PowerSI Tutorial

Product Version 16.6
January 2014

Document Updated on: November 5, 2013

Table of Contents

1 Preface

This tutorial provides all information required when you start using **PowerSI**.

This Preface has the following sections:

- Overview
- Who Should Read This Tutorial
- Additional Documentation
- Customer Support Information

1.1 Overview

This tutorial has the following chapters.

1.2 Who Should Read This Tutorial

PowerSI is intended to be used by engineers and researchers who work in the areas of design, modeling, and simulation of integrated circuits, chip carriers, and printed circuit boards (**PCB**). Users should have at least an entry-level experience in **SPICE** circuit simulations, and be aware of power and signal integrity issues in high-speed or high-frequency electronic systems.

1.3 Additional Documentation

For more information about **PowerSI**, see the following documents:

- ***PowerSI Getting Started Guide***
- ***PowerSI Tutorial***
- ***PowerSI_Chip_Cosimulation_Tutorial***
- ***Q&A***
- ***SPD_File_Format_RG***
- ***PowerSI User's Guide***
- ***Translators User's Guide***

1.4 How to Contact Technical Support

If you have questions about PowerSI, contact the *Cadence Online Support*.

2 Introduction

This chapter introduces **PowerSI** briefly and other related information.

This chapter covers the following sections:

- Software Overview
- PowerSI Workplace
- PowerSI and SPEED2000
- PowerSI and Broadband SPICE
- PowerSI Workflow

2.1 Software Overview

2.1.1 What's PowerSI

PowerSI is a new generation simulation tool to analyze electrical integrated-circuit (IC) packages and printed-circuit boards (PCB) for power and signal integrity.

It can provide full-wave analysis results quickly and accurately. This feature allows it to overcome tough high-speed design issues in power, ground, and signal integrity.

PowerSI uses frequency domain effectively in analysis of package and board performance due to its electromagnetic field computation engine.

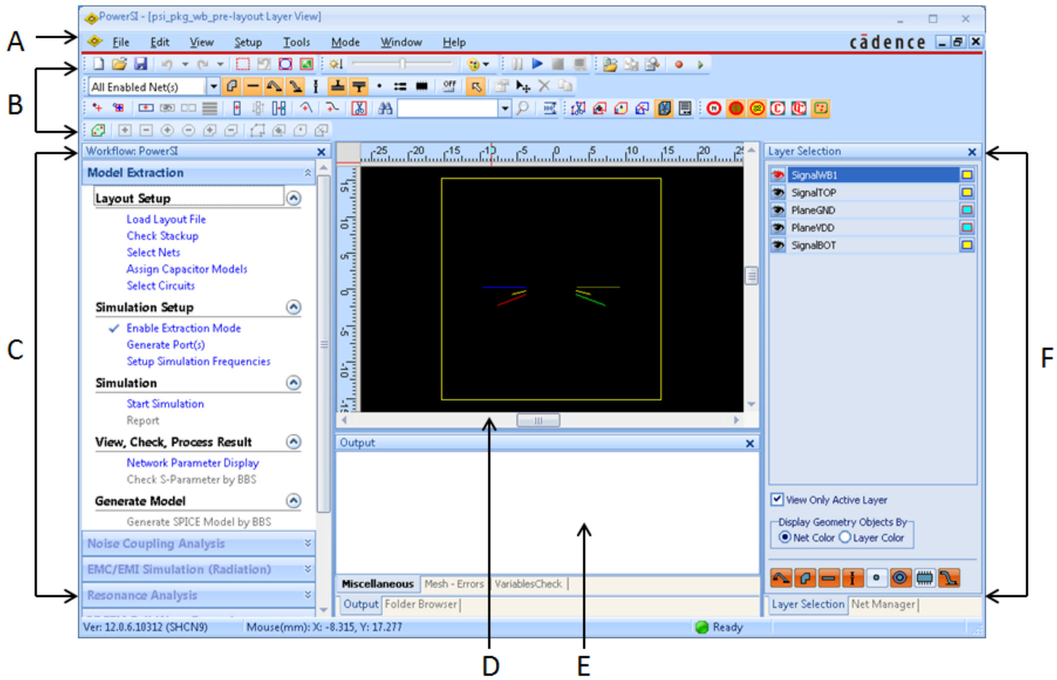
2.1.2 Applications

PowerSI can be used for analysis and design of electronic packages including chip carriers and PCBs. It is particularly effective for the following applications:

- Pre-layout and post-layout electrical analysis for entire packages and boards.
- Package and board resonance identification.
- Frequency-dependent impedances extraction for power ground systems.
- Decoupling capacitor placement optimization.
- Frequency-dependent S, Z, and Y network parameters determination.
- Signal return path discontinuities analysis.
- Electromagnetic coupling evaluation for planes, traces, and vias.
- What-if comparisons to optimize electrical performance.

2.2 PowerSI Workplace

This illustration indicates major parts of the workplace when you open a file in **PowerSI**. See ***PowerSI User's Guide*** for detailed information about menus, toolbars, and workflow pane.



A. Menu bar B. Toolbars C. Workflow pane

D. Show area E. Output/Folder Browser/TCL Reader pane

F. Layer Selection, Net Manager, or Circuit/Linkage Manager

2.3 PowerSI and SPEED2000

PowerSI and **SPEED2000** are complementary signal and power integrity. They share the same patented technologies of efficient full-wave analysis of complex packages and PCBs.

PowerSI and **SPEED2000** use unique field computation methodologies to perform analysis for an entire package or board with full-wave accuracy, allowing fine numerical meshes for the modeling of detailed structures, such as cuts and slots in planes, multiple power and ground layers, and any number of vias and traces.

PowerSI specifically focuses on frequency domain analysis while **SPEED2000** specializes in time domain. **PowerSI** and **SPEED2000** share the same data file and have a very similar user interface environment.

2.4 PowerSI and Broadband SPICE

The **Broadband SPICE** generates **HSPICE** and general **SPICE**-compatible equivalent circuit models for a passive N-port network characterized by its S, Z, and Y parameters.

The computation of N-port parameters for the package and board used by **PowerSI** can help users acquire accurate **SPICE**-compatible equivalent circuits. And these circuits can be used for time-domain simulations with other passive and active components, such as transistor level driver models in **SPICE**-like circuit simulators.

The second project in this tutorial describes the **PowerSI-Broadband SPICE** flow for I/O modeling and transient simulation. See ***Chapter 3 Project 2 Modeling and Transient Simulation***.

2.5 PowerSI Workflow

PowerSI has an integrated environment for graphically editing physical structures, and creating and editing circuits. The environment can build packages or boards for pre-layout analysis.

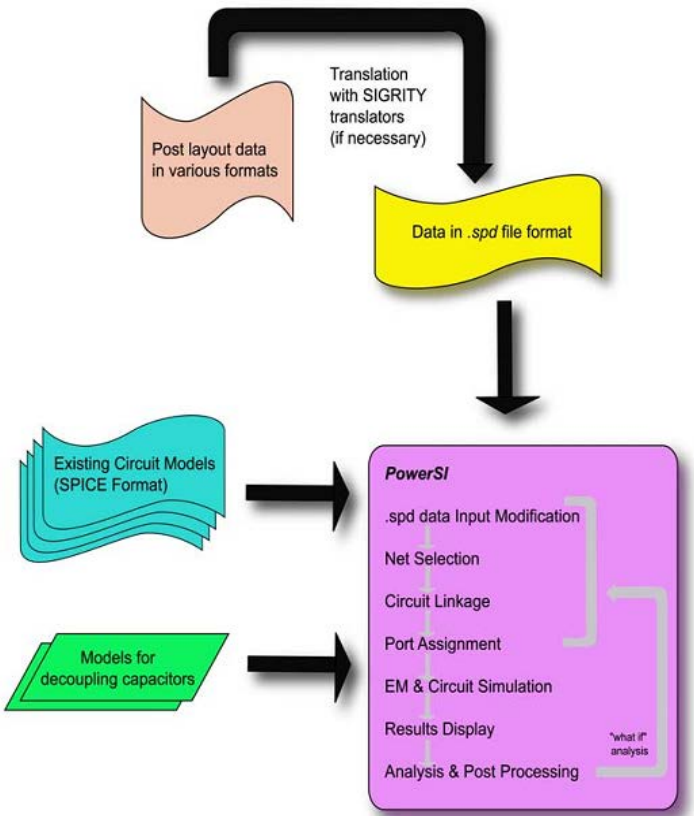
Many translators are developed to convert data files from other vendors to Sigrity's data file `.spd`. So you may use any data files generated under other tool environments to do the post-layout analysis of packages and boards.

PowerSI has three simulation modes: **Extraction Mode**, **Spatial Mode**, and **Resonance Mode**.

2.5.1 Extraction Mode

In the extraction mode, **PowerSI** can easily and conveniently extract S, Z, and Y parameters of multiple ports specified- for package and board structures, such as a power delivery system.

The following figure shows a typical workflow of **PowerSI** in the extraction mode:

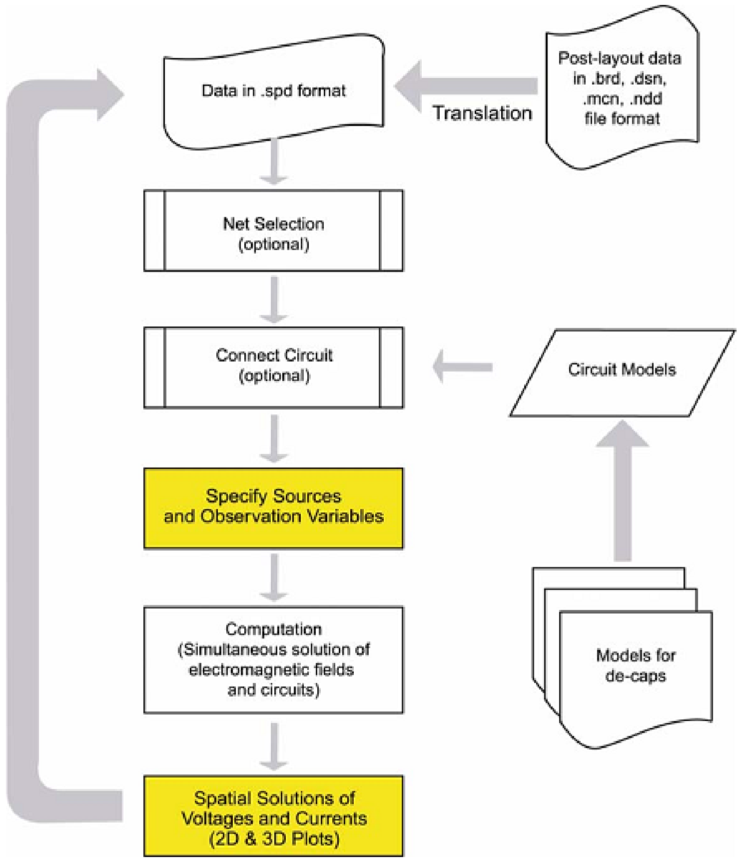


2.5.2 Spatial Mode

In the spatial mode, **PowerSI** can do AC analysis for circuits with many sources and obtain the following results:

- Spatial solutions of voltage distribution across planes.
- Voltage and currents in circuit components.
- Voltage and currents in physical structures such as traces and vias.

The following figure shows a typical flow of **PowerSI** in the spatial mode:



2.5.3 Resonance Mode

The resonance mode solver computes and displays all resonant modes between predefined frequency ranges. User can also specify the number of resonant modes directly.

PowerSI outputs a table which lists complex resonant frequencies and Q factors in the resonance mode.

3 Project 1 Impedance of Power Delivery System

This chapter describes how to use **PowerSI** to simulate the frequency-dependent power-ground impedance profiles at specified power and ground pin locations.

This chapter takes two files psi_brd_demoshort.spd and psi_brd_demodecaps.spd as examples to demonstrate how to use **PowerSI**. The two files are within the installation directory.

The chapter has the following sections:

- About the Examples
- Power Ground System Without Decoupling Capacitors
- Power Ground System With Decoupling Capacitors

3.1 About the Examples

Both psi_brd_demoshort.spd and psi_brd_demodecaps.spd have the same board with six metal layers including one power plane and one ground plane.

For psi_brd_demodecaps.spd, the board has decoupling capacitors mounted. The board dimensions are about 10 x 6 cm with FR4 dielectric material.

3.2 Power Ground System Without Decoupling Capacitors

This section takes psi_brd_demoshort.spd as an example to explain how to use **PowerSI**.

This section has the following topics:

- Start PowerSI and Load a File
- View Port Setup
- Set up Simulation Frequencies
- Run a Simulation
- View Simulation Report

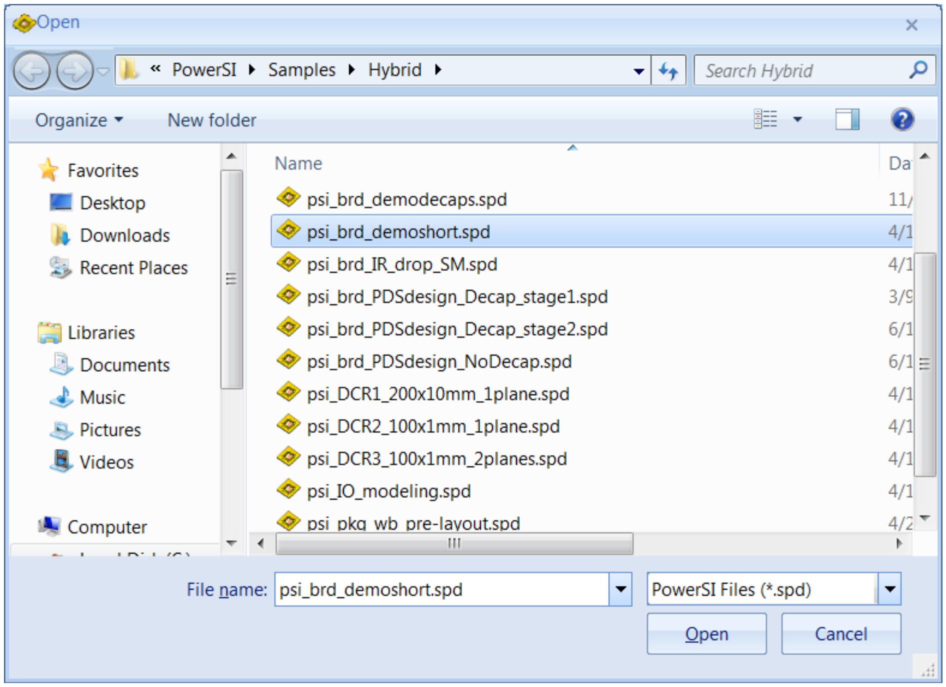
3.2.1 Start PowerSI and Load a File

Follow these steps to start **PowerSI** and load a file:

1. Launch **PowerSI.**
2. Select Mode > Extraction Mode.

   Extraction Mode is selected by default when PowerSI is launched. You can double-check to ensure the mode is correctly enabled.
3. Select File > Open….

   The **Open** dialog box opens.

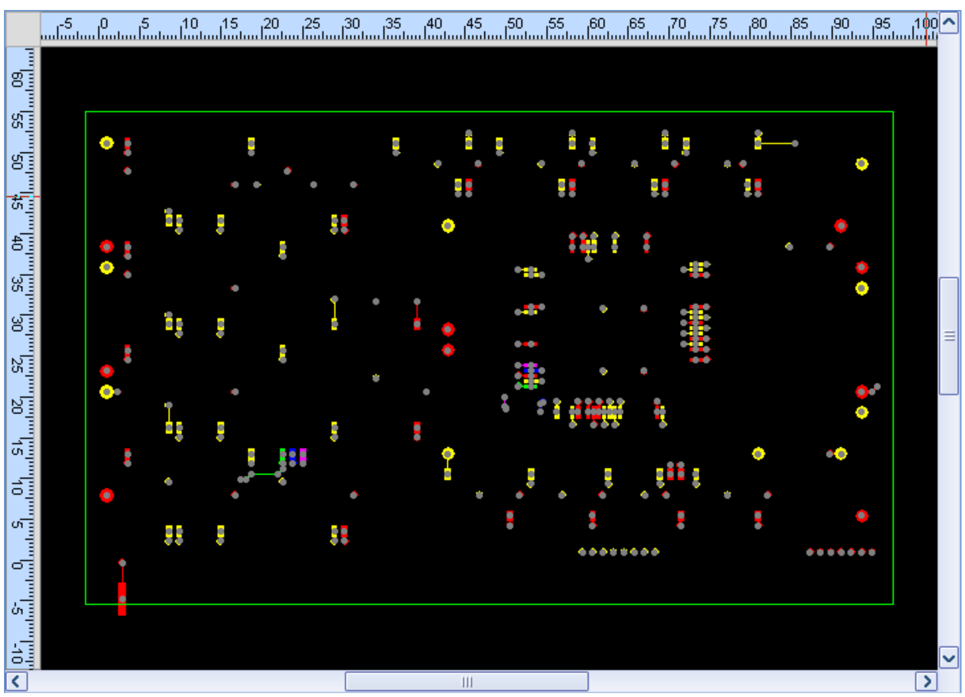


4. Double-click psi_brd_demoshort.spd.

or

Choose the file and click **Open**.

A 2D layer view displays.



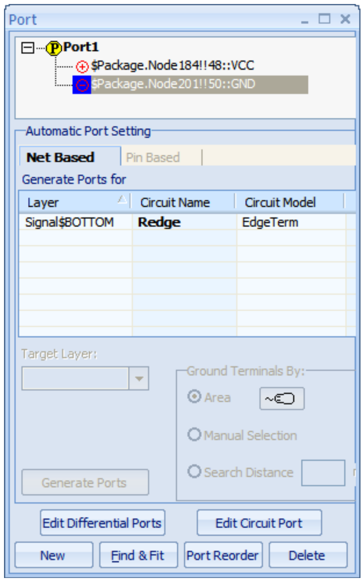
In this example, the circuit connects to an edge connector at the lower right corner of the board. The circuit has only one 0.01 Ω resistor which short circuits the power and ground pins of the edge connector.

3.2.2 View Port Setup

Follow these steps to check the setup:

1. Select Setup > Port….

   The **Port** setup dialog box opens.

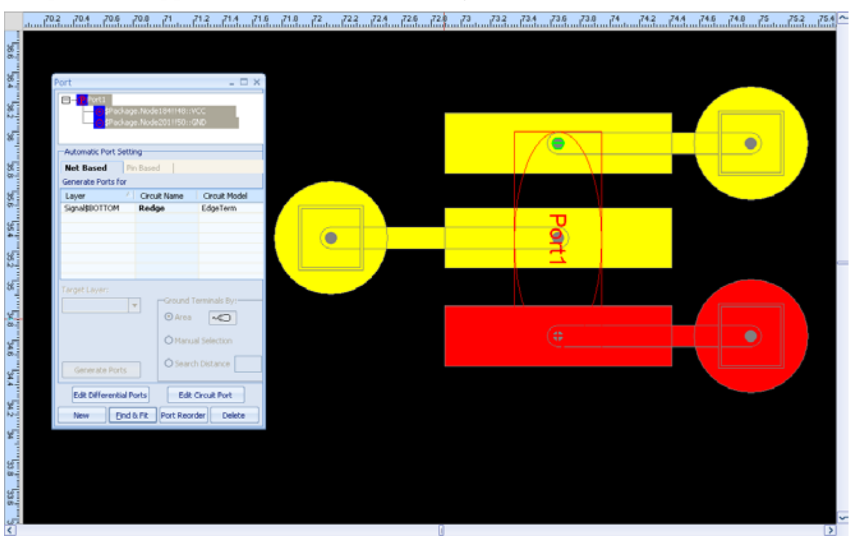


   This example has one predefined port.

   - The port positive terminal is connected to the package node — **Node184!!48::VCC**.
   - The port negative terminal is connected to the package node — **Node201!!50::GND**.

2. Select **Port 1** and click **Find & Fit**.

   The port area appears in the center of the layer view window.



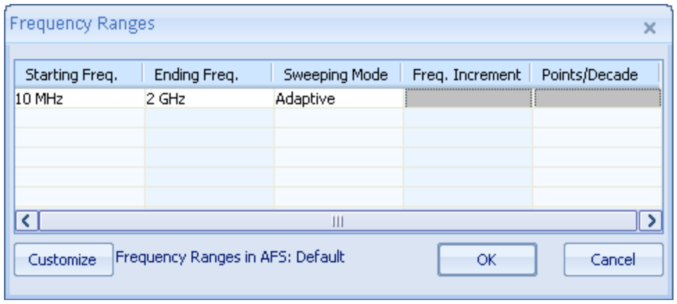
3. Click the **Fit** button  on the **Main Toolbar** to return to the overall view.
4. Close the **Port** dialog box.

3.2.3 Set up Simulation Frequencies

Follow these steps to set up simulation frequencies:

1. Select Setup > Simulating Frequencies….

   The **Frequency Ranges** dialog box opens.



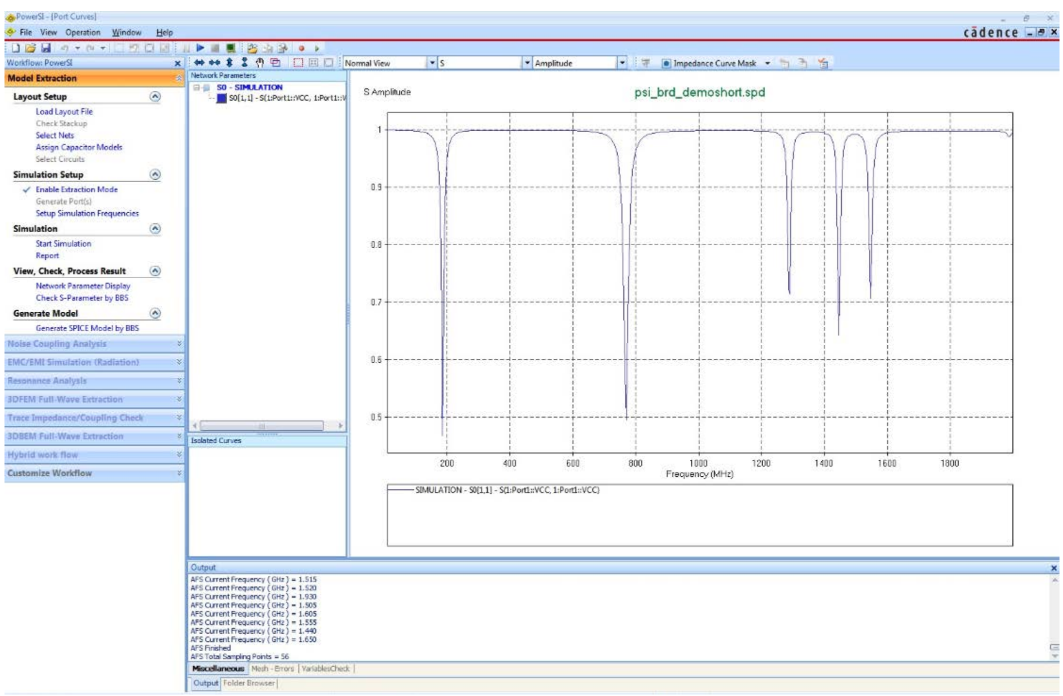
2. Change the following parameters in this window:
   - **Starting Freq.** to 10 MHz (or 0.01 GHz)
   - **Ending Freq.** to 2.0 GHz
   - **Sweeping Mode** to **Adaptive** from the drop-down menu.
3. Click **OK** to finish the setup.

3.2.4 Run a Simulation

Follow these steps to perform a simulation:

1. Click the **Start Simulation** button to start the frequency sweeping simulation for the network parameters at the port location.

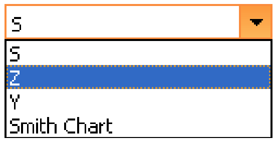
   An S-parameter displays first as follows.



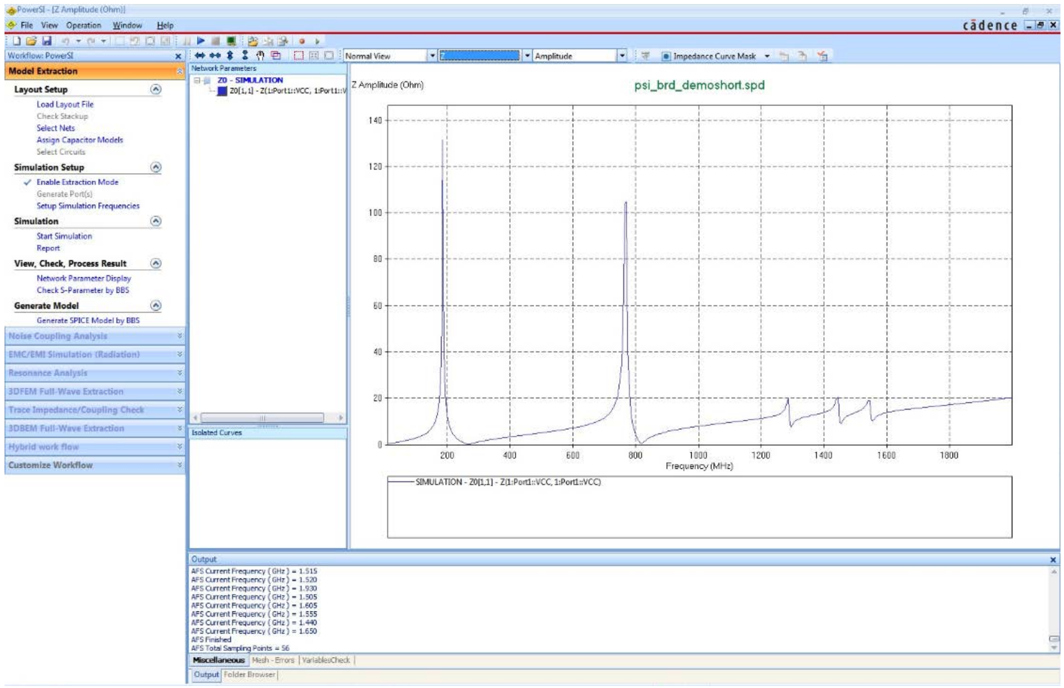
The **Output** window shows the process status during the simulation.

The **Status** bar shows **Computing completed** after the simulation is done.

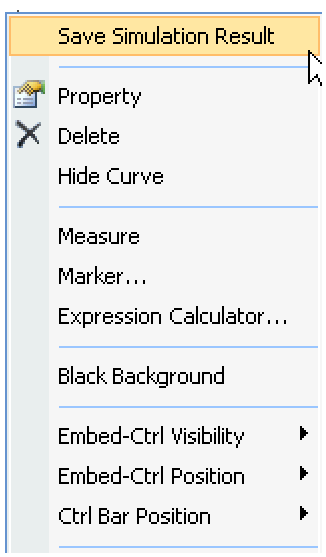
2. Click and choose the **Z** (impedance) parameter from the drop-down menu.



A Z-parameter curve displays as follows.

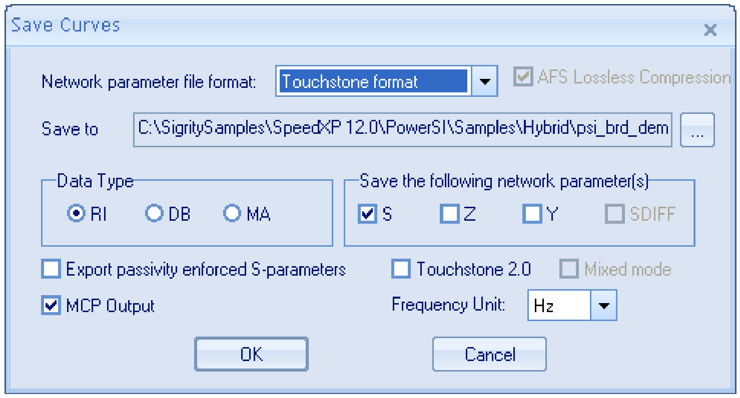


3. Right-click the plot to open a pop-up menu.



4. Select **Save Simulation Result**.

The **Save Curves** dialog box opens.



5. Select **Touchstone format** from the **Network parameter file format** drop-down menu and rename the file to psi_brd_demoshort.S1P .
6. Click **OK**.
7. Close the project without saving it.

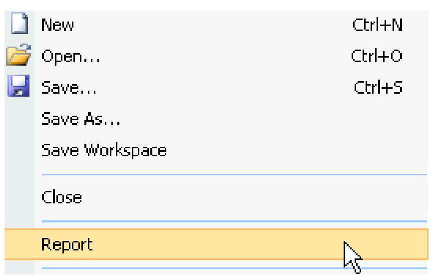
Tip!	Double-click a curve to show the **Curve Pattern Property** box. You can change **Style**, **Pattern**, **Width**, and **Mark** for the curve. See ***2.3.4 Run a Simulation*** for the details.

