# cādence<sup>®</sup>

## **PowerSI** Tutorial

Product Version 16.6 January 2014

Document Updated on: November 5, 2013

© 2014 Cadence Design Systems, Inc. All rights reserved.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522. All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

- 1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
- 2. The publication may not be modified in any way.
- Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
- 4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

## **Table of Contents**

1	Prefa	.ce	
	1.1	Overview	5
	1.2	Who Should Read This Tutorial	5
	1.3	Additional Documentation	. 5
	1.4	How to Contact Technical Support	5
2	Intro	duction	
	2.1	Software Overview	
	2.1.1		
	2.1.2		
	2.2	PowerSI Workplace	
	2.3	PowerSI and SPEED2000	
	2.4	PowerSI and Broadband SPICE	
	2.5	PowerSI Workflow	
	2.5.1	Extraction Mode	
	2.5.2		
	2.5.3	Resonance Mode	
3		ct 1 Impedance of Power Delivery System	
5	3.1	About the Examples	
	3.2	Power Ground System Without Decoupling Capacitors	
	3.2.1	Start PowerSI and Load a File	
	3.2.1	View Port Setup	
	3.2.2	Set up Simulation Frequencies	
	3.2.5	Run a Simulation	
	3.2.4		
	3.3	View Simulation Report	
	3.3.1	Power Ground System With Decoupling Capacitors Load a File	
	3.3.2	View Setup	
	3.3.3	Set up Simulation Frequencies	
4	3.3.4	Run a Simulation	
4	•	cct 2 Modeling and Transient Simulation	
	4.1	Overview	
	4.1.1	PCB	
	4.1.2	Modeling and Simulation Procedure	
	4.2	Constructing a Package	
	4.2.1	Construct a Package Model	
	4.2.2	Create Nets	
	4.2.3	Create a Stackup	
	4.2.4	Create Power Ground Shape and Setup Shape Properties	
	4.2.5	Add a Via	
	4.2.6	Add a Trace	
	4.2.7	Set up Trace Properties	
	4.2.8	Set up a Trace on Signal01	
	4.2.9	View Traces in 3D	
	4.3	Specifying Ports	
	4.3.1	Add Ports	
	4.3.2	Set up a Port	
	4.3.3	Set Reference Impedance for the Port	
	4.4	Running a Simulation	
	4.4.1	Set up Simulation Frequencies	
	4.4.2	Run a Simulation	
	4.4.3	Export Results	41

	4.5	Equivalent Circuit Extraction by Broadband SPICE	42
	4.6	Broadband SPICE Model Validation	43
	4.7	Transient Simulation by HSPICE	44
	4.7.1	Prepare an HSPICE Netlist	
	4.7.2	View HSPICE Simulation Results	45
5	Proje	ct 3 Decoupling Capacitors Placement	47
	5.1	Power Ground Noise Without Decoupling Capacitors	
	5.1.1	Start PowerSI and Load a File	47
	5.1.2	View Circuits	48
	5.1.3	View Circuit Voltages	
	5.1.4	View Voltage Distributions	
	5.1.5	Set up Simulation Frequencies	
	5.1.6	Run a Spatial Mode Simulation	
	5.2	Decoupling Capacitor Placement (First Round)	
	5.2.1	Review Spatial Voltage Distributions	
	5.2.2	Determine Decoupling Capacitor Locations	
	5.2.3	Determine Decoupling Capacitor Types and Values	54
	5.3	Decoupling Capacitor Placement (Second Round)	
	5.3.1	Review Spatial Voltage Distributions	
	5.3.2	Determine Decoupling Capacitor Locations	
	5.3.3	Determine Decoupling Capacitor Types and Values	
	5.4	Self and Transfer Impedance of the Power Delivery System	
	5.4.1	Simulate Self and Transfer Impedance Curves	
	5.4.2	Run Simulation with Decoupling Capacitors	
	5.5	Resonance Mode	
	5.5.1	Start PowerSI and Switch to the Resonance Mode	
	5.5.2	Set up Frequency Range	
	5.5.3	Run a Simulation	
6	Proje	ct 4 IR Drop Analysis for Power Delivery Systems	
	6.1	Power Delivery System Resistance (I)	
	6.1.1	Open a Project	
	6.1.2	Set up Simulation Frequencies	
	6.1.3	Turn on Inter-plane Coupling	
	6.1.4	Run a Simulation	
		Power Delivery System Resistance (II)	
	6.2.1	Compute Resistance	
		Power Delivery System Resistance (III)	
	6.3.1	1	
	6.4	Simulating IR Drop in the Spatial Mode	
	6.4.1	Start PowerSI and Load a File	
	6.4.2	View Circuits	
	6.4.3	Set up Voltages	
	6.4.4	Set up Spatial Voltage Distributions	
	6.4.5	Set up Simulation Frequencies	
	6.4.6	Turn on Inter-plane Coupling	
	6.4.7	Run a Simulation	84

## 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.

Chapter	Page
Introduction	6
Project 1 Impedance of Power Delivery System	10
Project 2 Modeling and Transient Simulation	21
Project 3 Decoupling Capacitors Placement	47
Project 4 IR Drop Analysis for Power Delivery Systems	74

#### 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.

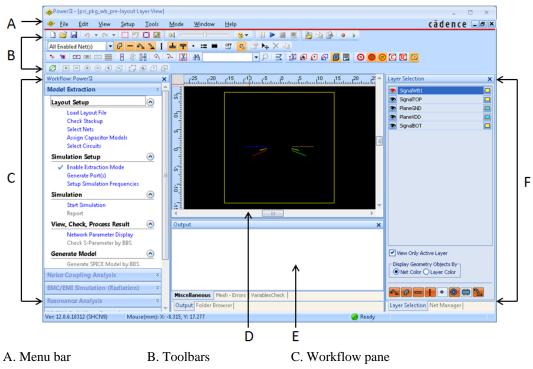
#### 2.1.2 Applications

**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.



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.

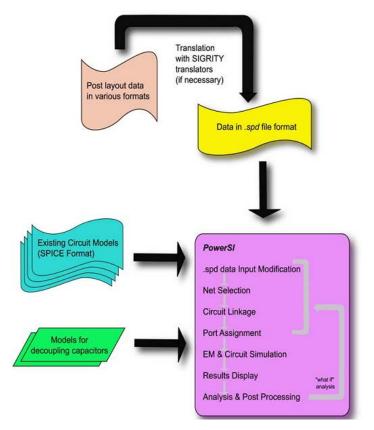
Many translators are developed to convert data files from other vendors to Sigrity's data file .spd. So you may use any data files generated under other tool environments to do the post-layout analysis of packages and boards.

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:



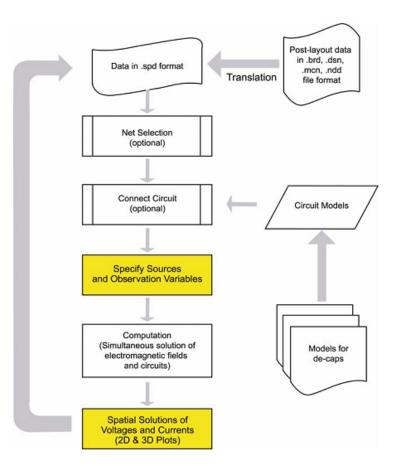
#### 2.5.2 Spatial Mode

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:

#### **PowerSI** Tutorial



#### 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

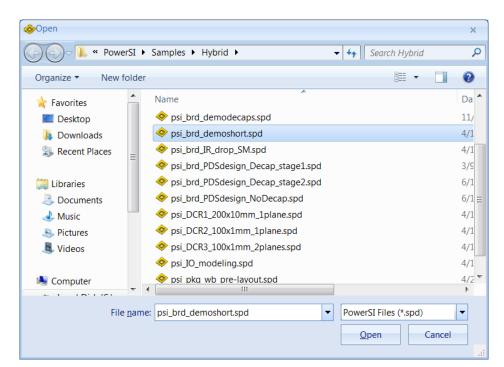
Follow these steps to start **PowerSI** and load a file:

- 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.

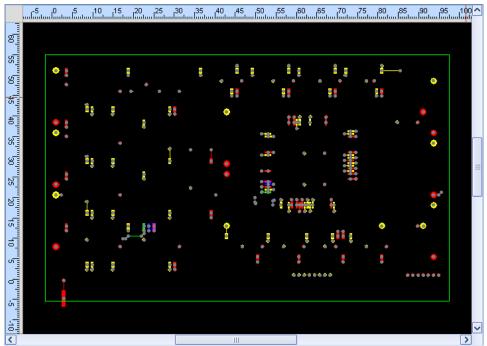


4. Double-click psi\_brd\_demoshort.spd.

#### 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  $\Omega$  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....

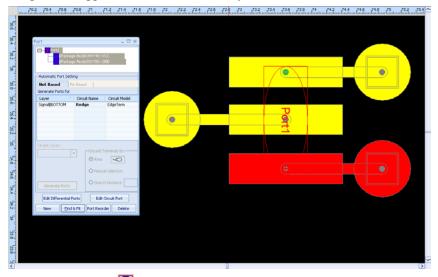
The Port setup dialog box opens.

Port _ 🗆 🗙										
Port1     •••      ••••     ••••     ••••     ••••     ••••     ••••     ••••     ••••     ••••     ••••     ••••										
Automatic Port Setting										
Net Based Pin Based										
Generate Ports for										
	Circuit Name	Circuit Model								
Signal\$BOTTOM	Redge	EdgeTerm								
Ground Terminals By:										
Generate Ports	Generate Ports O Search Distance									
Edit Differential	Ports Ed	lit Circuit Port								
New	d & Fit Port Re	order Delete								

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.

Starting Freq.	Ending Freq.	Sweeping Mode	Freq. Increment	Points/Decade
10 MHz	2 GHz	Adaptive		[

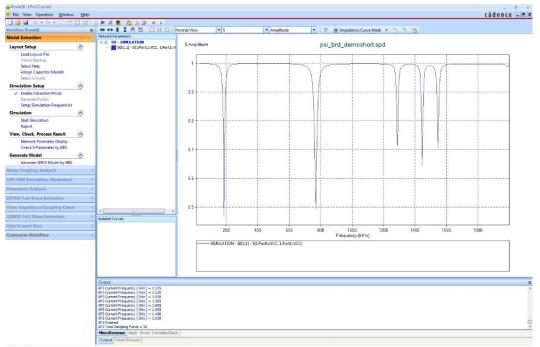
- 2. Change the following parameters in this window:
  - Starting Freq. to 10 MHz (or 0.01 GHz)
  - Ending Freq. to 2.0 GHz
  - Sweeping Mode to Adaptive from the drop-down menu.
- 3. Click **OK** to finish the setup.

