroll down until you find the **ST_CTL** and **ST_ERROR** nets
  - Place check marks next to
    - **ST_CTL**
    - **ST_ERROR**
  - The nets will show up in the **Selected Nets** workspace.
  - The nets should now be highlighted. For the most part, they run horizontally on layer L3
    - You can also highlight nets by checking the checkbox next to Nets in the **Selection Filter** workspace and then clicking on the nets in the layout window

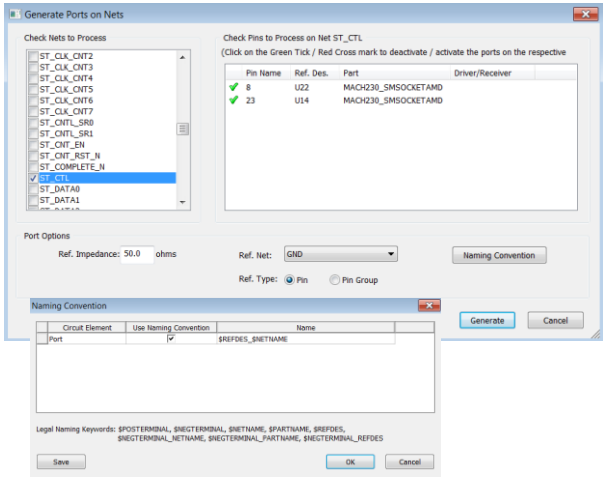


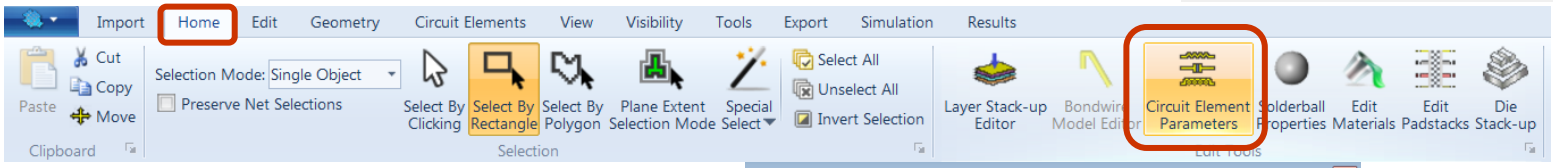
SYZ Parameters For Signal Integrity

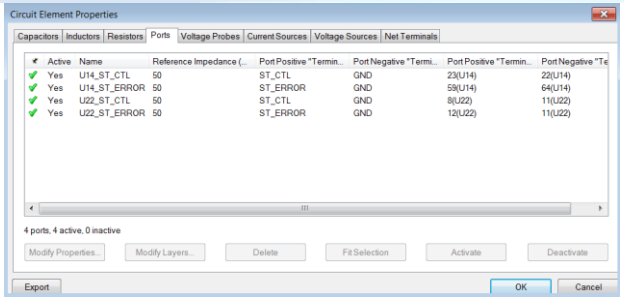
- **Generate Ports**
  - From the **Tools** tab, Select ***Generate Ports on Selected Nets…***



  - Click the **Naming Convention** button:
    - Enter Port naming convention: **$REFDES_$NETNAME**
    - Select **Use Naming Convention** ☑
    - Press the **OK** button
  - Make sure the **Ref. Net** is set to **GND**
  - Click the **Generate** button to automatically create 50 ohm ports at each end of the selected nets
  - Verify that the 4 ports have been properly created
    - From the **Home** tab, Select ***Circuit Element Parameters…***
      - Select the **Ports** tab
      - Click **OK**