3.2.5 View Simulation Report

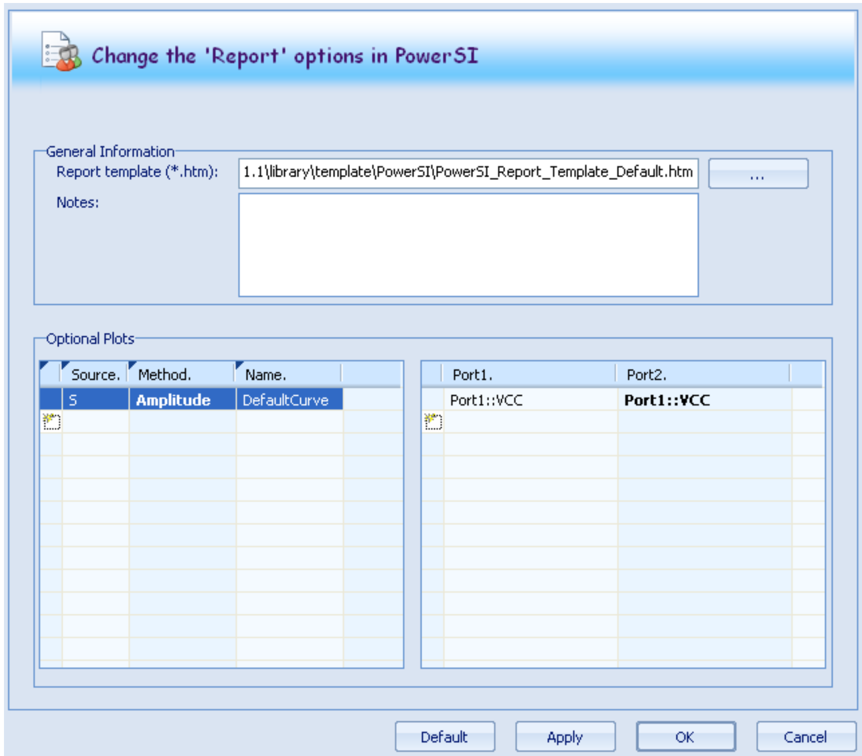
You can view the simulation report after running a simulation.

Follow these steps to open the simulation report window:

1. Click  to return to the **Layer View** window.
2. Select File > Report.



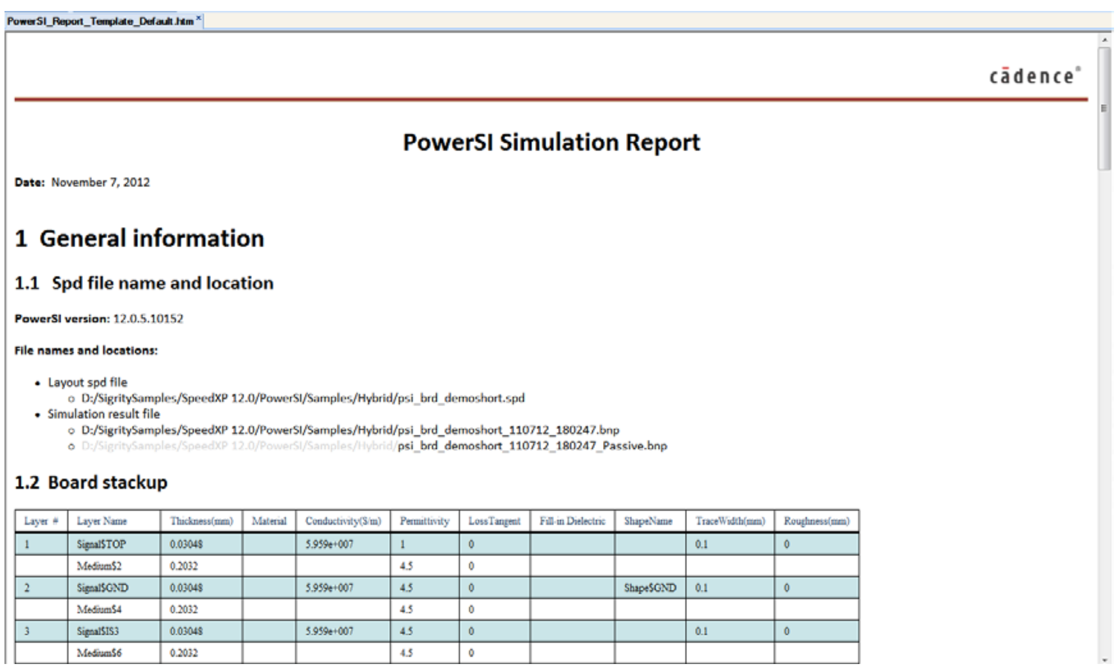
The **Simulation Report Setting** window opens.



NOTE!	You can select Tools > Options > Edit Options… > Simulation (Basic) > Report to open the window, too.

3. Click **OK** directly, or set up the **Simulation Report Setting**.

   The **Simulation Report** window appears as shown below:



PowerSI_Report_Template_Default.htm

cādence

PowerSI Simulation Report

Date: November 7, 2012

1 General information

1.1 Spd file name and location

PowerSI version: 12.0.5.10152

File names and locations:

- Layout spd file
  - D:/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort.spd
- Simulation result file
  - D:/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.bnp
  - D:/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247_Passive.bnp

1.2 Board stackup

Layer #	Layer Name	Thickness(mm)	Material	Conductivity(S/m)	Permittivity	LossTangent	Fill-in Dielectric	ShapeName	TraceWidth(mm)	Roughness(mm)
1	Signal$TOP	0.03048		5.959e+007	1	0			0.1	0
	Medium$2	0.2032			4.5	0				
2	Signal$GND	0.03048		5.959e+007	4.5	0		Shape$GND	0.1	0
	Medium$4	0.2032			4.5	0				
3	Signal$IS3	0.03048		5.959e+007	4.5	0			0.1	0
	Medium$6	0.2032			4.5	0				

You can see from the window that the **Simulation Report** includes:

- **General information: Spd file name and location**, **Board stackup**, and **Layout top and button layer views**.
- **Simulation setup: Net enabled for simulation**, **Circuit enabled for simulation**, **Port setup**, and **Frequency setup**.
- **Results: Simulation resources**, **Topology and port topology block diagrams**, and **Simulation results curve displays**.

NOTE!	You may need to select Window > Simulation Report to show the **Simulation Report** window.

3.3 Power Ground System With Decoupling Capacitors

This section takes the file psi_brd_demodecaps.spd as an example to describe how to use **PowerSI**.

This section has the following topics:

- Load a File
- View Setup
- Set up Simulation Frequencies
- Run a Simulation

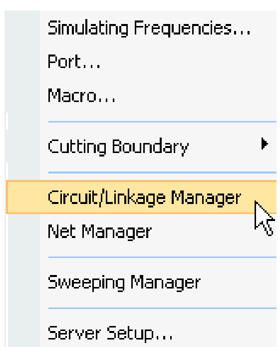
3.3.1 Load a File

Follow steps in **3.2.1** ***Start PowerSI and Load a File*** to load the file psi_brd_demodecaps.spd.

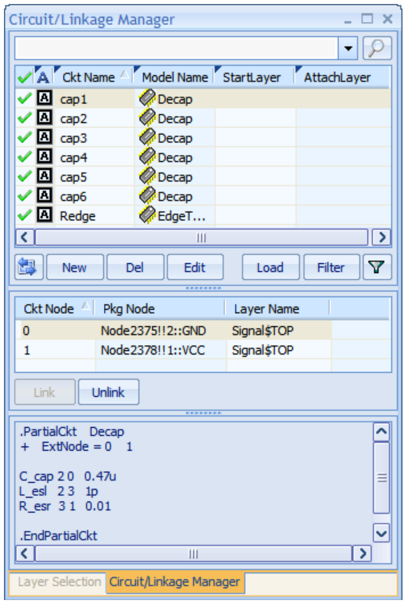
3.3.2 View Setup

Follow these steps to check the setup:

1. Select Setup > Circuit/Linkage Manager.

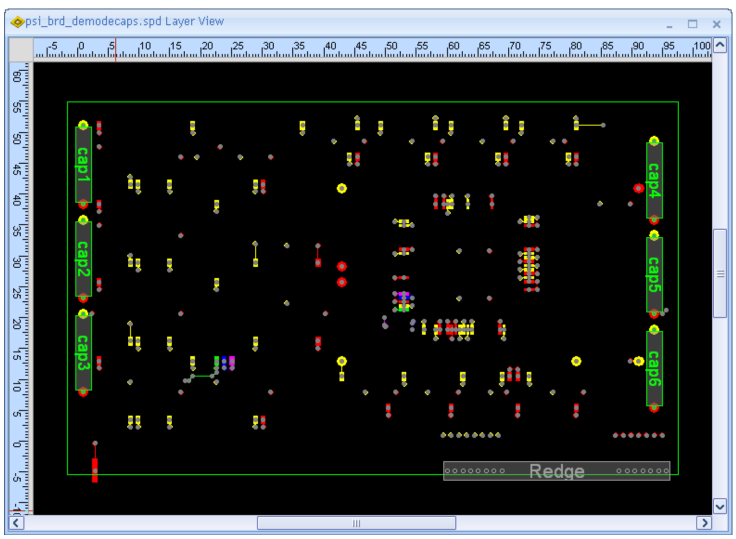


The **Circuit/Linkage Manager** window opens in the right side of the main window.



This example contains six decoupling capacitors connected through the power ground to the supply planes at the side edges of the board.

NOTE!	For how to set up decoupling capacitors, please refer to **Working with Circuits** in the document ***SPD_Layout_UG***.



Ensure that you have clicked to enable **Show Circuit** in the **Layer Selection** window as follows:

The file psi_brd_demodecaps.spd has the same mesh settings (90 x 50) and port specifications as the file psi_brd_demoshort.spd.

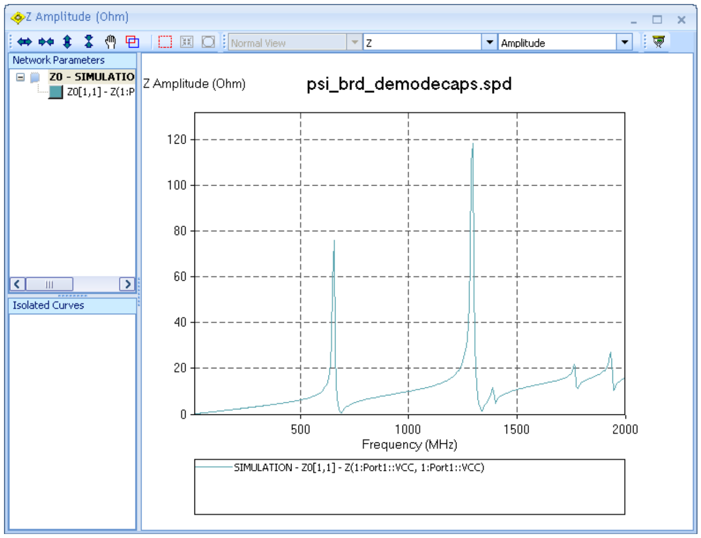
3.3.3 Set up Simulation Frequencies

Follow the steps in **3.2.3 *Set up Simulation Frequencies*** to set up the simulation frequencies.

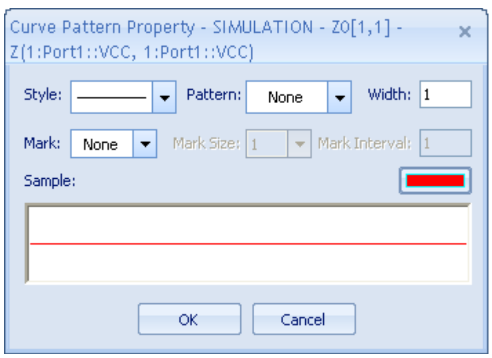
3.3.4 Run a Simulation

Follow these steps to perform a simulation:

1. Click the **Start Simulation** button to start the simulation.
2. Change the display to a Z-parameter curve. See ***2.2.4 Run a Simulation*** for details.

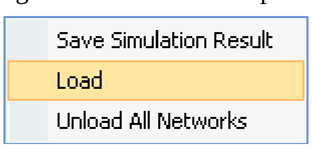


3. Double-click the curve to open the **Curve Pattern Property** dialog box.

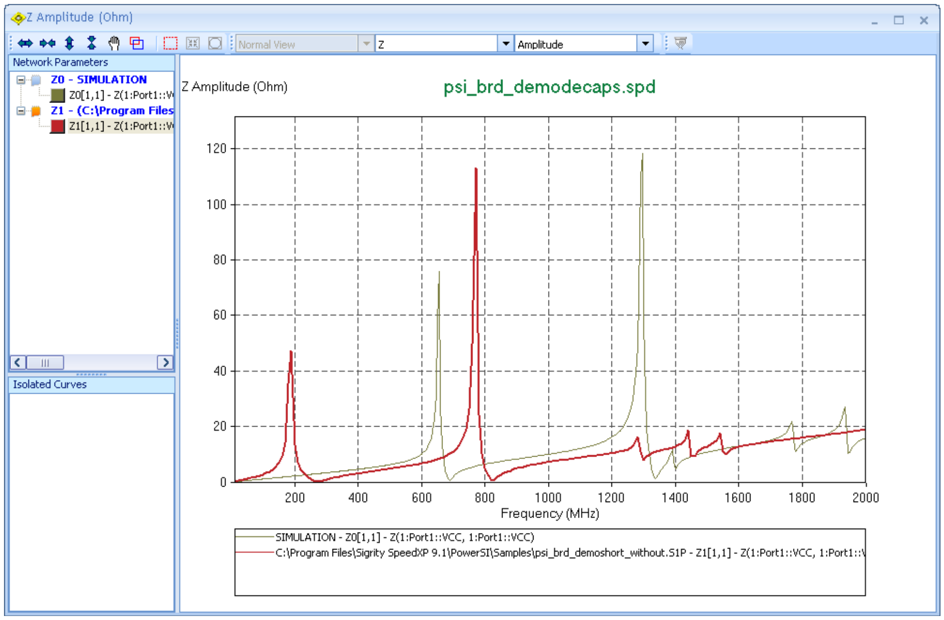


Change the curve color to red. You can also change **Style**, **Pattern**, **Width**, and **Mark** for the curve.

4. Right-click in the blank space of the **Network Parameters** pane to open a pop-up menu.



5. Select **Load** to load the psi_brd_demoshort.S1P file which is saved during operation on psi_brd_demoshort.spd.
6. Compare the power ground performance with and without decoupling capacitors.



The loaded curve overlaps with the present one. So the difference is clear.

7. Close the project without saving it.

4 Project 2 Modeling and Transient Simulation

This chapter describes how to use **PowerSI** and **Broadband SPICE** in I/O modeling and transient analysis.

The chapter contains the following sections:

- Overview
- Constructing a Package
- Specifying Ports
- Running a Simulation
- Equivalent Circuit Extraction by Broadband SPICE
- Broadband SPICE Model Validation
- Transient Simulation by HSPICE

4.1 Overview

This section introduces PCB and overall modeling and simulation procedures.

This section covers two topics:

- PCB
- Modeling and Simulation Procedure

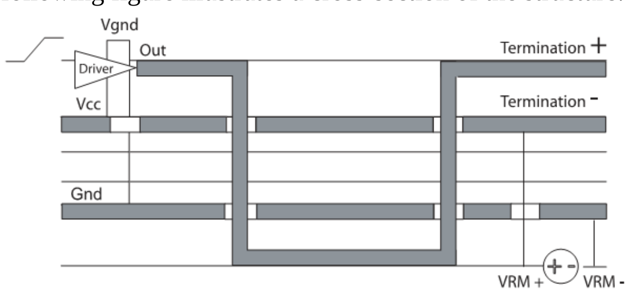
4.1.1 PCB

The printed circuit board (PCB) sample has six metal layers, with traces, interconnections, a driver with power supply connections and a DC-voltage source.

These components can be described as follows:

- **Six metal layers**: A top signal layer, a bottom signal layer, a power plane and a ground plane, and two inner signal layers between the power and ground planes.
- **Traces**: Routes on the top and bottom signal layers.
- **Interconnections**: Between traces on different layers linked by vias.
- **Driver**: Connected to the trace end on the left side of a board. The driver is described by a SPICE transistor level-3 model.
- **Driver power supply nodes**: Connected to the power and ground planes.
- **DC-voltage source**: Connected to the PCB at the edge.

The following figure illustrates a cross-section of the structure:



4.1.2 Modeling and Simulation Procedure

The general steps of modeling and simulation are as follows:

1. Construct a package in **PowerSI**, or translate a board or package layout file into `.spd` format and load it into **PowerSI**.

   (See ***3.2 Constructing a Package***.)

2. Define ports at locations where drivers, terminations, and DC sources are connected.

   (See ***3.3 Specifying Ports***.)

3. Run simulation and obtain multiple-port S-parameter in **Touchstone** data format.

   (See ***3.4 Running a Simulation***.)

4. Load a network parameter file into **Broadband SPICE**, and generate an **HSPICE** compatible equivalent circuit.

   (See ***3.5 Equivalent Circuit Extraction by Broadband SPICE***.)

5. Generate an **HSPICE** netlist for an AC analysis to check the frequency response of the model.

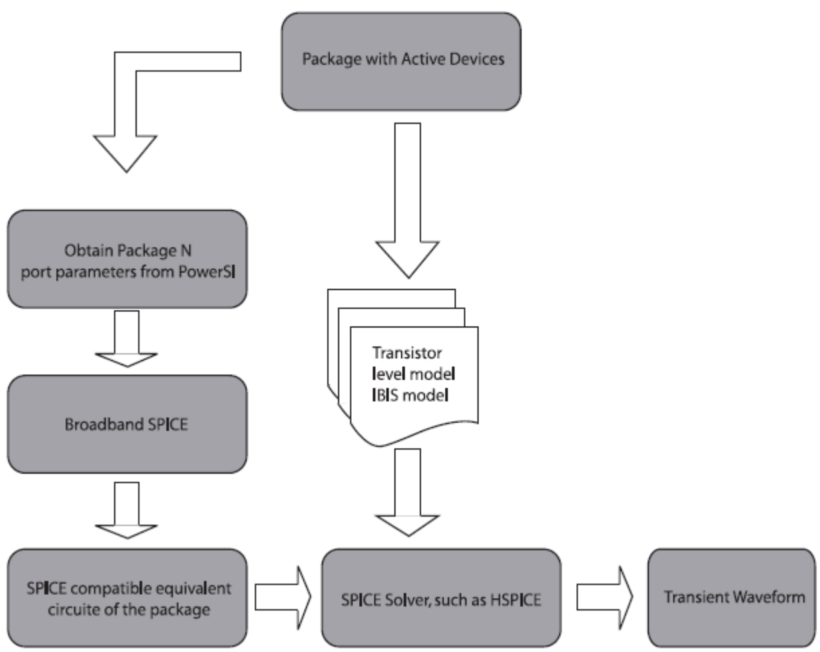
   (See ***3.6 Broadband SPICE Model Validation***.)

6. Generate an **HSPICE** netlist for the transient simulation.

   a. Connect a driver model, DC source, and termination to a multi-terminal equivalent circuit for the package obtained from **Broadband SPICE**.

   b. Run an **HSPICE** simulation for the netlist and observe power ground noise and waveforms at the driver and receiver (termination) ends.

   (See ***3.7 Transient Simulation by HSPICE***.)



4.2 Constructing a Package

This section describes how to construct, create, and add components into a package.

This section covers the following topics:

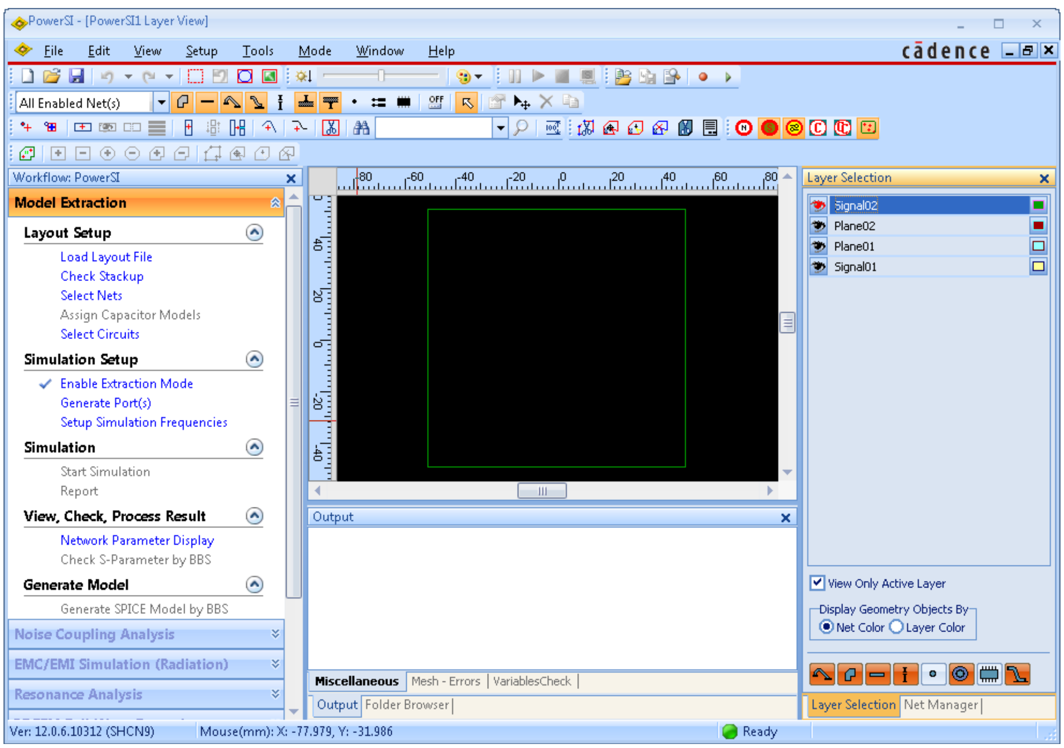
- Construct a Package Model
- Create Nets
- Create a Stackup
- Create Power Ground Shape and Setup Shape Properties
- Add a Via
- Add a Trace
- Set up Trace Properties
- Set up a Trace on Signal01
- View Traces in 3D

4.2.1 Construct a Package Model

Follow these steps to construct a package model in **PowerSI**:

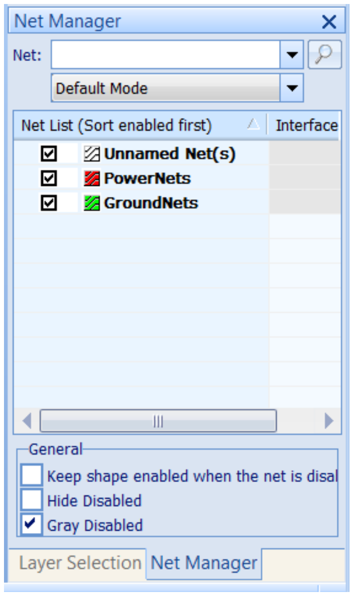
1. Launch **PowerSI**.
2. Select File > New.

   A default four-layer stackup opens.

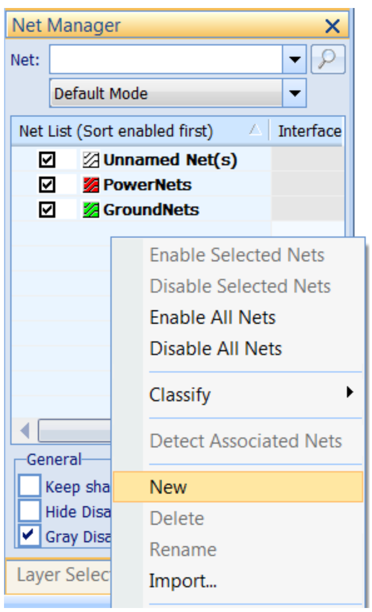


4.2.2 Create Nets

1. Click the **Net Manager** tab to switch to **Net Manager**.

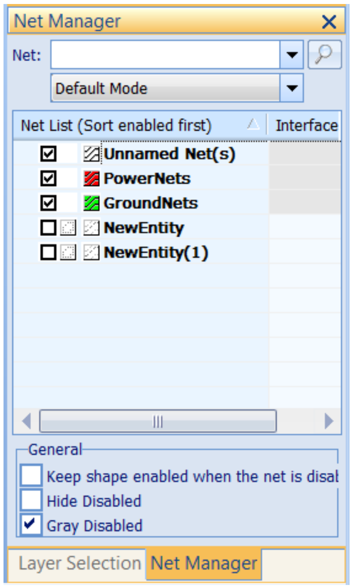


2. Right-click in the area and select **New** from the pop-up menu to create a new net.

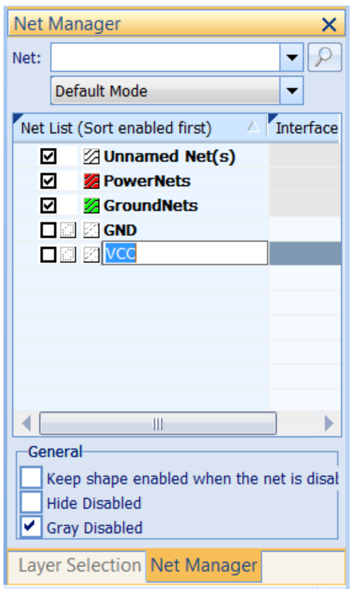


3. Repeat the above step to create one more new net.

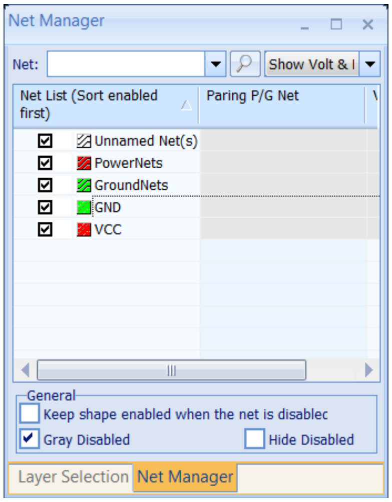
   Two new nets are created.



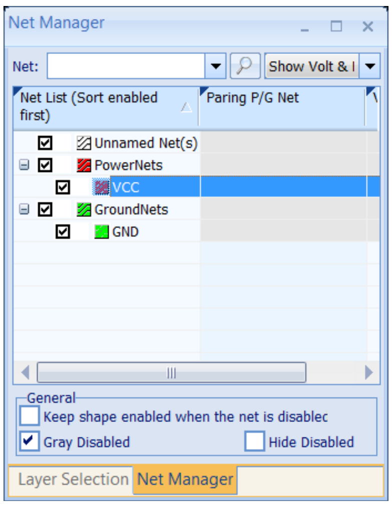
4. Assign the two new nets with name **GND** and **VCC**.



5. Define the colors for **GND** and **VCC** and select to enable both nets.



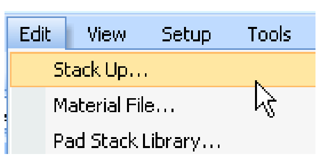
6. Move **GND** to **GroundNets** and **VCC** to **PowerNets** as the following figure shows.



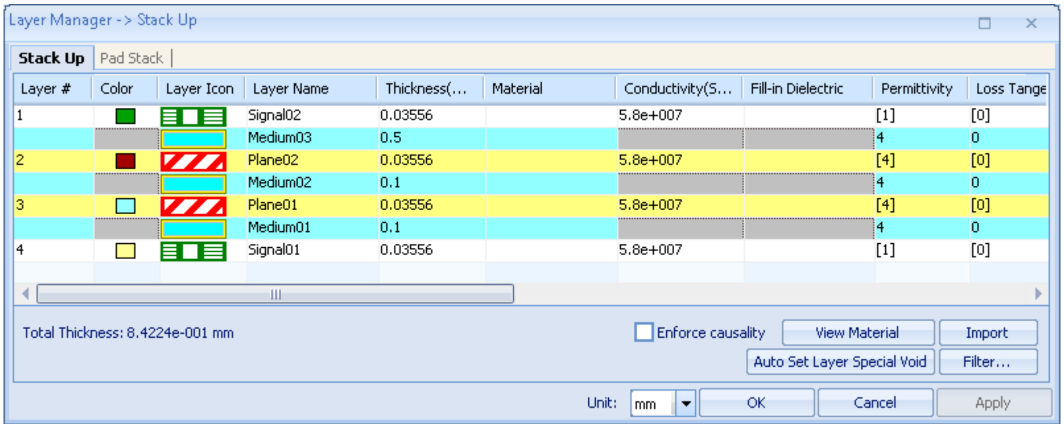
4.2.3 Create a Stackup

Follow these steps to create a six-layer stackup:

1. Select Edit > Stack Up....



The **Stackup** dialog box opens.

