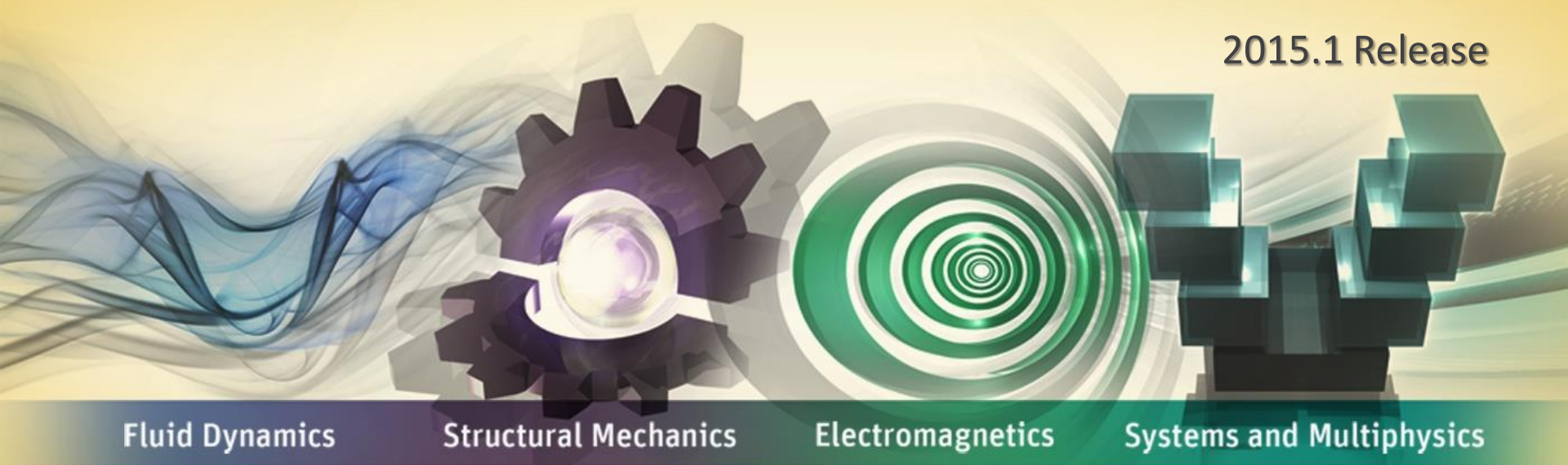


# Workshop 10: Near/Far-Field Analysis

2015.1 Release



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

## Introduction to ANSYS SIwave

- **Far Field Simulation**

- This example is intended to show how to setup and run a far field simulation using SIwave and Electronics Desktop. A differential driver will be driven by a pair of differential nets and a differential receiver will be used at the opposite end. We will look at the far field radiated from the PCB up to 10 MHz with a 1MB/s bit signal.
- The simulation is divided into 3 sections:
  - **SIwave** - Compute the S-parameter model for signal, power & ground nets.
  - **Electronics Desktop** - Run a transient (time domain) simulation of the overall circuit, which includes the driver, receiver, source, load, and the SIwave S-parameter model. Use push excitations to convert the time domain waveforms into the frequency domain sources in SIwave.
  - **SIwave** - Use the new frequency domain sources in SIwave so that we can compute the far field response.
- You will learn how to:
  - Setup and Compute the S-parameters.
  - Create a dynamic link SIwave model in ANSYS Electronics Desktop for our time domain simulation.
  - Generate the frequency domain source files.
  - Compute the far field response in SIwave.
  - Plot the far field result in dBuV/m at 3 meters.
- This chapter was written with the assumption that the user has a basic knowledge of SIwave and ANSYS Electronics Desktop

## Example – Far Field from PCB


- **ANSYS Siwave and Electronics Desktop Design Environment**
  - The following features of the ANSYS Tools Environment are used in this exercise
    - Pre-processing
      - FFT
    - Boundaries/Sources
      - Ports
      - PRBS Source
      - Driver and Receiver
    - Solutions
      - S-Parameters
      - Transient
      - Far Field
    - Plots
      - Transient Waveforms
      - Far Field in dBuV/m

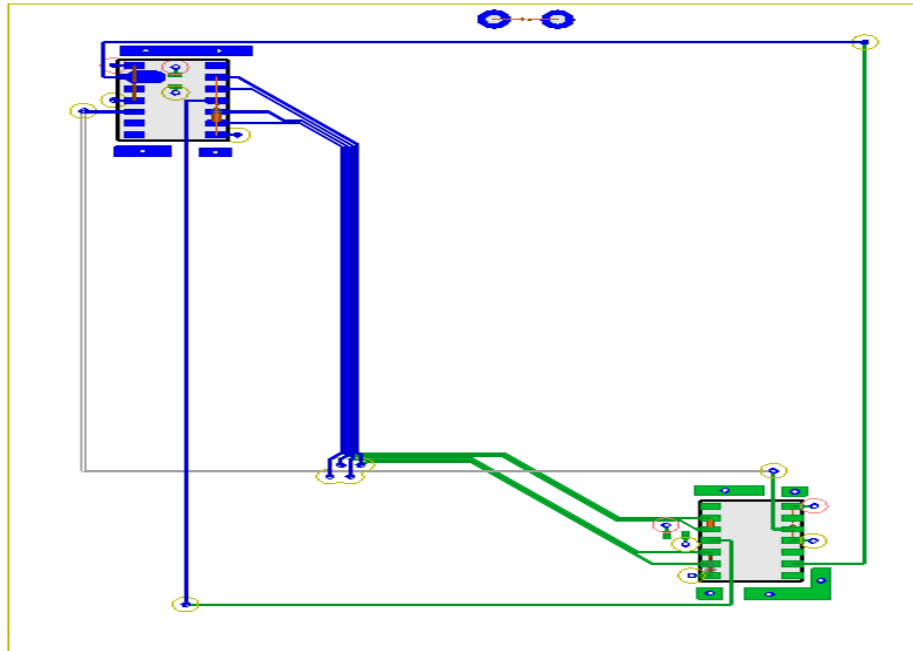
## Example – Far Field from PCB

- **Open Slwave**

- Select the Windows **Start** menu and select: **All Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.1 > ANSYS Slwave 2015.1**