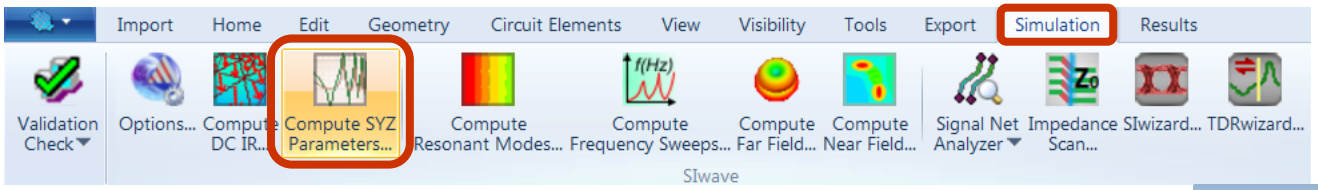




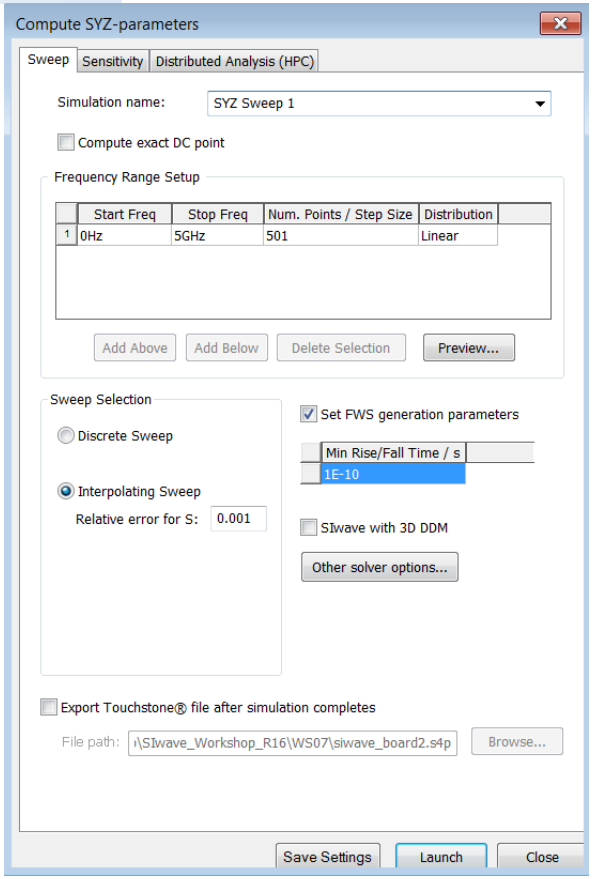


SYZ Parameters For Signal Integrity

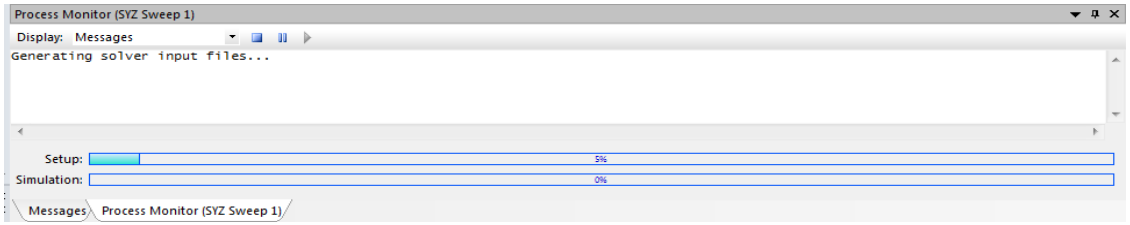
- **Run an SYZ Simulation**
  - Form the menu ***Simulation,*** select ***Compute S-, Y-, Z-Parameters…***



- Change the settings as shown below:
  - Start Freq: **0 Hz** and Stop Freq: **5GHz**
  - Num. Points: **501** and Distribution: **Linear**
  - Sweep Selection:
    - Option button: **Interpolating Sweep**
    - Error Tolerance: **0.001**
  - Set FWS generation parameters: ☑ **Checked**
  - Uncheck ***Export Touchstone file after simulation completes***
- Click the **Launch** button to start the simulation