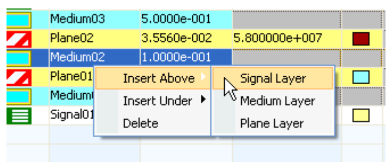


2. Add signal layers above and below the layer **Medium02**.

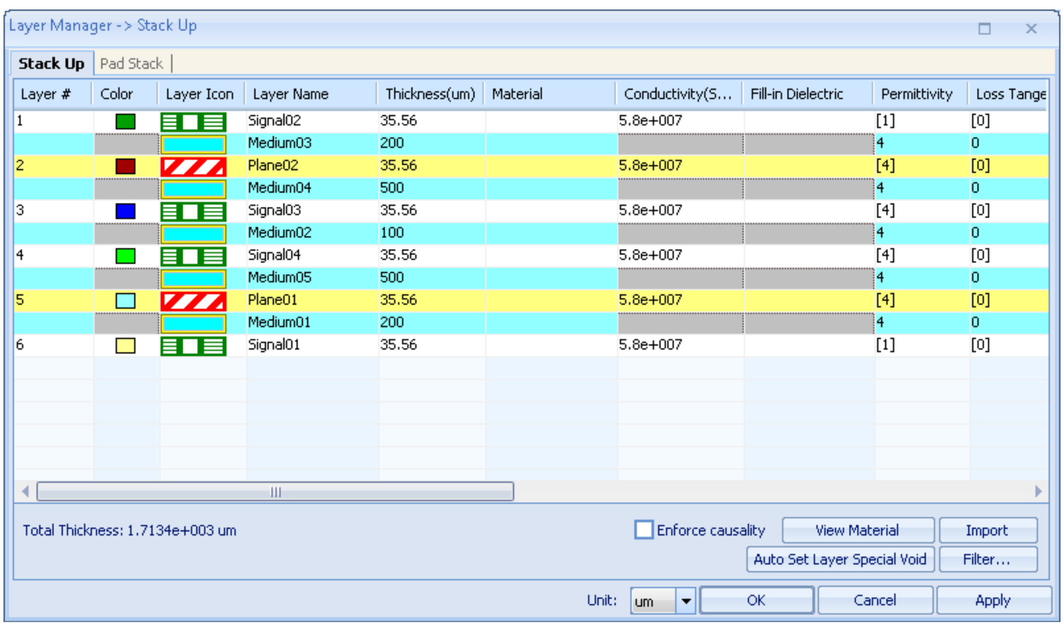
   a. Select the layer **Medium02**.

   b. Right-click **Medium02** to open a pop-up menu.

   c. Select Insert Above > Signal Layer to add a new signal layer while a medium layer **Medium04** is automatically added below **Plane02**.



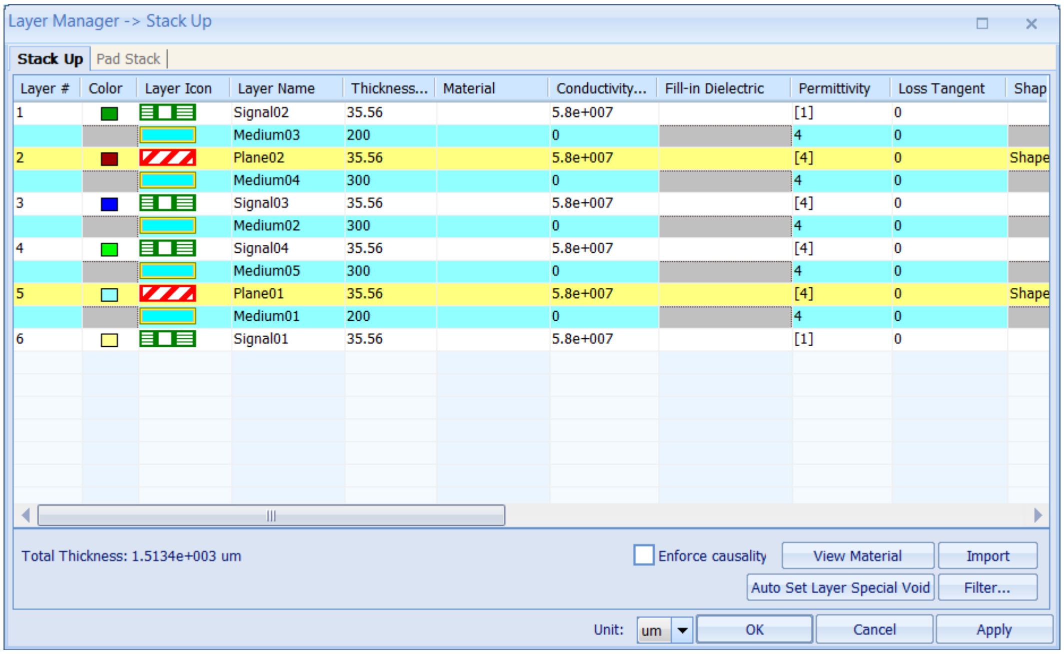
   d. Repeat **Step b** and select Insert Under > Signal Layer to add a signal layer below **Medium02** while a medium layer **Medium05** is automatically added above **Plane01**.



3. Change the **Thickness** unit to **um** from the **Unit** drop-down menu.

4. Double-click the **Thickness** cells to change the values:

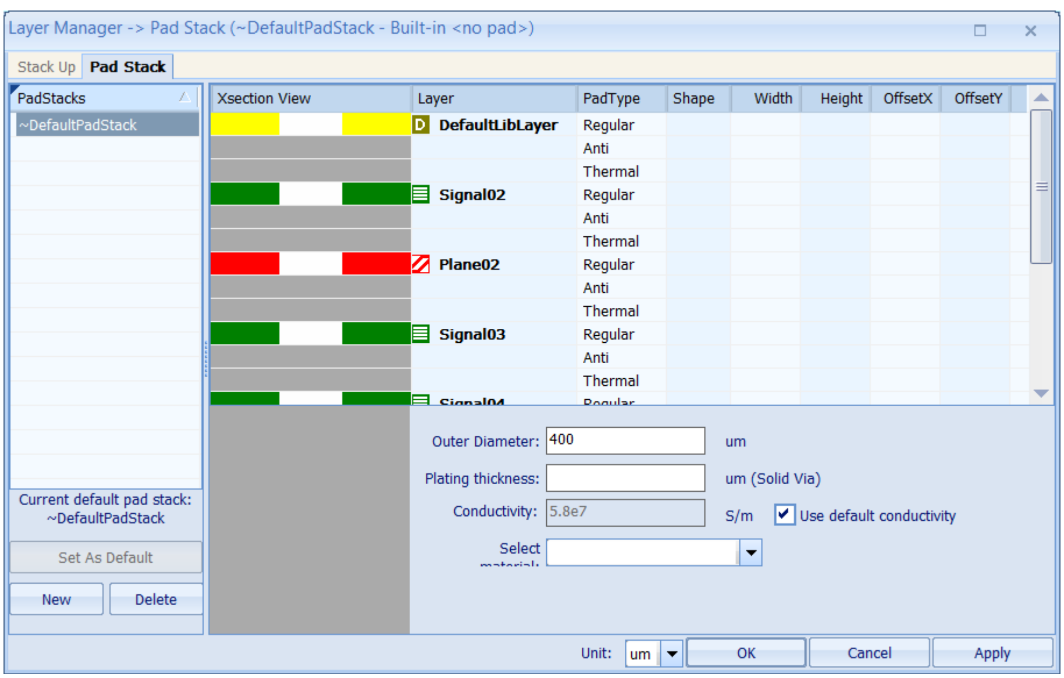
   - Click the **Medium01** and **Medium03 Thickness** cells and input 200 um.
   - Click the **Medium02**, **Medium04** and **Medium05 Thickness** cells and input 300 um.



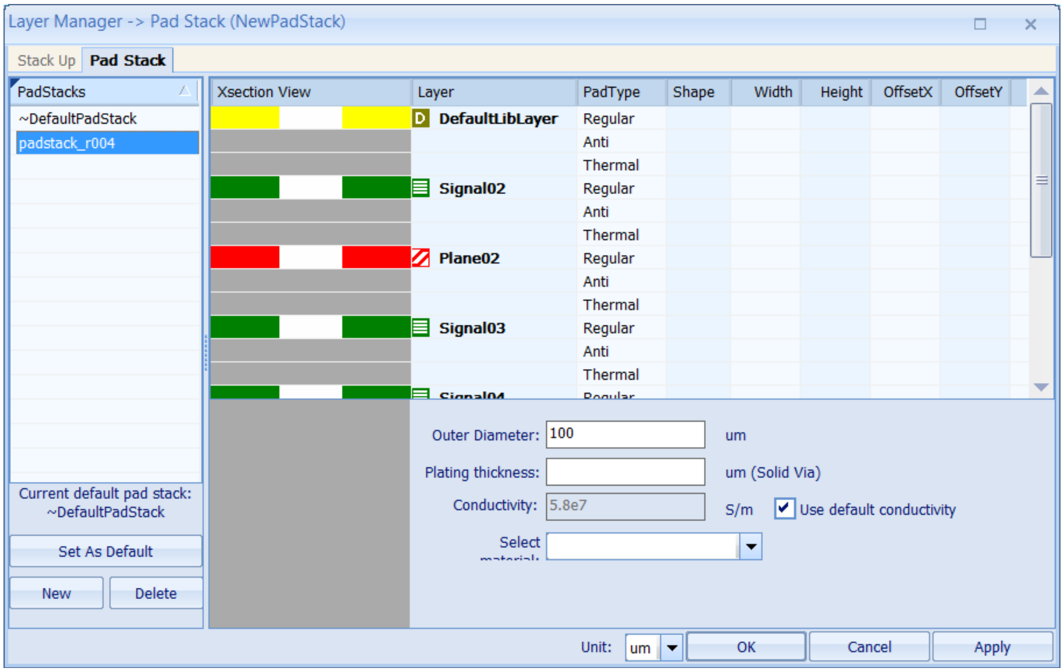
Here, you can also set up **Trapezoidal Trace Angle** and **Surface roughness** for signal layers, or **Permittivity** and **Loss Tangent** for all layers. The default values are used for them in this example.

TIP!	Use the **Ctrl** or **Shift** key to select multiple rows in one column. When you select multiple rows in one column, a value box appears at the bottom of the **Stackup** dialog box, where you can change the value for the selected rows.

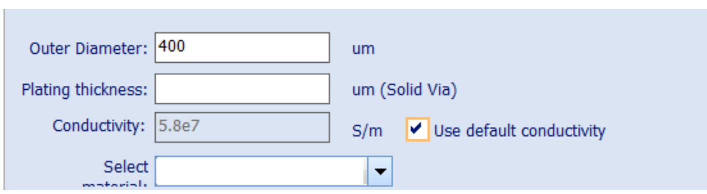
5. Click the **Pad Stack** tab to switch to **Pad Stack** window.



6. Click the **New** button to create a new padstack and assign it with the name **padstack_r004**.



7. Input **400** in the **Outer Diameter** field.

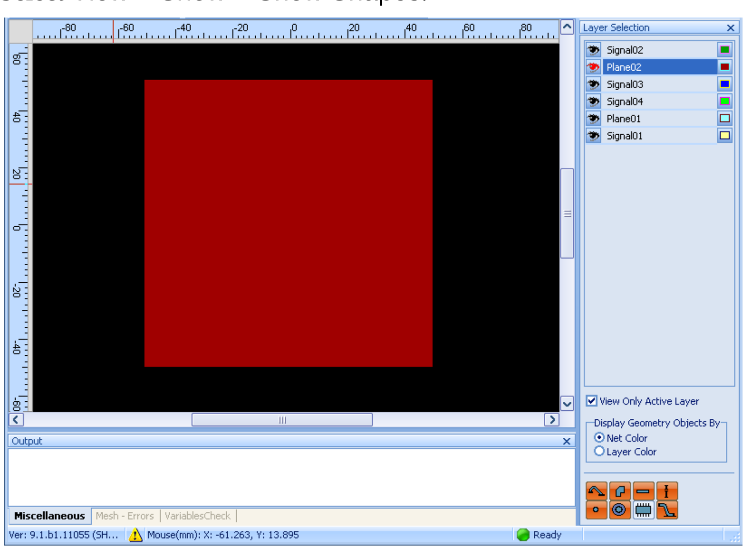


8. Click **OK** to save your changes and exit the **Layer Manager** window.

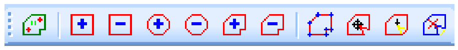
4.2.4 Create Power Ground Shape and Setup Shape Properties

Follow these steps to create a power ground shape:

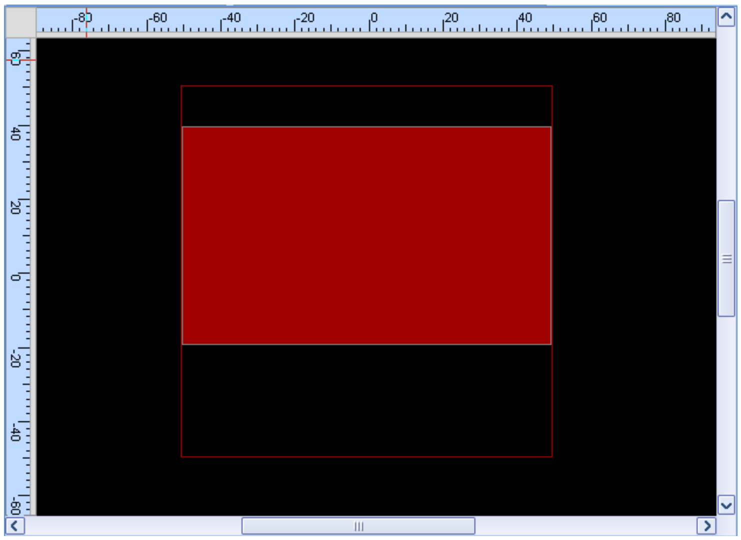
1. Activate **Plane02** in the **Layer Selection** window.
2. Select View > Show > Show Shapes.



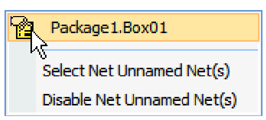
3. Choose the **Select** button  on the **Select** toolbar, and click any location on the shape to select the shape.
4. Press the **Delete** key to delete the shape.
5. Select View > Toolbars > Shape to bring up the **Shape** toolbar.



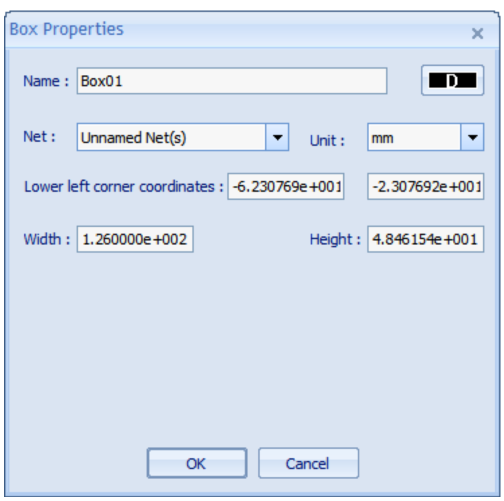
6. Select the **Box+** button  and draw a rectangle in the **Layer View** window. The screen automatically zooms into the new shape.



7. Right-click the created shape to open a pop-up menu.



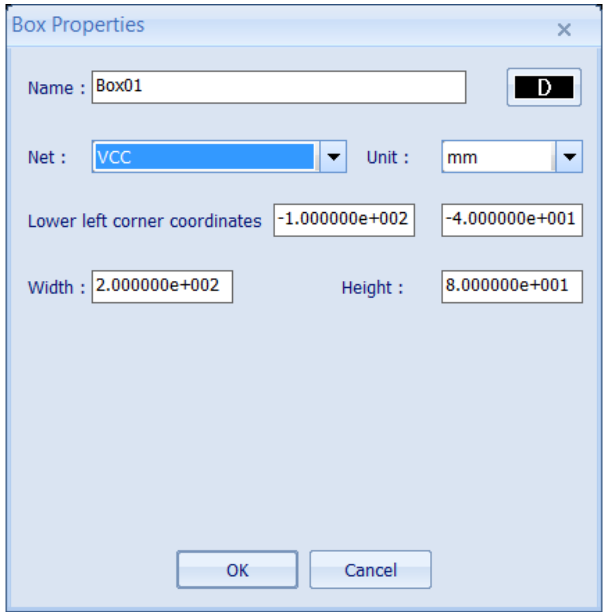
8. Select **Package1.Box01** to open the **Box Properties** dialog box.



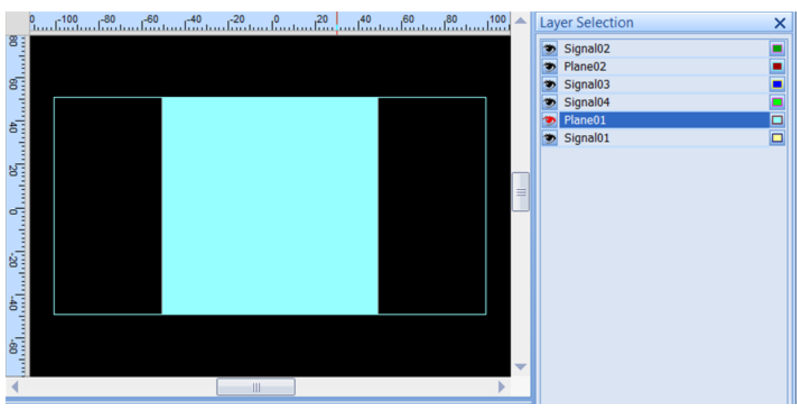
9. Set up the coordinates for **Box01**:
   - **Unit**: mm
   - **Lower left corner coordinates**: –1.00000e + 002 and –4.00000e + 001

- **Width**: 2.00000e + 002
- **Height**: 8.00000e + 001

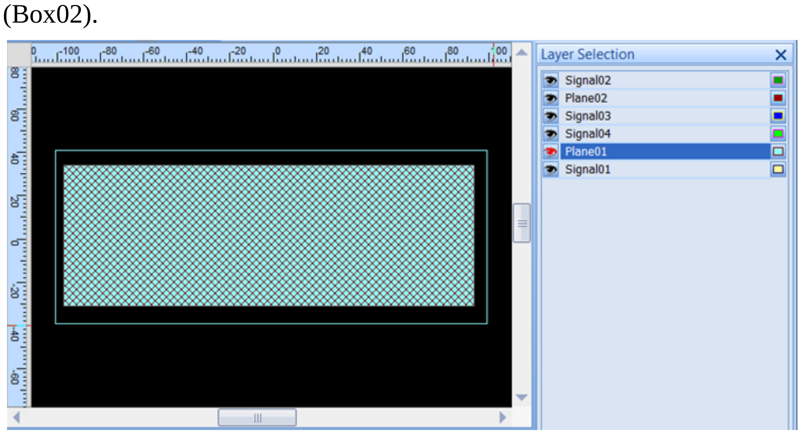
10. Select **VCC** from the **Net** drop-down list and assign it to Box01.



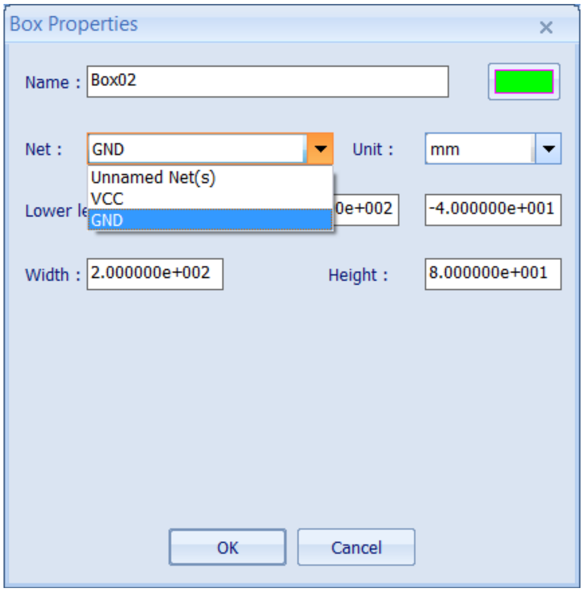
11. Click **OK** to save the changes.
12. Activate **Plane01** in the **Layer Selection** window.



13. Repeat the above steps to delete the original shape in **Plane01**, and create a new shape (Box02).



14. Set up the coordinates for **Box02**.
    - **Unit**: mm
    - **Lower left corner coordinates**: –1.00000e + 002 and –4.00000e + 001
    - **Width**: 2.00000e + 002
    - **Height**: 8.00000e + 001
15. Select **GND** from the **Net** drop-down list and assign it to Box02.



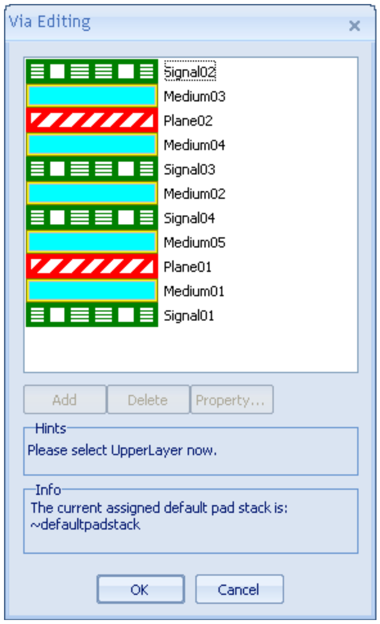
16. Click **OK** to save the changes.

4.2.5 Add a Via

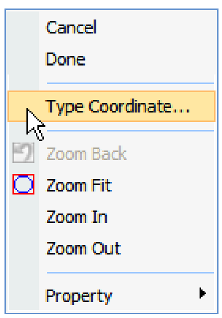
Follow these steps to add a via:

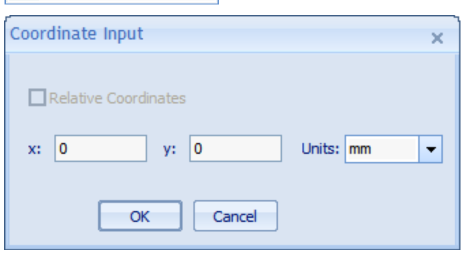
1. Click anywhere in the **Layer View** window to make it active.
2. Select the **Add Via** button  on the **Object** toolbar.
3. Click a location (–25, 0) on the **Layer View** window.

   The cursor transforms into a + sign. The **Via Editing** dialog box opens.



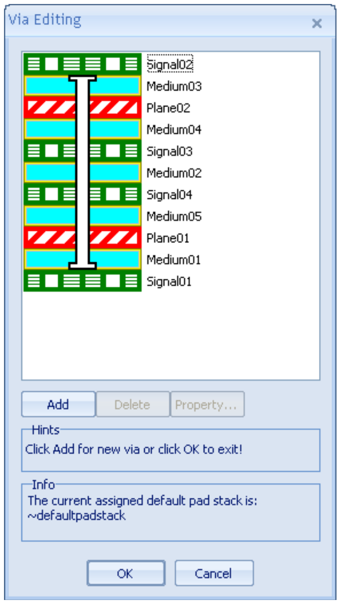
TIP!	The **Status Bar** shows the coordinates of the cursor timely. You can right-click to open a pop-up menu and select **Type Coordinate…** to open the **Coordinate Input** dialog box. You can input exact coordinates directly in the box.





4. Select **Signal02** and then **Signal01** in the **Via Editing** dialog box.

   A via appears to connect the **Signal02** and **Signal01** layers.



5. Click **OK**.

   The added via is automatically named as **Via01**, which connects the traces routed on **Signal01** and **Signal02**.

NOTE!	When adding vias, you may not get the exact coordinates as specified. Simply locate the coordinates as close as possible. The simulation result should not be highly affected by coordinates that are not exact.

TIP!	If you add a via at a wrong location, you can either: • Click the **Select** button on the **Select** toolbar, drag to select the via, and then press the **Delete** key. • Use the **Move** button on the **Select** toolbar to move the node to the right coordinates. For detailed information about the **Select** toolbar, see ***PowerSI User's Guide***.

Repeat **Step 2** to **6** to add the following vias.

Via	Coordinates	Layers	Descriptions
Via02	25, 0	**Signal02, Signal01**	Specify the traces routed on Signal01 and Signal02.
Via03	–75, 1	**Signal02, Plane02**	Specify the ports connecting the driver.
Via04	–75, –1	**Signal02, Plane01**	
Via05	75, 1	**Signal02, Plane02**	Specify the ports connecting the receiver.
Via06	75, –1	**Signal02, Plane01**	
Via07	–0.5, –39	**Signal02, Plane02**	Specify the ports connecting the VRM module. Use the zoom button to pinpoint the coordinates.
Via08	0.5, –39	**Signal02, Plane01**	

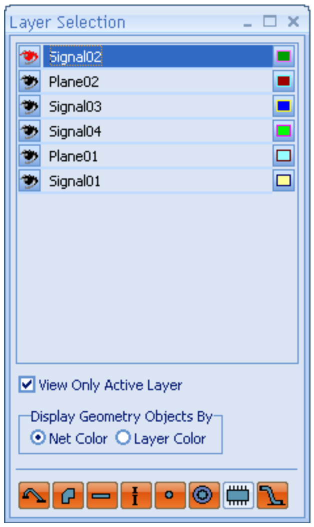
TIP!	Use **Snap to Grid** to set a 0.5 or –0.5 coordinates: Select Tools > Options > Edit Options… > Layout > Grid and Unit to check the **Snap objects to grid** option.

4.2.6 Add a Trace

Follow these steps to add a trace:

1. Click **Signal02** in the **Layer Selection** dialog box.

   The **Signal02** icon becomes red.



2. **Click** the **Add Trace** button  on the **Object** toolbar.

   The cursor becomes a + sign.

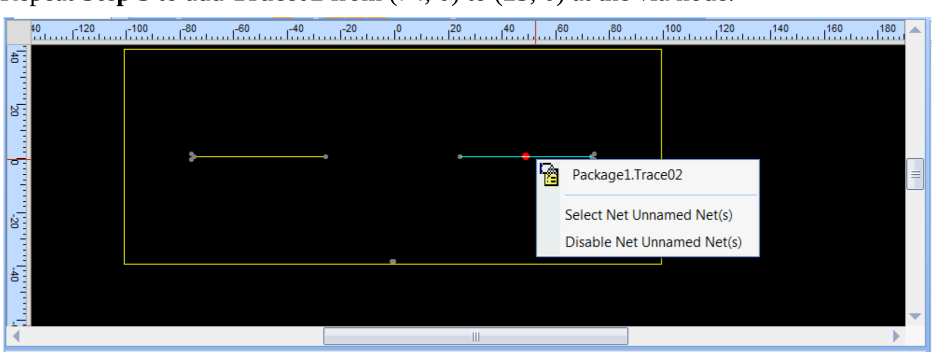
3. To add a trace:

   a Find the coordinates (–74, 0) and click.

   b Find **Via01** (–25, 0) and click.

   c Double-click to end this operation. **Trace01** is added.

   Repeat **Step 3** to add **Trace02** from (74, 0) to (25, 0) at the via node.

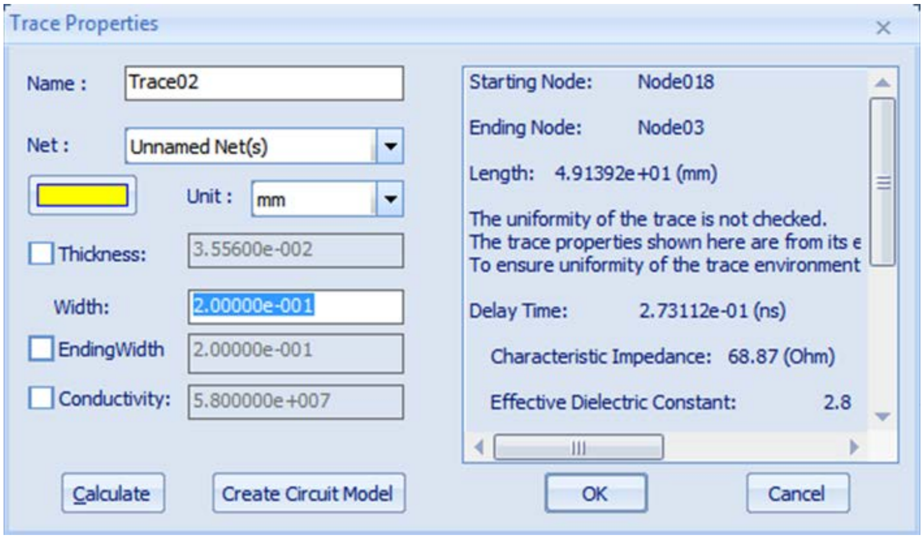


4.2.7 Set up Trace Properties

Follow these steps to change a trace’s properties:

1. Right-click **Trace02** and select the trace name in the pop-up menu.

   The **Trace Properties** dialog box opens.



2. Change the trace width to 0.2 mm.

Repeat **Step 1** and **2** to change the **Trace01** width to 0.2 mm.

4.2.8 Set up a Trace on Signal01

Follow these steps to change a trace on **Signal01**:

1. Click **Signal01** in the **Layer Selection** dialog box.

   The **Signal01** icon becomes red.

2. Add **Trace03** from (–25, 0) to (25, 0) on the **Signal01** layer.

   Via01 and **Via02** are now connected by **Trace03**.