- **Open a Slwave Project**

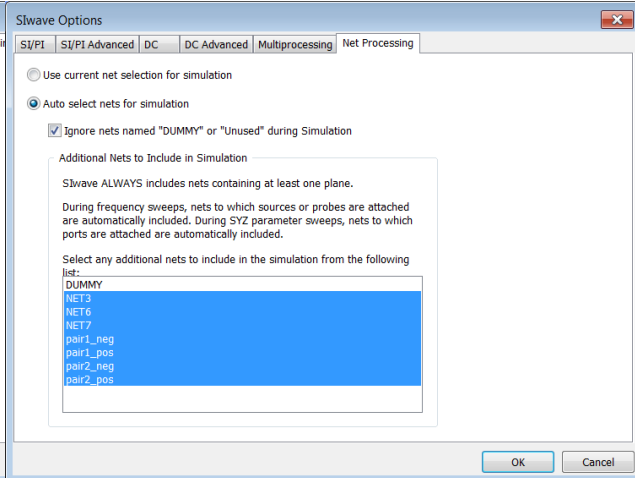
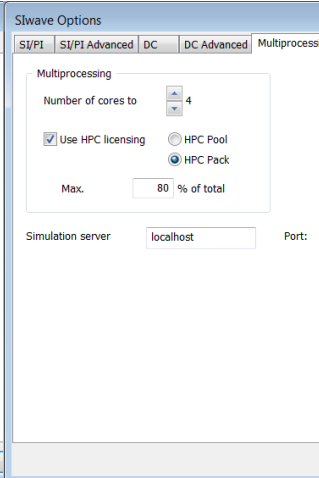
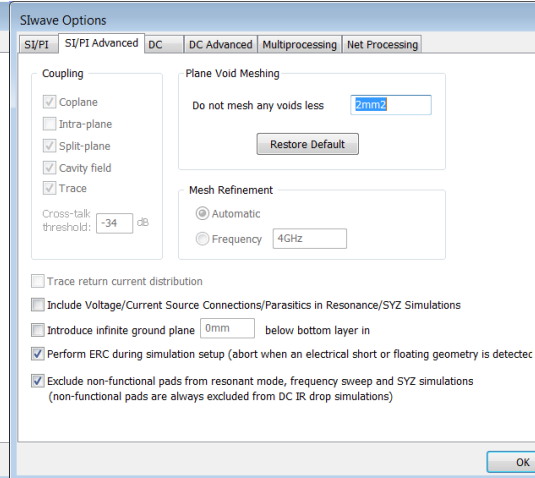
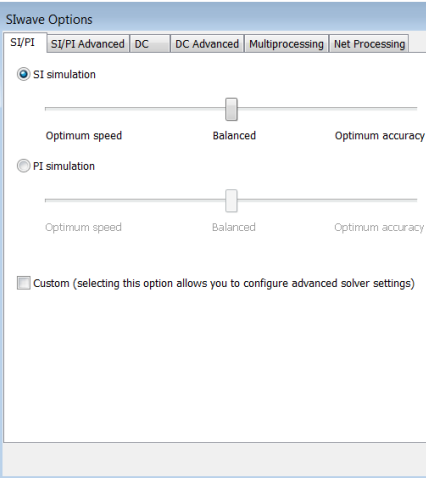
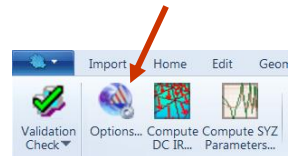
- Select the menu item **File > Open** or you can click on  found on the Quick Access Toolbar
  - Browse for file: **siwave\_emc\_farfield\_start.siw**,
  - Click **Open**
- **NOTE** - The file already has the s-parameters and far field results solved. This can be used to verify your work at a later time.



# Example – Far Field from PCB

## • Setting Simulation Global Options

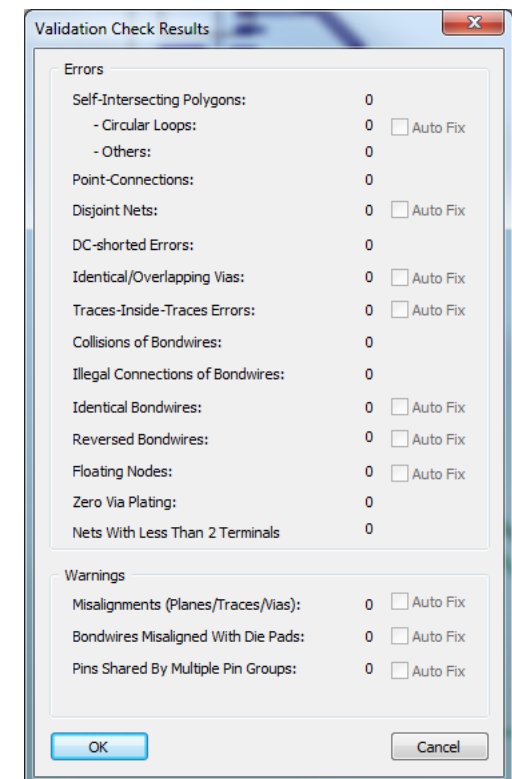
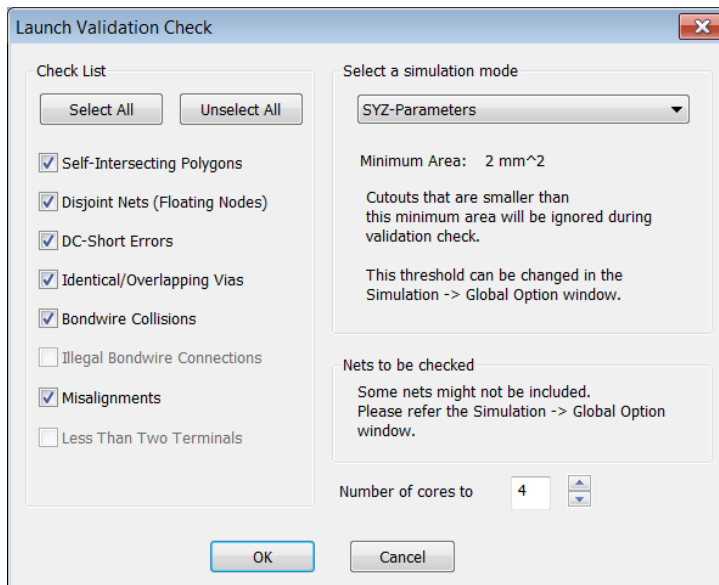
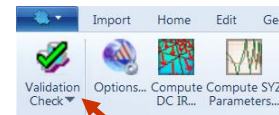
- Select the menu item **Simulation > S1wave Simulation Options**
  - Plane Void Meshing: **Automatically determine which voids to mesh**
  - Mesh Refinement: **Automatic**
  - Ignore nets named “DUMMY” or “Unused” during simulation: ☒ **Checked**
  - Perform ERC during simulation setup: ☒ **Checked**
  - Exclude non-functional pads...: ☒ **Checked**
  - Click **OK**
- **Note:** These are the default simulation settings.