#### 3.2.4 Run a Simulation

Follow these steps to perform a simulation:

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

An S-parameter displays first as follows.



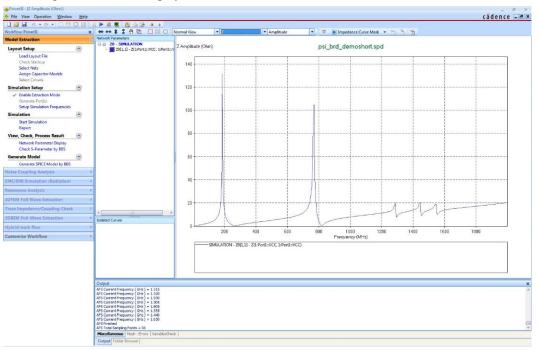
The **Output** window shows the process status during the simulation.

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

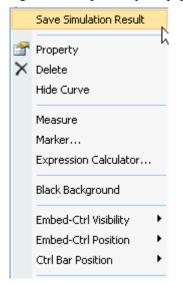
2. Click and choose the Z (impedance) parameter from the drop-down menu.



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.

Save Curves							×
Network par	ameter file format:	Touchsto	one format	•	🗹 AFS L	ossless Compr	essior
Save to	C:\SigritySamples	\SpeedXP	12.0\Pow	erSI\Sampl	es\Hybrid\p	osi_brd_dem	
─Data Type ⊙ RI		4	Save the	e following r	network par	rameter(s)	
Export pa	assivity enforced S-	parameters		Fouchstone	2.0	Mixed mode	_
MCP Out	tput		Fred	quency Unit	: Hz	-	
	ОК			Cano	:el		

- 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.

	Double-click a curve to show the <b>Curve Pattern Property</b> box.
Tip!	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 **X** to return to the **Layer View** window.
- 2. Select File > Report.

	New	Ctrl+N
2	Open	Ctrl+O
	Save	Ctrl+S
	Save As	
	Save Workspace	
	Close	
	Report	N
		N

The Simulation Report Setting window opens.

General Information Report template (*.htm): 1.1\library\template\PowerSI\PowerSI_Report_Template_Default.htm Notes:									
Optional Plo Source	Method.	Name. DefaultCurve		2	Port1. Port1::VCC		Port2. Port1::VCC		

NOTE	You can select Tools > Options > Edit Options > Simulation (Basic) >
NOTE!	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 Simulation Report Date: November 7, 2012 1 General information	owerSI_Rep	ort_Template_Defa	ult.htm *									
Date: November 7, 2012 <b>1 General information 1.1 Spd file name and location</b> PowerSI version: 12.0.5.10152 <b>File names and locations</b> • Layout spd file • 0.5/SignitySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort.spd         • Simulation result file • 0.5/SignitySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247_pascive.bnp         • Simulation result file • 0.5/SignitySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247_Pascive.bnp         • Dysimptisamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247_Pascive.bnp         • Dysimptisamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_100712_180247_pascive.bnp         • Dysimptisamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/Psi_brd_demosho												cādence°
1 General information         1.1 Spd file name and location         PowerSI version: 12.0.5.1052         File names and location         0.2/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort.spd         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_100712_180247.bnp         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         0.3/SigntySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         1.4/SigntSTOP       0.00444       5399+007       1       0       0.1       0       0         1       SigntSTOP       0.00444       5399+007       1       0       0.1       0       <						Powe	erSI Sir	nulatior	n Repo	rt		
1.1 Spd file name and location         PowerSI version: 12.0.5.10152         File names and locations:         • Layout spd file         • DySigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort.spd         • Simulation result file         • DySigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.bnp         • DySigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         • DysigritySamp	Date: No	vember 7, 2012										
Powersion : 12.0.5.10152 File names and locations: • Layout spd file • D-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_passive.bnp • D/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247_Passive.bnp • D/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_100712_180247_Passive.bnp • D/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_100712_180247_Passive.bnp • D/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_100712_180247_Passive.bnp • D/SigritySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_100712_180247_Passive.bnp	1 Ge	neral inf	ormatio	on								
File names and locations:         • Layout spd file         • 0.2/SigntySamples/SpeedXP 12.0/PowerSl/Samples/Hybrid/psi_brd_demoshort.spd         • Simulation result file         • 0.2/SigntySamples/SpeedXP 12.0/PowerSl/Samples/Hybrid/psi_brd_demoshort_110712_180247.bng         • Dr/SigntySamples/SpeedXP 12.0/PowerSl/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp         • Dr/SigntySamples/SpeedXP 12.0/PowerSl/Samples/Hybrid/psi_brd_demoshort_110712_180247_Passive.bnp         • Layer %       Layer Name         • SigntSoco       0.00444         • SigntSoco       0.1         • MediumS4       0.2032         • MediumS4       0.2032	1.1 Spd file name and location											
• Layout spd file <ul> <li>Dr/StgrttySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort.spd</li> </ul> • Simulation result file <ul> <li>Dr/StgrttySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp</li> <li>Dr/StgrttySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.Passive.bnp</li> </ul> <b>Layov</b> *       Layov Name       Thickness(mm)       Maternal <li>Conductivity(Sim)       Permittivity       LowsTargets       StapsName       TraceWalth(nm)       Roughness(nm)         1       SignalSTOP       0.03044       5.959e+007       1       0       0.1       0         2       SignalStOP       0.03044       5.959e+007       4.5       0</li>	PowerSI v	version: 12.0.5.10	0152									
<ul> <li>Simulation result file</li> <li>Or/StgrittySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_spd</li> <li>Or/StgrittySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.pns</li> <li>D/StgrittySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.pns</li> <li>D/StgrittySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.pns</li> <li>D/StgrittySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.pns</li> <li>D/StgrittySamples/SpeedXP 12.0/PowerSI/Samples/Hybrid/psi_brd_demoshort_110712_180247.pns</li> <li>SignitSOP</li> <li>0.00448</li> <li>SpS9e+007</li> <li>I</li> <li>I</li> <li>StgritSOP</li> <li>0.00448</li> <li>SpS9e+007</li> <li>I</li> <li>I<!--</td--><td>File name</td><td>s and locations:</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></li></ul>	File name	s and locations:										
I         SignalSTOP         0.03448         5.959±007         I         0         0.1         0           MediumS2         0.2032         4.5         0         -	• Sim	o D:/SigritySamp ulation result file o D:/SigritySamp o D:/SigritySamp	les/SpeedXP 12 les/SpeedXP 12	2.0/Power	SI/Samples/Hybrid	l/psi_brd_de	moshort_110					
MediumS2         0.2032         0         4.5         0         0         0         0           2         SignalSOND         0.00448         5.559e+007         4.5         0         SignalSOND         0.10         0           MediumS4         0.2032         0         4.5         0.0         SignalSOND         0.10         0	Layer #	Layer Name	Thickness(mm)	Material	Conductivity(S/m)	Permittivity	LossTangent	Fill-in Dielectric	ShapeName	TraceWidth(mm)	Roughness(mm)	
2         SemiSOND         0.0048         5.959+007         4.5         0         ShapeGOND         0.1         0           Medium54         0.2032         4.5         0         1         0         1         0	1	SignalSTOP	0.03048		5.959e+007	1	0			0.1	0	1
Medium54 0.2032 4.5 0												1
	2				5.959e+007				Shape\$GND	0.1	0	1
3 SignalSIS3 0.03048 5.959e+007 4.5 0 0.1 0												
Medium56 0.2012 4.5 0	3	-			5.959e+007		-			0.1	0	4

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<br/>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:

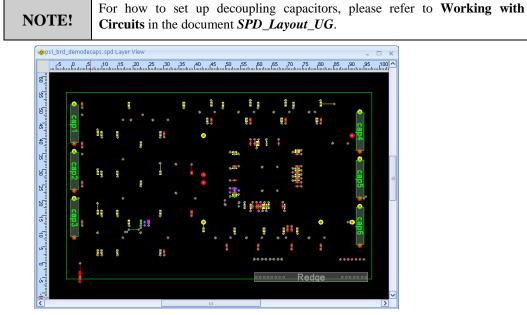
1. Select Setup > Circuit/Linkage Manager.

Simulating Frequencies Port Macro	
Cutting Boundary	۲
Circuit/Linkage Manager	
Net Manager	Ś
Sweeping Manager	
Server Setup	

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

			- 9
🗸 🗛 Ckt Name	4 Model Name	StartLayer	AttachLayer
V 🗛 cap1	Decap		
🗸 🖪 cap2	Decap		
🗸 🖪 cap3	Decap		
🗸 🖪 cap4	Decap		
🗸 🖪 cap5	Decap		
🗸 🖪 cap6	Decap		
🗸 🖪 Redge	EdgeT		
<	111		>
New	Del Edit	Load	Filter 🕎
Ckt Node 🔶 Pk	g Node	Layer Nam	e
0 No	de2375!!2::GND	Signal\$TOP	•
1 No	de2378!!1::VCC	Signal\$TOP	
Link Unlin	k		
.PartialCkt Deca + ExtNode = 0			^
C_cap 2 0 0.47u L_esl 2 3 1p R_esr 3 1 0.01			=
.EndPartialCkt			~
<	111		>

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



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.

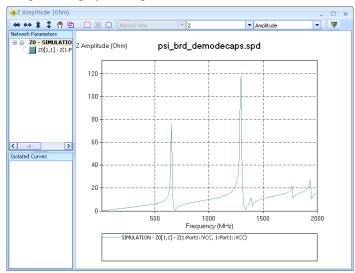
#### 3.3.3 Set up Simulation Frequencies

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 **b** 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.

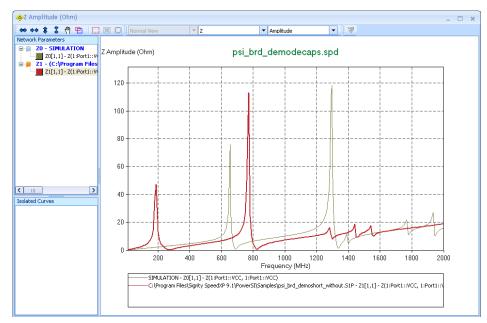
	attern Property - SIMULATION - ZO[1,1] - 💦 🗙 t1::VCC, 1:Port1::VCC)
Style:	Pattern: None Vidth: 1
Mark:	None  Mark Size: 1  Mark Interval: 1
Sample	:
	OK Cancel

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.

Save Simulation Result
Load
Unload All Networks

- 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

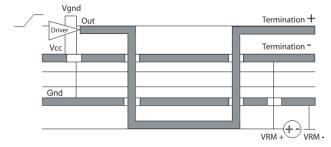
#### 4.1.1 PCB

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.