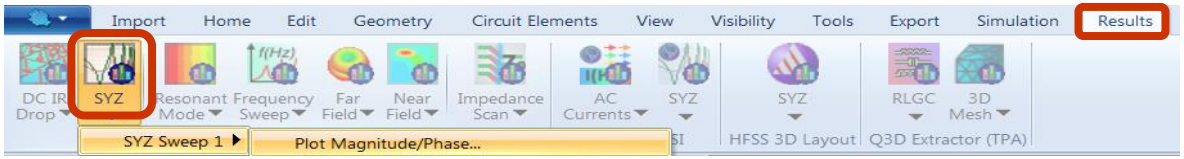


- Wait for the simulation to complete

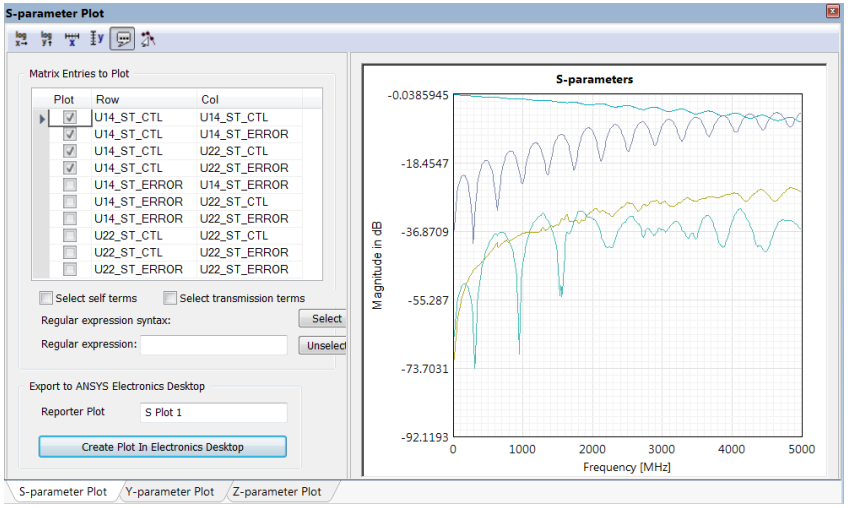


SYZ Parameters For Signal Integrity

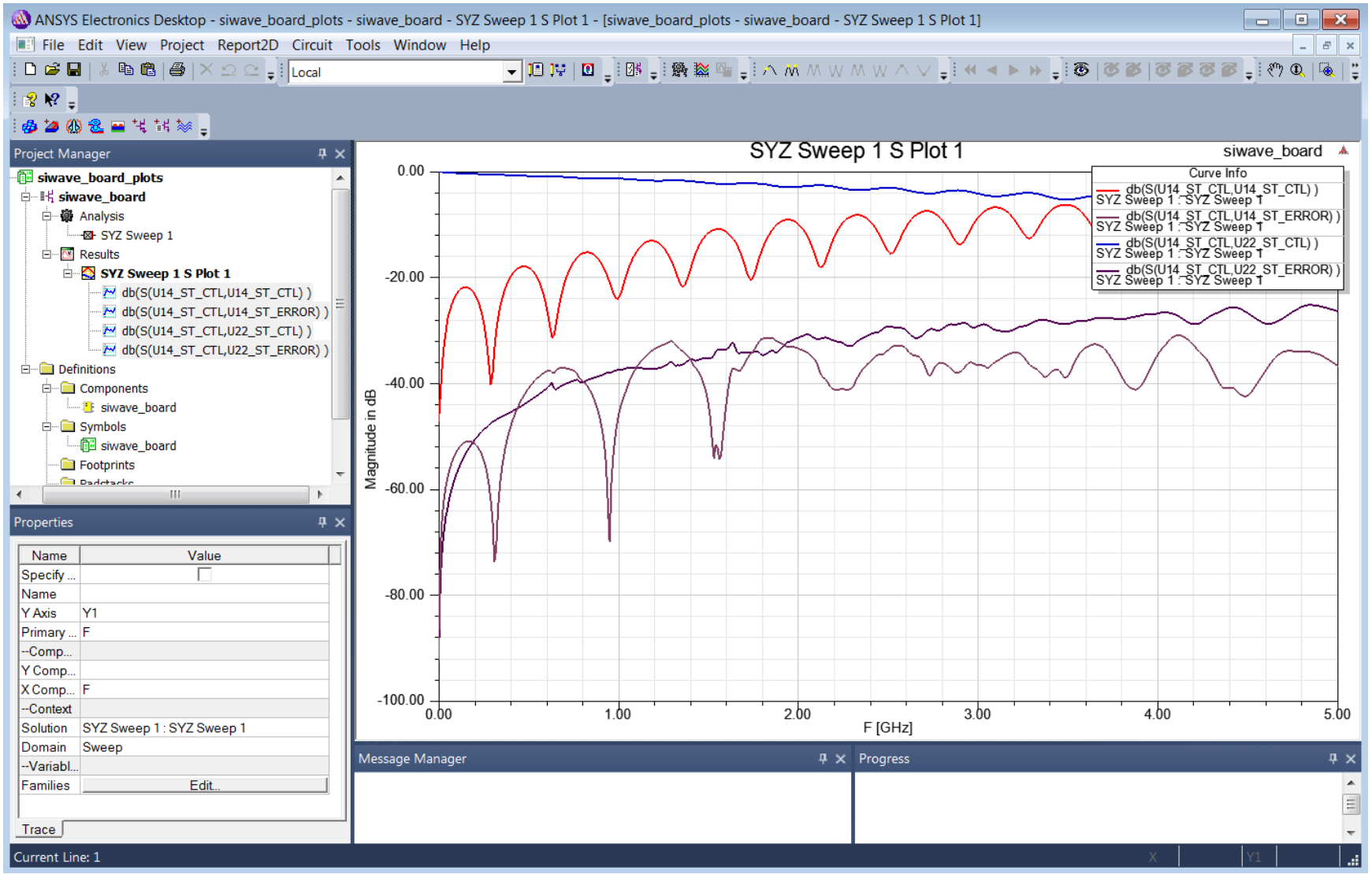
- **Plot the results**
  - When the simulation has finished, plot the results
  - From the ***Results*** tab, Select ***SYZ > SYZ Sweep 1 > Plot Magnitude/Phase…***