## Example – Far Field from PCB

### • Validation Check

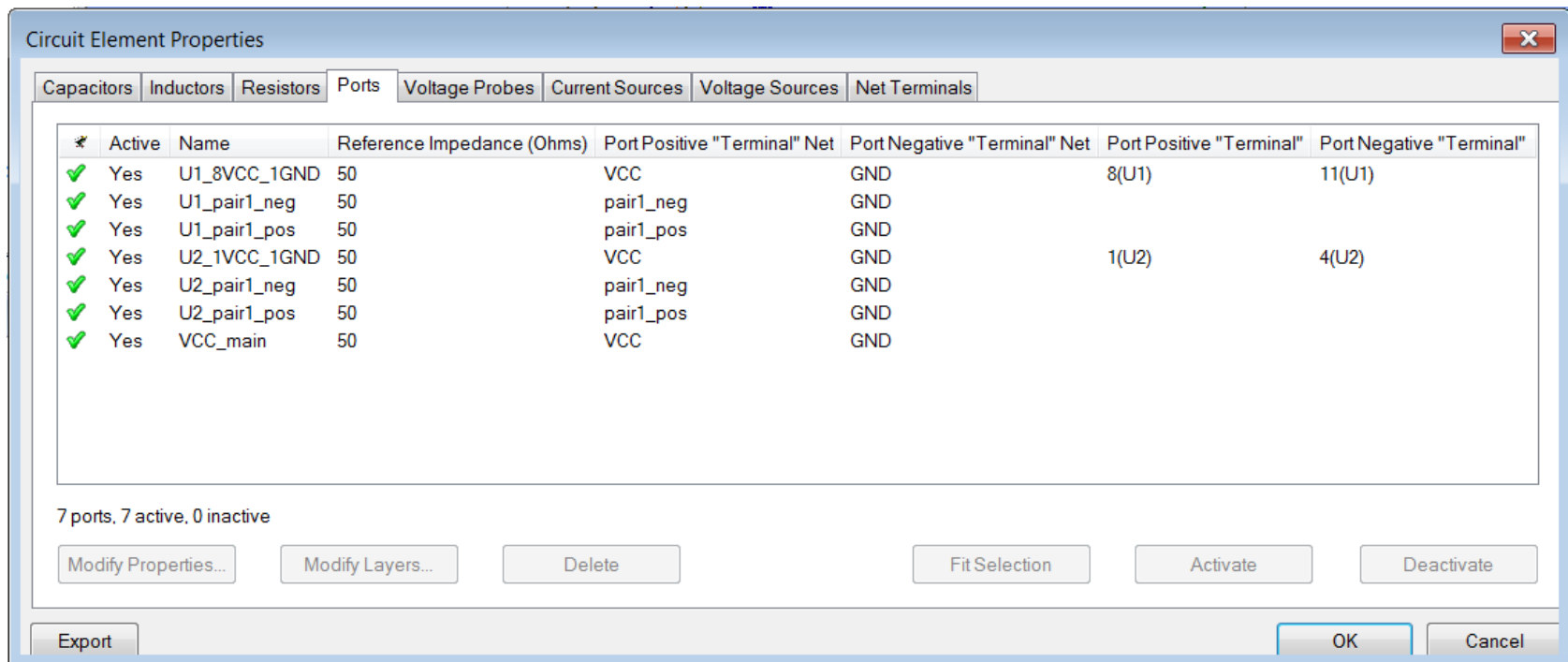
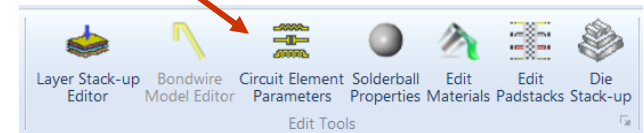
- It is a good idea to do a validation check before you start working on any design in SIwave for the first time
- The validation check tests for self-intersecting polygons, disjoint nets, overlapping (DC-shorted) nets and nets with overlapping vias.
- This helps you avoid finding layout errors after all the setting up ports and other solution settings.
- To run the validation check:
  - Select the menu item **Simulation > Validation Check**
    - Click **OK**
  - 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**



# Example – Far Field from PCB

## • Verify Ports

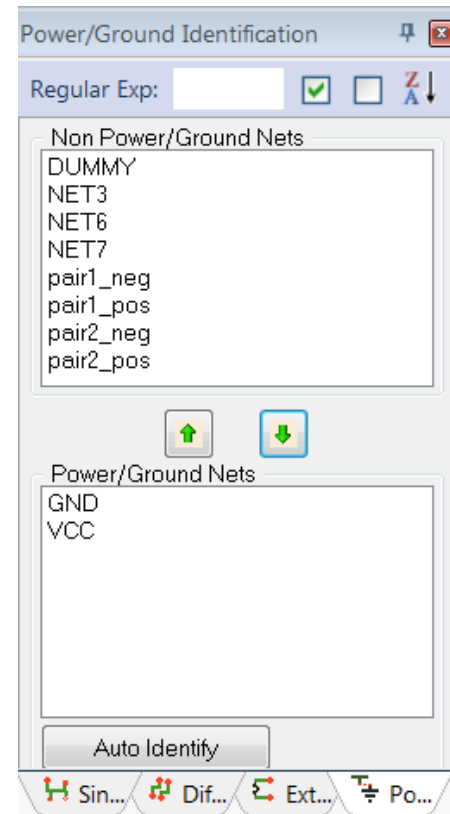
- Select the menu item **Home > Edit Tools > Circuit Element Parameters**
  - Click on Ports tab
    - Verify that you have 7 ports as shown below
  - Click the **OK** button



## Example – Far Field from PCB

### • Defining Power and 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**.

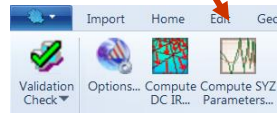




## Example – Far Field from PCB

### • Compute S-Parameters

- For Full-Wave SPICE export, the S-parameter model must be solved from DC up to the knee frequency (Fknee), where  $F_{knee} \approx 0.5/\text{rise\_time}$ .
- We also recommend an additional logarithmic sweep at low frequencies for better accuracy in the tail of the step response
  - For a 100 ps rising edge source, the knee frequency is 5 GHz
- Click **Simulation > Siwave > Compute S-, Y-, Z-Parameters...**
  - ☒ **Compute Exact DC point**
  - Row 1
    - Start Freq: **0 Hz**
    - Stop Freq: **0 Hz**
    - Num. Points: **1**
    - Distribution: **Linear**
  - Row 2
    - Start Freq: **1 kHz**
    - Stop Freq: **5 MHz**
    - Num. Points: **100**
    - Distribution: **By Decade**
  - Row 3
    - Start Freq: **5 MHz**
    - Stop Freq: **5 GHz**
    - Num. Points: **1001**
    - Distribution: **Linear**
  - Sweep Selection:
    - Select **Interpolating Sweep**
    - Error Tolerance: **0.001**
  - Click **Launch** to start the simulation



Compute SYZ-parameters

Sweep | Sensitivity | Distributed Analysis (HPC)

Simulation name: SYZ Sweep 1

☐ Compute exact DC point

Frequency Range Setup

	Start Freq	Stop Freq	Num. Points / Step Size	Distribution
1	0Hz	0	1	Linear
2	1kHz	5MHz	100	By Decade
3	5MHz	5GHz	1000	Linear

Add Above Add Below Delete Selection Preview...

Sweep Selection

☐ Discrete Sweep

☒ Interpolating Sweep

Relative error for S: 0.001

☐ Set FWS generation parameters

Min Rise/Fall Time / s: 1E-10

☐ Siwave with 3D DDM

Other solver options...

☒ Export Touchstone® file after simulation completes

File path: C:\Users\fecastro\Documents\Training\SIWave\_Intro Browse...


Save Settings Launch Close

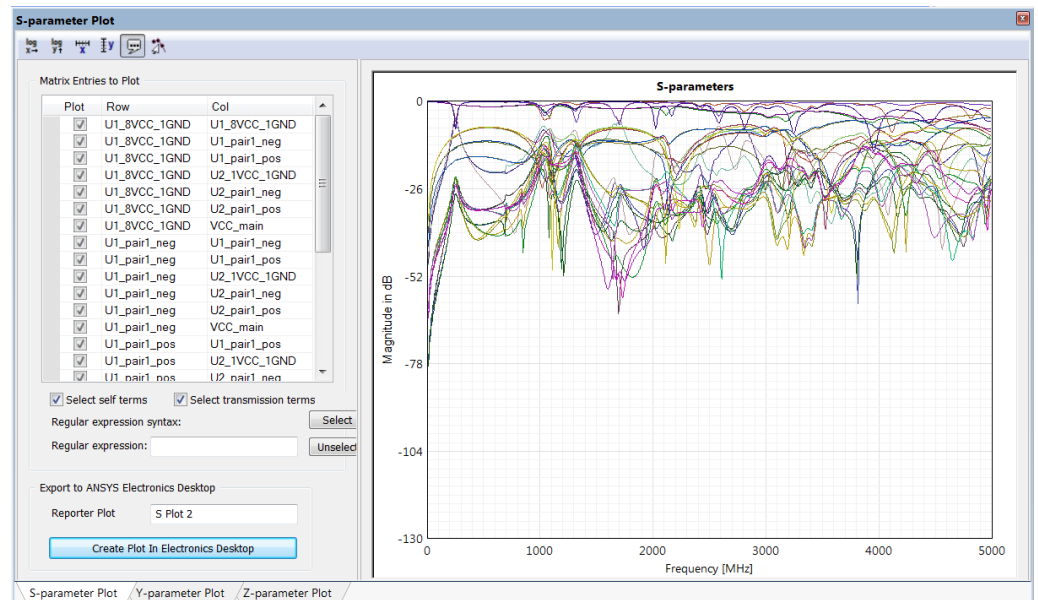
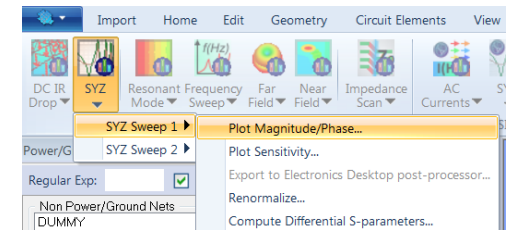
## Example – Far Field from PCB