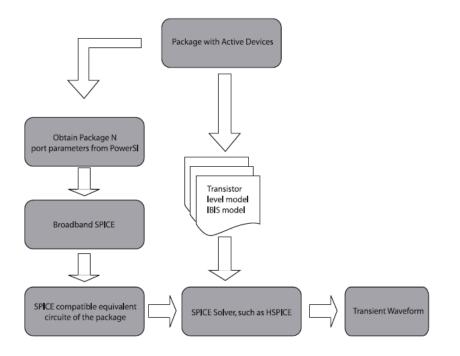
(See 3.5 Equivalent Circuit Extraction by Broadband SPICE.)

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:

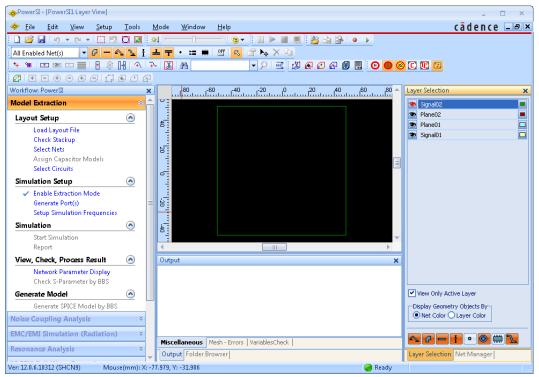
- Construct a Package Model
- Create Nets
- Create a Stackup
- Create Power Ground Shape and Setup Shape Properties
- Add a Via
- Add a Trace
- Set up Trace Properties
- Set up a Trace on Signal01
- View Traces in 3D

#### 4.2.1 Construct a Package Model

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

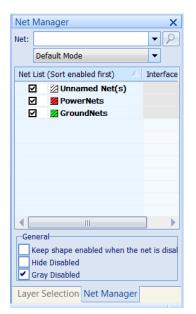
- 1. Launch PowerSI.
- 2. Select File > New.

A default four-layer stackup opens.

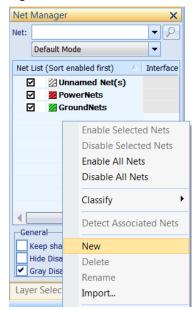


#### 4.2.2 Create Nets

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.



 Repeat the above step to create one more new net. Two new nets are created.

Net Manager	×
Net:	• 2
Default Mode	-
Net List (Sort enabled first) $ riangleq$ I	nterface
Unnamed Net(s)	
PowerNets	
GroundNets	
NewEntity	
NewEntity(1)	
	•
General	
Keep shape enabled when the ne	t is disal
Hide Disabled	
Gray Disabled	
Layer Selection Net Manager	

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

Net Manager 🗙
Net:
Default Mode
Net List (Sort enabled first)
🗹 🛛 Unnamed Net(s)
PowerNets
GroundNets
General
Keep shape enabled when the net is disal
Hide Disabled
Gray Disabled
Layer Selection Net Manager

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

Net:		Show Volt & I	•
Net first	List (Sort enabled $ riangle$	Paring P/G Net	\
E	Unnamed Net(s)		
Ū	PowerNets		
Ē	GroundNets		
Ū	GND		
Ē	VCC		
. ◀ 📘	111		►
	ieral Geep shape enabled whe Gray Disabled	n the net is disablec Hide Disabled	

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

Net Manager		- 🗆 ×
Net:	▼ 🔎 Show	v Volt & I 🔻
Net List (Sort enabled first)	△ Paring P/G Net	
🗹 🛛 🖾 Unnamed Ne	t(s)	
🗉 🗹 🛛 🌌 PowerNets		
GroundNets		
🗹 🗾 GND		
General		
Keep shape enabled	when the net is dis	abled
Gray Disabled	Hide	Disabled
Layer Selection Net N	lanager	

#### 4.2.3 Create a Stackup

Follow these steps to create a six-layer stackup:

1. Select Edit > Stack Up....

Edit	View	Setup	Tools			
St	ack Up					
М	Material File					
Pa	ad Stack	Library				

The Stackup dialog box opens.

ric Permittivity [1] 4 [4] [4] 4 [4] [4] [4] [4] [4] [4] [4	<pre>v Loss Tange [0] 0 [0] 0 [0] 0 [0] 0 [0]</pre>
4 [4] 4	0 [0] 0
<mark>[4]</mark> 4	[0] 0
4	0
	-
[4]	[0]
4	0
[1]	[0]
	•
w Material ver Special Void	Import Filter

- 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**.

Me Z Pla	ane02 edium02 ane01		3.5560e-002 1.0000e-001 ert Above		5.800000e+007	
Z Pla						
	ane01	Ins	ort About			
			ert ADOVE		Signal Layer	
	edium	Ins	ert Under 🕨	h	Medium Layer	
Siç	ignal01	Del	ete		Plane Layer	
	_	-		-		

d. Repeat Step b and select Insert Under > Signal Layer to add a signal layer belowMedium02 while a medium layer Medium05 is automatically added above Plane01.

Layer #	Color	Layer Icon	Layer Name	Thickness(um)	Material	Conductivity(S	Fill-in Dielectric	Permittivity	Loss Tange
1			Signal02	35.56		5.8e+007		[1]	[0]
			Medium03	200				4	0
2		777	Plane02	35.56		5.8e+007		[4]	[0]
			Medium04	500				4	0
3			Signal03	35.56		5.8e+007		[4]	[0]
			Medium02	100				4	0
4			Signal04	35.56		5.8e+007	,	[4]	[0]
			Medium05	500				4	0
5		777	Plane01	35.56		5.8e+007		[4]	[0]
			Medium01	200				4	0
6			Signal01	35.56		5.8e+007	,	[1]	[0]
4									
			111						
Total Thick	mess: 1.7	134e+003 um				Enforce caus	ality View M	aterial	Import
							Auto Set Layer S		Filter

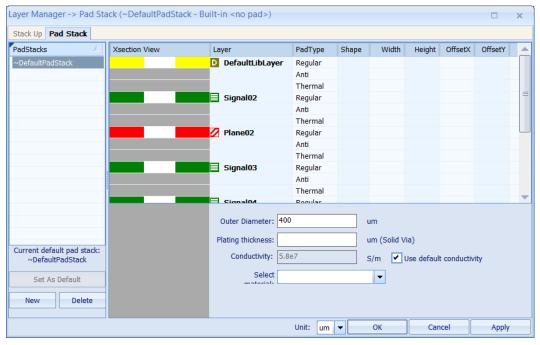
- 3. Change the **Thickness** unit to **um** from the **Unit** drop-down menu.
- 4. Double-click the **Thickness** cells to **c**hange the values:
  - Click the Medium01 and Medium03 Thickness cells and input 200 um.
  - Click the Medium02, Medium04 and Medium05 Thickness cells and input 300 um.

Stack U										
Layer #	Color	Layer Icon	Layer Name	Thickness	Material	Conductivity	Fill-in Dielectric	Permittivity	Loss Tangent	Sha
1			Signal02	35.56		5.8e+007		[1]	0	
			Medium03	200		0		4	0	
2			Plane02	35.56		5.8e+007		[4]	0	Shap
			Medium04	300		0		4	0	
3			Signal03	35.56		5.8e+007		[4]	0	
			Medium02	300		0		4	0	
4			Signal04	35.56		5.8e+007		[4]	0	
			Medium05	300		0		4	0	
5			Plane01	35.56		5.8e+007		[4]	0	Shap
			Medium01	200		0		4	0	
6			Signal01	35.56		5.8e+007		[1]	0	
◀			111							
Total Th	ickness:	1.5134e+003 I	um				Enforce causality	View Mate	rial Impo	ort
							· · ·	Set Layer Spec		

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.

Layer Manager -> Pad Sta Stack Up Pad Stack	ack (NewPadStack)								×
PadStacks	Xsection View	Layer	PadType	Shape	Width	Height	OffsetX	OffsetY	
~DefaultPadStack		D DefaultLibLayer	Regular						
padstack_r004			Anti						
			Thermal						
		Signal02	Regular						=
			Anti						
			Thermal						
		💋 Plane02	Regular						
			Anti						
			Thermal						
		Signal03	Regular						
			Anti						
			Thermal						
		Cional04	Popular						
		Outer Diameter: 10	0		um				
		Plating thickness:			um (Solid V	ia)			
Current default pad stack: ~DefaultPadStack		Conductivity: 5.8	e7		S/m 🔽	Jse defaul	t conductiv	ity	
Set As Default		Select			-				
New Delete									
			Unit: um	-	ОК	Car	icel	Apply	

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

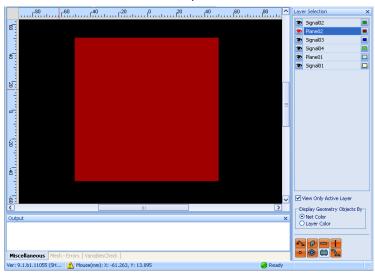


8. Click **OK** to save your changes and exit the **Layer Manager** window.

#### 4.2.4 Create Power Ground Shape and Setup Shape Properties

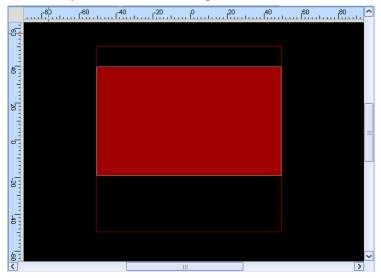
Follow these steps to create a power ground shape:

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

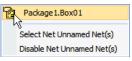


- 3. Choose the **Select** button select toolbar, and click any location on the shape to select the shape.
- 4. Press the **Delete** key to delete the shape.
- 5. Select View > Toolbars > Shape to bring up the Shape toolbar.

6. Select the **Box**+ button in 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.

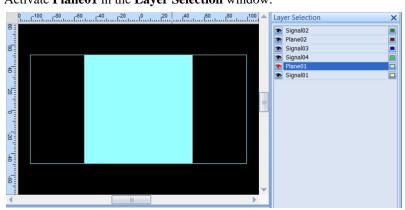
Box Properties ×
Name : Box01
Net : Unnamed Net(s)  Vuit : mm
Lower left corner coordinates : -6.230769e+001 -2.307692e+001
Width:         1.260000e+002         Height:         4.846154e+001
OK Cancel

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

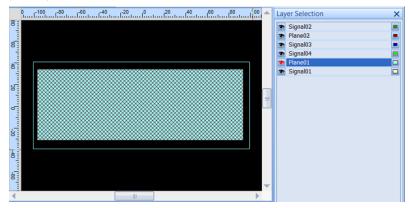
- Width: 2.00000e + 002
- **Height**: 8.00000e + 001
- 10. Select **VCC** from the **Net** drop-down list and assign it to Box01.