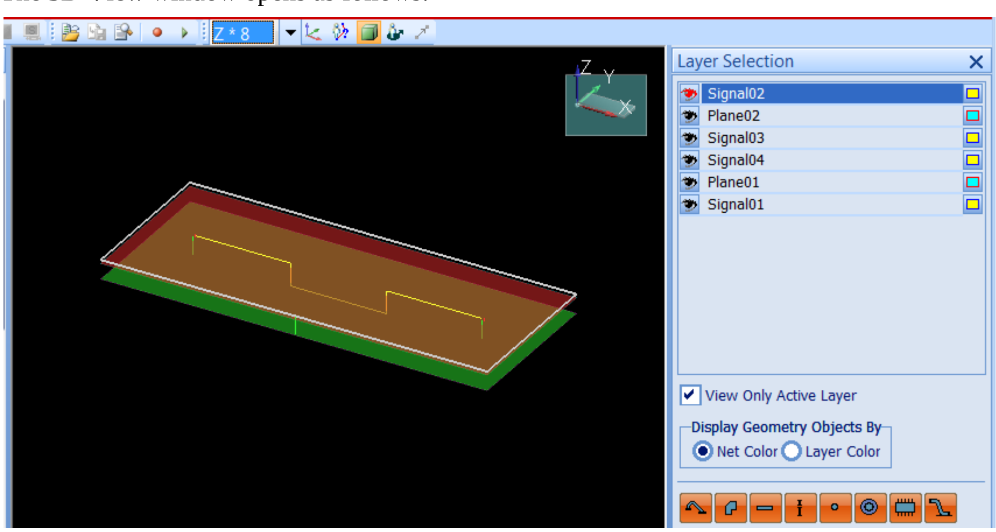
3. Change the **Trace03** width to 0.2 mm.

4.2.9 View Traces in 3D

Follow these steps to view traces in 3D:

1. Select View > 3D View > Full 3D View.

   The **3D View** window opens as follows:



2. Select View > Layer View to switch back to the **Layer View** window.

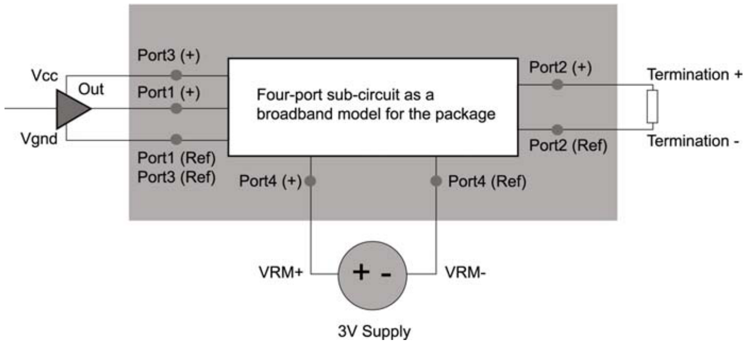
4.3 Specifying Ports

This section describes how to build a multiple terminal package model.

This section has the following two topics:

- Add Ports
- Set up a Port

The figure below shows a multiple terminal package model:



Three devices/components will be connected to the package. Ports are specified in the package structure where the connection to external circuit components is made in HSPICE simulation.

Follow these rules to specify PowerSI ports for connection with external circuit components in a SPICE-compatible circuit simulator:

- Specify N ports for connecting the package to an N+1 terminal device.
- Ensure that the specified N-port on the N+1 terminal device share the common reference (the negative “–” terminal of the port).
- Ensure that all N ports specified for connection to one device are physically local because the device is considered to be a lumped circuit component.
- **Driver**: A three-terminal device. Define two ports for it:

  `Port1: (+)Out (Ref)Vgnd Port3: (+)Vcc (Ref)Vgnd`
- **Termination:** A two-terminal device. Define one port for it:

  `Port2: (+)Termination + (Ref)Termination-`
- **DC Source:** A two-terminal device. Define one port for it:

  `Port4: (+)VRM+ (Ref)VRM-`

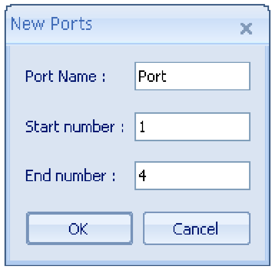
4.3.1 Add Ports

Add ports for connection to external components.

Follow these steps to add ports:

1. Select **Signal02** in the **Layer Selection** dialog box.
2. Click the **New Port** button on the **Port** toolbar.

   The **New Ports** dialog box opens.



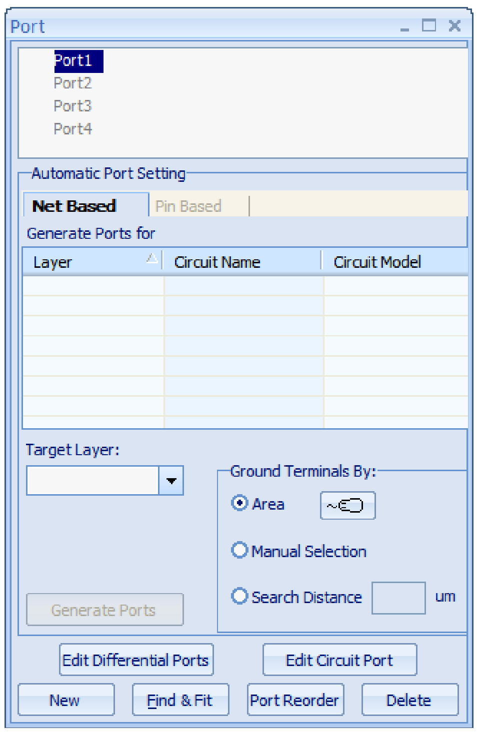
3. Set **Start number** to 1 and **End number** to 4 to add four ports.
4. Click **OK** to save and exit.

4.3.2 Set up a Port

Follow these steps to set a port:

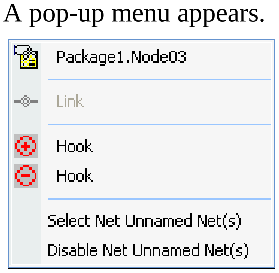
1. Select Setup > Port….

   The **Port** dialog box opens.



2. Select **Port**1.
3. Find and right-click the left end of **Trace01** (–74, 0).

   A pop-up menu appears.



4. Select **+Hook** to assign the left node of **Trace01** to the positive terminal of **Port1**.

5. Select the nearby **Via04** located at (–75, –1), a ground via and connected to **Plane01**, and right-click to show a pop-up menu.
6. Select **–Hook** to assign the upper node of **Via04** to the negative (reference) terminal of **Port1**.
7. Close the **Port** dialog box.
8. Select File > Save to save the structure in the file psi_IO_modeling.spd.

Repeat **Step 2** to **6** to set up **Port2** to **Port4**.

Port2

- Positive Terminal: The right end of **Trace02** (74, 0).
- Negative Terminal: The node of the ground via **Via06** located at (75, –1).

Port3

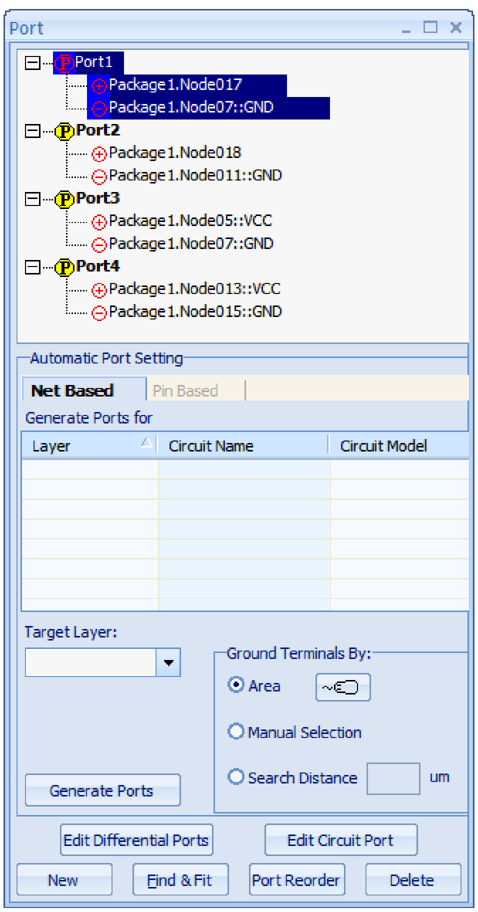
- Positive Terminal: The upper node of **Via03** located at (–75, 1).
- Negative Terminal: The lower node of **Via04** located at (–75, –1)

Port4

Use the zoom button to enlarge the area so that you can select the correct node.

- Positive Terminal: The node of the power via **Via07** located at (–0.5, –39).
- Negative Terminal: The node of the ground via **Via08** located at (0.5, –39).

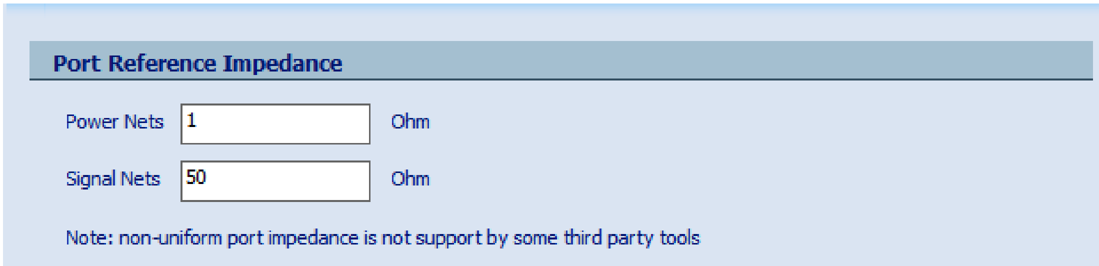
The **Port** dialog box looks like the following figure after four ports are defined.



4.3.3 Set Reference Impedance for the Port

To set reference impedance for the port, select

Tools > Options > Edit Options… > Simulation(basic) > Network Parameters



By default, Power Nets is set to 1Ohm and Signal Nets is set to 50Ohm.

4.4 Running a Simulation

This section describes how to simulate the package with ports defined in the file psi_IO_modeling.spd generated in ***Section 4.3 Specifying Ports***.

This section has the following topics:

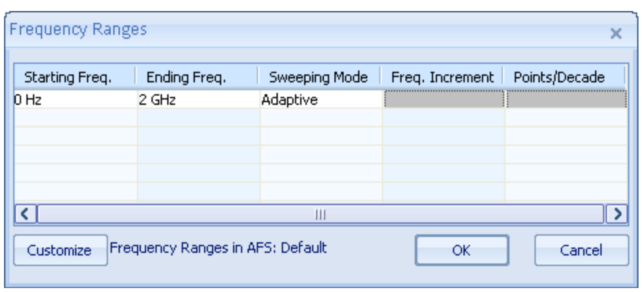
- Set up Simulation Frequencies
- Run a Simulation
- Export Results

4.4.1 Set up Simulation Frequencies

Follow these steps to set up simulation frequencies:

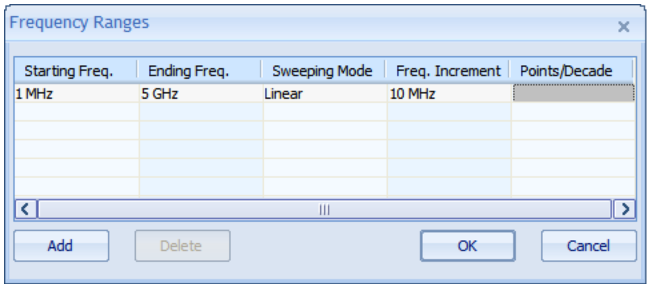
1. Select Setup > Simulating Frequencies….

   The **Frequency Ranges** dialog box opens.



2. Change the following parameters:
   - **Starting Freq.** to 1 MHz
   - **Ending Freq.** to 5.0 GHz
   - **Sweeping Mode** to **Linear**

   Freq. Increment becomes 10 MHz automatically.

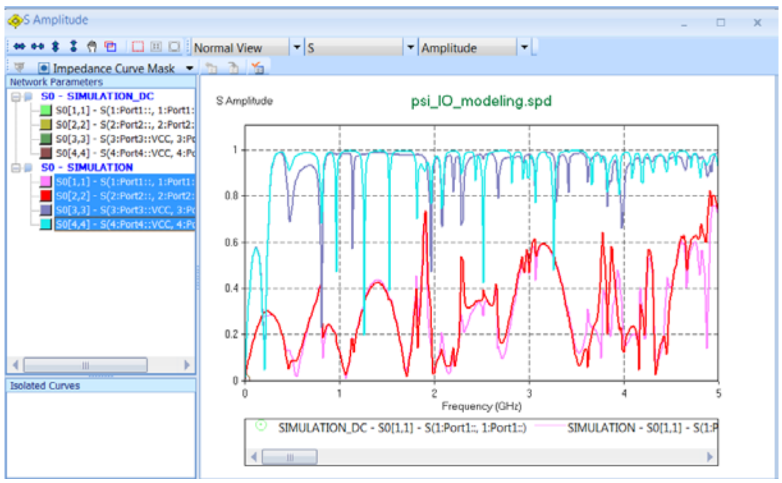


3. Click **OK** to save and exit.

4.4.2 Run a Simulation

1. Press the **Start Simulation** button on the **Simulating** toolbar.

   The **Port Curves** window appears.



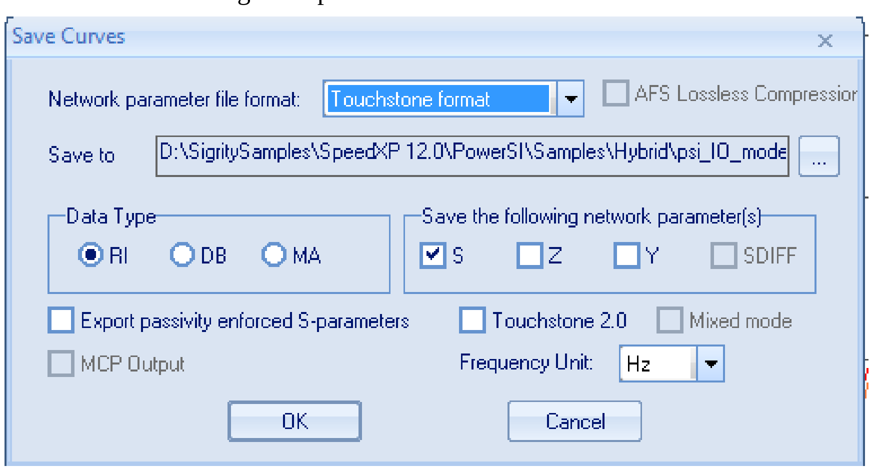
The curve window automatically displays the S-parameter curves for four ports.

4.4.3 Export Results

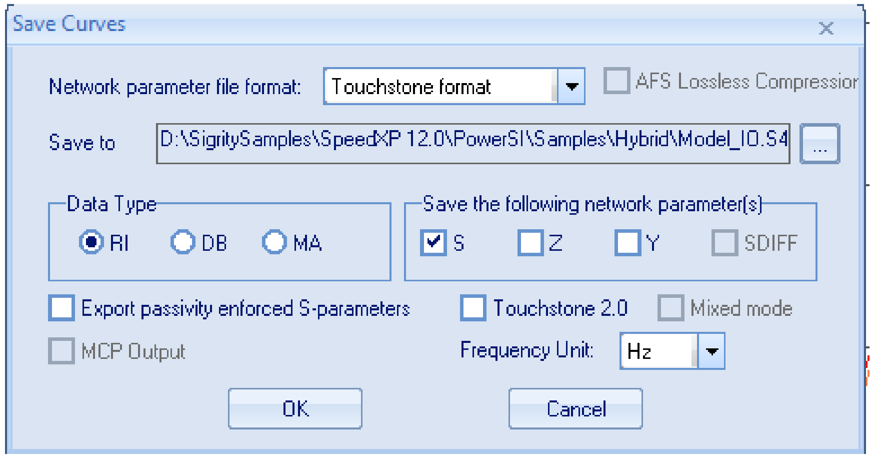
Follow these steps to export the simulation results in different formats:

1. Select File > Save Simulation Result.

   The **Save Curves** dialog box opens.



2. Select **Touchstone format** as the file format.
3. Select **RI** as the data type.
4. Select **S** under **Save the following network parameter(s)**.
5. Rename the file to model_IO.s4p.



6. Click **OK** to save the file and exit the **Save Curves** dialog box.

   The following header format for S parameter is supported in touchstone file:

```
! GNDNET NET014
! NETSLIST NET029, NET019
! Port 1 = U3_1 NET029
! Port 2 = U1_2 NET019
! Port 3 = U4_1 NET019
! Port 4 = U6_2 NET029
```

   The nets and ports are specifically bound to each other. No implicit assumptions are made in this case and only the explicit binding specified in the header is honored.

   GNDNET and NETSLIST are solver specific optional parameters:

   - GNDNET specifies the net to which all the ground pins in the model should be connected
   - NETSLIST is a comma separated list of nets which should be deleted from the schematic when the model gets assigned

7. Select File > Exit to close the project.

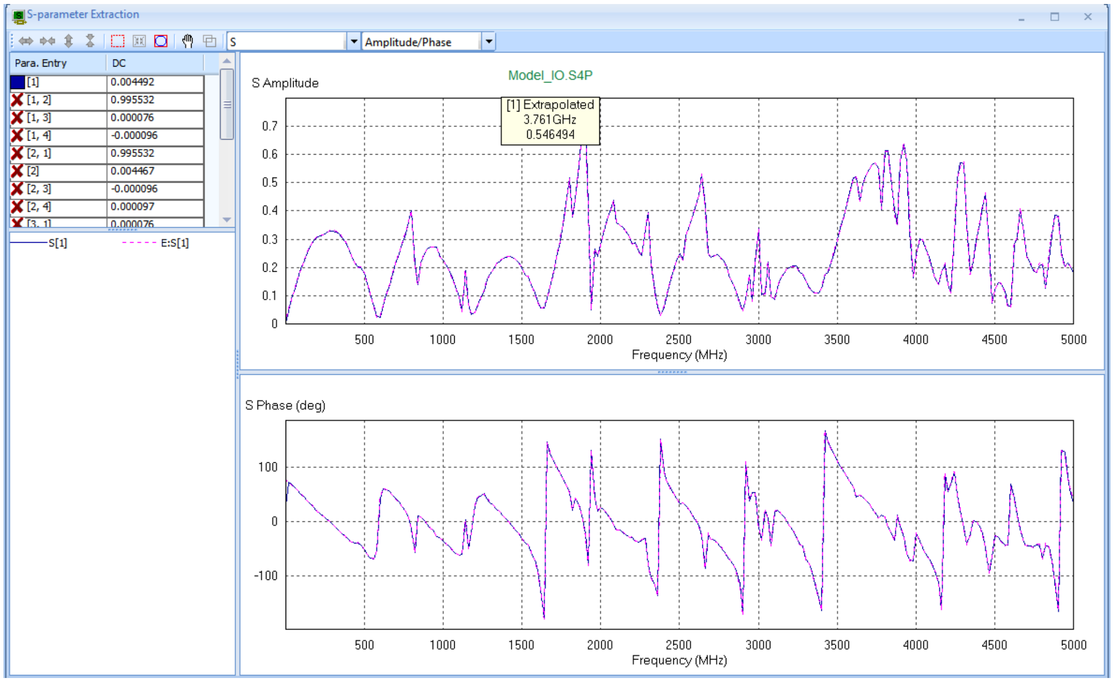
4.5 Equivalent Circuit Extraction by Broadband SPICE

This section uses the network parameter file model_IO.s4p to extract equivalent circuits in **Broadband SPICE**.

Follow these steps to extract equivalent circuits:

1. Launch **Broadband SPICE**.
2. Select File > Open… to open the file model_IO.s4p.
3. Choose the **Extract** button  on the **Action** toolbar to start an extraction.

   The following window shows the extraction result:



4.6 Broadband SPICE Model Validation

This section describes how to use **HSPICE AC Analysis** to check the frequency response of the model before proceeding with the transient simulation of the package models with the driver, DC source, and terminations. **HSPICE AC Analysis** should match the **PowerSI** simulation results.

The .net function is used in **HSPICE**. Because the .net function can only simultaneously obtain two port parameters, you need to compute the partial entries of the port S-parameters of an N-port, by connecting the reference impedance (50 ohms) to the rest of the ports.

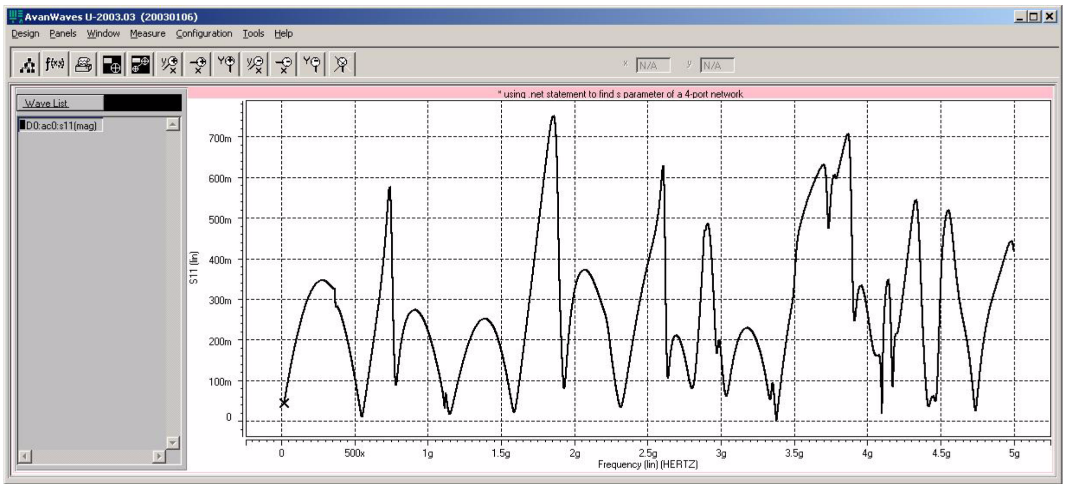
The following **HSPICE** netlist (Validate_Model_IO.sp) exemplifies the computation of S11, S12, S21, and S22 of the four-port. The following **AvanWave** screen capture shows the results of an S11 (magnitude) from the 1 MHz to 5 GHz, which matches the **PowerSI** simulation.

The Validate_Model_IO.sp file includes a file Model_IO.txt. This file is a **Broadband SPICE**-extracted equivalent circuit file. Both files should be within the same directory.

```
* Using .net statement to find S parameter of a 4-port network
.include 'Model_IO.txt'
.AC lin 1Meg 20Meg 5000Meg.
.net v(p2) vin rout = 50 rin = 50
vin p1 0 AC 1
X1 p1 p2 p3 p4 0 Model_IO
R3 p3 0 50
R4 p4 0 50
.probe AC S11(m)
.probe AC S12(m)
.probe AC S21(m)
.probe AC S22(m)
.end
```

See your **HSPICE** application user guide for information on how to check the frequency response.

NOTE!	• The equivalent circuit model (in terms of **SPICE** sub-circuit) for an N-port S-parameter includes N+1 nodes, where the first N nodes corresponds to the N-port terminals. The last node corresponds to the reference of all the N-ports. • In a SPICE simulation, the N-terminal nodes of the sub-circuit should be connected to the device terminals for which the positive terminals of the ports are specified in PowerSI.



4.7 Transient Simulation by HSPICE

This section describes how to perform a transient simulation for IO analysis and to observe the power supply noise at the driver's power ground pins.

This section has the following topics:

- Prepare an HSPICE Netlist
- View HSPICE Simulation Results

4.7.1 Prepare an HSPICE Netlist

You can find the file Driver_Package.sp and MC74LCX16244.txt under the Broadband SPICE > Samples directory.

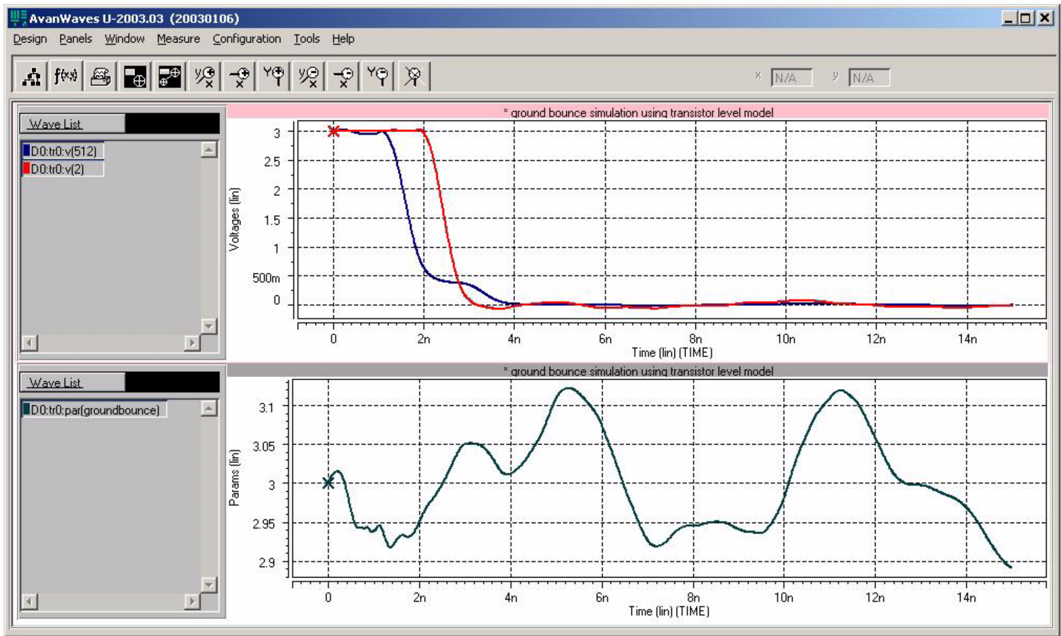
- The file MC74LCX16244.txt contains a transistor-level model for a particular driver.
- The file Driver_Package.sp contains the following information:

```
* Ground bounce simulation using transistor level model
* Last Updated: May 23, 2002
* File name: Driver_Package.sp

.option probe post=2 ingold = 2 numdgt = 10
.param Z0 = 50

* Include the HSPICE level-3 model for the driver
```

```
.include 'MC74LCX16244.txt'

* Include the equivalent circuit model for the package
.include 'app1_drv.txt'

* The termination

R2 2 0 10meg

* The DC supply model

Vdc_supply 4 t4 DC 3
Rdc_supply t4 0 1e-5

* Connect the equivalent circuit to the driver model etc.
* 1: Driver output node       2: Termination+ node
* 511: Vcc node of the driver 4: DC source positive node

X1 1 2 511 4 0 app1_drv

.tran  0.005ns 15ns

.probe tran v(512) V(510) GroundBounce = par('V(518)-V(522)')
v(518) v(522) v(1) v(2) v(4) v(511) v(510)
.print tran v(512)
.print tran GroundBounce = par('V(518)-V(522)')
.print tran v(2)

.end
```

4.7.2 View HSPICE Simulation Results

After the **HSPICE** simulation is completed, you should get a view as follows:



- The upper half shows the waveform at the driver-end (in blue) and the waveform at the termination end (in red).

  The input signal to the driver is a pulse with 1 ns rise-time specified in the netlist.
- The lower half shows the voltage between the VCC node and the Vgnd node of the driver.

  The figure shows that the actual supply at the driver fluctuates around three volts due to the driver's switching activity.

5 Project 3 Decoupling Capacitors Placement

This chapter describes how to select and place decoupling capacitors to minimize the power delivery system noise. This chapter takes the following files as examples:

- psi_brd_PDSdesign_NoDecp.spd
- psi_brd_PDSdesign_Decap_stage1.spd
- psi_brd_PDSdesign_Decap_stage2.spd

These files have been installed when you install **PowerSI** on your computer.

This chapter has the following sections:

- Power Ground Noise Without Decoupling Capacitors
- Decoupling Capacitor Placement (First Round)
- Decoupling Capacitor Placement (Second Round)
- Self and Transfer Impedance of the Power Delivery System

Project Specifications:

- The PCB size is 12” x 8” and has a Signal-Power-Ground-Signal stackup.
- The power and ground metal layers are copper and 1.4 mils thick.
- The dielectric layers are FR4 material (dielectric constant = 4.0, lossless).
- The dielectric layer thickness between power and ground planes is 5 mils.