### • Viewing the S-Parameters

- When the simulation is completed, you will see SYZ Sweep 1 in the Results panel.
  - Click **Results > Slwave > SYZ > SYZ Sweep1 > Plot Magnitude/Phase**
- Select the individual matrix entries or categories (self terms, transmission terms) you want to examine.
- Click **Create Plot in Electronics Desktop**. This will launch ANSYS Electronics Desktop where the plotted S-parameters are shown. You will also see the created plot in the Results panel: SYZ Plot 1
- Close plot window

### • Save Slwave Project

- Select the menu item **File > Save** or  
you can click on  found on the Quick Access Toolbar



# Example – Far Field from PCB

## • ANSYS Electronics Desktop

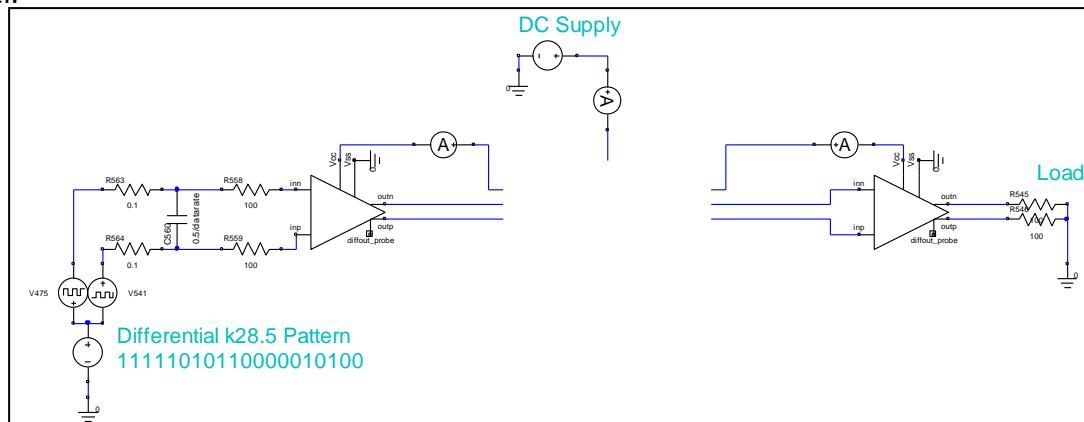
- We have just finished the first part of this exercise. We computed the S-parameters in Siwave and reviewed the results.
- Next, we are going to bring this S-parameter block into ANSYS Electronics Desktop so that we can run a transient simulation and then compute the FFT for each port of our channel model. Finally, the far field simulation will be setup and run in Siwave.

## • Open ANSYS Electronics Desktop Project

- Click the **Start** button then select:

**All Programs > ANSYS Electromagnetics > ANSYS Electromagnetics Suite 16.1 > ANSYS Electronics Desktop 2015.1**

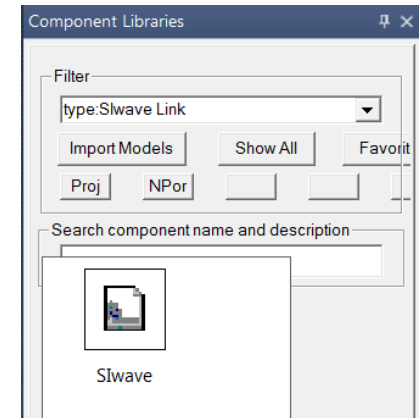
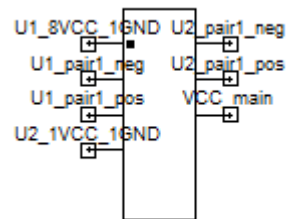
- Click **File > Open**, browse to the folder that contains **siwave\_ch6\_emc\_far\_field\_start.adns**
  - select **siwave\_ch6\_emc\_far\_field\_start.adns**
  - Click **Open**.
  - Since ANSYS Electronics Desktop is a new feature, it will be asked you to convert the **.adns** (format from ANSYS Designer) file to the **.aedt** (new format) file. You are able to keep your **.adns** file choosing **copy** option.
- This circuit already has the power supply, driver, and receiver defined. All that is needed to complete the schematic is to import the 7-port Siwave model.



# Example – Far Field from PCB

## • Add Siwave Design

- We are going to create a Dynamic Link to the Siwave model. This link allows ANSYS Electronics Desktop to keep track of the Siwave project for the latest results and push simulated currents and voltages back to Siwave for field simulations
- To create our Dynamic Link:
  - On the side bar **Component Libraries** type **Siwave Link**
  - Choose the symbol **Siwave**
  - Browse to the project directory, and select **siwave\_emc\_far\_field\_start.siw**
    - Click **Open**
- The symbol below will appear and you'll be able to add it to the schematic.

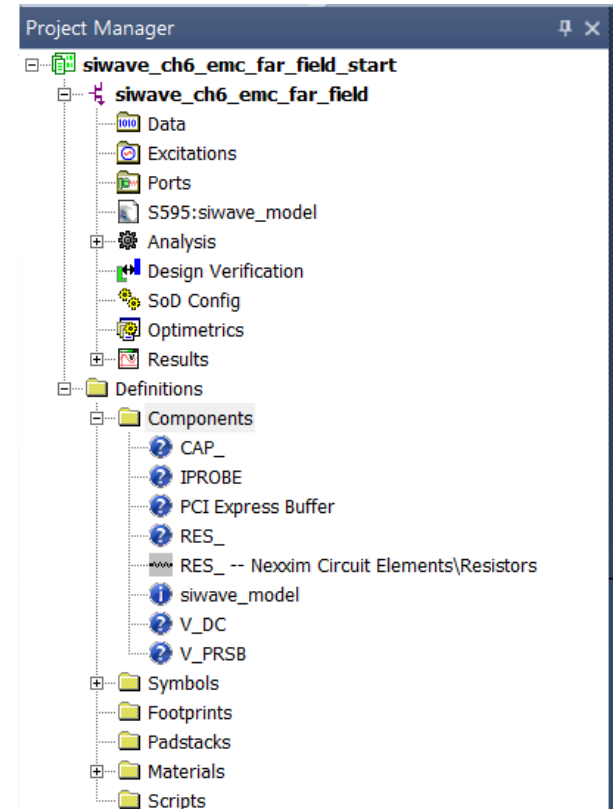
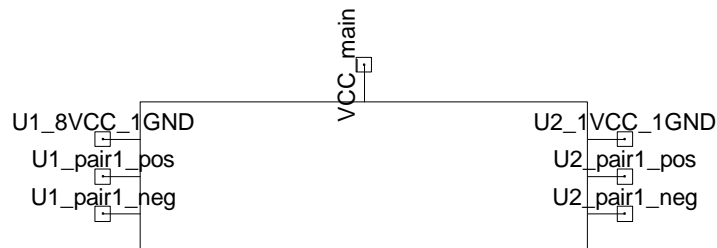


- Press the **ESC** key to end component placement mode
- If you need more than one of this component, you can search directly on the side bar for the name of the component you created (i.e. **siwave\_model**) and drag it to the schematic.
- You can also expand **Definitions > Components** in the project manager side bar and drag **siwave\_model** onto the schematic.