Box Prop	perties		×
Name :	Box01		
Net :	VCC	▼ Unit :	mm
Lower l	eft corner coordinates	-1.000000e+002	-4.000000e+001
Width :	2.000000e+002	Height :	8.000000e+001
	ОК	Cancel	]

- 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.

Box Prop	perties x
Name :	Box02
Net :	GND Unit : mm V
Lower l	
Width :	2.000000e+002 Height : 8.000000e+001
	OK Cancel

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.

#### PowerSI Tutorial

The Status Bar shows the coordinates of the cursor timely.

### TIP!

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.

	Cancel		
	Done		
	Type Coordinate		
ю 19	S Zoom Back		
	Zoom Fit		
	Zoom In		
	Zoom Out		
	Property •		
Coo	ordinate Input		×
	Relative Coordinates		
-			
x	c: 0 y: 0	Units: mm	ı 🔽
x	(; <mark>0 y: 0</mark>	Units: mm	•

4. Select Signal02 and then Signal01 in the Via Editing dialog box.A via appears to connect the Signal02 and Signal01 layers.

Via Editing	×
	Signal02 Medium03 Plane02 Medium04 Signal03 Medium02 Signal04 Medium05 Plane01 Medium01 Signal01
Add Dele Hints Click Add for new via	
Info The current assigne ~defaultpadstack	d default pad stack is:
ОК	Cancel

5. Click **OK**.

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

<b>NOTE!</b> 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.
---

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

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		
Via06	75, -1	Signal02, Plane01	Specify the ports connecting the receiver.	
Via07	-0.5, -39	Signal02, Plane02	Specify the ports connecting the VRM module.	
Via08	0.5, -39	Signal02, Plane01	Use the zoom button to pinpoint the coordinates.	

	Use <b>Snap to Grid</b> to set a 0.5 or -0.5 coordinates: Select Tools > Options > Edit Options > Layout > Grid and Unit to check the <b>Snap objects to grid</b> 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.

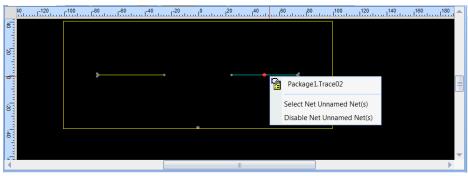
aye	er Selection 💦 💶 🗙
۲	Signal02
9	Plane02
9	Signal03
9	Signal03
9	Plane01  Signal01
9	Signal01
	/iew Only Active Layer
	splay Geometry Objects By Net Color O Layer Color

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.

Name :	Trace02	Starting Node: Node018
Net:	Unnamed Net(s)	Ending Node: Node03
	Unit : mm	Length: 4.91392e+01 (mm)
		The uniformity of the trace is not checked. The trace properties shown here are from its e
Thickn	ess: 3.55600e-002	To ensure uniformity of the trace environment
Width:	2.00000e-001	Delay Time: 2.73112e-01 (ns)
Ending	Width 2.00000e-001	Characteristic Impedance: 68.87 (Ohm)
Condu	ctivity: 5.800000e+007	Effective Dielectric Constant: 2.8
		< <u> </u>
Calc	ulate Create Circuit N	1odel OK Cancel

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**:

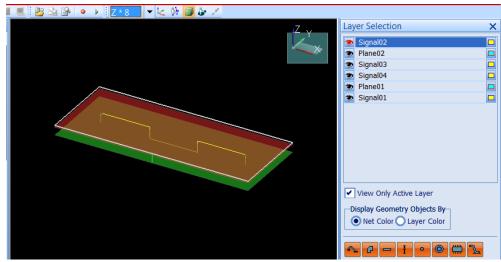
- Click Signal01 in the Layer Selection dialog box. The Signal01 icon becomes red.
- Add Trace03 from (-25, 0) to (25, 0) on the Signal01 layer.
   Via01 and Via02 are now connected by Trace03.
- 3. Change the **Trace03** width to 0.2 mm.

#### 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.

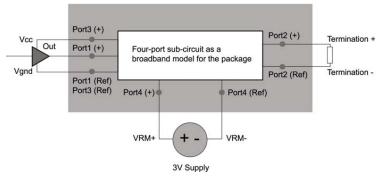
# 4.3 Specifying Ports

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.
- Click the New Port button in on the Port toolbar.
   The New Ports dialog box opens.

New Ports	×
Port Name :	Port
Start number :	1
End number :	4
ОК	Cancel

- 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.

Port			_ 🗆 X
Port1 Port2 Port3 Port4			
Automatic Port Set	ting		
Net Based	Pin Based	d	
Generate Ports for			
Layer 🛆	Circuit	Name	Circuit Model
Target Layer:			
	-	Ground Termi	
		⊙ Area	~©
		O Manual Se	lection
Generate Ports		O Search Dis	tance um
Edit Differentia	al Ports	Edit C	Circuit Port
New Ei	nd & Fit	Port Reord	ler Delete

- 2. Select Port1.
- 3. Find and right-click the left end of **Trace01** (-74, 0).

A pop-up menu appears.

7	Package1.Node03
-0-	Link
⊕ ⊙	Hook Hook
	Select Net Unnamed Net(s) Disable Net Unnamed Net(s)

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.

Port _ 🗆 🗙
Image: Part i         Image: Package 1.Node017         Image: Package 1.Node018         Image: Package 1.Node05::VCC         Image: Package 1.Node013::VCC         Image: Package 1.Node013::VCC         Image: Package 1.Node015::GND
Automatic Port Setting
Net Based Pin Based
Generate Ports for
Layer 🛆 Circuit Name Circuit Model
Target Layer: Ground Terminals By: O Manual Selection O Search Distance um
Generate Ports
Edit Differential Ports Edit Circuit Port
New Find & Fit Port Reorder Delete

#### 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

Port Refere	nce Impedance	
Power Nets	1	Ohm
Signal Nets	50	Ohm
Note: non-un	iform port impedance is	s not support by some third party tools

By default, Power Nets is set to 10hm and Signal Nets is set to 500hm.

# 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.

Starting Freq.	Ending Freq.	Sweeping Mode	Freq. Increment	Points/Decade
) Hz	2 GHz	Adaptive		
<		111		

- 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.

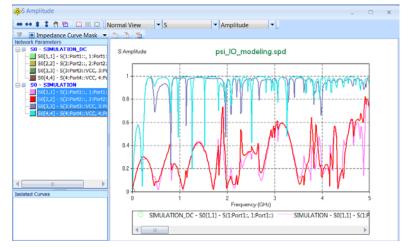
Starting Freq.	Ending Freq.	Sweeping Mode	Freq. Increment	Points/Decade
MHz	5 GHz	Linear	10 MHz	(
		111		

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

### 4.4.2 Run a Simulation

1. Press the **Start Simulation** button  $\triangleright$  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.

Save Curves ×
Network parameter file format: Touchstone format  AFS Lossless Compression
Save to D:\SigritySamples\SpeedXP 12.0\PowerSI\Samples\Hybrid\psi_I0_mode
● RI       ● DB       ● MA       Save the following network parameter(s)         ● RI       ● DB       ● MA       Image: Save the following network parameter(s)
Export passivity enforced S-parameters Touchstone 2.0 Mixed mode
MCP Output Frequency Unit: Hz -
OK Cancel

- 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.

Save Curves ×
Network parameter file format: Touchstone format
Save to D:\SigritySamples\SpeedXP 12.0\PowerSI\Samples\Hybrid\Model_10.S4
Data Type       Save the following network parameter(s)         ● RI       DB       MA         ✓ S       Z       Y         SDIFF
Export passivity enforced S-parameters Touchstone 2.0 Mixed mode
MCP Output Frequency Unit: Hz 💌
OK Cancel

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:

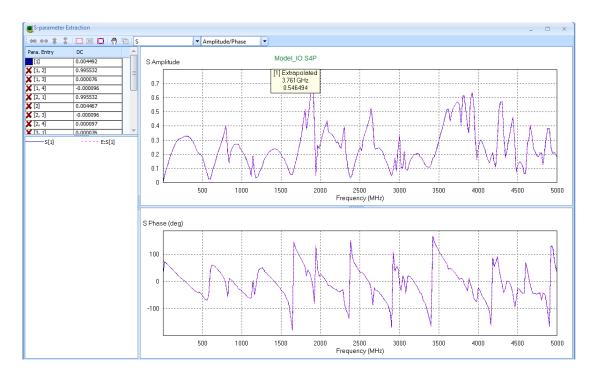
- GNDNET specifies the net to which all the ground pins in the model should be connected
- NETSLIST is a comma separated list of nets which should be deleted from the schematic when the model gets assigned
- 7. Select File > Exit to close the project.

### 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.

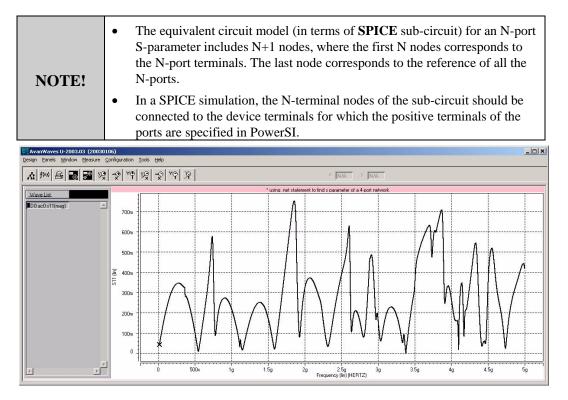
The .net function is used in **HSPICE**. Because the .net function can only simultaneously obtain two port parameters, you need to compute the partial entries of the port S-parameters of an N-port, by connecting the reference impedance (50 ohms) to the rest of the ports.

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 pl 0 AC 1
X1 pl 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.



# 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
```

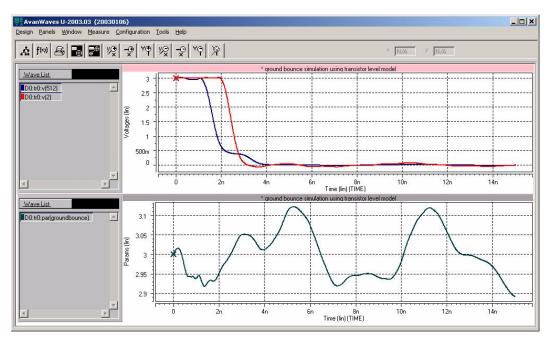
\* Include the HSPICE level-3 model for the driver

```
.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 le-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

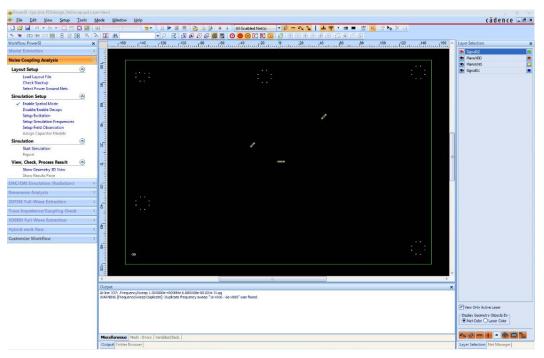
This section has the following topics:

- 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.