5.1 Power Ground Noise Without Decoupling Capacitors

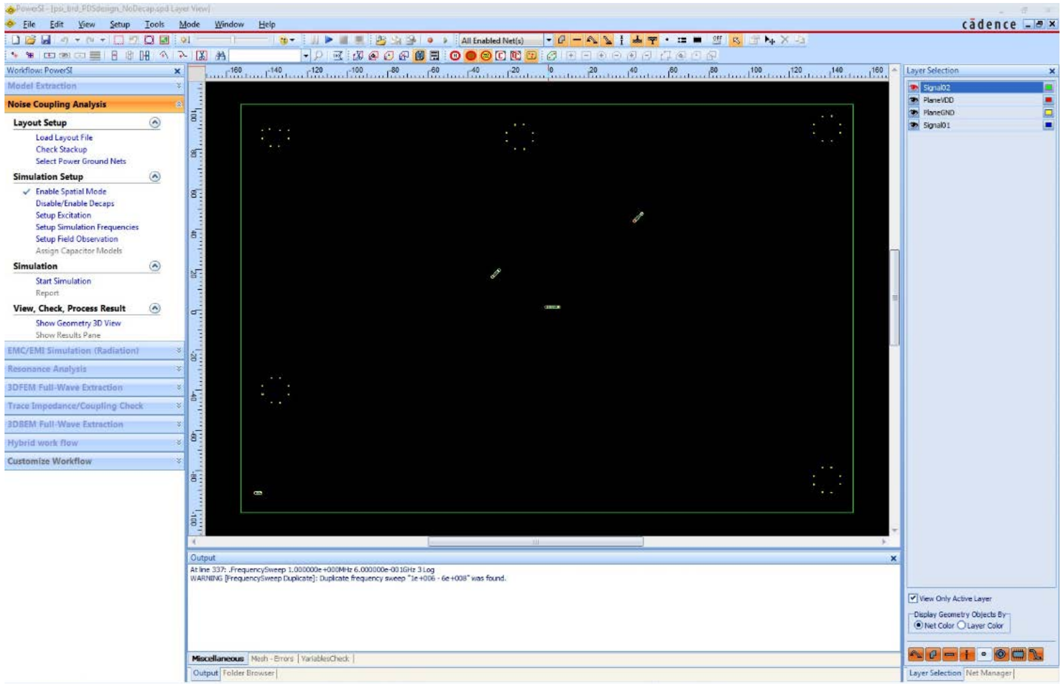
This section has the following topics:

- Start PowerSI and Load a File
- View Circuits
- View Circuit Voltages
- View Voltage Distributions
- Set up Simulation Frequencies
- Run a Spatial Mode Simulation

5.1.1 Start PowerSI and Load a File

Follow these steps to start **PowerSI** and load a file:

1. Launch **PowerSI**.
2. Select Mode > Spatial Mode to set the **Spatial Solution** mode for this simulation.
3. Select File > Open… and choose psi_brd_PDSdesign_NoDecap.spd.

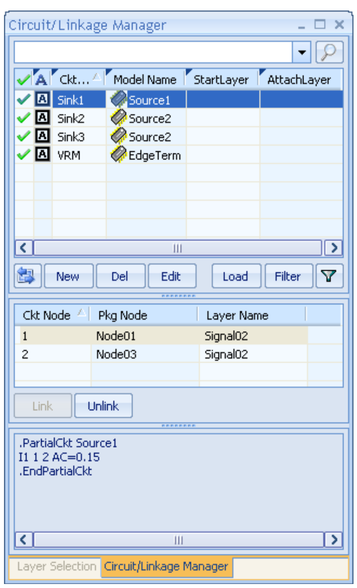


In this example, three current sources around the center of the board represent the current sink at these locations. The center current source is 150 mA and the other two are 75 mA. A resistor at the lower left of the board represents the Voltage Regulator Module and its value is 0.01 ohm.

5.1.2 View Circuits

Follow these steps to view connected circuits in details.

1. Select Setup > Circuit/Linkage Manager.



Spatial Mode requires at least one active circuit to act as the stimulus for the simulation. AC current sources are typically used and can be automatically created under the Noise Coupling Analysis Workflow > Setup Excitation.

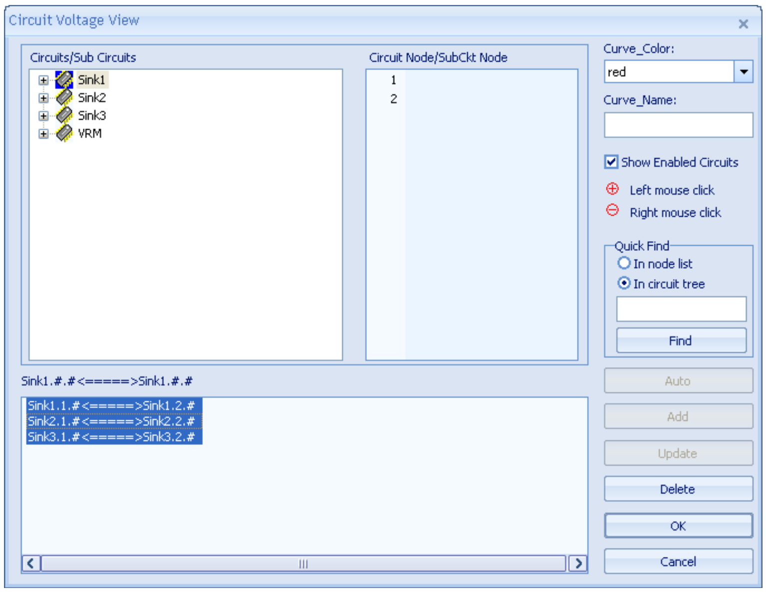
2. Close the **Circuit/Linkage Manager** window without any changes.

NOTE!	This sample contains one VRM and three Sinks. For more information about how to set up VRM and Sink, please refer to **Working with Circuits** in the document ***SPD_Layout_UG***.

5.1.3 View Circuit Voltages

Follow these steps to view the circuit voltage variables:

1. Select Setup > Simulation View > Ckt Voltage… to open the following window.



Three minimized **Sink** voltages are selected.

2. Click **OK**.

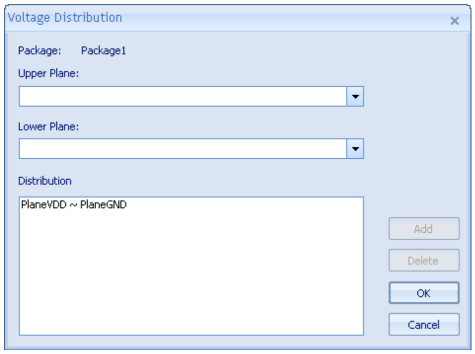
   In this example, the voltages of all three sink current locations have been defined. These voltages are the ones being minimized in this tutorial.

For more information about how to set circuit voltages, see Preparing for the Simulation > Setup the Sources and Observations in Spatial Mode > Specify Circuit Voltage in the document ***PowerSI_UG***.

5.1.4 View Voltage Distributions

Follow these steps to view voltage distributions:

1. Select Setup > Simulation View > 3D Spatial… to review the voltage distribution. The **Voltage Distribution** dialog box appears.



In this example, a plane pair **PlaneVDD ~ PlaneGND** is selected as shown in the **Distribution** text box.

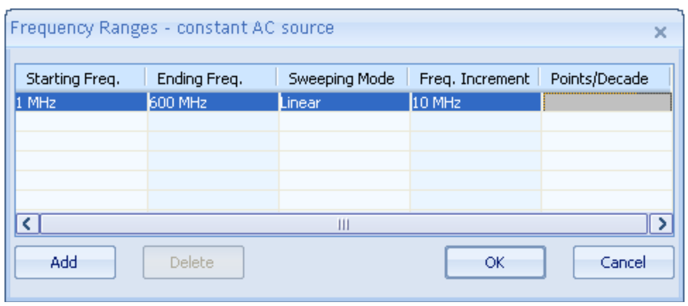
2. Click **OK**.

5.1.5 Set up Simulation Frequencies

Follow these steps to set up simulation frequencies:

1. Select Setup > Simulating Frequencies….

   The **Frequency Ranges** window opens.



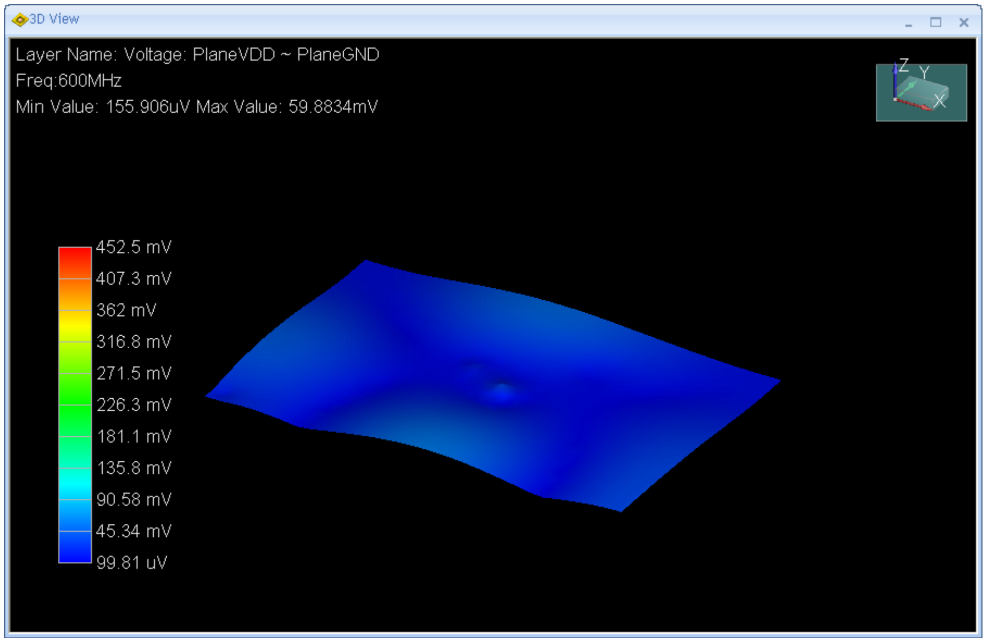
2. Change the following parameters:
   - **Starting Freq.** to 1 MHz
   - **Ending Freq.** to 600 MHz
   - **Sweeping Mode** to **Linear**
   - **Freq. Increment** to 10 MHz
3. Click **OK**.

5.1.6 Run a Spatial Mode Simulation

Follow these steps to perform a simulation and view the curves:

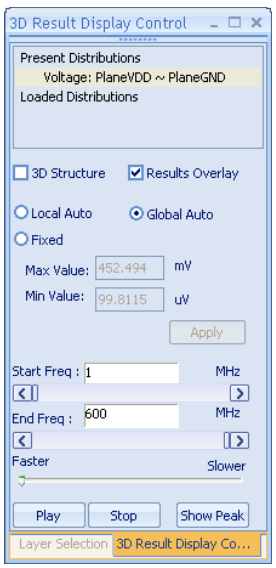
1. Click the **Start Simulation** button to start the simulation.

   The **3D View** window opens.

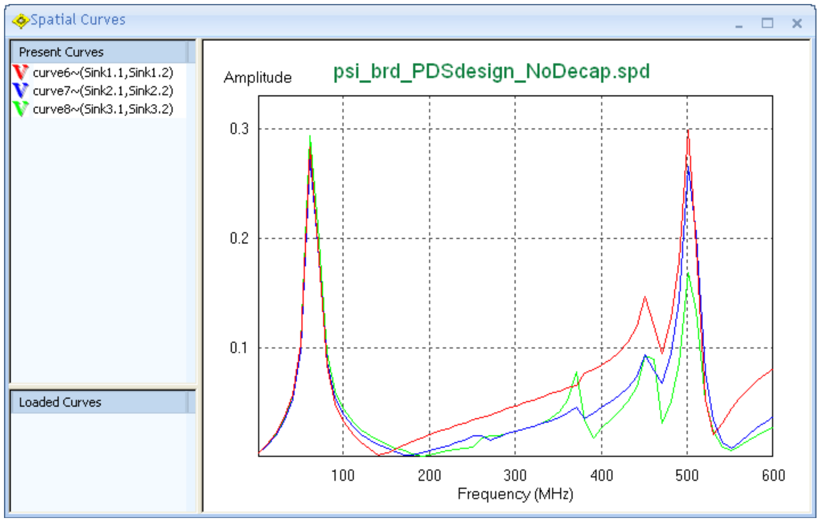


The window contains 3D plots for the voltages between the selected plane pairs.

The **3D Result Display Contro**l dialog box opens at the same time.



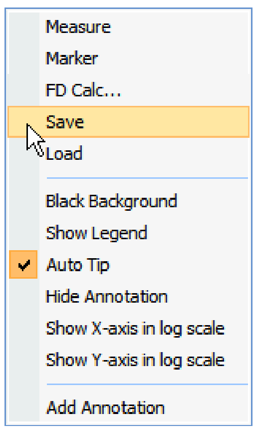
2. Select Window > Spatial Curves to view the **Spatial Curves** window.



This window contains 2D frequency responses of the selected voltage or current variables.

The red, blue, and green curves are the noise voltages at the three specified sink locations. Parallel resonance peaks appear in all three voltage locations.

3. Right-click the plot to show a pop-up menu.



4. Select **Save** to store the voltage curves as NoDecap.cur for later comparisons.

5.2 Decoupling Capacitor Placement (First Round)

This section describes how to reduce the noise voltages to below 100 mV up to 500 MHz.

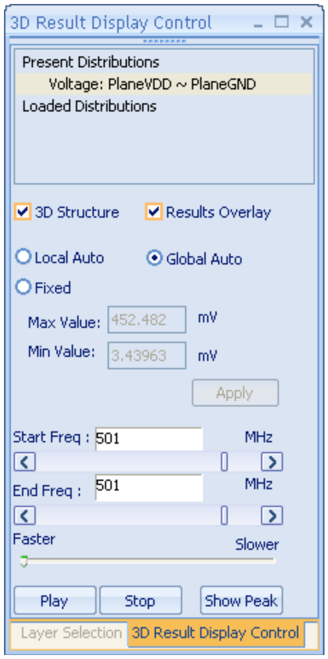
This section has the following topics:

- Review Spatial Voltage Distributions
- Determine Decoupling Capacitor Locations
- Determine Decoupling Capacitor Types and Values

5.2.1 Review Spatial Voltage Distributions

Follow these steps to review the spatial voltage distributions:

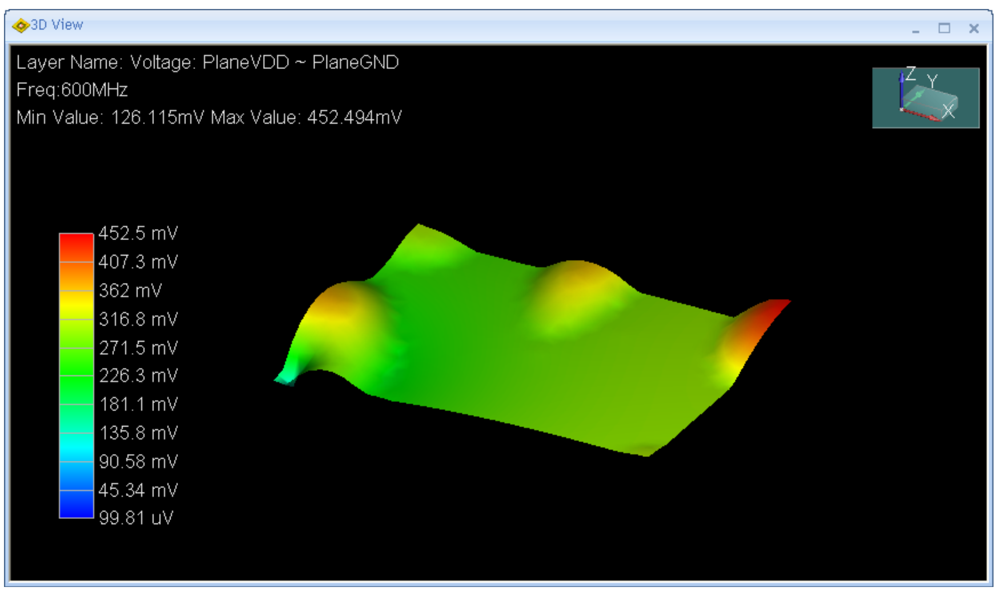
1. Select Window > 3D View and select the **3D Result Display Control** tab.



2. Input values in **Start Freq.** and **End Freq.**, or scroll the slide bars to adjust the values.

3. Click  to run.

4. Click  to display the maximum (peak) voltage distribution in the window. The maximum voltage within the frequency range appears for each mesh node.



The above figure shows three hot-spot locations where the color becomes orange to red. The peak noise voltages exceed 350 mV at these locations.

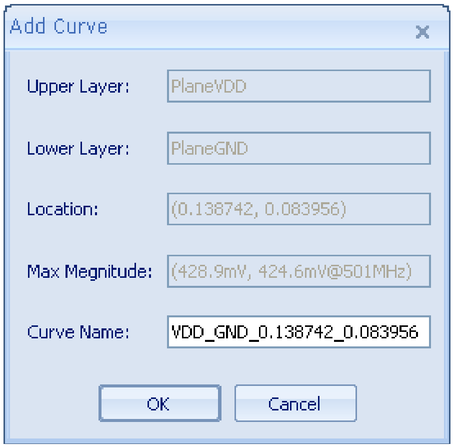
5.2.2 Determine Decoupling Capacitor Locations

Follow these steps to determine decoupling capacitor locations:

Add a Voltage versus Frequency curve at the hot-spot locations

1. Double-click a hot-spot location.

The **Add Curve** window opens.

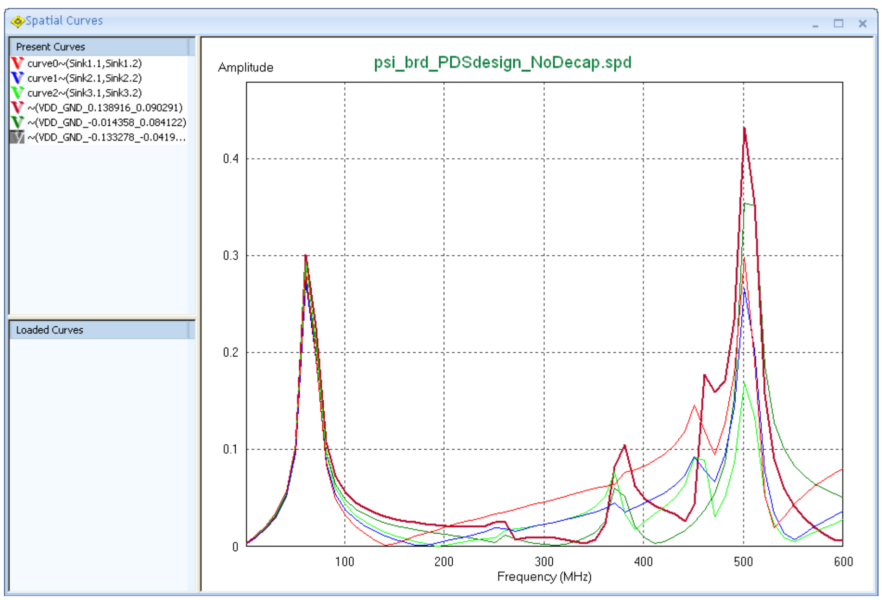


2. Click  to add a voltage response at this location to the **Spatial Curves** window.

Repeat **Step 1** and **2** to add other hot-spot voltage responses to the **Spatial Curves** window.

View the Voltage versus Frequency curve at the hot-spot locations

1. Select Window > Spatial Curves to display the **Spatial Curves** window.



Three voltage curves have parallel resonances at 61, 371, 461 and 501 MHz.

2. Close the current project without saving.

5.2.3 Determine Decoupling Capacitor Types and Values

This section describes how to determine decoupling capacitors and their values. The purpose of the determination is to remove the resonance peaks at the listed frequencies.

Use the capacitors with corresponding resonance frequencies below:

Capacitor Type	Resonance frequency (MHz)	Capacitor (F)	Inductance (H)	Resistance (ohm)
IDC0508	64.14	56 n	0.1 n	0.12

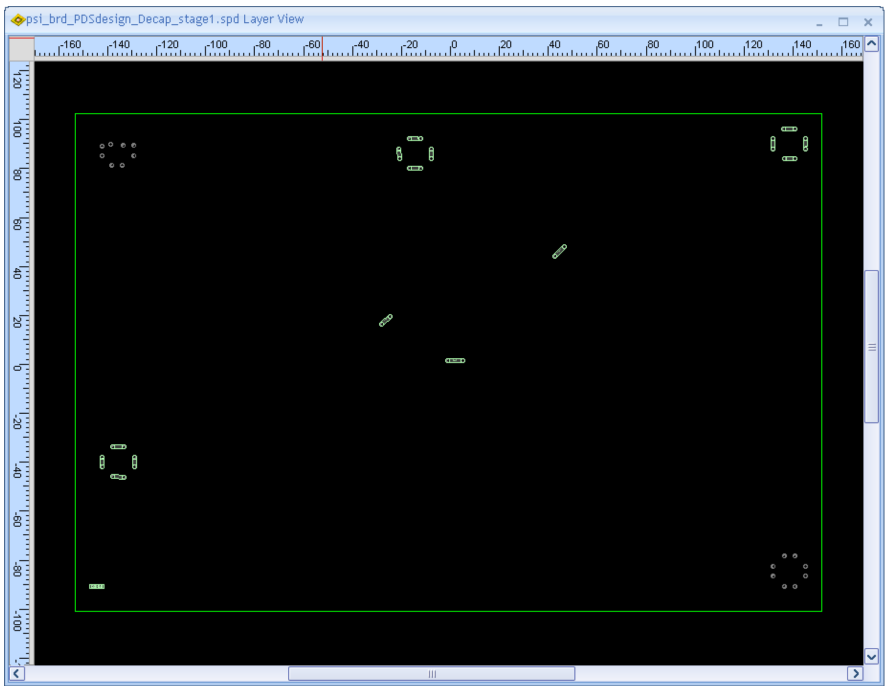
0508	357.75	330 p	0.6 n	1.39
0508	438.15	220 p	0.6 n	1.58
0508	484.39	180 p	0.6 n	1.65

The decoupling capacitors are located at where the 2D voltage curves are generated.

The pre-built file psi_brd_PDSdesign_Decap_stage1.spd contains three sets of decoupling capacitors as mentioned above. These capacitors are placed at the three hot-spot locations.

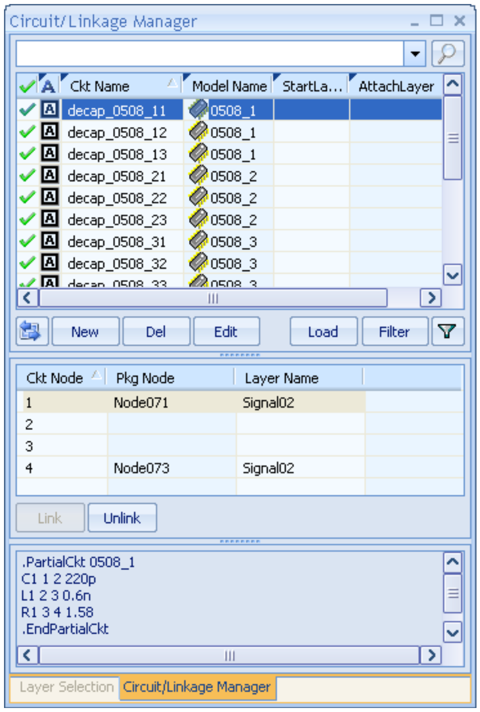
Follow these steps to run a simulation:

1. Select File > Open…, and open psi_brd_PDSdesign_Decap_stage1.spd from the <INSTALL_DIR>\SpeedXP\Samples\PowerSI\Hybrid\ directory.

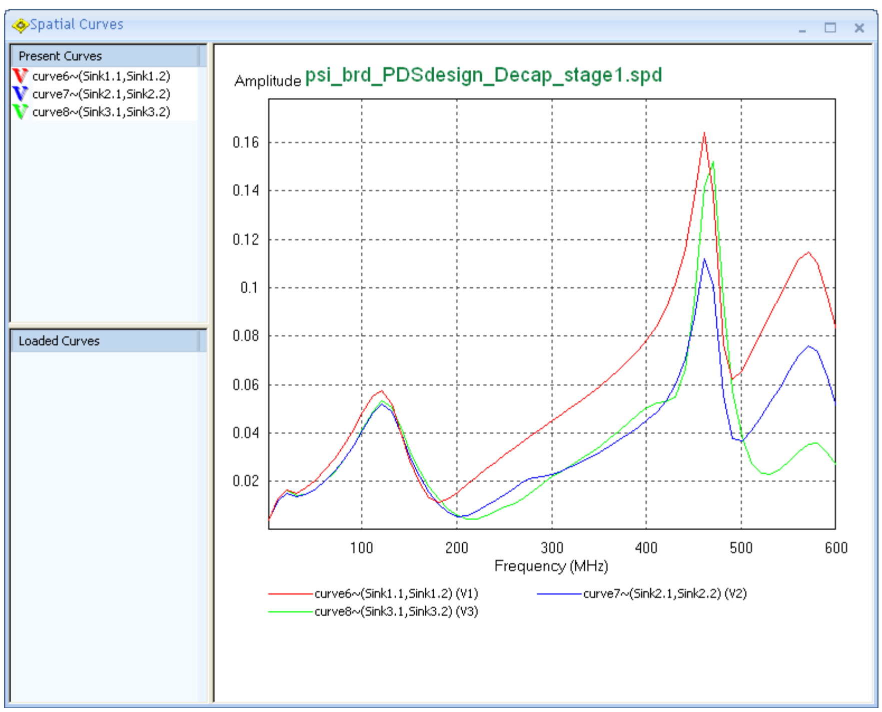


   This example has 12 decoupling capacitors placed at the desired locations.

2. Select Setup > Circuit/Linkage Manager to verify that the decoupling capacitors have been added.

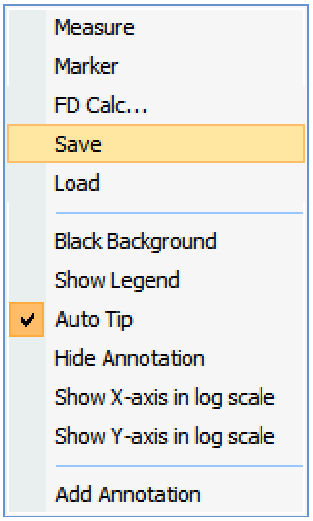


3. Click the **Start Simulation** button to start the simulation.
4. Select Window > Spatial Curves to switch to the **Spatial Curves** window.



The red, blue, and green curves are the noise voltages at the three specified sink locations with slightly reduced magnitude. In this example, the noise voltages still exceed the expected 100 mV in the frequency range between 400 to 500 MHz.

5. Right-click the plot to display a pop-up menu.



6. Select **Save** to save the voltage curves as Decap1.cur for later comparisons.

5.3 Decoupling Capacitor Placement (Second Round)

In the previous exercise, the noise voltages are lowered, but still violate specification in the frequency range between 400 to 500 MHz. In this section, the noise voltages between 400 to 500 MHz will be lowered to the expected 100 mV.

This section has the following topics:

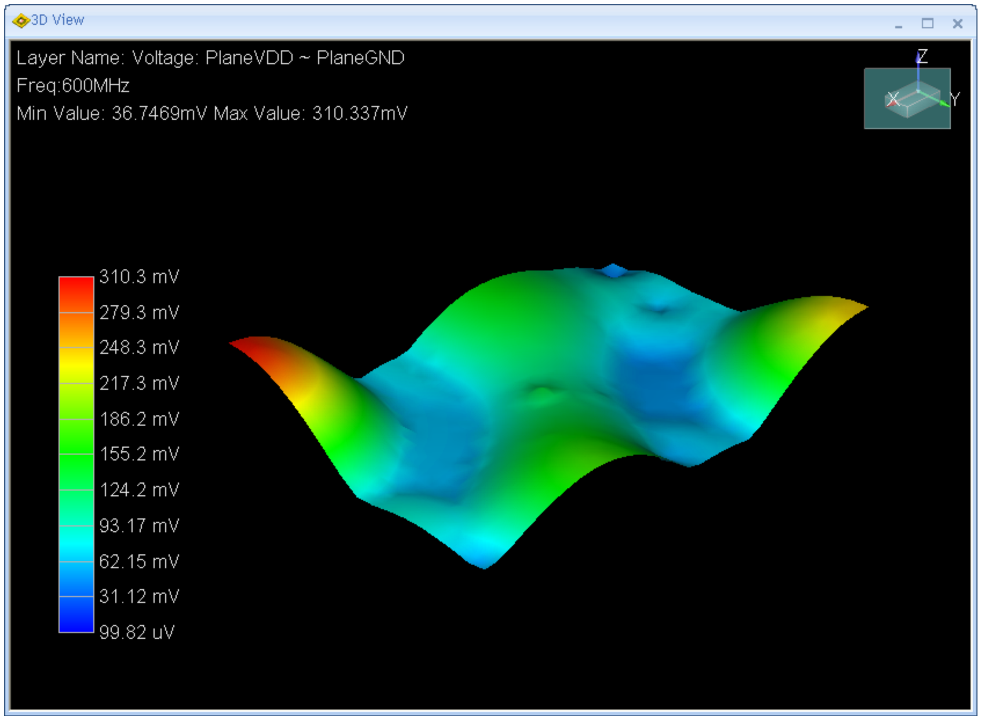
- Review Spatial Voltage Distributions
- Determine Decoupling Capacitor Locations
- Determine Decoupling Capacitor Types and Values

5.3.1 Review Spatial Voltage Distributions

Follow these steps to review the spatial voltage distributions:

1. Select Window > 3D View.
2. Select the **3D Result Display Control** dialog box and click the **Show Peak** button 

to display the maximum (peak) voltage distribution in the **3D View** window.