  - Uncheck the **Select Transmission Terms** checkbox to clear the plot
  - Uncheck the **log x->** button in the top left corner
  - Check the following four boxes in the list:
    - **U14_ST_CTL, U14_ST_CTL**
    - **U14_ST_CTL, U22_ST_CTL**
    - **U14_ST_CTL, U14_ST_ERROR**
    - **U14_ST_CTL, U22_ST_ERROR**
  - Click the **Create Plot in Electronics Desktop** button

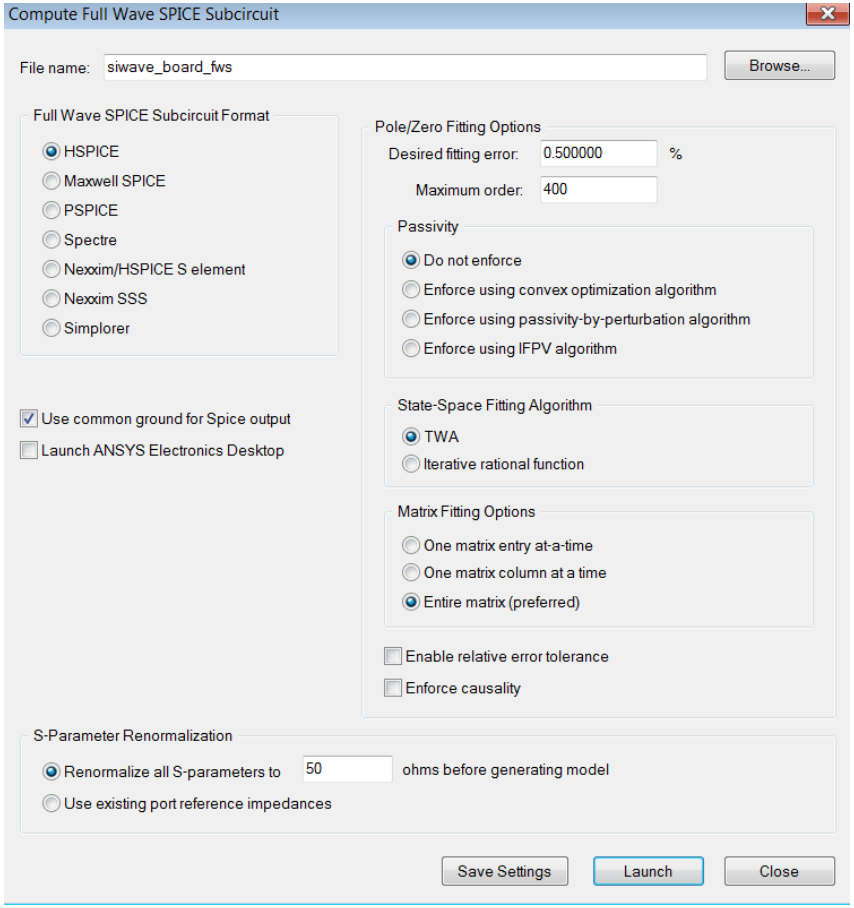


SYZ Parameters For Signal Integrity



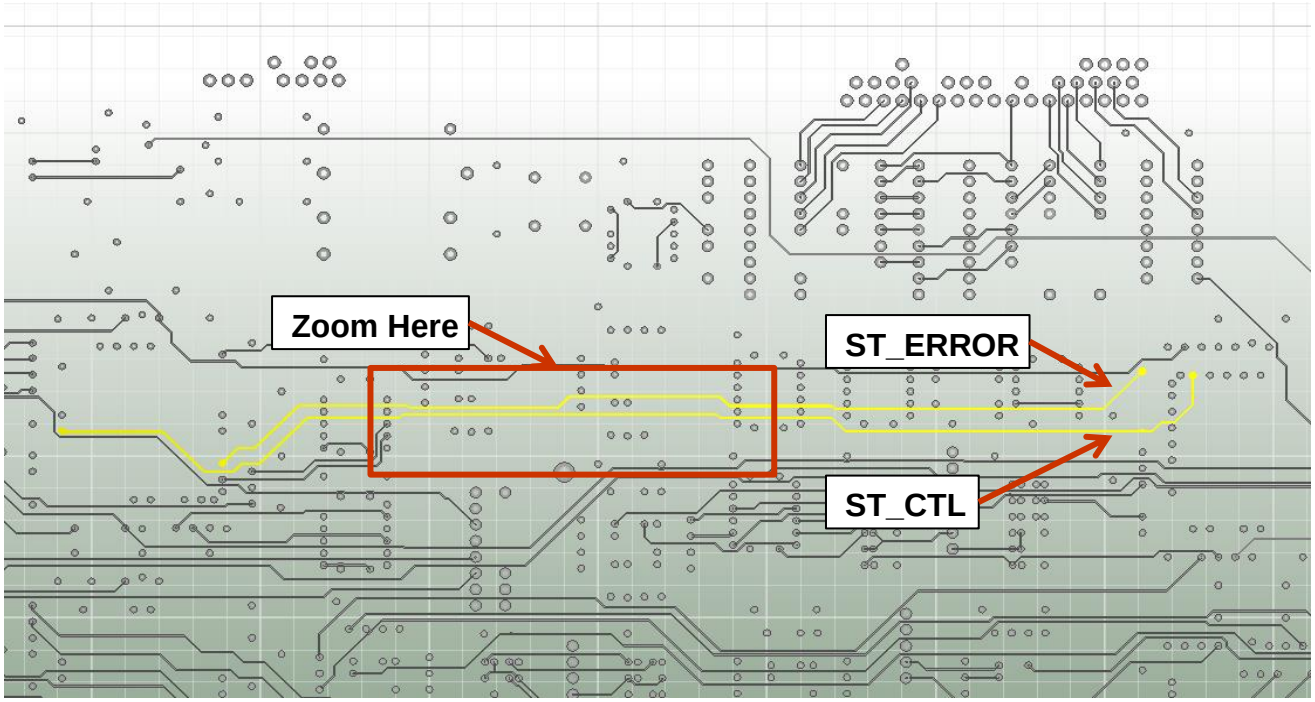
SYZ Parameters For Signal Integrity

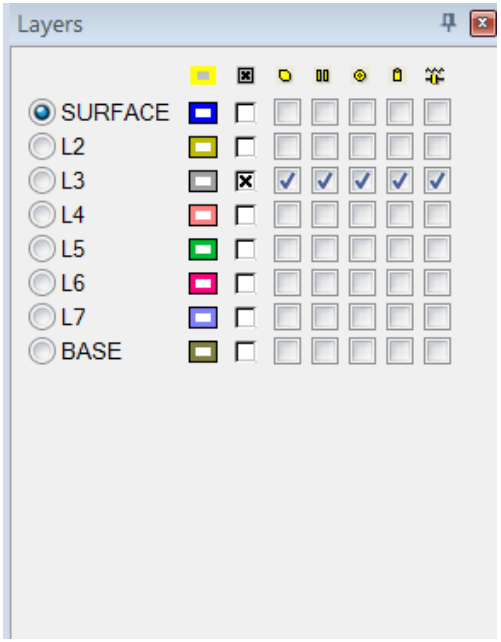
- **Export a Full-Wave SPICE sub-circuit model**
  - Close the **ANSYS Electronics Desktop**
  - Close the **Results** Viewer
  - From the ***Results*** tab **,** Select ***SYZ > SYZ Sweep 1 > Compute FWS sub-circuit***
    - File name: **siwave_board_fws**
    - Full Wave Spice Subcircuit Format: **HSPICE**