Ckt Node / Pkg Node Layer Name Node01 Signal02 Node03 Signal02 Link Unlink PartialCkt Source1 11 2 AC=0.15 EndPartialCkt	chLayer
1         Node01         Signal02           2         Node03         Signal02           Link         Unlink	er 🖓
Link         Unlink           .PartialCkt Source1         11 1 2 A C= 0.15	
.PartialCkt Source1 II 1 2 AC=0.15	

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.

ircuit Voltage View		×
Circuits/Sub Circuits	Circuit Node/SubCkt Node	Curve_Color:
6: 20 Sink1 0: 0 Sink2 0: 0 Sink3 0: 0 VRM	1 2	red     ▼       Curve_Name:       ✓       ✓       Show Enabled Circuits       ⊕       Left mouse click       ○       Right mouse click       ○       In node list       ○       In circuit tree       Find
Sink1.#.#<====>Sink1.#.#		Auto
Sink1.1.#<====>Sink1.2.# Sink2.1.#<====>Sink2.2.# Sink3.1.#<=====>Sink3.2.#		Add
5000.174 C>50000.274		Update
		Delete
		ОК
<	III	Cancel

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.

Package: Package1		
Upper Plane:		
Lower Plane:		
	<b></b>	
Distribution		
PlaneVDD ~ PlaneGND		
		Add
		Add

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.

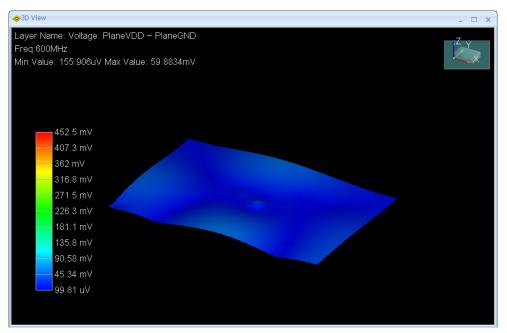
Starting Freq.	Ending Freq.	Sweeping Mode		Points/Decade
MHz	600 MHz	Linear	10 MHz	
		111		

- 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:

 Click the Start Simulation button ▶ to start the simulation. The 3D View window opens.

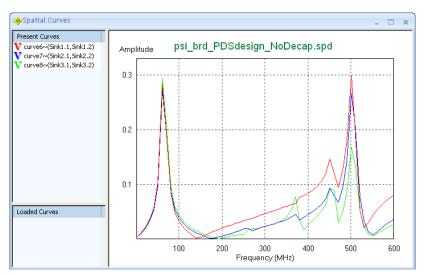


The window contains 3D plots for the voltages between the selected plane pairs.

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

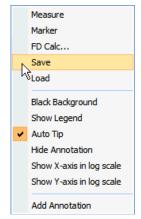
3D Result Display Contro	3D Result Display Control 🛛 🗕 🗖 🗙				
Present Distributions Voltage: PlaneVDD ~ PlaneGND Loaded Distributions					
☐ 3D Structure  ☑ Resul	ts Overlay				
🔾 Local Auto 🛛 💿 Globa	l Auto				
O Fixed					
Max Value: 452,494	nV				
Min Value: 99.8115 u	N				
	Apply				
Start Freq : 1	MHz				
<li></li>	>				
End Freq : 600	MHz				
<					
Faster	Slower				
3					
Play Stop	Show Peak				
Layer Selection 3D Result I	Display Co				

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

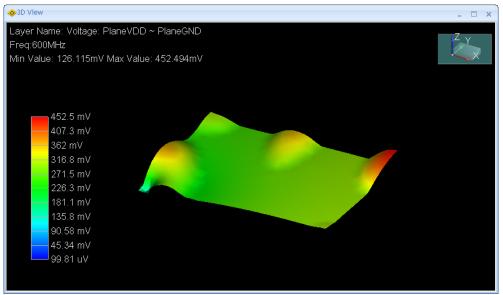
#### 5.2.1 Review Spatial Voltage Distributions

Follow these steps to review the spatial voltage distributions:

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

3D Result Display Control 🛛 – 🗆 🗙				
Present Distributions Voltage: PlaneVDD ~ PlaneGND Loaded Distributions				
✓ 3D Structure ✓ Results Overlay				
O Local Auto 💿 Global Auto				
OFixed				
Max Value: 452.482 mV				
Min Value: 3,43963 mV				
Apply				
Start Freq : 501 MHz				
End Freq : 501 MHz				
Faster Slower				
Play Stop Show Peak				
Layer Selection 3D Result Display Control				

- 2. Input values in **Start Freq.** and **End Freq.**, or scroll the slide bars to adjust the values.
- 3. Click Play to run.
- 4. Click **Show Peak** 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

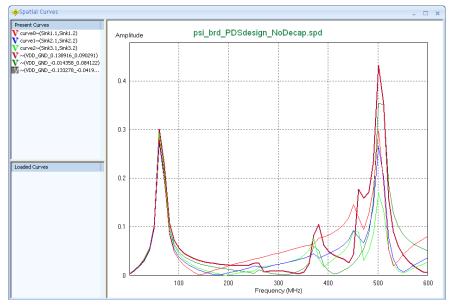
1. Double-click a hot-spot location.

The Add Curve window opens.

Add Curve	×
Upper Layer:	PlaneVDD
Lower Layer:	PlaneGND
Location:	(0.138742, 0.083956)
Max Megnitude:	(428.9mV, 424.6mV@501MHz)
Curve Name:	VDD_GND_0.138742_0.083956
	K Cancel

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.

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

Use the capacitors with corresponding resonance frequencies below:

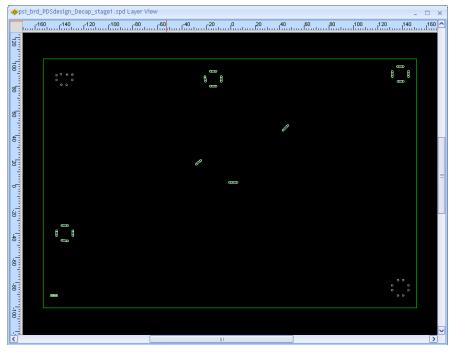
0508	357.75	330 p	0.6 n	1.39
0508	438.15	220 p	0.6 n	1.58
0508	484.39	180 p	0.6 n	1.65

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.

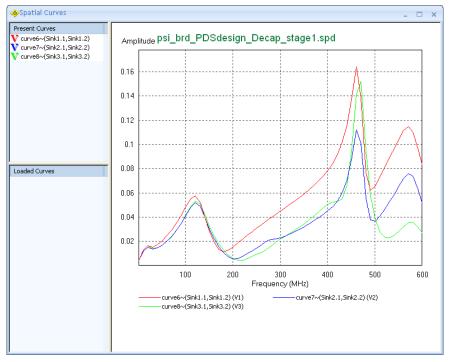


This example has 12 decoupling capacitors placed at the desired locations.

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

	Ckt Na	ame	Δ	Mod	el Name	Sta	artLa	Att	achLay	/er
<ul> <li>A</li> </ul>	decap	0508	3 11		508 1					
🗸 🖂	decap	0508	3_12	0	508_1					=
🗸 🖂	decap	0508	13	0	508_1					_
🗸 📐	decap.	0508	21	Øo	508_2					
🗸 🗸	decap.	_0508	3_22	Øo	508_2					
🗸 🗸	decap.	_0508	3_23	Øo	508_2					
🗸 📐	decap.	_0508	3_31	ØO	508_3					
	decap.				508_3					~
<u>~</u> 🕅	decan	0509	1 33	<u> 22 n</u>	508.3					<b>⊳</b>
						_		7		
	New		Del	E	dit		Load		Filter	
_	ode 🛆					er Na	me			
1		Node	e071		Sigr	al02				
2										
3						laa				
4		Node	8073		Sigr	al02				
Link Unlink										
.PartialCkt 0508_1         ▲           C1 1 2 220p         □           L1 2 30.6n         □           R1 3 4 1.58         □           EndPartialCkt         ✓										

- 3. Click the **Start Simulation** button  $\triangleright$  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.

	Measure
	Marker
	FD Calc
	Save
	Load
	Black Background
	Show Legend
4	Auto Tip
	Hide Annotation
	Show X-axis in log scale
	Show Y-axis in log scale
	Add Annotation

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.

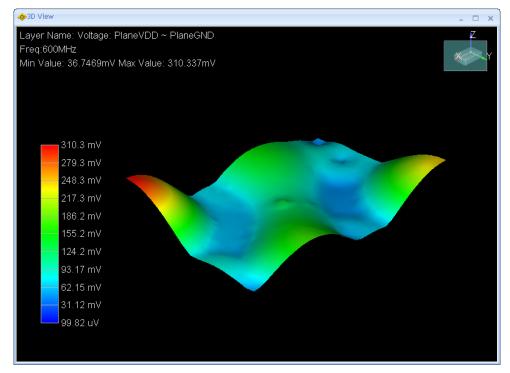
This section has the following topics:

- 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 Show Peak 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.

Add Curve	×
Upper Layer:	PlaneVDD
Lower Layer:	PlaneGND
Location:	(0.141551, -0.085770)
Max Megnitude:	(297.3mV, 296.1mV@471MHz)
Curve Name:	VDD_GND_0.1415510.085770
0	K Cancel

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

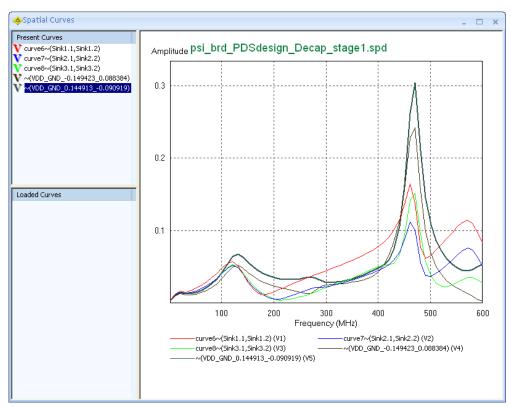
Repeat Step 1 and 2 to add other hot-spot voltage responses to the Spatial Curves window.

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.

**PowerSI** Tutorial



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.

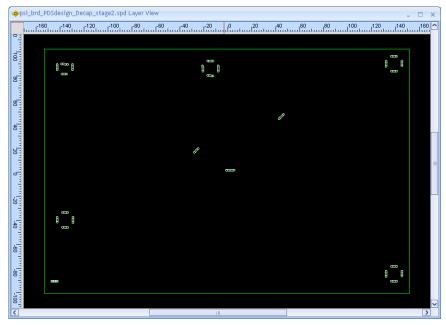
Use the capacitors with corresponding series resonance frequencies below:

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.