Two hot-spots are at the lower right and upper-left corner. The peak noise voltages exceed 200 mV at both locations.

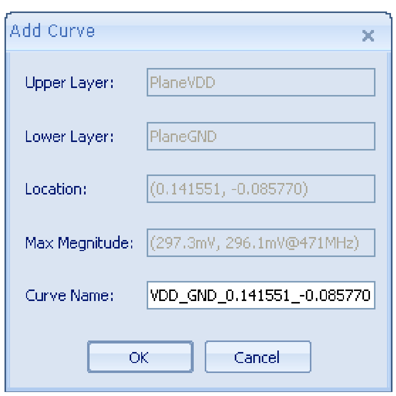
5.3.2 Determine Decoupling Capacitor Locations

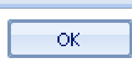
Follow these steps to determine decoupling capacitor locations:

Add a Voltage versus Frequency curve at hot-spot locations

1. Double-click a hot-spot location.

   The **Add Curve** dialog box opens.



2. Click 

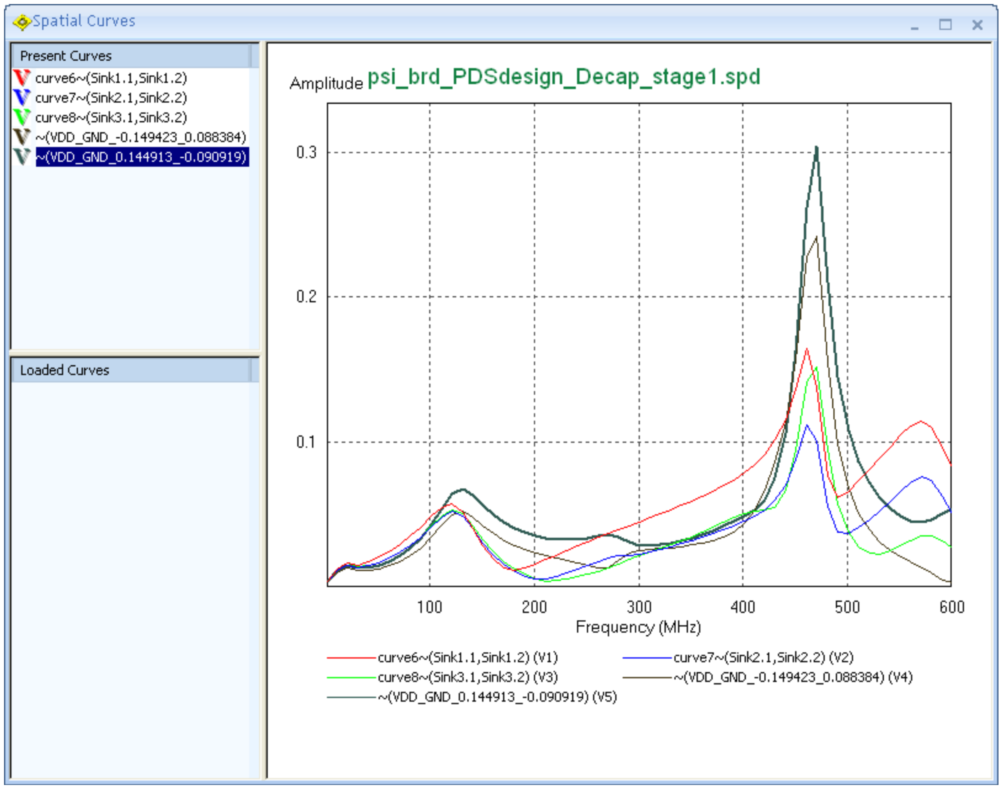
to add the voltage response at this location to the **Spatial Curves** window.

Repeat **Step 1** and **2** to add other hot-spot voltage responses to the **Spatial Curves** window.

View the Voltage versus Frequency curve at hot-spot locations

1. Select Window > Spatial Curves to display the **Spatial Curves** window.

   Two new voltage response curves are shown in the panel: VDD_GND_–0.149423_0.088384 and VDD_GND_0.144913_–0.090919.



Both of voltage curves have parallel resonances at 131 and 471 MHz.

2. Close the current project without saving.

5.3.3 Determine Decoupling Capacitor Types and Values

This section describes how to determine decoupling capacitors and their values. The purpose of the determination is to remove the parallel resonance peaks at the frequencies listed.

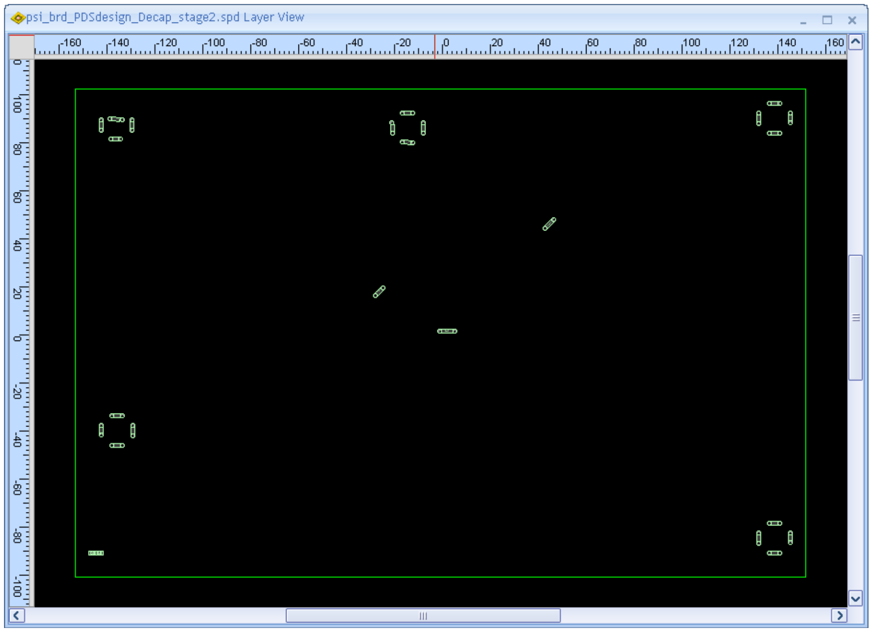
Use the capacitors with corresponding series resonance frequencies below:

Capacitor Type	Resonance frequency (MHz)	Capacitor (F)	Inductance (H)	Resistance (ohm)
IDC0508	138.56	12 n	0.1 n	0.23
0508	438.15	12 p	0.6 n	1.58

The decoupling capacitors are located at the hot-spots where the 2D voltage curves are generated.

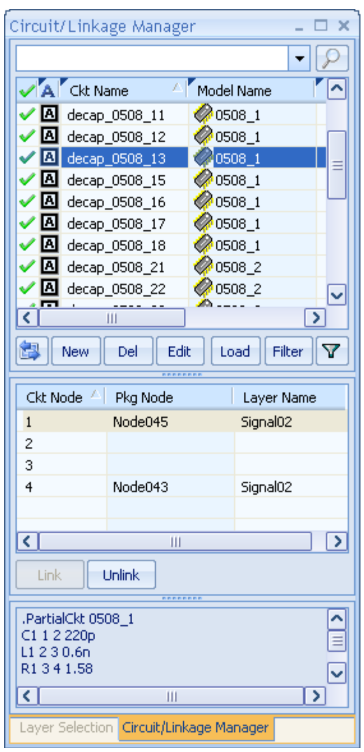
The prebuilt file psi_brd_PDSdesign_Decap_stage2.spd contains two IDC0508 capacitors and two 0508 capacitors placed at each of the two hot-spot locations.

1. Select File > Open…, and open psi_brd_PDSdesign_Decap_stage2.spd from the <INSTALL_DIR>\SpeedXP\Samples\PowerSI\Hybrid\ directory.



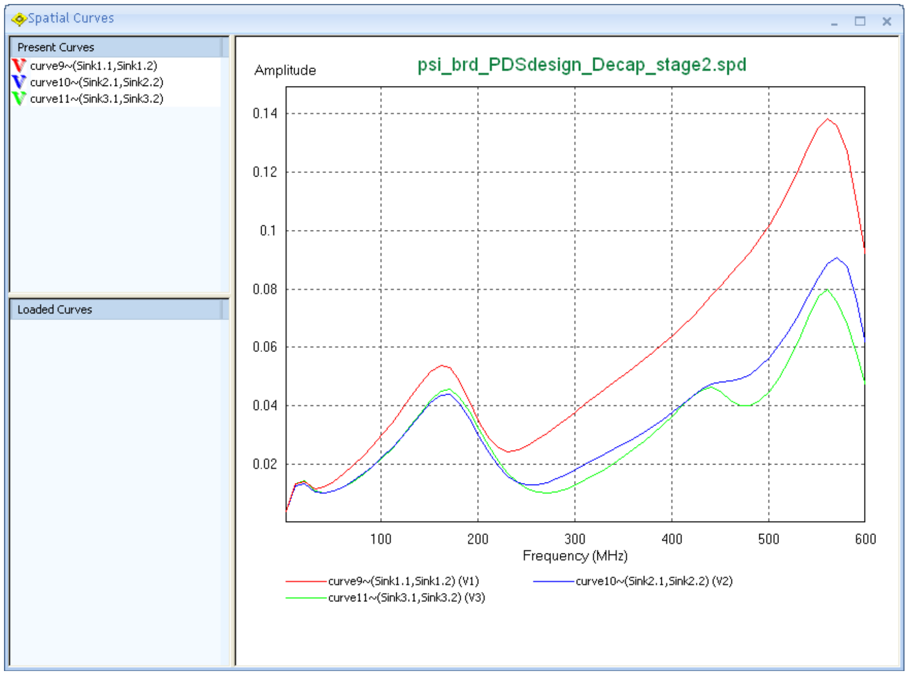
Eight additional decoupling capacitors have been added.

2. Select Setup > Circuit/Linkage Manager to display the **Circuit/Linkage Manager** dialog box to verify that the decoupling capacitors have been added.



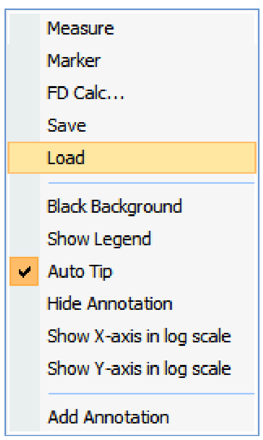
3. Click the **Start Simulation** button to start the simulation.

The **Spatial Curves** window opens like the following figure.



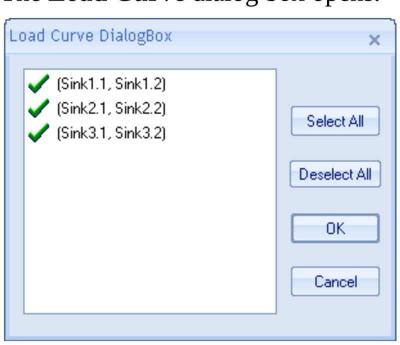
The red, blue, and green curves are the noise voltages at the three specified sink locations with slightly reduced magnitude. In this example, the noise voltages have been lowered up to 500 MHz. The noise voltages up to 400 MHz are lower than 60 mV.

4. Right-click the plot to display the pop-up menu.

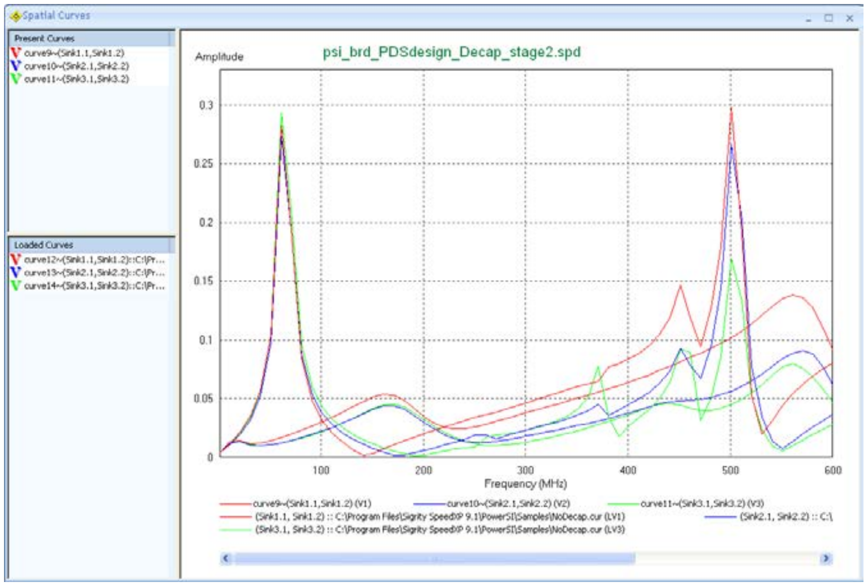


5. Select **Load** to load the stored voltage curves in NoDecap.cur.

The **Load Curve** dialog box opens.



6. Click Select All to load all the curves and click OK.



There has been significant noise voltage reduction at the various sink locations.

7. Close the current project without saving.

5.4 Self and Transfer Impedance of the Power Delivery System

This section describes how to view the self and transfer impedance (impedance parameter matrix) with and without decoupling capacitors, and further demonstrate the improvements on the performance of the power delivery system.

This section has the following topics:

- Simulate Self and Transfer Impedance Curves
- Run Simulation with Decoupling Capacitors

5.4.1 Simulate Self and Transfer Impedance Curves

This section describes how to show hooked ports, run a simulation without decoupling capacitors, and view different ports after the simulation.

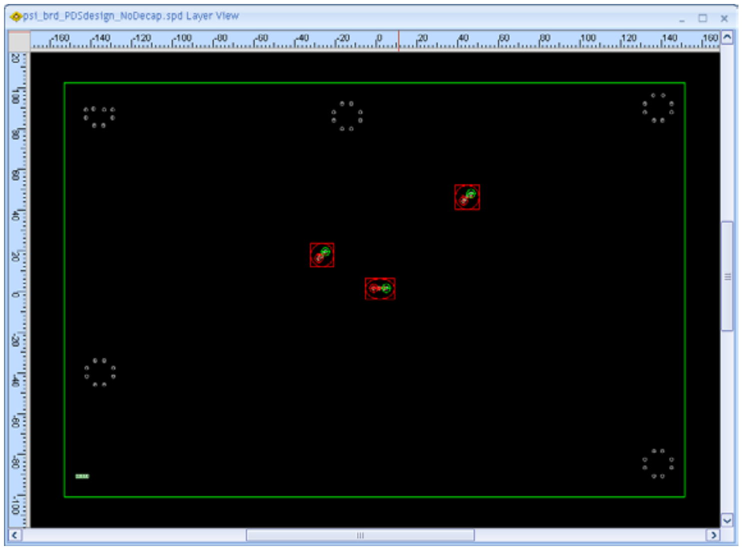
Show hooked ports and run a simulation without decoupling capacitors

1. Select Mode > Extraction Mode.
2. Select File > Open…, and open the file psi_brd_PDSdesign_NoDecap.spd from the <INSTALL_DIR>\SpeedXP\Samples\PowerSI\Hybrid\ directory.

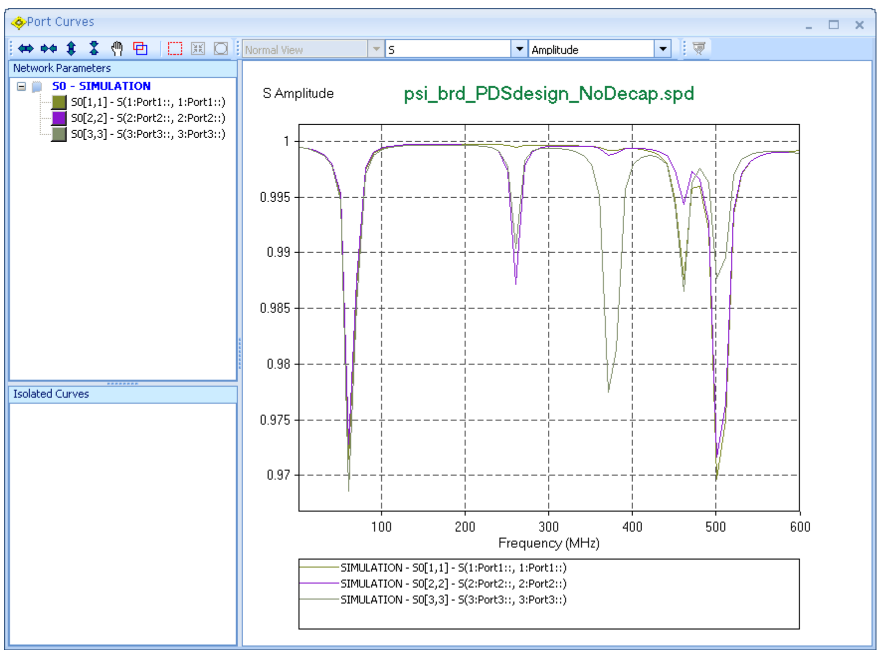
NOTE!	In some cases, you might see a warning regarding the circuit definitions of the source components. The **Extraction Mode** does not accept independent voltage or current sources that do not equal zero, and gives out a warning. In this example, the voltage source = 0. If you see this warning and wish to keep the circuit definitions of the source components, click Yes.

3. Select View > Show > Show Hooked Ports to enable hooked port view.

   Three pre-specified ports at the sink locations appear as in the following figure:

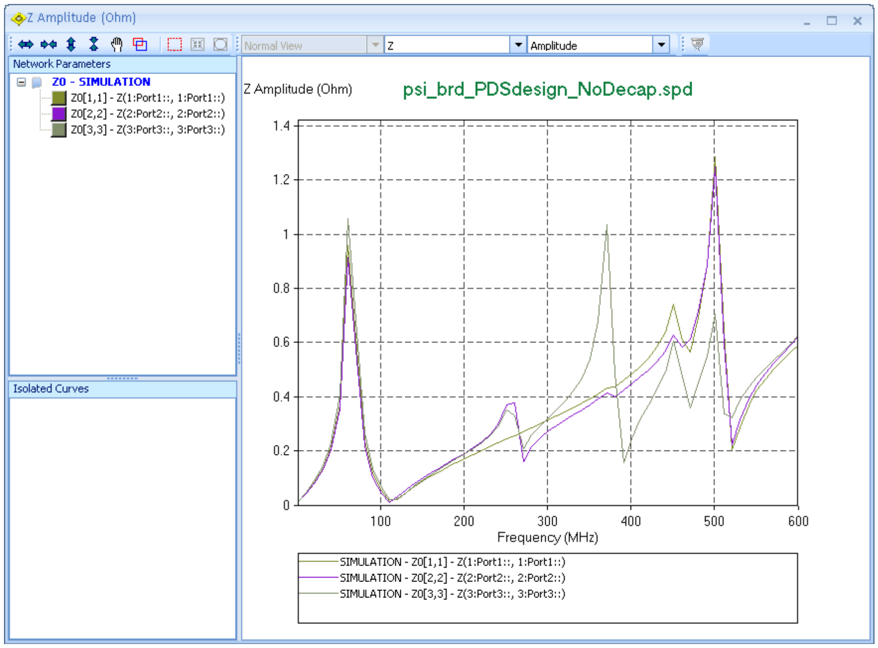


4. Click the **Start Simulation** button  to start the simulation without decoupling capacitors. All port curves appear on the plot.



5. Choose the **Z** (impedance) parameter from the **Curve Settings** toolbar.

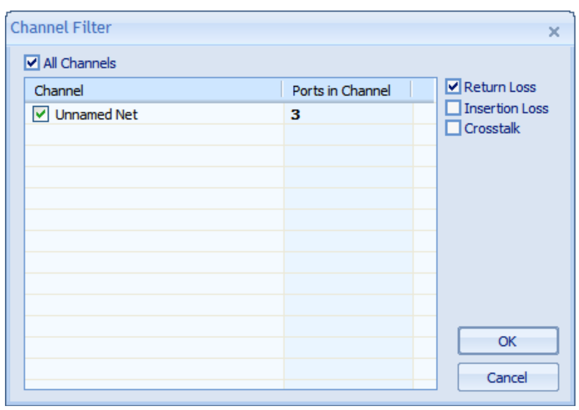
The resulting curve presents a Z-parameter.



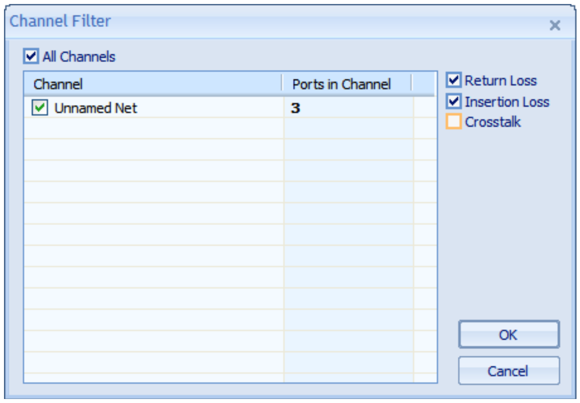
View all different ports

By default the display shows the S-parameter matrix diagonal components or reflection. When you select **Z** from the **Curve Settings** toolbar, the self impedance at each port shows. To view transfer impedance, plot the transmission parameters.

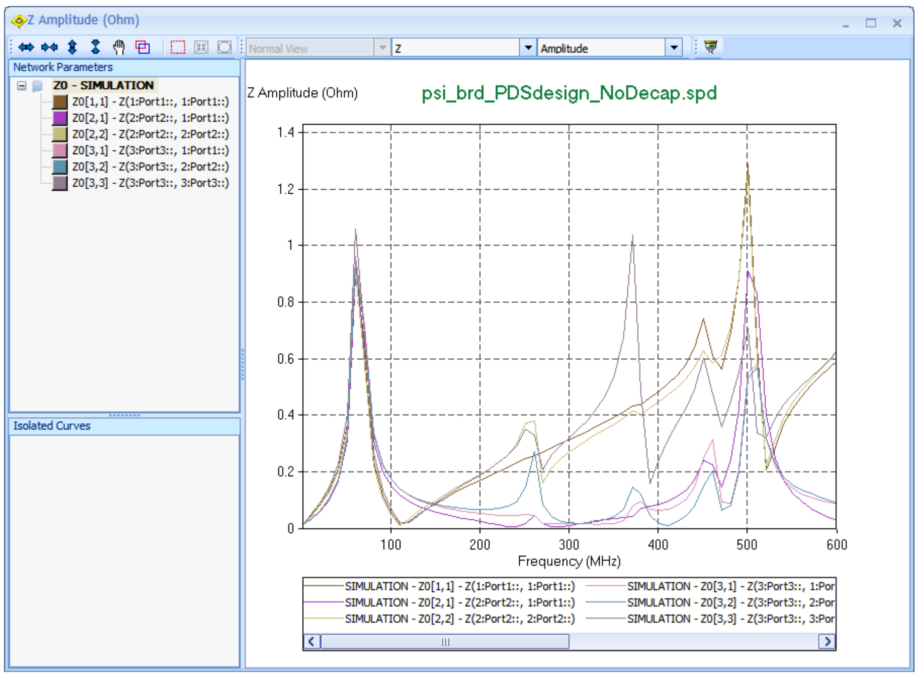
1. Click the **Channel Filter** button on the **Curve Operations** toolbar to open the **Channel Filter** dialog box.



2. Check **Insertion Loss**.

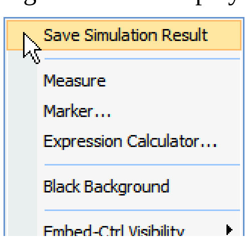


3. Click  to show impedance parameters.

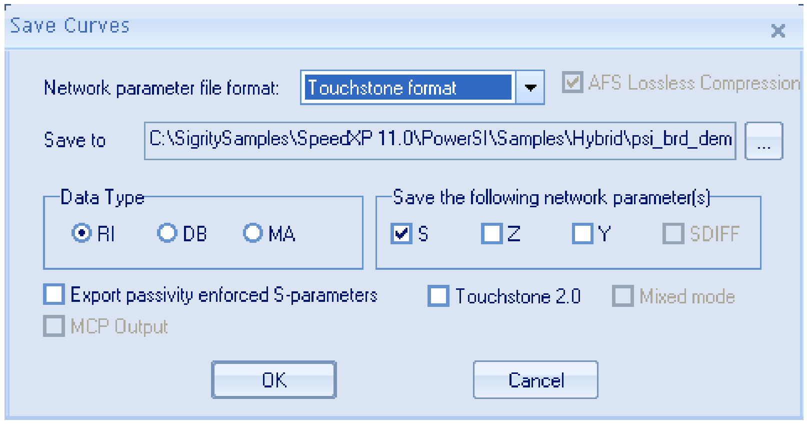


This window shows all of the impedance parameters, both self and transfer impedance for the top half matrix.

4. Right-click the display to open a pop-up menu.



5. Select **Save Simulation Result** to save the S-parameters and impedance matrix in the psi_brd_PDSdesign_NoDecap.S3P for later comparison.

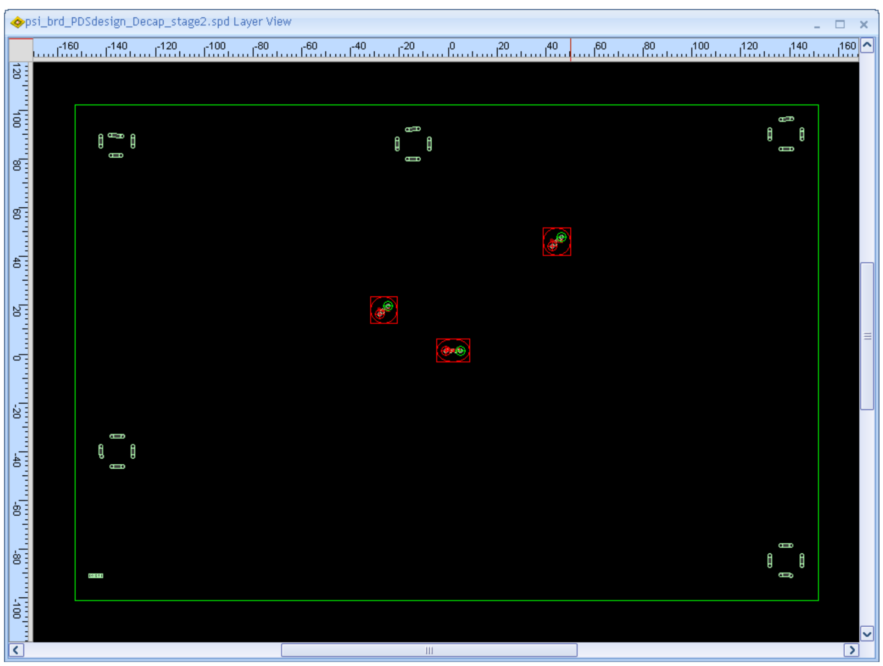


6. Select Window > Layer View to switch to the **Layer View** window.
7. Select File > Close to close the current project.

5.4.2 Run Simulation with Decoupling Capacitors

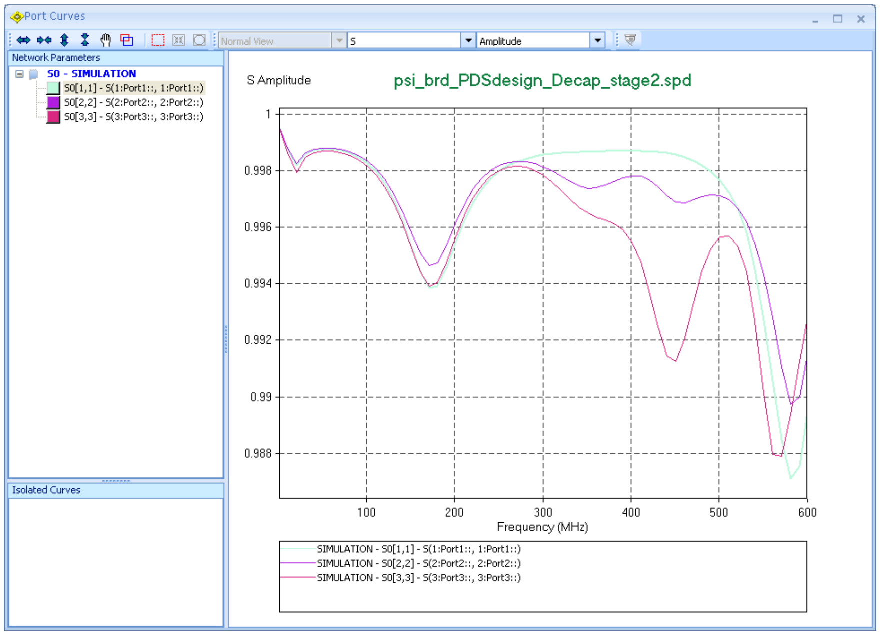
1. Select File > Open… to open the file psi_brd_PDSdesign_Decap_stage2.spd from the PowerSI Samples directory.

NOTE!	In some cases, you might see a warning regarding the circuit definitions of the source components. The Extraction Mode does not accept independent voltage or current sources that do not equal zero, and gives out a warning. In this example, the voltage source = 0. If you see this warning and wish to keep the circuit definitions of the source components, click **Yes**.