    - Use common ground for Spice output: ☑ **Checked**
    - Click the **OK** button



SYZ Parameters For Signal Integrity

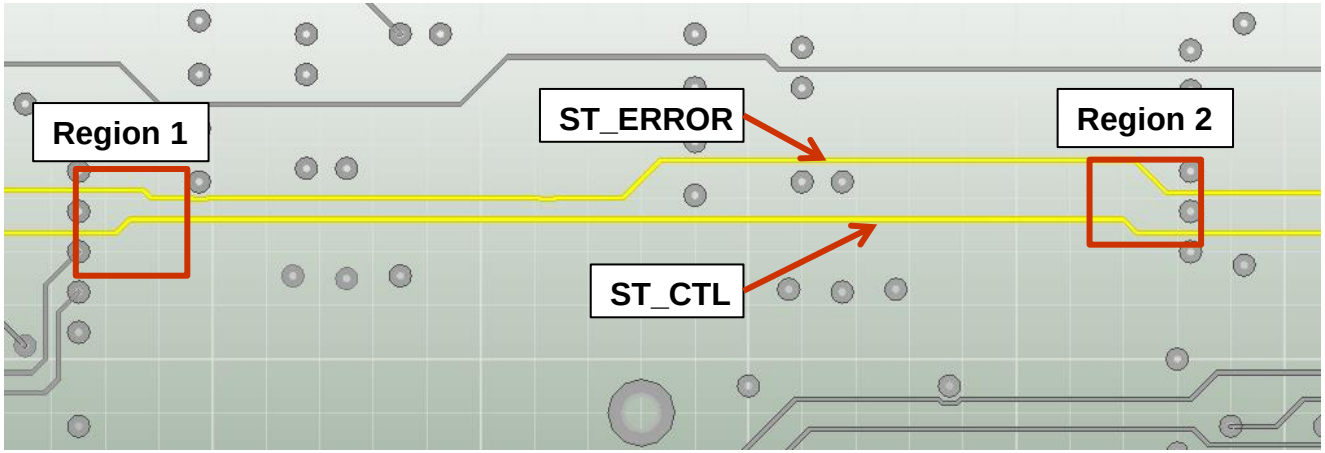
- **Edit the traces to lower crosstalk**
  - The S-parameter results show fairly low near-end crosstalk, but we'll assume that a system simulation using the model indicated a need to reduce the coupling. We will edit the traces by increasing the minimum spacing between them to attempt to lower the crosstalk.
  - Use the checkboxes in the Layers workspace to turn off the visibility of all layers except for L3 on which the ST_CTL and ST_ERROR traces are routed.
    - If the ST_CTL and ST_ERROR nets are not still highlighted, highlight them by clicking the boxes next to their names in the Nets workspace. The main window should look roughly like that below.
    - Zoom into the region outlined in the black box below.