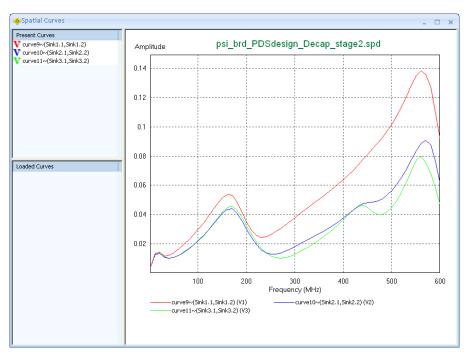


Eight additional decoupling capacitors have been added.

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

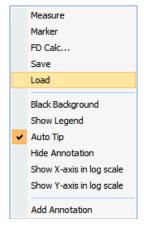
Circuit/Linka	age Manag	er	-	□ ×	
			-	ρ	
Kt N	ame 🗠	Model	Name		
🗸 🖪 decap	_0508_11	050	08_1		
	_0508_12		08_1		
V 🛆 decap			08_1	_ ≡	
VA decap		050	-		
✓ A decap✓ A decap		050	J8_1 D8 1		
V A decap			JO_1 D8 1	-	
V A decap		1	08_1 08_2		
V 🖪 decap		1	 08_2		
<		<i>A</i> 0	· · ·	2	
New		, 			
New	Del Ed		ad Filter	7	
Ckt Node 🛆	Pkg Node		Layer Nam	e	
1 Node045 Signal02					
2					
3					
4	Node043		Signal02		
Link					
.PartialCkt 0508_1           C1 1 2 220p           L1 2 3 0.6n           R1 3 4 1.58					
<	111			>	
Layer Selectio	n Circuit/Lin	kage Ma	nager		

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.

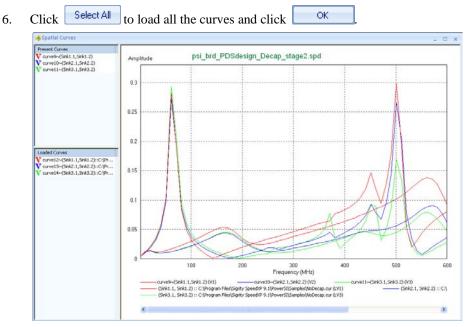
4. Right-click the plot to display the pop-up menu.



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

The Load Curve dialog box opens.

.oad Curve DialogBox	×
<ul> <li>(Sink1.1, Sink1.2)</li> <li>(Sink2.1, Sink2.2)</li> <li>(Sink3.1, Sink3.2)</li> </ul>	Select All Deselect All OK Cancel



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.

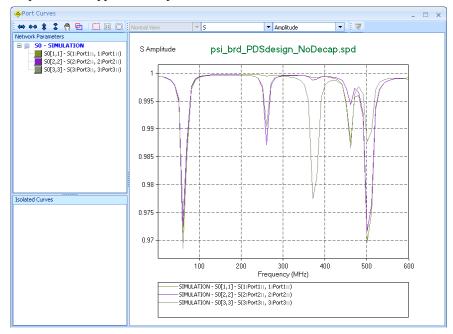
	In some cases, you might see a warning regarding the circuit definitions of the source components. The <b>Extraction Mode</b> does not accept independent
NOTE!	voltage or current sources that do not equal zero, and gives out a warning. In this example, the voltage source $= 0$ . If you see this warning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

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:



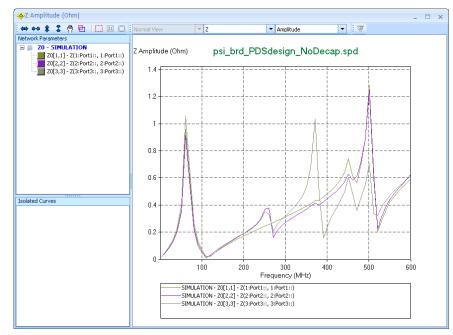
4. Click the **Start Simulation** button to start the simulation without decoupling capacitors. All port curves appear on the plot.



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

iitude 💌

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.

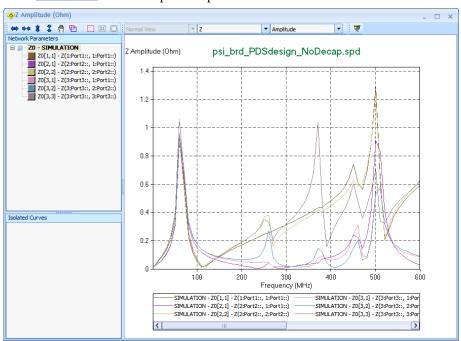
1. Click the **Channel Filter** button 💆 on the **Curve Operations** toolbar to open the **Channel Filter** dialog box.

		Return Loss
Channel	Ports in Channel	
Unnamed Net	3	Insertion Loss
		Crosstalk

2. Check **Insertion Loss**.

Ports in Channel 3	Return Loss
3	Insertion Los
	Crosstalk
	Crosstaik

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.

	Save Simulation Result
5	Measure
	Marker
	Expression Calculator
	Black Background
	Embed-Ctrl Visibility

5. Select **Save Simulation Result** to save the S-parameters and impedance matrix in the psi\_brd\_PDSdesign\_NoDecap.S3P for later comparison.

Save	Curves			x
N	etwork par	rameter file	format: Touch	hstone format
Sa	ave to	C:\Sigrity	Samples\Speed>	XP 11.0\PowerSI\Samples\Hybrid\psi_brd_dem
	Data Type	O DB	ОМА	Save the following network parameter(s)
	Export pa MCP Ou		orced S-paramet	ters Touchstone 2.0 Mixed mode
		E	ОК	Cancel

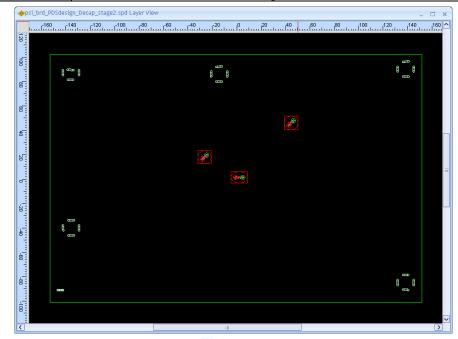
- 6. Select Window > Layer View to switch to the Layer View window.
- 7. Select File > Close to close the current project.

#### 5.4.2 Run Simulation with Decoupling Capacitors

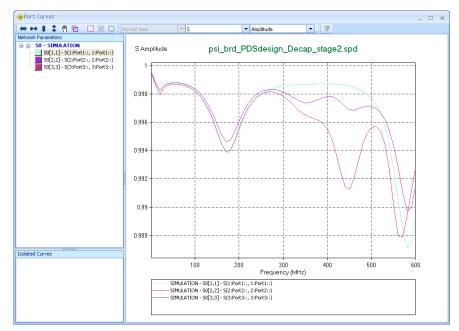
1. Select File > Open... to open the file psi\_brd\_PDSdesign\_Decap\_stage2.spd from the PowerSI Samples directory.

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**.



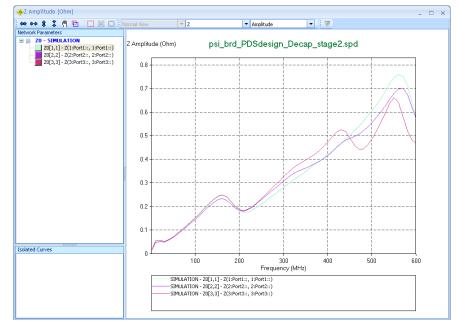
Press the Start Simulation button ▶ to start the simulation.
 All port curves appear on the plot.



•

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

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  $\mathbf{Z}$  from the **Curve Settings** toolbar, the self impedance at each port shows. To view transfer impedance, plot the transmission parameters.

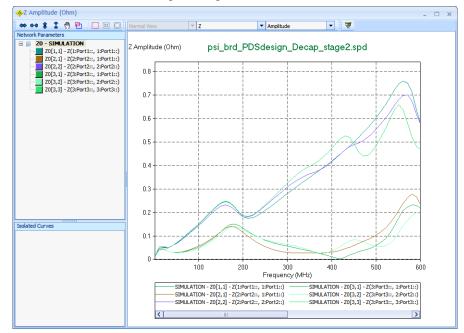
1. Click the **Channel Filter** button 💆 on the **Curve Operations** toolbar to open the **Channel Filter** dialog box.

<ul> <li>All Channels</li> </ul>		
Channel	Ports in Channel	Return Loss
Unnamed Net	3	Insertion Loss
		Crosstalk

2. Check Insertion Loss.

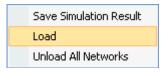
C	iannel Filter		×
	All Channels		
	Channel	Ports in Channel	Return Loss
	Unnamed Net	3	✓ Insertion Loss Crosstalk
			ОК
			Cancel
			Cancel

3. Click to show impedance parameters.

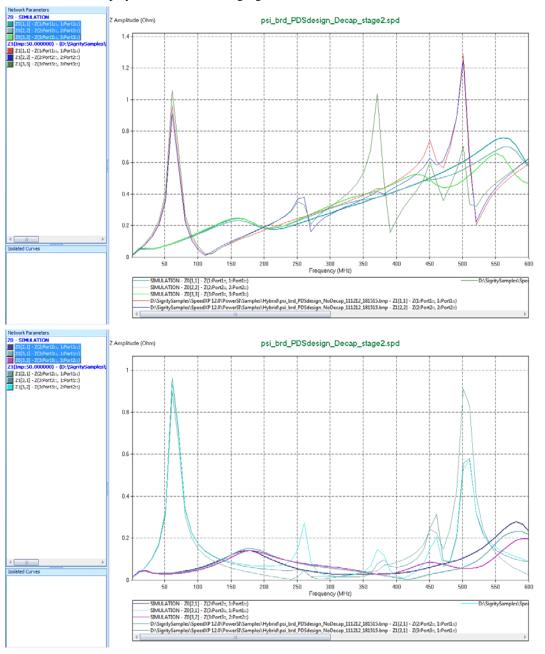


This window shows all of the impedance parameters, both self and transfer impedance for the top half matrix.

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.



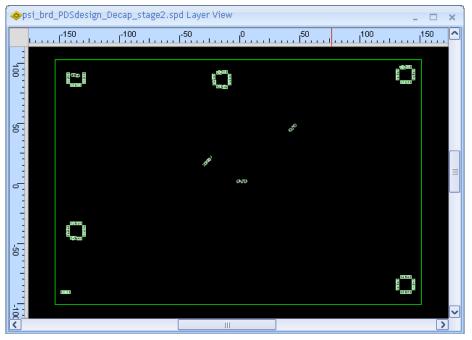
- 6. Select Window > Layer View to switch to the Layer View window.
- 7. Select File > Close to close the current project.