# Example – Far Field from PCB

## • Edit Component Symbol

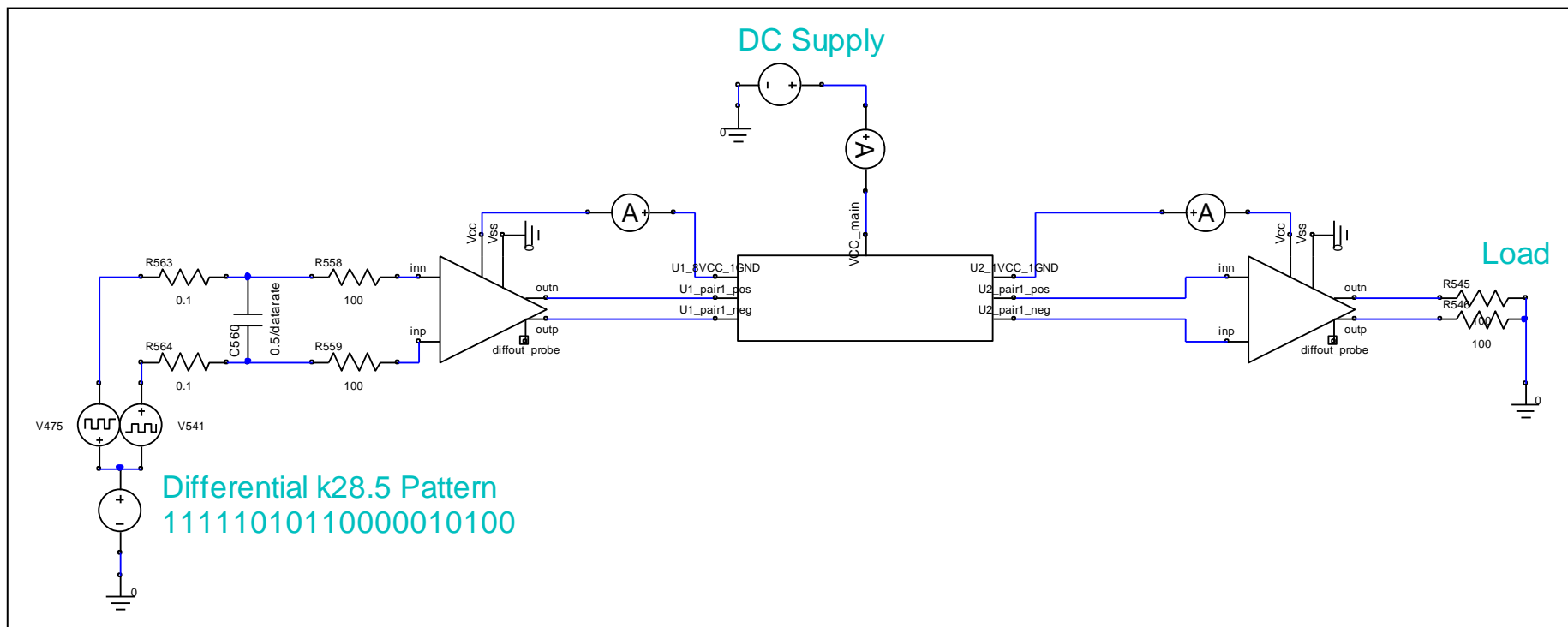
- Unfortunately the ports of the default symbol are not ordered in the way we want for the transient simulation
- We need to edit the symbol and re-order the ports. To edit the symbol:
  - Expand **Definitions > Symbols** in the Project Manager side bar
  - Double-click on **siwave\_model**
  - A symbol editor window will appear.
  - Edit the symbol and arrange the ports as shown below
  - Save the project to update the symbol
  - Close the symbol editing window by clicking on “X” at the top right hand corner of the subwindow.



## Example – Far Field from PCB

- **Add Siwave component**

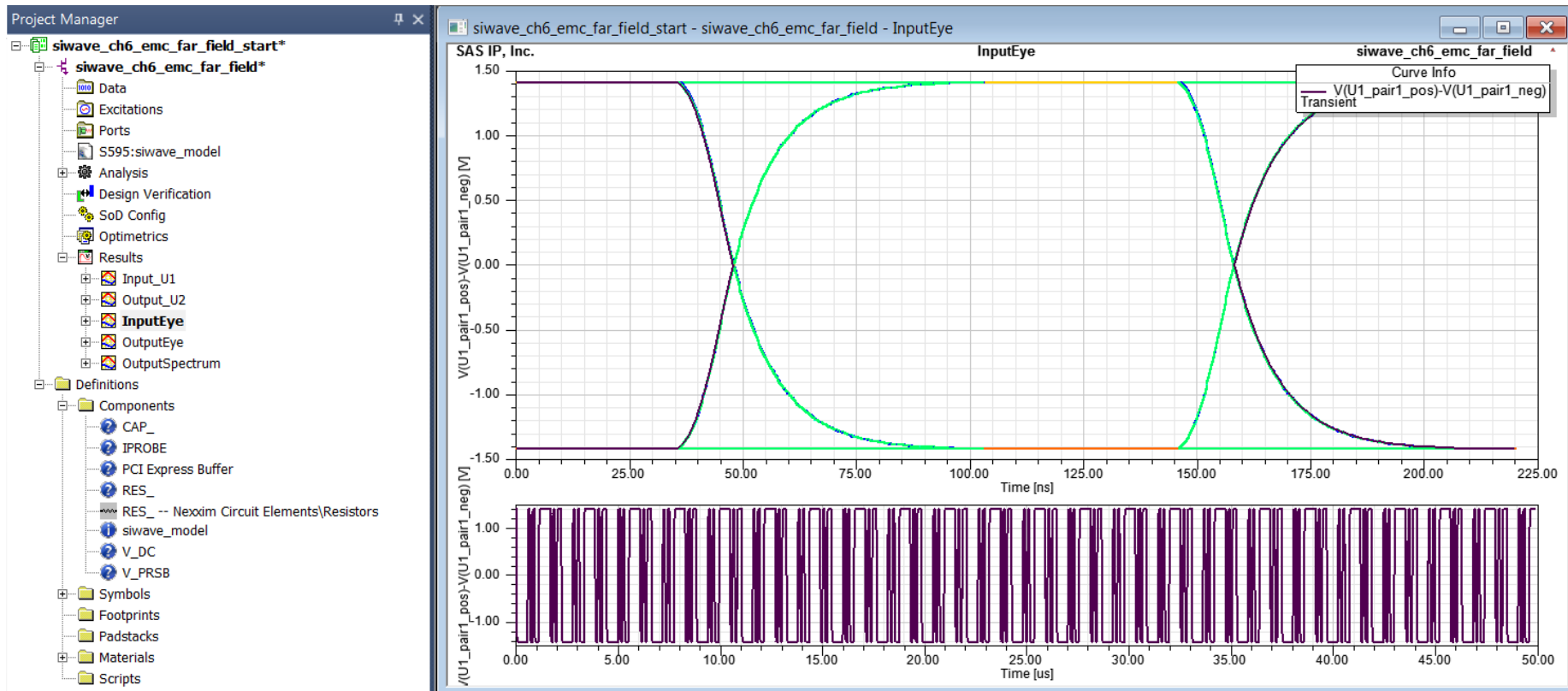
- In the schematic drag the instance of the siwave\_model component you created earlier and drop it into the rest of the circuit as shown below