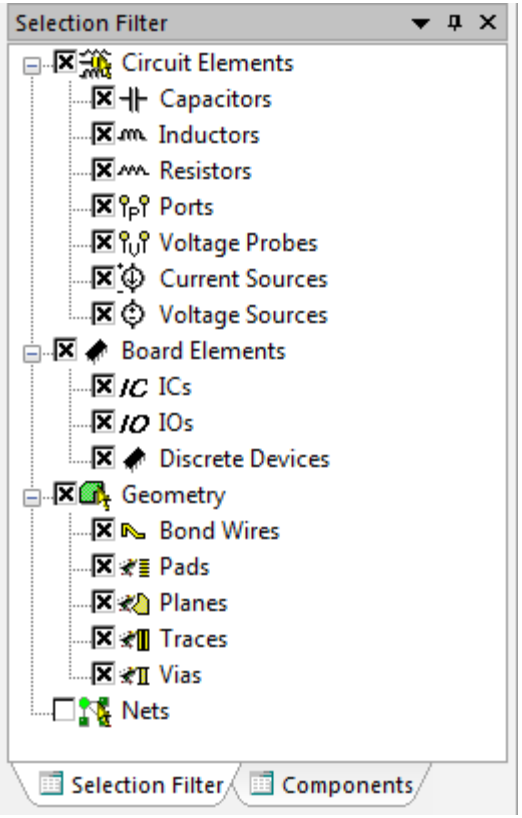




SYZ Parameters For Signal Integrity

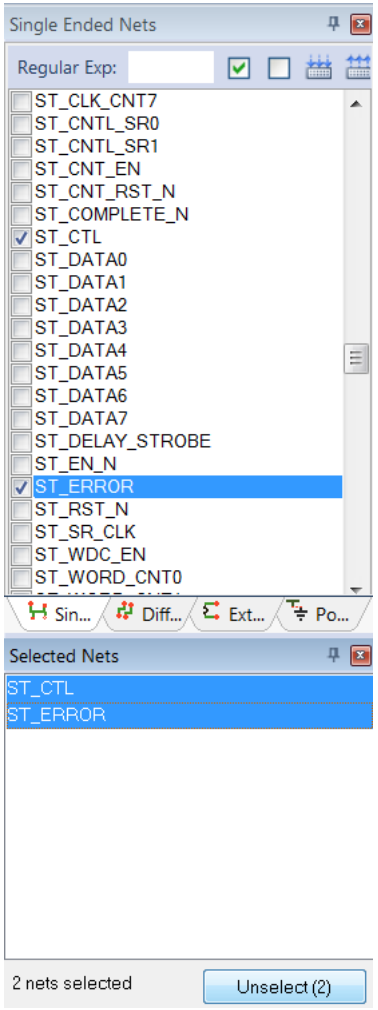
- Now the main window shows a portion of the ST_CTL and ST_ERROR nets where they are routed close together.
- In the **Selection Filter** workspace, ensure that the box next to Geometry is checked.
- Zoom into the area labeled Region 1 in the above figure. The top left corner of Region 1 is at approximately (2700, 3340) mils.