## 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.

Enable Total Mode	Total Frequency Number	10
Frequency Min 1KHz	Frequency Max	100MHz

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**.

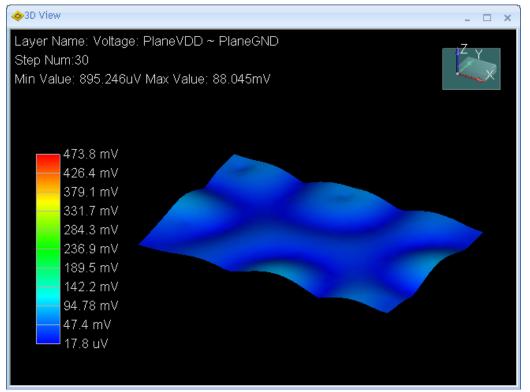
- 3. Click OK
- 4. (Optional) Select Setup > Simulation View > 3D Spatial... to set up the 3D spatial voltage display.

Voltage Distribution	×
Package: Package1	
Upper Plane:	
	]
Lower Plane:	
	]
Distribution	
PlaneVDD ~ PlaneGND	
	Add
	Delete
	ОК
	Cancel

#### 5.5.3 Run a Simulation

Click the **Start Simulation** button  $\blacktriangleright$  to start a simulation.

The 3D view looks as shown below.



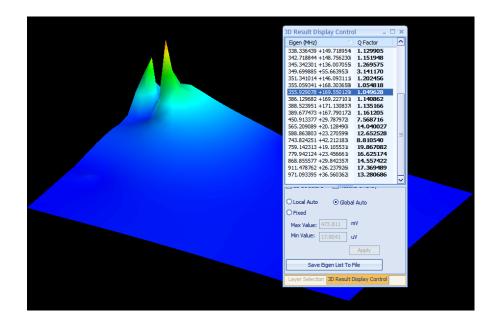
**PowerSI** outputs a table which lists complex resonant frequencies and Q factors in the **3D Result Display Control** window as shown below:

3D Re	esult Display Control	X
No.	Resonance freq (MHz)	Q Factor
1 2 4 5 6 7 8 9 10	15.138267 +6.71 23.035848 +29.00 24.248250 +22.95 40.878040 +28.60 53.929644 +78.09 55.541386 +73.26 172.153045 +36.3 283.261371 +119 347.819705 +56.4	1.127364 0.397155 0.528253 0.714645 0.871770 0.345262 0.379047 2.368178 1.187772 3.079277
1	III ent Distributions /oltage: PlaneVDD ~ PlaneGN led Distributions	D
Load	ent Distributions /oltage: PlaneVDD ~ PlaneGN	
Load	International Content of the second s	
Load 3D O Loa O Fix	International Content of the second s	
Load 3D O Loa O Fix Max	ent Distributions /oltage: PlaneVDD ~ PlaneGN led Distributions Structure  Results Ove cal Auto  Global Auto red	
Load 3D O Loa O Fix Max	ent Distributions /oltage: PlaneVDD ~ PlaneGN led Distributions Structure  Results Ove cal Auto  Global Auto red (Value: 471.379 mV	rlay
Load 3D O Loa O Fix Max	ent Distributions /oltage: PlaneVDD ~ PlaneGN led Distributions Structure	rlay

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.

#### **PowerSI** Tutorial



## 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.

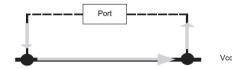
This chapter has the following sections:

- 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:

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

The PowerSI simulation results should closely match the formula results.

This section has the following sections:

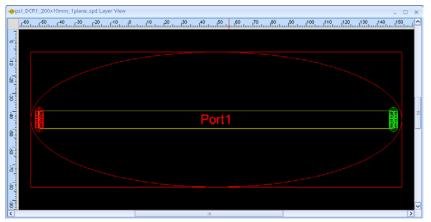
Open a Project

- 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....

Frequency Rang	jes			×
Starting Freq.	Ending Freq.	Sweeping Mode	Freq. Increment	Points/Decade
100 Hz	200 Hz	Linear	100 Hz	
<		111		
Add	Delete		ОК	Cancel

- 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.

<b>TIP:</b> 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).

File Manager         Save Options         Hotkeys         Layout       ●         Ord and Unit       ●         View       ●         Processing       ●         Trace       E         Error Checking       ●         3D Layout View       ●         Daplay       ●         Quity       ●         Simulation (Basic)       ●         Special Vid       ●         Simulation Report       ●         Simulation (Advanced)       ●         Electric Models       ●         Fried Domain       ●         Meth       Reference Handing         Nets and Shapes       Special Handing         Special Handing       ■         Enctric Models       ■         Fried Domain       ●         Meth       Reference Handing         Nets and Shapes       ■         Special Handing       ■	File 💿	Channel Althe UT field Domains and in Domain ST
Layout       ●         Grid and Unit       ●         View       ●         Processing       Trace         Trace       Error Checking         3D Layout View       ●         Display       ●         Quality       ●         Simulation (Basic)       ●         General       ●         Net and Coupling       ●         Network Parameters       ●         Special Void       When the frequency is low enough so that the skin depth is larger than the metal thickness, the field might couple through the metal shapes. This phenomena can be captured by the inter-plane coupling formulation. Note:         Simulation (Advanced)       ●         Bectric Models       ●         Pred Doman       ●         Meth       Automatic RLGC adjustment for antipad array         Nets and Shapes       ■         Special Handing       ■         ©       Enable automatic RLGC adjustment	Save Options	Change the 'Field Domain' options in PowerSI
View       O Natural boundary condition (considers boundary radiation)         Trace       Error Checking         3D Layout View       O         Deplay       Compensate for shape width         Simulation (Basic)       O         General       O Natural boundary condition (considers boundary radiation)         Network Parameters       O Not considered         Special Void       D Enhanced mode         Network Parameters       O Enhanced mode         Simulation Report       O Enhanced mode         Electric Models       Field Domain         Mesh       Automatic RLGC adjustment for antipad array         Nets and Shapes       Enable automatic RLGC adjustment		Boundary condition
Display Quality       Inter-plane coupling (IPC)         Simulation (Basic) (*)       O Not considered         General Net and Coupling Network Parameters       Issue mode         Simulation (Advanced) (*)       D Enhanced mode         Simulation Report       When the frequency is low enough so that the skin depth is larger than the metal thidness, the field might couple through the metal shapes. This phenomena can be captured by the inter-plane coupling formulation. Note: Including IPC in the simulation may result in heavy memory usage and long simulation time         Electric Models Field Domain Meth Reference Handling       Automatic RLGC adjustment for antipad array         Letable automatic RLGC adjustment       Enable automatic RLGC adjustment	Grid and Unit View Processing Trace Error Checking	Natural boundary condition (considers boundary radiation)
Quality       O Not considered         Simulation (Basic)       Image: Constraint of the second		
	Simulation (Basic) (*) General Net and Couping Network Parameters Special Viol Simulation Report Simulation (Advanced) (*) Electric Models Field Domain Mesh Reference Handling Nets and Shapes	Basic mode     Crhanced mode     Crhanced mode     When the frequency is low enough so that the skin depth is larger than the metal thickness, the field might     coupling through the inter-share coupling formulation.     Note: Induding IPC in the simulation may result in heavy memory usage and long simulation time  Automatic RLGC adjustment for antipad array
Detault Appry OK		Default Apply OK C

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

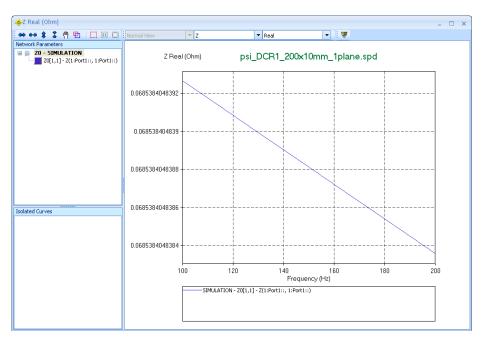
#### 6.1.4 Run a Simulation

Follow these steps to run a simulation:

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



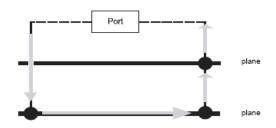
The curve looks as shown below:



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 ( $\sigma$ ).

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

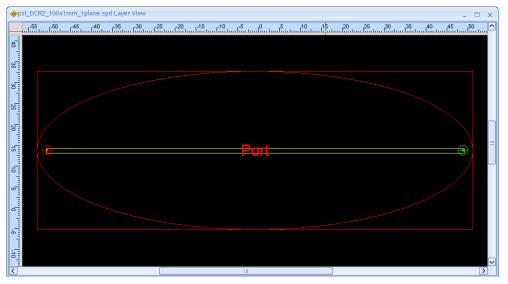
 $R = L/(\sigma \times W \times T) = 0.345Ohms$ 

The PowerSI simulation results should closely match the formula results.

#### 6.2.1 Compute Resistance

Follow these steps to compute resistance:

 Open a Project and load the file psi\_DCR2\_100x1mm\_1plane.spd. See 5.1.1 Open a Project.



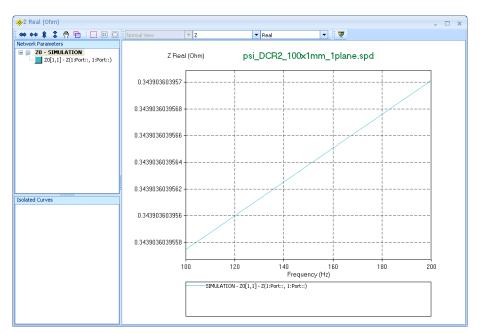
In this example, the positive terminal is connected to the Vcc plane on the left through a through-hole via, and a negative terminal connected to the Vss and Vcc planes on the right.

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

Frequency Rang	es			×
Starting Freq.	Ending Freq.	Sweeping Mode	Freq. Increment	Points/Decade
100 Hz	200 Hz	Linear	100 Hz	
<		111		>
Add	Delete		ОК	Cancel

- 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 OK
- 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.

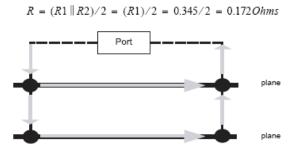


The resistance result is 0.3430 ohms closely matching the analytical results.