# Example – Far Field from PCB

- **Run the Transient Simulation**

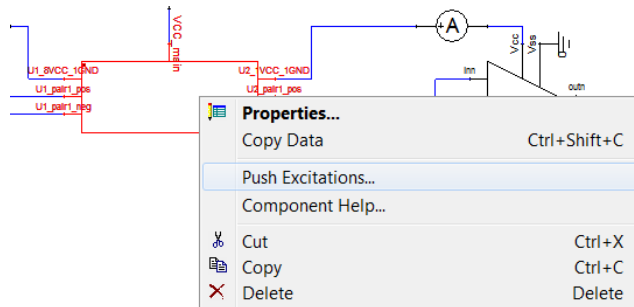
- To run the existing transient analysis
  - Select the menu item **Circuit > Analyze**
- When the simulation is done you can examine the results
- Double click on the plots under **Results** in the Project Manager to see some of the waveforms in this circuit



# Example – Far Field from PCB

## • Create Push Excitations Source Files

- Designer can “push excitations” to transfer the current and voltage waveforms at the boundary of the SIwave component back to SIwave for use in field simulations
- Right-click on the **siwave\_model** component in the Schematic and select **Push Excitations**
- For some products the user must provide FFT parameters when using Push Excitations, but for SIwave the raw time-voltage/current data is passed across the link and SIwave handles the FFT during field simulation
- Push Excitation Information
  - Solution: **Transient**
- Click **OK**
- Select the menu item **File > Save**
- Select the menu item **File > Exit**



The dialog box titled "Push Excitation Information" contains the following fields and options:

- Solution:** Transient (dropdown)
- Frequency:** (empty dropdown)
- Transient Parameters:**
  - Start:** 0s (dropdown)
  - Stop:** 10s (dropdown)
  - Max:** 100 (text input)
  - Window Type:** Hamming (dropdown)
  - Kaiser:** 0 (text input)
- Transient Spectrum Information:**
  - Maximum (text input)
- Buttons:** OK, Cancel

- We just concluded the second part of this exercise. We are now left with running a frequency sweep in SIwave using the above source files created with Push Excitations. When the sweep completes we will be able to plot the far field results.



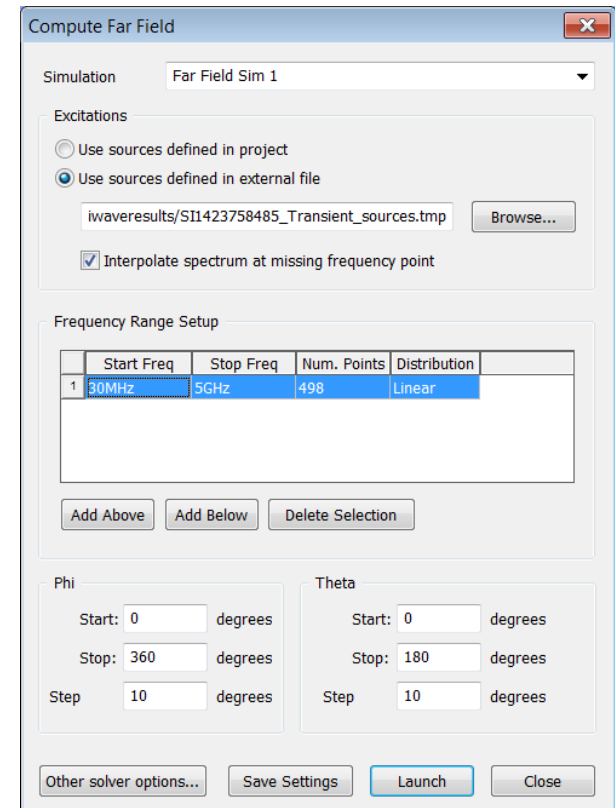
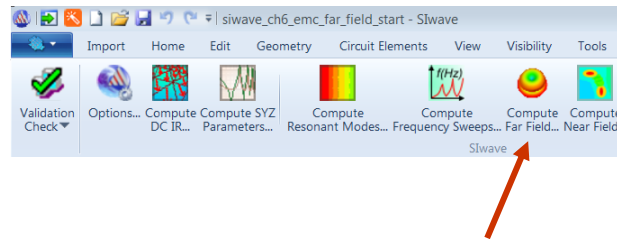
## Example – Far Field from PCB

### • Far Field Simulation

- If you don't have the SIwave project **siwave\_emc\_far\_field\_start.siw** opened, please re-open it.
- Click **Simulation > SIwave > Compute Far Field...**
- The option button **Use sources defined in external file** should be selected and pointing to a file
- This file contains pointers to the current/voltage data for each of the ports that was pushed from Designer

### • Frequency Range Setup

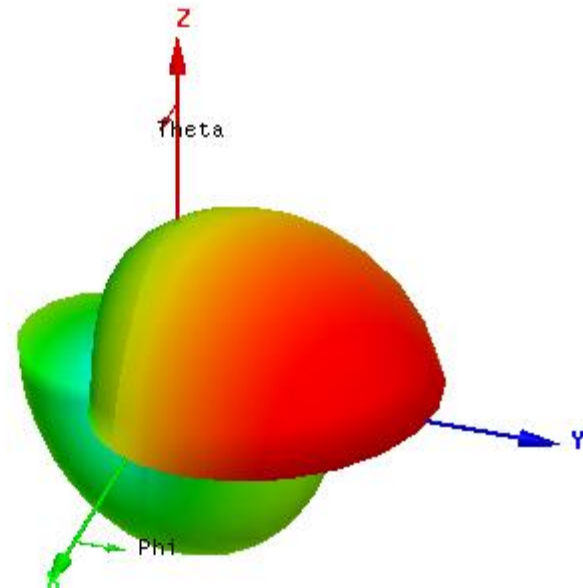
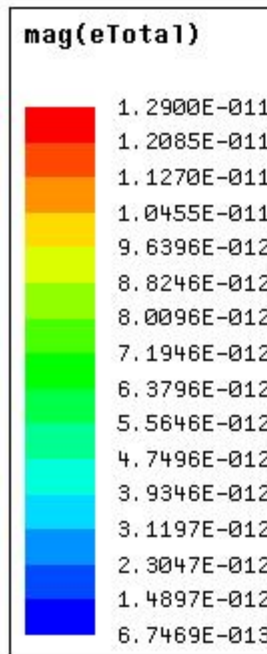
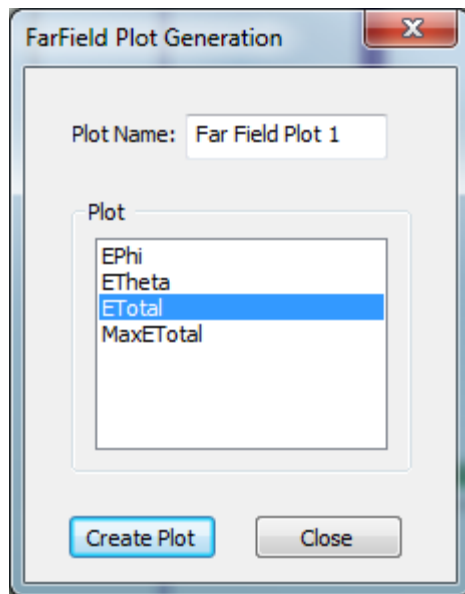
- Start Freq: **30 MHz**
- Stop Freq: **5 GHz**
- Num. Points: **498**
- Distribution: **Linear**
- **Phi**
  - Start: **0 degrees**
  - Stop: **360 degrees**
  - Step Size: **10 degrees**
- **Theta**
  - Start: **0 degrees**
  - Stop: **180 degrees**
  - Step Size: **10 degrees**
- Press the **Launch** button.