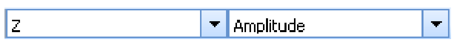


2. Press the **Start Simulation** button to start the simulation.

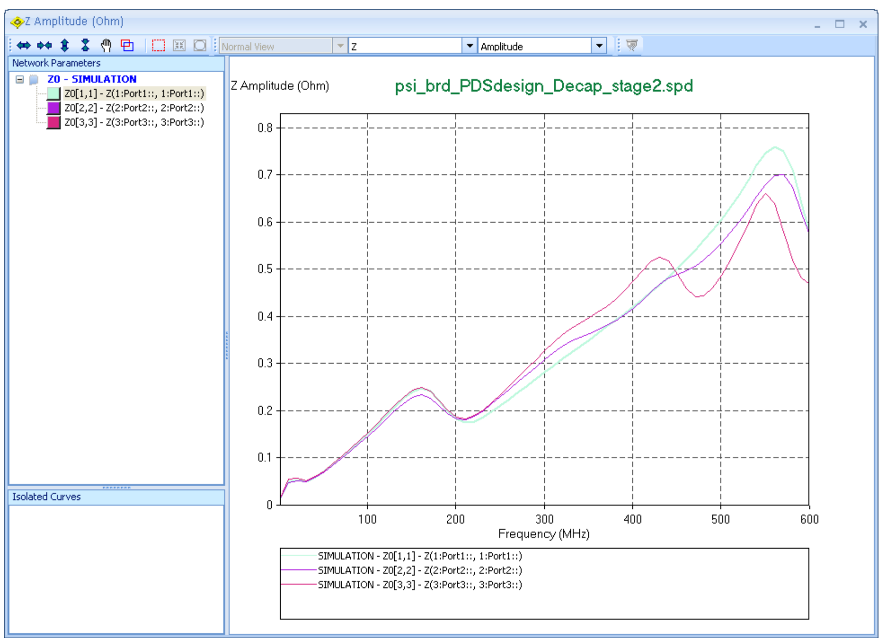
All port curves appear on the plot.



3. Choose the **Z** (impedance) parameter from the **Curve Settings** toolbar.



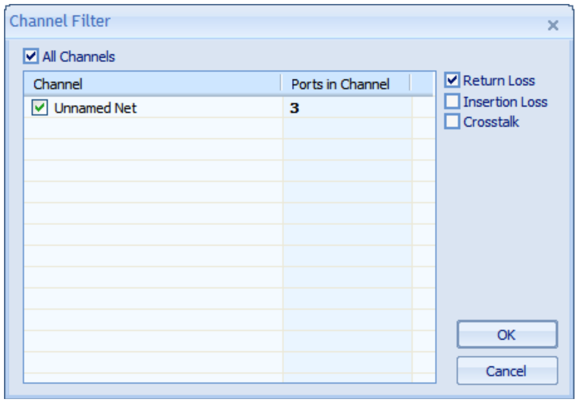
The resulting curve presents a Z-parameter.



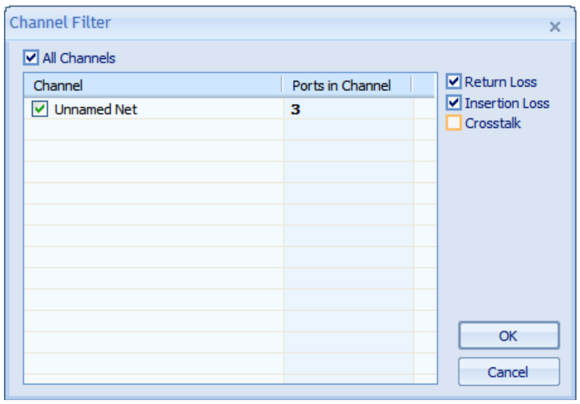
View all different ports

By default the display shows the S-parameter matrix diagonal components or reflection. When you select **Z** from the **Curve Settings** toolbar, the self impedance at each port shows. To view transfer impedance, plot the transmission parameters.

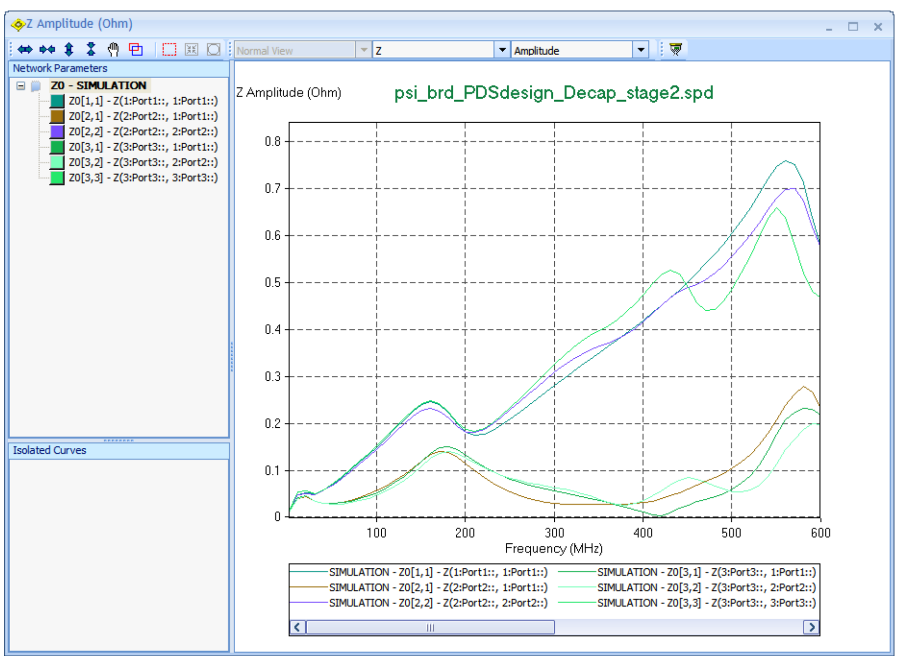
1. Click the **Channel Filter** button  on the **Curve Operations** toolbar to open the **Channel Filter** dialog box.



2. Check **Insertion Loss**.

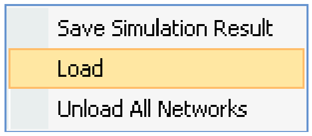


3. Click  to show impedance parameters.



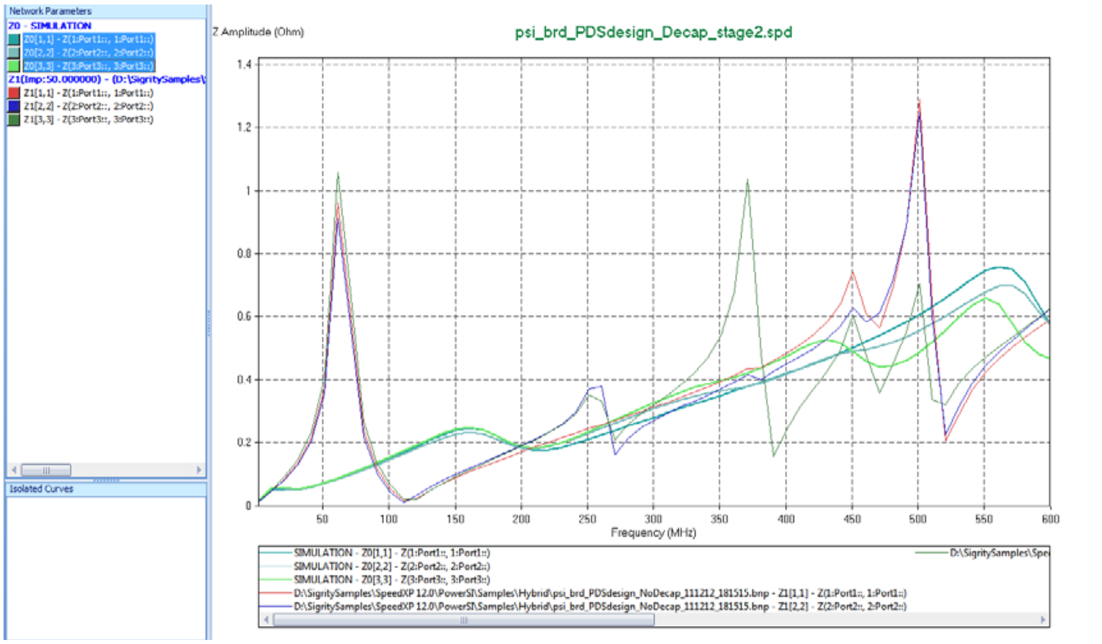
This window shows all of the impedance parameters, both self and transfer impedance for the top half matrix.

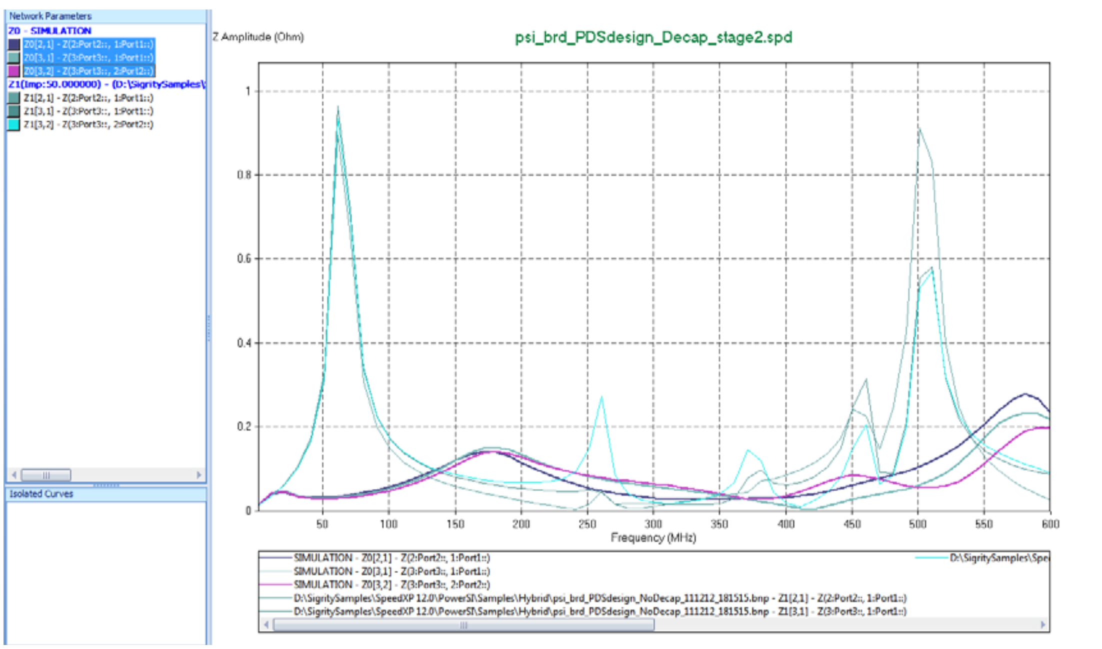
4. Right-click the **Network Parameters** to open a pop-up menu.



5. Select **Load** to load psi_brd_PDSdesign_NoDecap.S3P for comparison.

   The curves display like in the following figure.





6. Select Window > Layer View to switch to the **Layer View** window.
7. Select File > Close to close the current project.

5.5 Resonance Mode

In **PowerSI**, the resonance mode solver computes and displays all the resonant modes between predefined frequency ranges. User can also specify the number of resonant modes directly.

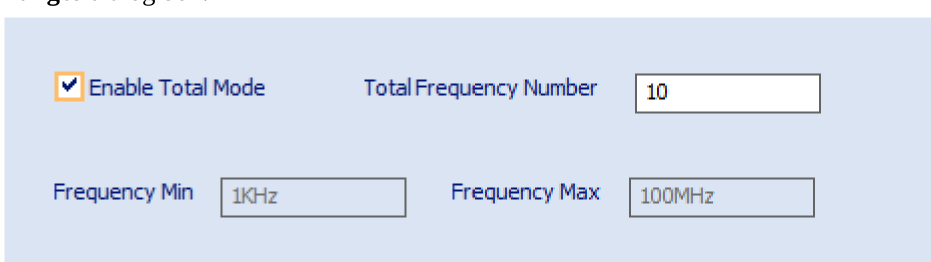
Follow these steps to start resonance mode analysis:

5.5.1 Start PowerSI and Switch to the Resonance Mode

1. Launch PowerSI.
2. Select Mode > Resonance Mode to switch to **Resonance Mode**.
3. Select File > Open… to open the file psi_brd_PDSdesign_Decap_stage2.spd from the <INSTALL_DIR>\SpeedXP\Samples\PowerSI\Hybrid\ directory.

5.5.2 Set up Frequency Range

1. Select Tools > Options > Edit Options… > Resonance > Setting to show the **Frequency Ranges** dialog box.



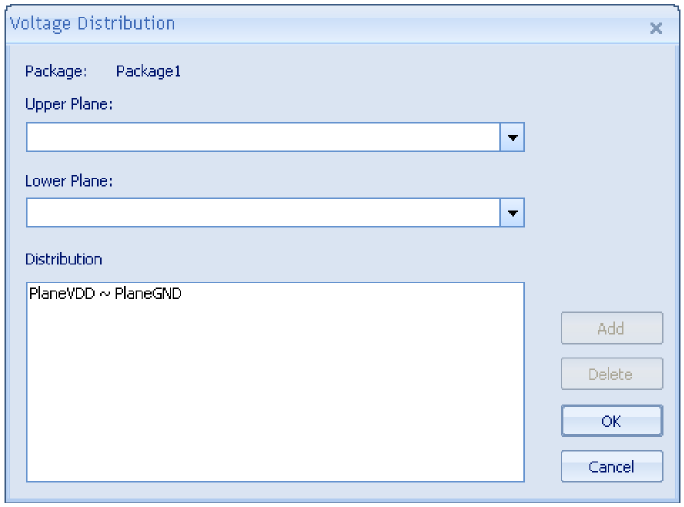
2. To extract all the resonant modes between frequency ranges, specify **Frequency Min** and **Frequency Max** in the field next to them.

   or

   To extract total number of resonant modes, check **Enable Total Mode** and enter a number in the field of **Total Frequency Number**.

3. Click 
.

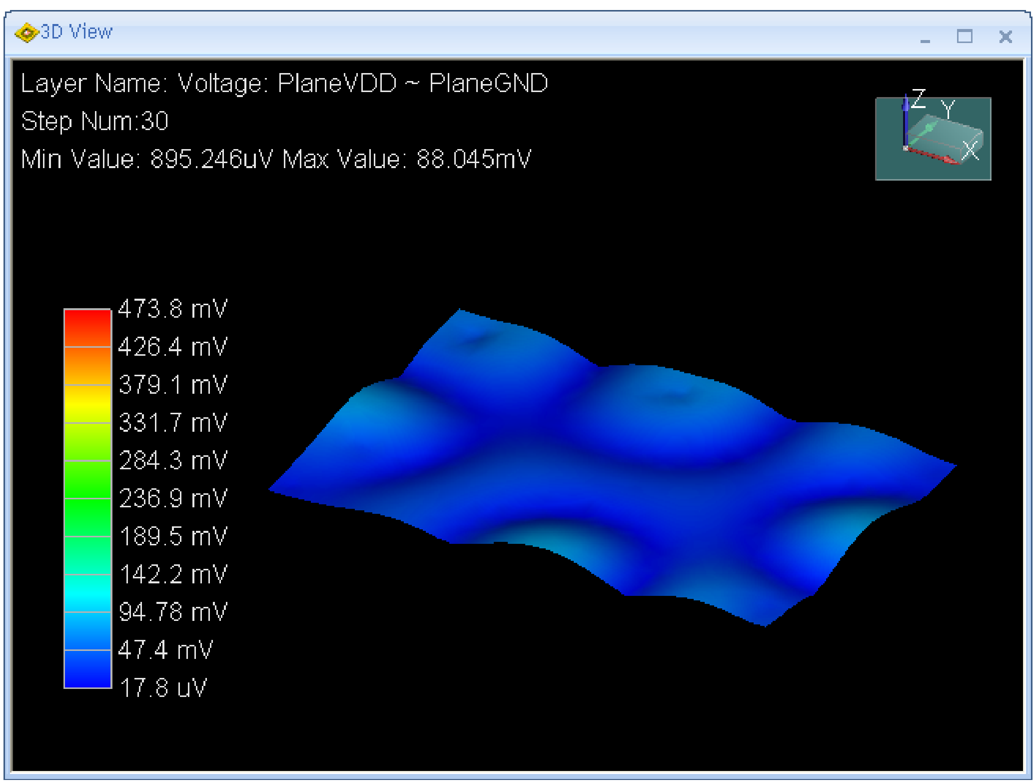
4. (Optional) Select Setup > Simulation View > 3D Spatial… to set up the 3D spatial voltage display.



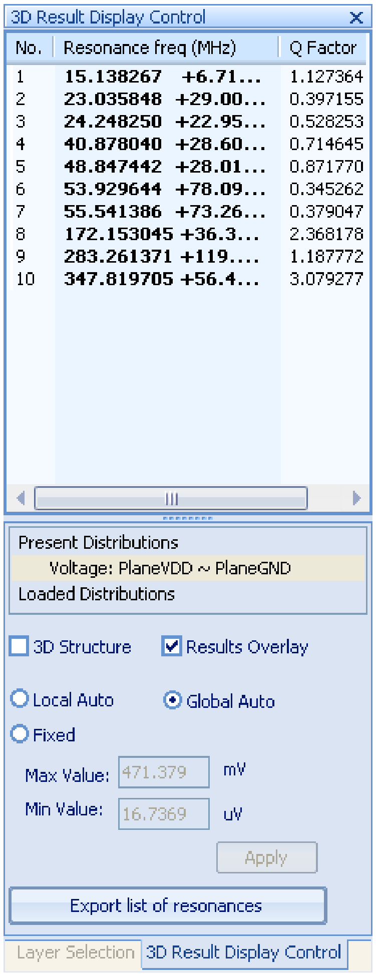
5.5.3 Run a Simulation

Click the **Start Simulation** button  to start a simulation.

The 3D view looks as shown below.

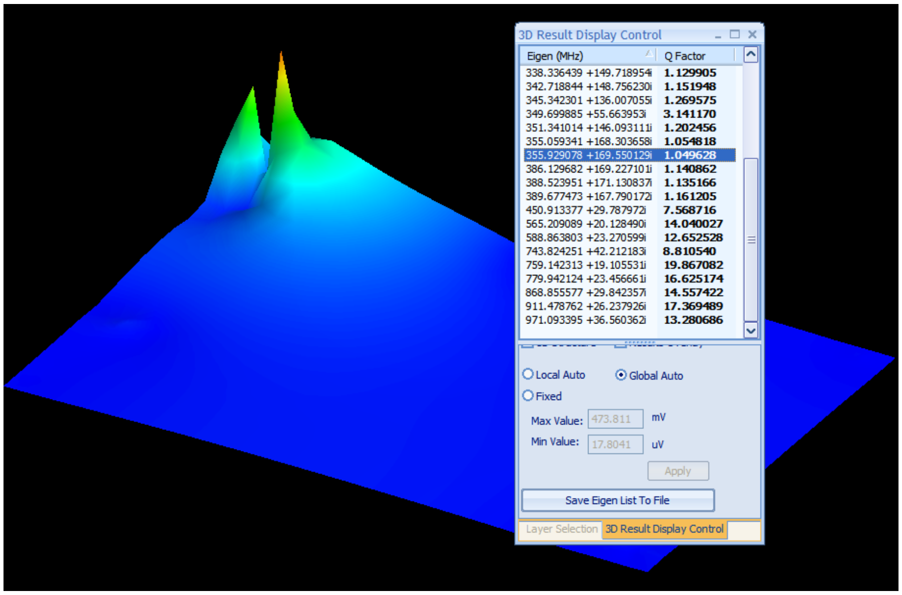


PowerSI outputs a table which lists complex resonant frequencies and Q factors in the **3D Result Display Control** window as shown below:



If the 3D spatial is set up, **PowerSI** also shows the normalized voltage field distribution at corresponding resonant frequencies.

You can click an **Eigen** entry to view the normalized voltage field distribution at each potential resonant frequency.


3D Result Display Control
Eigen (MHz)
Q Factor
338.336439 +149.718954i 1.129905
342.718844 +148.756230i 1.151948
345.342301 +136.007055i 1.269575
349.699885 +55.663953i 3.141170
351.341014 +146.093111i 1.202456
355.059341 +168.303658i 1.054818
355.929078 +169.550129i 1.049628
386.129682 +169.227101i 1.140862
388.523951 +171.130837i 1.135166
389.677473 +167.790172i 1.161205
450.913377 +29.787972i 7.568716
565.209089 +20.128490i 14.040027
588.863803 +23.270599i 12.652528
743.824251 +42.212183i 8.810540
759.142313 +19.105531i 19.867082
779.942124 +23.456661i 16.625174
868.855577 +29.842357i 14.557422
911.478762 +26.237926i 17.369489
971.093395 +36.560362i 13.280686
Local Auto
Global Auto
Fixed
Max Value: 473.811 mV
Min Value: 17.8041 uV
Apply
Save Eigen List To File
Layer Selection
3D Result Display Control

6 Project 4 IR Drop Analysis for Power Delivery Systems

This chapter describes how to analyze the DC current drop (IR drop) for power delivery system. The IR drop refers to the off-set of the DC voltage supply due to the distributed resistance associated with the metals, such as shapes and traces, in the power delivery system.

This chapter uses the following files as examples:

- psi_DCR1_200x10mm_1plane.spd
- psi_DCR2_100x1mm_1plane.spd
- psi_DCR3_100x1mm_2planes.spd
- psi_brd_IR_drop_SM.spd

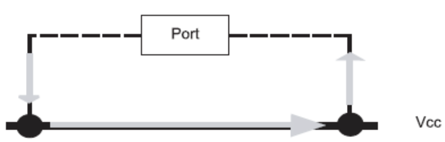
These examples give detailed steps of how to perform DC analysis for most situations and analysis. The first two examples in this chapter show the computation of resistance associated with power/ground nets. The last example explains how to determine the voltage drop in the spatial mode.

This chapter has the following sections:

- Power Delivery System Resistance (I)
- Power Delivery System Resistance (II)
- Power Delivery System Resistance (III)
- Simulating IR Drop in the Spatial Mode

6.1 Power Delivery System Resistance (I)

In this example, you compute the resistance associated with the VCC net as shown below:



The Vcc plane has the following specifications:

- 200 mm long (L)
- 10 mm wide (W)
- 5 um thick (T)
- Conductivity 5.8e7 S/m (σ)
- Two rows of vias, acting as testing points, are 0.5 mm away from the plane edge.

Consequently, the actual DC path for the plane is 199 mm (L). The vias are 1 um long and have negligible DC resistance. Because the plane is a thin rectangular strip, the DC resistance of the Vcc plane can be found analytically using the following formula:

$$R = L/(\sigma \times W \times T) = 0.06862 Ohms$$

The **PowerSI** simulation results should closely match the formula results.

This section has the following sections:

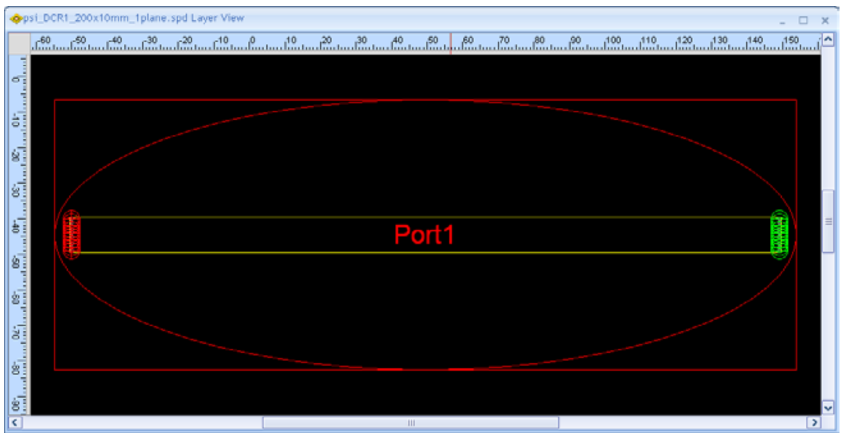
- Open a Project

- Set up Simulation Frequencies
- Turn on Inter-plane Coupling
- Run a Simulation

6.1.1 Open a Project

This section describes how to open a project and show hooked ports in the **Extraction Mode**.

1. Launch PowerSI.
2. Select File > Open… and choose the file psi_DCR1_200x10mm_1plane.spd from the <INSTALL_DIR>\SpeedXP\Samples\PowerSI\Hybrid\ directory.
3. Select View > Show > Show Hooked Ports to make sure that the **Show Hooked Ports** option is enabled.

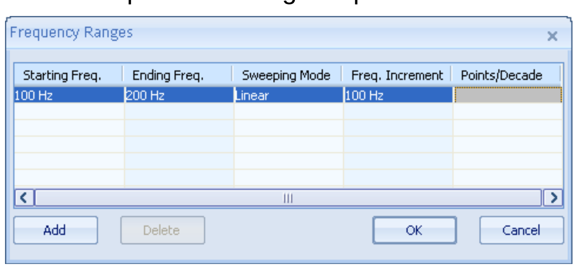


In this example, a port has been specified at where the positive terminal connects to multiple points at the left end of the VCC plane, and the negative terminal connects to multiple points at the right end of the VCC plane.

6.1.2 Set up Simulation Frequencies

Follow these steps to set up simulation frequencies:

1. Select Setup > Simulating Frequencies….



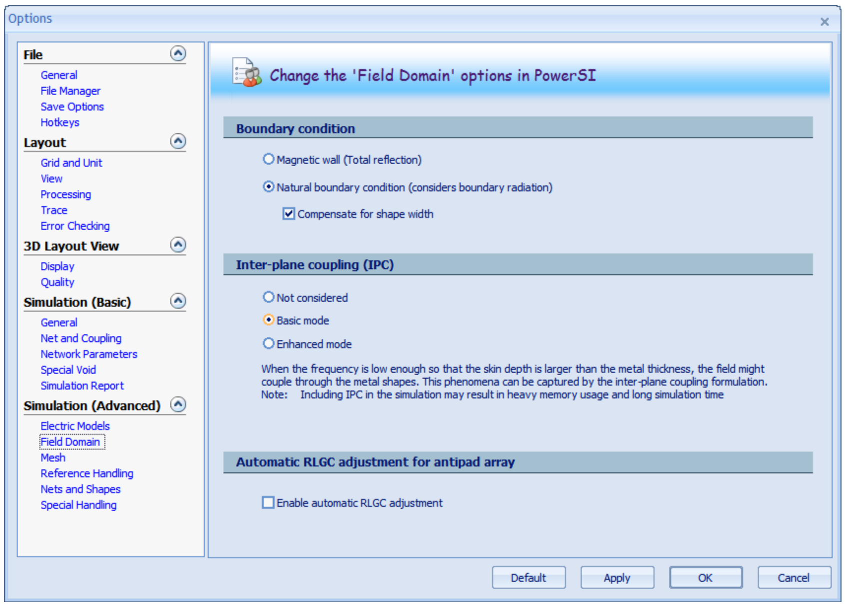
2. Change the following parameters:
   - **Starting Freq.** to 100 Hz
   - **Ending Freq.** to 200 Hz
   - **Sweeping Mode** to **Linear**
   - **Freq. Increment** to 100 Hz
3. Click **OK**.

TIP!	Set the simulation frequencies as these values when you use PowerSI to analyze DC.

6.1.3 Turn on Inter-plane Coupling

Follow these steps to turn on the inter-plane coupling:

1. Select Tools > Options > Edit Options… > Simulation (Advanced) > Field Domain.
2. Select **Basic mode** under **Inter-plane coupling (IPC)**.



3. Click 

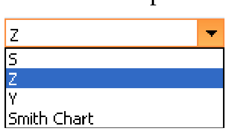
.

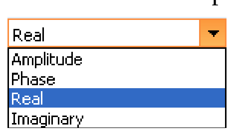
TIP!	Always enable **Inter-plane coupling** to ensure accuracy during the DC analysis.