## 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.

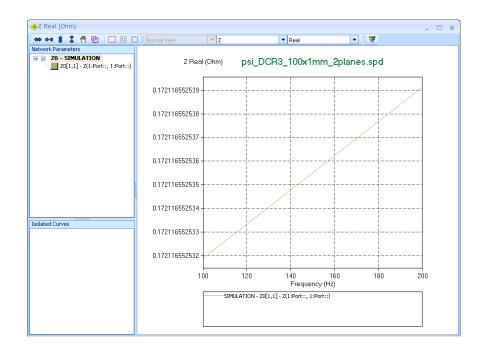
The result can be analytically computed by the following equation:



#### 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.

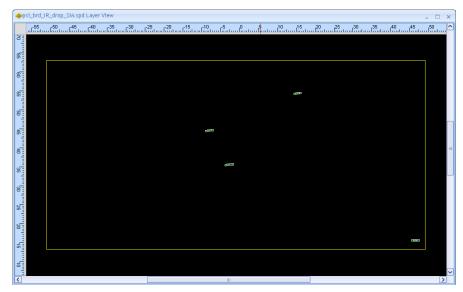
This section has the following topics:

- 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.

= 1	odel Name		
🗸 🖪 Sink1 🛛 🍕			- 🔎
🗸 🖪 Sink1 🛛 🍕			
🗸 🖪 Sink1 🛛 🍕		StartLaver	AttachLave
=	Isource1		
🗸 🖪 Sink2 🛛 🐇	Isource2		
🗸 🖪 Sink3 🛛 🏈	Isource2		
🗸 🖪 VRM 🛛 🏈	Vsource		
<	111		
New Del	Edit	Load	Filter
Ckt Node 🛆 🛛 Pkg Node	e	Layer Name	
1 Node017		Signal02	
2 Node019		Signal02	
Link Unlink			
.PartialCkt Isource1			-
I1 1 2 AC=2.0			_
			=
.EndPartialCkt			
<			<u> </u>

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.

Circuits/Sub Circuits	Circuit Node/SubCkt Node	Curve_Color:
B Sink1 B Ø Sink2 B Ø Sink3 B Ø VRM		green Curve_Name: ✓ Show Enabled Circu ⊕ Left mouse click ○ Right mouse click ○ Quick Find ○ In node list ○ In circuit tree Find
ink1.#.#<====>5ink1.#.# 5ink1.1.#<====>5ink1.2.# 5ink2.1.#<====>5ink3.2.# 5ink3.1.#<====>5ink3.2.#	11	Auto Add Update Delete OK Cancel

## 6.4.4 Set up Spatial Voltage Distributions

2.

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.

Voltage Distribution	×
Package: Package1 Upper Plane:	
Lower Plane:	
Distribution	
PlaneVSS ~ PlaneVCC	Add Delete OK Cancel

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

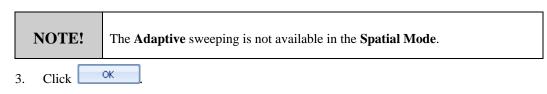
Follow these steps to set up simulation frequencies:

1. Select Setup > Simulating Frequencies....

The Frequency Ranges dialog box opens.

Starting Freq.	Ending Freq.	Sweeping Mode	Freq. Increment	Points/Decade
00 Hz	200 Hz	Linear	100 Hz	
<				

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



## 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).

<ul> <li>Magnetic wall (Total reflection)</li> </ul>	
• Natural boundary condition (co	onsiders boundary radiation)
Compensate for shape wid	ith
nter-plane coupling (IPC)	
O Not considered	
• Basic mode	
O Enhanced mode	h so that the skin denth is larger than the metal thickness. The field minbt
When the frequency is low enoug couple through the metal shapes. Note: Including IPC in the simula	h so that the skin depth is larger than the metal thickness, the field might This phenomena can be captured by the inter-plane coupling formulation. ation may result in heavy memory usage and long simulation time
When the frequency is low enoug couple through the metal shapes. Note: Including IPC in the simula	This phenomena can be captured by the inter-plane coupling formulation. ation may result in heavy memory usage and long simulation time
When the frequency is low enoug couple through the metal shapes. Note: Including IPC in the simula <b>owerDC Option</b>	This phenomena can be captured by the inter-plane coupling formulation. ation may result in heavy memory usage and long simulation time
When the frequency is low enoug couple through the metal shapes. Note: Including IPC in the simule <b>owerDC Option</b>	This phenomena can be captured by the inter-plane coupling formulation. ation may result in heavy memory usage and long simulation time e pr antipad array

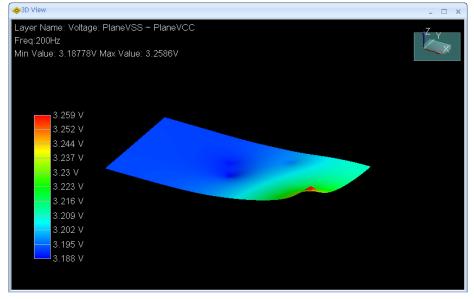
#### 6.4.7 Run a Simulation

3.

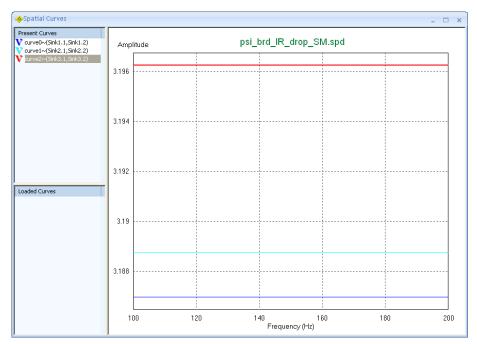
Follow these steps to perform a simulation and view the curves:

1. Click the **Start Simulation** button **b** 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.

Change the 'Displ	ay'	options in PowerSI			
Color Scheme					
Background		Line			
Node		Selected Node			
Antipad Display					_
Антирай Діяріаў					
○ As Void      Outline		O Hide			
Walk Through					
Walk pace: 50 um/Step	10	· · · · · · · · · · · · ·	100um/step		
Camera distance: 500 um	0	· · · · · · · · · · · · · · · · · · ·	1mm		
Camera angle: 50 d	30	·	100d		
		Default	Apply	ОК	Cancel

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

3D Result Display Contro	ol _□×
Present Distributions	
Voltage: PlaneVSS ~ Pla	aneVCC
Loaded Distributions	
3D Structure 🗹 Resul	ts Overlay
O Local Auto 💿 Globa	l Auto
O Fixed	
Max Value: 3.2586	/
Min Value: 3, 18778	/
	Apply
Start Freq : 100	Hz
<	>
End Freq : 200	Hz
< 1	>
Faster	Slower
3	
Play Stop	Show Peak
3D Result Display Control	ayer Selection

# Index

•		Ε	
.spd	8	Ending Freq.	13, 40, 50, 75, 78, 83
3		Export results	40 8
3D Result Display Control	50, 52, 57, 85	extraction mode	0
3D Spatial	49	F	
3D View	36, 50	Features	6
Α		Find & Fit	12
AC analysis	22	Fit button	12 40, 50, 75, 78, 83
Adaptive	13	Freq. Increment Frequency Ranges	13, 40, 50, 70, 83
add a trace	35	frequency sweeping	13, 40, 50, 70, 85
add a via	33	inequency sweeping	15
Add Curve	53	G	
Add Trace	35	ground plane	21
Add Via	32	Н	
В			
-	21	HSPICE	7, 22, 43, 44
bottom signal layer	21	HSPICE simulation	22, 37, 45
Box Properties	30	Ι	
Broadband SPICE Broadband SPICE Model Val	7	L/O madalina	21
Broadband SPICE Woder van	lidation 22	I/O modeling Inductance	21 54, 59
С		installation directory	54, 59 10
Capacitor Type	54, 59	inter-plane coupling	76, 83
capacitors	18	Inter-plane coupling	70, 83
change a trace	35, 36	inter-plane coupling	70
circuit 6, 7, 8, 11, 22, 37, 42,	,	L	
81	, 10, 10, 02, 00,	Layer Selection	29, 35, 36, 37
circuit voltage	48, 81	layer view	11
Circuit/Linkage Manager	17, 48, 59, 81	Layer View	30, 32
construct a package model	23	Linear	40, 50, 75, 78, 83
Constructing a Package	22	load a file	10
create a power ground shape	29	Load a file	17
cross-section structure	21	м	
Curve Pattern Property	15	Μ	
Curve Settings	63, 66, 67, 76	modeling	21, 22
Customer Support Informatio	n 5	Modeling and Transier	nt Simulation 7
D		Ν	
data file	8	negative terminal	12
data structures	47	Negative Terminal	39
Decoupling Capacitor Placem	nent 47	netlist	22, 43, 44, 46
decoupling capacitors	10, 18, 19	network parameters	6, 13
design		Network Parameters p	ane 19
architecture	10	New Ports	37
component level	5, 7, 21	Node Move	34
user interface	74		
documentation	5		

## Р

package	7, 12
PCB	21
planes	6
Port	38
Port setup	12
ports 8, 22, 34, 37, 38, 39, 40, 41, 43	, 62, 75
positive terminal	12
Positive Terminal	39
Power Delivery System	47
power ground	18
power ground noise	22
Power Ground Noise Without Decoup	pling
Capacitors	47
power ground performance	19
power plane	21
Present Curves	64, 67

#### R

Resistance	54, 59
Resonance frequency	54, 59
resonance mode	9, 69
Run a simulation	10, 17
Running a Simulation	22

## S

Save Curves	14, 41, 42	
Save Simulation Result	14, 41	
Self and Transfer Impedance	47	
Set up simulation frequencies	10, 17	
Shape	30	
simulation 13, 18, 21, 22, 50, 54, 55, 60, 62,		
66, 76, 78, 84		
simulation frequencies 13, 18, 40, 4	49, 75, 82	
simulation report	15	
Software Overview	6	
S-parameter 13, 2	22, 41, 43	
Spatial Curves 51, 5	53, 55, 58	
Spatial Mode 8, 47, 63, 6	67, 80, 83	

spatial mode simulation	on 80
spatial voltage distribu	
Specifying Ports	22
SPEED2000	7
stackup	23, 26, 47
Start Simulation butto	n 13, 19, 41, 50, 55,
60, 62, 66, 71, 76, 7	78, 84
start the PowerSI	10
Starting Freq.	13, 40, 50, 75, 78, 83
Status Bar	33
Sweeping Mode	13, 40, 50, 75, 78, 83
system architecture	6

#### Т

Thickness	27
top signal layer	21
Touchstone	15
trace	6, 7, 8, 21, 34, 35, 36, 74
Trace Property	35
traces	6, 8
transient analysis	21
transient simulation	n 7, 43, 44
Transient Simulati	on 22

#### $\mathbf{V}$

via 6, 7, 8, 21, 33, 34, 35, 39	, 74, 78, 79
Via Editing	32
vias	6, 8, 21
View the port setup	10
View the setup	17
view traces in 3D	36
Voltage Distribution	82
voltage distributions	49, 52, 82
W	

## workflow workplace

## Z

Z-parameter 14, 19, 63, 66

8 6