## Example – Far Field from PCB

### • Plot Far Field Results

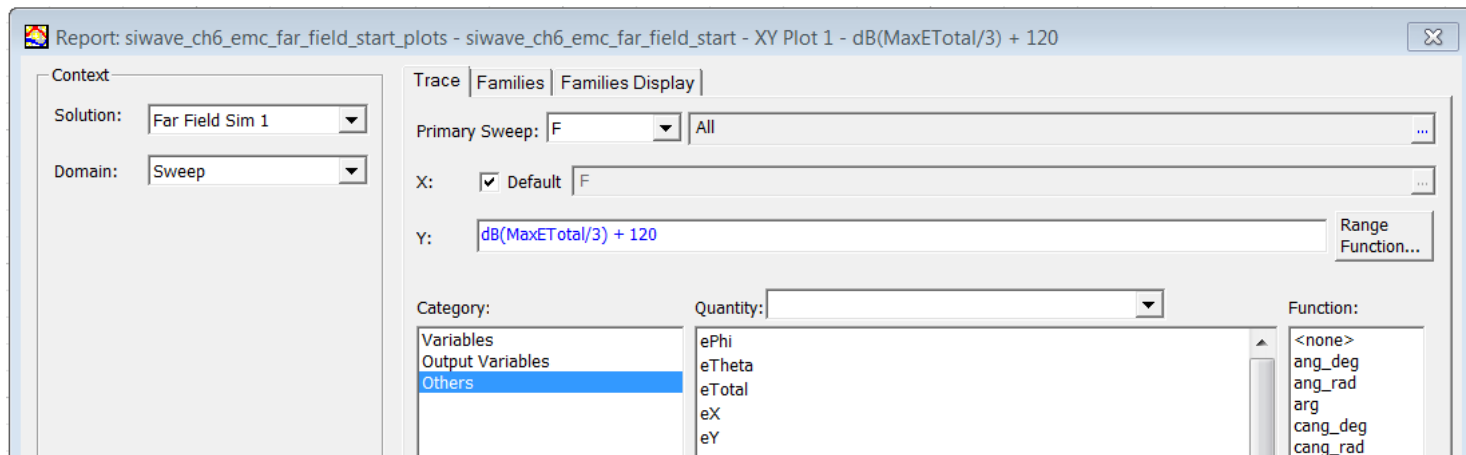
- To plot the far field EMI results
  - Select the menu item **Results > Slwave > Far Field > Far Field Sim 1 > Plot Far Fields**
  - FarField Plot Generation window
    - Select **ETotal**
    - Click **Create Plot**
  - Then ANSYS Electronics Desktop will be launched as below
  - Once Plot is create click on **Close** in FarField Plot Generation pop-up GUI



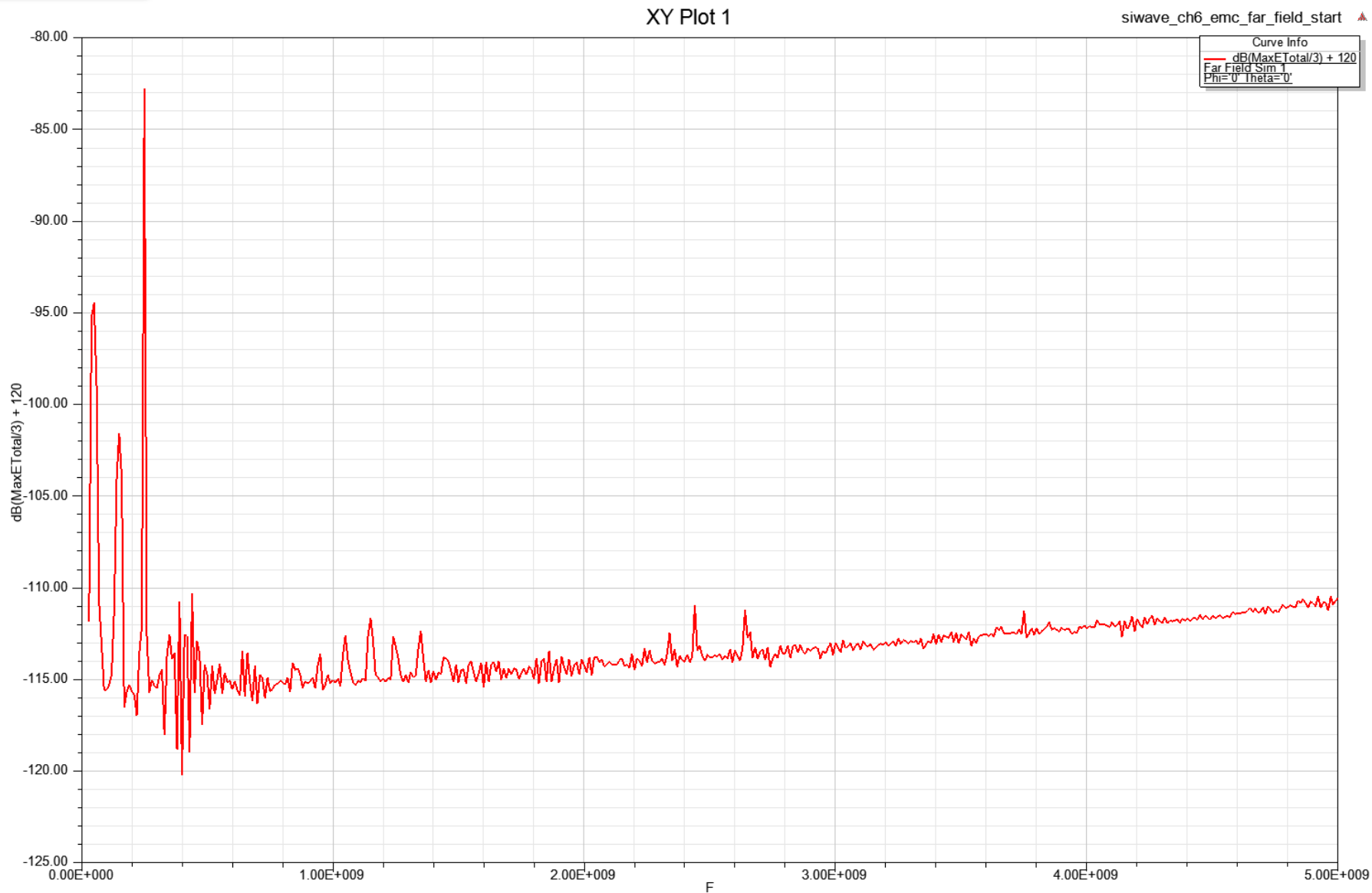
# Example – Far Field from PCB

## • Plot Far Field Results – Cont.

- In the **ANSYS Electronics Desktop**
- Select the menu item **Circuits > Results > Create Standard Report > Rectangular Plot**
  - Traces Window:
    - Primary Sweep: **F**
    - Category: **Others**
    - Quantity: **MaxEtotal**
    - Function: **dB**
    - Y axis:  **$\text{dB}(\text{MaxEtotal}/3) + 120$** 
      - This will give the plot units of  $\text{dB}\mu\text{V}/\text{m}$  at 3 m
    - Click the **New Report** button
    - Click the **Close** button