6.1.4 Run a Simulation

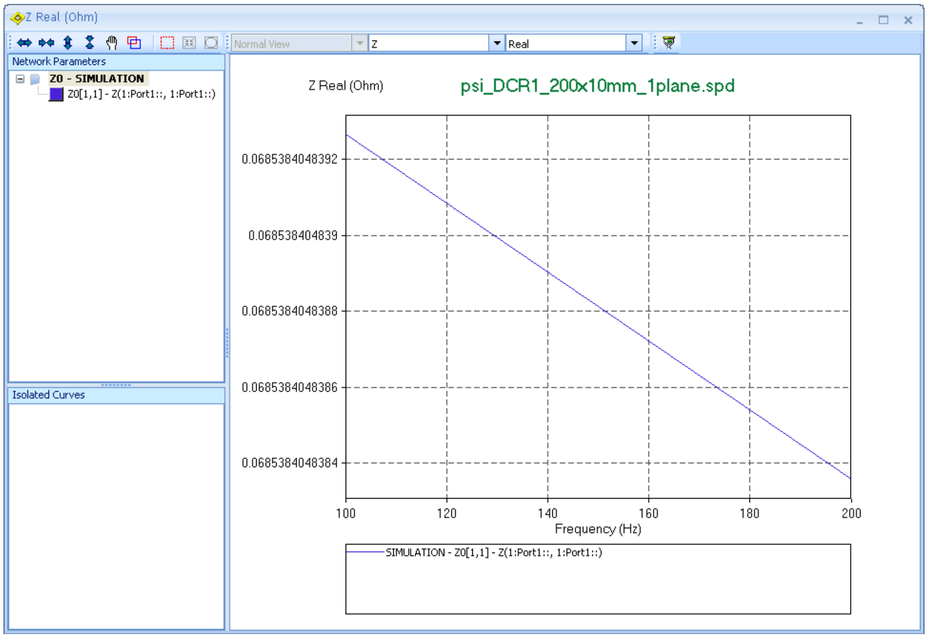
Follow these steps to run a simulation:

1. Press the **Start Simulation** button  to start the simulation.
2. Choose the **Z** parameter and **Real** from the drop-down menus on the **Curve Settings** toolbar.





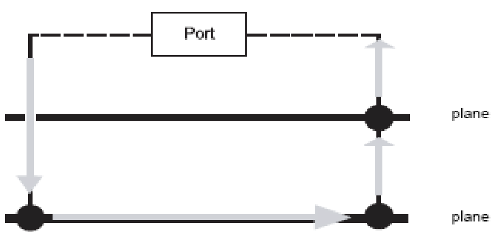
The curve looks as shown below:



The resistance result is around 0.068500 ohms which is consistent with the analytical result.

6.2 Power Delivery System Resistance (II)

In this example, you compute the resistance associated with the Vcc net as shown below.



Both the Vcc and Vss planes have the same specifications:

- 100 mm long (L)
- 1 mm wide (W)
- 5 um thick (T)
- Conductivity is 5.8e7 S/m (σ).

Because the plane is a thin rectangular strip, the DC resistance of either plane can be found analytically using the following formula:

$$R = L/(\sigma \times W \times T) = 0.345 Ohms$$

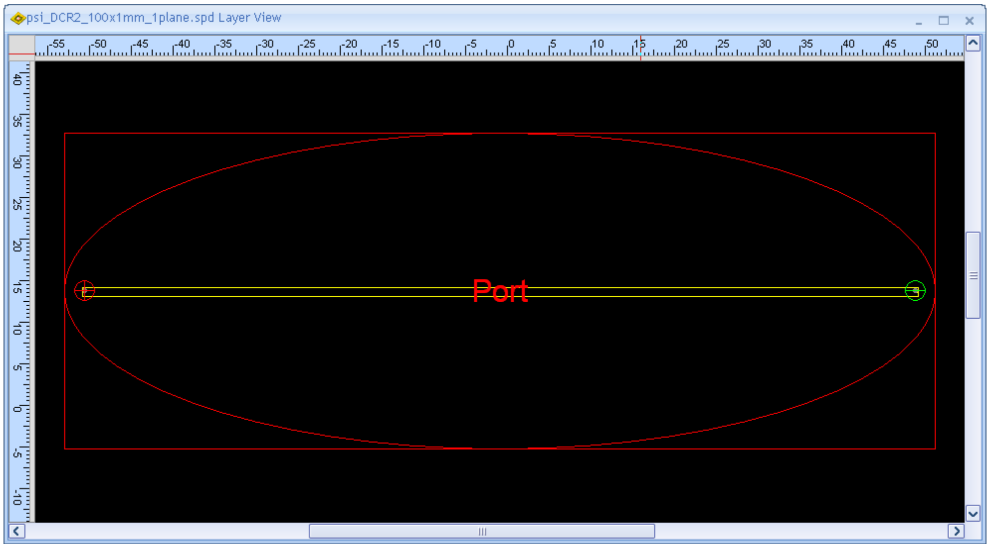
The **PowerSI** simulation results should closely match the formula results.

6.2.1 Compute Resistance

Follow these steps to compute resistance:

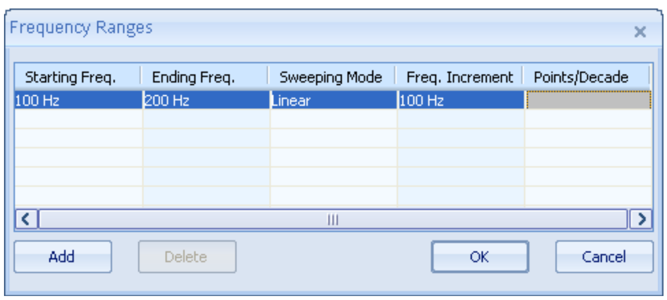
1. Open a Project and load the file psi_DCR2_100x1mm_1plane.spd.

   See ***5.1.1 Open a Project***.



In this example, the positive terminal is connected to the Vcc plane on the left through a through-hole via, and a negative terminal connected to the Vss and Vcc planes on the right.

2. Select Setup > Simulating Frequencies…. See ***5.1.2 Set up Simulation Frequencies***.

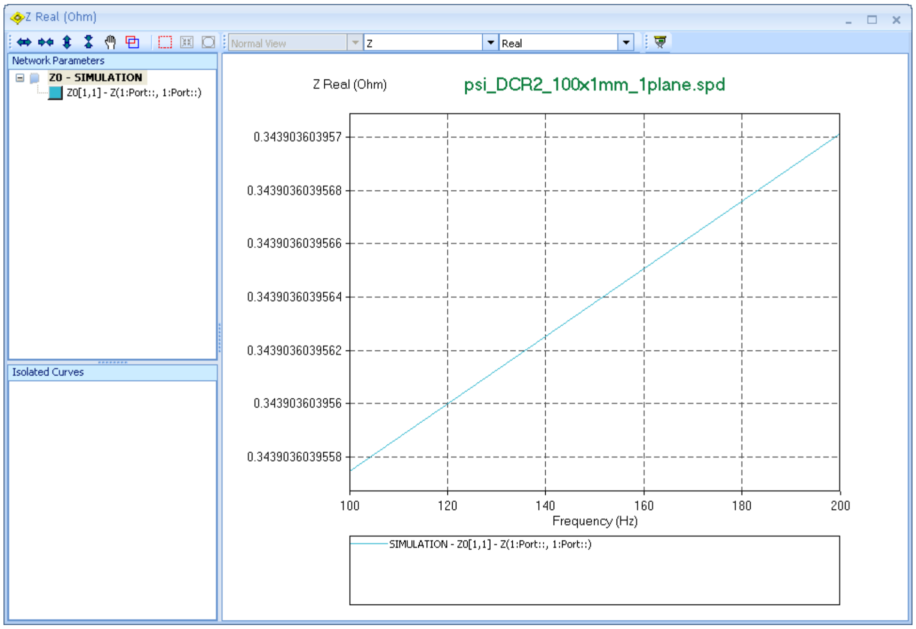


3. Change the following parameters:
   - **Starting Freq.** to 100 Hz
   - **Ending Freq.** to 200 Hz
   - **Sweeping Mode** to **Linear**
   - **Freq. Increment** to 100 Hz
4. Click .

5. Press the **Start Simulation** button  to start the simulation.

   See ***5.1.4 Run a Simulation***.

   The **Z**-real parameter curve appears as shown below.



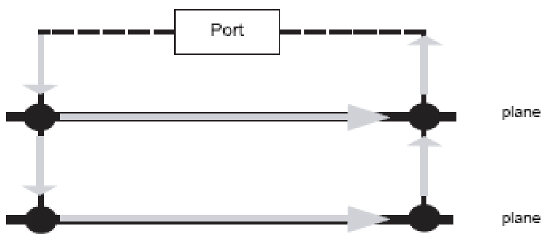
The resistance result is 0.3430 ohms closely matching the analytical results.

6.3 Power Delivery System Resistance (III)

In this example, the structure is similar to the previous one. However, the via on the left end also stops at the Vss plane, making contact with the Vss plane. Because the Vss plane is identical to the Vcc plane, the resistance seen from the port is half that of the previously computed Vcc net in the previous example.

The result can be analytically computed by the following equation:

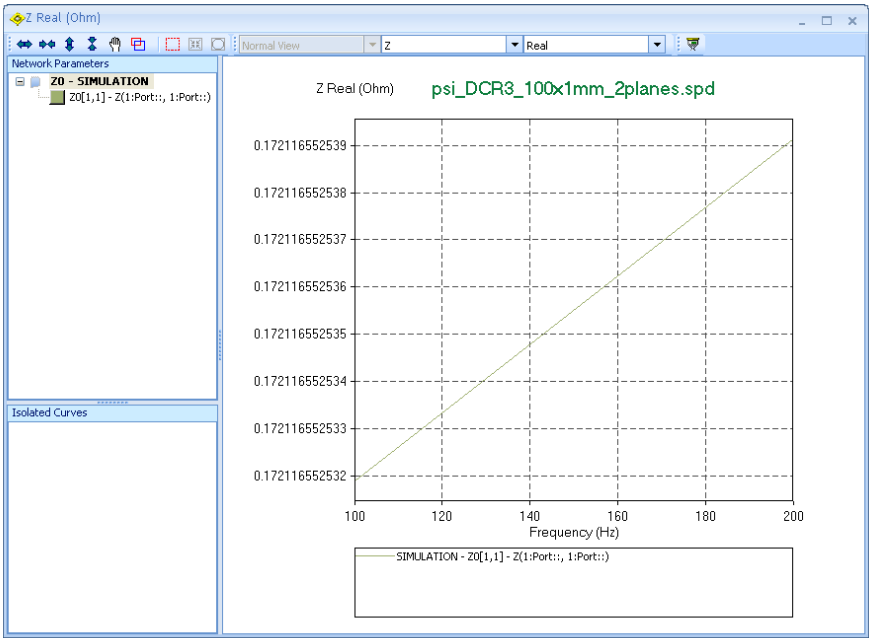
$$R = (R1 \parallel R2)/2 = (R1)/2 = 0.345/2 = 0.172\,Ohms$$



6.3.1 Compute Resistance

Follow the steps in ***5.1 Power Delivery System Resistance (I)***, but use this file psi_DCR3_100x1mm_2planes.spd.

The following figure shows the results and is consistent with the analytical solution: ~0.172 ohms.



6.4 Simulating IR Drop in the Spatial Mode

This section describes how to use **PowerSI** to simulate IR drop in the spatial mode and DC voltage distribution across power ground planes.

In this example, a 3.3 V voltage source (DC) is connected to the power ground planes at the lower right corner of the board. Three pre-specified sink locations are modeled using current sources.

The spatial mode simulation easily identifies the locations where the voltage reduction exceeds a defined threshold.

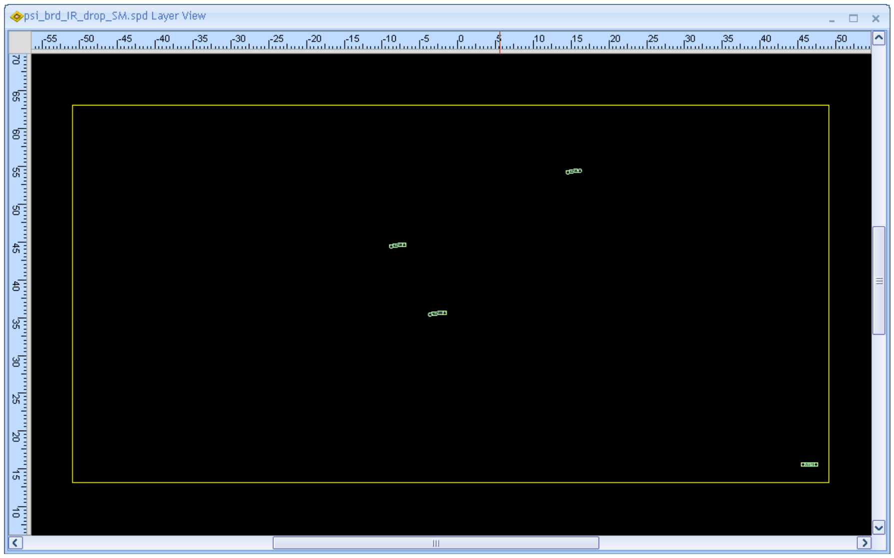
This section has the following topics:

- Start PowerSI and Load a File
- View Circuits
- Set up Voltages
- Set up Spatial Voltage Distributions
- Set up Simulation Frequencies
- Turn on Inter-plane Coupling
- Run a Simulation

6.4.1 Start PowerSI and Load a File

Follow these steps to start **PowerSI** and load a file:

1. Launch **PowerSI.**.
2. Select Mode > Spatial Mode.
3. Select File > Open… and choose psi_brd_IR_drop_SM.spd.

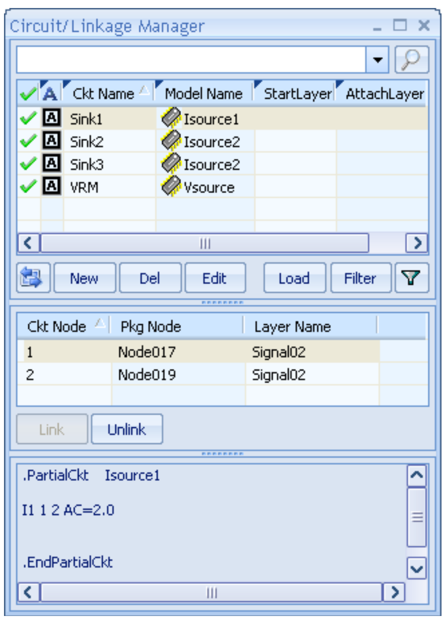


The figure shows three sink locations and one voltage source connected.

6.4.2 View Circuits

Follow these steps to view the information for each of these locations:

1. Select Setup > Circuit/Linkage Manager.



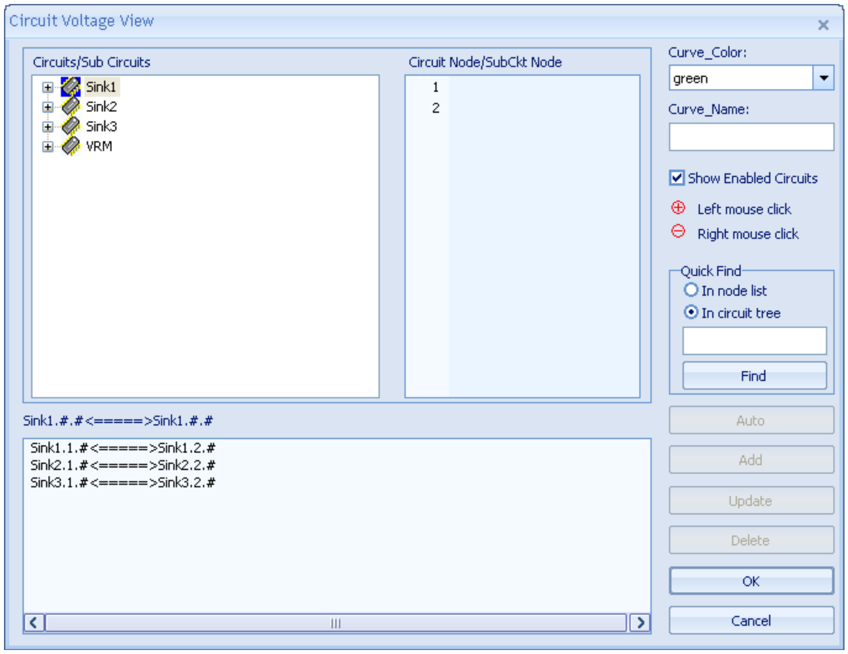
2. Close the Circuit/Linkage Manager window without any changes.

6.4.3 Set up Voltages

Follow these steps to set up the circuit voltage:

1. Select Setup > Simulation View > Ckt Voltage… to specify the voltages of the sink locations.

   For this example, the sink locations have already been selected for viewing voltage.



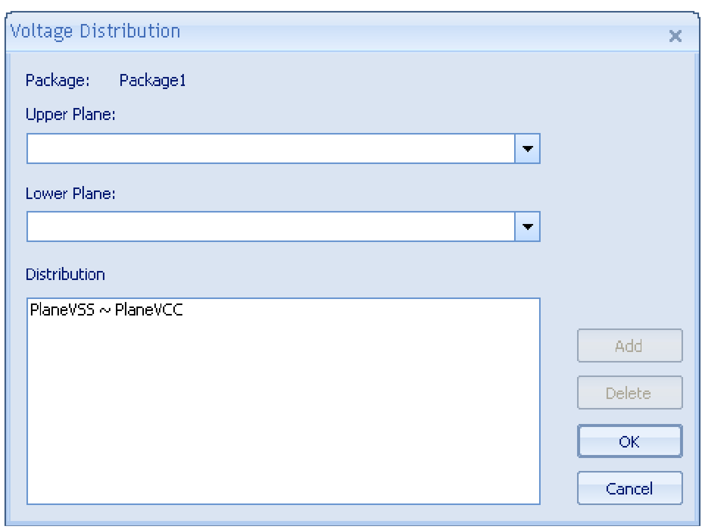
2. Click OK or Cancel to exit.

6.4.4 Set up Spatial Voltage Distributions

Follow these steps to set up or view voltage distributions:

1. Select Setup > Simulation View > 3D Spatial to review the voltage distribution.

   The **Voltage Distribution** dialog box opens.



   For this example, the spatial voltage distributions have already been specified.

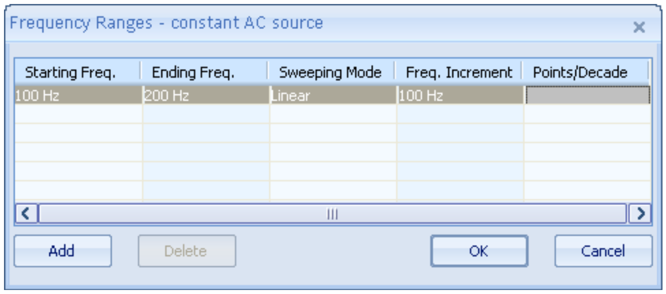
2. Click OK or Cancel to exit.

6.4.5 Set up Simulation Frequencies

Follow these steps to set up simulation frequencies:

1. Select Setup > Simulating Frequencies….

The **Frequency Ranges** dialog box opens.



2. Change the following parameters:
   - **Starting Freq.** to 100 MHz
   - **Ending Freq.** to 200 MHz
   - **Sweeping Mode** to **Linear**
   - **Freq. Increment** to 100 MHz

NOTE!	The **Adaptive** sweeping is not available in the **Spatial Mode**.

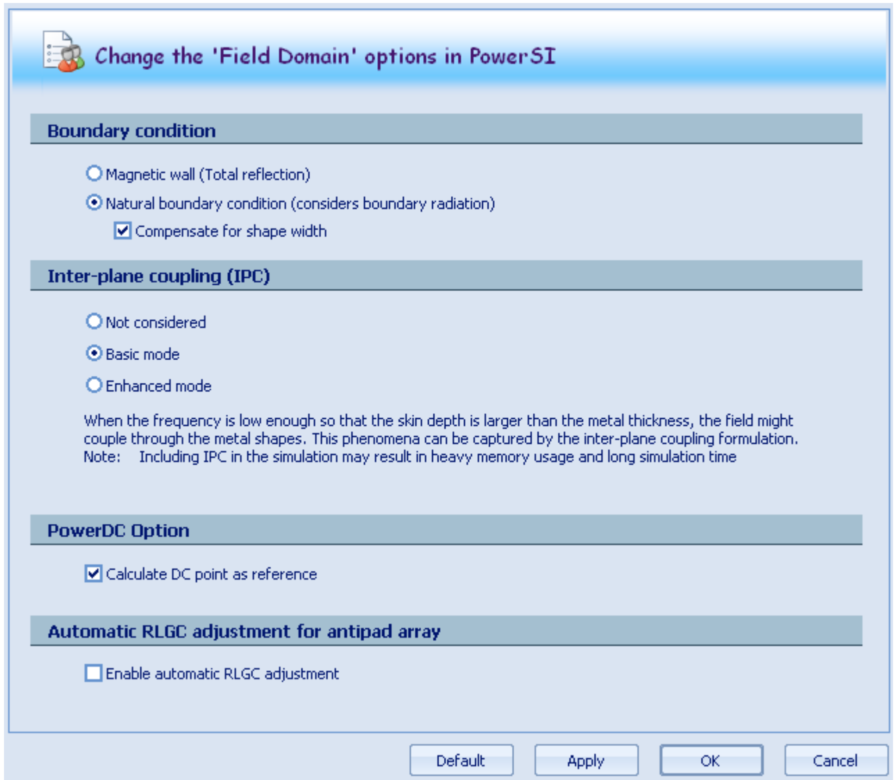
3. Click 

.

6.4.6 Turn on Inter-plane Coupling

Follow these steps to turn on the inter-plane coupling:

1. Select Tools > Options > Edit Options… > Simulation (Advanced) > Field Domain.
2. Select **Basic mode** under **Inter-plane coupling (IPC)**.



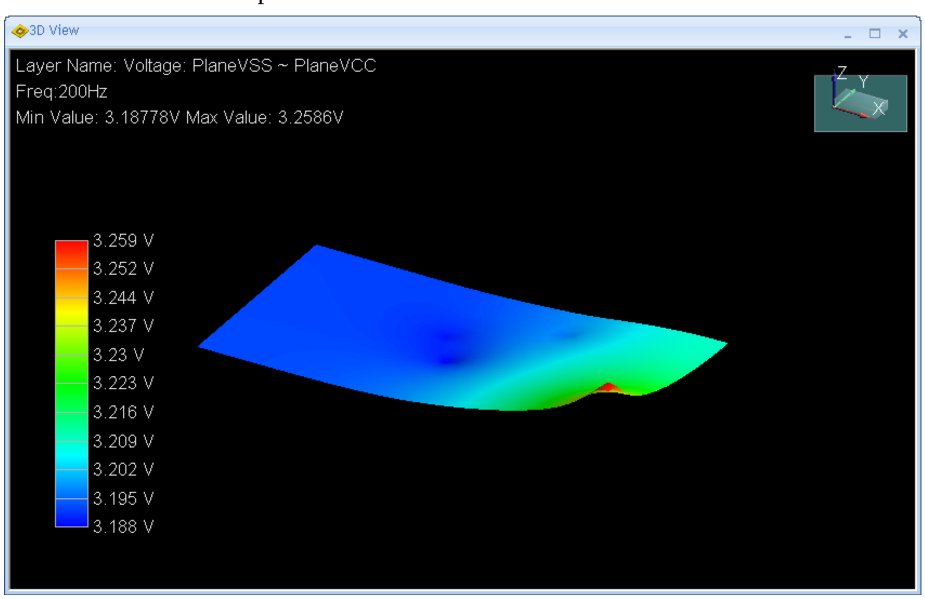
3. Click OK.

6.4.7 Run a Simulation

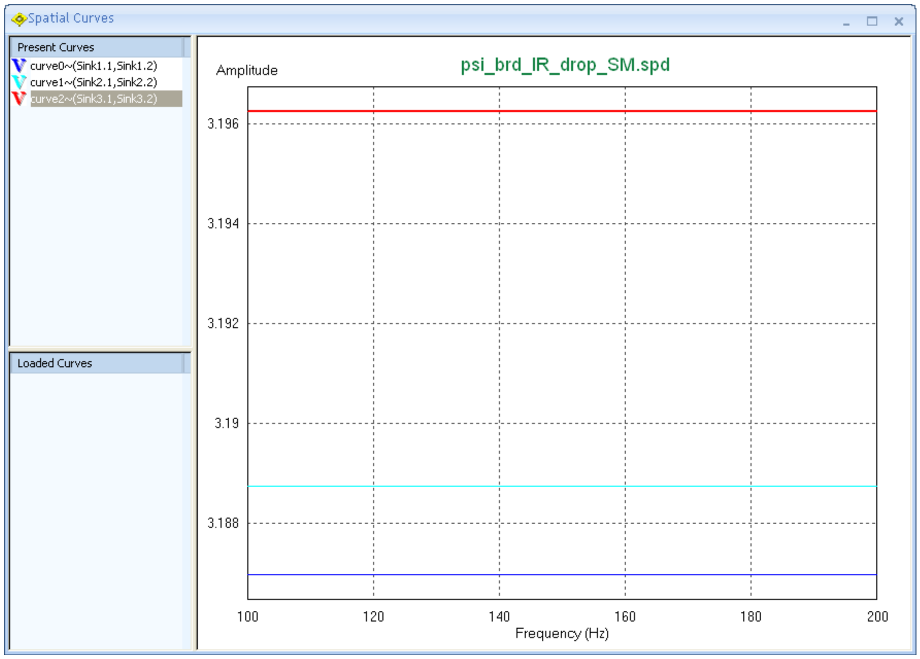
Follow these steps to perform a simulation and view the curves:

1. Click the **Start Simulation** button to start the simulation.

   The **3D View** window opens.



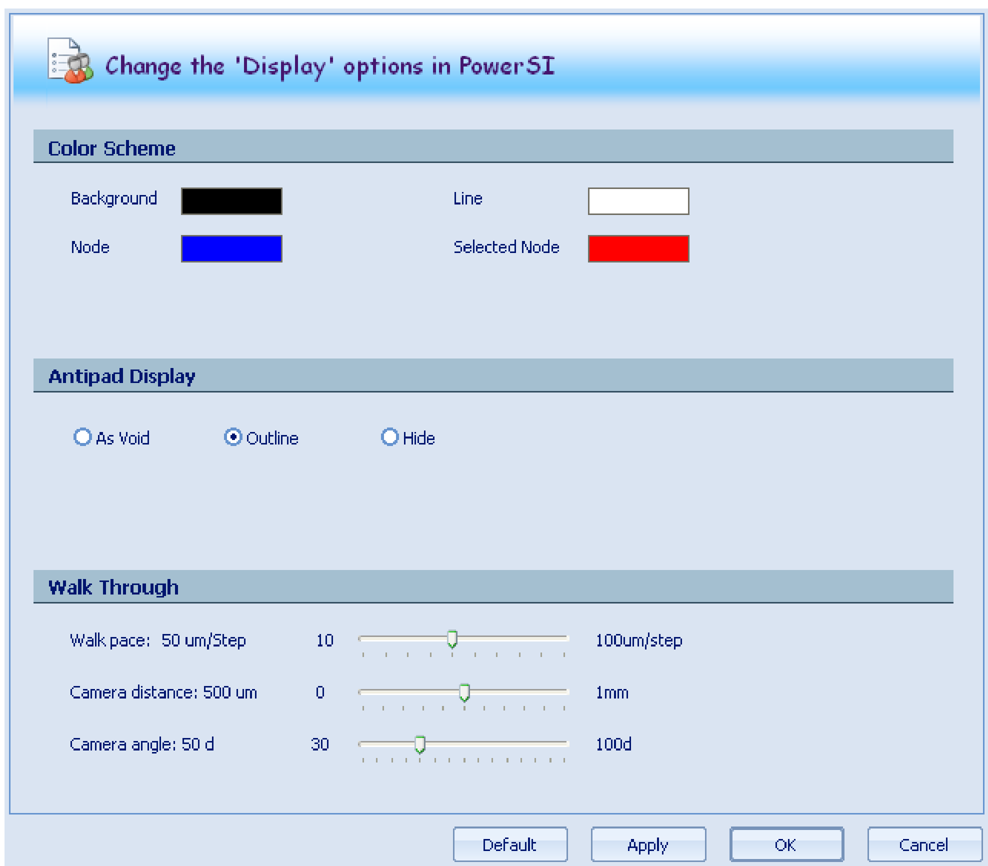
2. Select Windows > Spatial Curves to see the spatial curves.



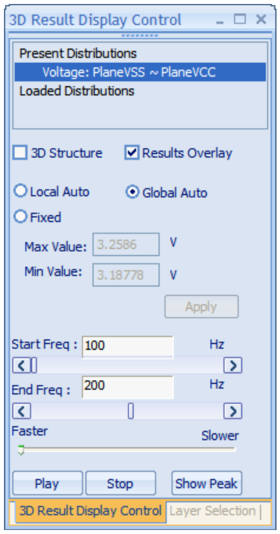
The spatial curve window displays the actual voltages across the sink locations.

3. Select Tools > Options > 3D View Settings… to set up **Display** and **Quality**.

   Set up display options in the following window.



4. Set up parameters for the 3D view in the **3D Result Display Control** dialog box.


3D Result Display Control
Present Distributions
Voltage: PlaneVSS ~ PlaneVCC
Loaded Distributions
3D Structure
Results Overlay
Local Auto
Global Auto
Fixed
Max Value:
3.2586
V
Min Value:
3.18778
V
Apply
Start Freq :
100
Hz
End Freq :
200
Hz
Faster
Slower
Play
Stop
Show Peak
3D Result Display Control
Layer Selection

Index