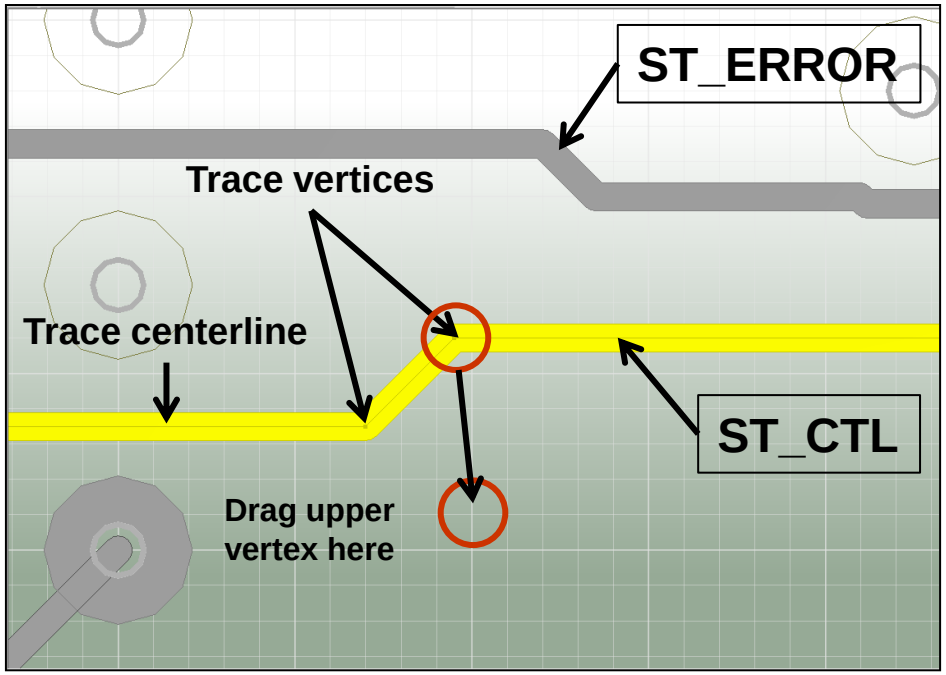
SYZ Parameters For Signal Integrity

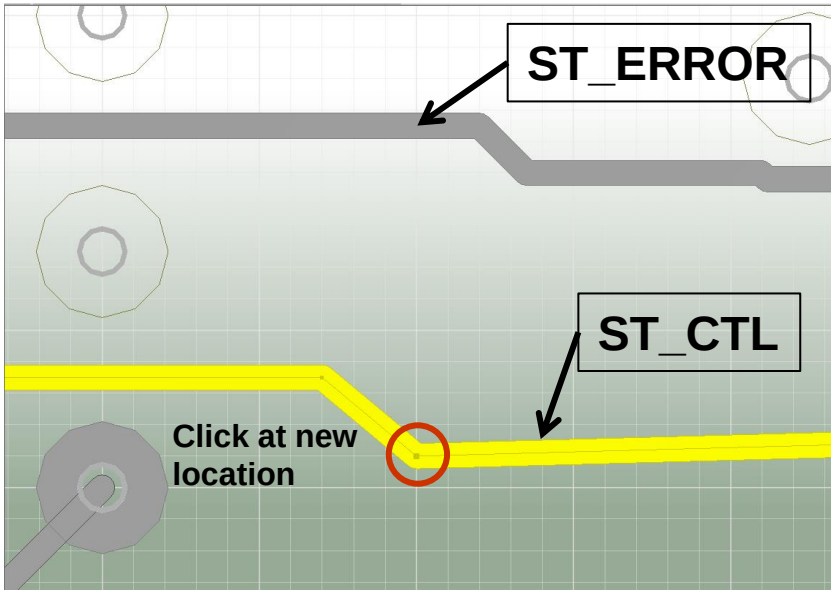
- Unselect the selected nets by selecting both of the nets in the **Selected Nets** sidebar box and press the **Unselect** button. This will allow the nets to be manipulated.