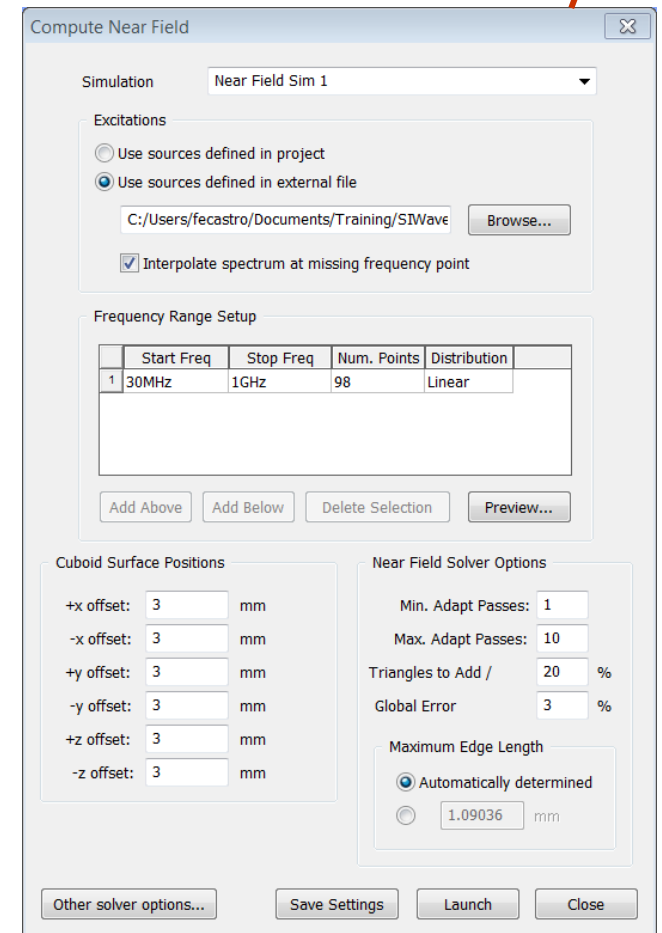
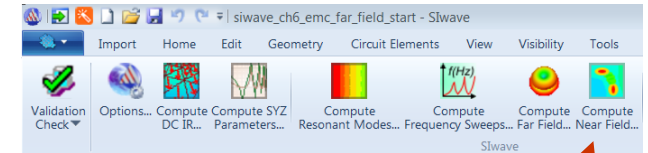
# Example – Far Field from PCB



# Example – Near Field from PCB

## • Near Field Simulation

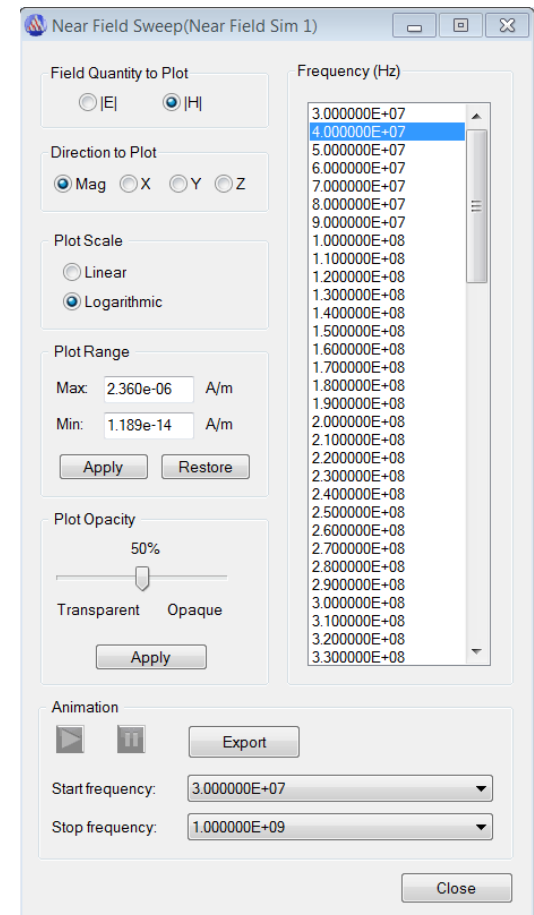
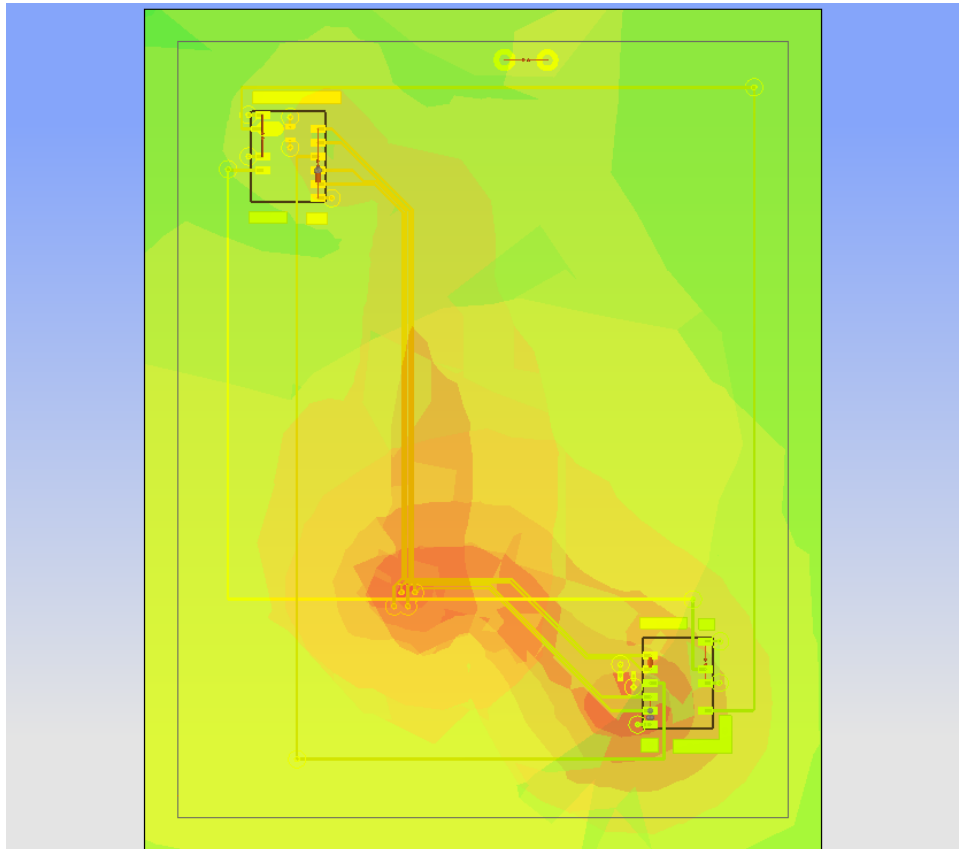
- Return to the SIwave GUI for the next steps
- To simulate Near Field
  - Select the menu item **Simulation > SIwave > Compute Near Field...**
  - Frequency Range Setup:
    - Start Freq: **30 MHz**
    - Stop Freq: **1 GHz**
    - Num Points: **98**
  - Cuboid Surface Positions:
    - +x offset: **3mm**
    - -x offset: **3mm**
    - +y offset: **3mm**
    - -y offset: **3mm**
    - +z offset: **3mm**
    - -z offset: **3mm**
  - Near Field Solver Options
    - Min Adapt Passes: **1**
    - Max Adapt Passes: **10**
    - Maximum Edge Length: **Automatically determined**
  - Click **OK**



# Example – Near Field from PCB

- **Near Field Simulation Continued:**

- When the simulation is complete select : **Results > SIwave > Near Field > Near Field Sim 1 > Plot Fields**
- A window appears that allows you to select either the E- or H-field magnitude to plot as well as the frequency at which the near-field is plotted
- Animation capabilities are also provided for sweeping through the solved frequencies



**This page intentionally left blank**

**Visit us at [www.ansys.com](http://www.ansys.com)**