SYZ Parameters For Signal Integrity

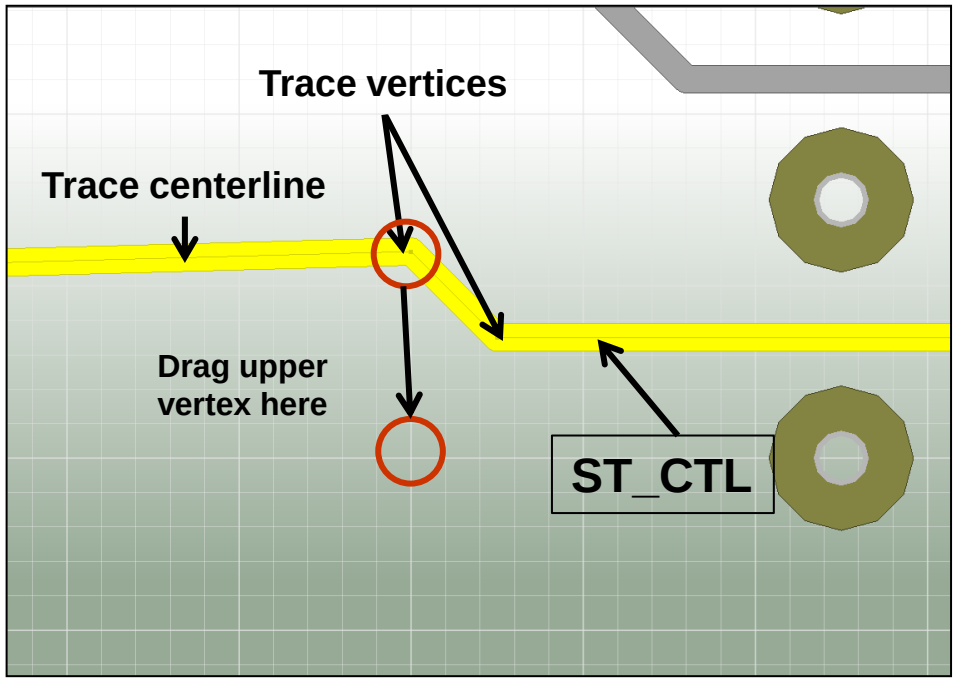
- Click on the ST_CTL trace. The ST_ERROR net will de-highlight.
- Click again on the ST_CTL trace and the trace vertices and centerline will become visible.
- Drag the upper vertex within Region 1 to the location shown, approximately coordinates (2850, 3210) mils.
- Click on the new location to place the vertex.

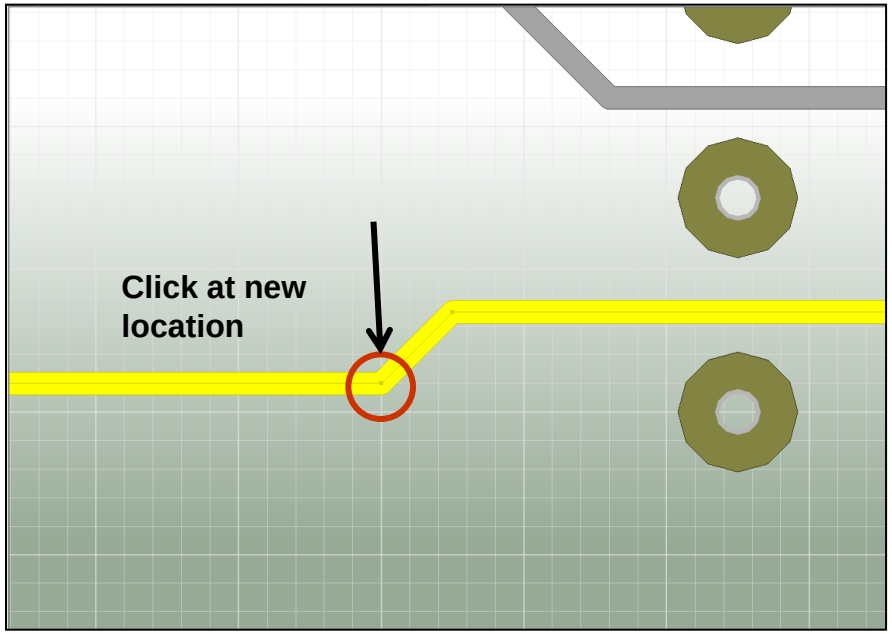




SYZ Parameters For Signal Integrity

- Zoom out, then zoom into Region 2, which has an upper left corner at approximate coordinates (4660, 3300) mils.
- Click on net ST_CTL to highlight the trace.
- Click on the trace again to display the trace vertices and centerline.
- Drag the upper vertex to the location shown in the figures, approximately (4700, 3210) mils.
- Click on the location to place the vertex.





SYZ Parameters For Signal Integrity

- **Re-run the simulation**
  - Form the menu ***Simulation,*** select ***Compute S-, Y-, Z-Parameters…***
    - Enter **SYZ Sweep 2** in the Simulation Name field.
    - Uncheck ***Export Touchstone file after simulation completes***
    - Keep the other settings the same as the first simulation
    - Click **Launch**

- **Plot the new results**
  - From the ***Results*** tab, Select ***SYZ > SYZ Sweep 2 > Plot Magnitude/Phase…***
    - Repeat the earlier process for selecting the four matrix entries in the list and click the **Create Plot in Electronics Desktop** button
    - The trace modification has slightly reduced the peak crosstalk. The frequency at which crosstalk first exceeds -32dB is roughly 1.2GHz in the original design, and 2GHz in the modified design.
    - Save the modified design
      - Click ***File > Save As…***
      - Filename***:*** **siwave_board2.siw**
      - Click **Save**

SYZ Parameters For Signal Integrity